SOLIDWORKS

Basics and Modelling Fundamentals

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.

- In SolidWorks, you sketch ideas and experiment with different designs to create 3D models.

SolidWorks

- SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.

- Designing in a modeling package such as SolidWorks is beneficial because it saves time, effort, and money that would otherwise be spent prototyping the design.

SolidWorks Components - PARTS

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

Pieza

- 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

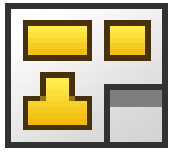
- The second component is the assembly. Assemblies are collections of parts which are assembled in a particular fashion using mates (constraints).

- Any complex model will usually consist of one, or many assemblies.

Ensamblaje

SolidWorks Components - DRAWINGS

- The third, and final component in SolidWorks is the Drawing.

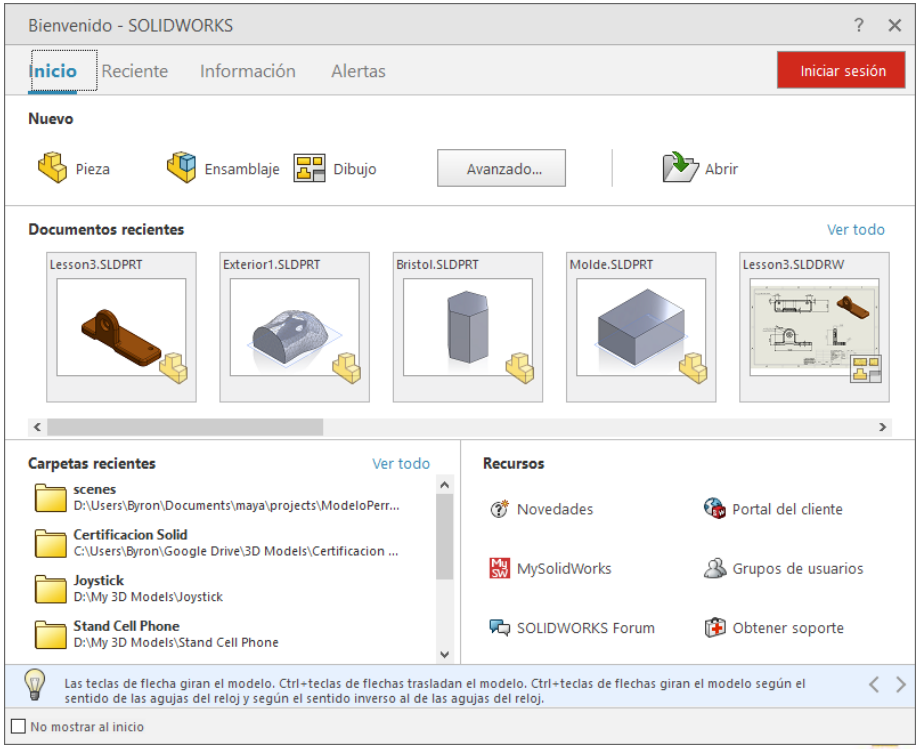


Dibujo

- A drawing is the typical way to represent a 3D model such that any engineer (or manufacturer) can recreate your part.

- Drawings are important because they provide a standard way of sharing your design.

SolidWorks – Let's Begin

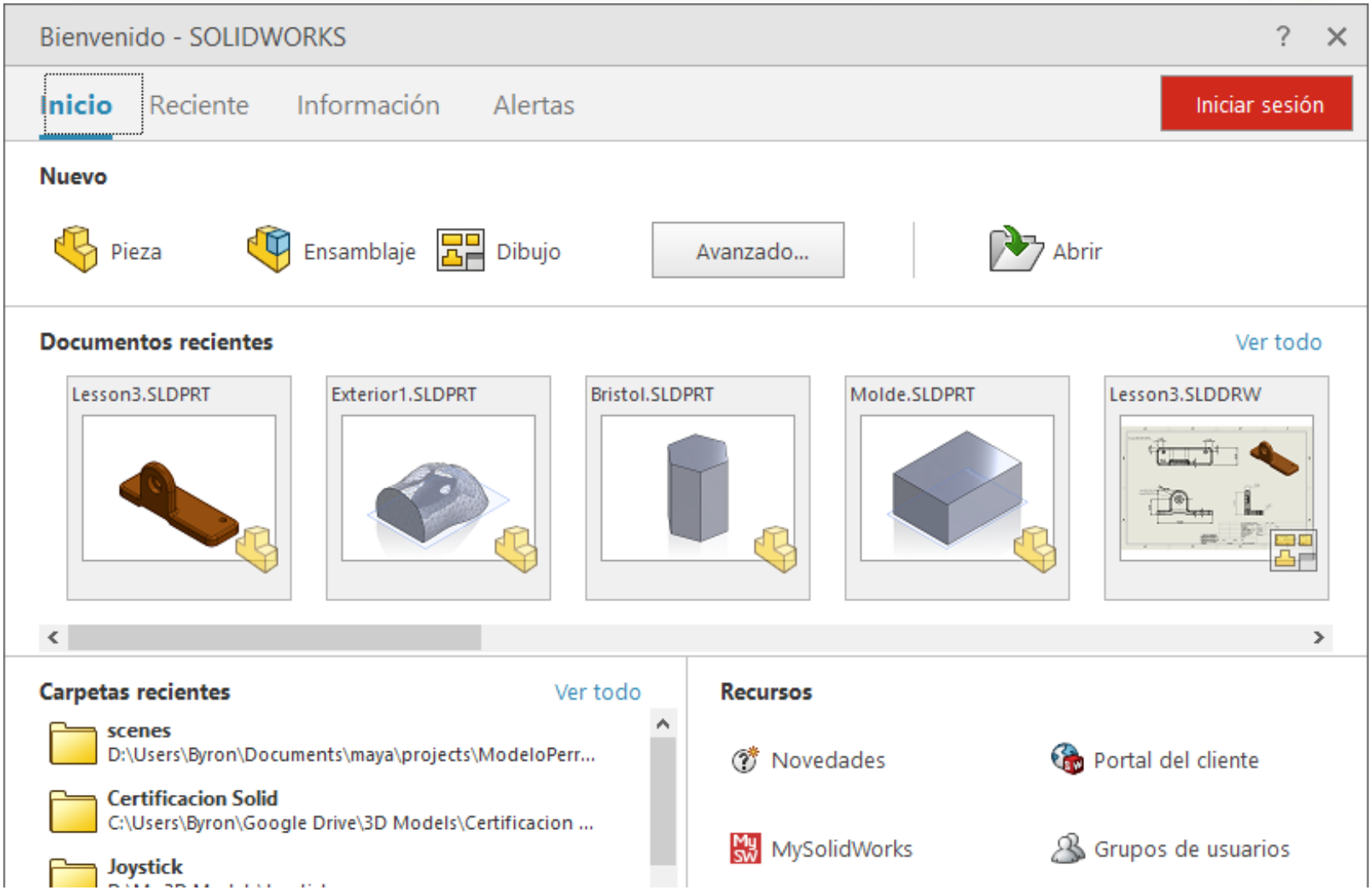


By default, no file is opened automatically when you start the program.

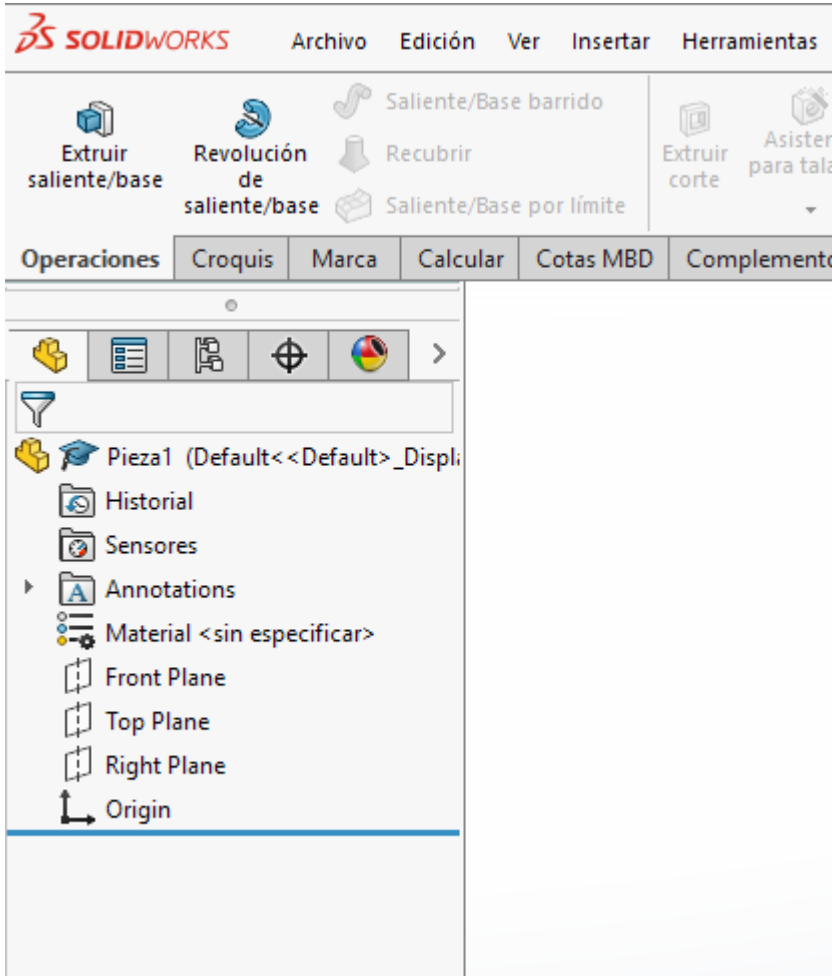
To create a new file, click on File > New or click the New File icon in the main toolbar.

This will open the New SolidWorks Document wizard.

SolidWorks Tour

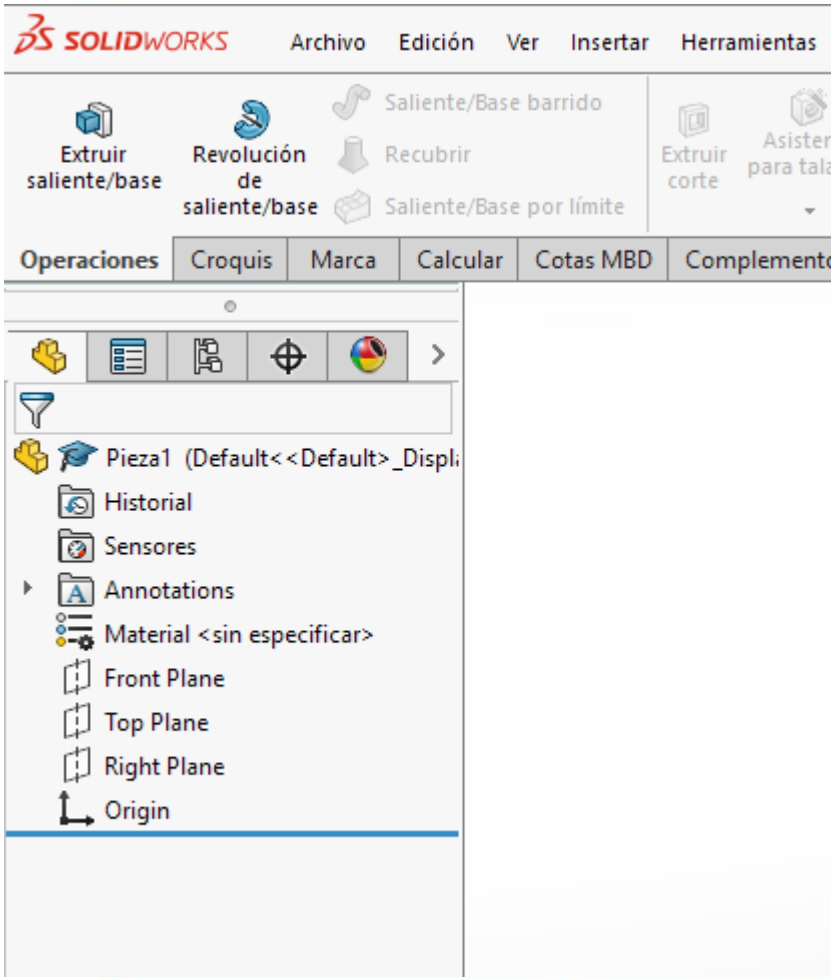


SolidWorks Tour



- 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

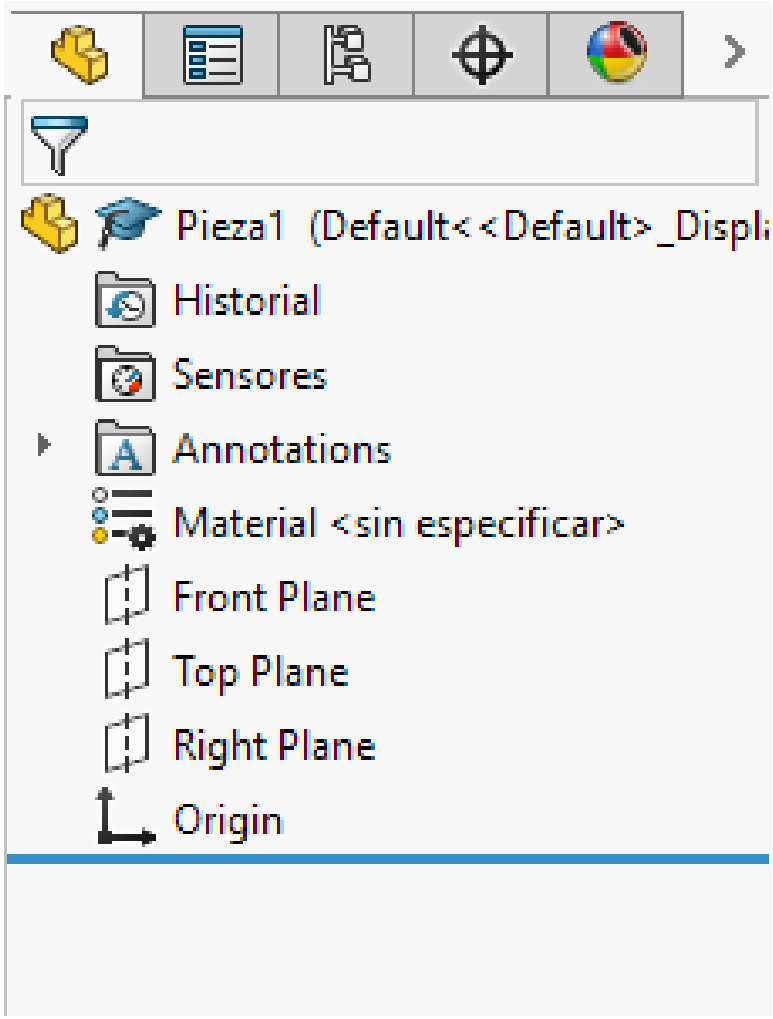
SolidWorks Tour



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

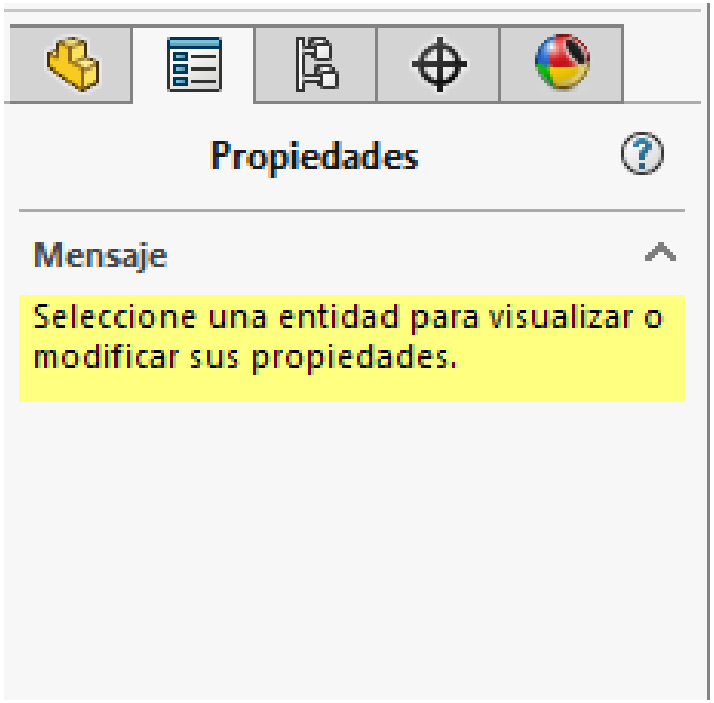
First, the left side of the screen consists of several tabbed panes that provide very important information regarding your model.

SolidWorks Tour



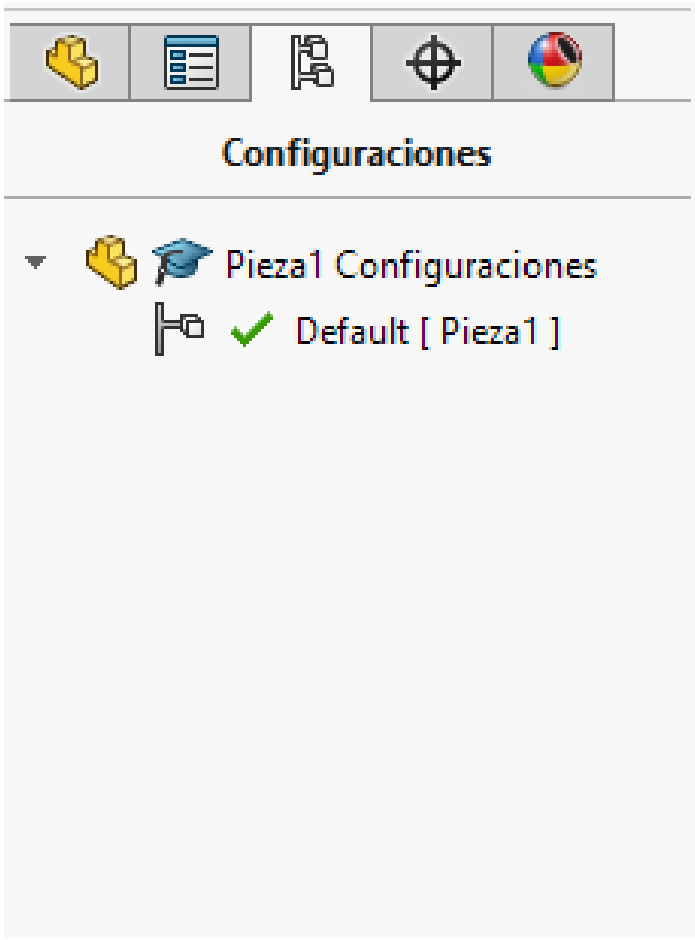
- The first tab, called the Feature Manager, lists all features that have been created within your model.
- This tab is extremely important as it will be from here that you select and change features once they have been created.

SolidWorks Tour



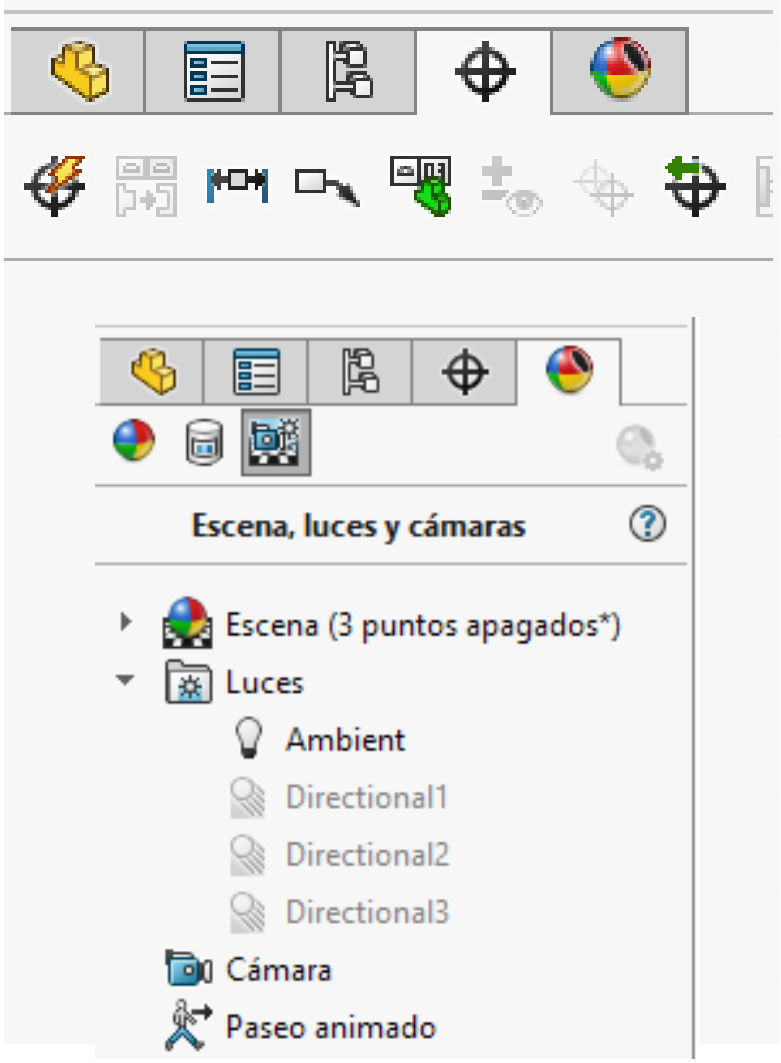
- The second tab, called the Property Manager, allows you to adjust the properties of various entities either during construction, or once it has been created.
- Note that generally you will not need to manually change the tab on the manager window

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.

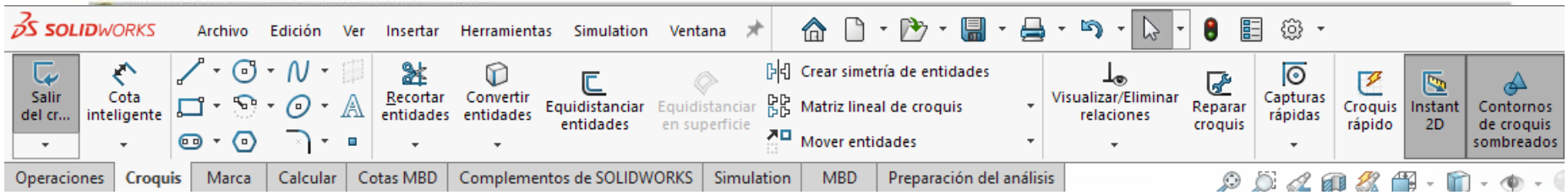
SolidWorks Tour



- 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

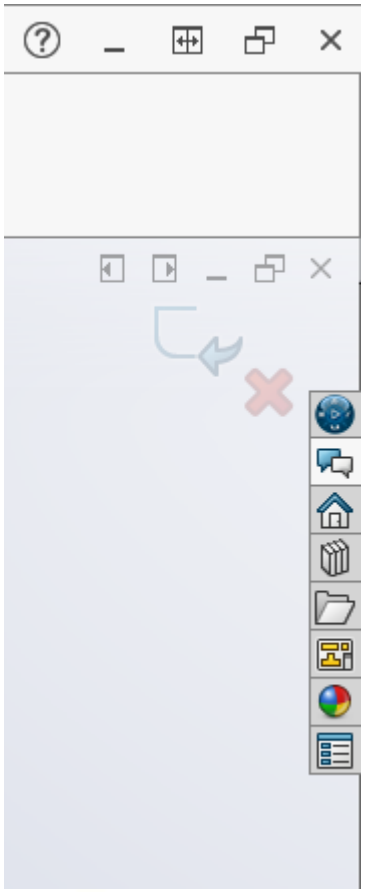
- The next important feature of the interface is the dynamic Toolbar



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

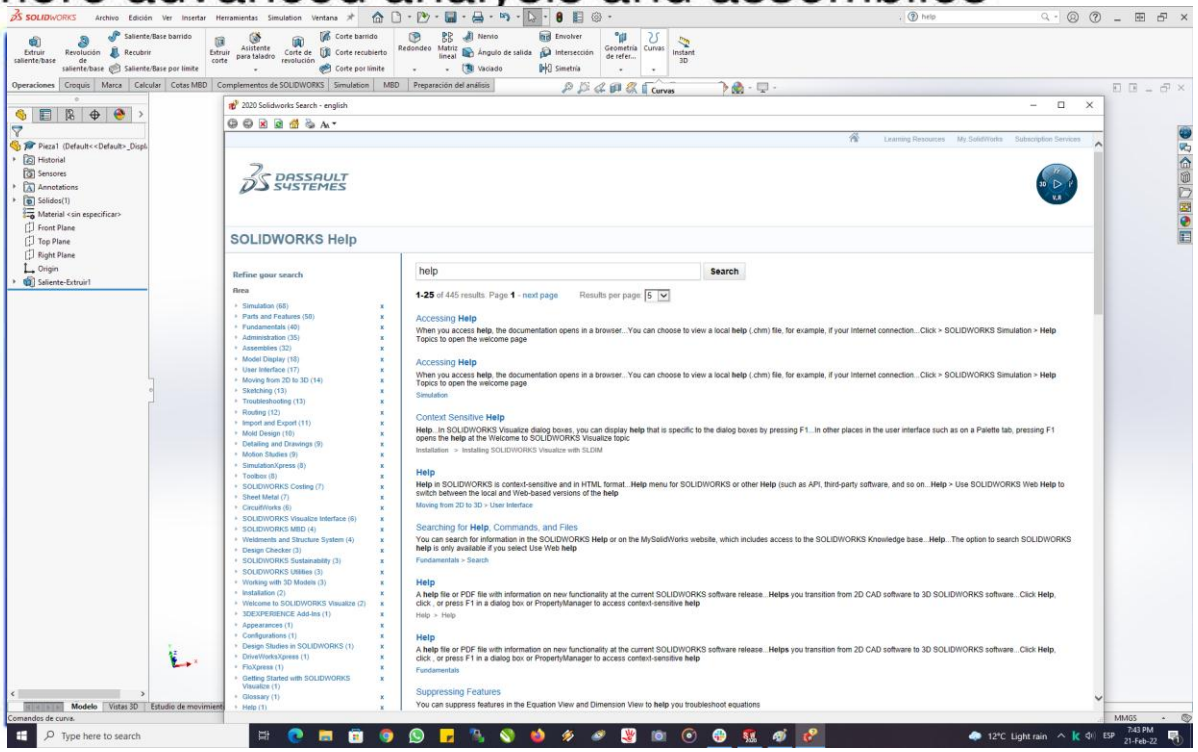
SolidWorks Tour

- 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

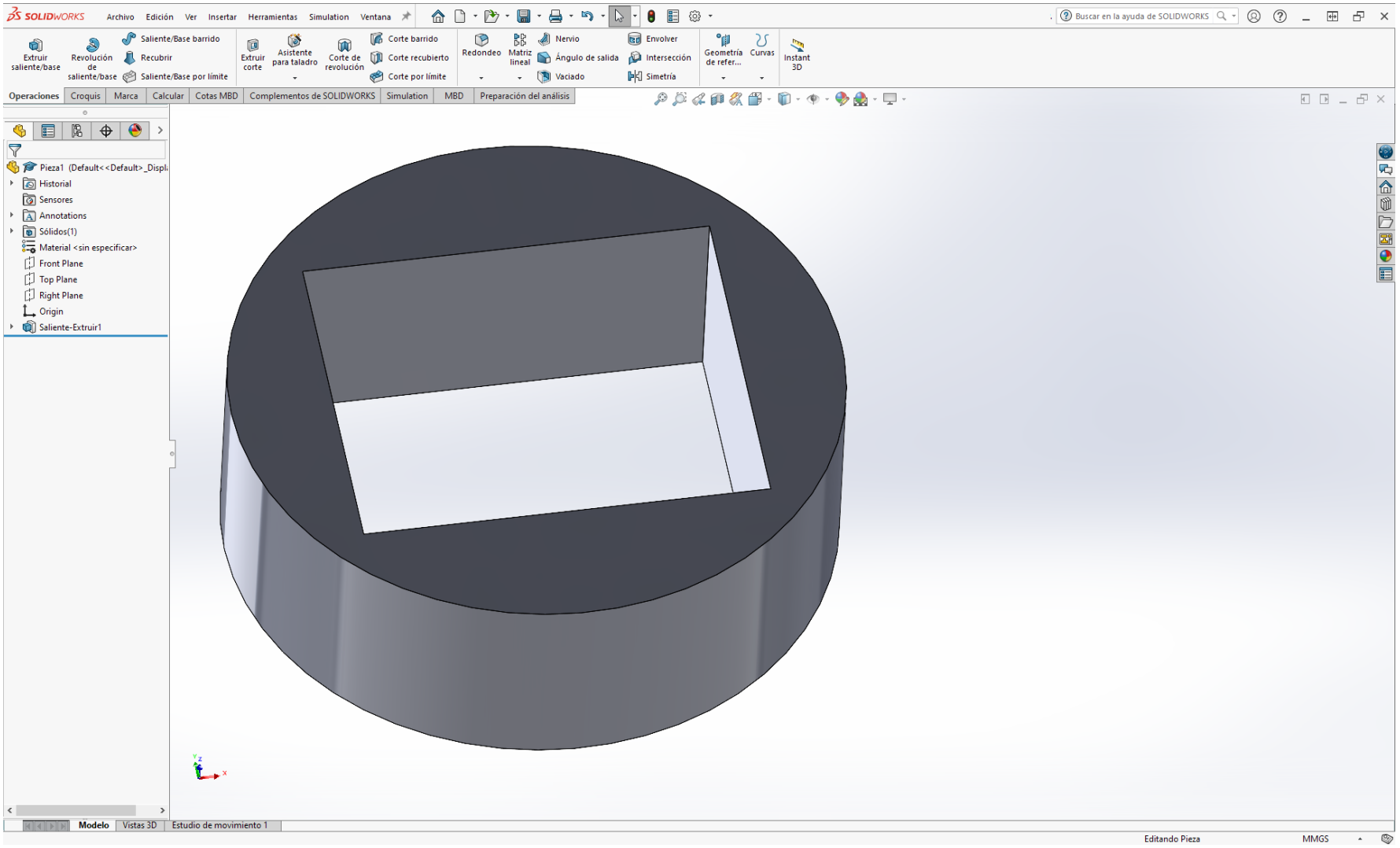


SolidWorks Exercise

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

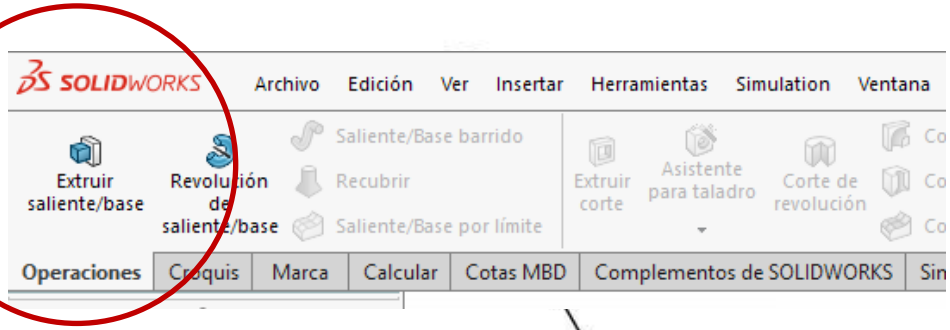
For now, we are only going to concern ourselves with two types of features, Extruded Boss/Base and Extruded Cut.

SolidWorks Exercise

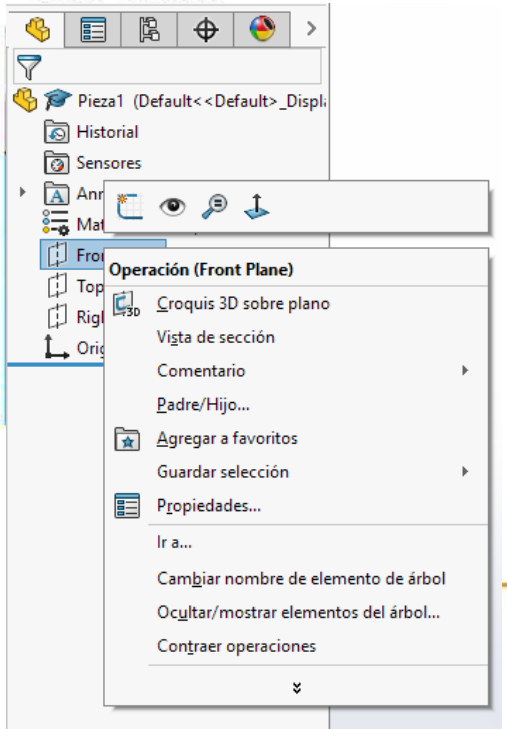


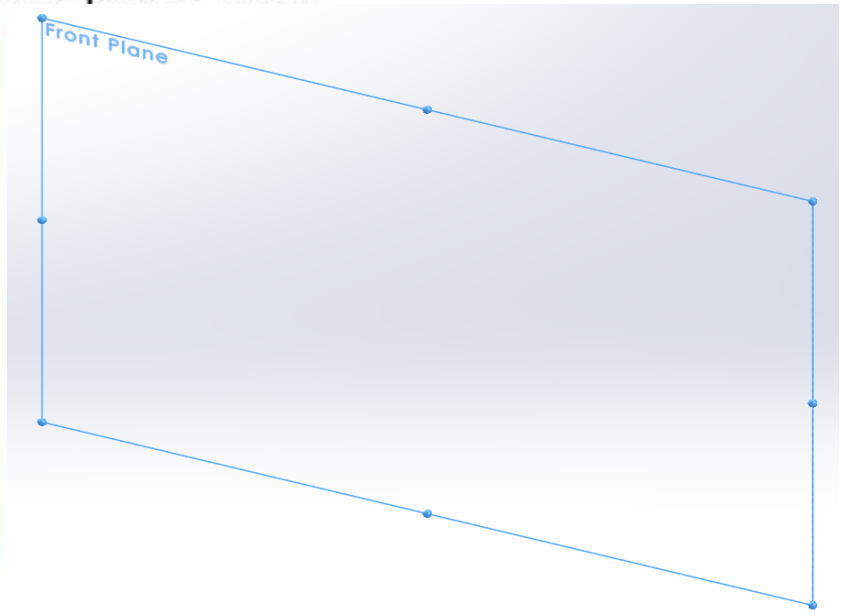
SolidWorks Exercise

Let's begin by selecting EXTRUDED Boss Base

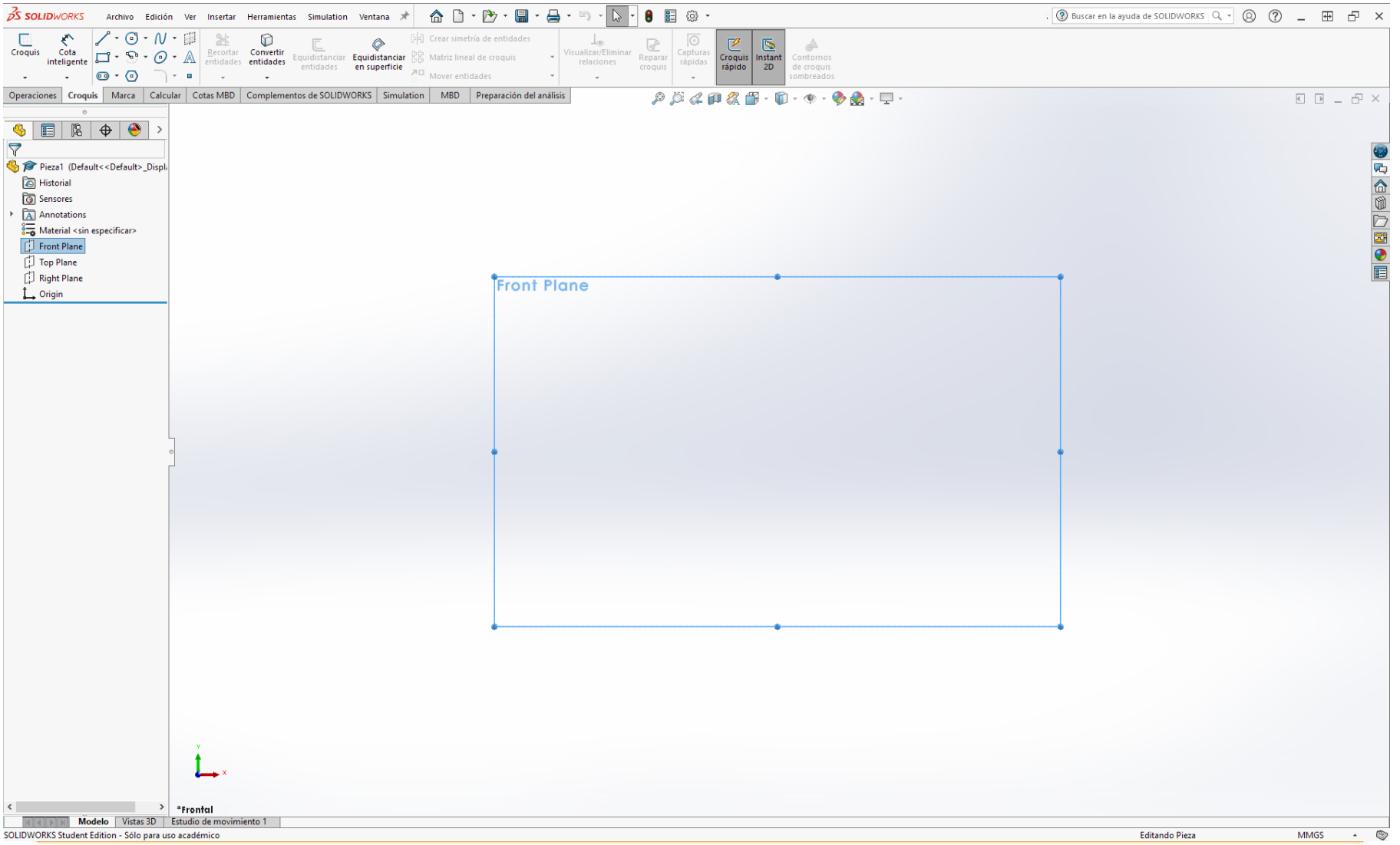


You should notice that your tab will change to property manager asking you to select a plane from the view.

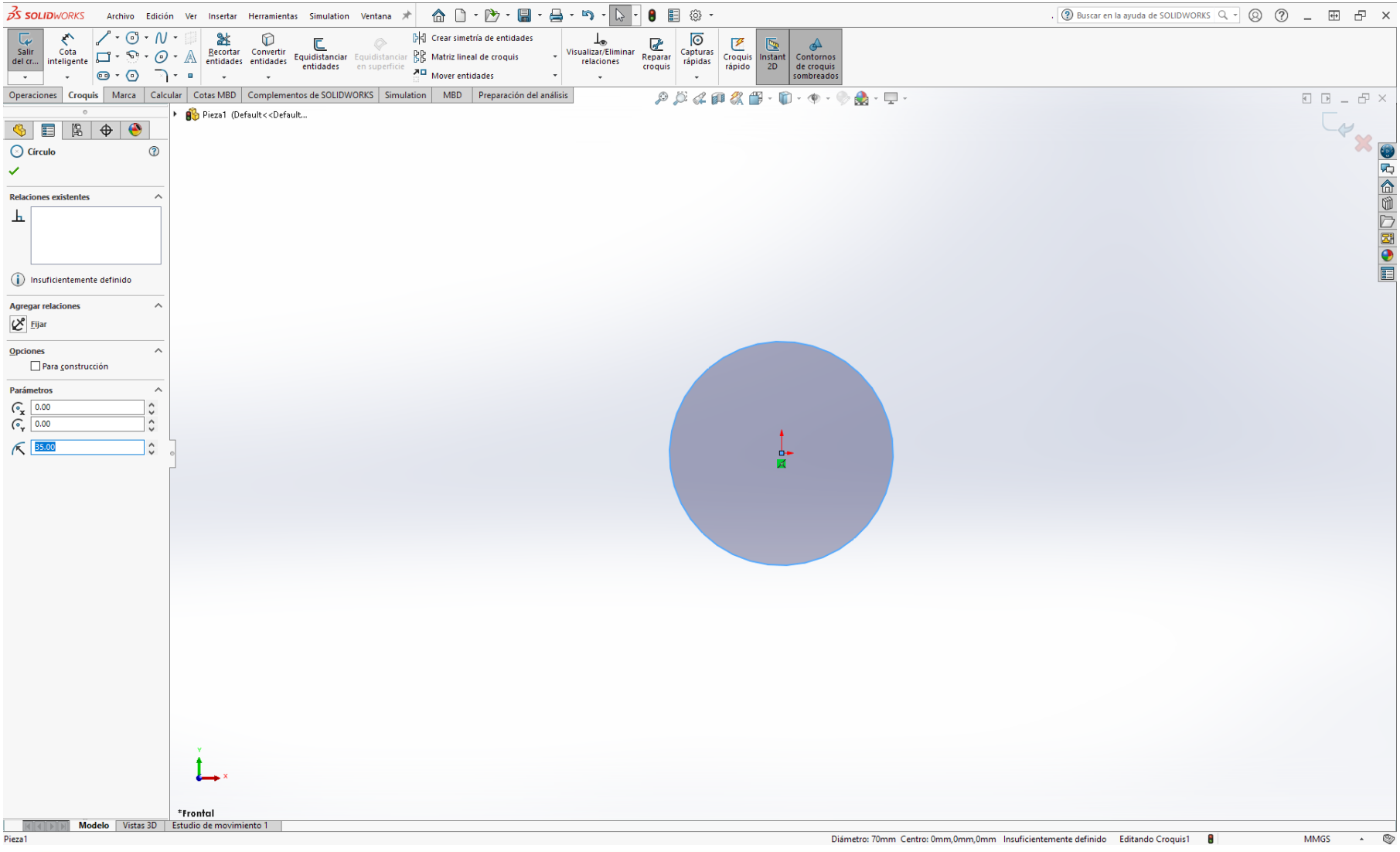




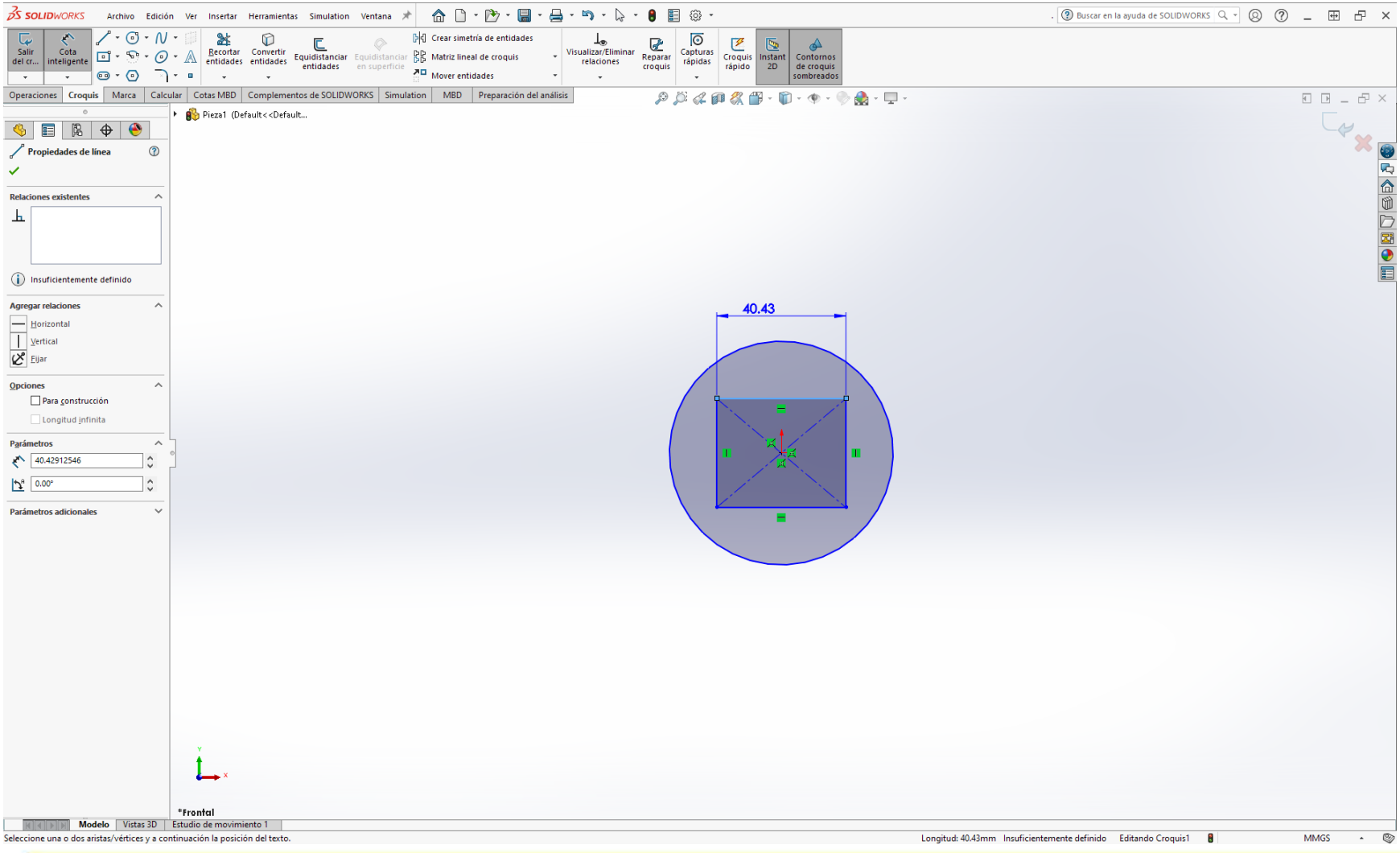
SolidWorks Exercise



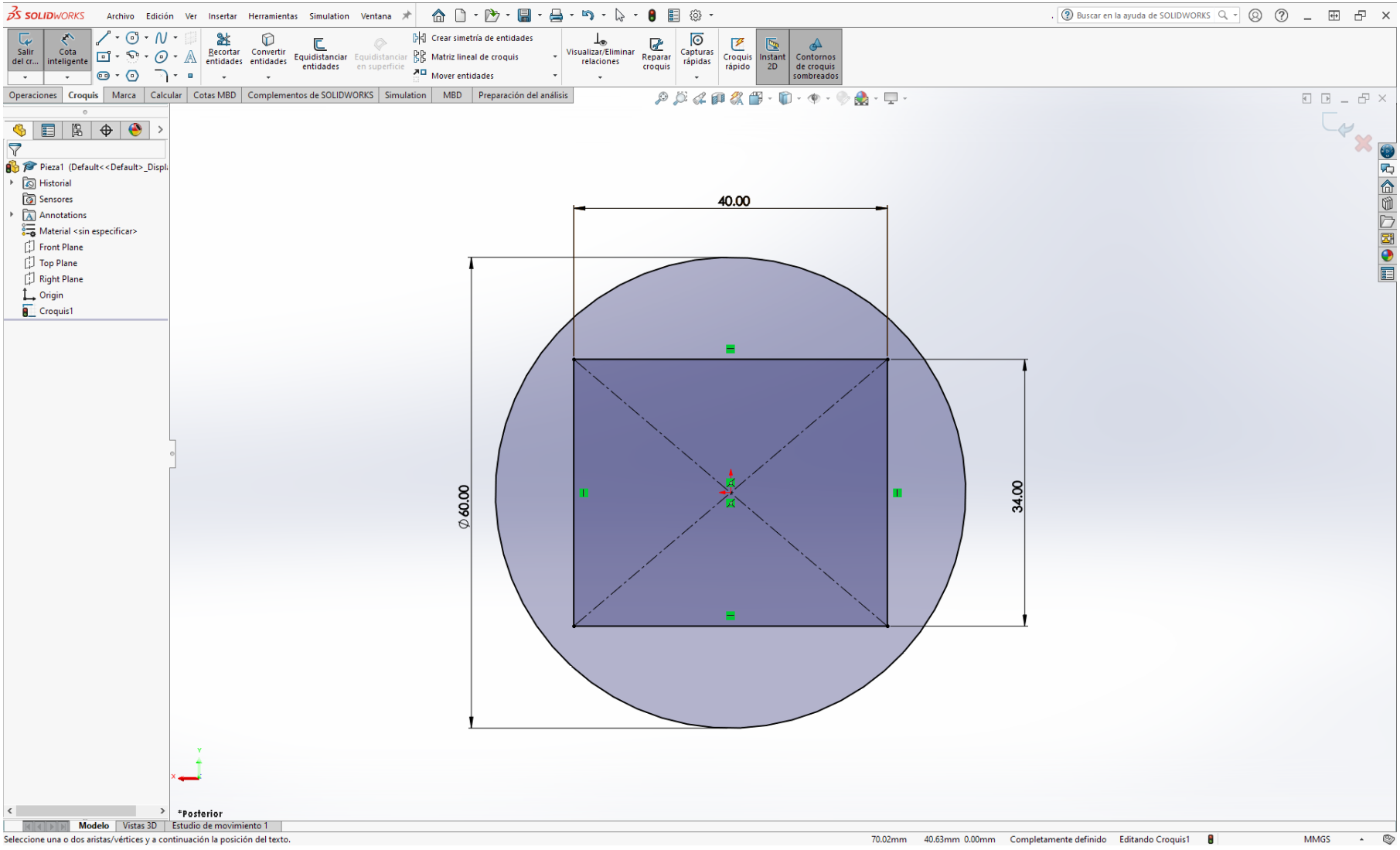
SolidWorks Exercise



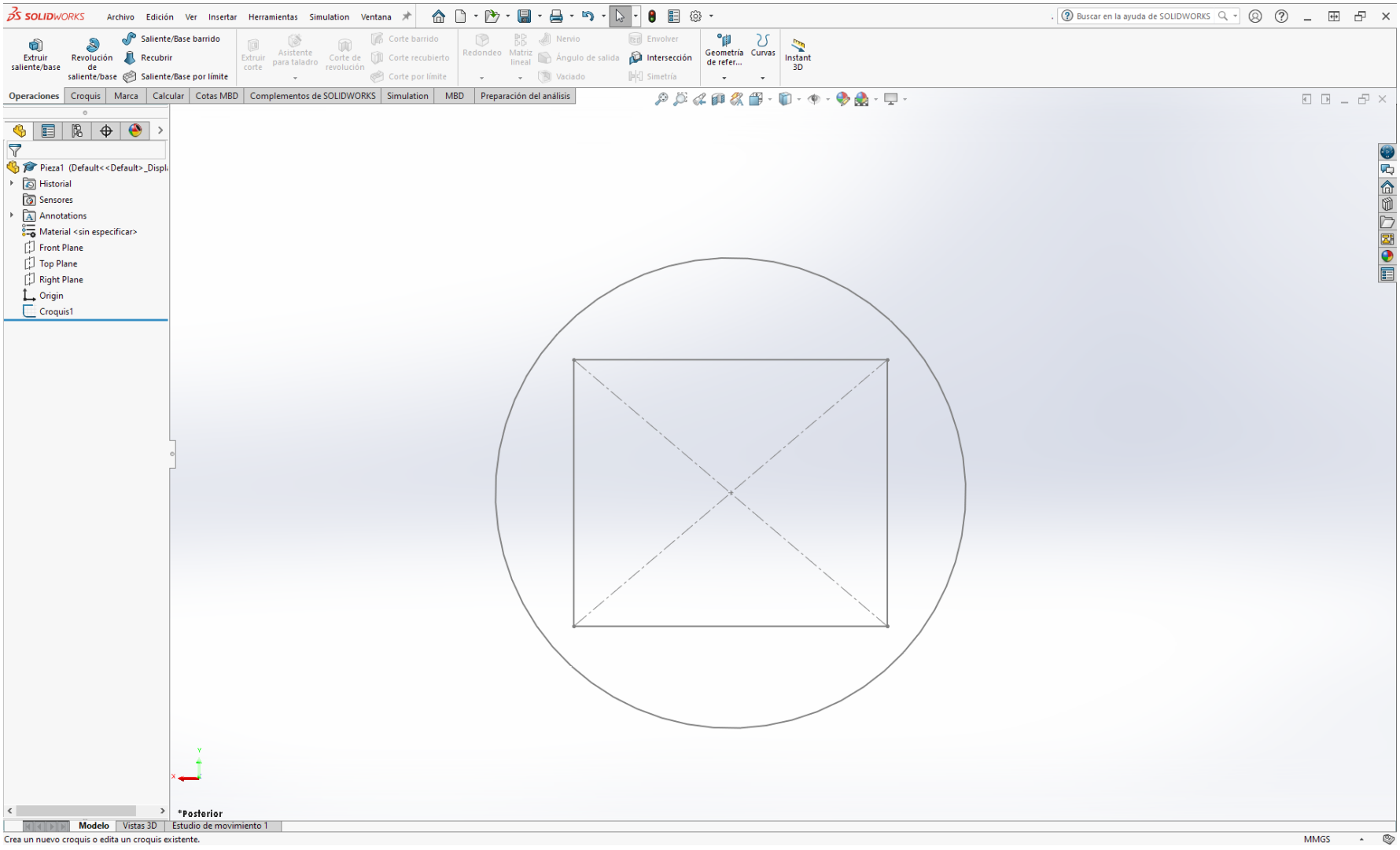
SolidWorks Exercise



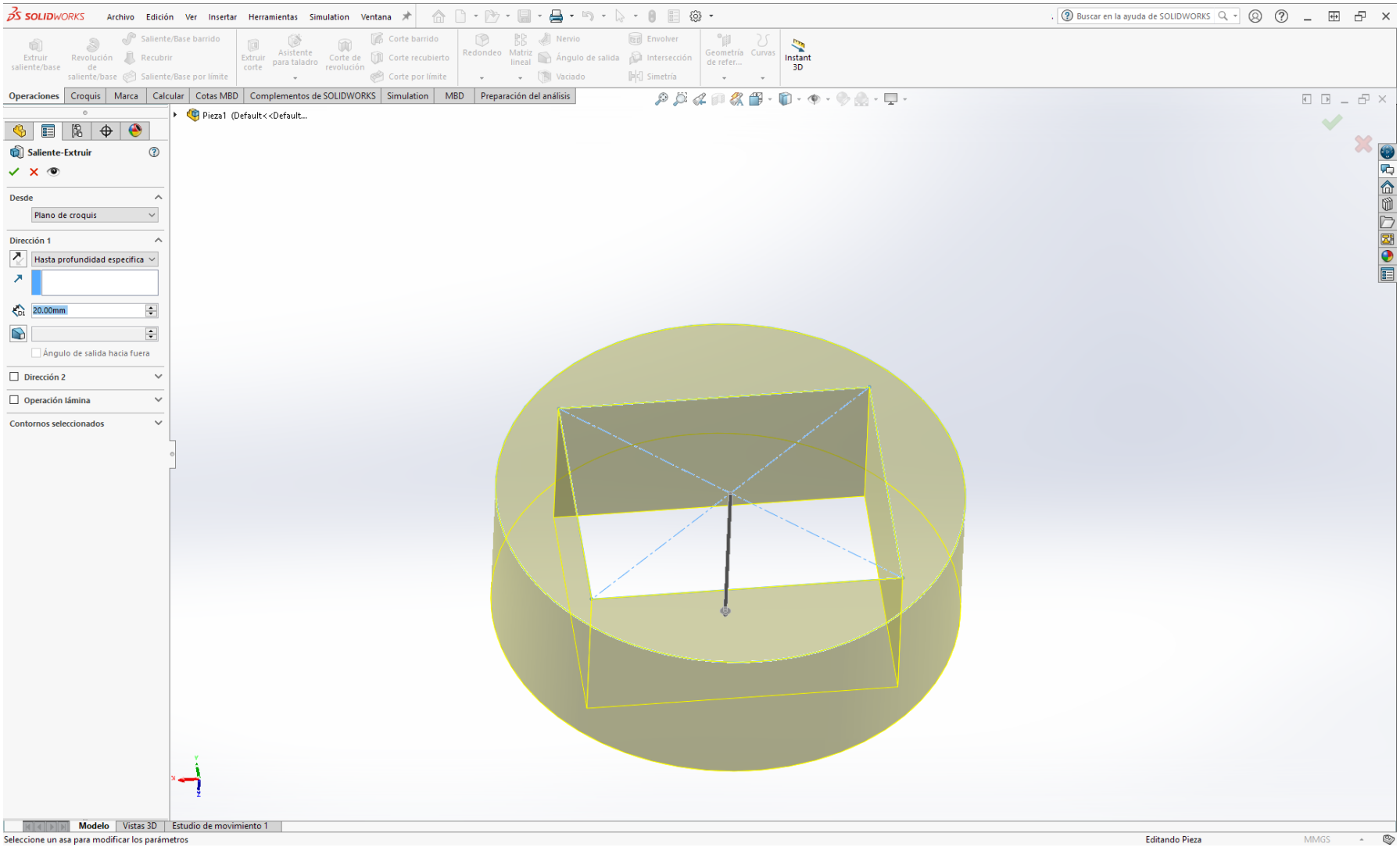
SolidWorks Exercise



SolidWorks Exercise



SolidWorks Exercise

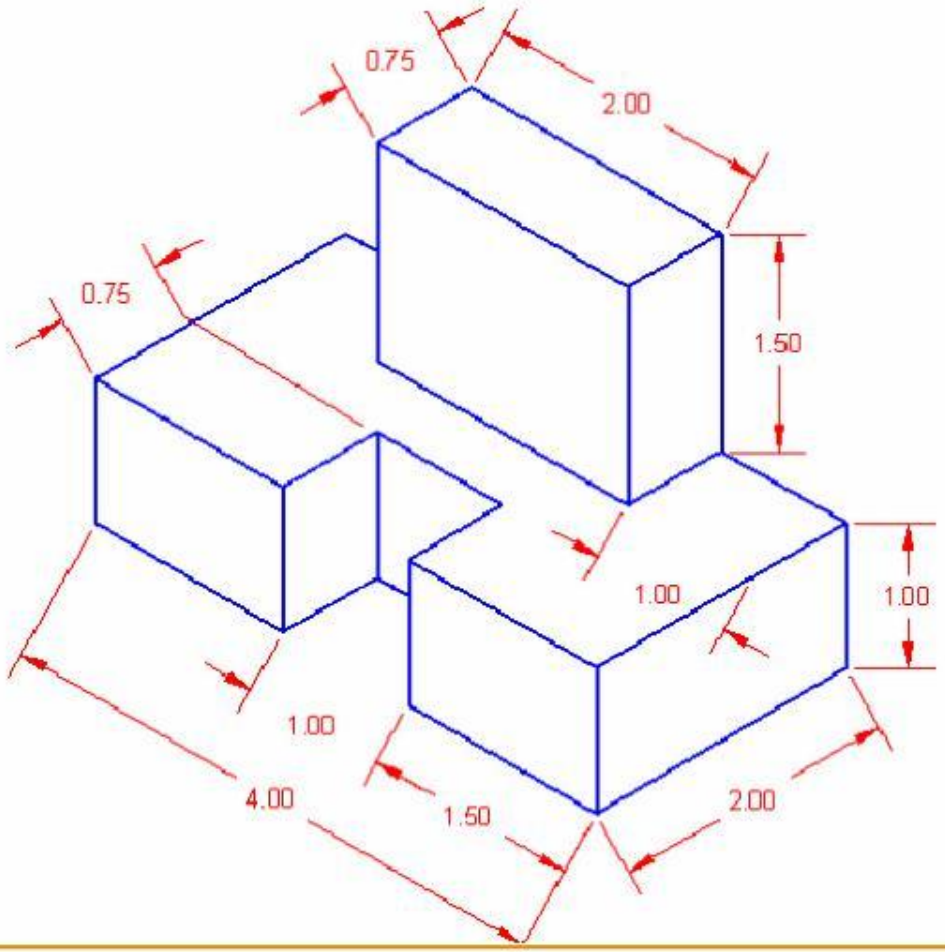


SolidWorks Exercise

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

SolidWorks Exercise 2 – Old version

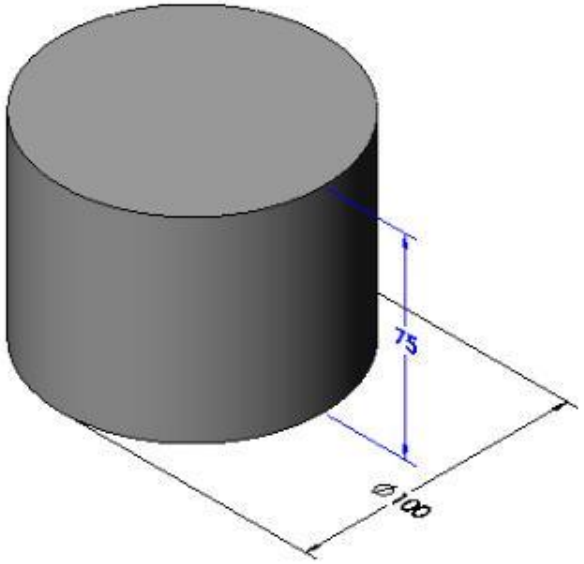
We wish to model the following part:

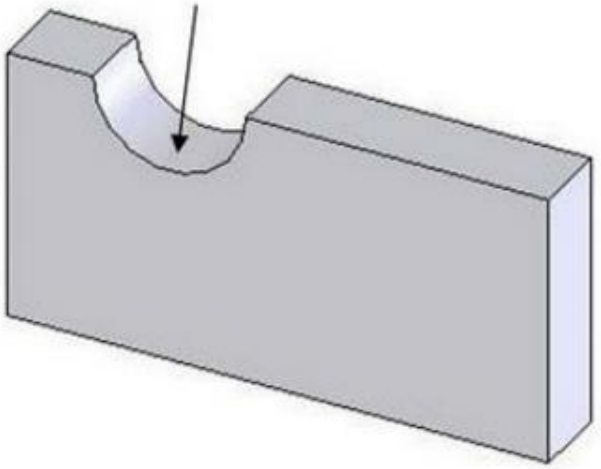


SolidWorks Exercise – What is Extrude?

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

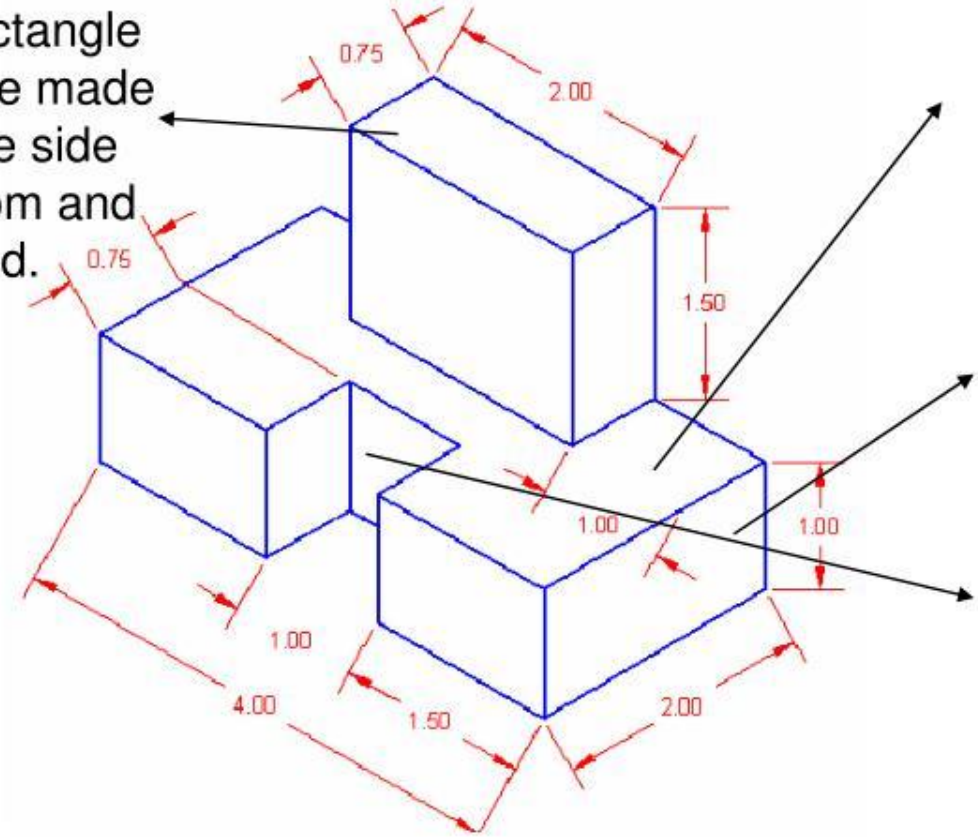




SolidWorks Exercise

There are MANY ways to EXTRUDE a surface

This rectangle could be made from the side or bottom and extruded.



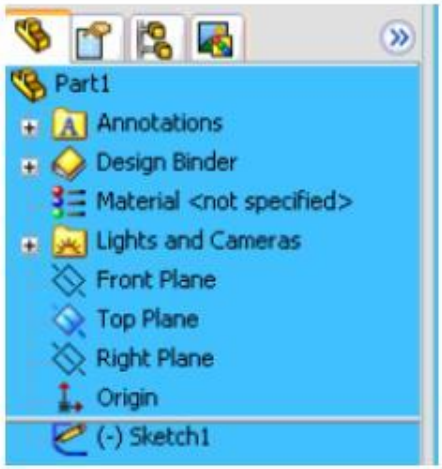
We could make this rectangle and EXTRUDE it DOWN or UP

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

A rectangle could be made here on top of the first surface, then extruded down to make the cut.

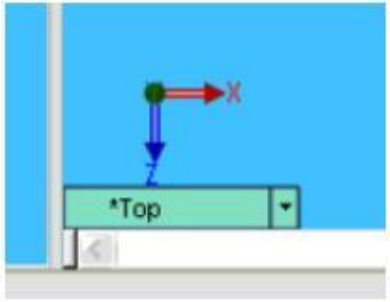
SolidWorks Exercise

Go ahead and select the horizontal plane shown.



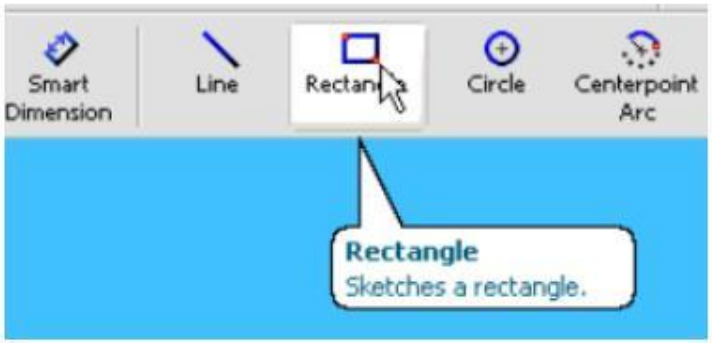
You should notice several things happen.

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.

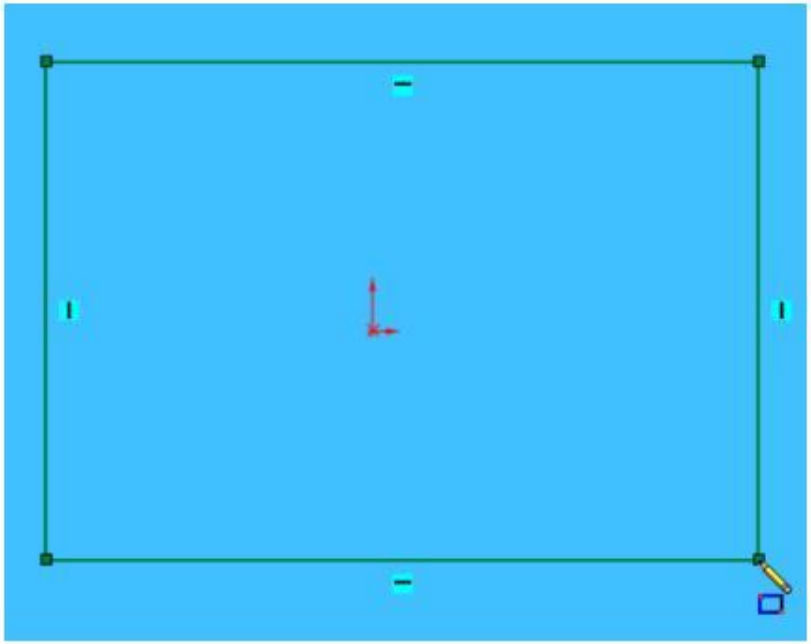


SolidWorks Exercise

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



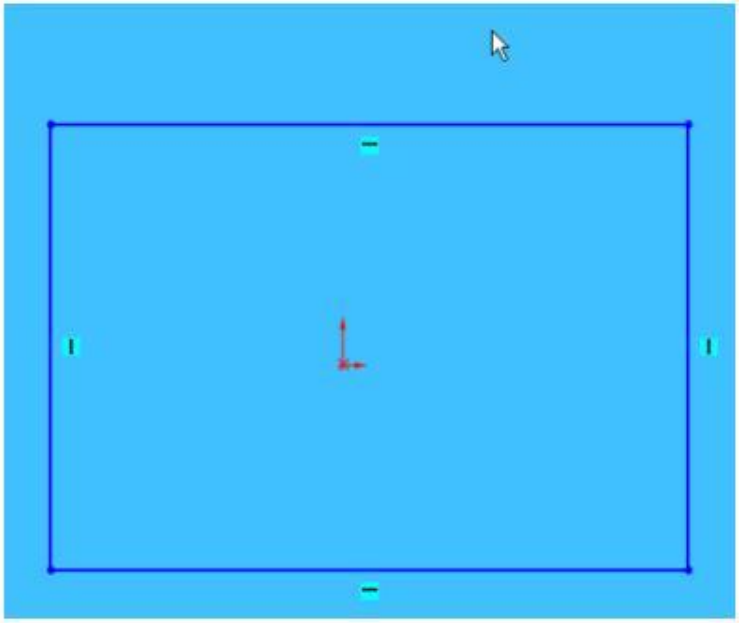
Make a rectangle by dragging the mouse from one corner of the screen to the other. The size does not matter at this point.



SolidWorks Exercise

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

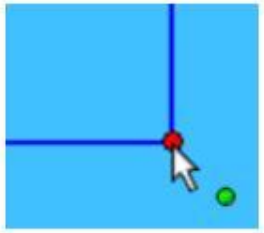
We need DIMENSIONS to our rectangle however.



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.

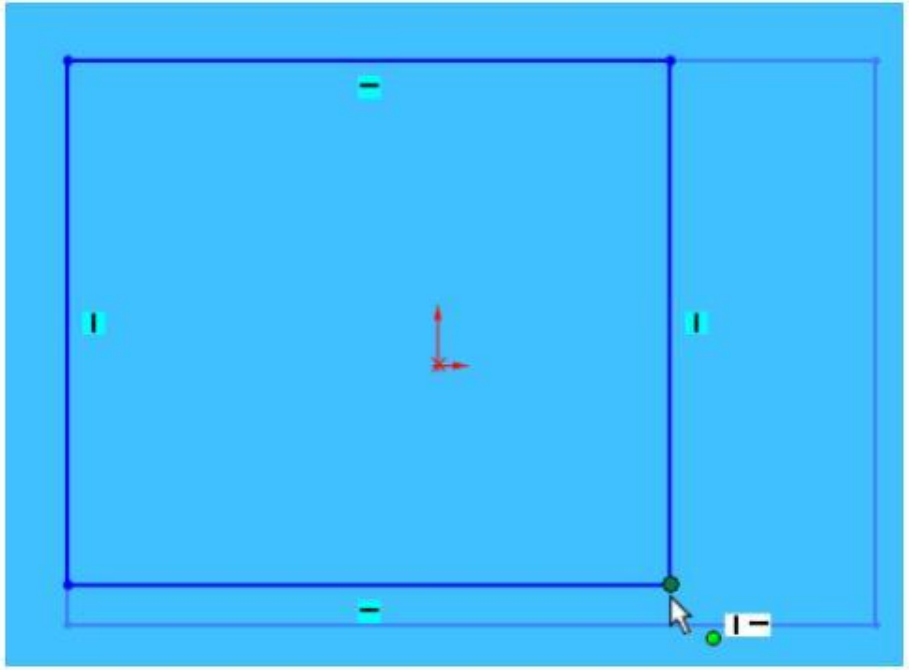
SolidWorks Exercise

Click the bottom right point on the rectangle.



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

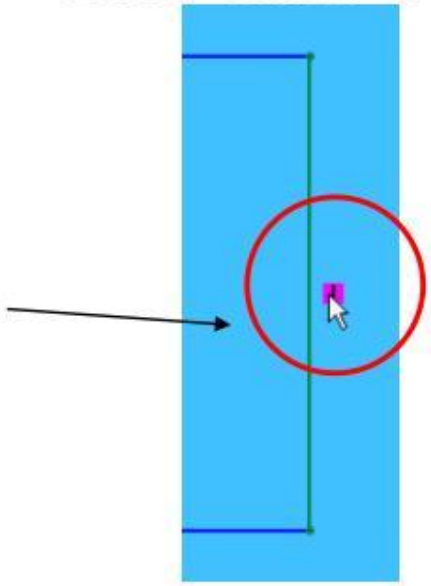
This point has degrees of freedom and is NOT constrained.



SolidWorks Exercise

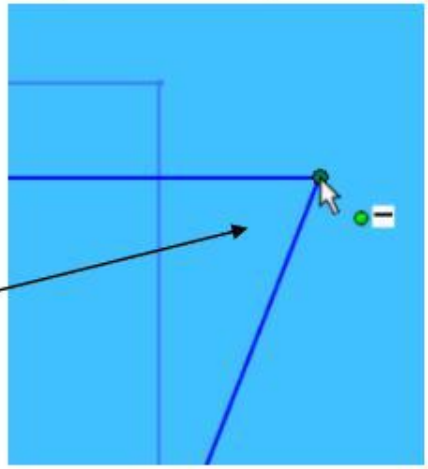
Let's save our work!

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.

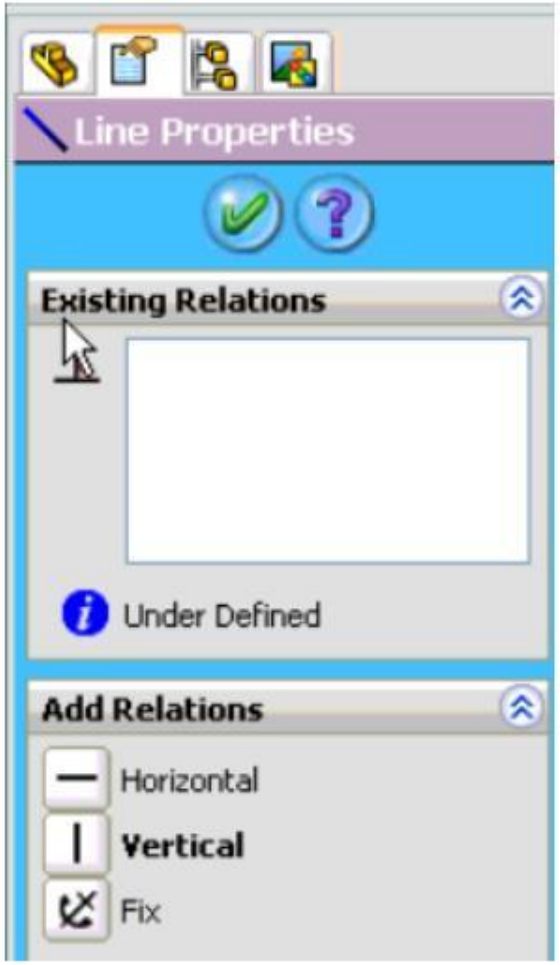
The horizontal line still moves up or down, but notice the vertical line can be moved in any direction as you deleted the relationship.

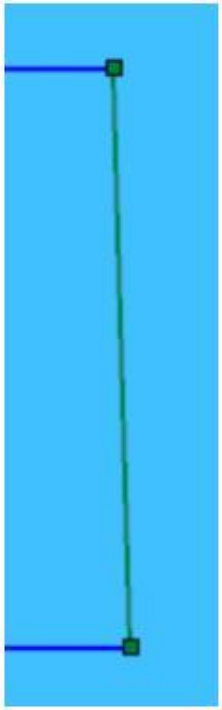


SolidWorks Exercise

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

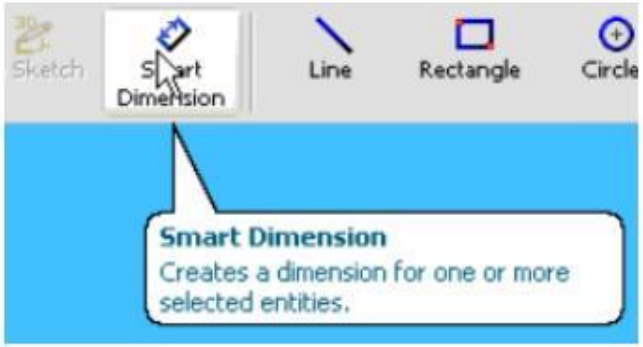
Line properties will appear and you can click on the VERTICAL button on the ADD RELATIONS windows to constrain the line to just the vertical direction.

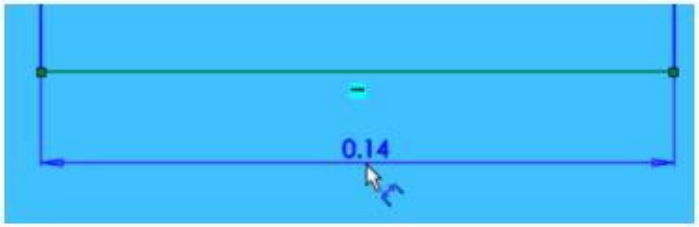




SolidWorks Exercise

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



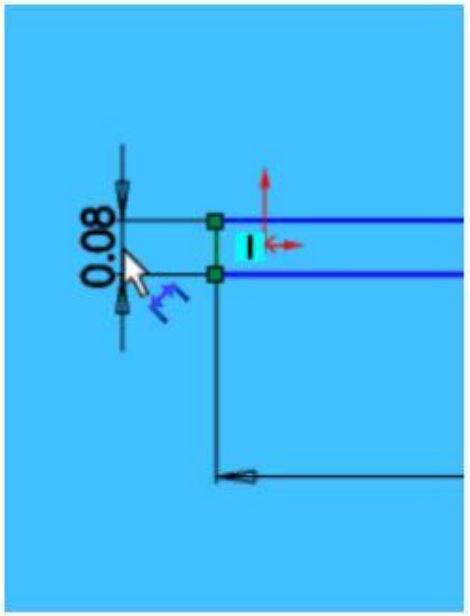


Click the bottom horizontal line of our rectangle and drag the dimension down.

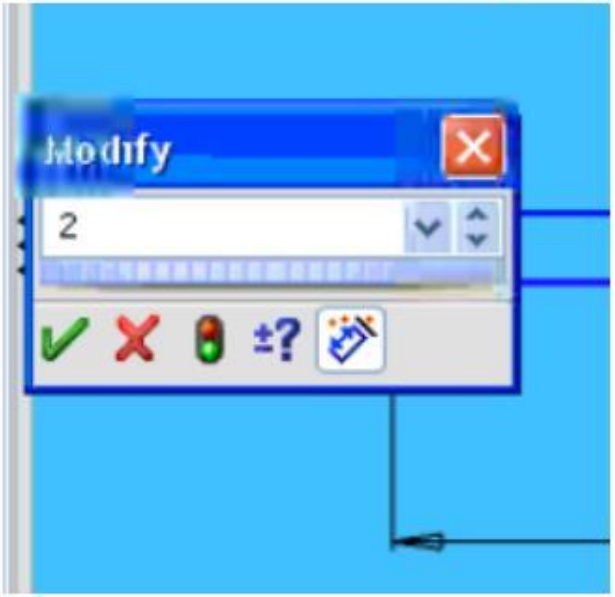
You can change the length of the line by using the slider bar or you can simply enter in 4 m in the box shown. Enter the value 4 then click the green check. Since the box is way too big, press “f” on the keyboard to fit. Then drag down the dimension so that you can see it after hitting escape.

SolidWorks Exercise

Now using Smart Dimension again lets dimension the left vertical line. You may need to zoom or pan to view the line to click on.

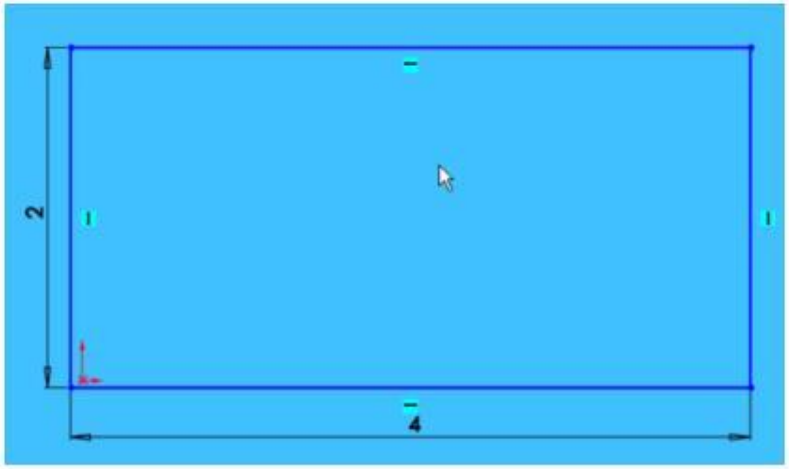


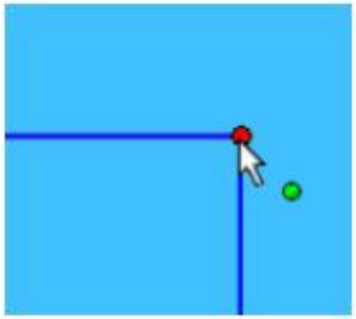
Modify the Length to be 2 units as shown in the original drawing.



SolidWorks Exercise

Hit Escape, then “f” to fit.
You should see a 2 x 4 rectangle.

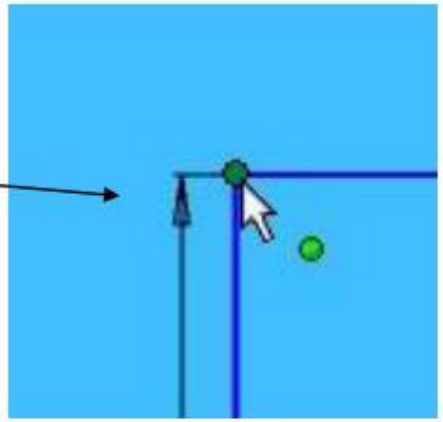




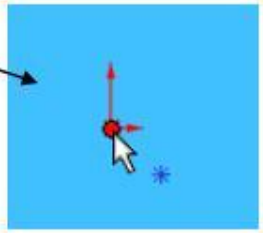
If you click the top right corner, you see you can move the entire rectangle. The “entire” drawing has yet to be defined in terms of its location relative to the plane,

SolidWorks Exercise

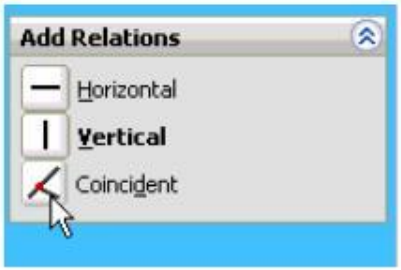
To make the drawing fixed in the plane we can make one point coincident with the origin. Choose the top left point.



Press CTRL, then select the origin.

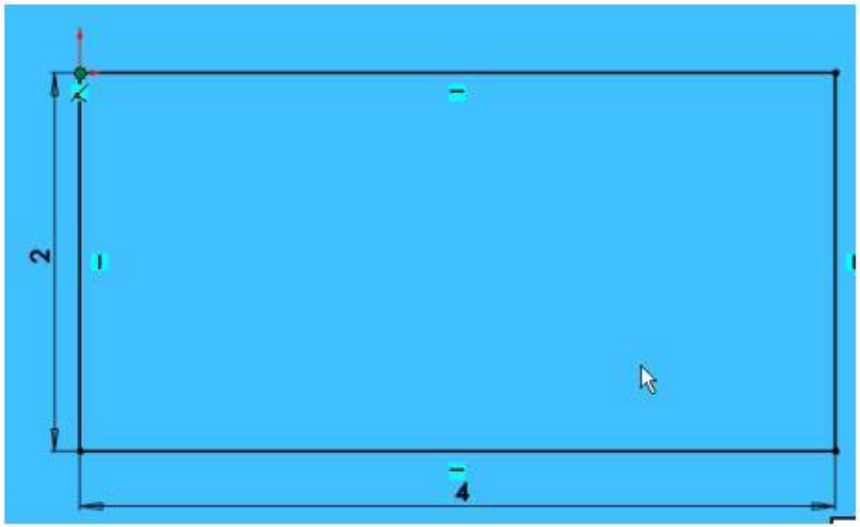


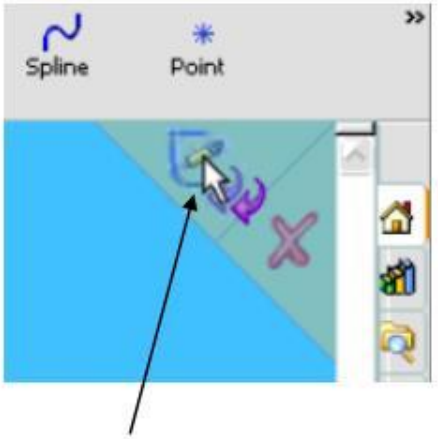
You will notice that under selected entries you have TWO items chosen. Under ADD RELATIONS you see "Coincident". Choose this button and you will see the origin move to that point.



SolidWorks Exercise

Choose ok (green check). You should see all of the lines turn black.

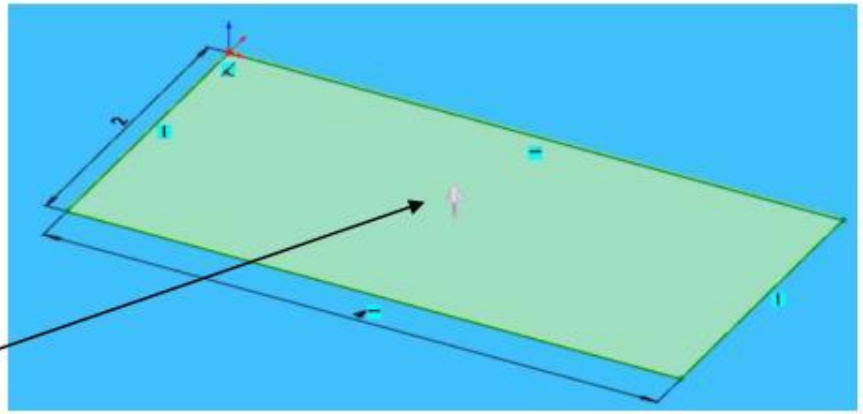


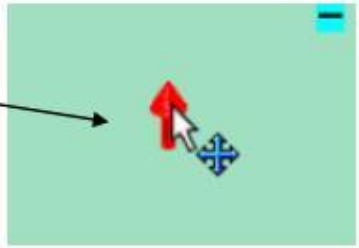


Click in the top right corner to EXIT the SKETCH.

SolidWorks Exercise

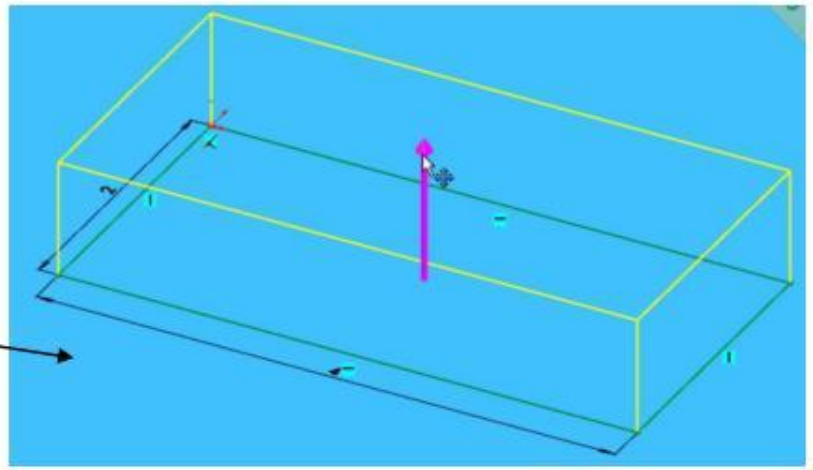
When you exit the sketch you should see your dimensions and sketch in the plan you choose. An arrow will be located in the middle of your sketch. Click on the arrow and move upward.



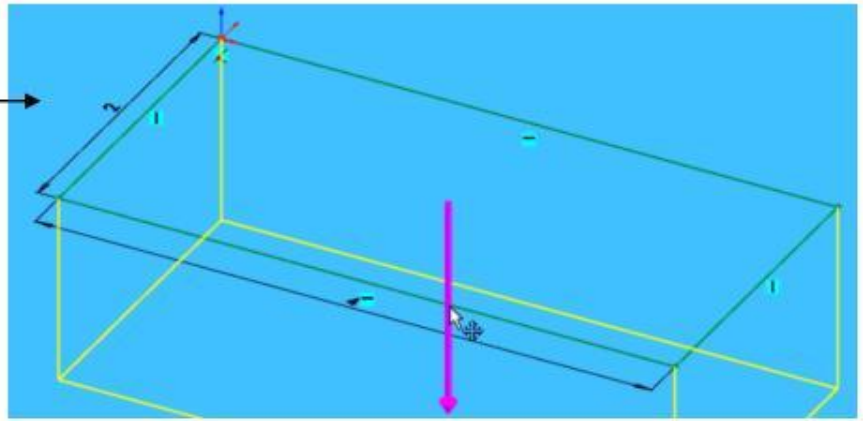


SolidWorks Exercise

As you drag it upward you should see the volume like structure shown.

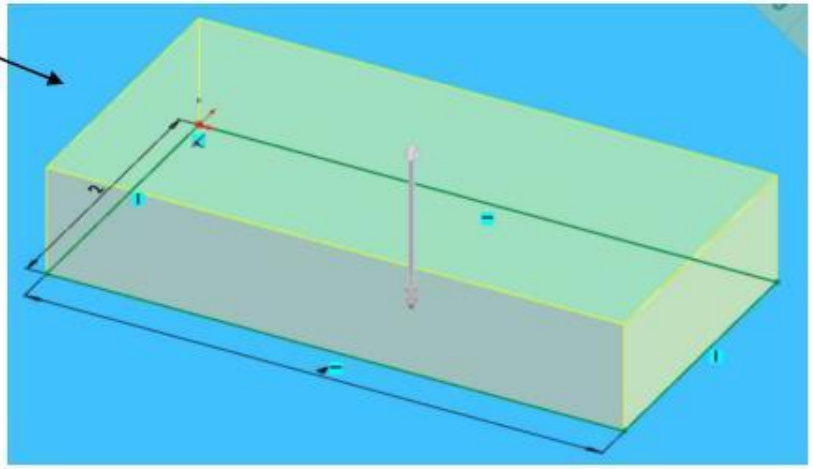


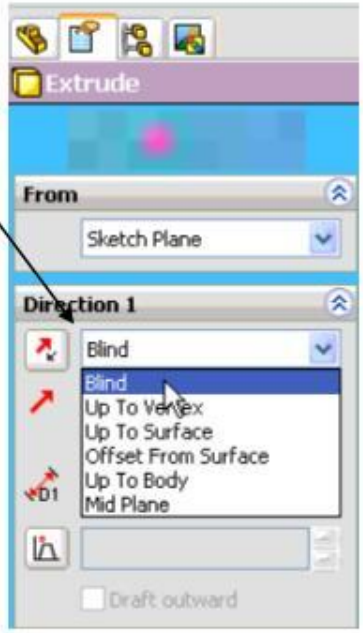
You can also EXTRUDE the shape down.



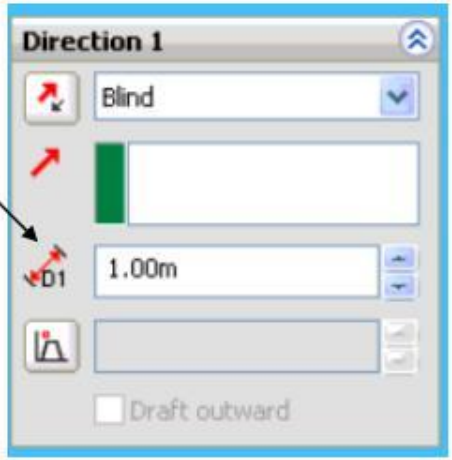
SolidWorks Exercise

Drag the surface upward and release to select a volume. Under the properties manager we see several DIRECTIONS we can take this surface. Just choose BLIND.



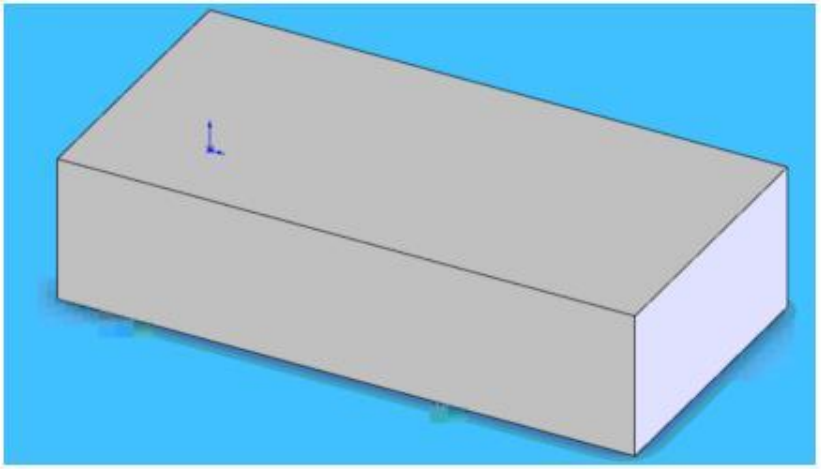


The “D1” you see means depth and we see from our original picture we want a depth of 1. So enter 1, then click OK.

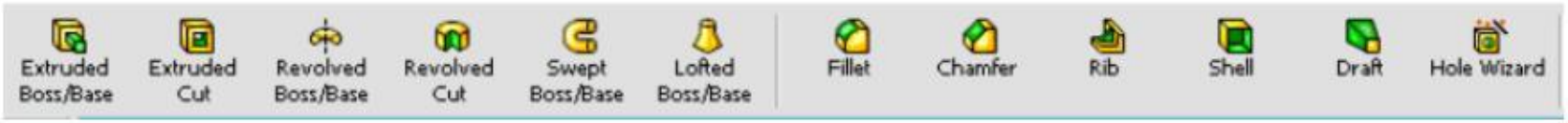


SolidWorks Exercise

We now have our first SOLID piece.

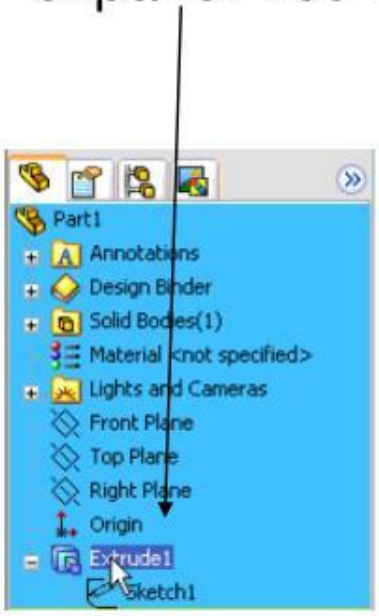


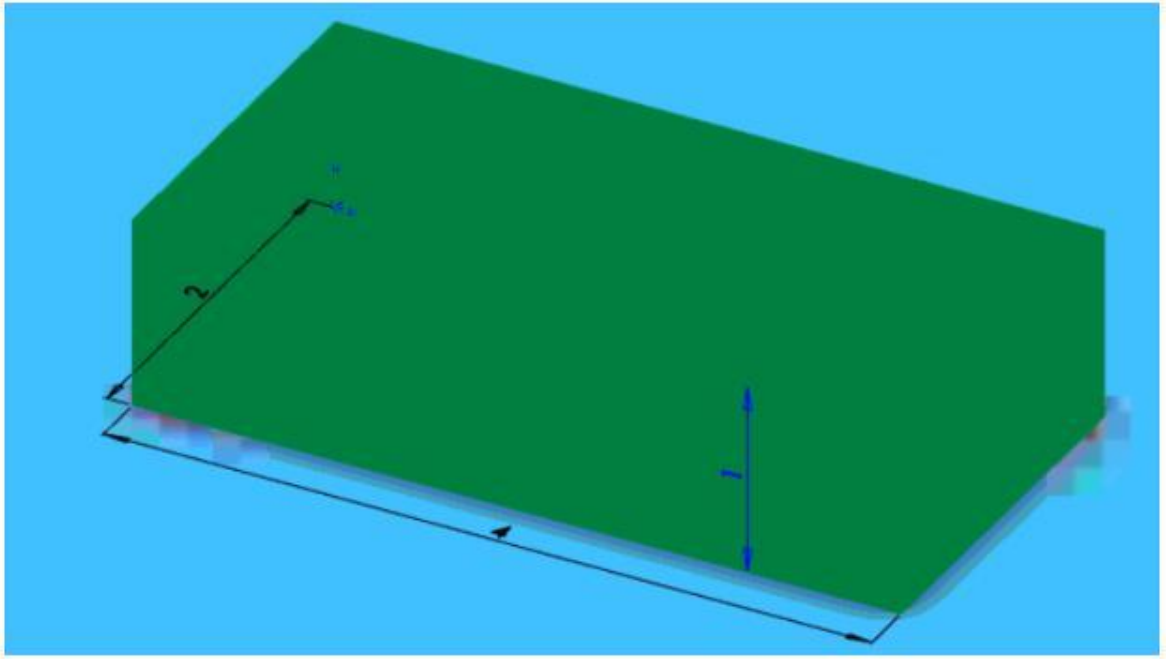
You should also notice we have a ton of new features available as well.



SolidWorks Exercise

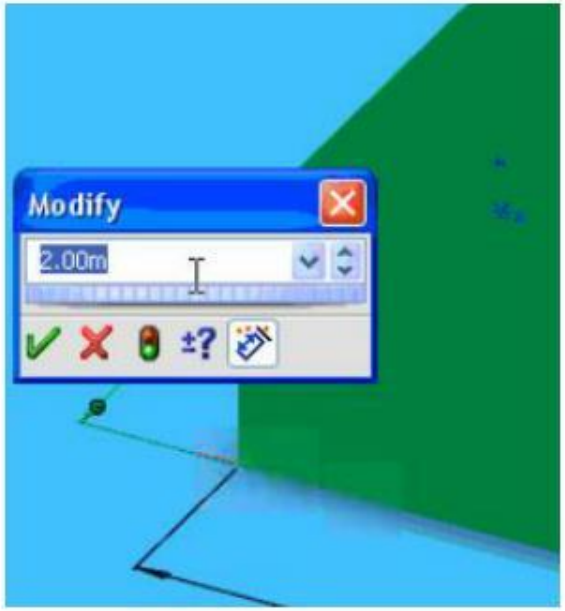
Solidworks is a PARAMETRIC design tool, meaning the edges are always driven by dimensions. So it is very important that your drawing be fully defined. Notice under features manager EXTRUDE now appears on the list. Double click on this or expand. You now see the design in terms of its dimensions.

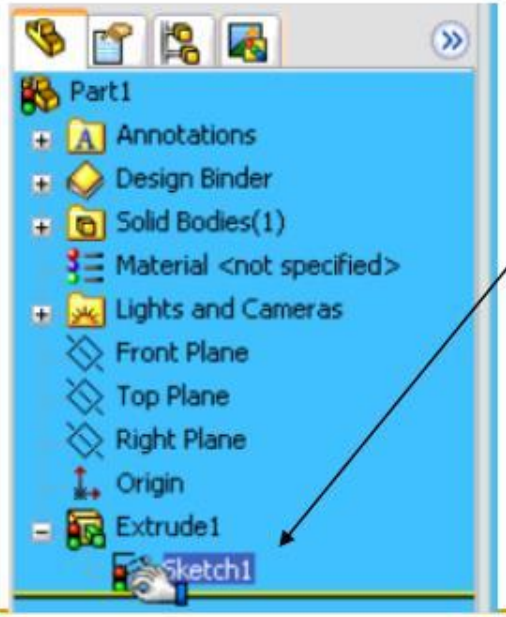




SolidWorks Exercise

Click on the width dimension and change the value from 2 to 3. Then click OK. Then click exit in the properties manager window. What happened?





Notice the STOPLIGHT icon next to our sketch under features manager. What this means is that we have CHANGED the design but we have yet to REGENERATE the part.

SolidWorks Exercise

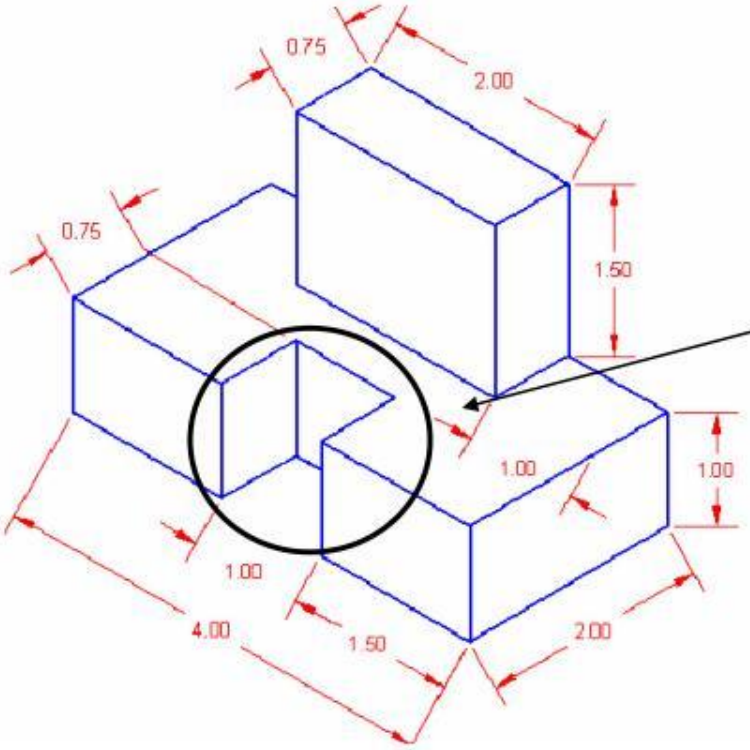
Up on the dynamic toolbar you will see another “stoplight” icon, but this time it is a button. If you click this button it will REBUILD your part according to the changes you made.

Notice the changes in your part.

Go ahead and make changes to the other dimensions and rebuild to see the effect.

THEN CHANGE THE DIMENSIONS BACK TO THE ORIGINAL DIMENSIONS of 4x2x1.

SolidWorks Exercise

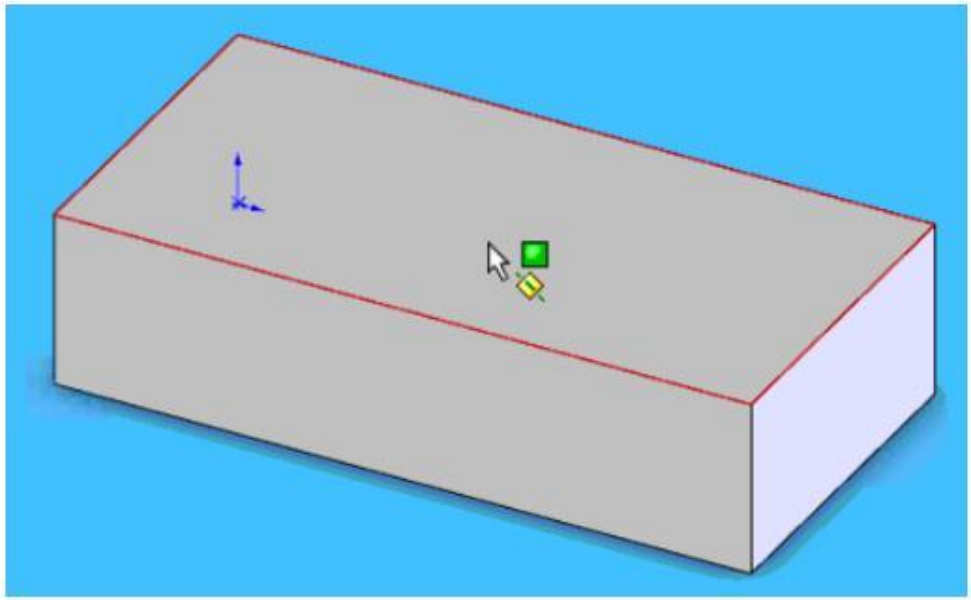


Lets go ahead and make this cut!

To do this, we need to use what is called an EXTRUDED CUT.

SolidWorks Exercise

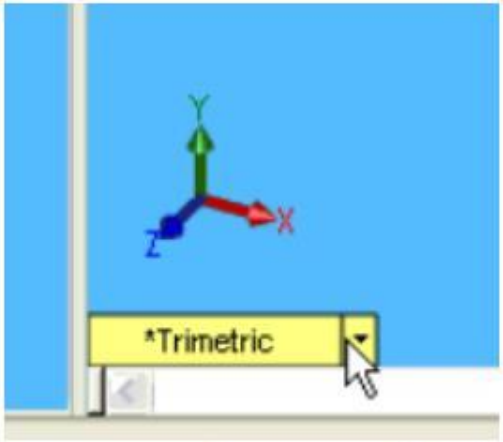
Click extruded cut and notice what it is asking us in the properties manager tab.



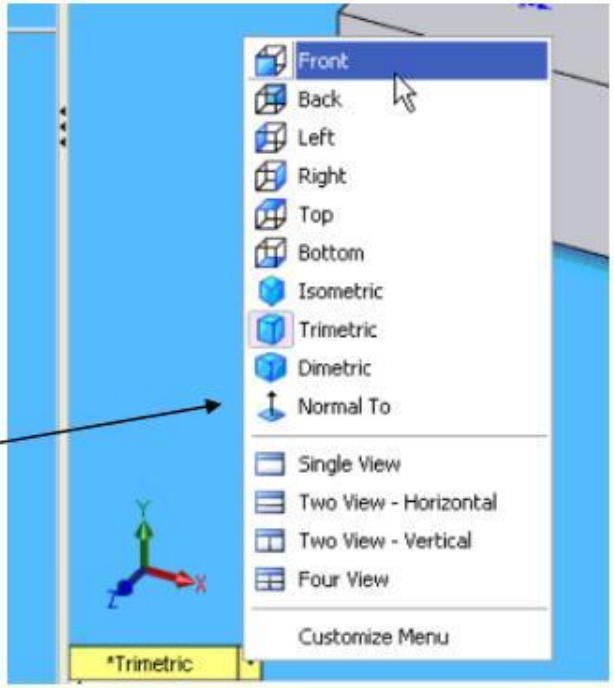
We want to select planar face which is the top surface as shown.

SolidWorks Exercise

Click on that top surface. **What happened?** We are sketching on top of the plane but unfortunately we are not LOOKING at the plane itself. At the bottom left you see the VIEW selection. Trimetric is chosen now, but if you click the arrow beside it other options become available.

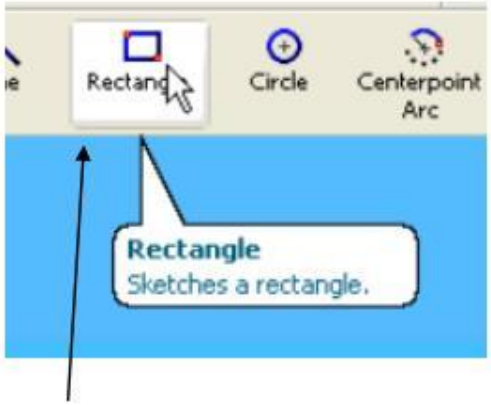


Click the **NORMAL TO** view.

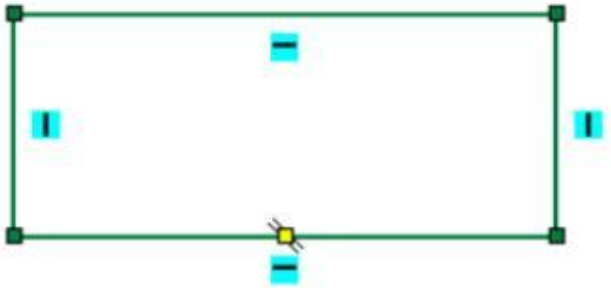


SolidWorks Exercise

You should see now that you are looking directly DOWN at the plane.



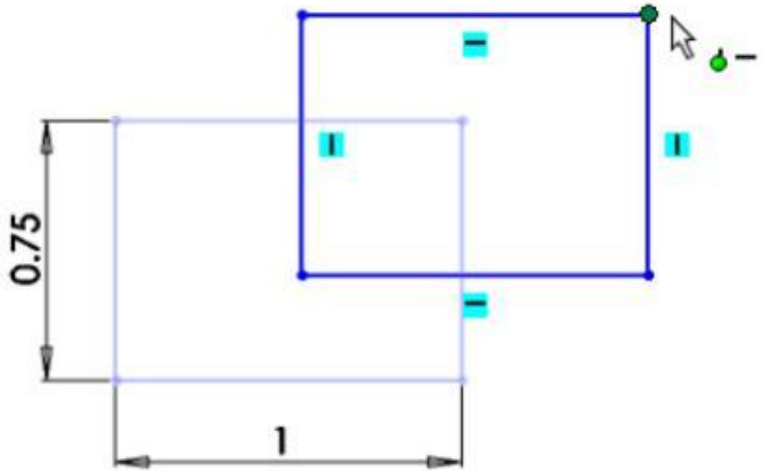
Click the rectangle button at the top and draw a rectangle of ANY SIZE within the surface of your plane.



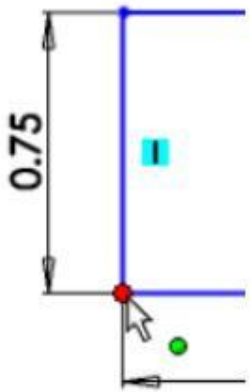
SolidWorks Exercise

If you hit escape, you see that the rectangle is certainly UNDER DEFINED just as our last one was. So lets go ahead an DIMENSION the rectangle using our given figure.

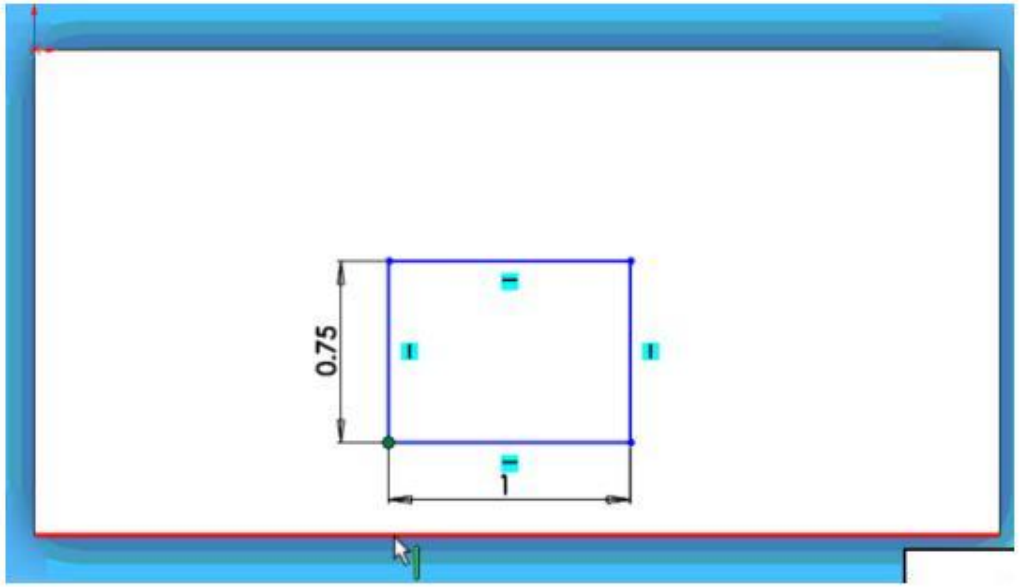
When you are finished you basically have a rectangle than you can freely move within the planar surface.



SolidWorks Exercise

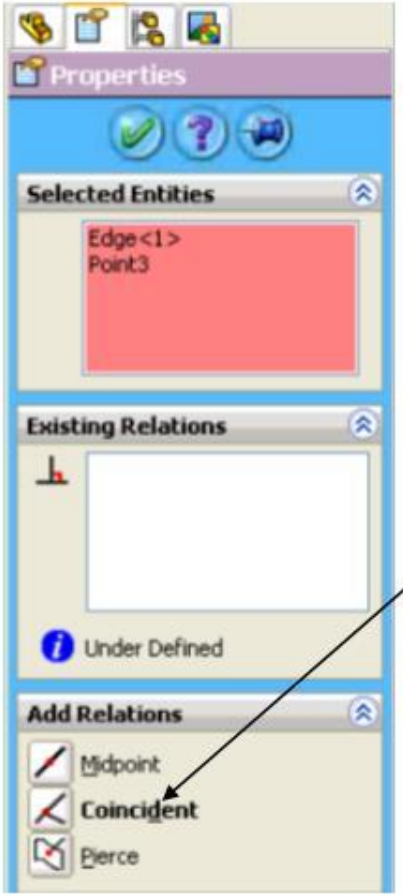


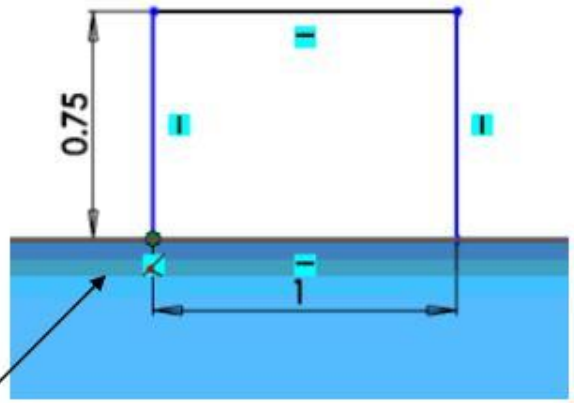
Lets define the location of the rectangle. Click on the bottom left point and press the CTRL button to select MULTIPLE entries. Click on the bottom horizontal line.



SolidWorks Exercise

Under properties manager notice the entries selected in pink. Choose coincident under ADD RELATIONS.

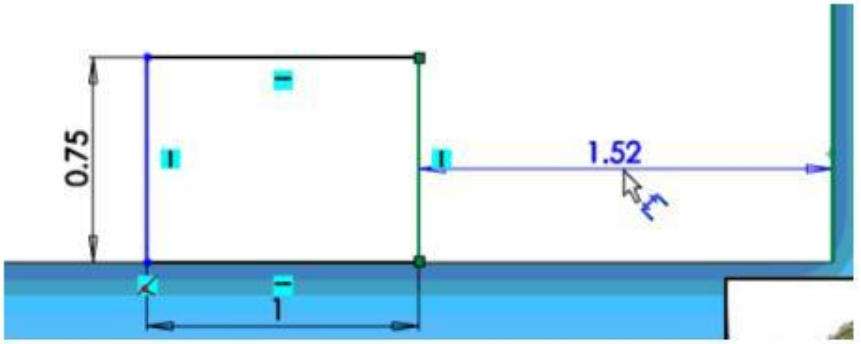




Notice your point is now coincident with the bottom surface. Click OK in the properties manager window. Notice that your horizontal lines are black meaning they are fully defined. Try dragging the box

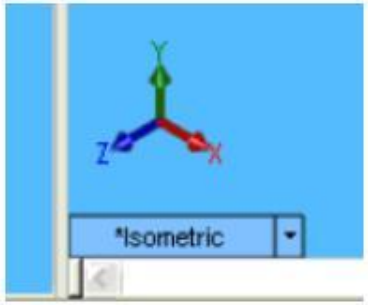
SolidWorks Exercise

Clink on smart dimension and choose the right vertical line of our smaller rectangle then click on the vertical line of our planar surface edge. Then modify the distance to 1.50

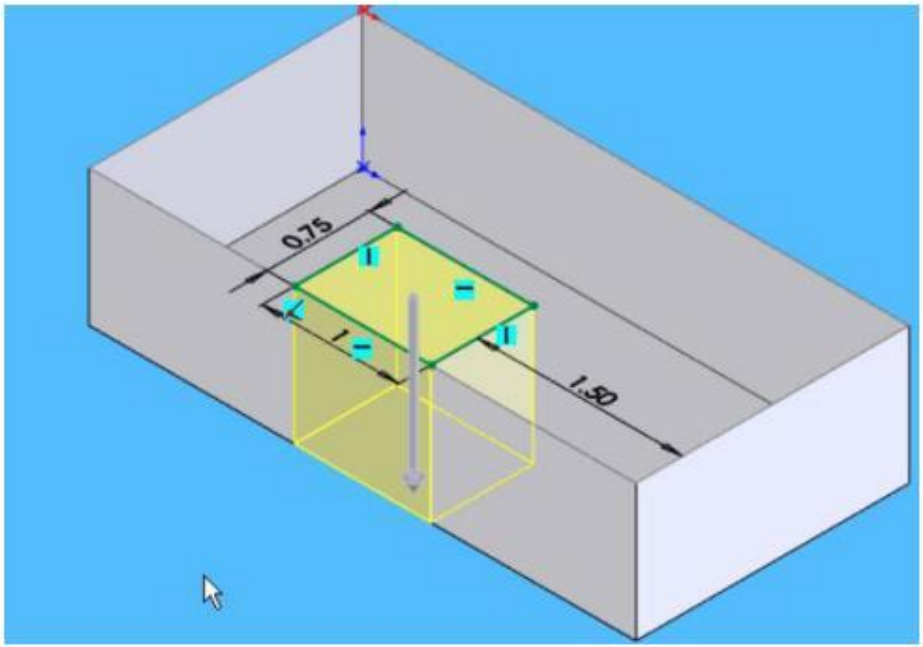


Notice we see FULLY DEFINED at the bottom of our screen and that all of our lines are black meaning NO freedom of movement.

SolidWorks Exercise

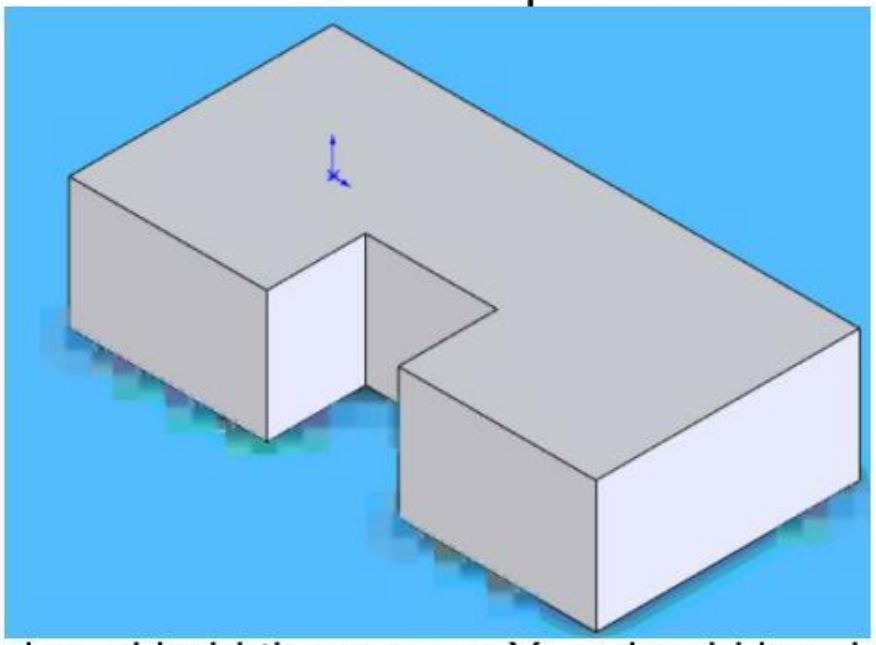


Click OK in properties manager then EXIT sketch in the top right. The view will not be seen from the point of view we can extrude so choose a different view from below. Choose ISOMETRIC.



SolidWorks Exercise

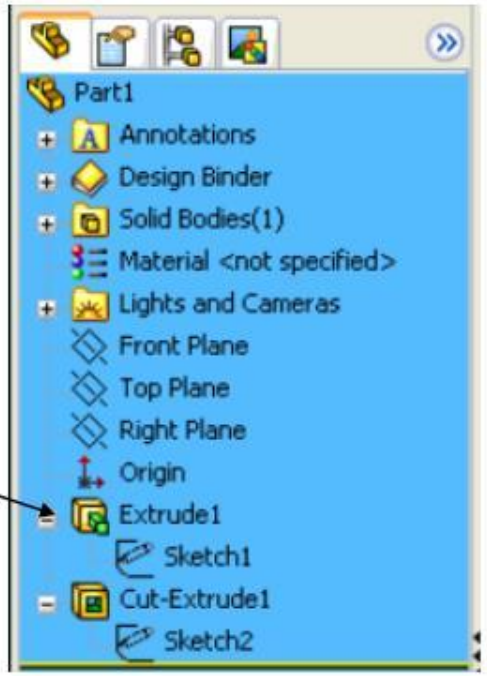
From the property manager window under Cut-Extrude we can choose a direction. Choose THROUGH ALL as we want our cut to go all the way through. Then click OK. You should see the complete cut.



Click and hold the mouse. You should be able to ROTATE the part in 3D space.

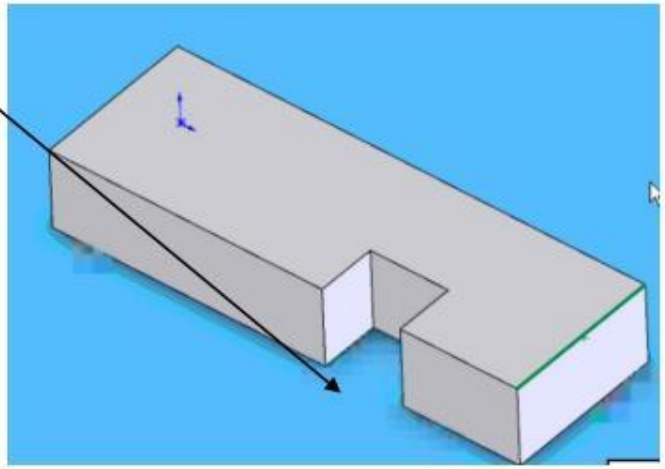
SolidWorks Exercise

Suppose we change the ORIGINAL dimension of the planar surface length from 4m to 6m.In feature manager, click on the original extrude and then modify the length dimension to 6m. Then regenerate the part. You should see that the CUT is fixed at 1.5 m from the right edge.



So the big question is ….**WAS THIS DESIGN INTENT?**

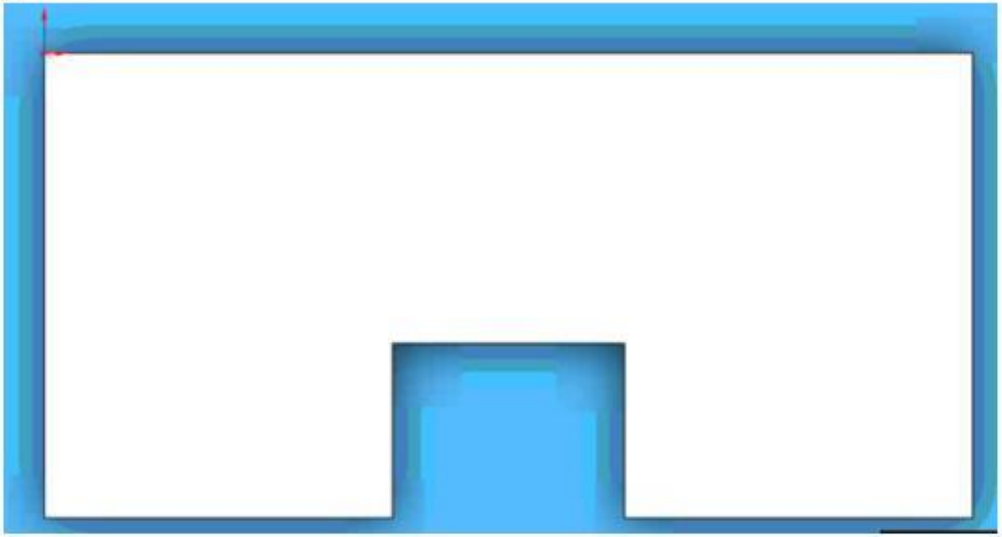
Did we really want this 1.5 m from the right or did we want the cut to be CENTERED?



SolidWorks Exercise

Go ahead and change the length dimension back. In this case, the designer had the cut centered in the original drawing. Just keep DESIGN INTENT in mind when you are trying to decide HOW to model a part.

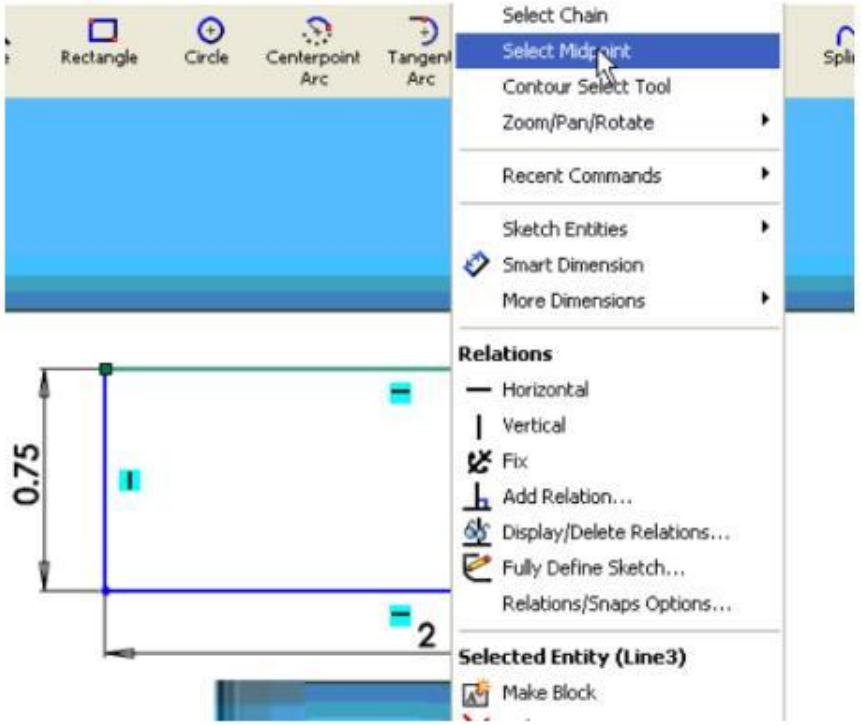
Choose Extruded Boss/Base then choose the top planar surface. Then change your view to NORMAL TO so that you see the top planar surface as shown.



SolidWorks Exercise

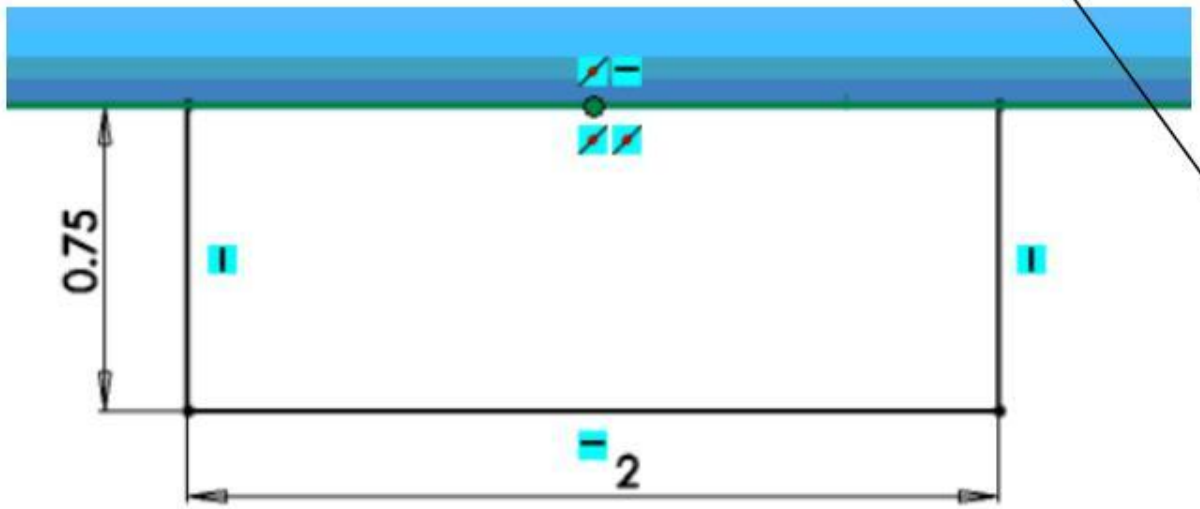
Now lets finish the top part.

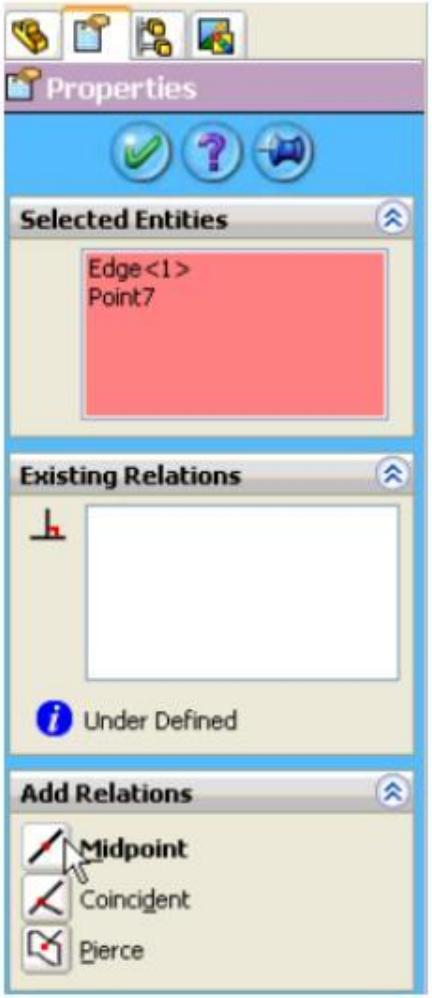
Choose rectangle, sketch it and dimension it according to the original drawing. But now lets try to center the sketch as we know it was the designer's intent to do this. RIGHT CLICK on the top horizontal line of your rectangle and a window will open. Choose SELECT MIDPOINT.



SolidWorks Exercise

You should notice a point on your line. Press CTRL then CLICK on the top horizontal line of your planar surface. You should see them selected in properties manager in pink. Choose MIDPOINT in the ADD RELATIONS window.

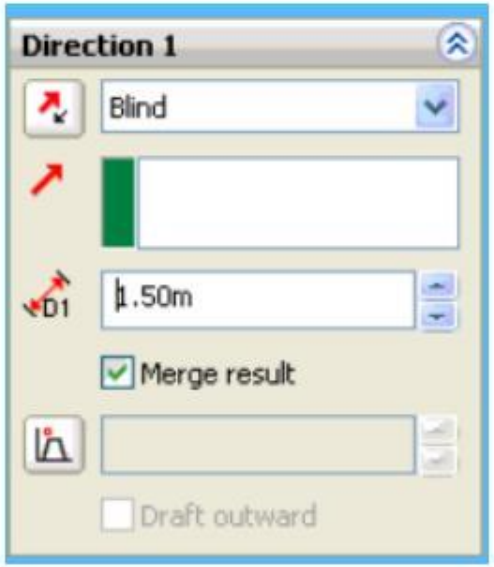


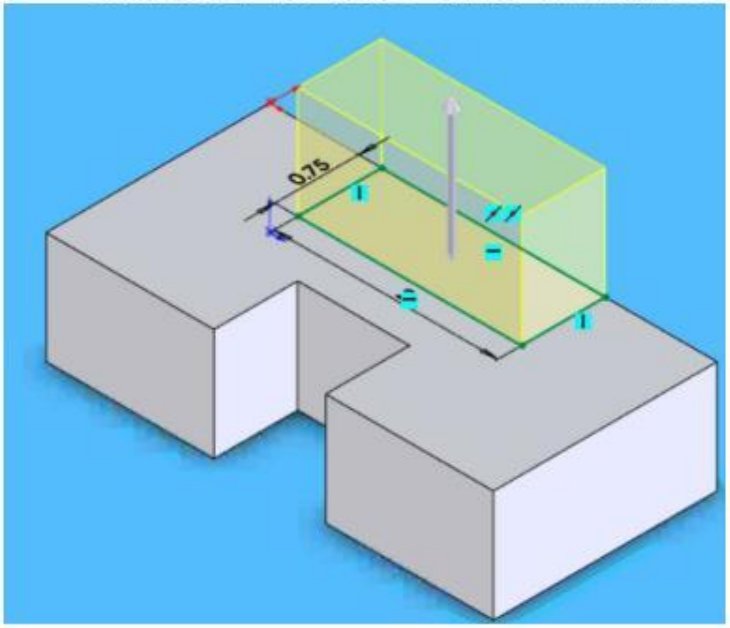


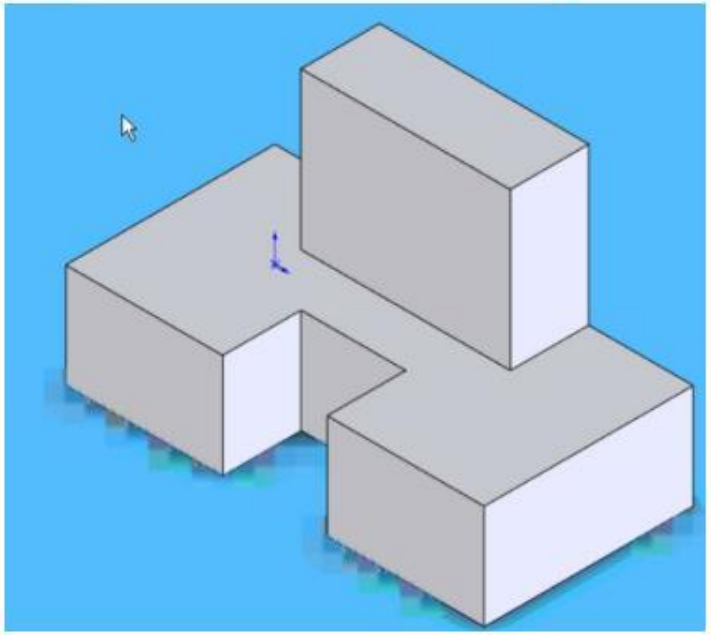
Your rectangle is now centered at the midpoint on the top edge of the planar surface.

SolidWorks Exercise

EXIT your sketch then change your view to isometric. In properties manager, use BLIND for the direction and change the height(D1) to be 1.50 according to our original drawing. Choose select to see the finished part.

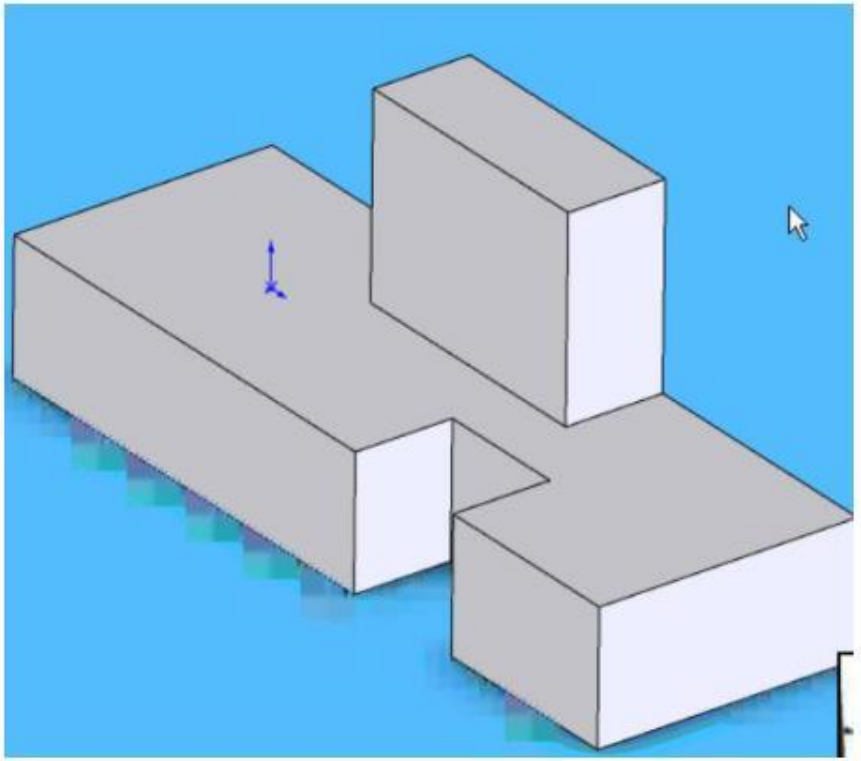






SolidWorks Exercise

To establish DESIGN INTENT, change the original length dimension from 4 to 6m. Then rebuild the part. You should see the top part remain centered and the cut 1.5 m from the right edge.



Sources

- University of Central Florida
- Modelado solidworks: https://www.instagram.com/solidwparaestudiantes/?hl=es-la