
Solidworks Lesson 6 - Assembly & Part Drawings

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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.

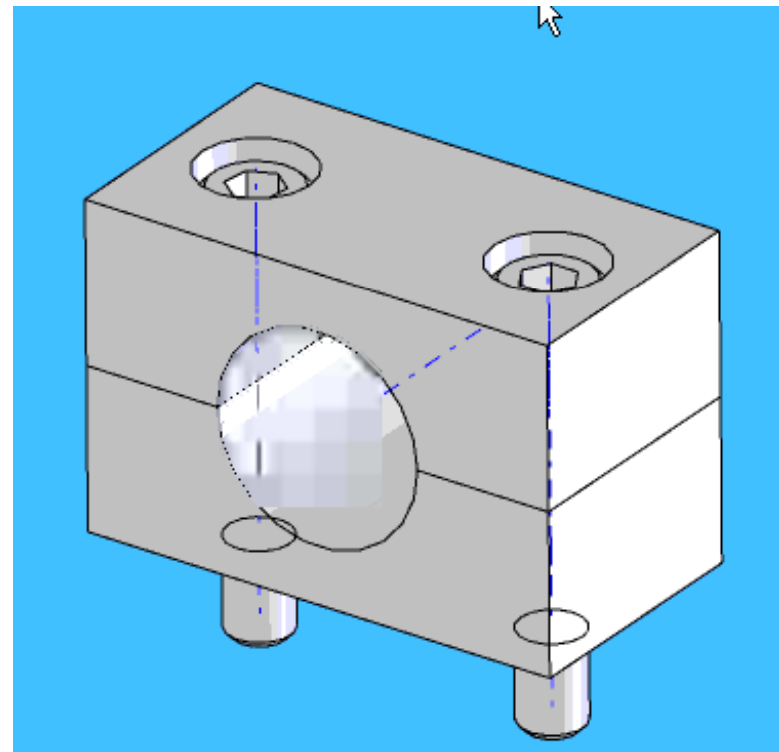
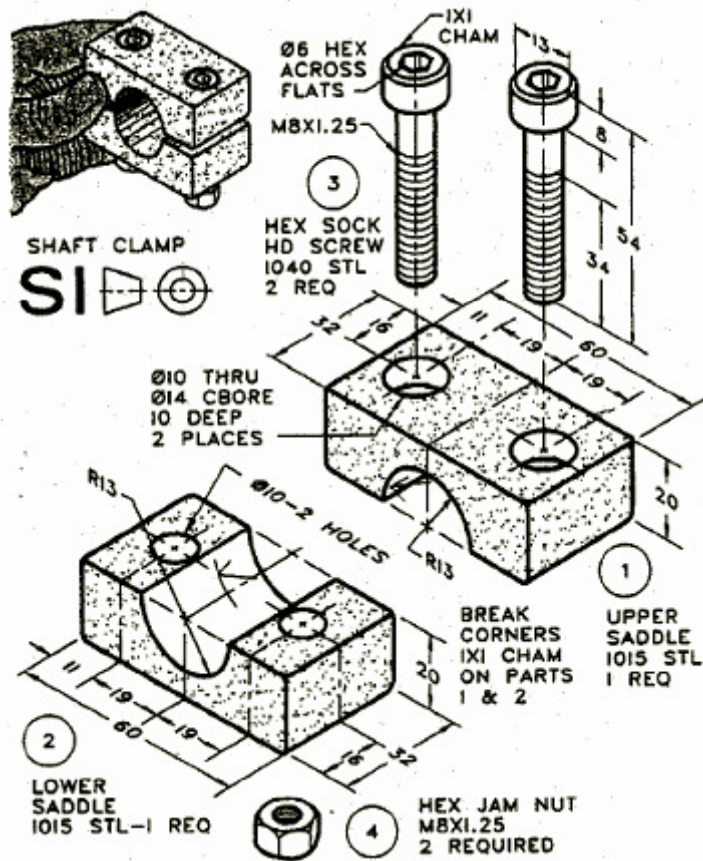
Drawings in Solidworks

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!

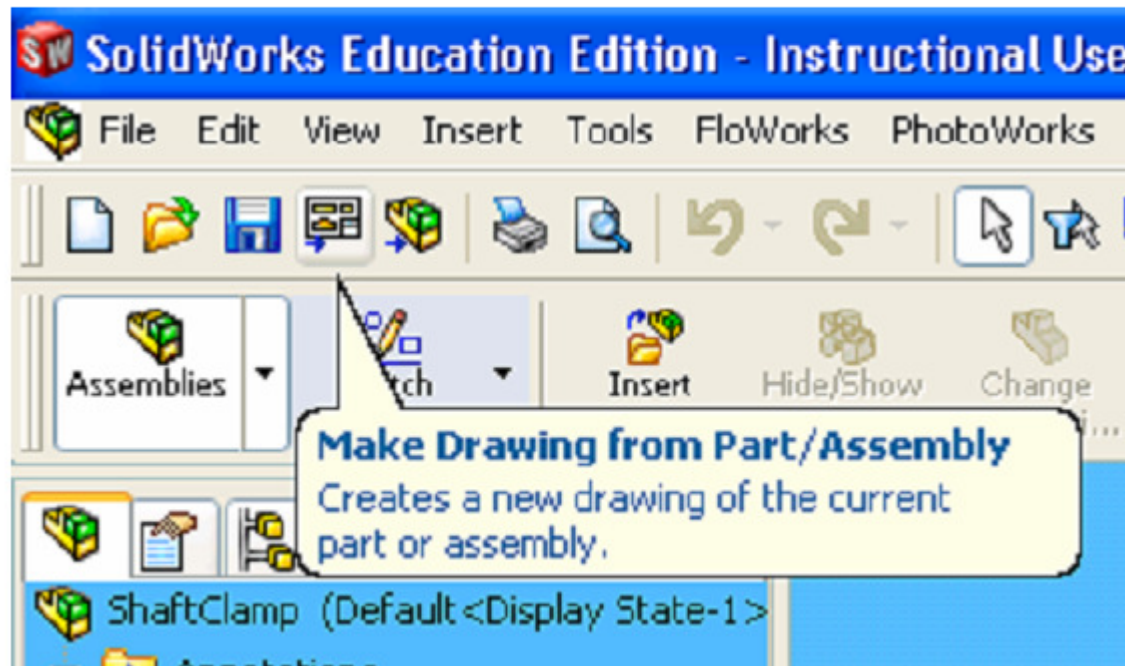
Drawings in Solidworks

Rather than start a new part from scratch, lets open the same assembly that we have been using for the past several lectures:



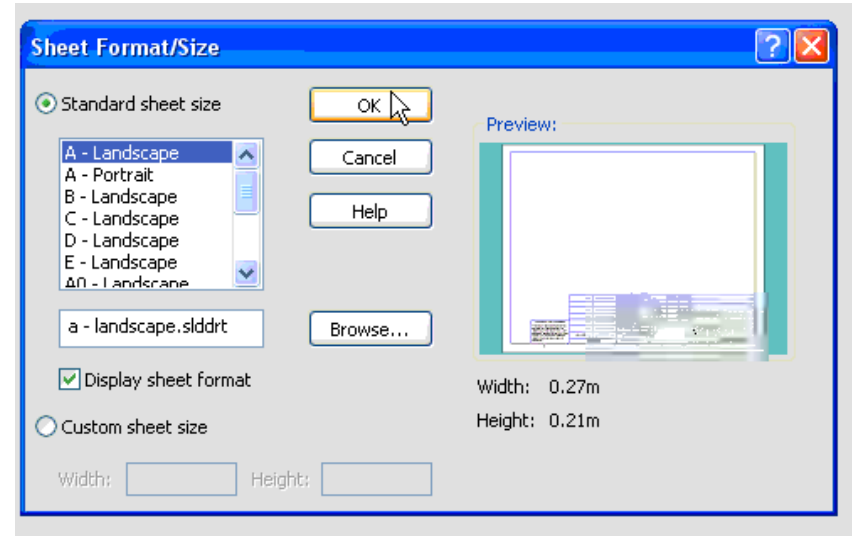
Drawings in Solidworks

Once we open the assembly, we can click on the following button which will automatically create a drawing file from our assembly:



Drawing Format

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



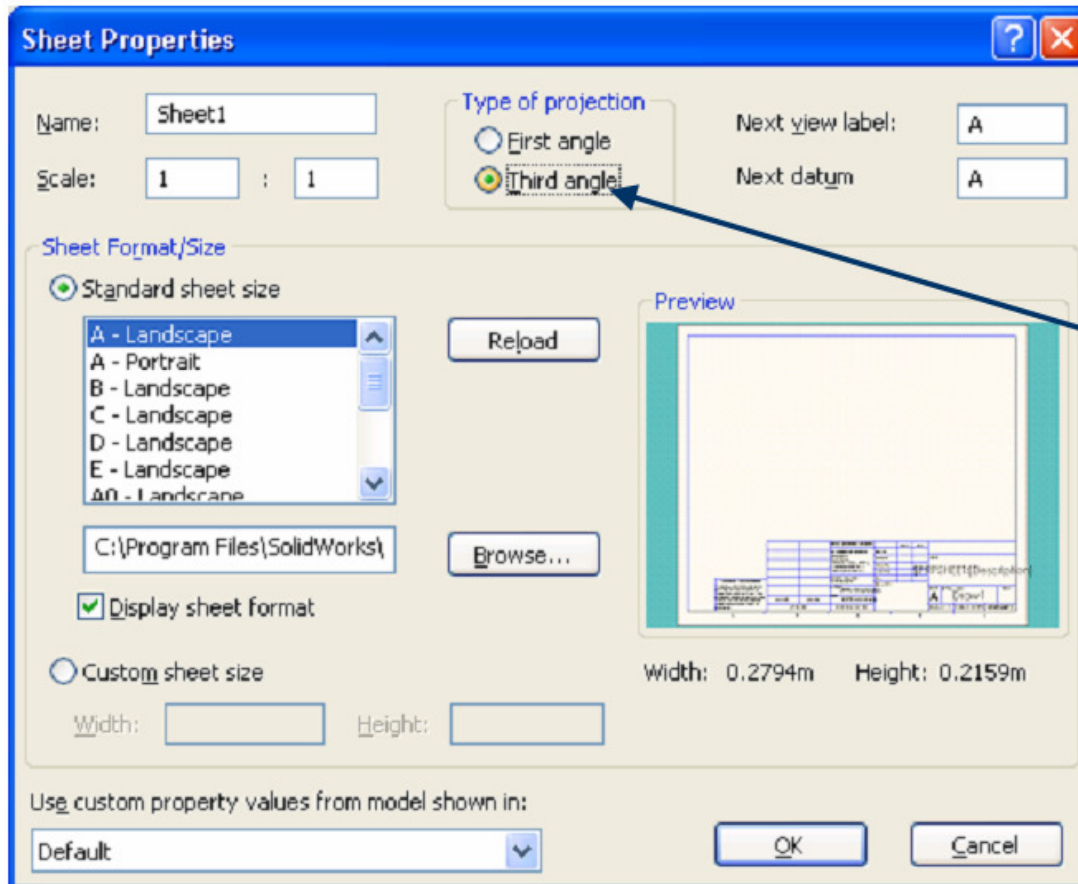
Drawings in Solidworks

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.

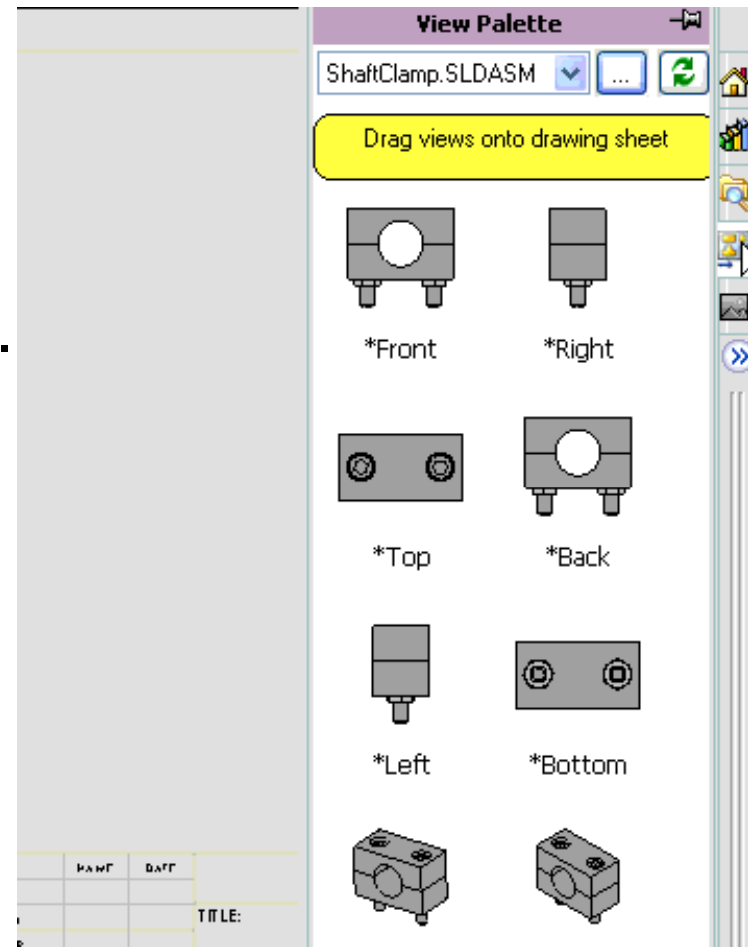
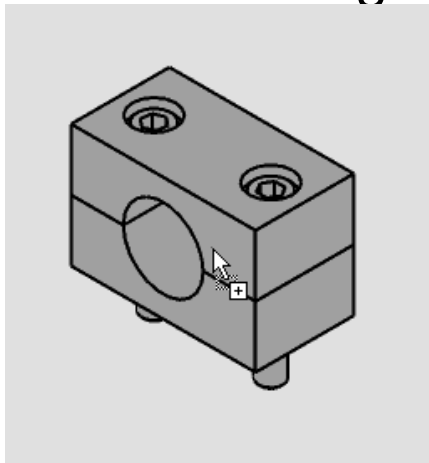


Drawings in Solidworks

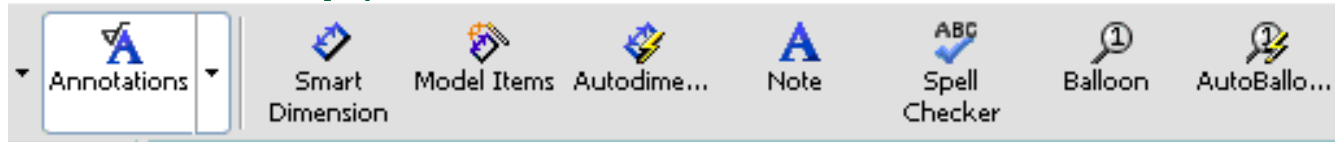
Open up the palette on the RIGHT side menu

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

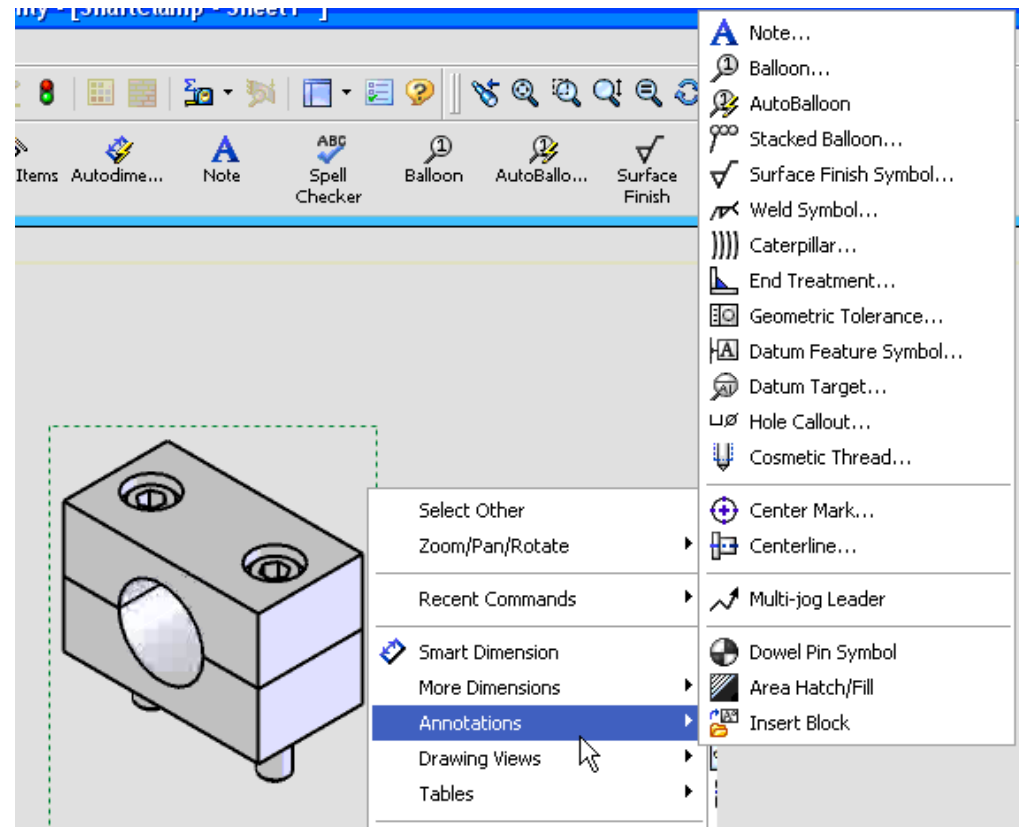
Choose the ISOMETRIC view and drag it to the drawing field.



Drawings in Solidworks



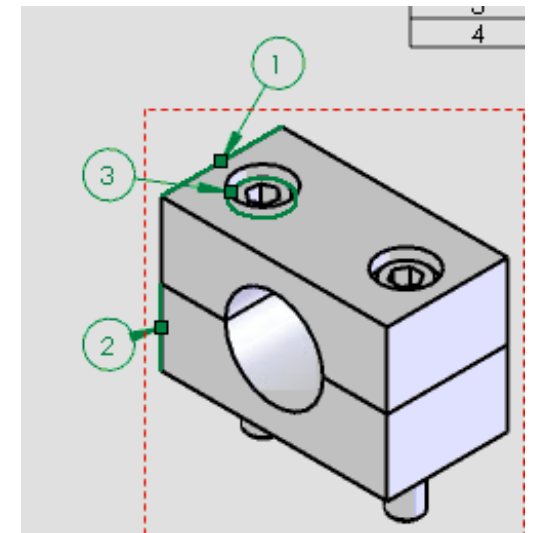
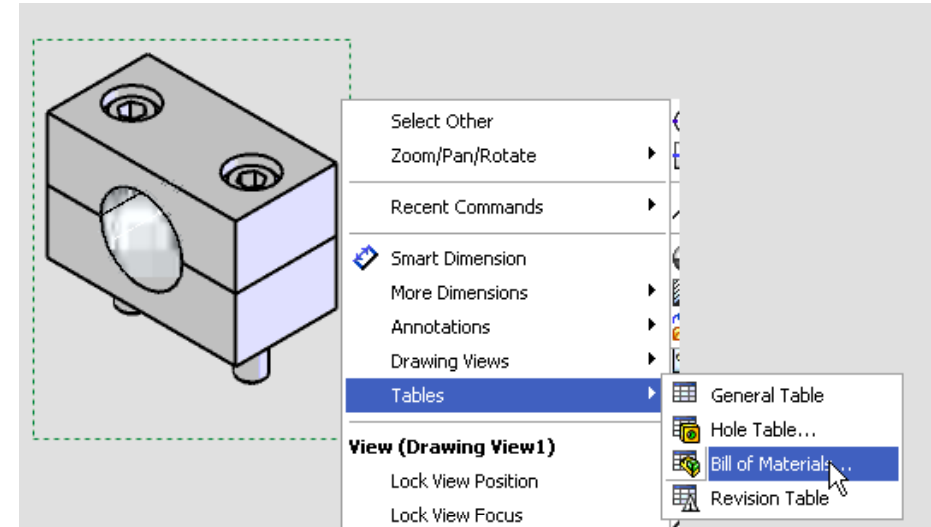
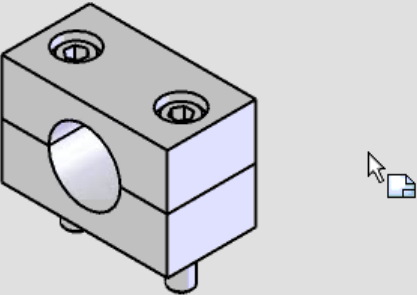
Once you have placed a view into the drawing field you can access any type of annotations by either clicking on the appropriate button on the dynamic toolbar or RIGHT clicking on the design.



Drawings in Solidworks

Let's add a bill of materials.
Right click on design,
choose TABLES, then bill of materials. Property
Manager will open, click OK

	A	B	C	D
1	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
2	1	UpperSaddle		1
3	2	LowerSaddle		1
4	3	HexSocketScrew		2
5	4	HexJamNutM8x1.25		2

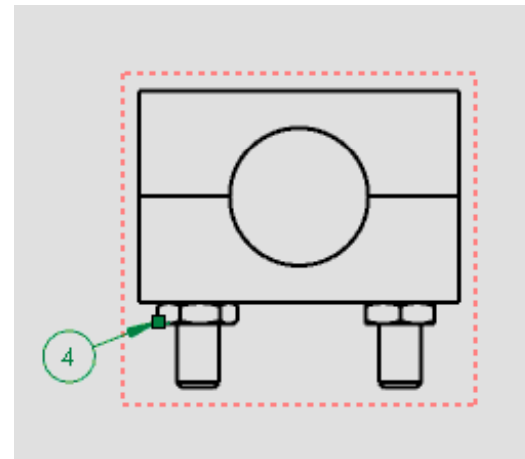
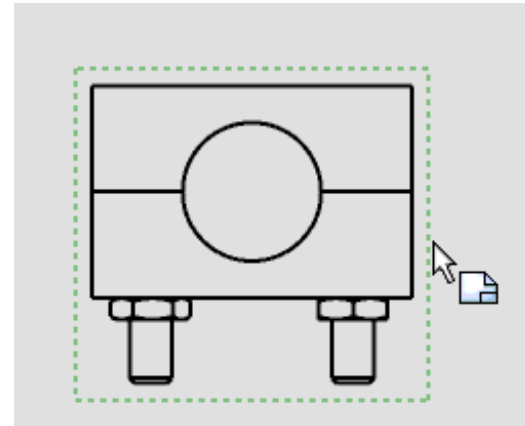


Now click the design, then AUTOBALLOON on the toolbar.
Click OK in property manager.

Drawings in Solidworks

Now add a FRONT VIEW from the palette and click OK in property manager. Then click the design and then AUTOBALLOON again.

You will notice the #4 part is now being shown.

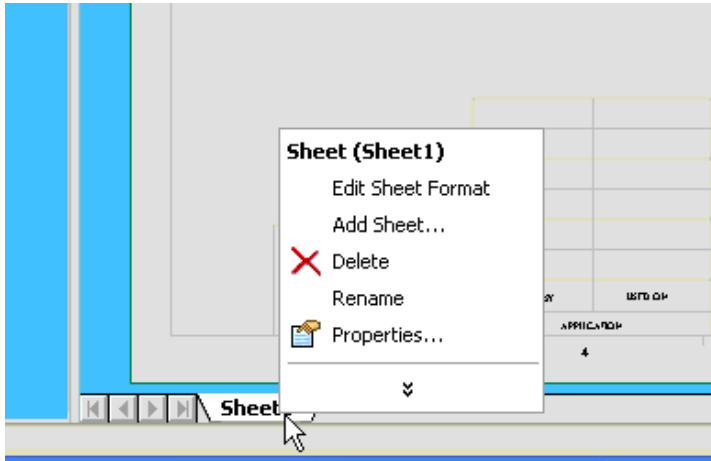


Drawings in Solidworks

Using SolidWorks to create the drawing for your assemblies can save a lot of time, however, the real power is when you go to create a drawing for a specific part.

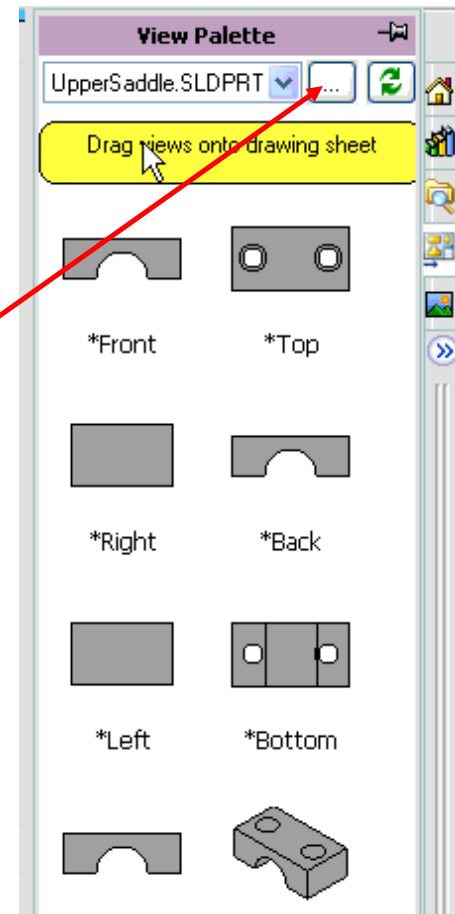
So, we can do this by creating a new sheet within the drawing file and linking to one of our part files.

Drawings in Solidworks



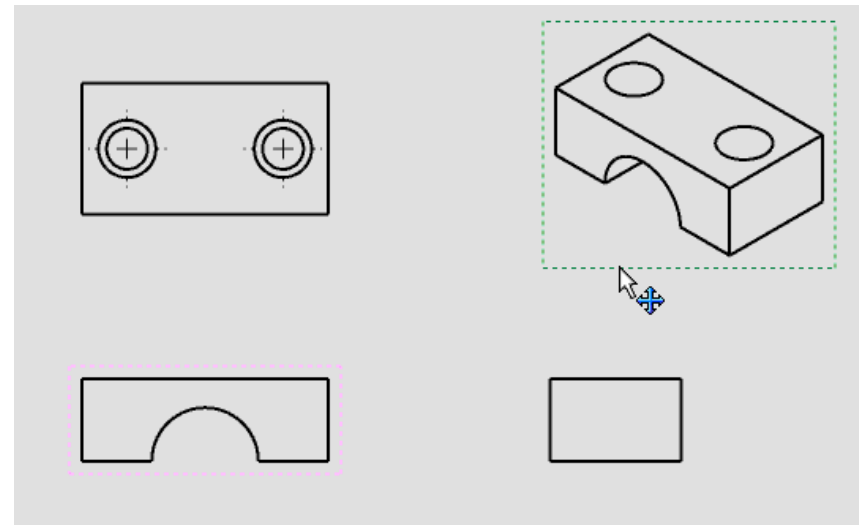
Right click on the SHEET 1 tab to add a new sheet.

Unfortunately, only the assembly file is shown in the palette. Click the box with the 3 dots to browse for the upper saddle. Choose the upper saddle from the screen and click OPEN. The upper saddle views will then appear in the palette for you to choose from.



Drawings in Solidworks

Choose the FRONT view, the click, then move UPWARD. Another VIEW will appear, click. Then move right, another view will appear, click, then move it another direction so that you get a 3D orthogonal view plus isometric.

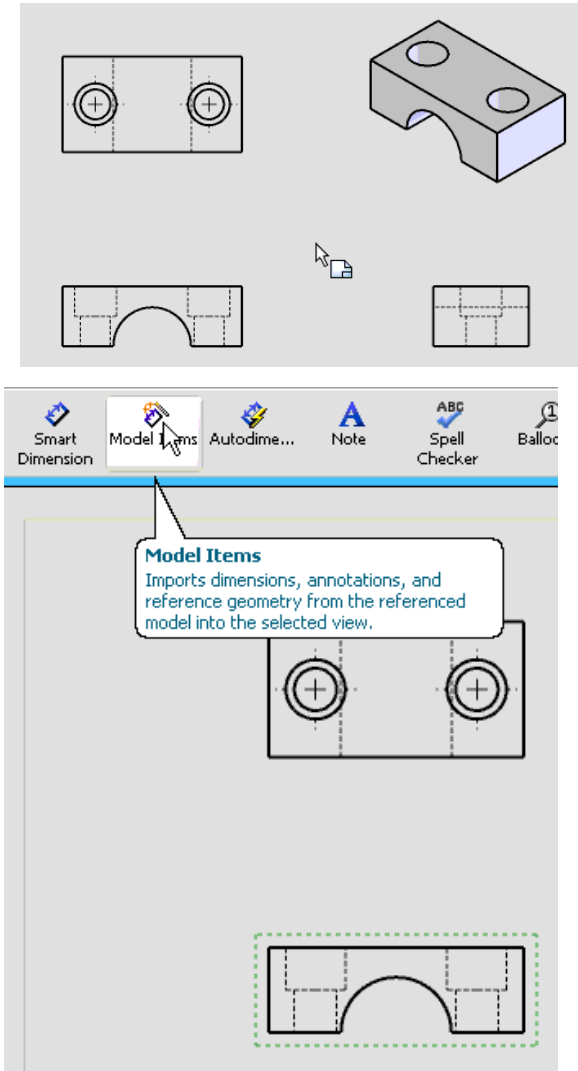


If you click the FRONT VIEW and drag it around, you will notice the other view move with it so that it is aligned.

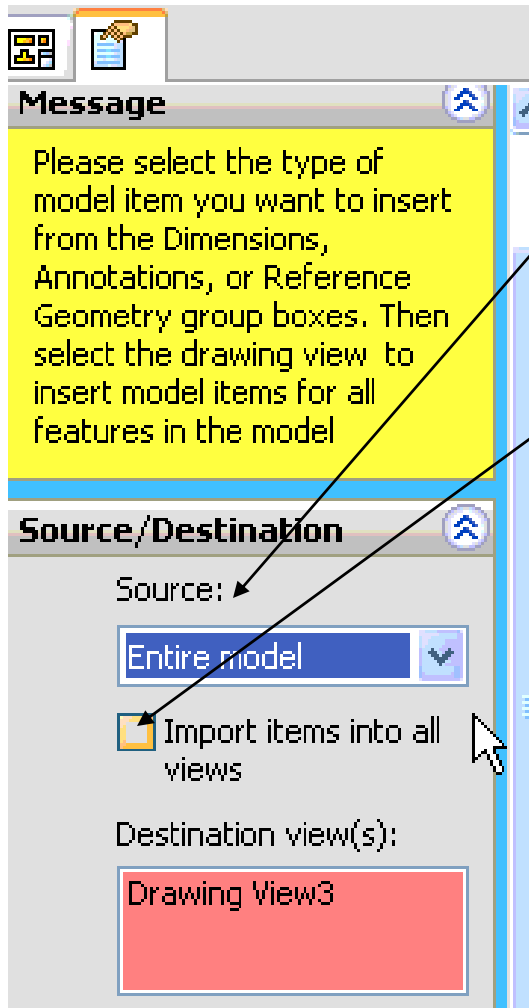
Drawings in Solidworks

Change the view of the FRONT to “hidden line” and the Isometric view to “shaded with edges”. Simply click on the design and select the change.

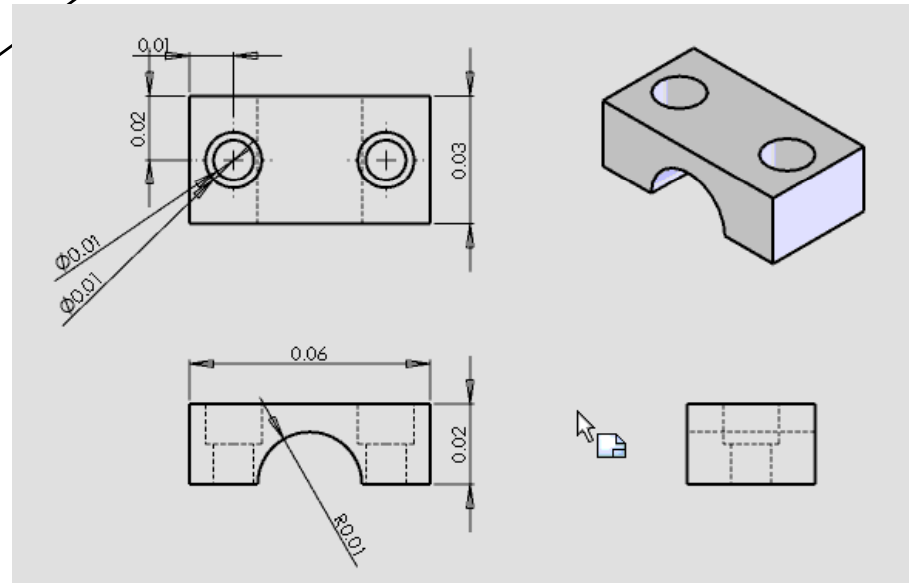
It would be nice to have some dimensions too. Click on the FRONT view as that is the one we used to project all the others and on the top toolbar choose MODEL ITEMS.



Drawings in Solidworks



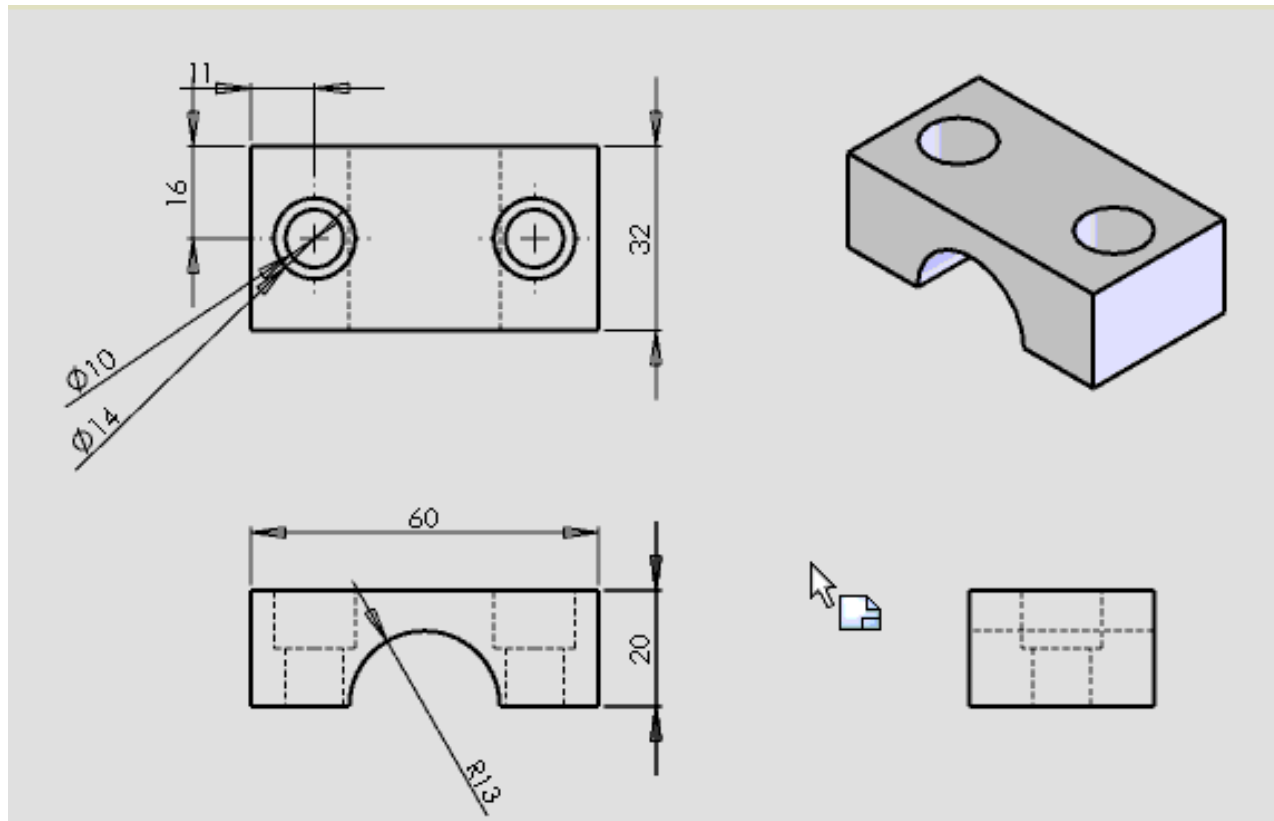
In property manager, choose SOURCE and select “entire model” from the menu. And also check, “import items into all views”. Click OK



The dimensions will automatically appear, however, you may need to shift the dimensions around for aesthetics.

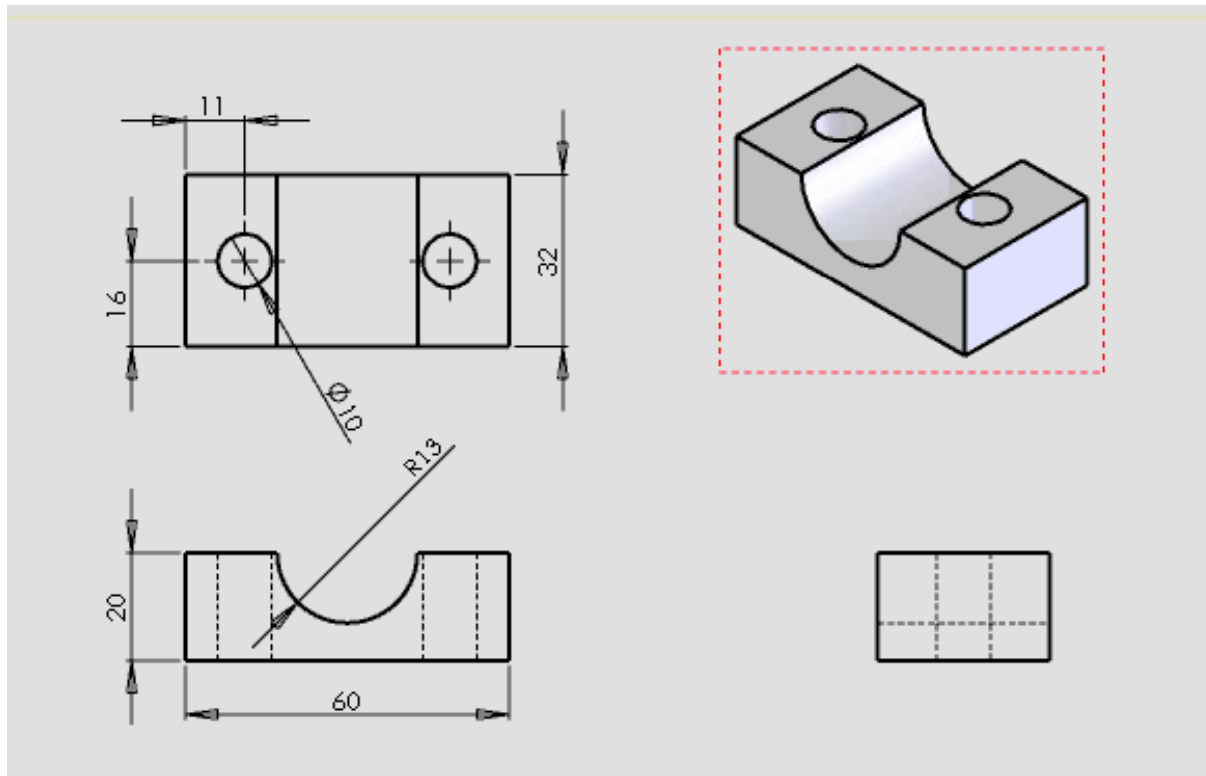
Drawings in Solidworks

If the UNITS are wrong, right click on Shaft Clamp in feature manger and change the units to MMPS.



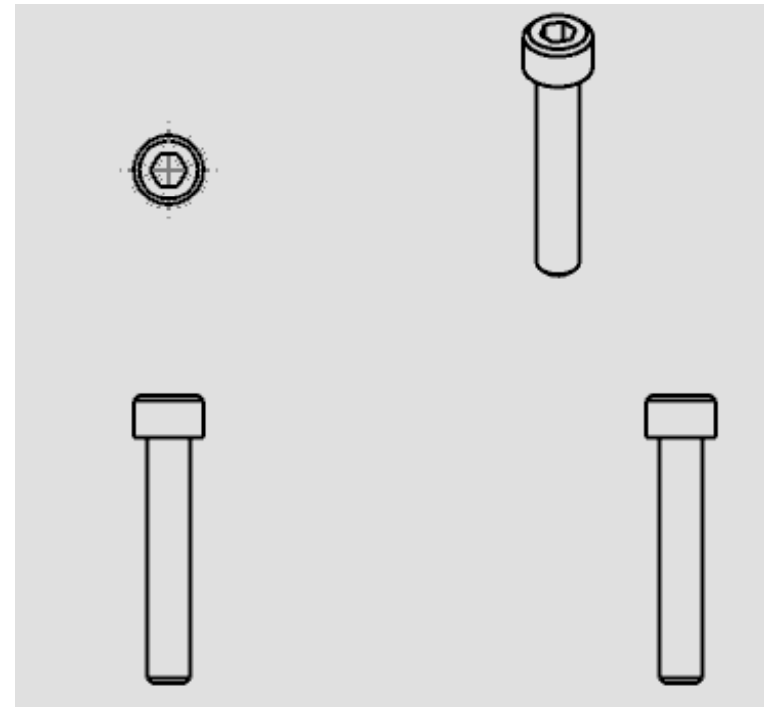
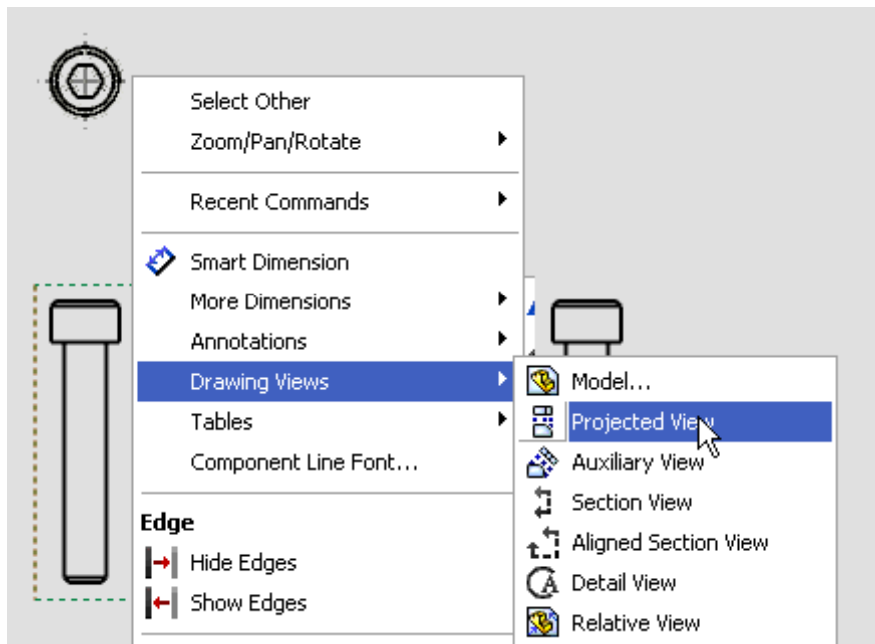
Drawings in Solidworks

Now you can ADD another SHEET. Import the lower saddle. And repeat the process.

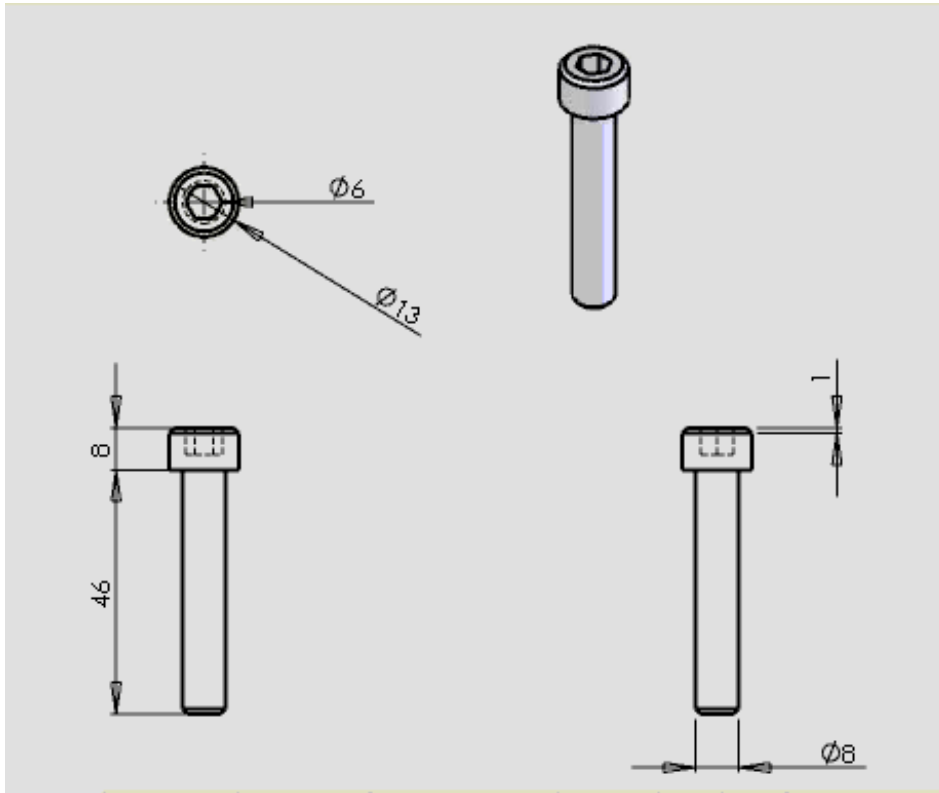


Drawings in Solidworks

Repeat for the HEW Screw. To get the projected view you may have to RIGHT CLICK on the part, then choose DRAWING VIEWS, then Projected view.



Drawings in Solidworks



Save this drawing!

Drawings in Solidworks

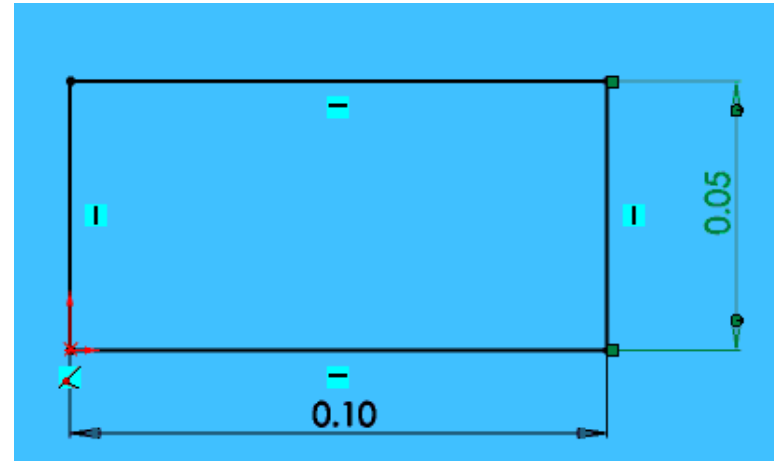
As you can see, if properly built, SolidWorks can construct the necessary drawing files almost automatically.

This is another reason why it is important to fully constrain your sketches when you are modeling your parts, if you dimension them properly, SolidWorks will be able to import your dimensions directly into your drawings.

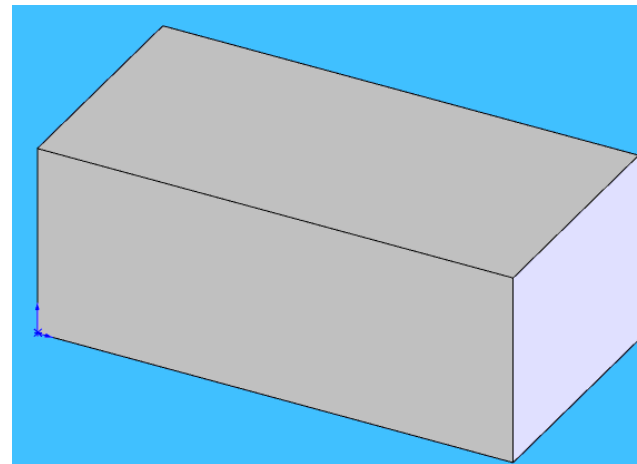
Drawings in Solidworks

Lets make a new part.

Start by choosing
extruded boss/base
from the TOP PLANE.
Make a rectangle and
dimension it 0.10 x
0.05.



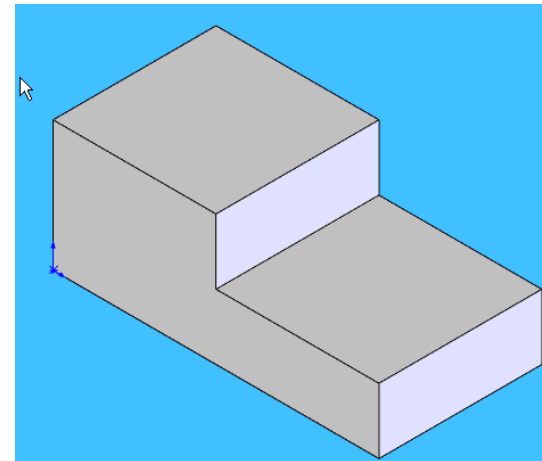
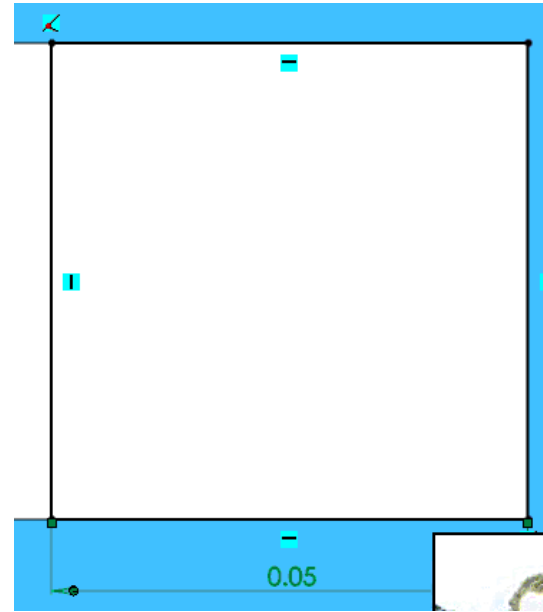
Exit sketch and extrude to
0.04



Drawings in Solidworks

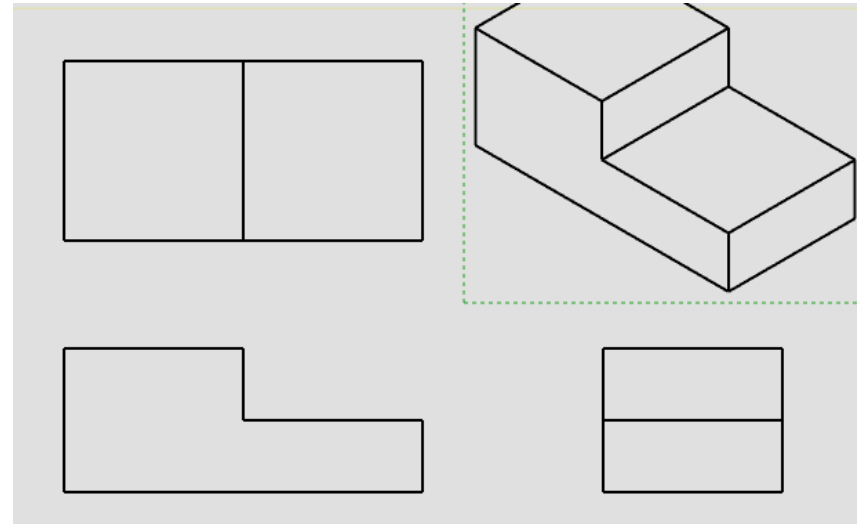
Do an EXTRUDED CUT on the top surface. Change view to Normal and draw a rectangle on the entire right side. Dimension the lower edge to be 0.05.

Exit sketch and extrude downward 0.02 then click OK. Save part as block.



Drawings in Solidworks

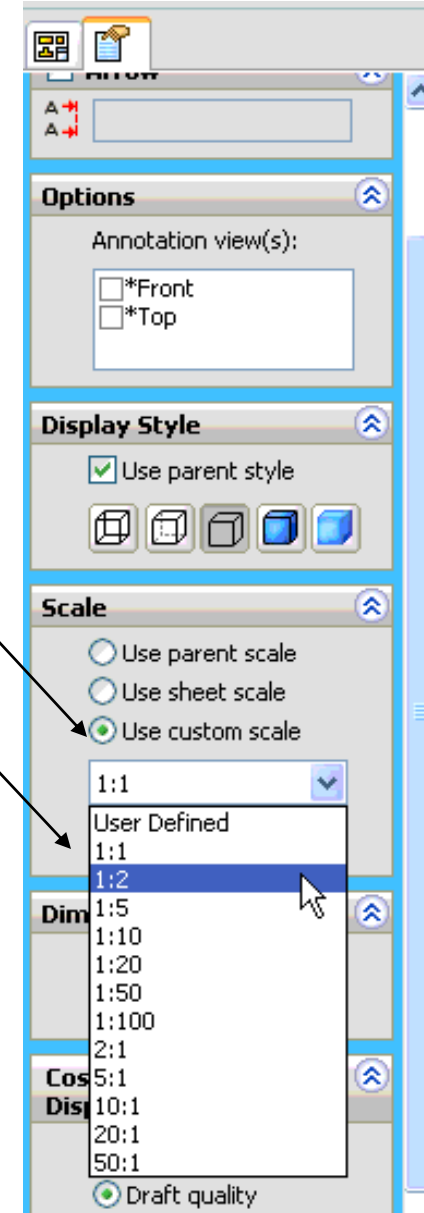
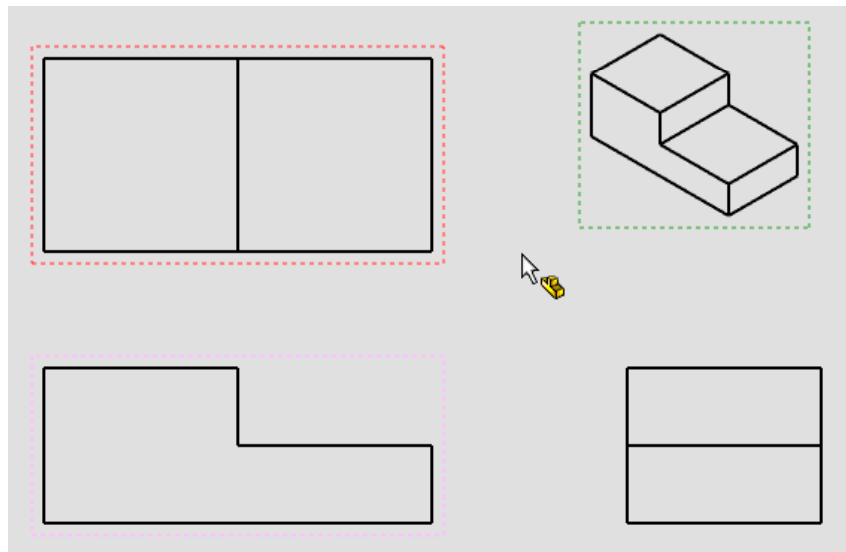
Now choose “make drawing from part” on toolbar above. Choose A-Landscape. Right click, choose properties to change view to **THIRD ANGLE**. Open the palette and drag in the Front View, clicking to reveal all the other views.



You may notice that the isometric view is too large. Click on the isometric design.

Drawings in Solidworks

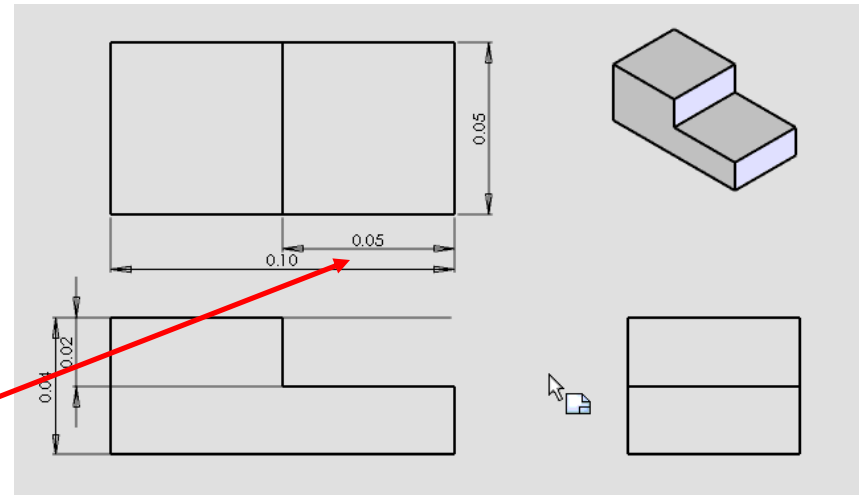
In property manager, you can change the scale by selecting CUSTOM SCALE and choosing the scale you want from the menu. Choose 1:2



Drawings in Solidworks

Change the isometric view to shaded with edges and add the MODEL ITEMS to reveal the dimensions.

Let's say we want to change the width of the cut!

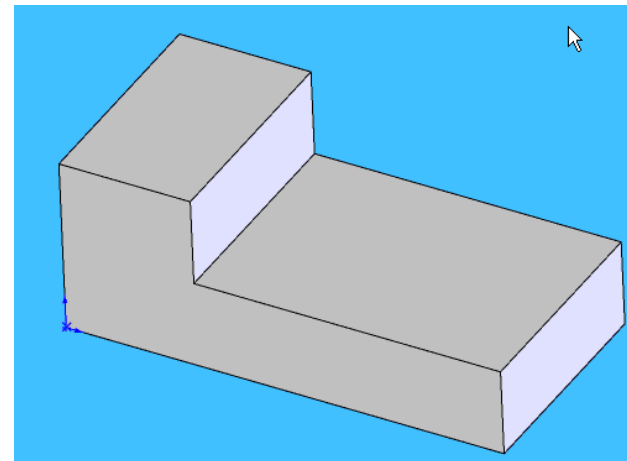
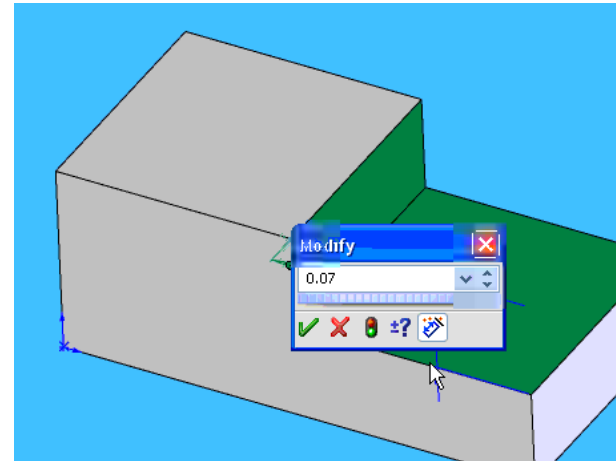


Before we doing anything,
save the drawing.

Drawings in Solidworks

Change your window back to the part and double click on the cut part to reveal the dimensions. Change the width to 0.07.

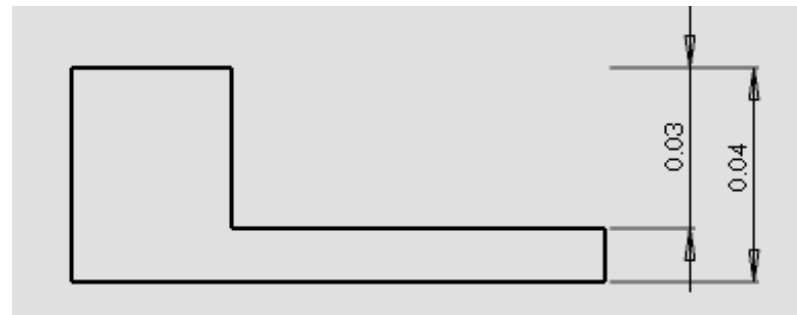
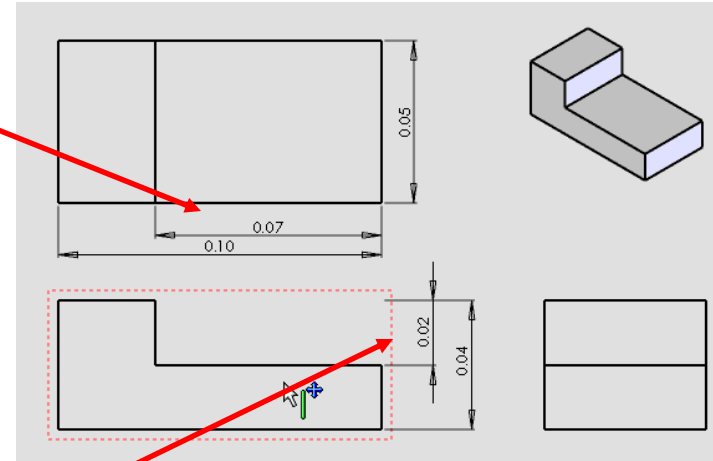
Click OK, then RE-BUILD



Drawings in Solidworks

Change your window back to the drawing. Notice the changes are already fixed.

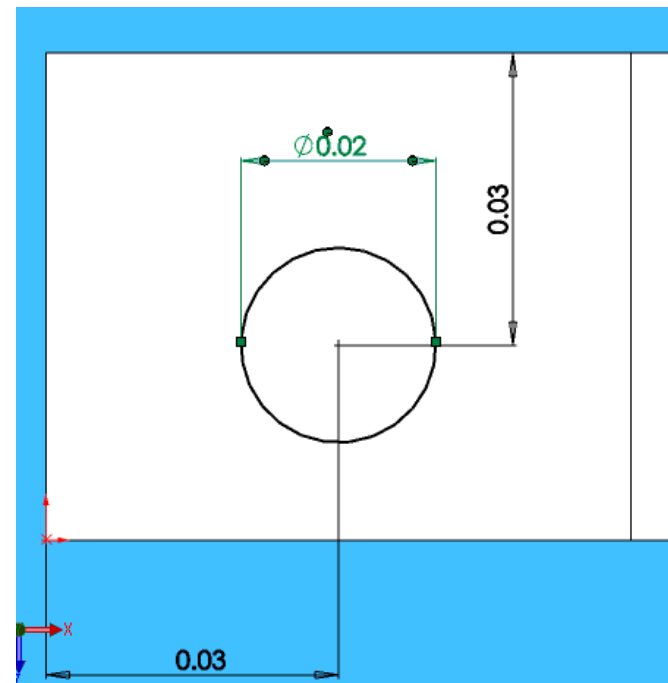
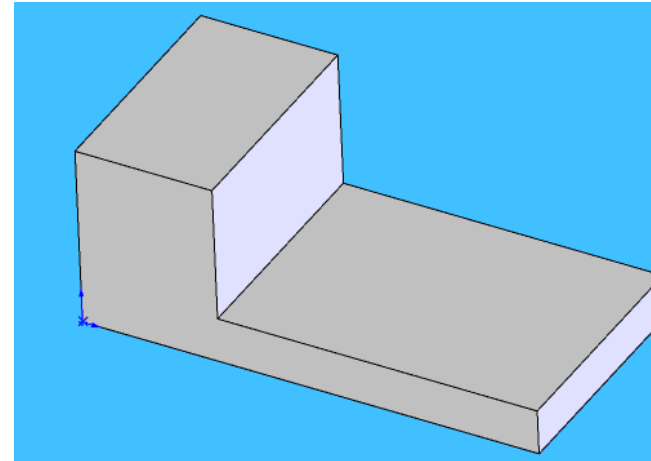
Let's make a change on the drawing now. Change the cut depth to 0.03 by clicking on the dimension in the drawing. Click RE-BUILD



Drawings in Solidworks

Change the window back to the part. Notice the changes. Solidworks uses REFERENCES whereas other 3D CAD programs do not.

Lets add a hold to the top surface. Choose Extruded CUT. Draw a circle with a diameter of 0.02 and 0.03 from each edge. Be sure o change your views.



Drawings in Solidworks

Change the window back to the drawing and you will discover the hole there. If you go through the ADD MODEL ITEMS procedure again, it will add the additional missing dimensions you just added.

