



Basic SolidWorks

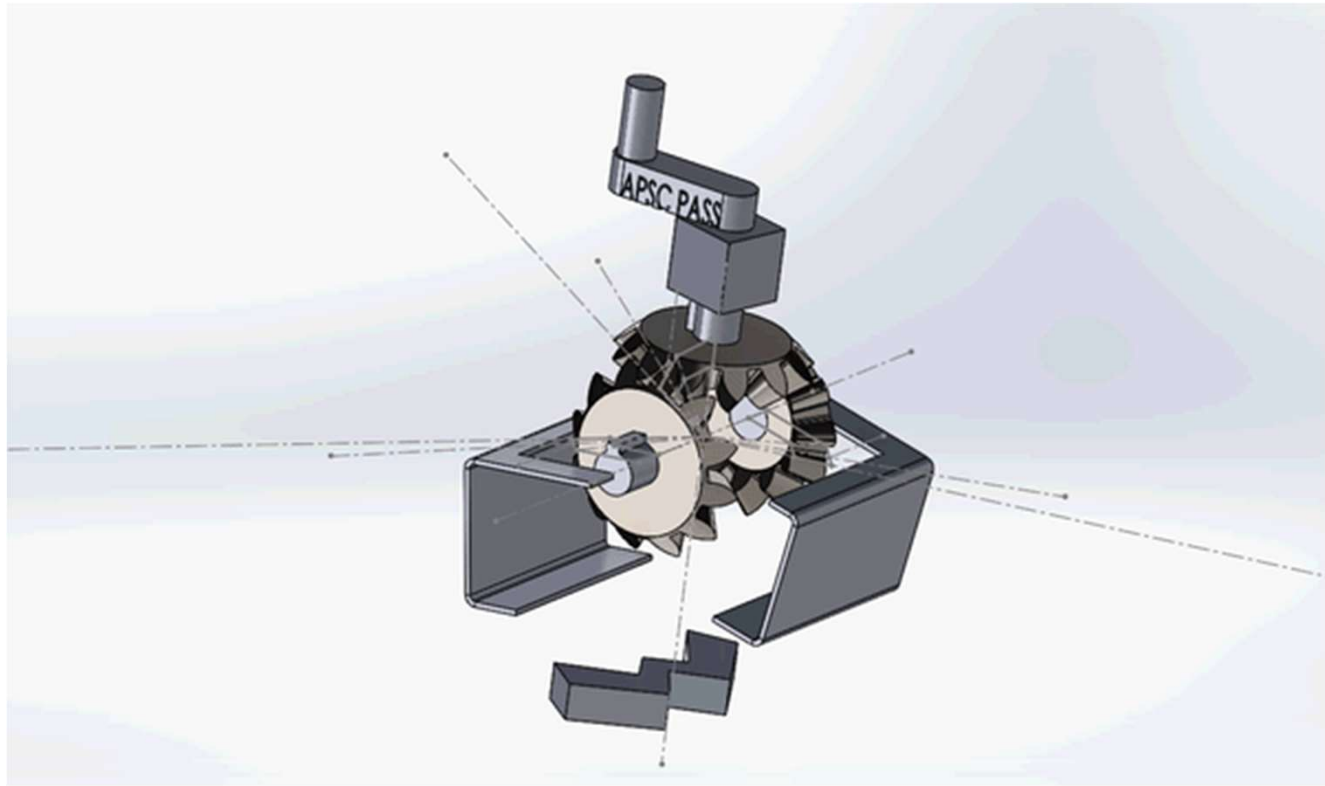
By Friend

Pre-workshop

- Download and install SolidWorks
https://ubc.service-now.com/kb_view_customer.do?sysparm_article=KB0015800
- The CAD team may use different software such as Inventor, but use Solidworks for this workshop
 - The fundamentals are all the same

Agenda

- Sketching, dimensioning, mirroring
- Extrusion, adding text
- Sheet metal



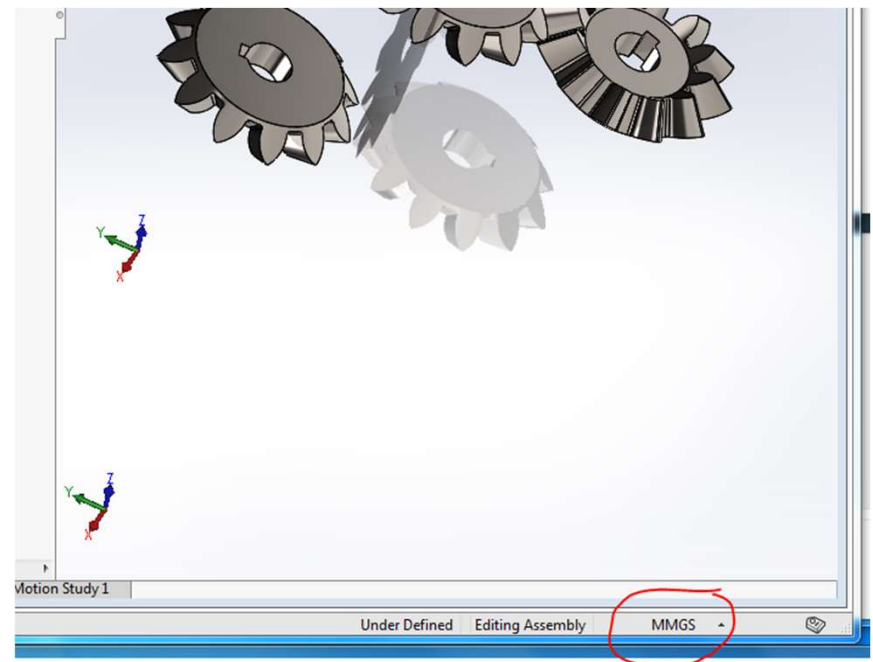
Final product from Basic SolidWorks and Adv. SolidWorks workshops



0. Setting up

Units

- Make sure you're in MMGS for this tutorial!

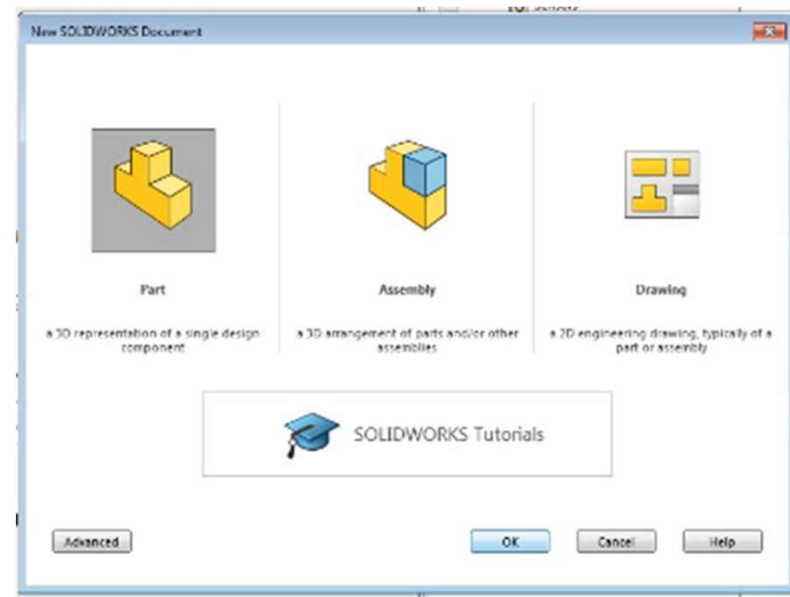




1. Sketching, dimensioning, mirroring

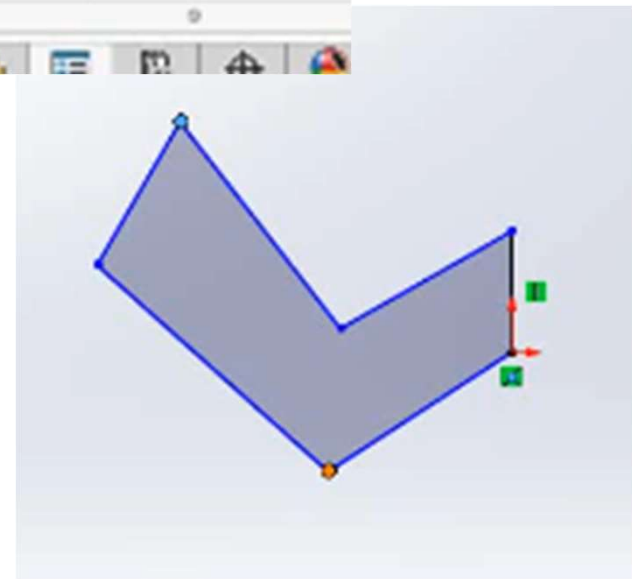
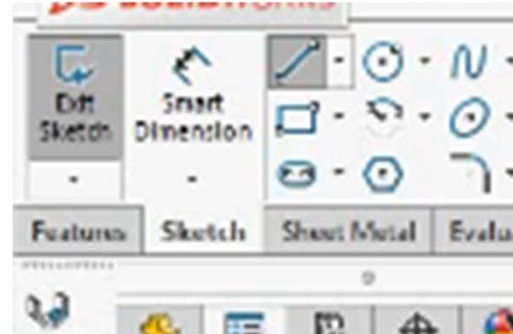
Create a new part

- File -> New -> Part
- This window should pop up



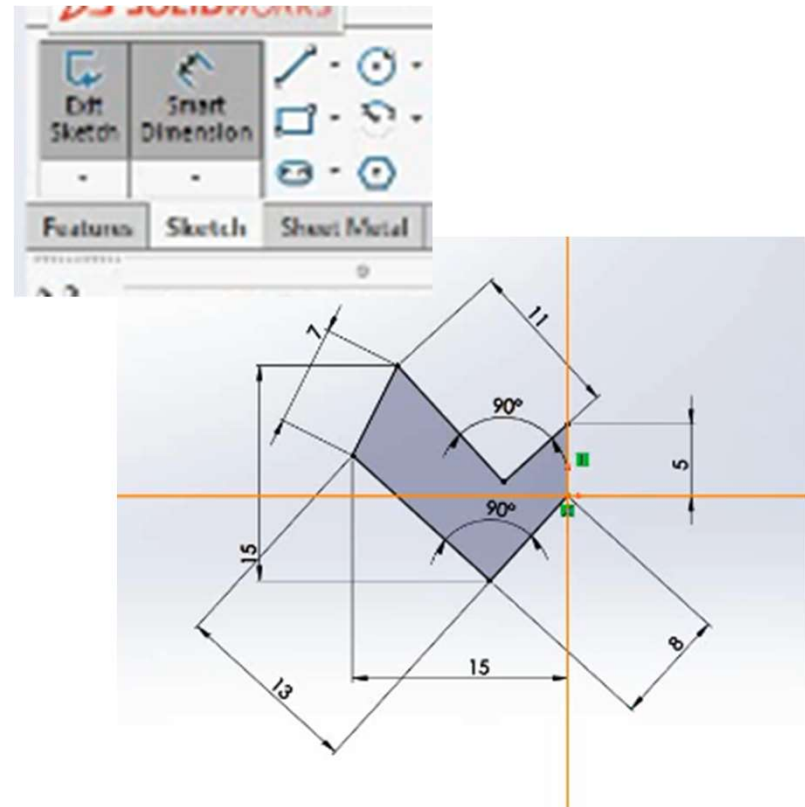
Sketching any shape

- Select any plane, click “Sketch” at Sketch toolbar, and start sketching the rabbit.
- Do not worry about dimensioning or relationships yet. Just make sure it looks similar to this.



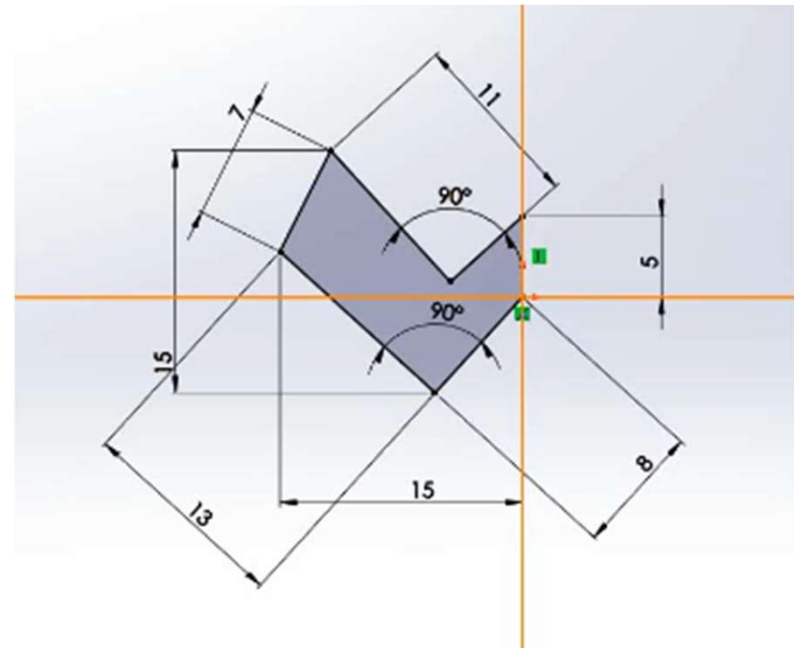
Adding dimensions

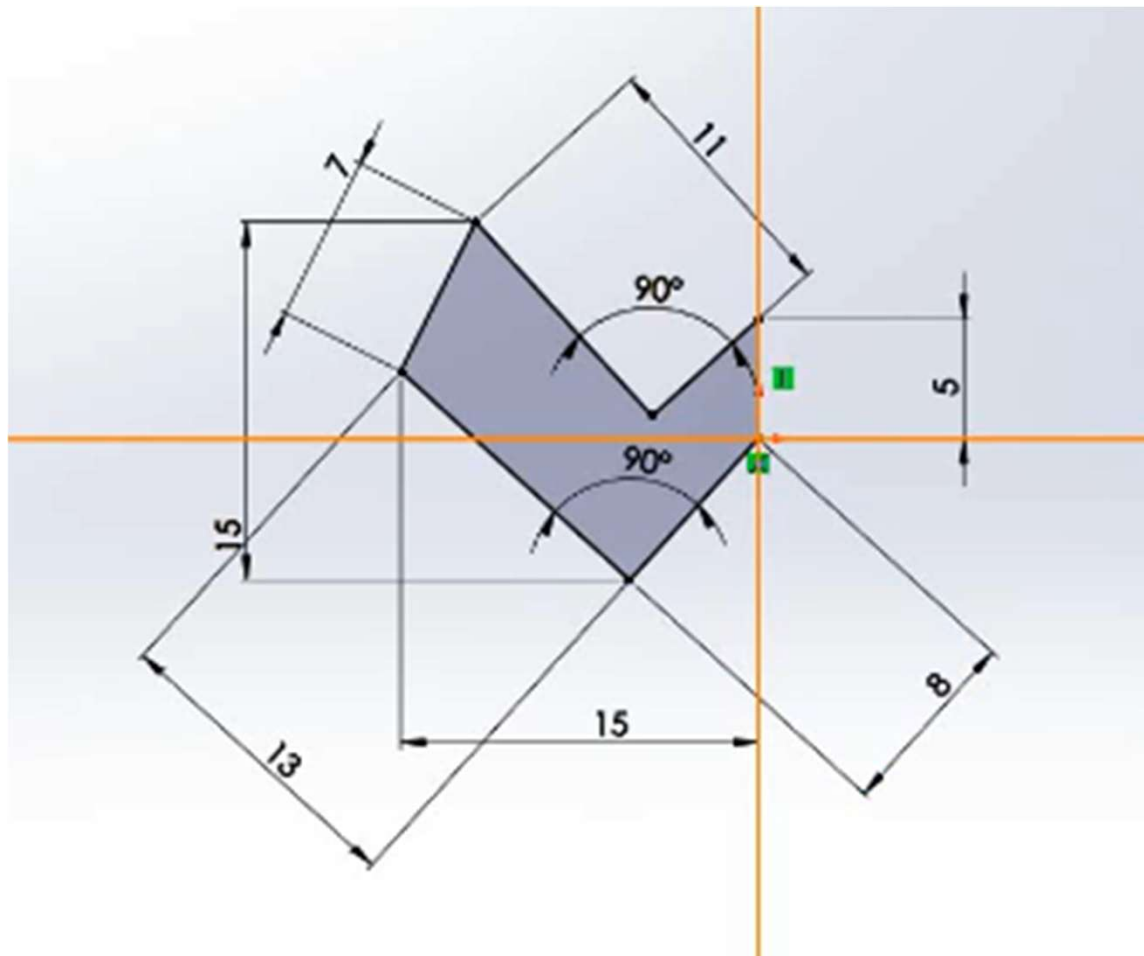
- One by one, add the dimensions and relationship according to the image here.
- As long as your original drawing somewhat resembles this, it will come together by itself once you start to add more dimensions.
- Make sure you don't have any extra dimensions or relationships, or you might not be able to add some of the required dimensions shown here.



Adding dimensions

- Make sure all the lines are black when you're done.
 - Black line means that drawing is fully defined. There's no ambiguous part or any moving part.
 - Blue, under-defined.
 - Red or yellow, over-defined.

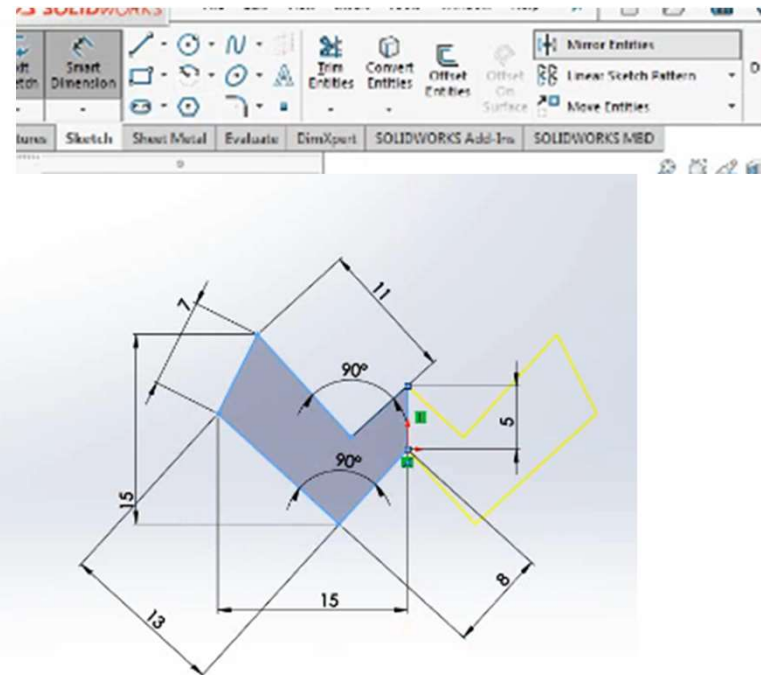




Dimensions

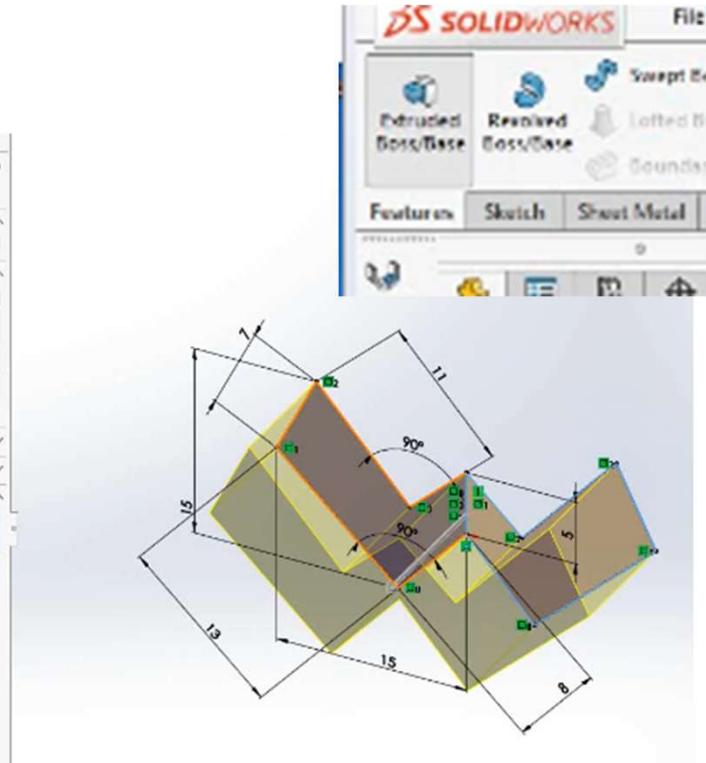
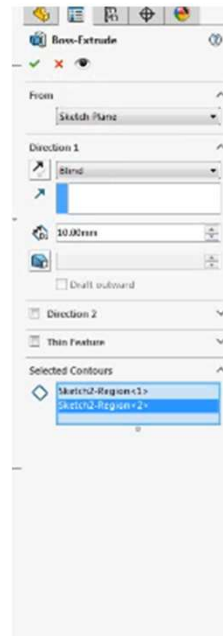
Mirroring

- Select to “Mirror Entities”
- Select all the lines except for the leftmost line to mirror, while choosing the leftmost line for everything to be mirrored about



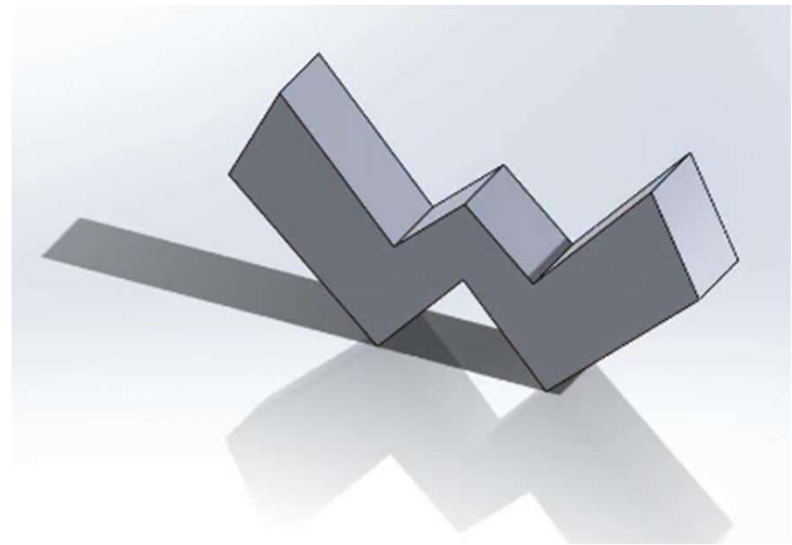
Extrusion

- Click on “Extrude Boss/Base” at Features toolbar, and extrude by 10mm.



Done!

- Save the file as “Item”(not that the file will be relevant later on).

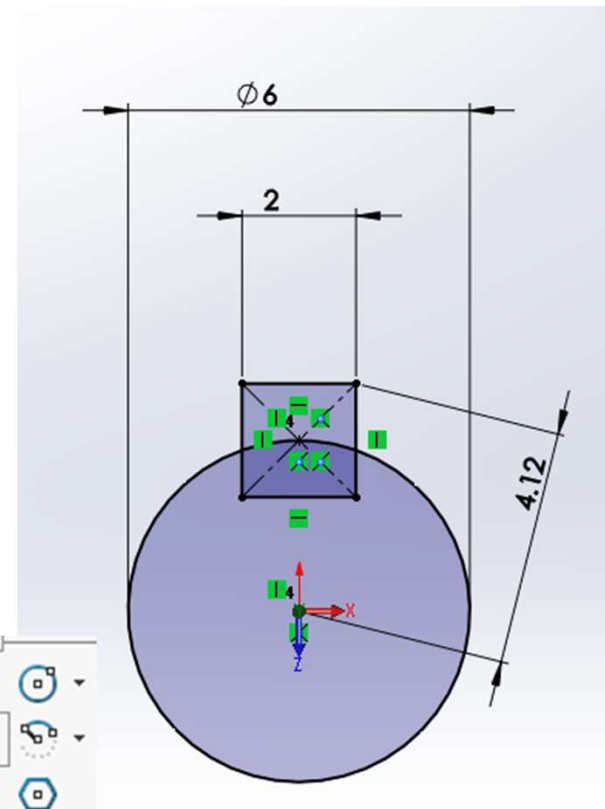
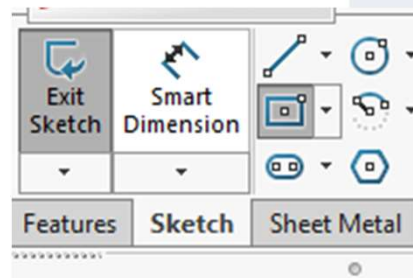




2. Extrusion, adding text

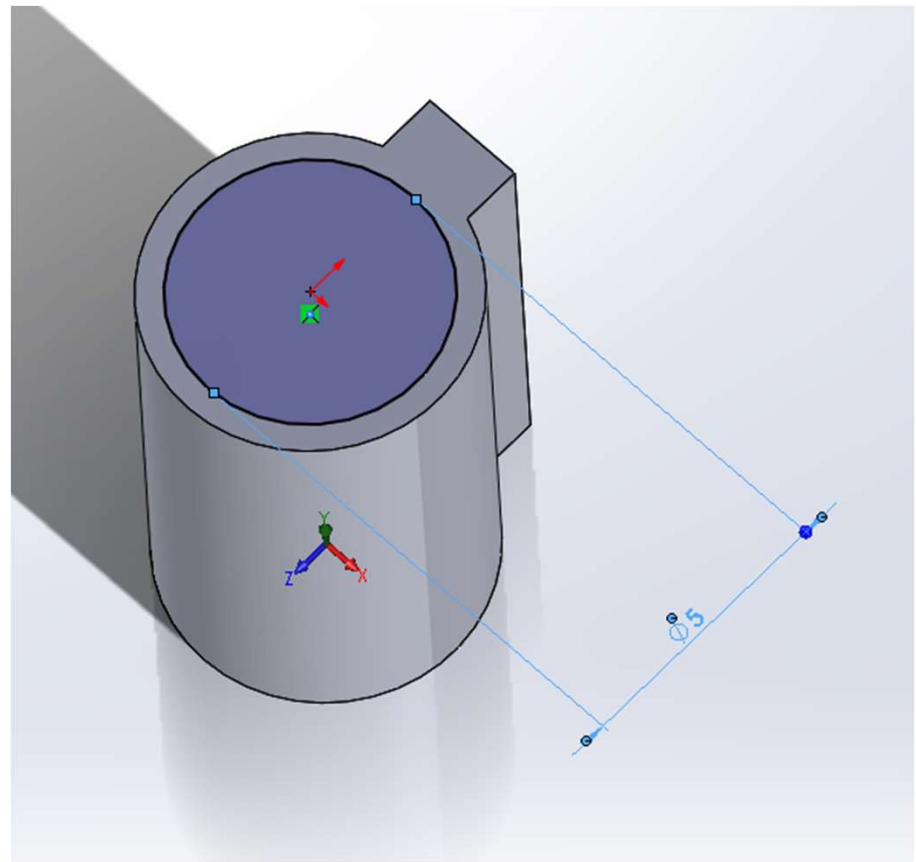
Sketching the base

- On any plane, sketch this, then extrude by 10mm.
 - To draw the rectangle, it may be helpful to use center-point rectangle instead of the regular two-corner rectangle.



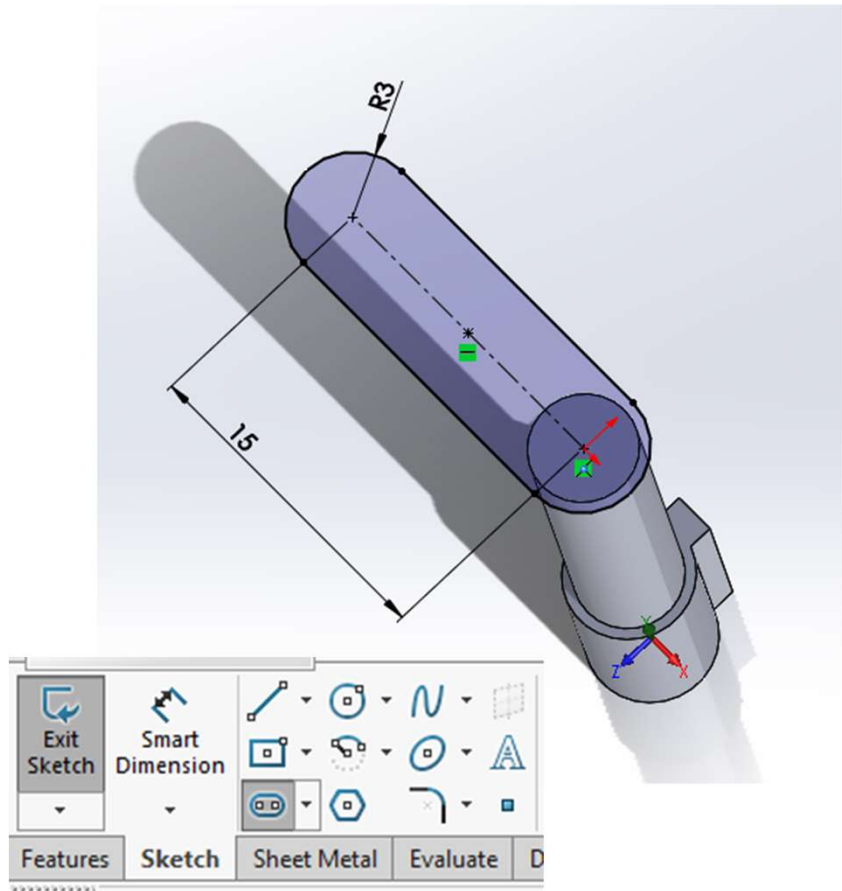
Extruding the arm

- Draw a concentric circle on the extrusion, then extrude by 10mm.



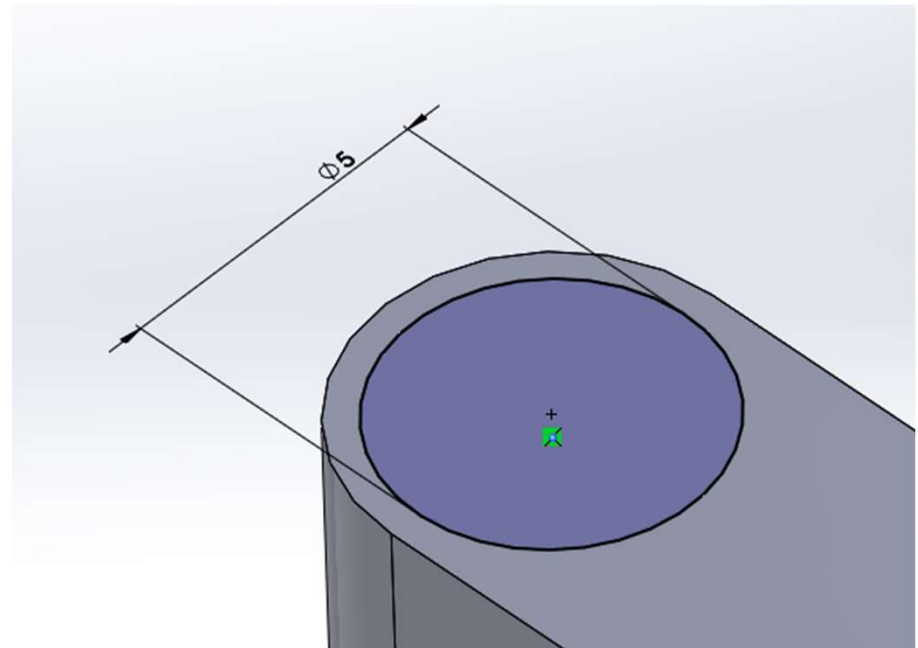
Extruding the arm

- Draw a slot to make the arm, then extrude by 5mm.
 - To draw the slot, it may be helpful to use “Straight slot” in Sketch toolbar.



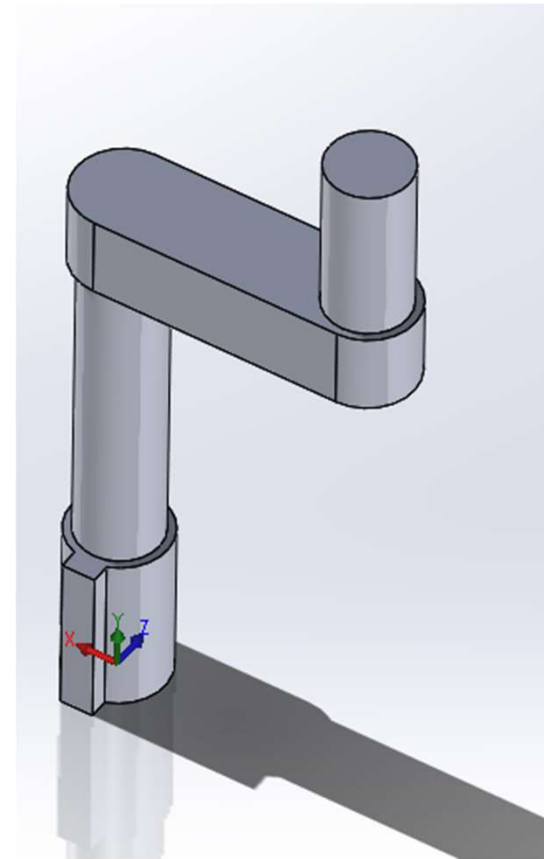
Extruding the arm

- Draw a concentric circle on the arm, then extrude by 10mm.



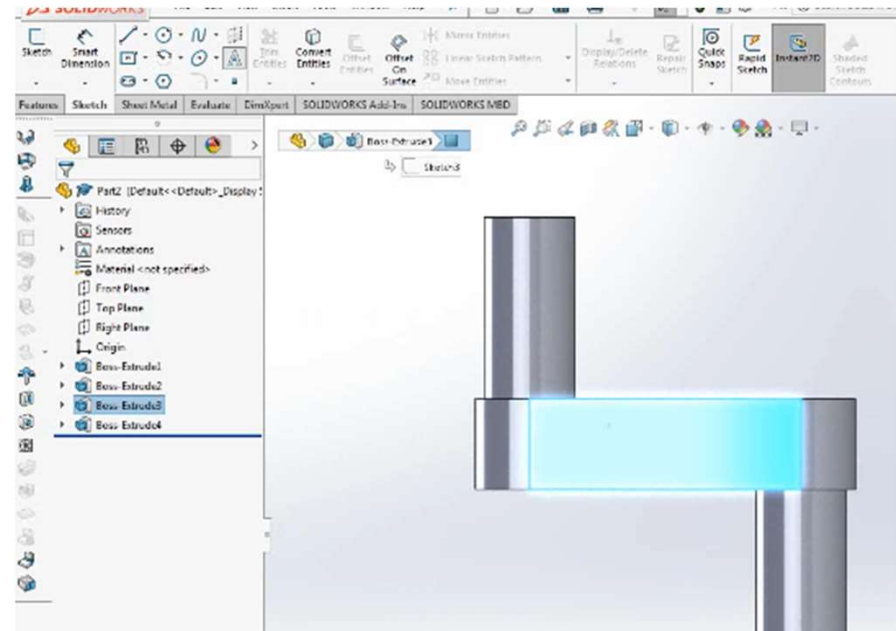
Extruding the arm

- You should have this form at this point.



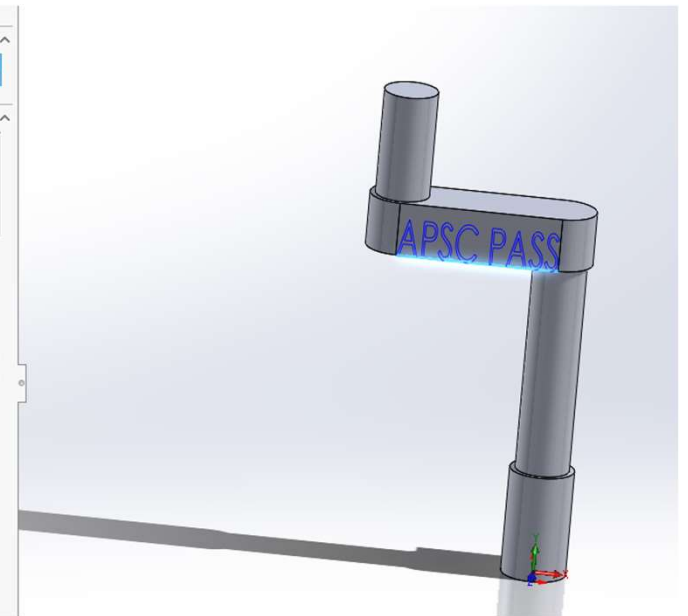
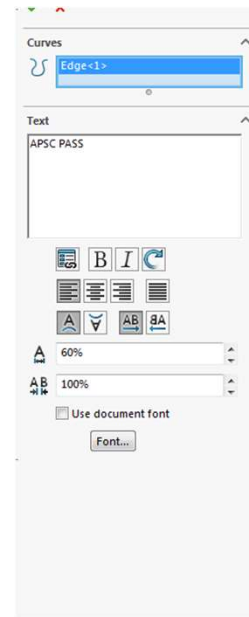
Adding text

- Select a side face of the arm.
- Go to Sketch toolbar and click “Sketch”. Click “Text” to add text.



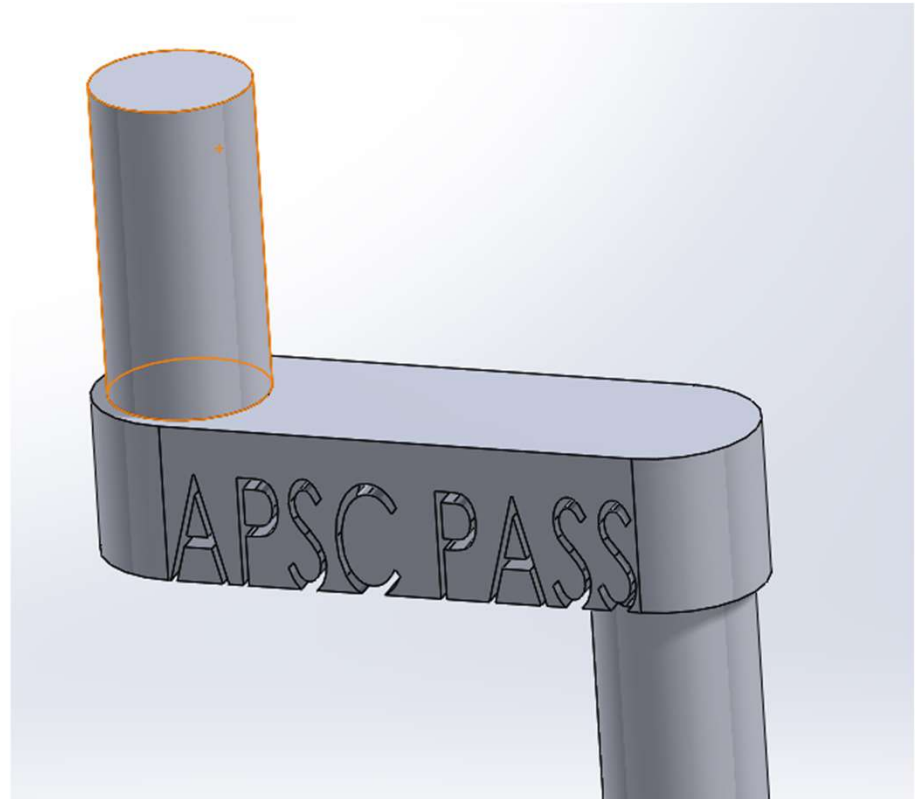
Adding text

- Choose any text(I'll go for "APSC PASS").
 - Default setting for the text is the document's. Untick the option so you can modify text to squeeze it into the given space.
- Extrude 2mm cut.



Done!

- Save the file as “Handle”





3. Sheet metal

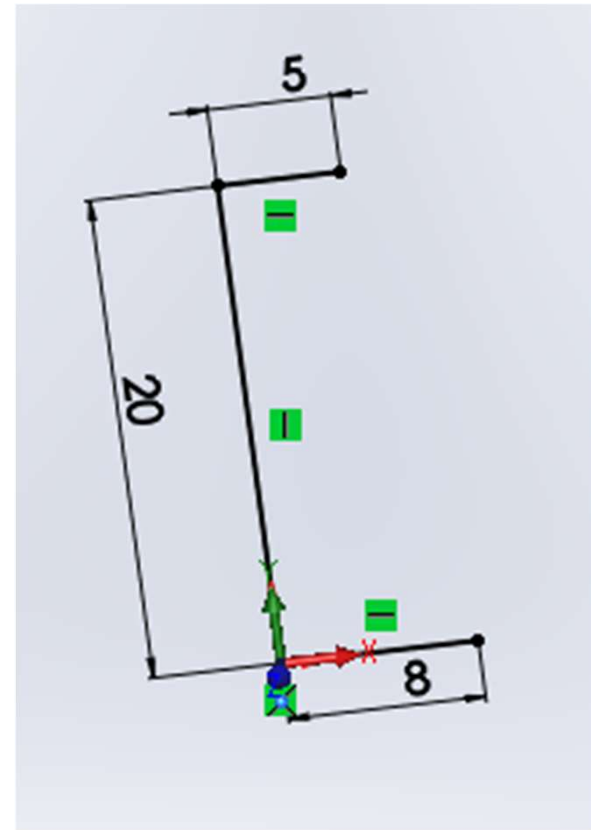
Adding sheet metal to Toolbar

Two ways:

1. Right click on Toolbar and a options will appear. Tick on "Sheet Metal"
2. View -> Toolbars -> Sheet Metal

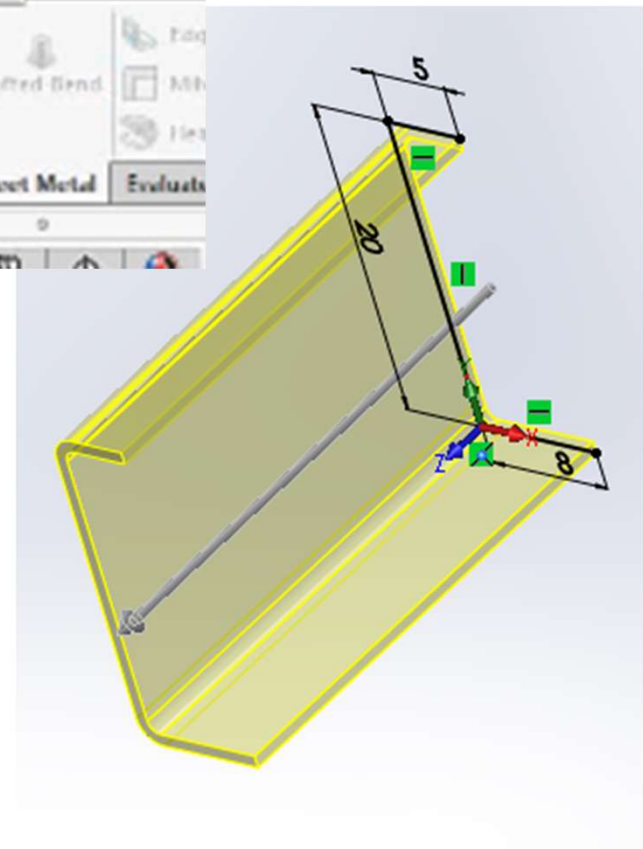
Sketching

- Select any plane and sketch accordingly.



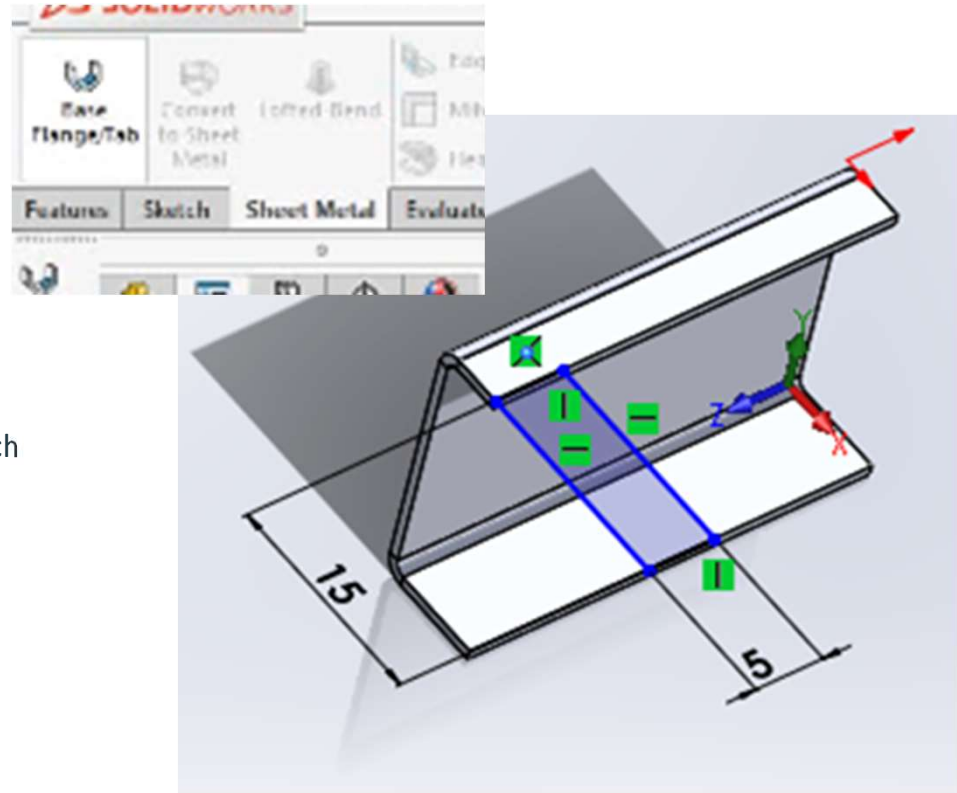
Sheet metal

- Go to Sheet metal toolbar, click “Base Flange/Tab”.
- Extrude by 30mm.



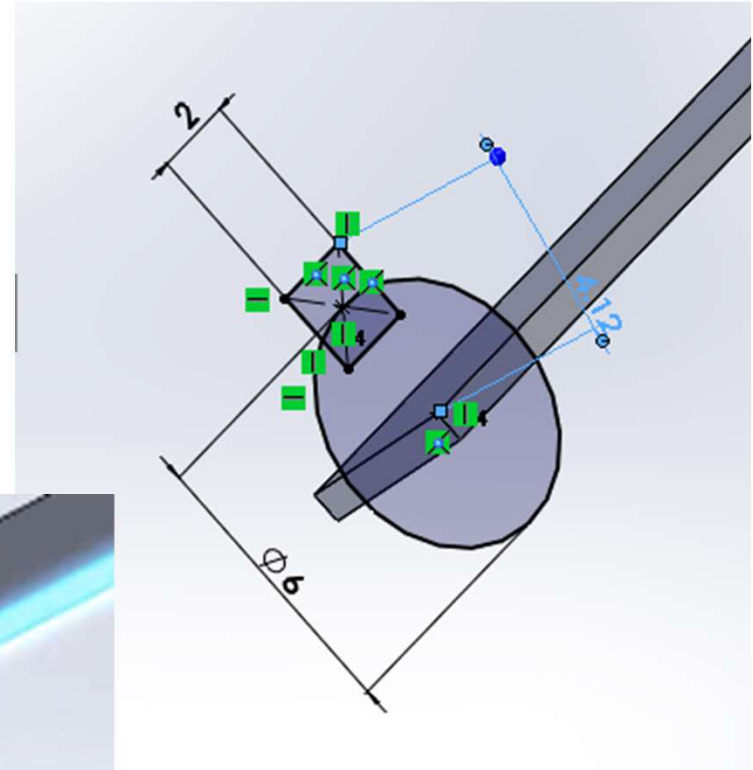
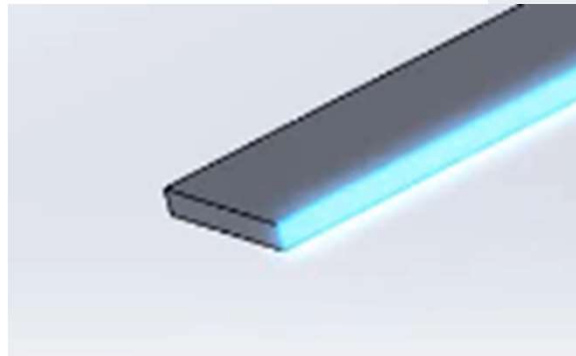
Sheet metal

- Select the top face of the claw, and sketch accordingly.
- Go to Sheet metal toolbar, click “Base Flange/Tab” to add the tab to the claw.



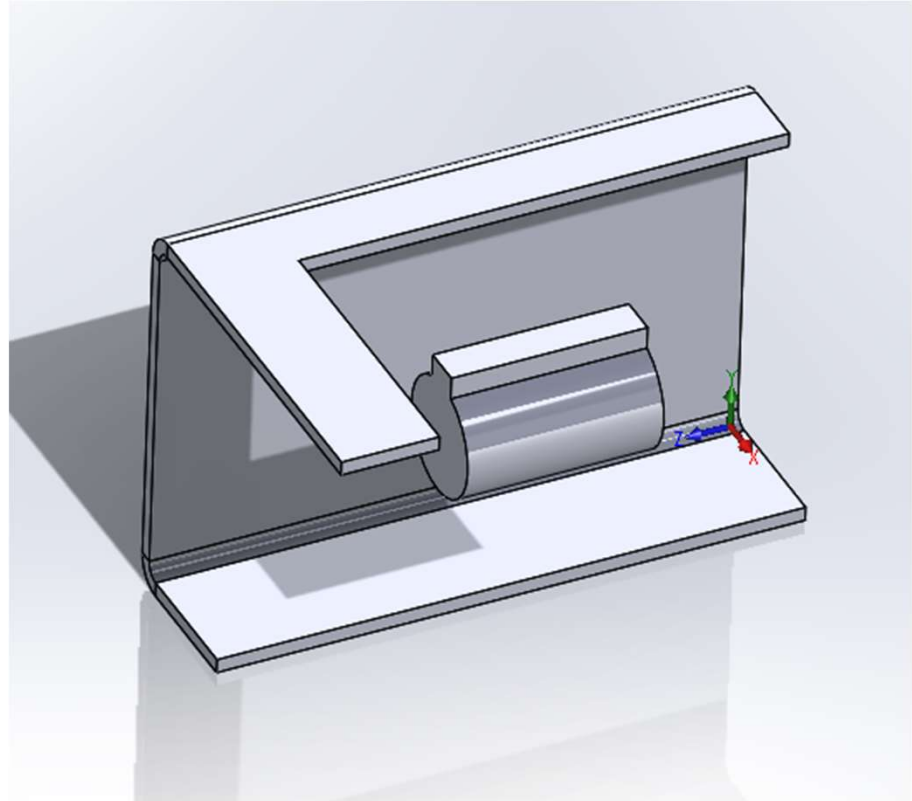
Sheet metal

- On the inner side of the tab, sketch accordingly.
- Extrude by 10mm.



Done!

- Save the file as “Claw”



Some further readings

- Get certified with CSWA
<https://www.solidworks.com/sw/support/cs-wa-academic.htm> talk to Rapid's External Coordinator for this
 - There's a good cert. prep. course on Lynda (free with VPL account)