CAD 101 in Fusion 360

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Promo Copy

Fusion 360 is a cloud-based CAD/CAM tool by Autodesk and this software is free for students, educators and start-ups. Fusion 360 incorporates parametric design, so you can change the materials you use in fabrication without having to re-design your work, which is a great asset when prototyping or testing your designs.

This workshop will take you through an introduction to the software and simple design workflow.

101 skills development workshops give you the basic skills you need to start your new creative journey. Each workshop is delivered by an experienced facilitator and no prior experience required, just basic computer skills, a willingness to learn and a bit of patience.

Acknowledgement

We acknowledge Aboriginal and Torres Strait Islander peoples and their continuing connection to land and as custodians of stories for millennia. We respectfully acknowledge the land on which we all meet today, and pay our respects to elders past, present and emerging.

Summary

Design a hook, but make it parametric.

This model can be used as an example of a functional 3D print.

Tools: Sketches, Extrude thin, Extrude Cut, Chamfer, Fillet.
Concepts: Accuracy for fitting parts, tolerance for 3D printing, sketch constraints, design flexibility, test your parameters.

Skills Introduced

- Parametric 3D modelling
- Design a thing in Fusion
- Tools: Sketching, Extrude, Revolve, Pattern, etc
- Modify a 3D model
- Export an STL

Tools

- Autodesk Fusion360
- Computer
- A mouse!
- Notepad (optional)

Preparation

Prepare a 3D printed version to demonstrate final product. Ensure Fusion is up to date.

Workshop Walk through

Hello

What are you hoping to learn today?
What would you like to use Fusion360 for in the future?

Intro

What is CAD and why is it useful?

Precision

Modification
Outputs (e.g. 3D printing or Laser Cutting)

Fusion360 and how it fits into the CAD family *(optional)*

Why Fusion360 ?

Accessibility
Push / pull modelling
Timeline (Design History)
DFM

Approach

Design intent and good practice
Model strategy (be the architect of your model), basic parametric modelling
Maintain the freedom of flexibility by having a sound model

Overview of some tools

Planes X,Y,Z Explore by hovering over toolbars. **Sketch,**
then Feature,
then Modify
Generally: add fillets and chamfers last!

Don't panic, everything can be undone or traced back ! (timeline, CTRL Z, the X).

This will make sense once we get modelling…so, let's get started !

Step 1 : Layout Sketch #1

Create a Sketch on the Front Plane.
Select Line Tool (L).
Draw from above, let go at origin, then hold while moving the mouse to the side. This is called an arc.
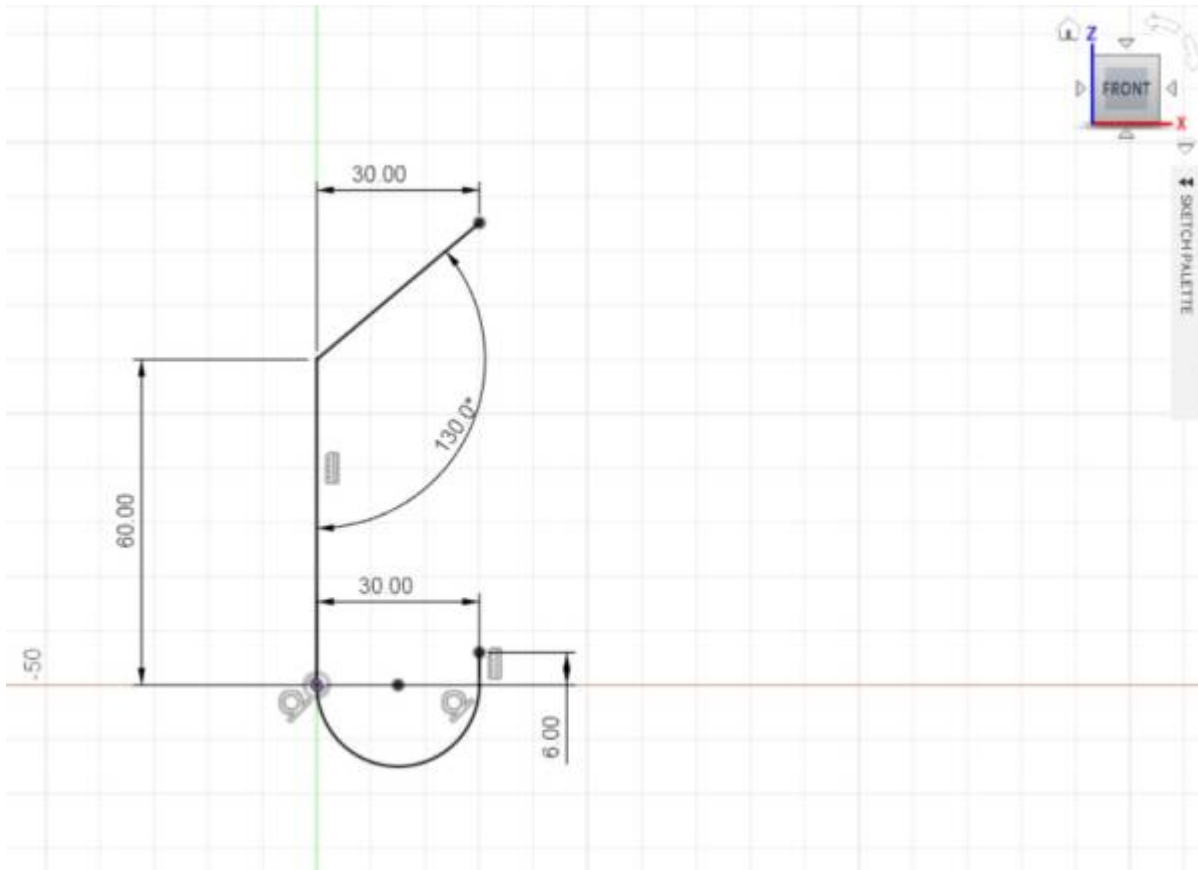Add angled line from top.
Add dimensions and constraints.
Ensure the sketch is fully constrained (i.e. black)
Finish Sketch and Rename the sketch. (It can be handy to leave the word 'sketch' in the name e.g. "Sketch: Profile", as down the line you may also have a Feature called 'Profile'.)

Step 2: Create Layout Sketch #2: Holes

Select Line Tool.
Change line type to construction.
Snap to centre, and draw it the length of the middle segment.
Circle tool - Centre Diameter Circle.
To get this sketch to be fully constrained, we need to dimension the circle diameters and also their position on the line.
Note: Sketch dimension will not snap to the centre of the construction line,so we can add a point.
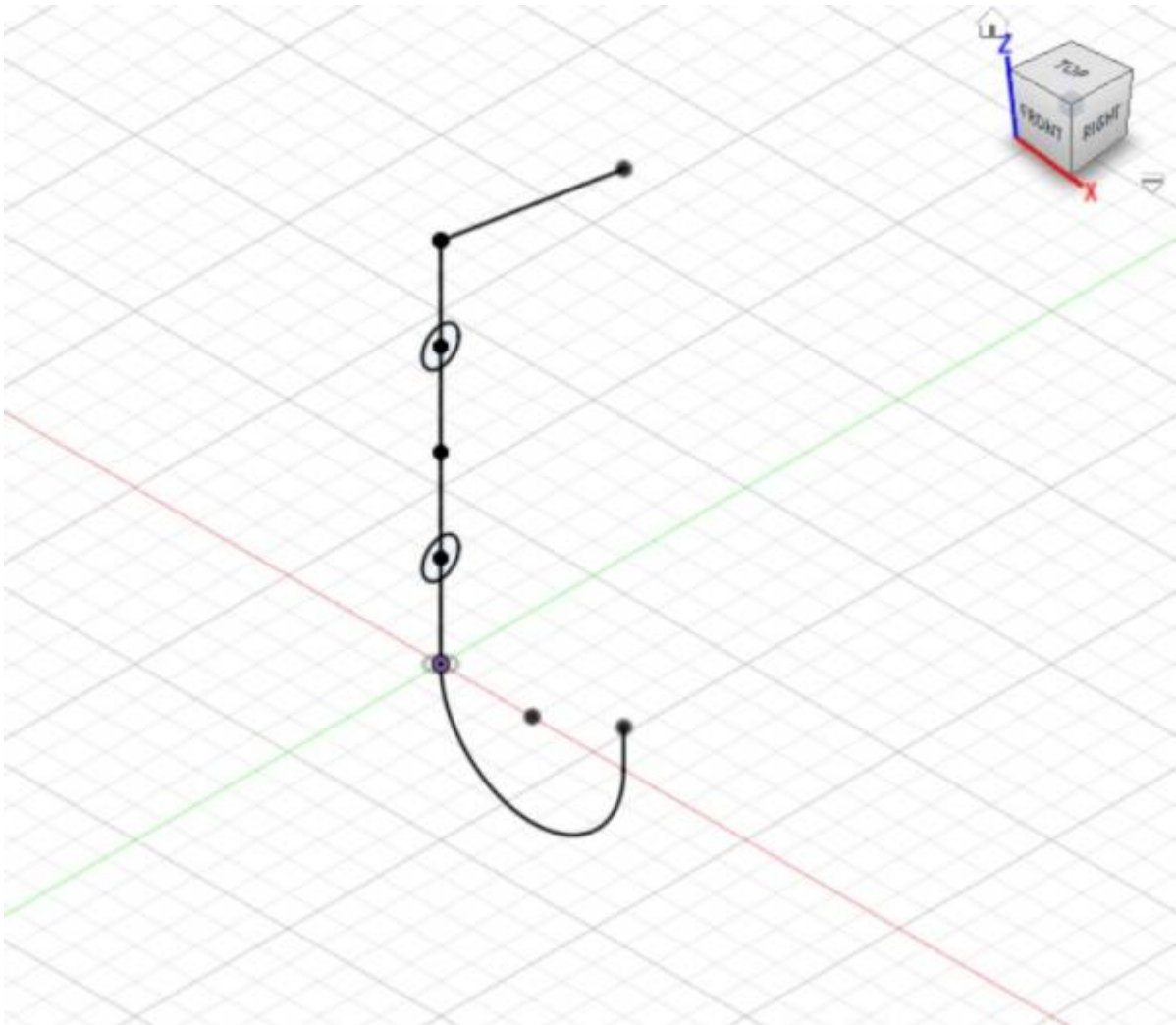Ensure the sketch is fully constrained.
Name the sketch.
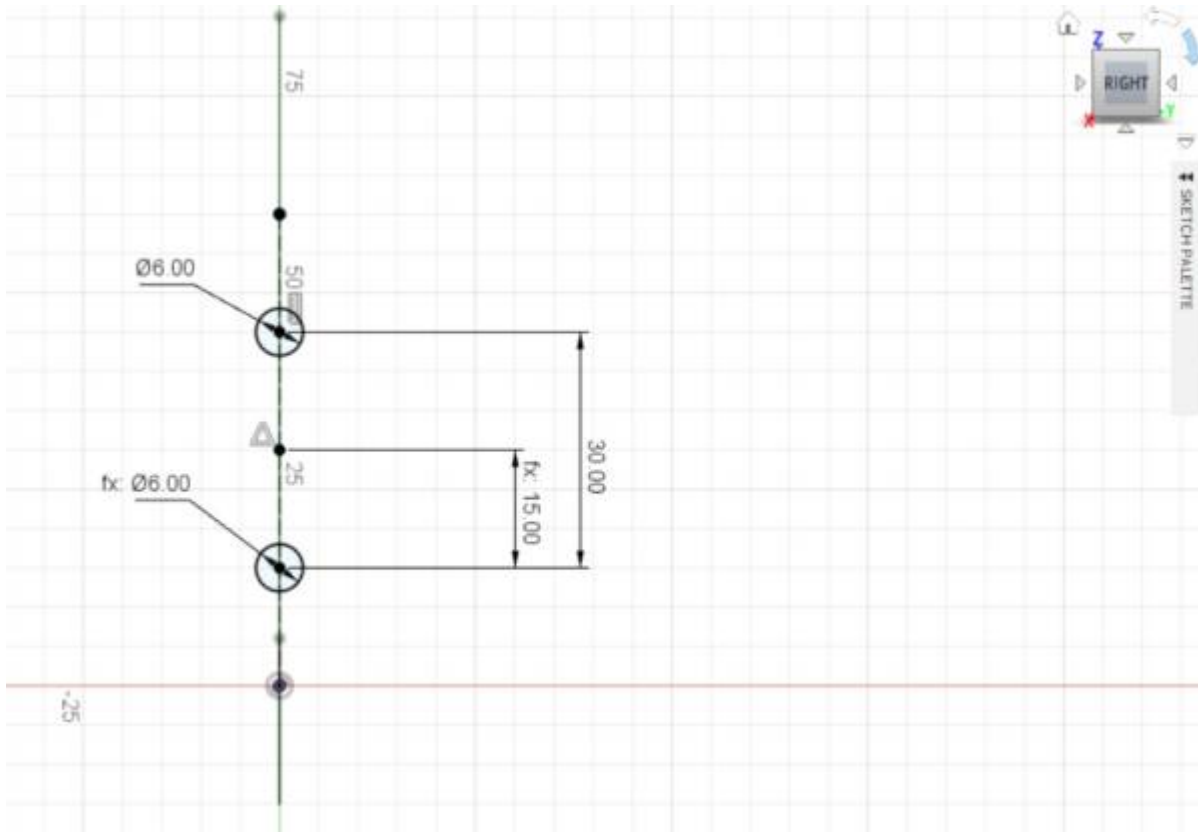
Step 3: Inspect

Use the corner of the Home-cube to inspect in 3D
Toggle Sketches on/off to check progress and alignment.
Hide Sketch #2 for next part.

Step 4: Extrude Profile

Select Extrude tool (e).
Note, you cannot select a profile to extrude.
Change 'type' to Thin Extrude and enter a dimension.
Note: this is a one sided extrude.
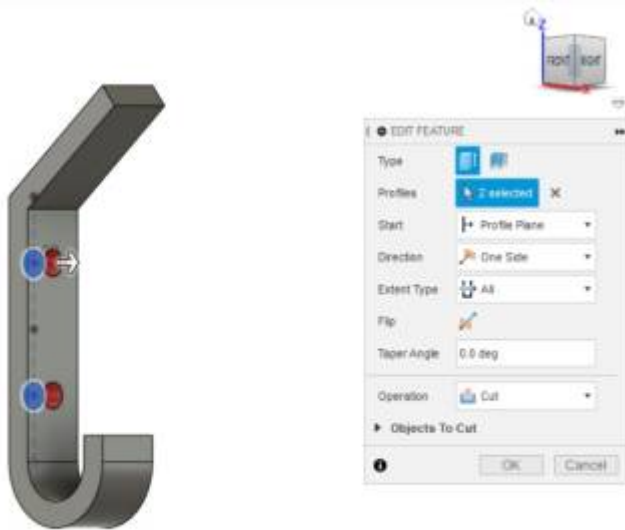Change direction to symmetrical.
Finish.

Step 5: Extrude Cut

The previous sketch will have automatically hidden to make way for your next move.
Show Layout Sketch #2.
Select Extrude tool.
Cut through all.
Finish.

Step 6: Chamfer

We're now up to modifying.
Select Chamfer Tool.
Add chamfer to suit your screw.

Step 7: Fillets

Add fillets, pay attention to the order.

Step 7: Test it and Break it !?

Click File > Save As.
Make a new copy to test and change parameters.
Modify Layout Sketch #1, and add a bounding box to improve scalability.

Export STL

What's an STL ?
This file type represents an object by tessellating it, which means creating a mesh of small triangular shapes.

Your part is now ready for 3D Printing 😃

BONUS TIPS !

If time allows…

#1: Set current view as… #2: Good shortcuts to know

MORE WITH FUSION 360

Encourage curiosity…see how other models are made.
Data Panel > Design Samples > Lamp
Overview of multi-component modelling
Activate components
Demonstrate Sweep tool
Roll back

CAM workspace
Free-form modelling
Sheet Metal

Continue Learning...

Interested in Learning more ? Product Design Online is one of the best resources for Beginners Fusion 360: https://www.youtube.com/playlist?list=PLrZ2zKOtC_-C4rWfapgngoe9o2-ng8ZBr

- Reference Link

Downloads

Add any slide presentations, instructions, software etc. here

If you wish to make a slide deck to include, please follow a similar example of colour and Wiki styling.