

MFG46366

Fusion 360 and 3D Printing: Tips and Tricks for a Successful Workflow

Elizabeth Bishop
University of Warwick, UK

Learning Objectives

- Learn how to use the Design workspace in Fusion 360.
- Learn how to avoid common design mistakes when 3D printing.
- Explore different available 3D-printing technology.
- Explore the additive manufacturing workspace in Fusion 360.

Description

This instructional demo will show you how to take your ideas to reality through the use of design in Fusion 360 software, and how to utilize 3D printing to manufacture your idea. 3D printing (or additive manufacturing) is a great technology not only for prototyping, but also for all the way through to end-use components in industry. 3D printing is now easily affordable for the everyday user, and it can be difficult to know where to start. This talk will show you how to get started with Fusion 360, introduce some different types of 3D printing with the focus on fused filament fabrication (FFF), then go on to cover some common mistakes when using 3D-printing technology and how to avoid them.

Speaker(s)



Elizabeth Bishop is a Postgraduate Researcher at the University of Warwick researching Large-Scale Additive Manufacturing (3D Printing). She has been interested in 3D printing for several years now, following a successful project surrounding designing and making a humanitarian rescue UAV. Elizabeth also volunteers as a Maker in Residence in the Engineering Build Space at Warwick University where she explores making, CAD and CAM alongside 3D printing.

@LizBish94

What is 3D Printing?

The terms Additive Manufacturing (AM) and 3D Printing are often used interchangeably. The standard ASTM 52900 defines AM as “the process of joining materials to make parts from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing and formative manufacturing methodologies.” The standard splits AM into 7 distinct technologies which are summarised really well in the graphic from 3D HUBS in Figure 1 below.

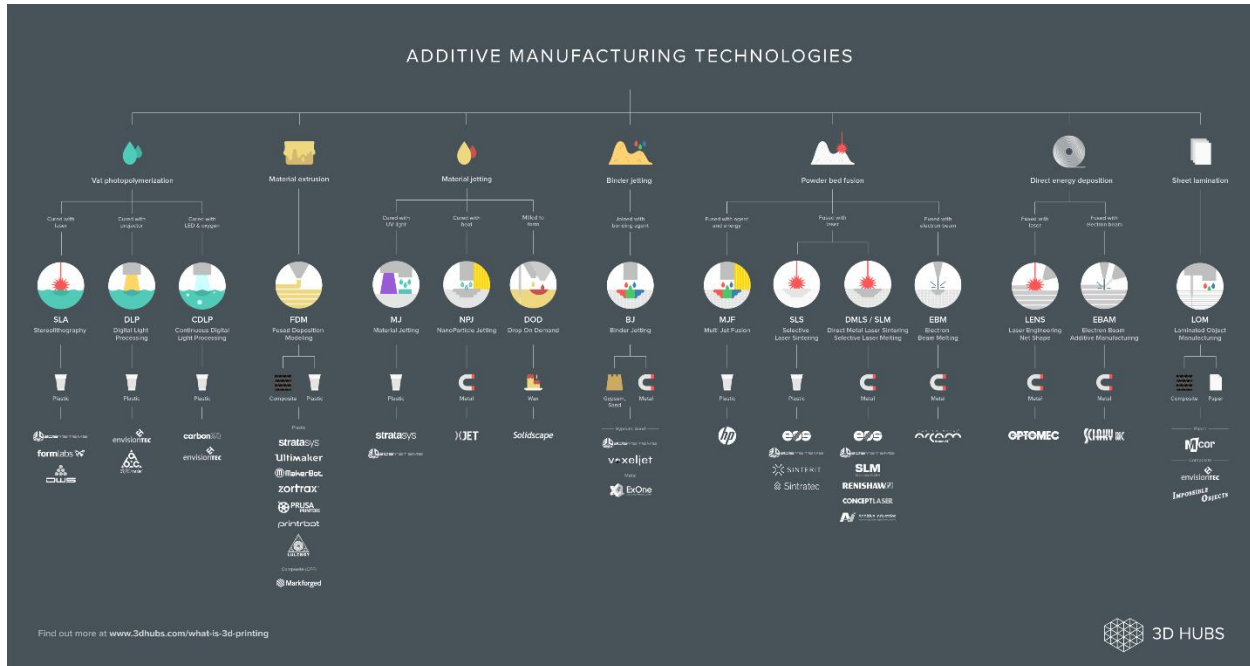


Figure 1: Graphic from 3D HUBS explaining the 7 distinct technologies of AM.

When someone refers to 3D Printing they are usually referring to the branch of AM which covers material extrusion. This technology includes Fused Deposition Modelling (FDM) or Fused Filament Fabrication (FFF).

3D Printing Workflow

The 3D Printing workflow can be summarised as in Figure 2 below. In summary you create a design in a CAD package, save that design as a .stl file, import the .stl into a slicer, generate the toolpaths or gcode. The 3D printer can then read the gcode and this tells it how to manufacture the final part. By using Fusion 360, the steps up to loading the gcode onto a 3D printer can now be completed entirely in Fusion 360 using the Additive Manufacturing workspace.

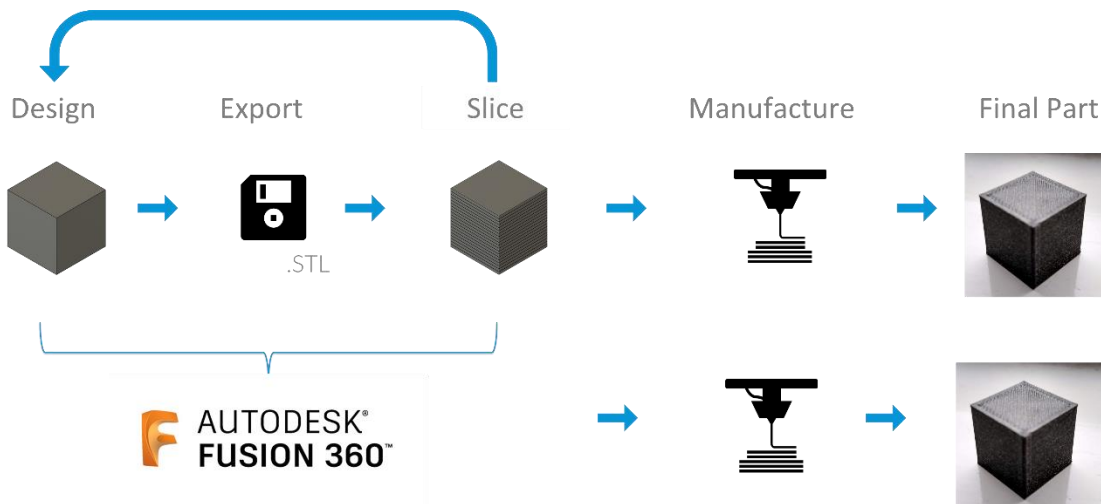


Figure 2: The 3D Printing Workflow

Slicers

A slicer is a program which takes the 3D model and splits it into layers or slices and generates the gcode which can be read by the 3D printer. The slicer allows the user to set parameters such as: layer height; number of shells; infill percentage; infill type; print speed and temperatures. It then calculates the toolpaths necessary based on those parameters.

Design Tips for 3D Printing

As with any manufacturing method it can take time to learn all of the intricate details and best practices. However, when starting out the following are some great tips to follow. They focus mainly around designing to not need support material. Support material is used when prints cannot support themselves, it can lead to waste in both time and material. It can be the same material or a secondary material. In most cases it leaves marks on the underside of your part. There are some really easy changes you can make to your designs to reduce the need for support material.

1. *Change the print orientation*
Try changing the print orientation on the bed to reduce overhangs and increase part stability.
2. *Change the overhang angle*
Try and aim for a 45° angle, this won't need support material. The smaller your layer height the more overhang you can achieve.
3. *Bridging design*
Changing the design of a bridging feature from a flat to a self-supporting arch, or even better a tear drop can remove the need for support material.

My final tip is around feature size, always try and aim for your features to be AT LEAST two times your extrusion width.

3D Printing Workflow in Fusion 360

The 3D printing workflow in Fusion 360 can be summarised below in Figure 2.

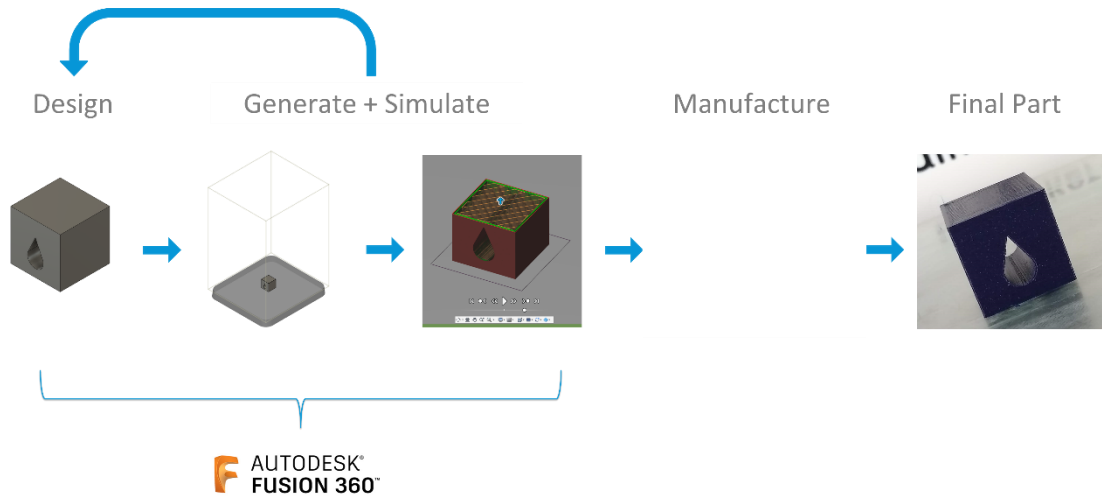
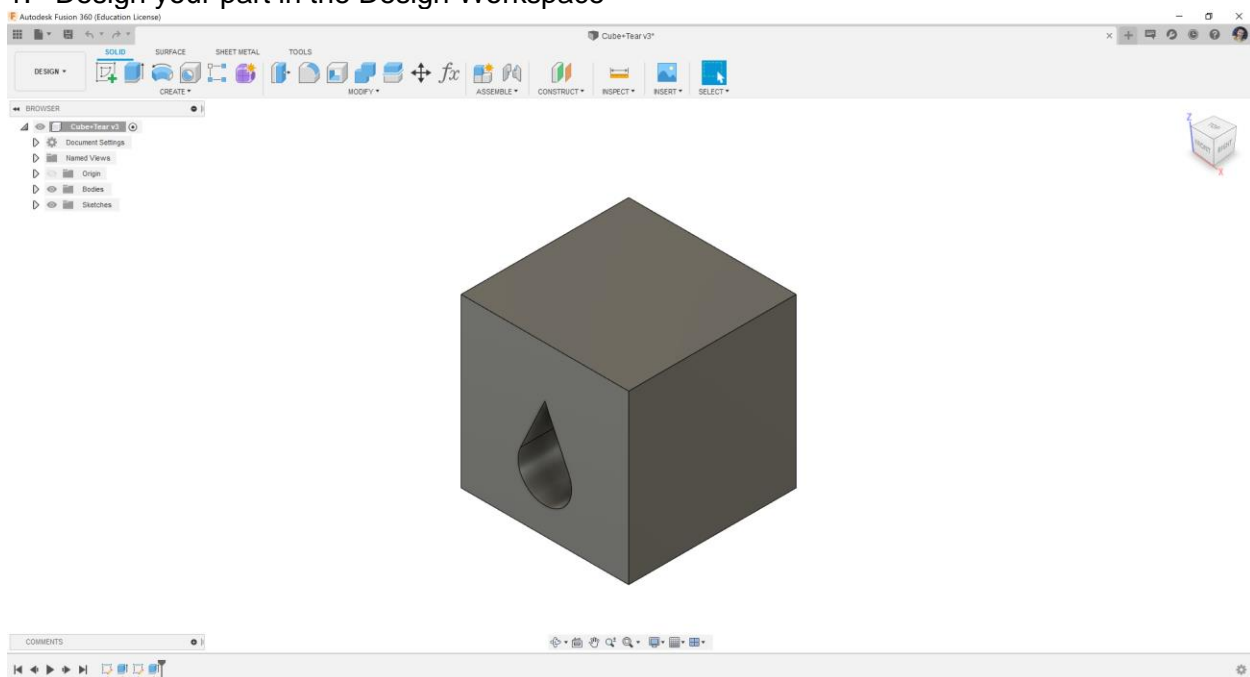


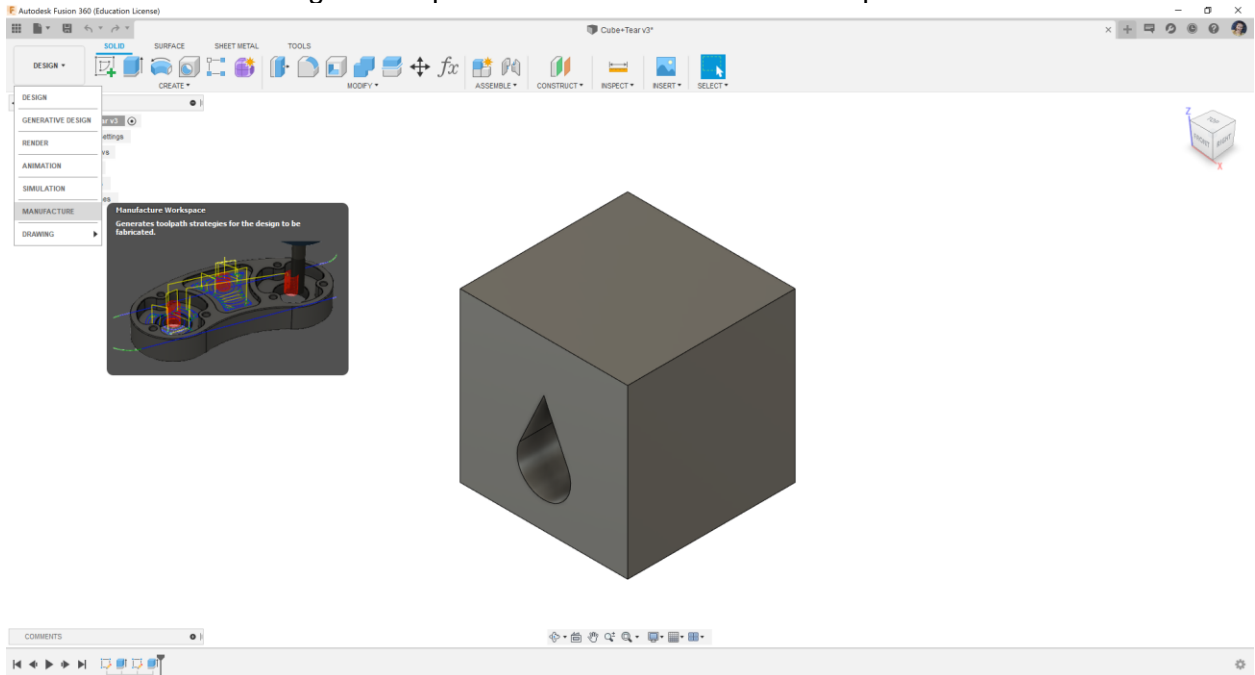
Figure 3: 3D Printing workflow in Fusion 360.

The steps are as follows.

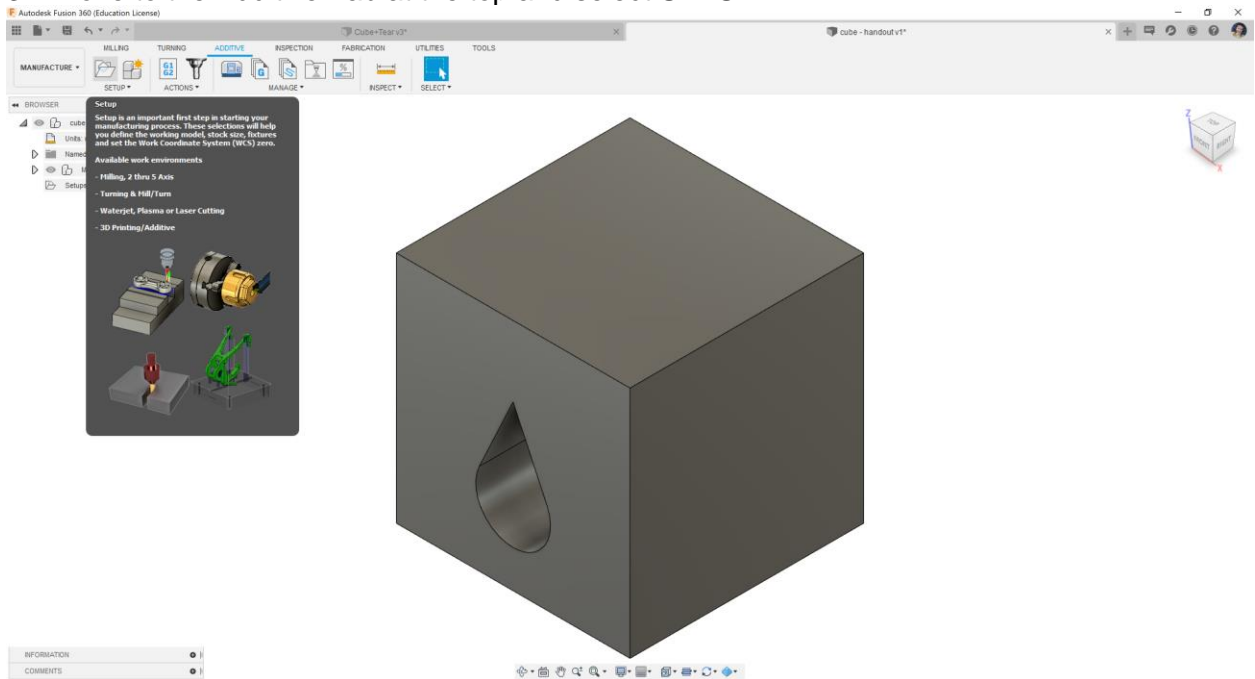
1. Design your part in the Design Workspace



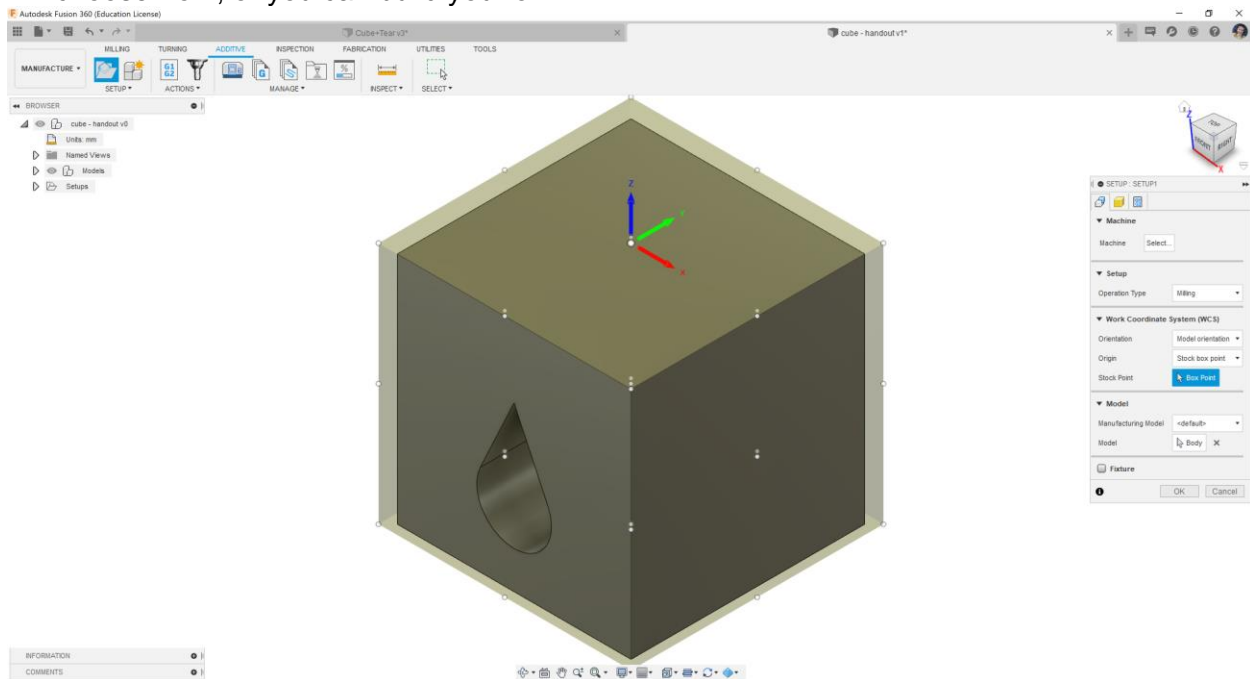
2. Move from the Design Workspace into the Manufacture Workspace



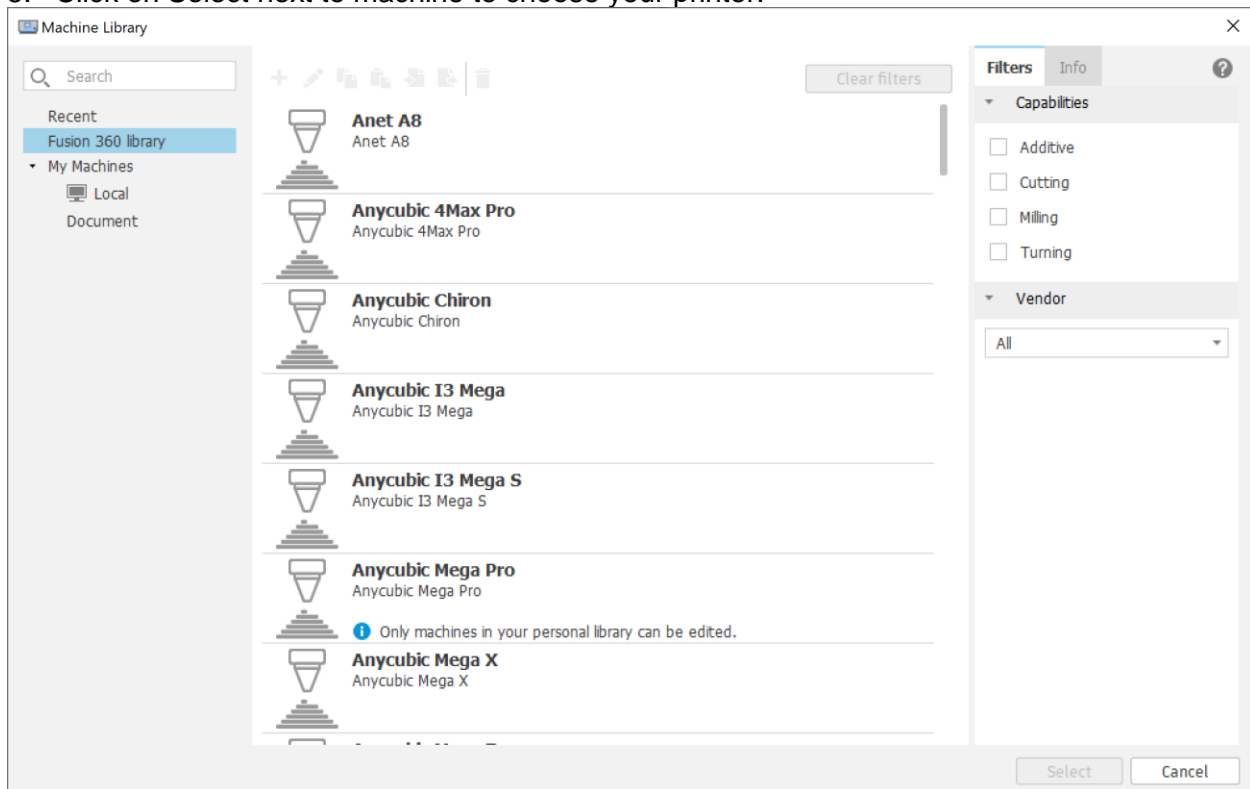
3. Move to the Additive Tab at the top and select SETUP



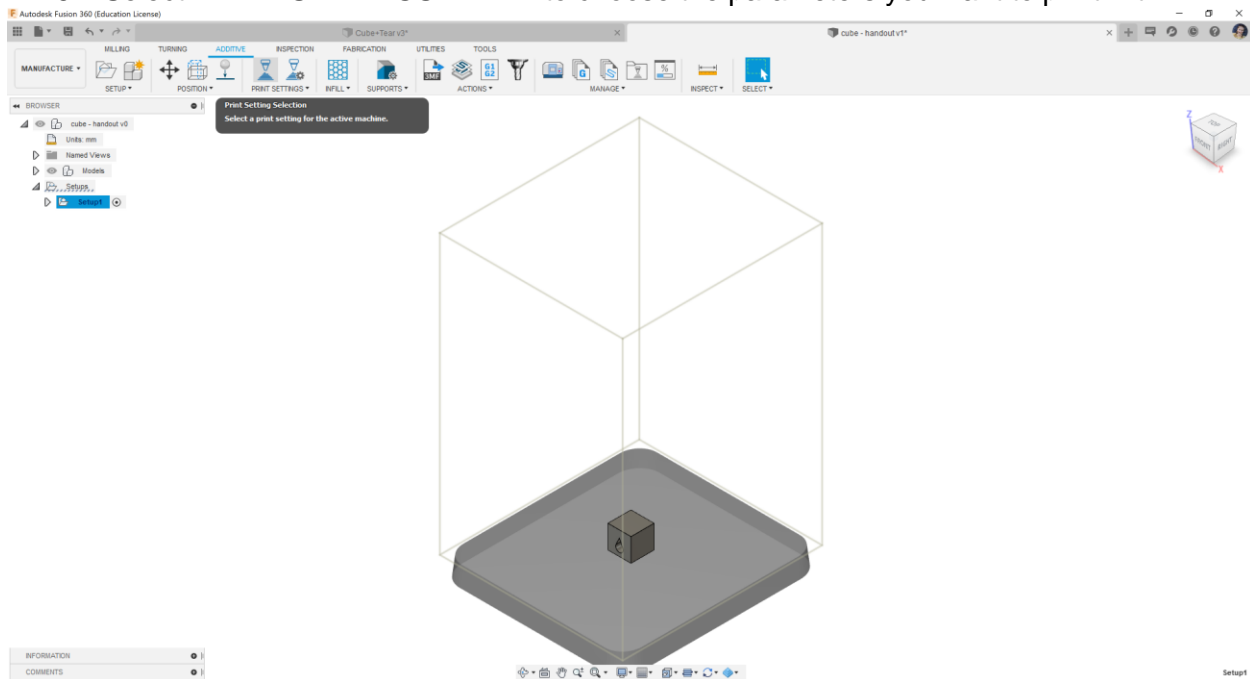
4. Setup your machine i.e. your 3D Printer. There are some built in machines for you to choose from, or you can build your own.



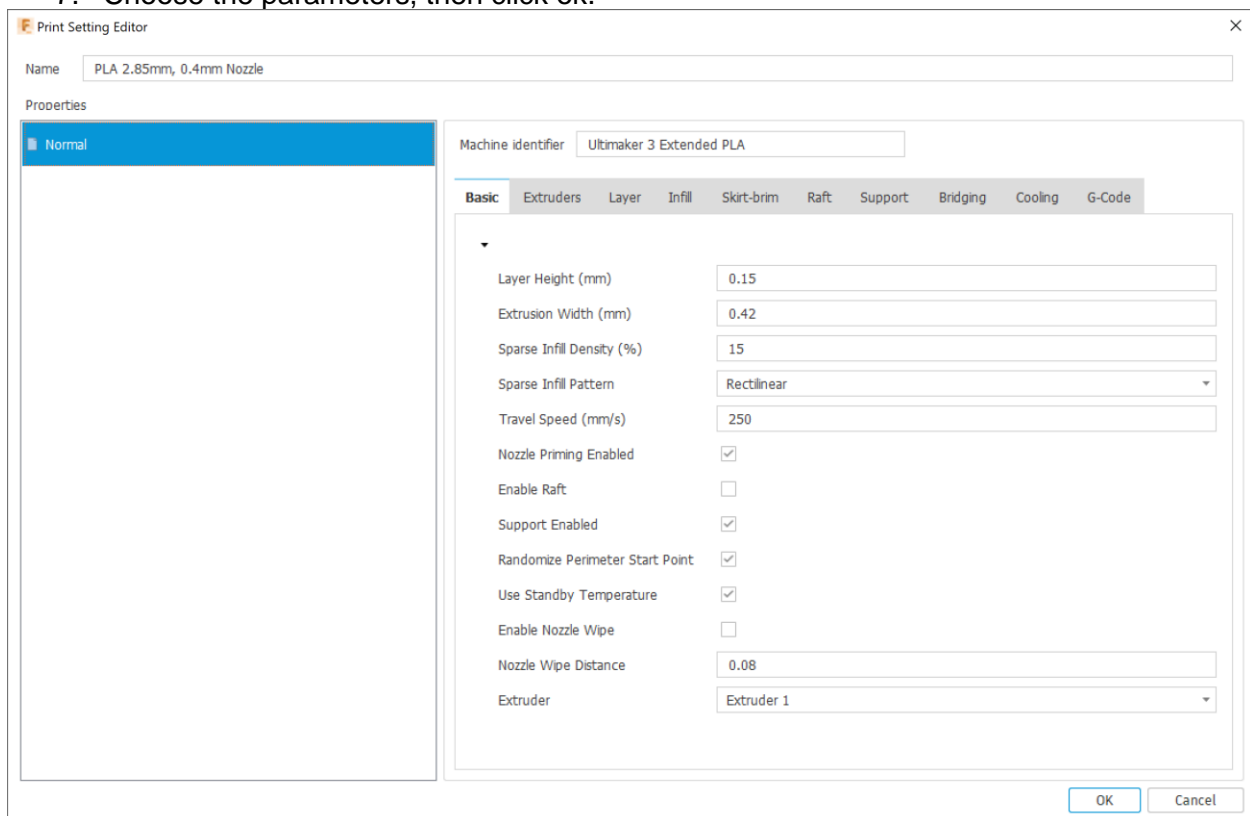
5. Click on Select next to machine to choose your printer.



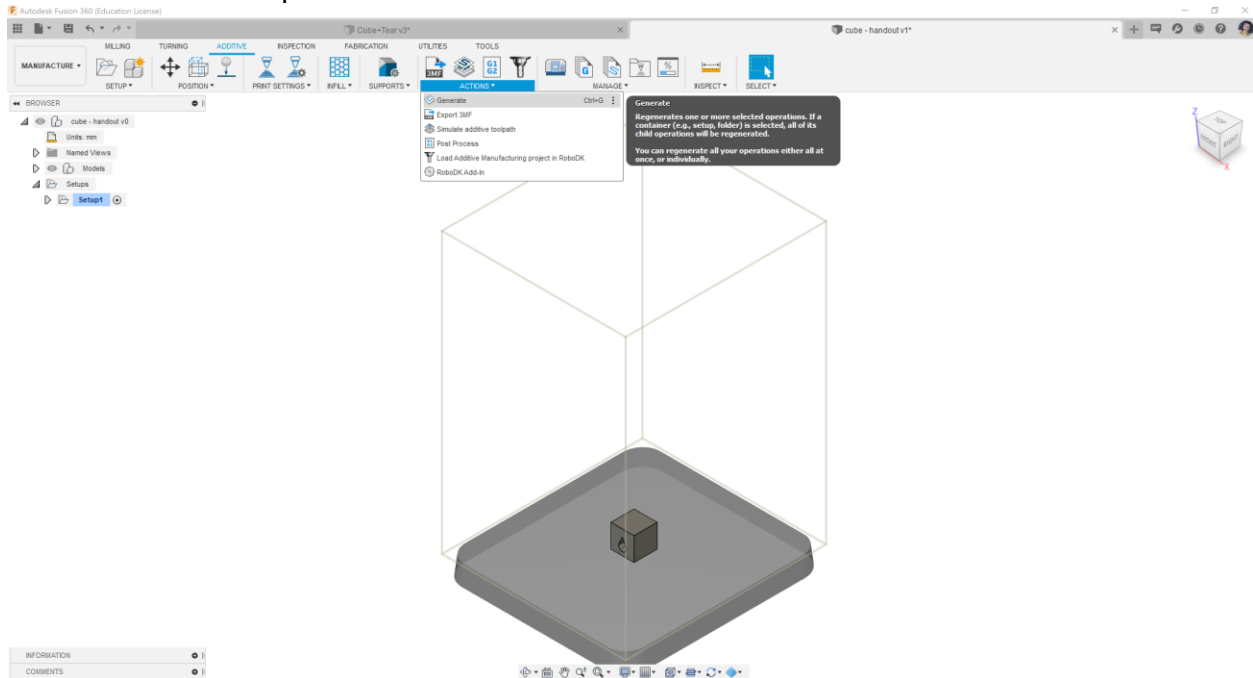
6. Select PRINT SETTINGS > EDIT to choose the parameters you want to print with.



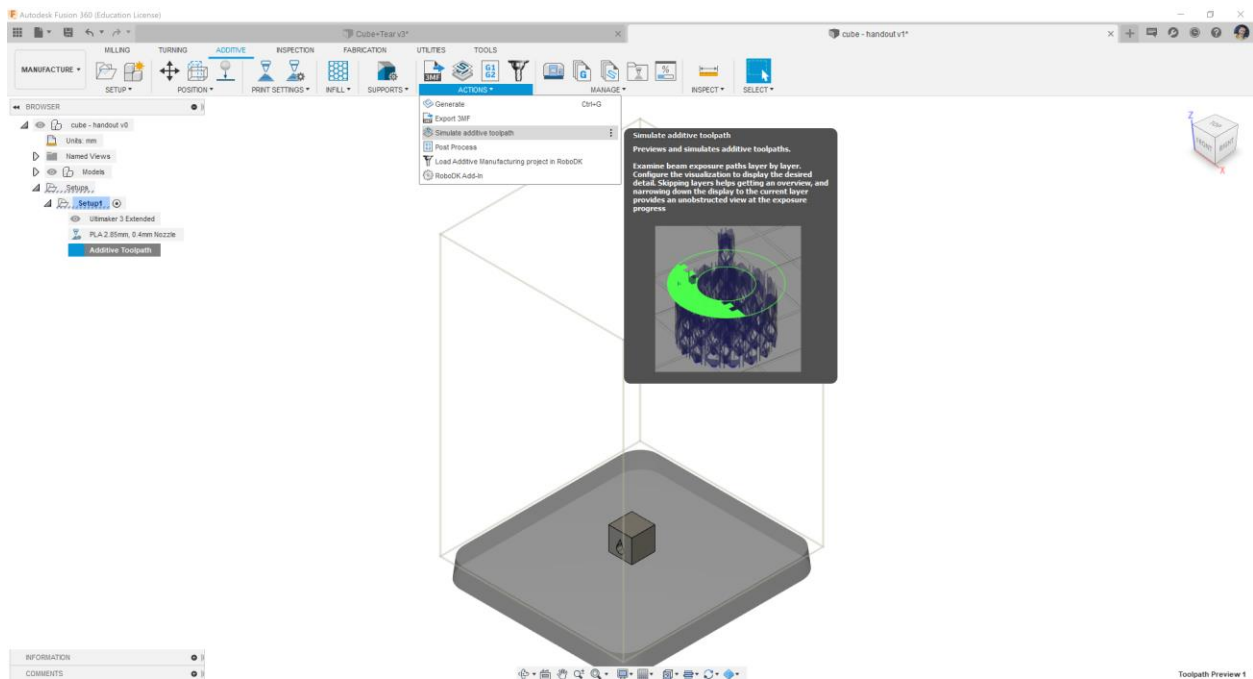
7. Choose the parameters, then click ok.



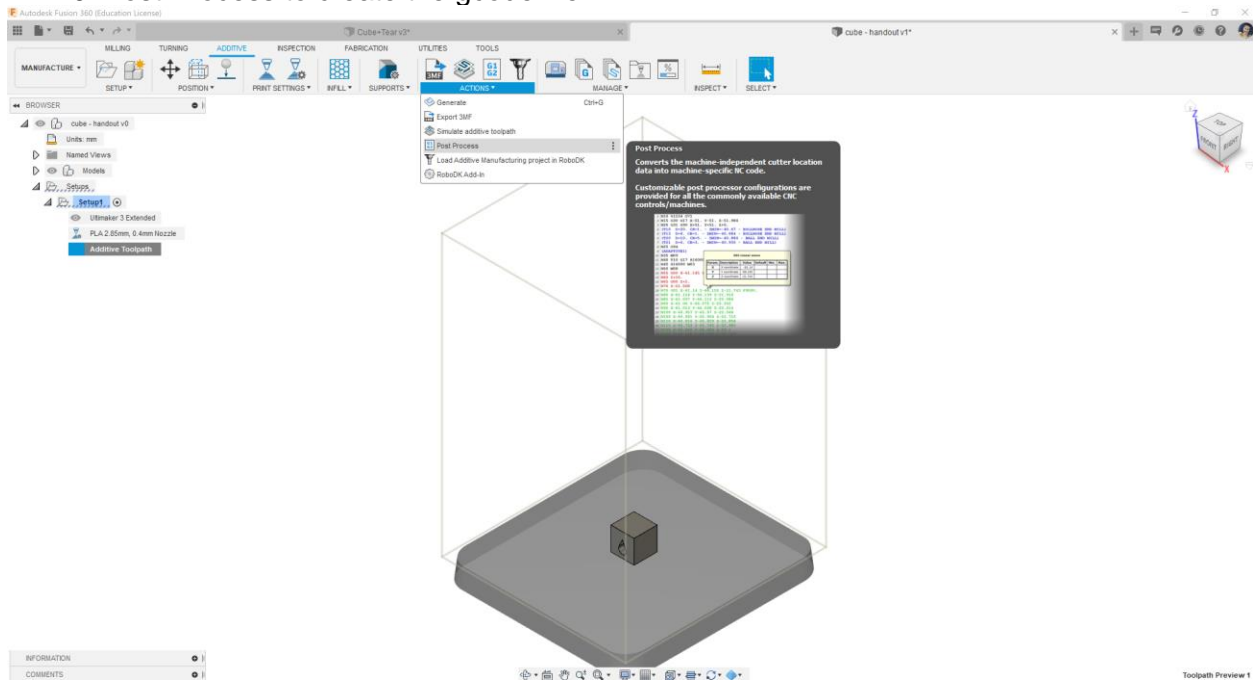
8. Generate the toolpaths



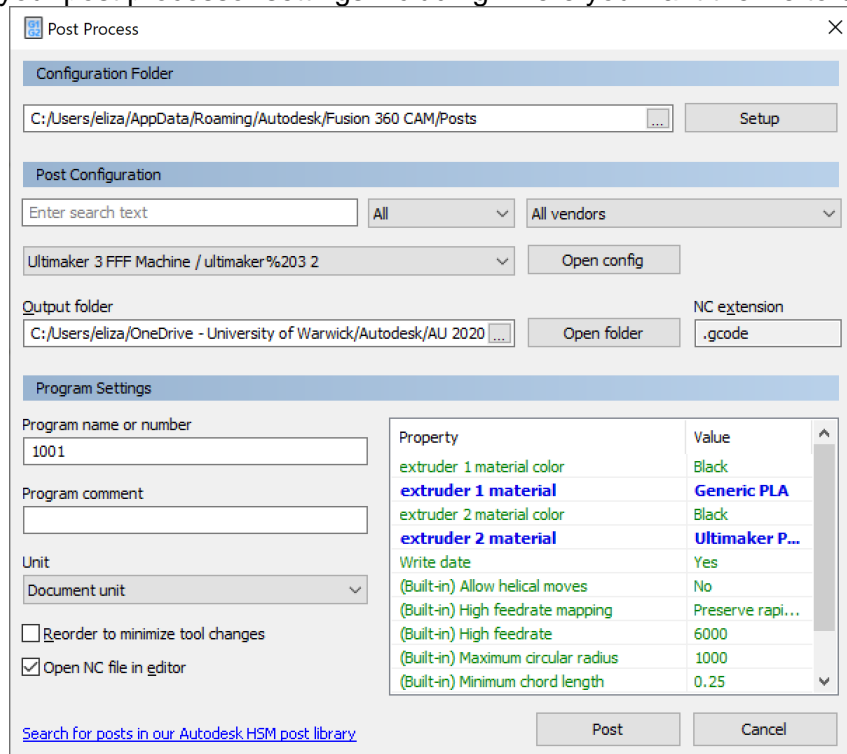
9. Simulate the toolpath to check you are happy



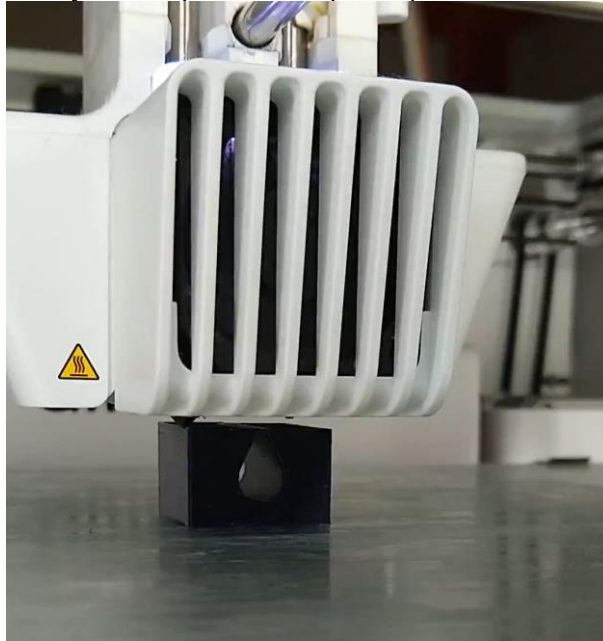
10. Post Process to create the gcode file



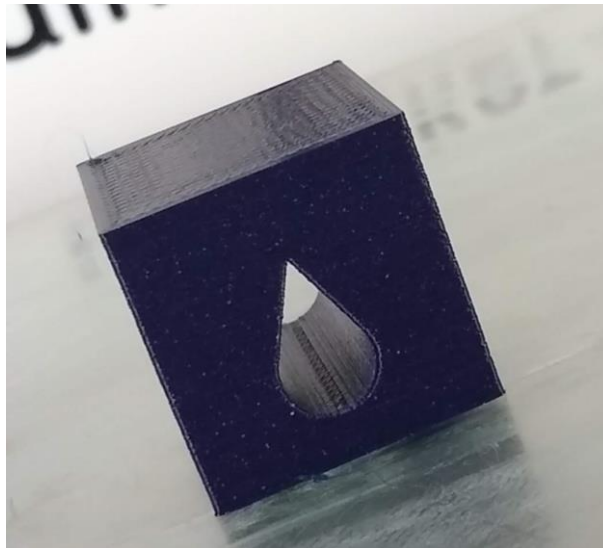
11. Check your post processor settings including where you want the file to be saved.



12. Load your gcode file onto your 3D printer and press print



13. Enjoy your 3D Printed part.



Conclusions

This handout has covered the basics of 3D Printing, and shown a 3D Printing workflow using Fusion 360 to process the toolpaths.