

INTRODUCTION TO 3D PRINTING

 Makerspace Sessions



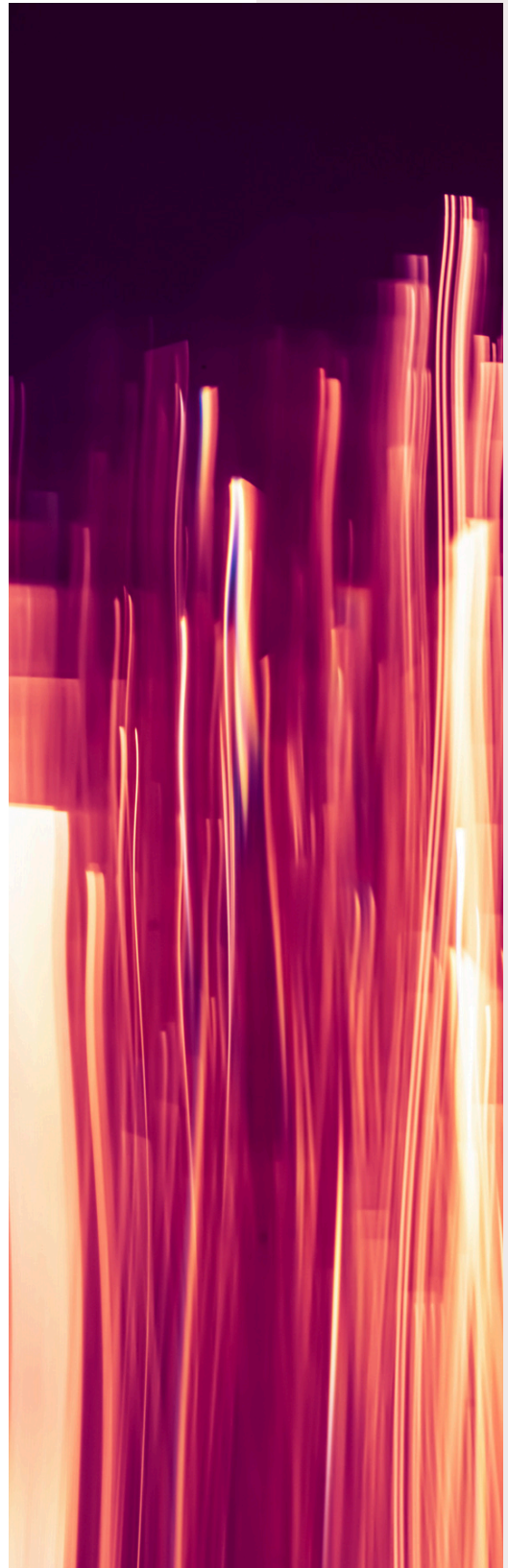
SESSION PLAN

What is 3D Printing?

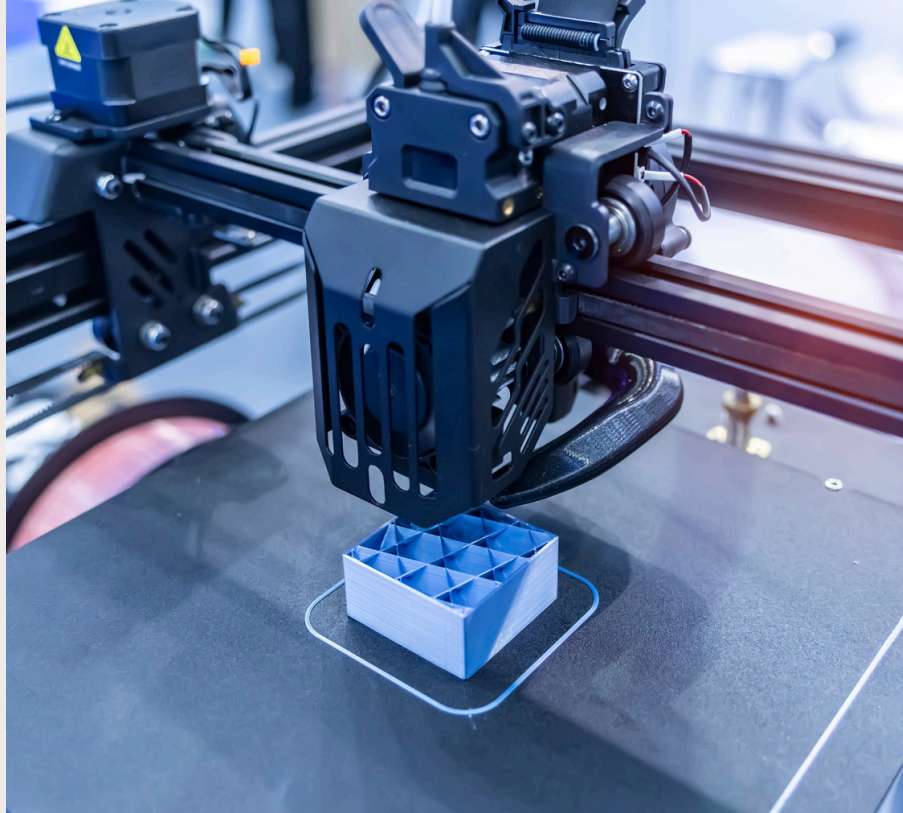
Slicer Tutorial

3D Printing in Makerspace

Further Information



WHAT IS 3D PRINTING?



An Introduction

3D printing turns a digital design into a physical object, whether the object is made with PLA or Resin, the design is broken down into individual layers which the printer completes one at a time until the design is fully formed. At Ipswich Libraries Makerspaces, we print with 2 different materials.

PLA: PLA printing builds objects layer by layer using melted plastic. Once the plastic cools down, the printer continues onto the next layer until the object is completed. This is the primary material models are printed in.

Resin: Resin printing build objects layer by layer using a special liquid that hardens when exposed to UV light in short flashes. This is a more complicated and lengthy process due to safety procedures when handling uncured resin. Consultation with a Makerspace Officer is required before this material is considered for use.

Types of Printers

3D printers are now being used in many areas of industry including medicine, manufacturing and even in culinary fields. There are many different types of 3D printers available on the market and in industry. These printers can vary in size and other factors depending on their intended use.

Ipswich Library Makerspaces use filament printers and resin printers.



Bambu Lab A1 Mini
Filament Printer



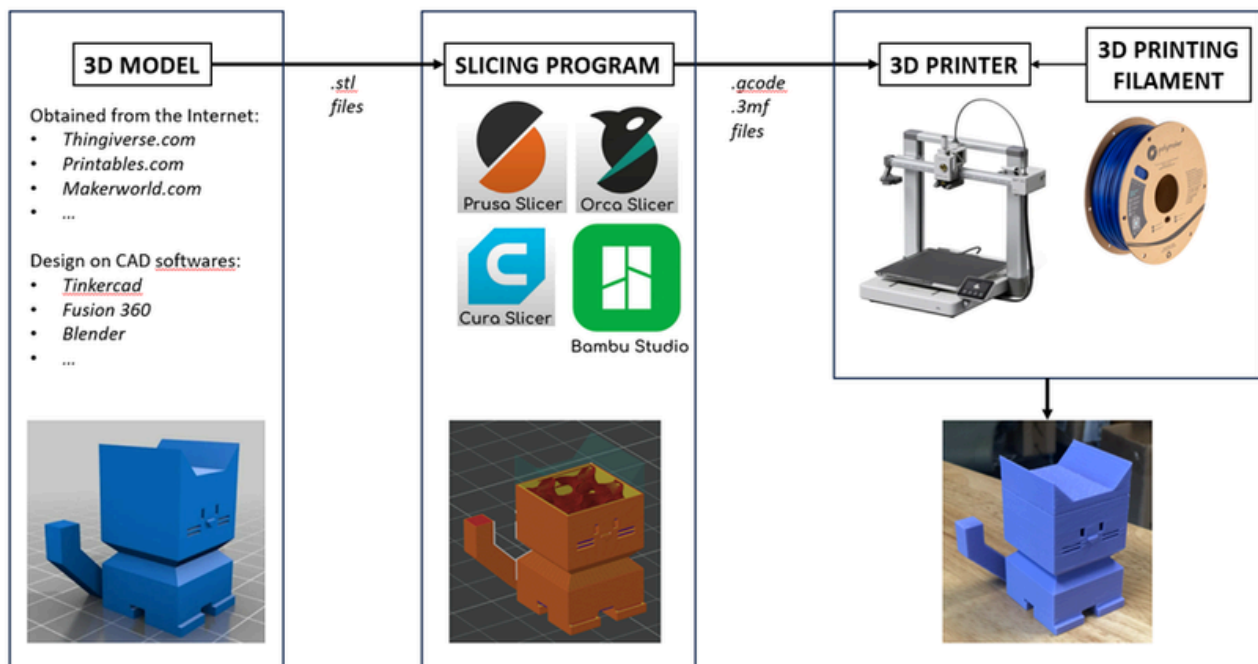
Bambu Lab P1P
Filament Printer

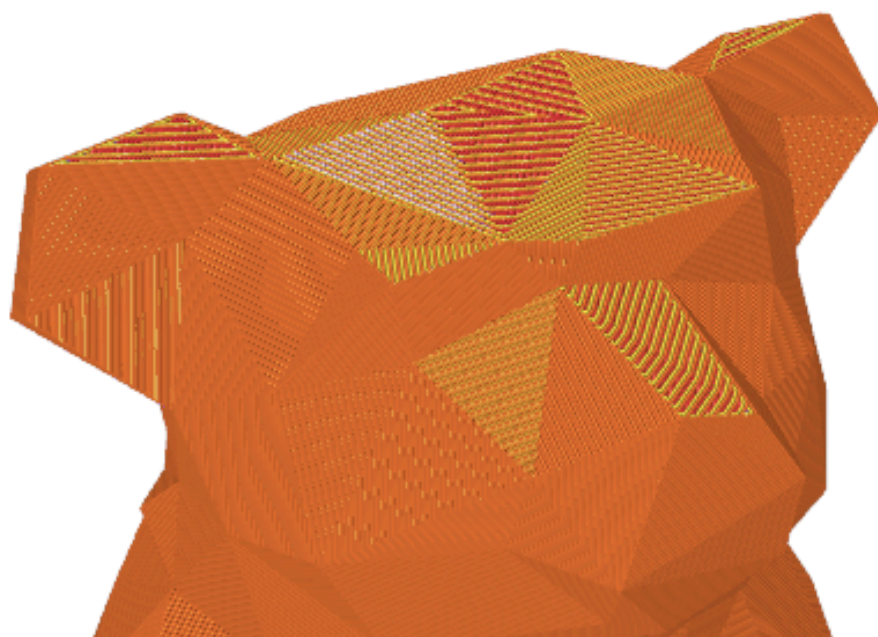


Elegoo Saturn 4 Ultra
Resin Printer

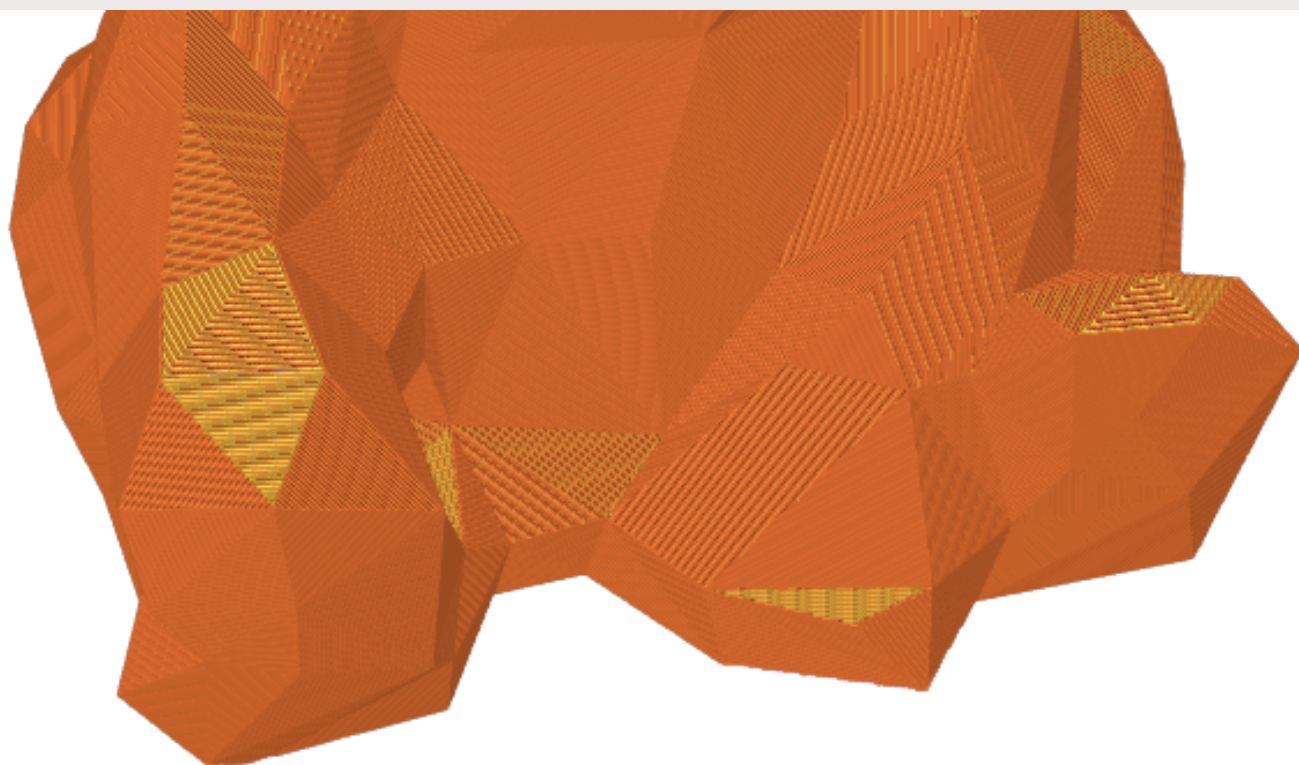
How They Work

The filament is loaded on to the printer and fed into the extruder nozzle. This nozzle heats to between 210 -230 degrees, as per the requirements for PLA filament. The filament is then extruded in the required width (0.4 of a millimetre) on to the plate which is also heated (to between 60 - 65 degrees) as per the requirement of this filament. The printer extrudes the filament onto the plate in layers to make up the desired shape.





SLICER TUTORIAL



What is a Slicer?

A slicer is a specialised program relative to the 3D printer you are using that takes a 3D model and 'slices' it into sections the printer can understand and print in. It is important to note that a Slicer is not a modelling program, therefore you cannot create a model within it but rather make basic tweaks to the model to finally prep it for print. Such tweaks include;

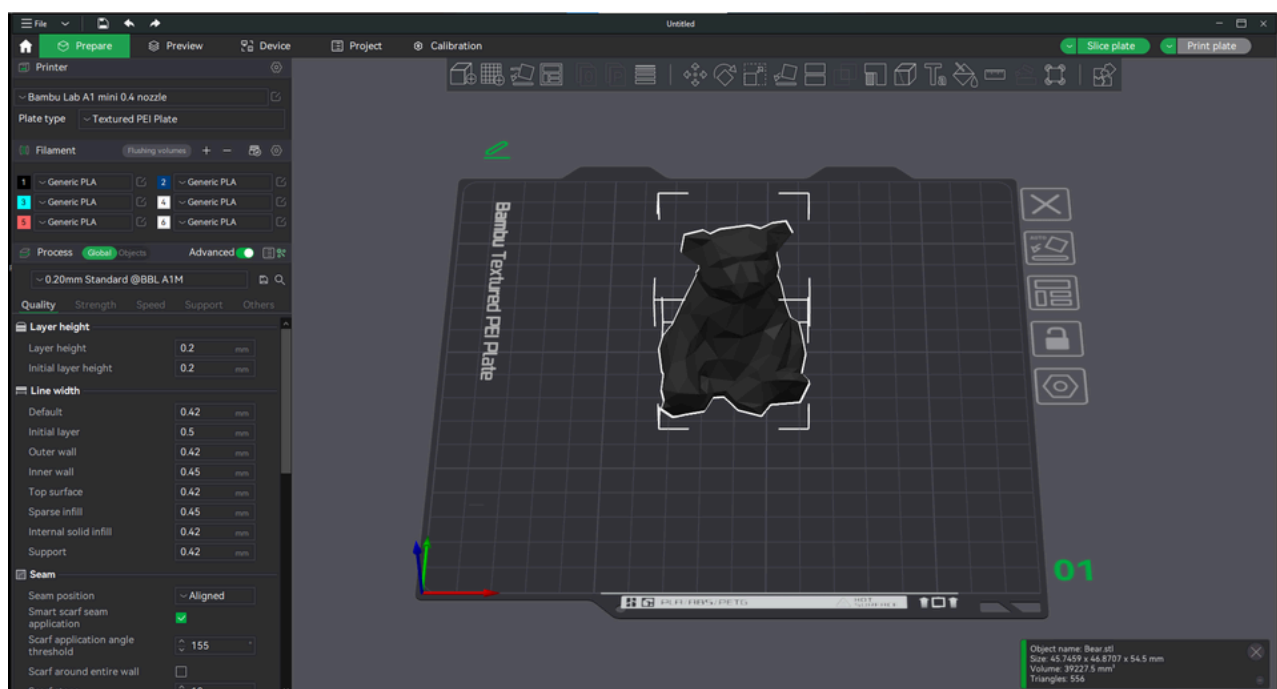
- Changing the size of the model
- Changing the angle of the model
- Cutting the model into more easily printable parts

Slicers generally take 3D models with the file extensions of .STL, .3MF and .OBJ. However, this can vary depending on the slicer and advancements in software.

Though there are many different types of slicers the same terminology and basic functions are transferable between them. This tutorial is based using Bambu Studio.

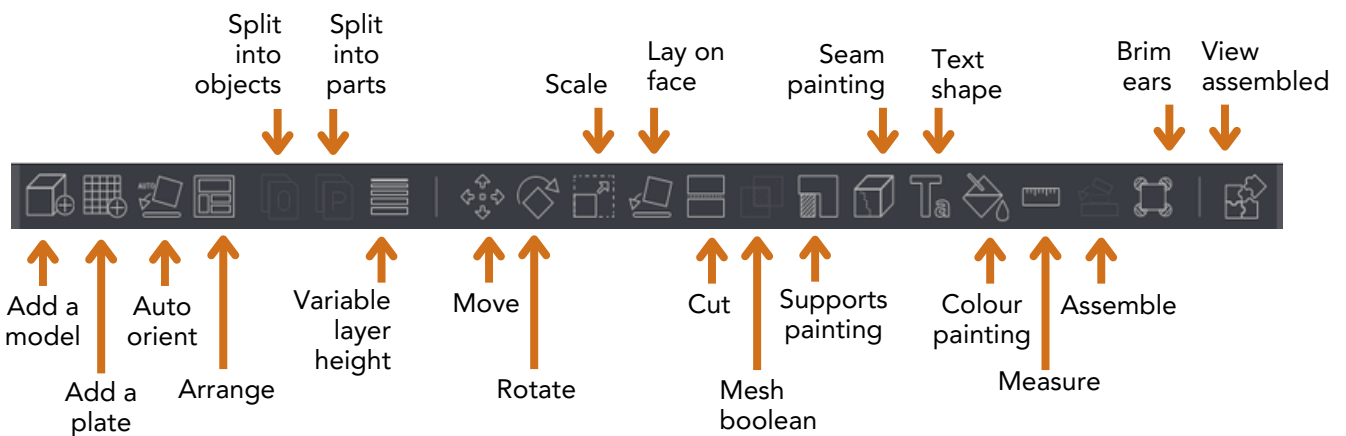
The Makerspace Officer will Import the model you wish to 3D print or if you are at home you can click on the 'add' element in the toolbox or go to 'file', and 'import' your model. Remember that the file must be in a .stl, .3mf or .obj file format.

The Slicer Workspace



The Tool Bar

In the top bar of the space you will see the Tool Bar. This is where you can import your model and make adjustments via the tools available. Explore the tools to better understand their use.



Navigation with the Mouse

You can navigate the workspace by using your mouse.

LEFT MOUSE BUTTON

This moves your model when moused over it. Also rotates the plate on all axis by clicking on the plate and dragging.

RIGHT MOUSE BUTTON

By clicking on the plate and dragging you can manoeuvre the plate without changing the axis of the plate



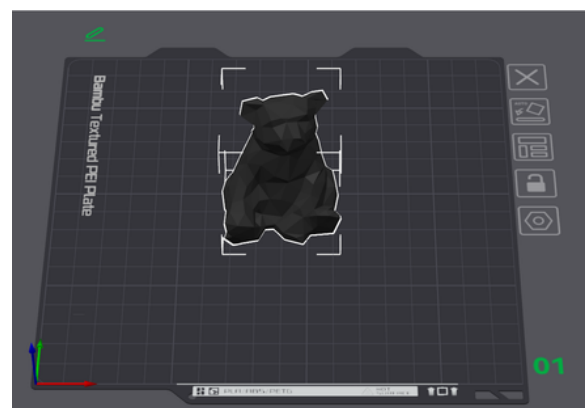
MOUSE WHEEL

Rolling up and down will zoom in or out of your model

The Plate or Bed

You will notice that your model sits upon a grid square. This is called the plate or bed. This area size corresponds to the exact plate size of the printer you have selected to use (see the very top of the left hand menu window).

You can navigate around the plate by using your mouse. Note that you can not lift your model to float above the plate, have it too big or over the edge of the plate.



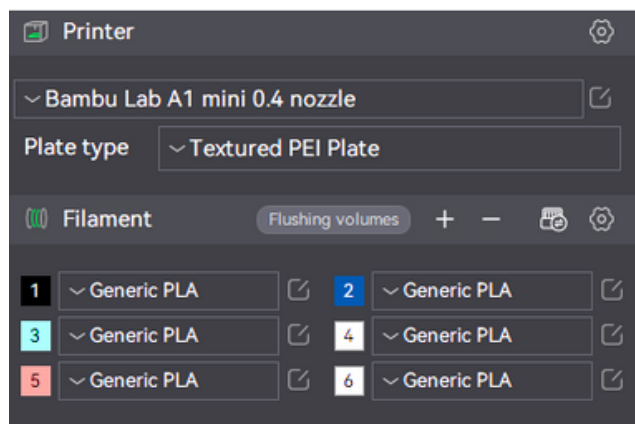
Preparing Your Model

In the left hand menu window you will see a variety of options. We will go through these options step by step to prepare the model

1 The Printer Details

You can choose which printer you are printing with from the Bambu selection of printers. Selecting the right printer will also change the size of the plate to suit the printer. It also details the nozzle size.

Below this you can also stipulate what type of plate you are using. We will be using a textured plate.



2 Filaments

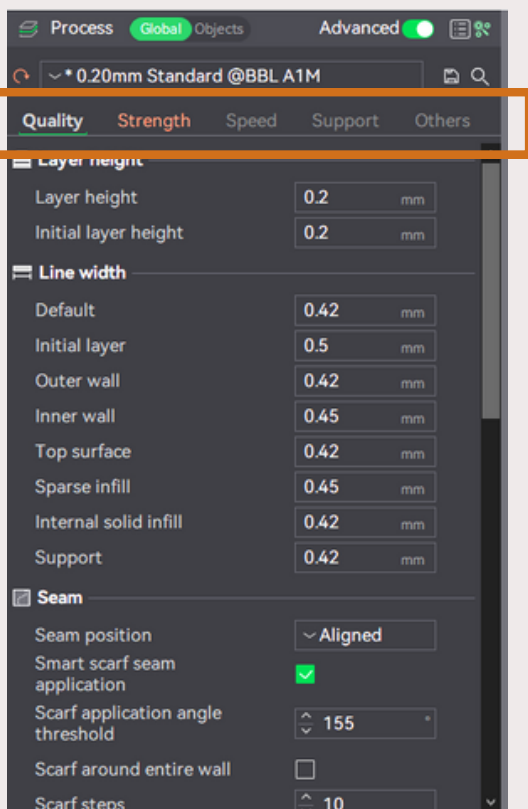
For the purposes of fairness we only print models in one colour. That said it is possible to print in multiple colours if your printer has more than one spool slot.

In this selection you would identify which colour and type of filament is in each slot from left to right. We use PLA or Poly Lactic Acid filaments that are biodegradable. These filaments become malleable at 60 degrees and are extruded out at temperatures between 210 and 230 degrees celsius.

Process Section

This section is where much of the intricate detail regarding the printing of your model will take place.

You will notice 5 tabs; Quality, Strength, Speed, Support and Others. We will work through these tabs to ensure the model is ready for print.



3 The Quality Tab

In this tab you can define layer height, line widths, seams, ironing, wall generation and advanced setting for this tab.

We will only cover the main elements of layer height, line widths and seams in this tutorial.

Layer Height

A 3D printer builds a model up from its base like a sandwich one piece of bread at a time. The layer height is the height of each layer as it builds.

Smaller layer height results in smoother surface finish and higher detail, which can increase the print time.

Thicker layer height generally gives rougher surface but helps print faster, which is better for quick prototyping.

For a 0.4mm nozzle, we can set the layer height from 0.08mm to 0.32mm. Our default layer height is 0.2 of a millimetre.

Line Width

This is the width of each line it draws or extrudes. This can be altered depending on your model and what you require.

It is important to note the terms 'wall' and 'infill', which will be covered in more detail in the next tab.

Seam

On some models you may notice a line along your model that may look out of place like a seam line in a stuffed animal toy. The placement and visibility of this line can be altered using this section.

4 Strength Tab

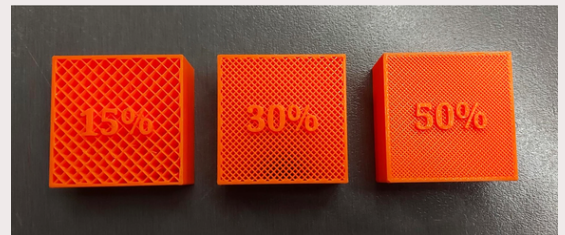
Shells/Walls

Also known as a perimeter, it is the outer edges of your model. You can adjust the thickness of these as you see fit. For example if you would like to print an aeroplane it is best to increase your shells thickness in order to strengthen the thin wings for the craft.

Infill

It is like the fleshy insides of your model. This can be altered by percentages. You can also alter the pattern desired for the infill. Although you will not see it on completion it can play a role in the dexterity and time taken in completion of your model.

The maximum percentage density of infill recommended by manufacturers is 50%. This is because anything more than that is considered a waste of filament. In saying this, it is possible to print in 100% infill but it will substantially increase the weight and print time of your model.



5 Speed Tab

You can adjust the speed of each section of your model if you desire. However, it is important to remember the below formula when deciding on changing these elements.

Faster speed = less detail + less time
Normal or slower speed = more detail
+ more time

You can reduce the time of your print by altering the print speed of your infill and inner walls. This will not compromise your external detail.

6 Support Tab

Supports are an external scaffold that builds up to support any overhanging part of your model. For example if you have a model of a person with their arms outstretched you will need to have supports to be able to print the arms. There are many different types of supports you can use for different purposes depending on your printer software. Supports are easily removed from prints on completion of the model.

For this model the options are:

Lattice supports (Normal) – are good for many models particularly mechanical, geometric and those with minimal detail. This form of support can use more filament than the below and can leave more noticeable support marks on more rounded or organic style models.

Tree supports (Tree or Organic) – are good for more organic shapes such as people, nature and models where you want minimal support marks on your model. This form is also more economical on your filaments as only the specific support point is targeted.

Raft

A raft is a type of support that provides a strong layer of filament that the printer can put around a model to increase the model adhesion to the bed. It is best used on thin tall models and models with delicate points that touch the bed. An example of this would be a ballerina on the point of her toes.

If you make the raft of your model too thick it can be difficult to remove and end up acting more like a plate connected to the model.

7 Others Tab

Bed Adhesion

This tab refers to assisting in helping tall, delicate, thin or unusually balanced models to remain stable throughout the print process and avoid failure of the print. The two types of elements you can use are:

Skirt

A skirt is designed to increase adhesion to the plate or bed and is not connected to the model in any way. Instead, it forms a ring around the model base.

Brim

A Brim is designed to provide a thin layer on the plate around the print to prevent warping of the model. This is printed connected to the print and is easy to remove. Warping can occur when using different filaments such as PETG and ABS filaments.

Slicing Your Model

Now that you have gone through the details it is time to slice your model. This option is available in the top right hand corner in the drop down menu select 'Slice Plate'.

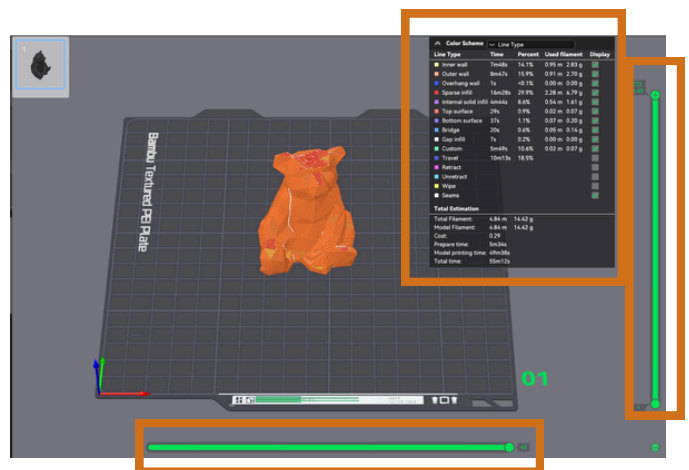


8 Viewing the Sliced Model

In this mode you can see the model as it will be printed on the printer including and supports, skirt, brim or rafts you have selected. In the top right corner you will also see a breakdown of each print element and how long it will take and how much filament they will use.

At the bottom of the window you can see how much filament overall the model will use, cost of filament used and the time it will take to print.

You can also view each layer of your print and the path of the extruder on each layer by adjusting the nodes on the vertical and horizontal green bars.



If you wish to make any further changes or amendments to your model at this time you can click on the 'Prepare' tab at the top left of the screen to do so. Remember that you will have to slice your model again until you get it the way you want it.

 Prepare

 Preview

9 Exporting to G-code

Once a model is ready to be printed it is exported as .gcode file. Now your model is ready to be sent to the printer. Go to the top right hand corner and choose 'Export Plate Sliced File'. In other slicers this might be called 'Export' or 'export to gcode'. Clicking on this option converts the model file to a .gcode file that is recognisable only to the printer you programmed it to and will have saved it in a language the printer can understand and work with. For Makerspace, all models are prepared and sent to print by the Makerspace Officer. Please keep reading for more information on 3D printing in the Makerspace.

If you are looking to have your model printed by an external printer you will need to have your model in .stl, .3mf or .obj for them, not .gcode. This is because every printer has it's own specific requirements and will not recognise a gcode from another slicer.

Thank you for Participating!

Thank you for attending Ipswich Libraries 3D printing session. We strongly encourage you to participate in our 3D modelling session as well, to learn how to make your own 3D models for print. Please go online or see staff for more details.

Other common links you may be interested in are:

Thingiverse.com

A website where you can begin your journey to procure free models for print. There are other websites as well but remember we don't print paid prints at Makerspace.

Tinkercad.com

Where you can sign up and create your own classroom for free and learn the basics of 3D modelling (see staff about our 3D modelling session hand book on this).

Makerspacechampions@ipswich.qld.gov.au

Our email for any requests, changes, feedback or questions.

3D Printing in Makerspace

Ipswich Libraries Makerspace afford 4 free prints a year to those whom complete our 3D printing or 3D modelling sessions.

We also highly recommend you keep your knowledge up to date by regularly participating in our 3D print challenges.

All prints are subject to a size restriction of 6cm cubed or the equivalent thereof, at the discretion of staff, per model.

All models will be printed using a singular PLA filament, unless the model comes in more than one piece or has been appropriately discussed and agreed to by the Makerspace staff. This is for fairness for all our customers.

Occasionally specialty filaments may be available for a limited time only, see Makerspace staff for more information.

Educational Opportunity not a Service

Our sessions and prints are free as council aims to create learning opportunities that are available and affordable to our community. As such Makerspace is not a printing service and we do not accept print requests from those whom have not completed one of our sessions. If you would like to have your models printed professionally there are a number of businesses that can accommodate this.

The Print Queue

Once you have selected or created a model and it has been accepted for print your model will be placed in our queue. Your model will usually be printed within a 2 week time frame, however, busy times like school holidays will increase this time period.

Upon completion of your print we will notify you that it is ready for pick up via the information you have given us in your session booking.

If these details change please notify makerspace as soon as possible through: makerspacechampions@ipswich.qld.gov.au

Requesting a Print

You can request a print at anytime once you have completed one of our 3D sessions. All print requests are added to the print queue and will be held to the same size restrictions.

Should there be any issues with your request we will notify you and encourage further discussion on alternative options.

To request a print all you need to do is email us at makerspacechampions@ipswich.qld.gov.au and include:

- Your name
- When you had your session (can be found in your email as we send you a confirmation email) and the name it was booked under
- A link to the model or have the file attached
- What colour you would like it in

Once we have this information we will be in touch to advise you of your requests status.

Please note that for copyright reasons we can not print files that have been purchased by you.

We will advise you if your colour choice can not be done or there are flaws in the model that make it unsuitable to be printed.

Resin Printing

Though we do have resin printers here it is not offered as our main material for printing. This is due to the extra time and procedure it takes in washing, curing and completing this type of material. As such this means that the time for you receiving your model is 4 weeks or more depending on school holidays and any unforeseen factors.

Should this be a preferred type of material you want your model printed in you will have to list your reasons why in your request. As this is a free service you are not guaranteed any specific material other than PLA for your print.

Personal Filaments

We do not accept personal filaments of any kind. A list of colours can be provided to you by Makerspace staff.

FURTHER INFORMATION

Ipswich Libraries Makerspaces offer a whole lot more!



3D MODELLING SESSIONS

Learn the basics of 3D Modelling! During this session, you will be able to learn how to make your very own 3D models. You can then use your new skills to learn more complex 3D modelling programs for animation, concept design and more! Once your model is complete, it will be added to the 3D Print Queue and you will be able to collect it once contacted by Makerspace staff.

(Suitable for adults and children, 8 years and up)

Duration: 45 minutes · Price: Free

Available at Springfield and Ipswich Makerspaces



MAKERSPACE CHALLENGES

Compete in the ultimate Makerspace Challenge to test your creative skills! Challenges change regularly and can be completed using the technology and software currently available in the Ipswich Libraries Makerspaces. Challenges are held in 2 different mediums:

paper crafting and 3D modelling

View the current challenges online or come into a makerspace to find out more! Price: Free

Available at Springfield and Ipswich Makerspaces



ASK - A - CHAMPION

Want to find more about creativity and innovation in the Makerspace? Want to learn more about our 3D Printers? or just have a general question or comment about the Makerspaces? Get in contact by clicking on the Ask-a-Champion link on the website or typing in the link below and a Makerspace Champion will be in contact with you.

<https://www.ipswichlibraries.com.au/ask-a-champion/>

EXPLORE MORE POSSIBILITIES!

<https://www.ipswichlibraries.com.au/ipswich-libraries-makerspaces/>



**IPSWICH
LIBRARIES**

MAKERSPACES