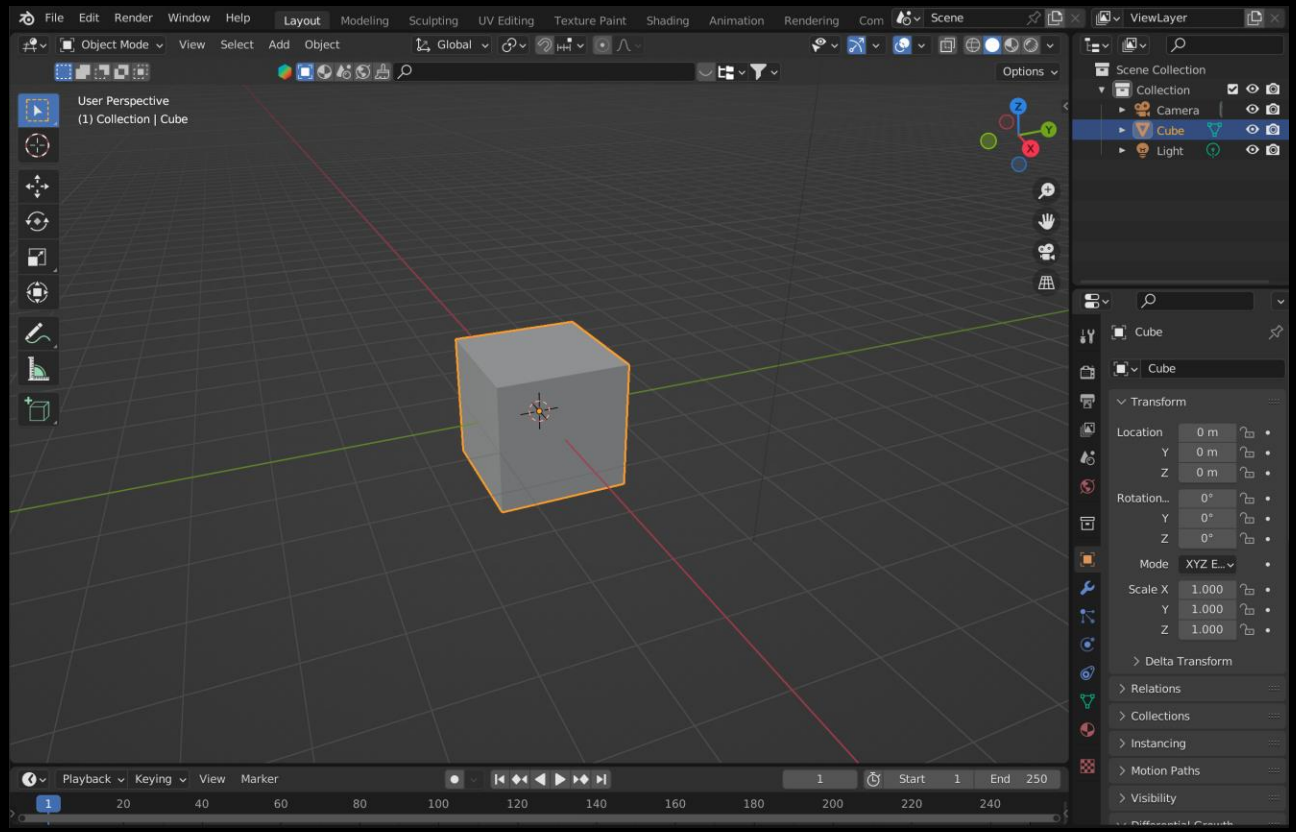


101 Workshop – 3D Modelling

THE EDGE

State Library of Queensland

Queensland Government

Acknowledgement of Country

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.

WORKSHOP SUMMARY

Learn the basics of 3D modelling with this introductory workshop. The session will focus on creating a basic character shape in Blender that can be used as a digital asset for gaming and animation for 3D printing.

Included in the workshop will be a 3D printer induction and how to preparing your 3D model for printing on The Edge's 3D printers and how to book and print your model.

Open Lab sessions are a chance to meet up with like-minded makers and tinkerers at The Edge with facilitators to support your creative needs in the space.

Bookings are required to use the equipment and you will be able to book with your SLQ account once you have completed the relevant induction.

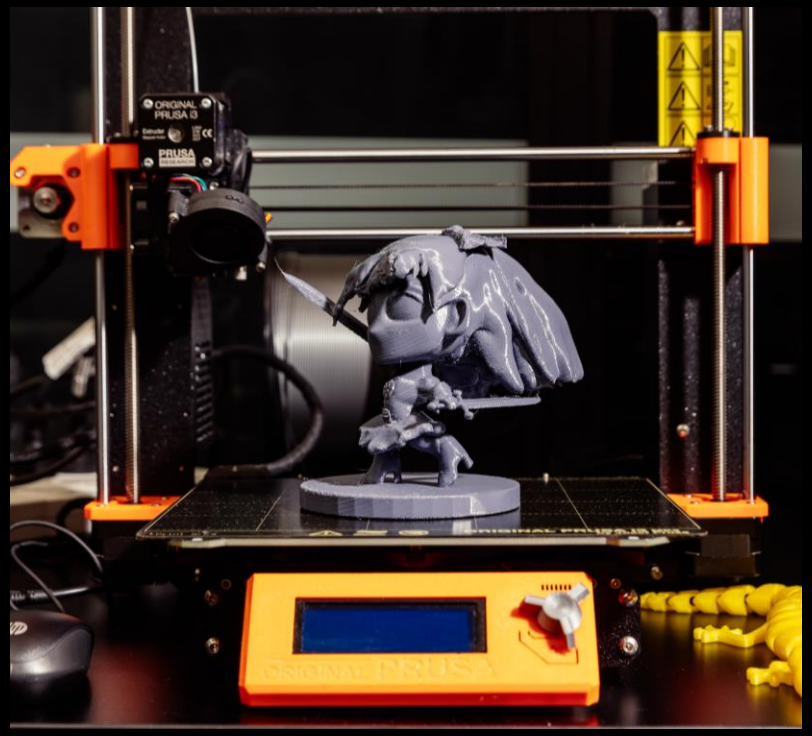
Wednesday 1.30pm – 8pm
Thursday 1.30pm – 8pm
Saturday 12pm – 6pm

For more info and to book, head to https://www.slq.qld.gov.au/visit/spaces/edge

What is a 3D Model?

- Allows for 3D printing which is the construction of a three-dimensional object. It does this through material been deposited, joined or solidified under computer control.

- The Edge has 5 Prusa model i3 MK3s+ printers, which are the successor to Original Prusa i3 MK3 and feature hardware and software upgrades offering improved reliability and ease of use.



OVERVIEW

Designing for 3D Printer

Fusion 360

Pros

- Powerful professional CAD, CAM, CAE software
- Lots of good tutorials online
- Great for designing close tolerance mechanical parts
- Free hobbyist licence available

Cons

- A lot to get your head around when starting out
- Full version relatively expensive (but not as expensive as others)

Designing for 3D Printer

Meshmixer

Pros

- Free
- Great software for sculpting 3D
- Good for repairing models made in other programs

Cons

- Probably want to try something more intuitive first to get your head around the concepts

Designing for 3D Printer

Blender

Pros

- Free and open source
- Great software for all types of 3D modelling and animation
- Lots of tutorials and guides online

Cons

- Probably want to try a simpler software first to get your head around the concept of 3D design

Designing for 3D Printer

TinkerCAD

Pros

- Free
- Great intuitive software for getting started
- Browser based

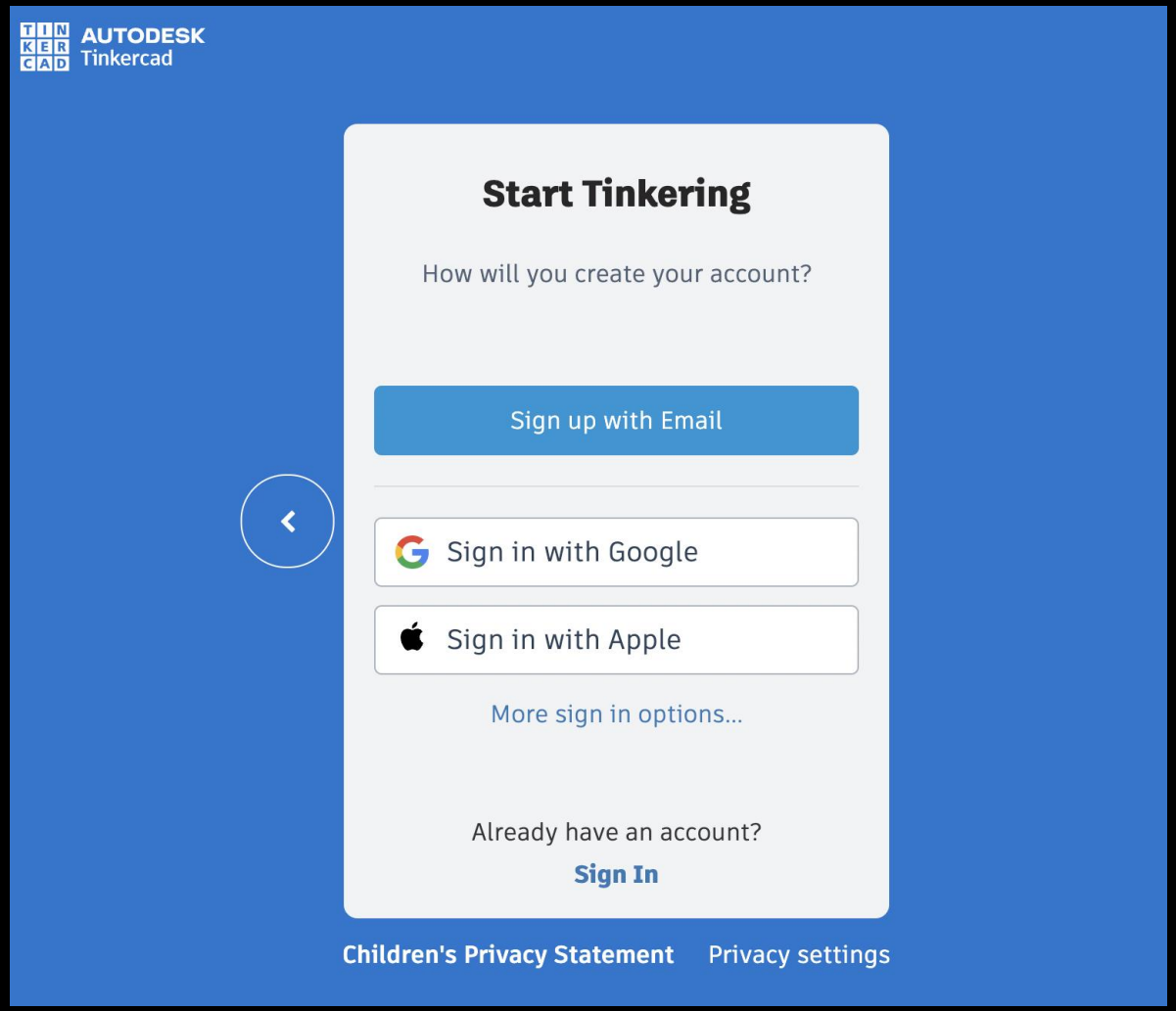
Cons

- Experienced designers might get frustrated with the lack of features

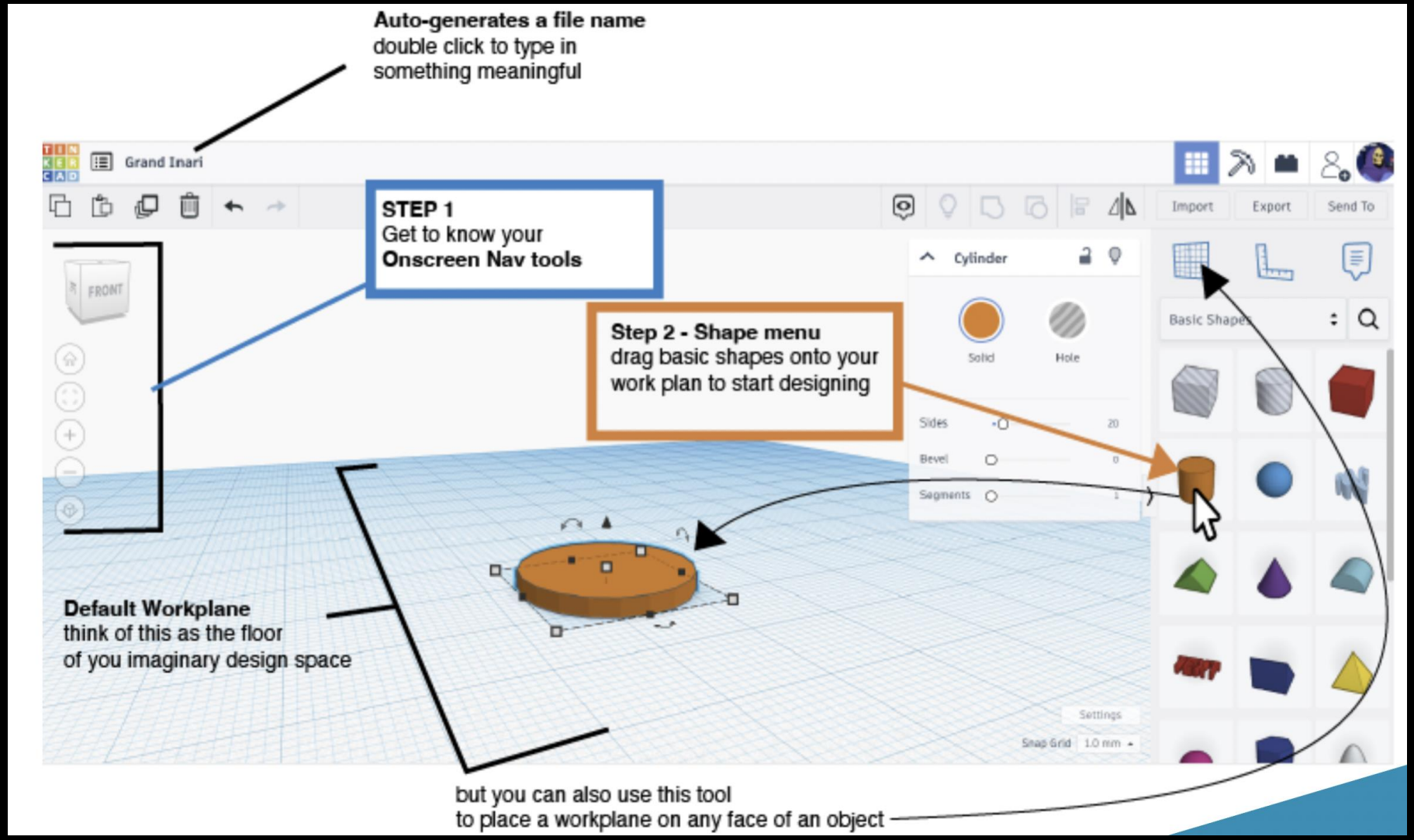
TinkerCAD

Let's have a go!

- Head to the internet and navigate to TinkerCAD.com
- Create a free account using an email address



The Interface



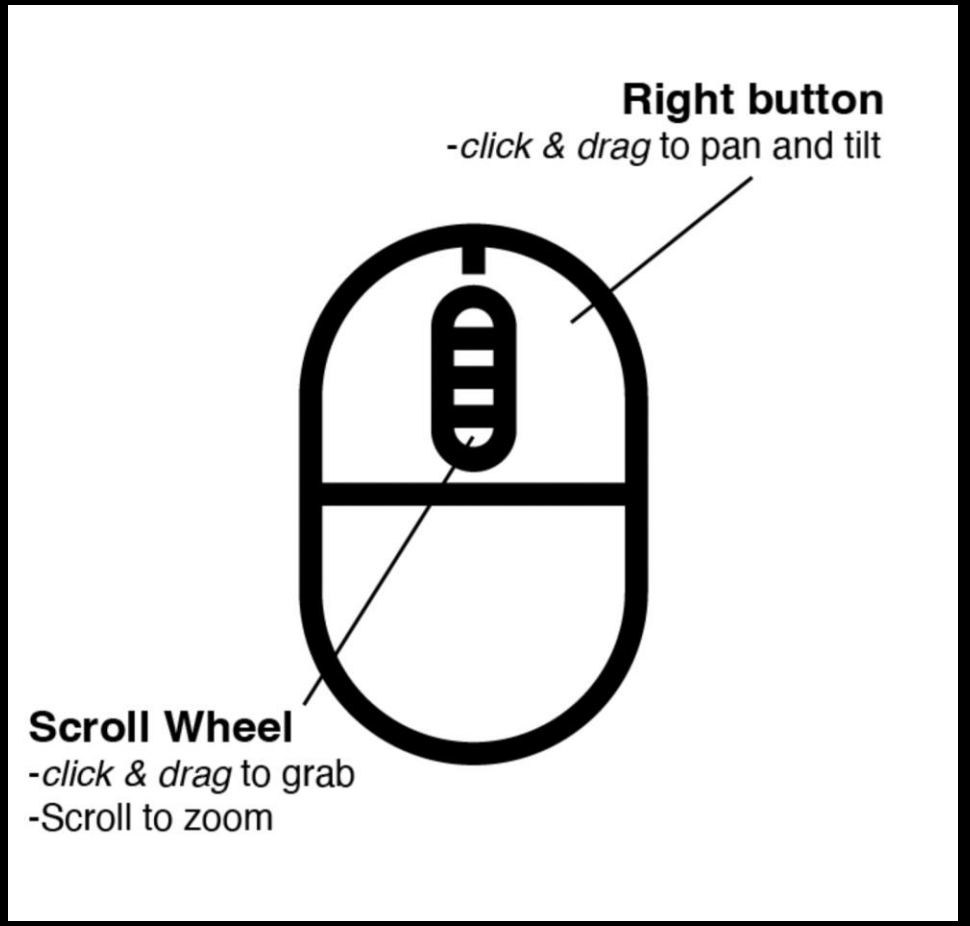
Navigating the 3-dimensional space

Moving around

- Zoom in and out using the scroll wheel

Pan & Tilt

- Right click and drag to change the aspect of your view or
- Click and drag the Cube device to rotate you view angle
- If you get lost click the **home icon** and it will take you back to the default view



Moving and manipulating shapes

Select

- Using the left click (hold shift select multiple objects)
- Or left click and drag a selection box around multiple objects

Resizing

- Left click and drag the black or white handles to resize your shapes
- Hold the SHIFT key to constrain proportions
- Left click and drag the black arrow to change the elevation of the shape in relation to the work plane

Moving

- You can nudge a selected object around the x & y axes with the arrow keys

Rotate

- Left click and drag the arched arrows (3 of them for X, Y, Z) to rotate an object, you can also type in an angle once it is highlighted.

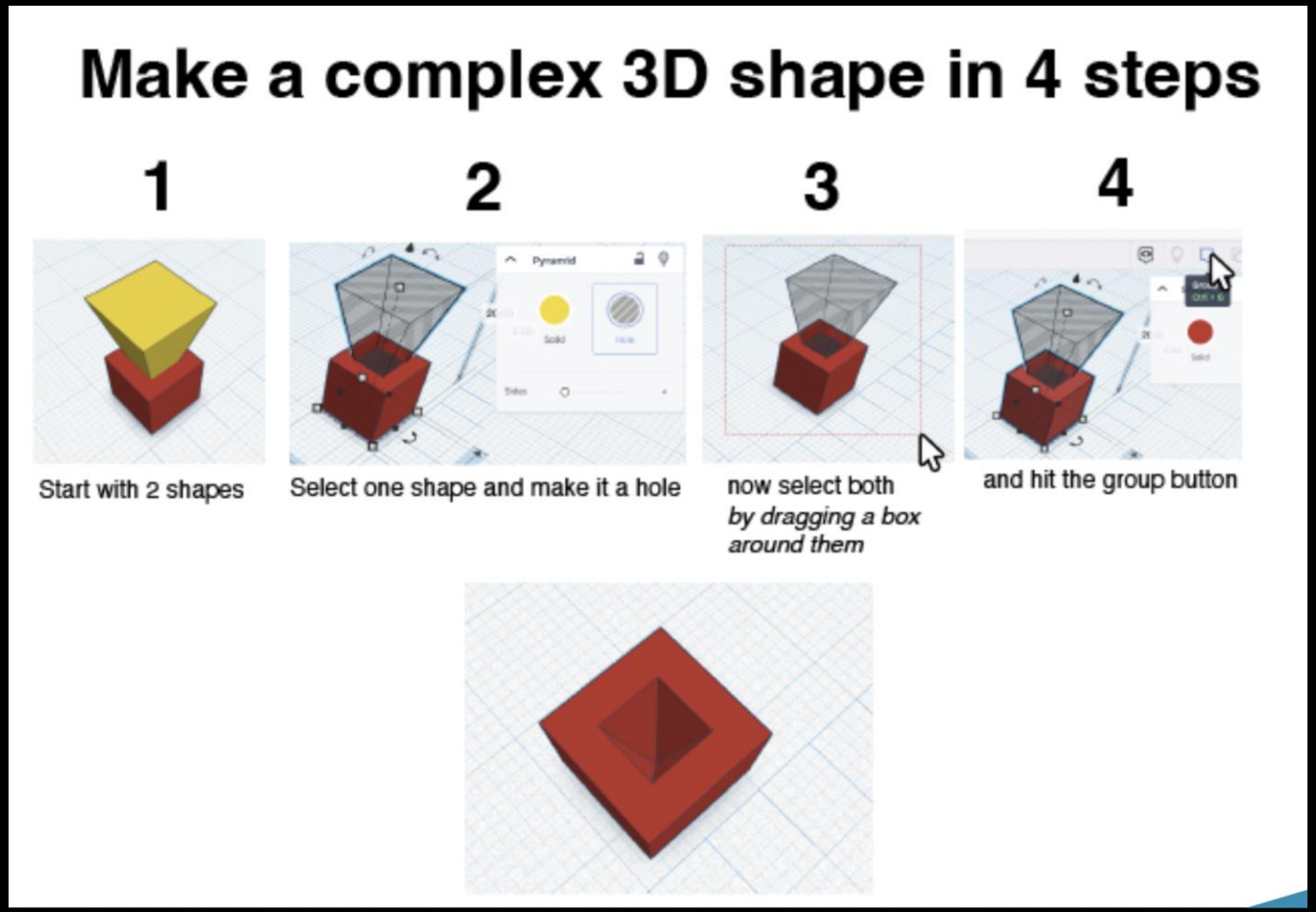
Ruler

- Drag the ruler out onto the workplace to resize or arrange shapes using typed in dimensions

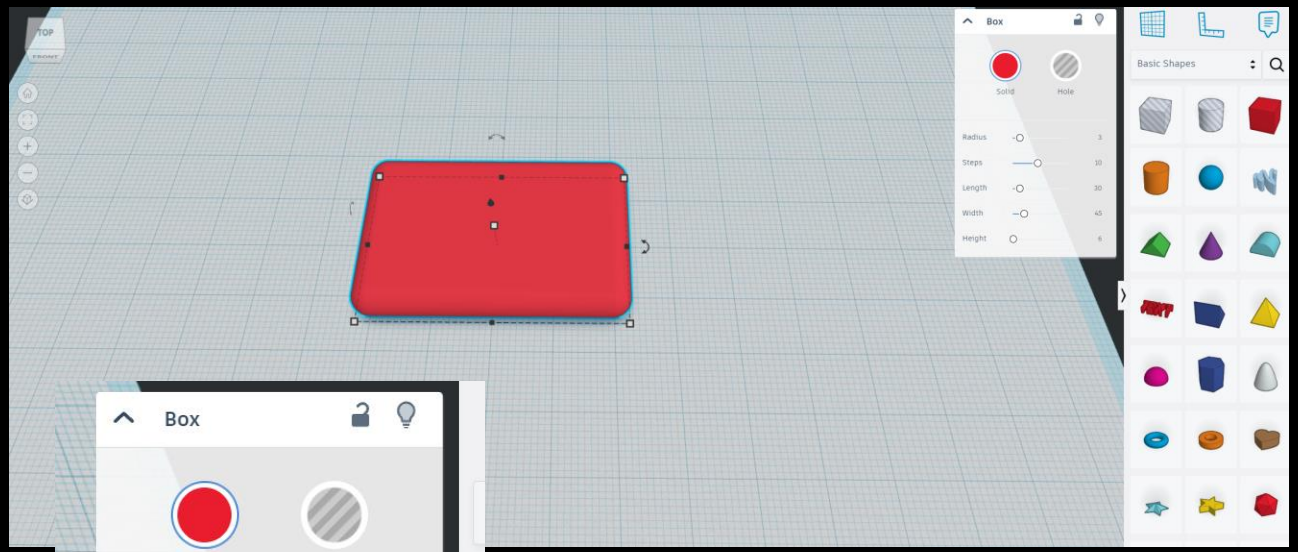
Make a complex shape

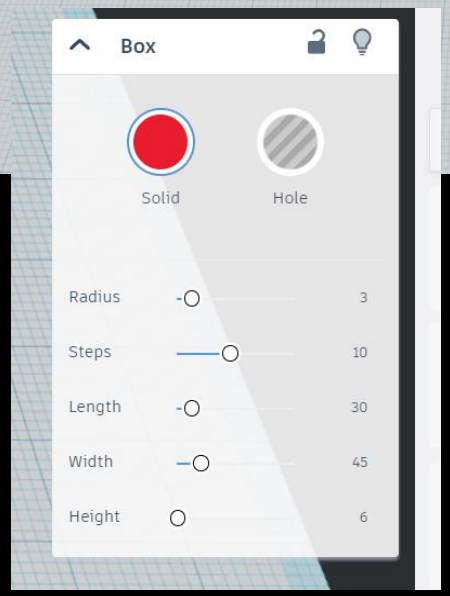
Grouping solids and holes

- You can make complex shapes by combining and subtracting the primitive (basic) shapes using the group tool.
- Select the objects you want to combine
- And then hit the group button



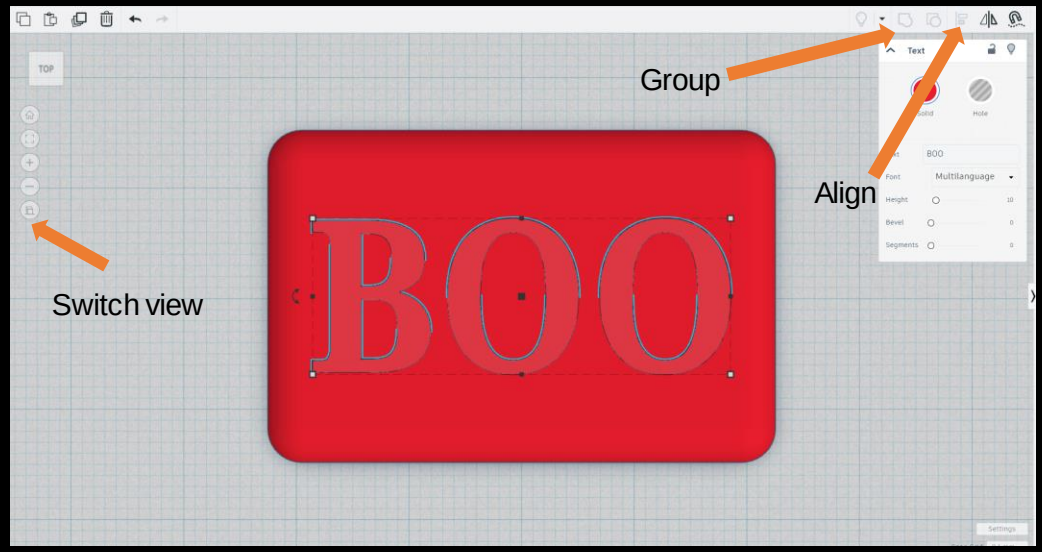
Make a keychain



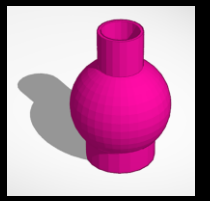


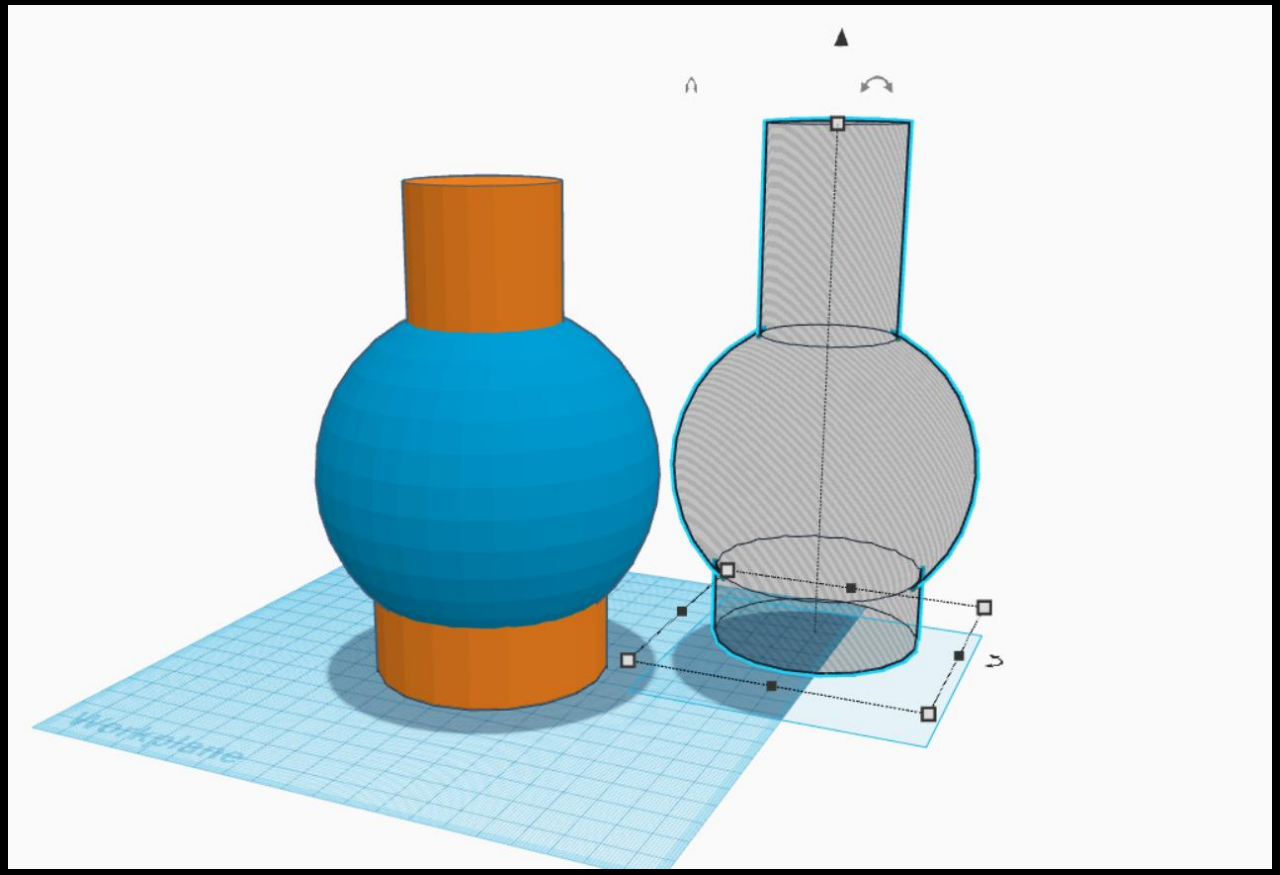
1. Bring in a cube to your workplane
2. Change the dimensions for your length, width and height to similar to the screenshot
3. To get rounded edges, change the radius

4. Hit the 'switch to flat view' button on the left menu
5. Choose the text shape and drag onto your base shape, it should align to the top
6. Write your own text in the Text field and resize to fit base shape, drag the text down show both sides
7. Select both shapes to align the text to center using the dots, choose 'hole' for the text
8. With both shapes selected, merge them together using the group tool so you have cut out text

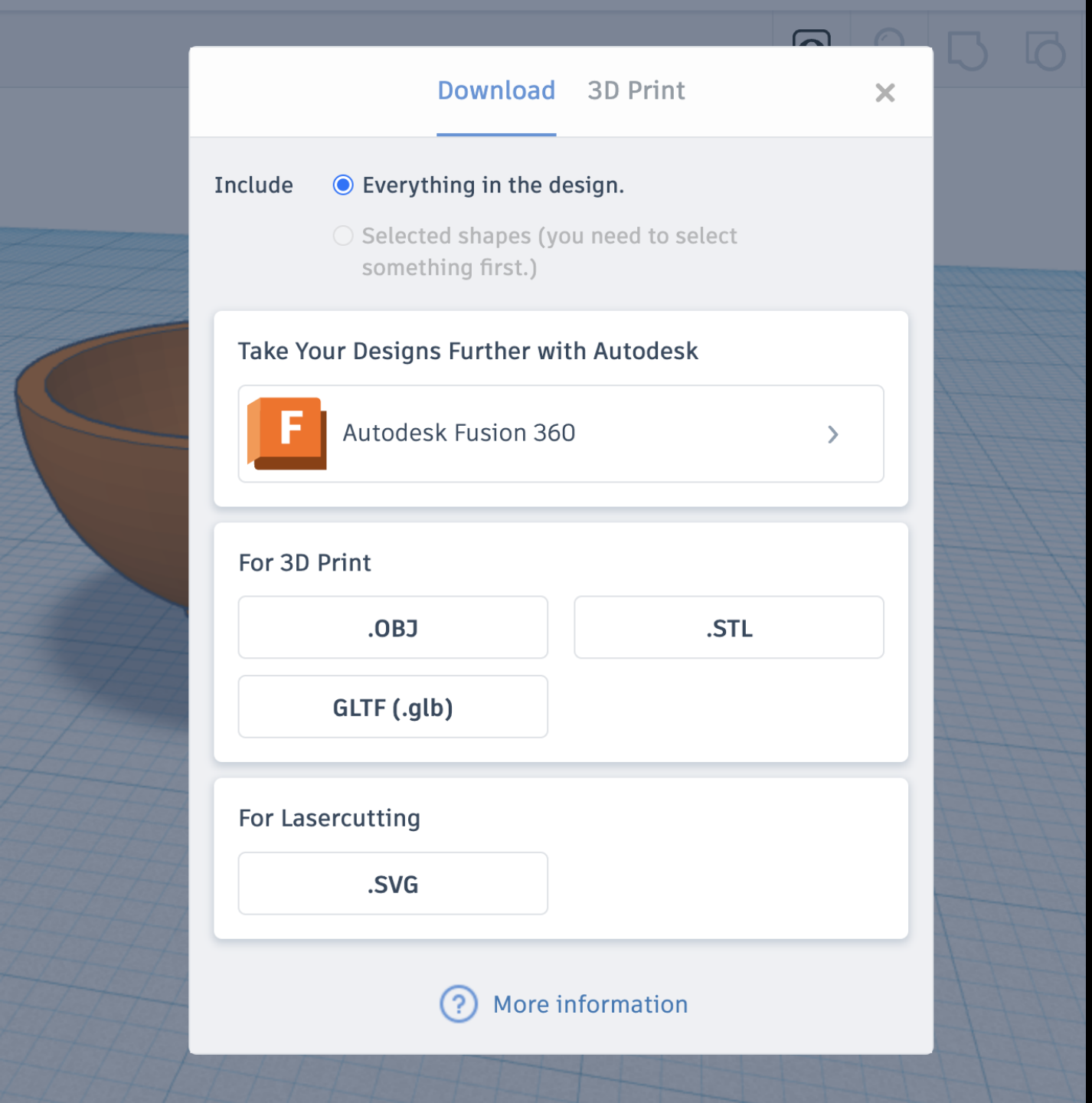


Make a vase





1. Create your vase with some shapes
2. Once happy with the shape, group them and duplicate so you have a combined one you can use to cut a hole. You can ungroup the duplicate to adjust neck height
3. Make sure to shrink the duplicate to at least 4mm smaller to allow for the vase wall thickness
4. Use your align tool and transparency to get the hole shape into position then group everything
5. You should have a final vase shape with thick enough walls to 3d print and use with water



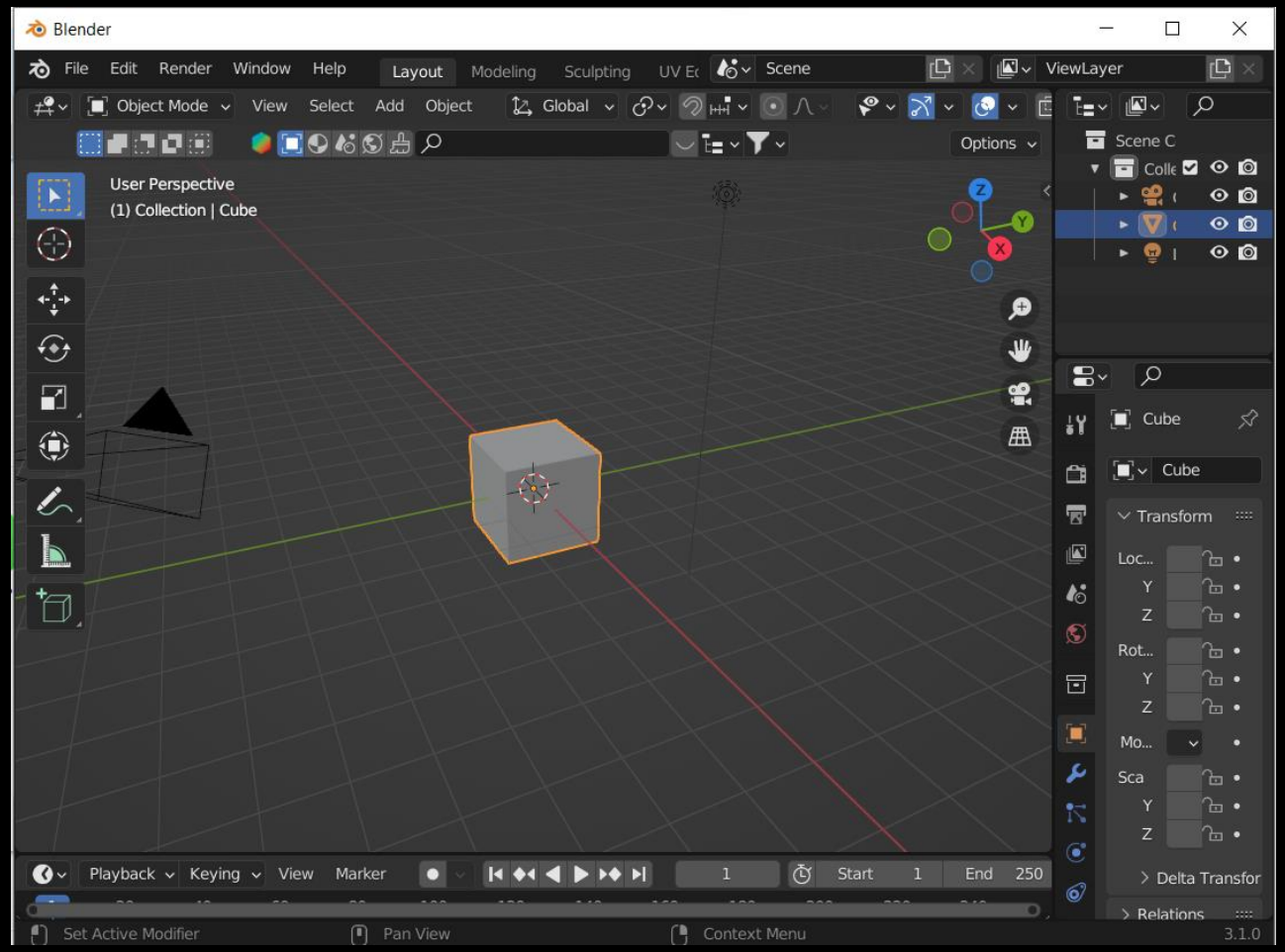
Exporting

- Hit the export button and select STL
- Save to your desired folder
- Open file up in PrusaSlicer using one of our facility computers. The software will be configured correctly on our computers.

Blender

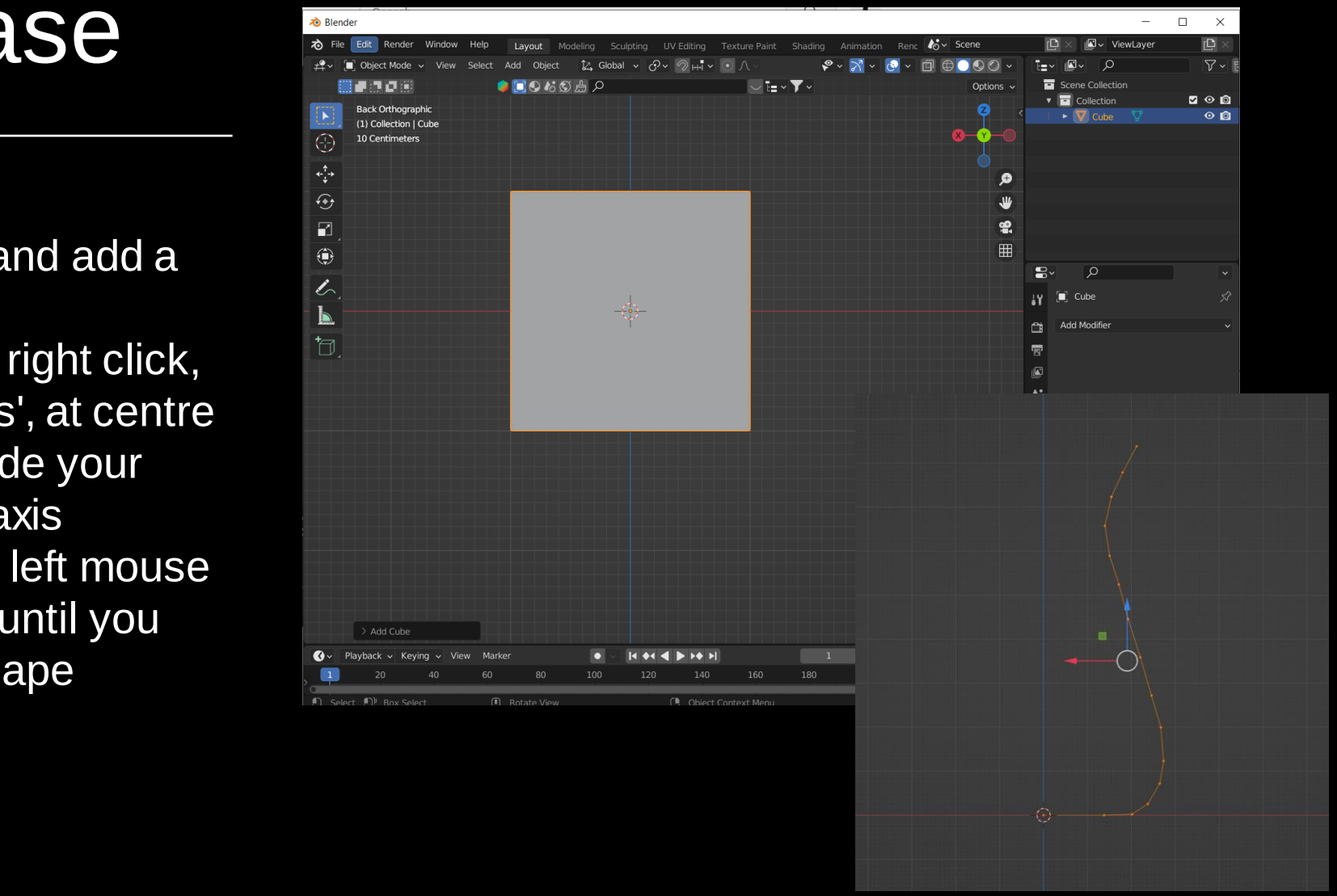
Let's have a go!

- Open the Blender app on your computer
- The default file has a cube, camera and a light
- Let's delete all those by selecting all the items and right click, delete



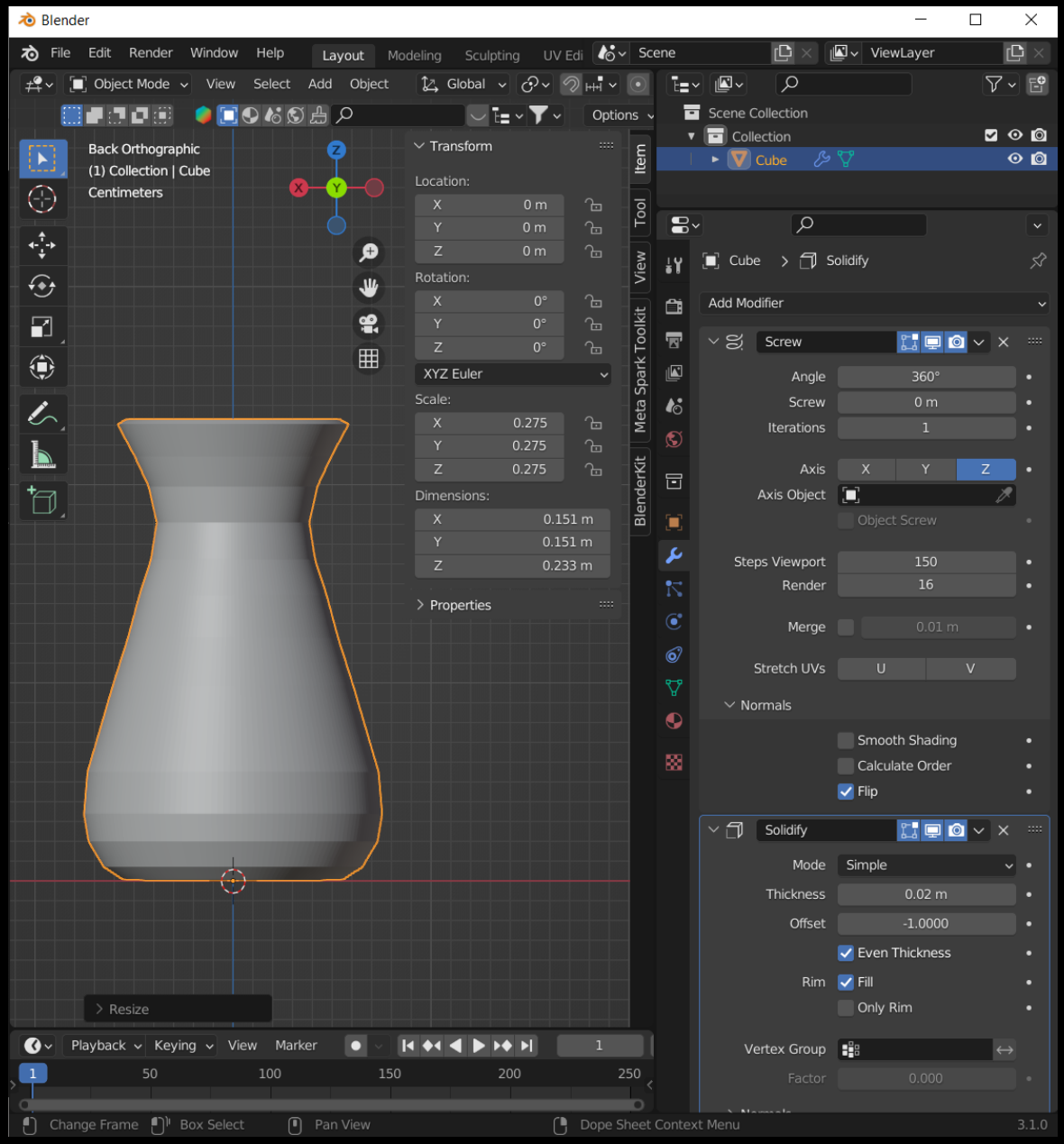
Make a vase

1. Fix view to the Y axis and add a cube
2. Go into edit mode and right click, choose 'merge vertices', at centre
3. Click E then X to extrude your first point along the X axis
4. Using E and click your left mouse button, extrude points until you create a basic vase shape



Make a vase

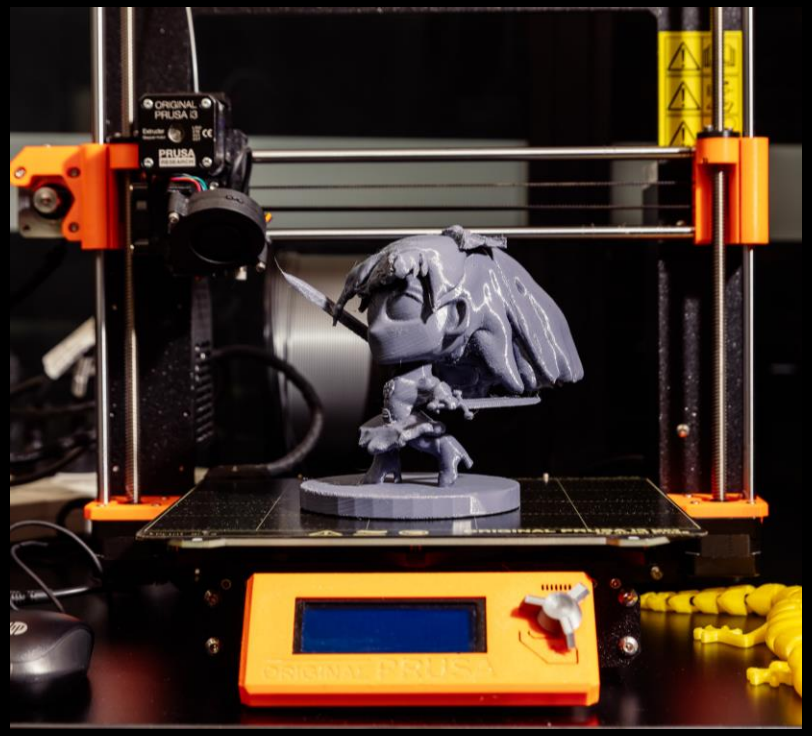
5. Add a 'screw' modifier to see your vase take shape, increase the steps to create a smoother look
6. Turn off smooth shading, and turn on Flip under normals
7. Add a 'solidify' modifier to create the vase walls to be at least 3mm thick (0.003m)
8. Once you are happy with your settings, choose the drop-down arrow and 'apply' the modifiers
9. Your vase is probably quite big so let's scale it down before we export, chose the scale tool and make it about 15cm height (0.15m)
10. Go to file menu, export, choose .stl



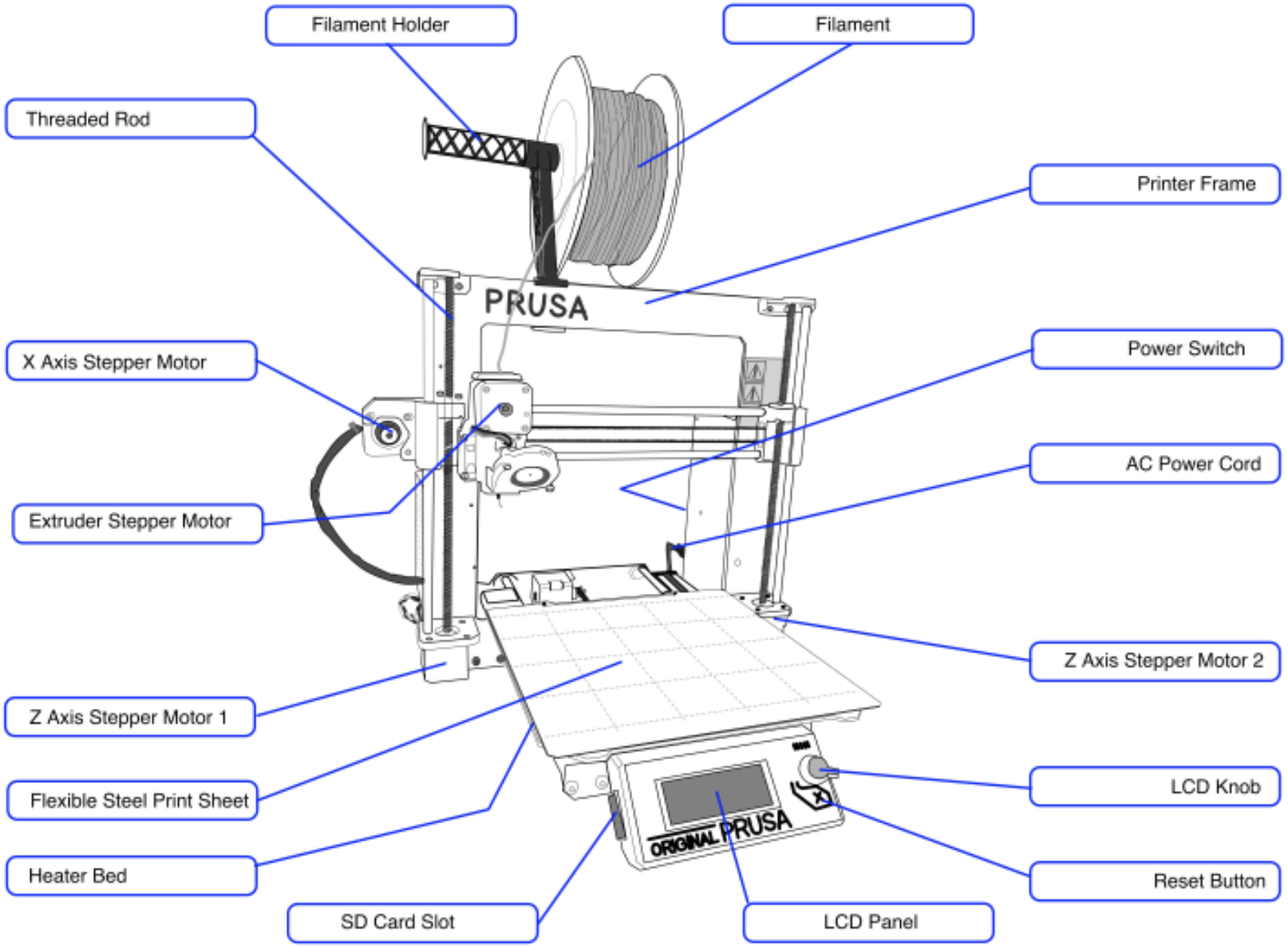
What is a 3D PRINTER?

- Allows for 3D printing which is the construction of a three-dimensional object. It does this through material been deposited, joined or solidified under computer control.

- The Edge has 5 Prusa model i3 MK3s+ printers, which are the successor to Original Prusa i3 MK3 and feature hardware and software upgrades offering improved reliability and ease of use.



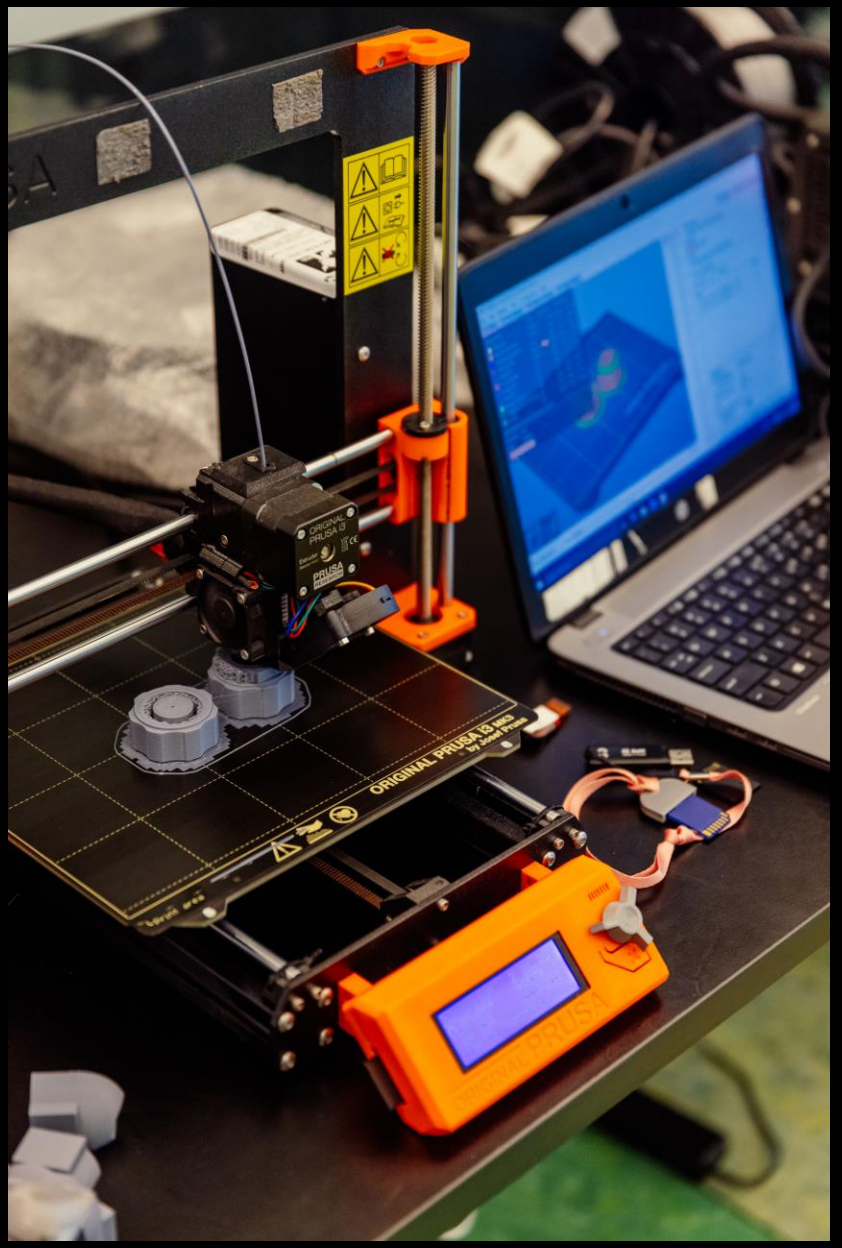
EQUIPMENT OVERVIEW



MACHINE OVERVIEW

Getting Started

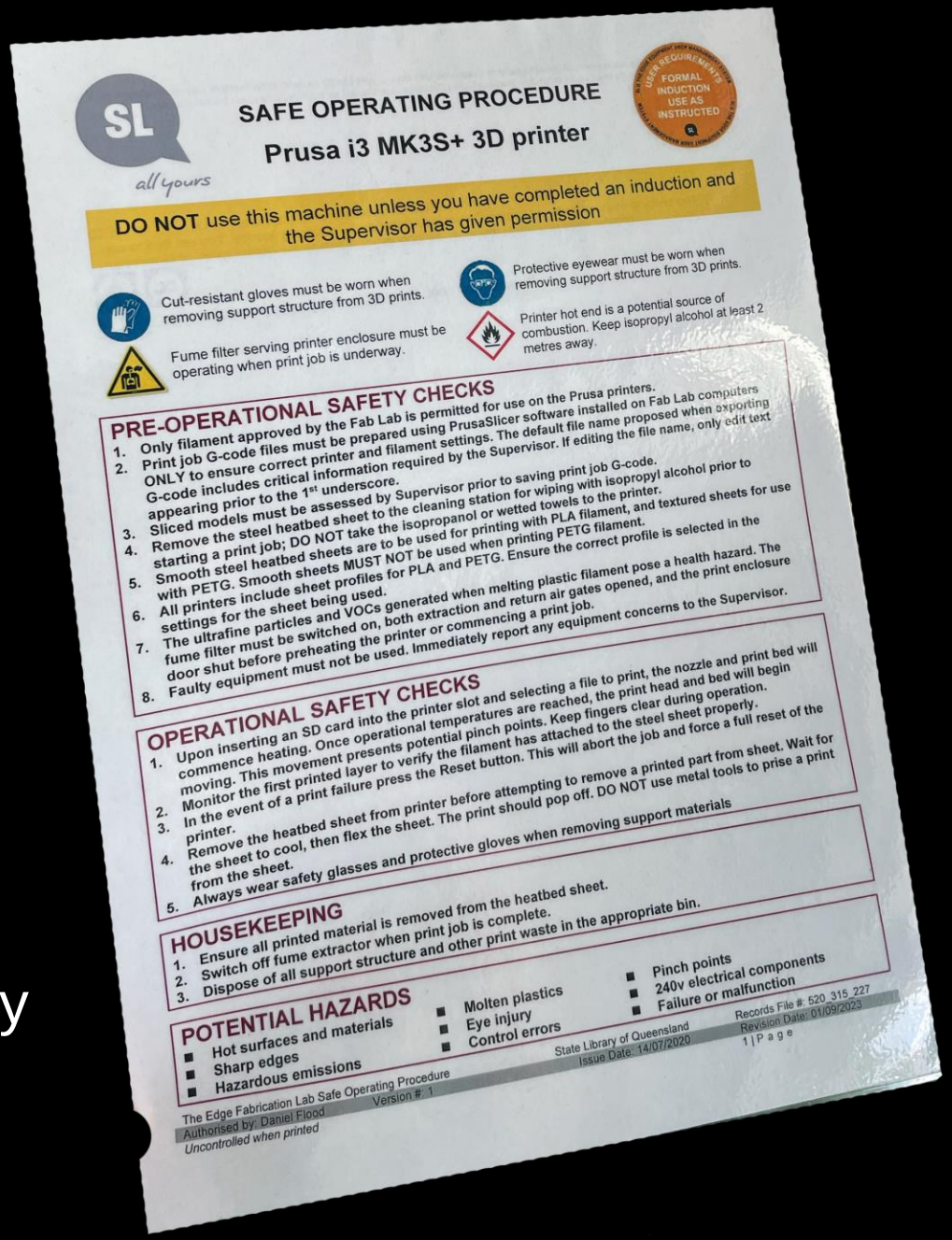
- Make a 3D printer booking.
- Set up design file or if already done, import your model into PrusaSlicer.
- Staff members can make recommendations regarding part orientation, support placement, and layer height.
- When ready to print grab a 3D printer form and ask an Open lab staff member to approve your job!



Safety

- Electricity
- Fire
- Respiratory
- Crush injury
- Isopropyl (eyes/breathing/flammable)

Check the safe operating procedures regularly and ask a staff member for help if you have any problems or concerns.



SAFE OPERATING PROCEDURE

Prusa i3 MK3S+ 3D printer

FORMAL INDUCTION USE AS INSTRUCTED

DO NOT use this machine unless you have completed an induction and the Supervisor has given permission

Cut-resistant gloves must be worn when removing support structure from 3D prints.

Protective eyewear must be worn when removing support structure from 3D prints.

Fume filter serving printer enclosure must be operating when print job is underway.

Printer hot end is a potential source of combustion. Keep isopropyl alcohol at least 2 metres away.

PRE-OPERATIONAL SAFETY CHECKS

1. Only filament approved by the Fab Lab is permitted for use on the Prusa printers.
2. Print job G-code files must be prepared using PrusaSlicer software installed on Fab Lab computers ONLY to ensure correct printer and filament settings. The default file name proposed when exporting G-code includes critical information required by the Supervisor. If editing the file name, only edit text appearing prior to the 1st underscore.
3. Sliced models must be assessed by Supervisor prior to saving print job G-code.
4. Remove the steel heatbed sheet to the cleaning station for wiping with isopropyl alcohol prior to starting a print job; DO NOT take the isopropanol or wetted towels to the printer.
5. Smooth steel heatbed sheets are to be used for printing with PLA filament, and textured sheets for use with PETG. Smooth sheets MUST NOT be used when printing PETG filament.
6. All printers include sheet profiles for PLA and PETG. Ensure the correct profile is selected in the settings for the sheet being used.
7. The ultrafine particles and VOCs generated when melting plastic filament pose a health hazard. The fume filter must be switched on, both extraction and return air gates opened, and the print enclosure door shut before preheating the printer or commencing a print job.
8. Faulty equipment must not be used. Immediately report any equipment concerns to the Supervisor.

OPERATIONAL SAFETY CHECKS

1. Upon inserting an SD card into the printer slot and selecting a file to print, the nozzle and print bed will commence heating. Once operational temperatures are reached, the print head and bed will begin moving. This movement presents potential pinch points. Keep fingers clear during operation.
2. Monitor the first printed layer to verify the filament has attached to the steel sheet properly.
3. In the event of a print failure press the Reset button. This will abort the job and force a full reset of the printer.
4. Remove the heatbed sheet from printer before attempting to remove a printed part from sheet. Wait for the sheet to cool, then flex the sheet. The print should pop off. DO NOT use metal tools to prise a print from the sheet.
5. Always wear safety glasses and protective gloves when removing support materials

HOUSEKEEPING

1. Ensure all printed material is removed from the heatbed sheet.
2. Switch off fume extractor when print job is complete.
3. Dispose of all support structure and other print waste in the appropriate bin.

POTENTIAL HAZARDS

- Hot surfaces and materials
- Sharp edges
- Hazardous emissions
- Molten plastics
- Eye injury
- Control errors
- Pinch points
- 240v electrical components
- Failure or malfunction

The Edge Fabrication Lab Safe Operating Procedure
Authorised by: Daniel Flood
Uncontrolled when printed
Version # 1
State Library of Queensland
Issue Date: 14/07/2020
Records File #: 520_315_227
Revision Date: 01/09/2023
1 | P a g e

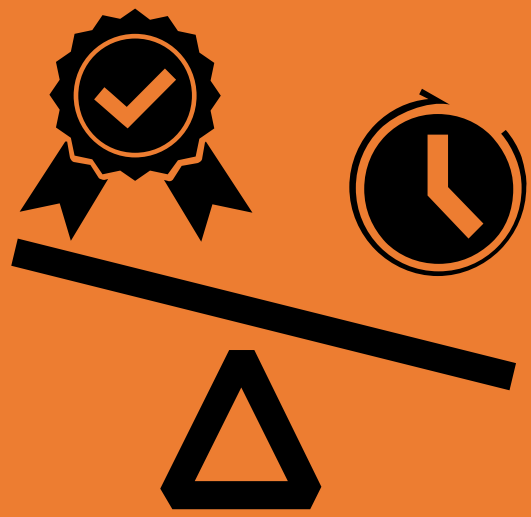
What is Slicing?

The act of converting a 3D model into a set of instructions for 3D printers is called **Slicing.** It is the intermediate and most important step in the process of 3D printing and where The Edge staff will need to check your settings.

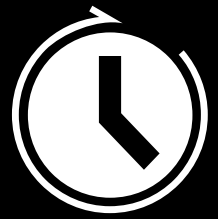
These are some of the main types of settings that can be controlled in a slicer software -

- Print Settings: quality and speed.
- Filament Settings: type of filament etc.
- Printer settings: the model of printer etc.
- Supports: added parts that support overhanging or bridge structures when printing.
- Infill: the internal structure (density/fullness) of a 3D printed part.

Slicing with PrusaSlicer

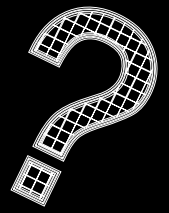


When choosing your print settings, you need to weigh up some priorities



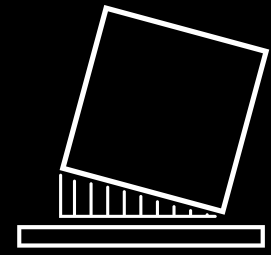
Does it need to be a quick print?

Or do you have time for high quality outcome?



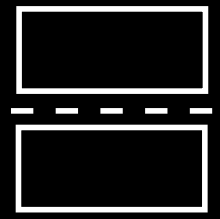
What percentage infill do you need?

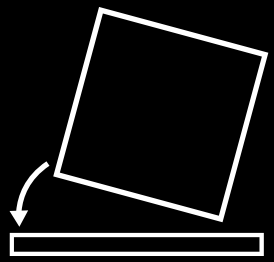
Do you need to print support?



How much filament will this use?

Would it be more time or material efficient to cut up your model?





Or maybe reorienting it will save time, plastic, support and or clean up?

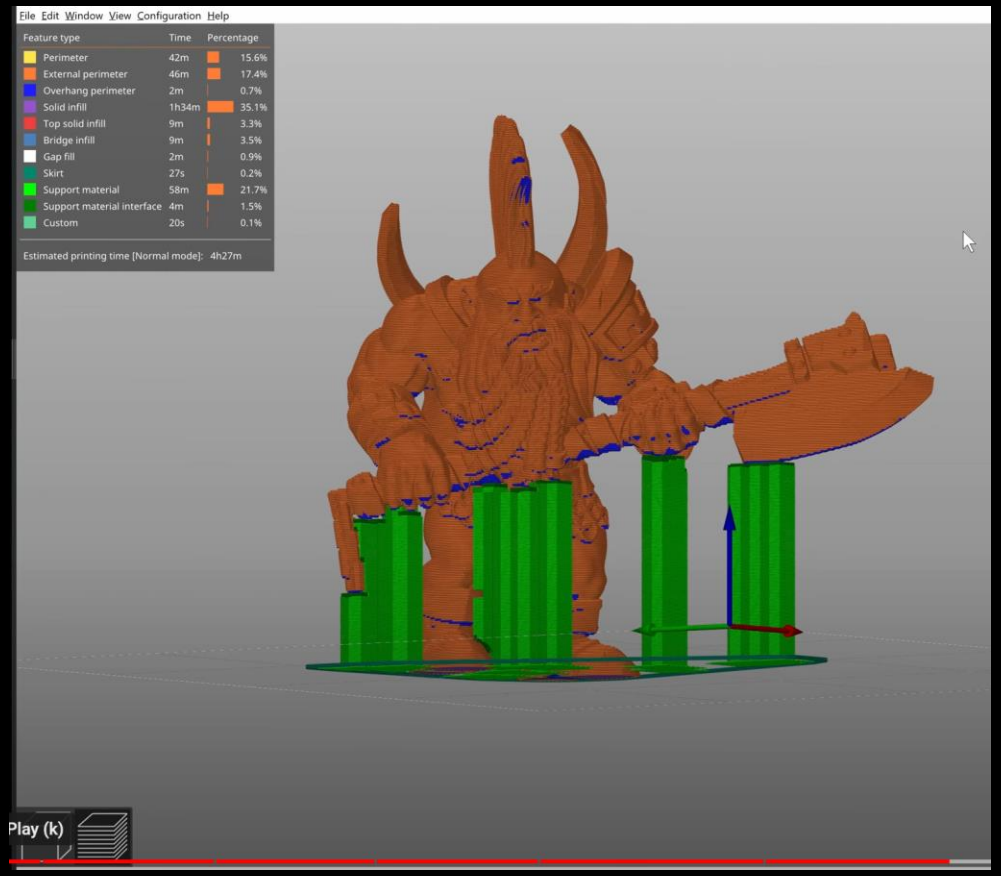
Further info on Supports

3D printers work by layering plastic to create a 3D object. Each new layer must be supported by the one beneath it. If part of your model starts in mid-air and is not supported by anything below, you need to add an additional support structure to ensure a successful print.

Consider if you able to alter the orientation of the model or split into multiple parts to reduce overhangs and the amount of supports generated.

PrusaSlicer can both detect areas that need supports and generate them in these places automatically.

The Paint On support option also works great to manually choose the places you want to add supports to.



Source Jakub Kočí

Further info on Infill

There are a few factors when choosing an infill pattern, listed below, and you can read up on these more on the Prusa website.

Prints (even mechanical parts) rarely benefit from infill over 20%, as uneven cooling can occur on more complex infill which leads to distortions and delamination.

Print speed: How quickly can the pattern be printed?

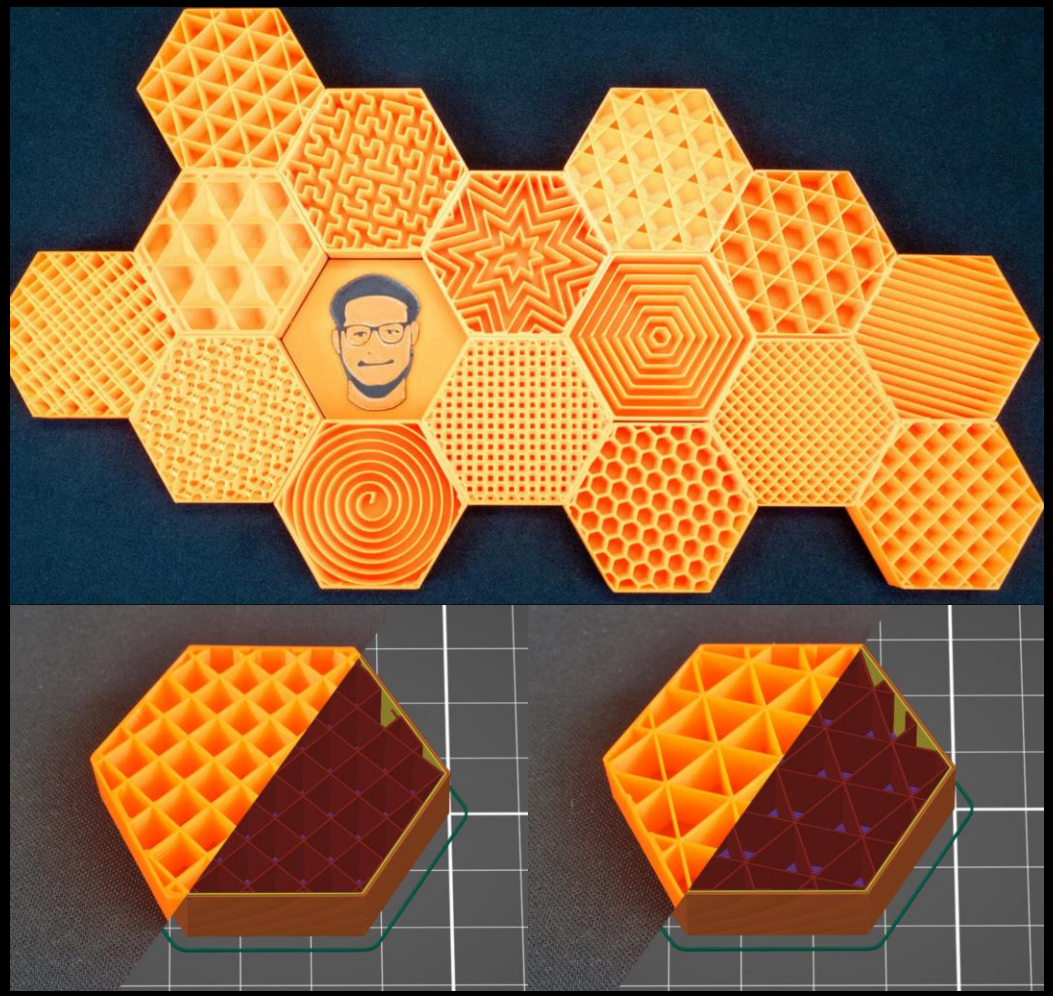
Density: How much filament is used to create the pattern?

Visuals: How does the pattern look when using transparent filament? How does it look if the model is exposed?

Support for top layers: How well will the top layer of a print be supported?

Flexibility: How flexible will the print be with the chosen infill pattern?

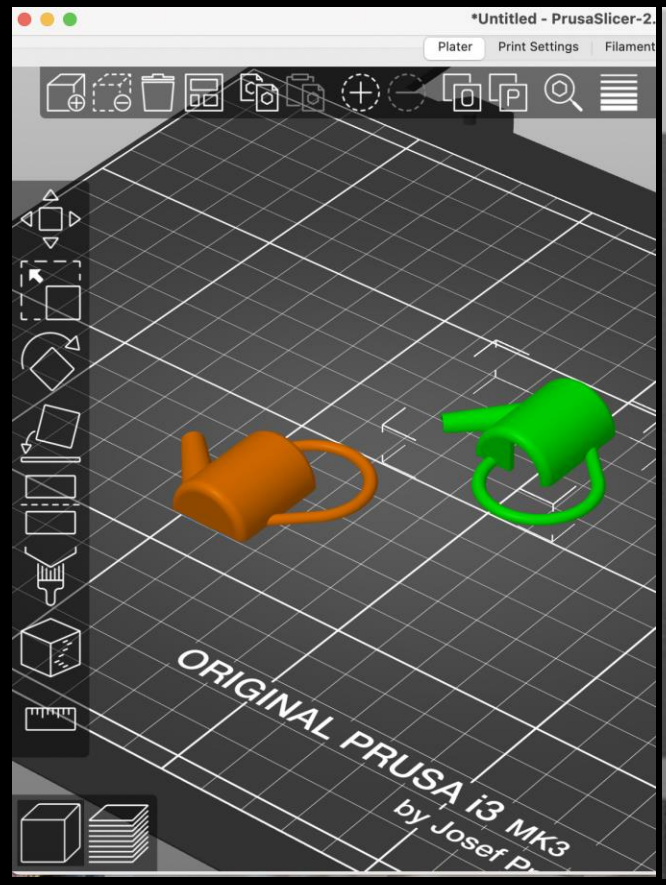
Filling: How suitable is the pattern for filling with resins or other substances? This is especially important for parts that may be used as molds for casting.

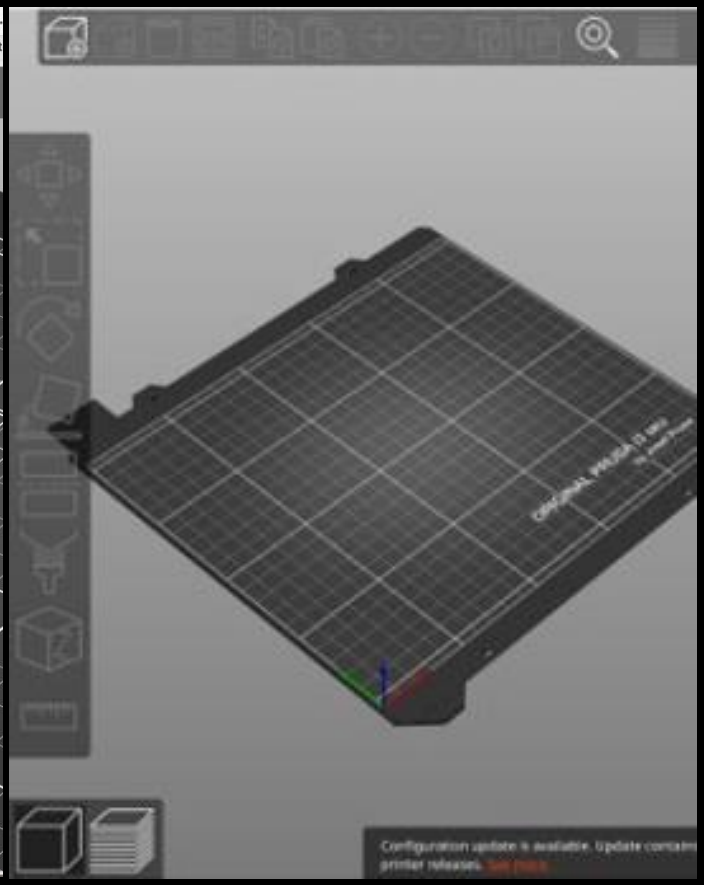


Source Jakub Kočí

Slicing with PrusaSlicer

- Open the **PrusaSlicer** application. If you see the pop-up 'Configuration Update' is available, click to install the update before proceeding further.
- Use the **Add** button to import your model/s into PrusaSlicer.
- **Delete** /**Delete All** buttons to remove models.
- Use **Move**, **scale**, **rotate**, **Place on Face** and **cut tools** to prepare the model to printing in the most efficient/effective orientation.

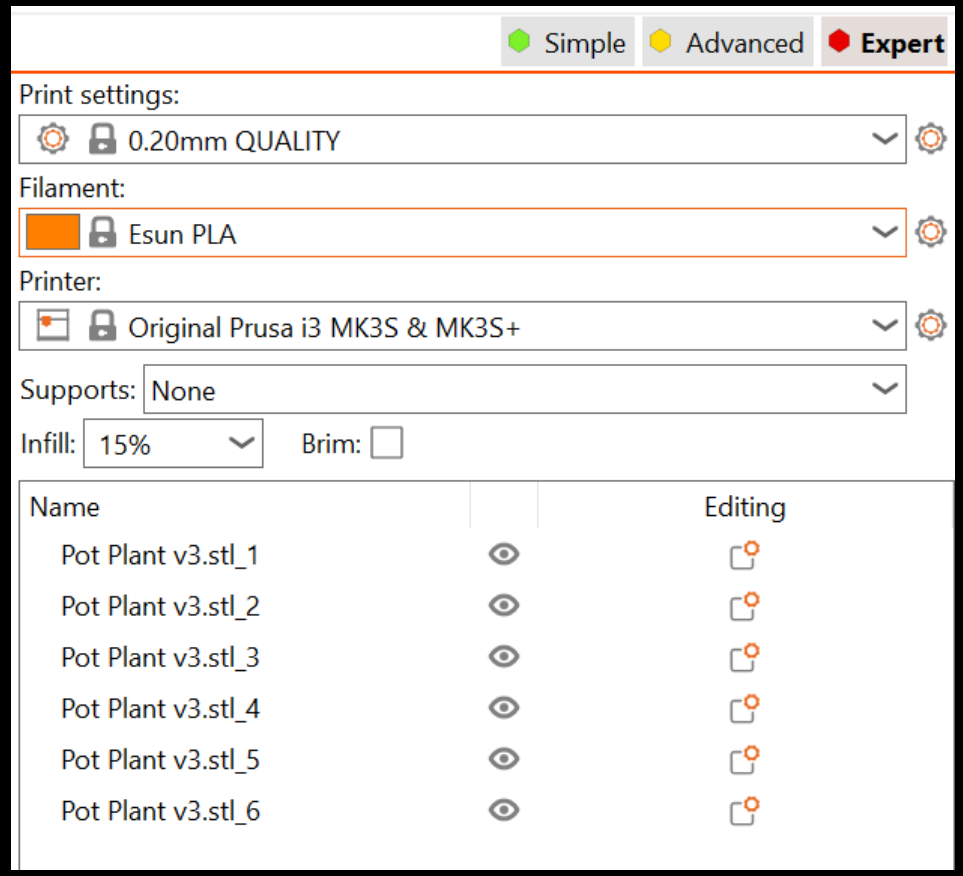




Slicing with PrusaSlicer

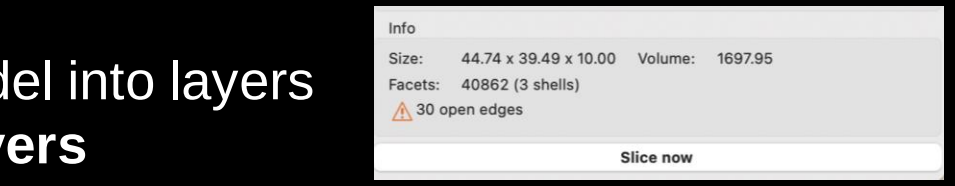
Print Settings

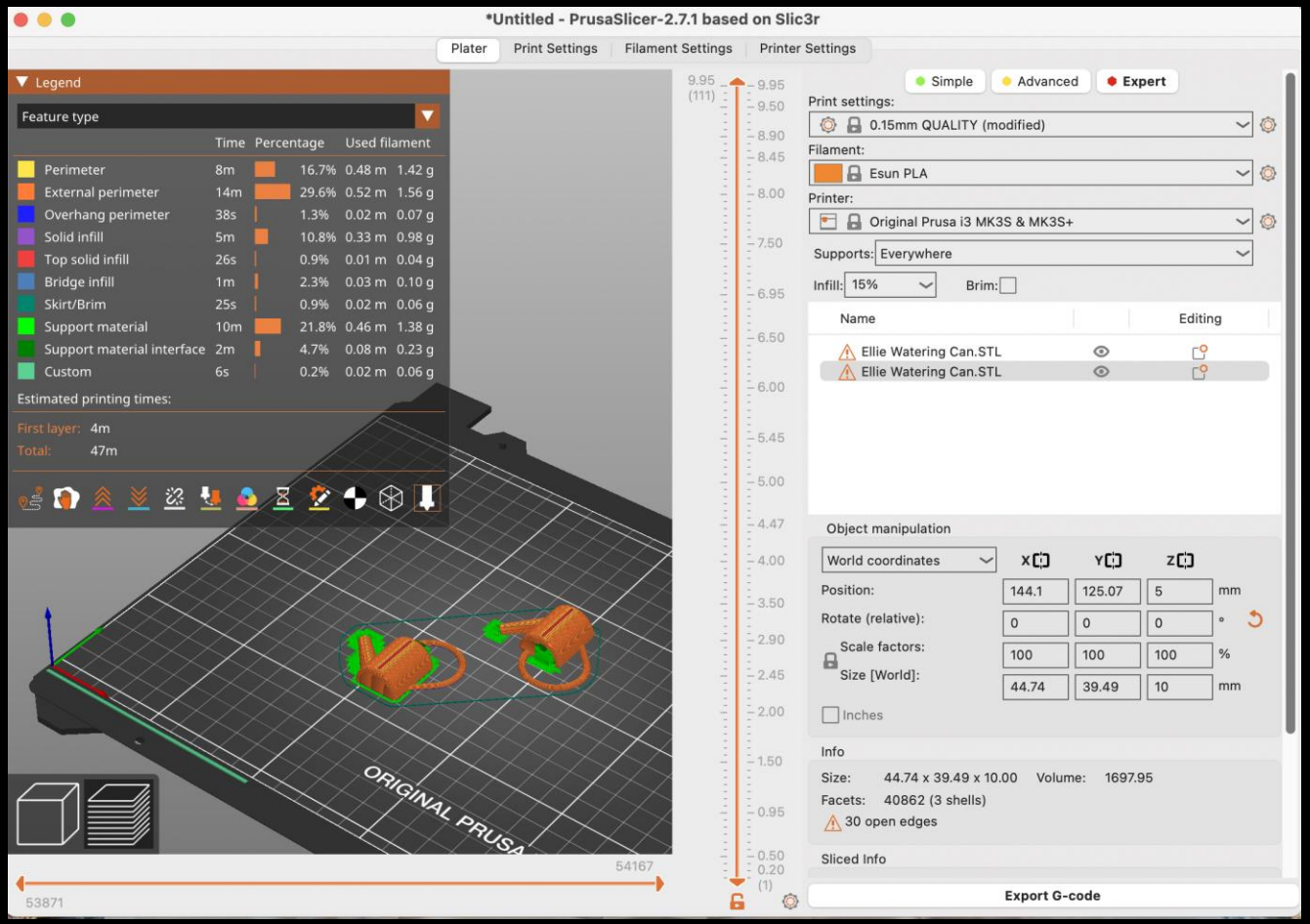
- Select a **Quality / Speed** setting for your print, the higher the quality, the longer the print will take.
- Select the **filament** that corresponds with the one on the printer (check the machine before you slice).
- Select a **Supports** setting
- Select an **Infill** percentage



Slicing with PrusaSlicer

- Click **Slice** to *slice* the model into layers in the Z axis and select **layers preview** to view infill land
- Check the **Sliced Info** for the cost (amount of filament used) and how long it's going to take to print.
- If you are happy with the time and settings, ask a staff member to check over and fill out the print form.
- Generate and export *G-Code* to the SD Card by clicking the **Export G-Code** button.





Setting up the printer

STEP BY STEP

After a staff member has approved your slicing, you can head to the 3D printer you have booked to start your 3D print.

1. Turn on the 3D printer (switch located on the back of the Prusa machines)
2. Ensure the printer is set up with the correct filament and build plate.
3. Switch on the extractor under the 3D printer. Each extractor serves 2 printer enclosures. Make sure BOTH the extraction and return air blast gates serving the printer you are using are open. If there is no job on the 2nd printer, ensure the blast gates serving its enclosure are closed.
4. Insert your media (SD card for Mk3s+ or USB thumb drive for the Mk4) into the printer.
5. Navigate to the file on your media by turning the knob on the screen and selecting the correct G-Code file you have prepared. Select by pushing the button. The printer extruder and bed will start heating up. Your print will start when the target temperatures are attained.
6. Monitor the start of the print to make sure it begins with no issues; you can then leave it to print.

Common print failures

- Birds nest (filament non-adhesion)
- X and/or Y-axis shift (belt or stepper motor errors)
- Print wholly or partially detaches from build plate
- Blocked nozzle
- G-code encoding error
- Thermal anomaly or runaway (hot end or bed)
- Head impact

Troubleshooting

Something is weird or going wrong?

- First reaction should aways be ***pause the job***
- Remember you can always ask a staff member for help!

The filament spool doesn't unwind?

- Check spool mounting and rotation
- Ensure filament on spool is able to unwind without binding

The machine announces an error?

- Refer to the manual for error code explanations
- Check for temperature, connection, or SD card issues

Troubleshooting

The print wholly or partially detaches from the build plate?

- Ensure correct build plate for specific filament type is being used.
- Ensure a clean and level build plate
- Adjust first layer Z height

The printer shifts in the x and/or Y axis?

- Tighten belts and pulleys (staff only)
- Ensure a stable, clear printing environment
- Verify correct print file generation

Filament birds nest forms

- Clean clogged nozzle (staff only)
- Ensure smooth filament path
- Calibrate extruder and temperature settings (staff only)

The spool runs out of filament or the filament breaks?

- Monitor spool and replace before empty (staff only)
- Replace filament with another spool known to be dry (staff only)

See staff for support with any issues labelled Staff only

Extra resources

Prusa

https://www.prusa3d.com/
https://blog.prusa3d.com/

Thingiverse

Checkout things other people have designed for 3D printing and save yourself a lot of time designing something someone else has successfully made. Remix / customise other people's designs.
https://www.thingiverse.com/categories

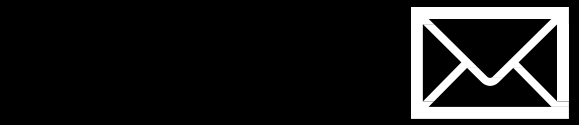
Functional Print

Check out the functional solutions people print up on Reddit.
https://www.reddit.com/r/functionalprint/

THANKS FOR ATTENDING

Please complete our survey that will be sent out via Eventbrite.

Contact us on appliedcreativity@slq.qld.gov.au



State Library of Queensland

Queensland Government

slq.qld.gov.au