

# 3D PRINTER INDUCTION - THE EDGE



## 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.



---

# INDUCTION SUMMARY

---

Using the State Library 3D printers, you can turn a digital 3D model into a plastic object.

In this induction you will learn the basics of 3D modelling with Tinkercad and how to print your own model, including how to load it into the printer and cleaning up the product.

Once you've completed the induction you can book and use the State Library of Queensland's 3D Printers at The Edge during Open lab.

**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**

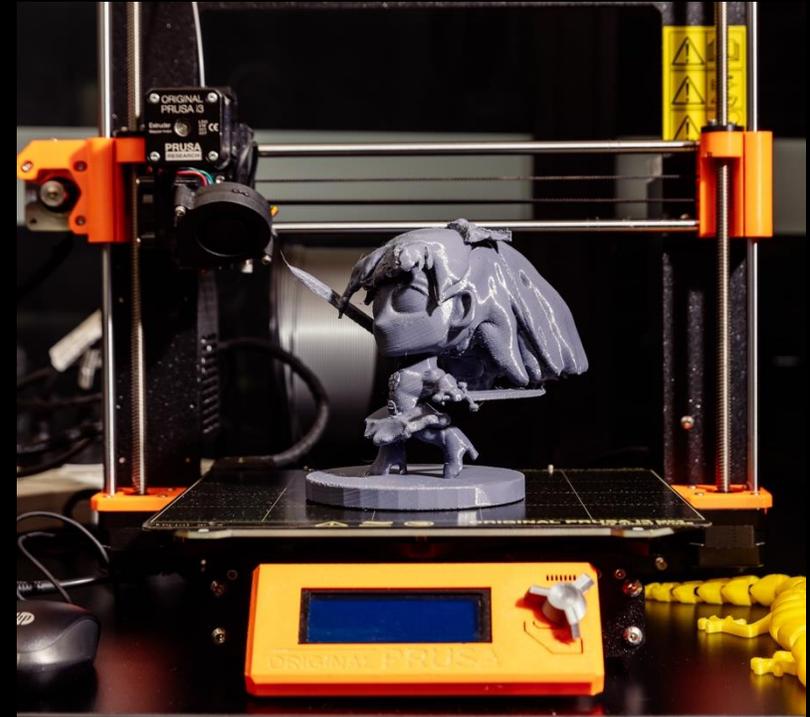
**Thursday 1.30pm – 8pm**

**Saturday 12pm – 5pm**

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

# What is a 3D PRINTER?

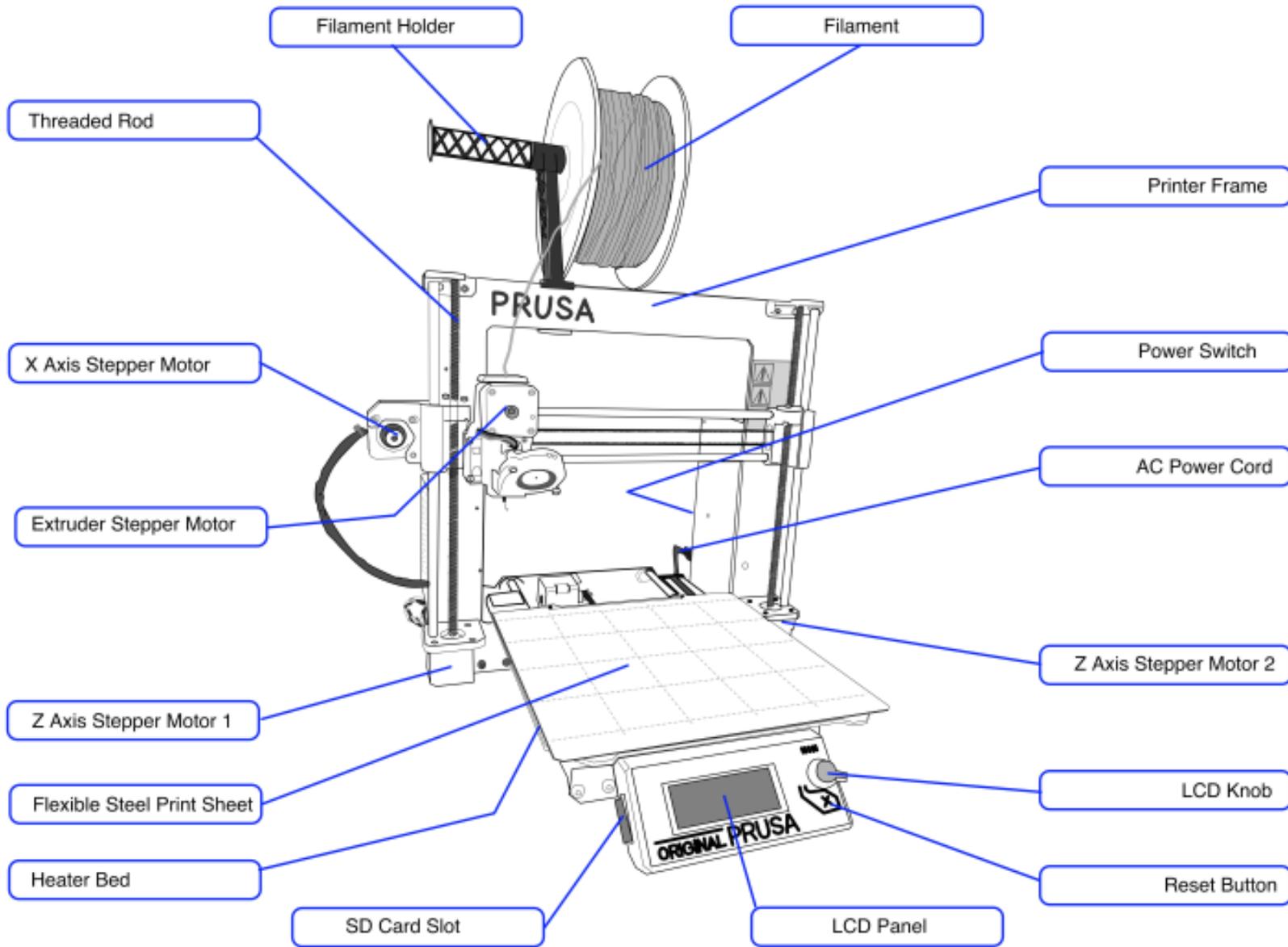
- Allows for 3D printing which is the construction of a three-dimensional object. It does this through material being deposited, joined or solidified under computer control.
- The Edge has 5 Prusa model i3 MK3s+ printers, which are the successor to Original Prusa i3 MK3 and feature hardware and software upgrades offering improved reliability and ease of use.



---

## EQUIPMENT OVERVIEW

---



---

# MACHINE OVERVIEW

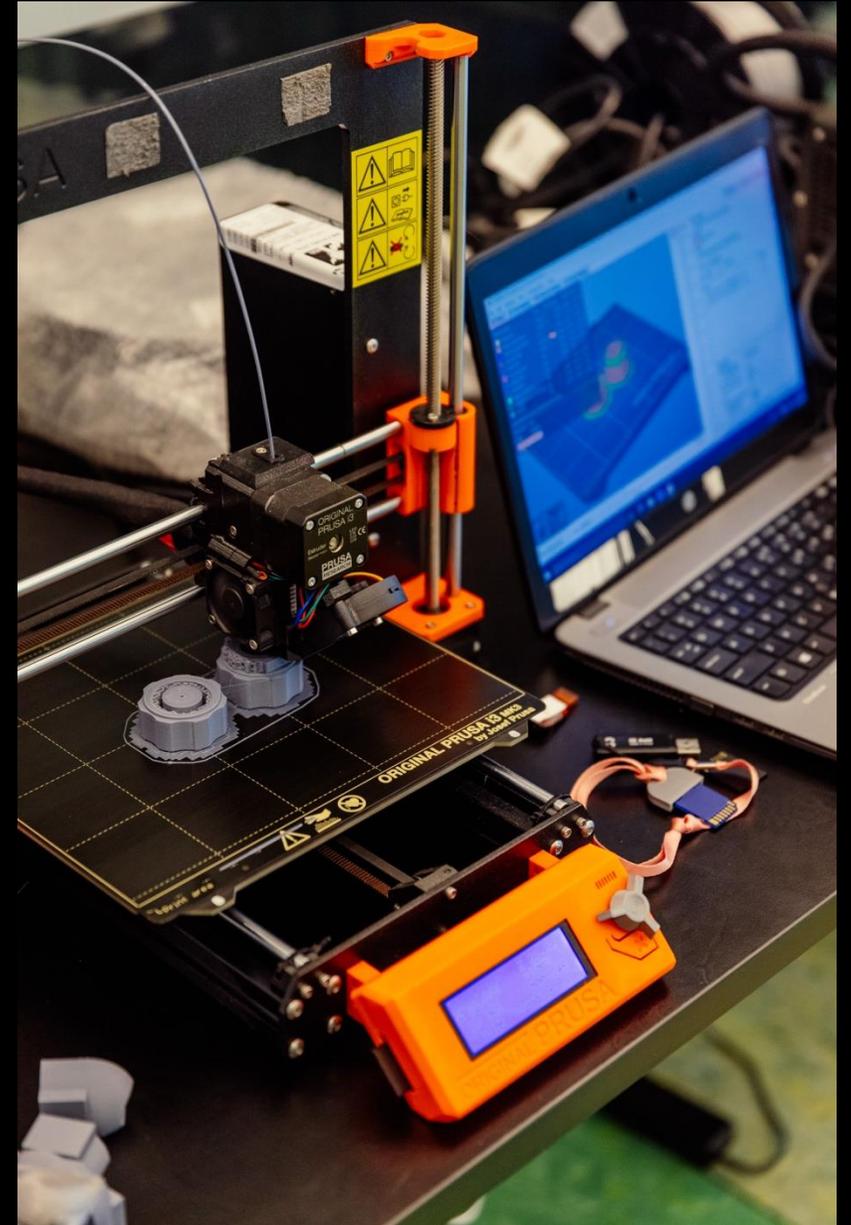
---

---

# 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.

**SL** *all yours*

## SAFE OPERATING PROCEDURE

### Prusa i3 MK3S+ 3D printer

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

**USE REQUIREMENTS**  
FORMAL INDUCTION  
USE AS INSTRUCTED

- 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 1<sup>st</sup> underscore.
3. Sliced models must be assessed by Supervisor prior to saving print job G-code.
4. Remove the steel heated 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 heated sheets MUST NOT be used when printing with PLA filament, and textured sheets for starting a print job; DO NOT use when printing with 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 heated 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 heated 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 | ■ Molten plastics | ■ Pinch points               |
| ■ Sharp edges                | ■ Eye injury      | ■ 240v electrical components |
| ■ Hazardous emissions        | ■ Control errors  | ■ Failure or malfunction     |

The Edge Fabrication Lab Safe Operating Procedure  
Authorised by: Daniel Flood  
Uncontrolled when printed

State Library of Queensland  
Issue Date: 14/07/2020

Records File #: 520\_315\_227  
Revision Date: 01/09/2023  
1 | Page

# 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

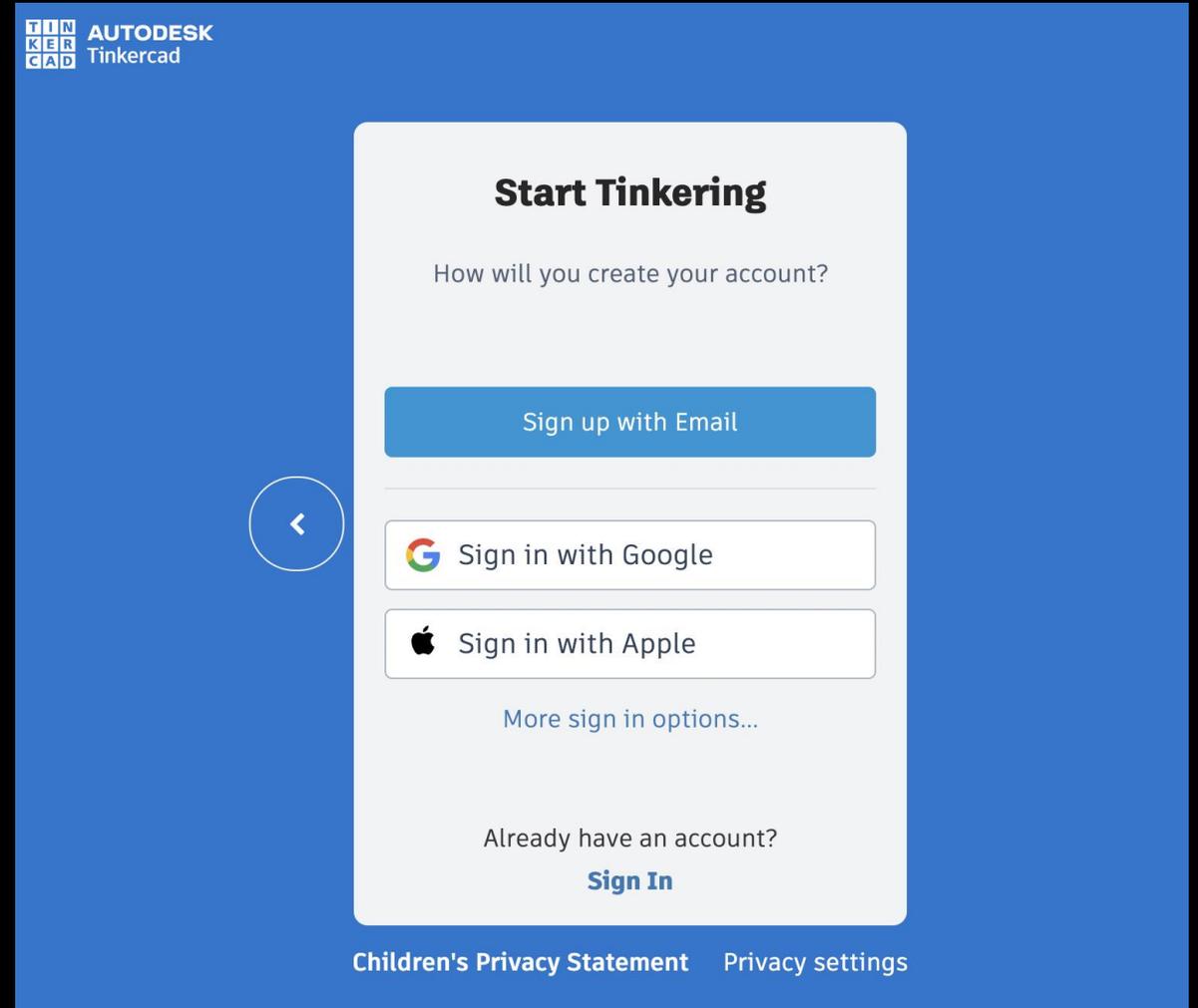
---

# 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

Auto-generates a file name  
double click to type in  
something meaningful

**STEP 1**  
Get to know your  
**Onscreen Nav tools**

**Step 2 - Shape menu**  
drag basic shapes onto your  
work plan to start designing

**Default Workplane**  
think of this as the floor  
of you imaginary design space

but you can also use this tool  
to place a workplane on any face of an object

The image shows a screenshot of a 3D CAD software interface. At the top, a window title bar displays 'Grand Inari'. Below it is a toolbar with various icons. On the left, a vertical navigation panel shows a 'FRONT' view icon and several circular navigation buttons. The main workspace features a blue grid floor representing the 'Default Workplane'. In the center of the grid, a brown cylinder is placed, with its top face highlighted in orange. To the right, a 'Basic Shapes' menu is open, showing a grid of various 3D shapes. A mouse cursor is hovering over a cylinder in this menu. A 'Properties' panel for the selected cylinder is visible, showing options for 'Solid' and 'Hole', 'Sides' (set to 20), 'Bevel' (set to 0), and 'Segments' (set to 1). At the bottom right, there are 'Settings' and 'Snap Grid' (1.0 mm) options. Several text annotations with arrows point to specific interface elements: one points to the window title, another to the navigation panel, a third to the cylinder in the shape menu, and a fourth to the cylinder in the workspace.

# Navigating the 3-dimensional space

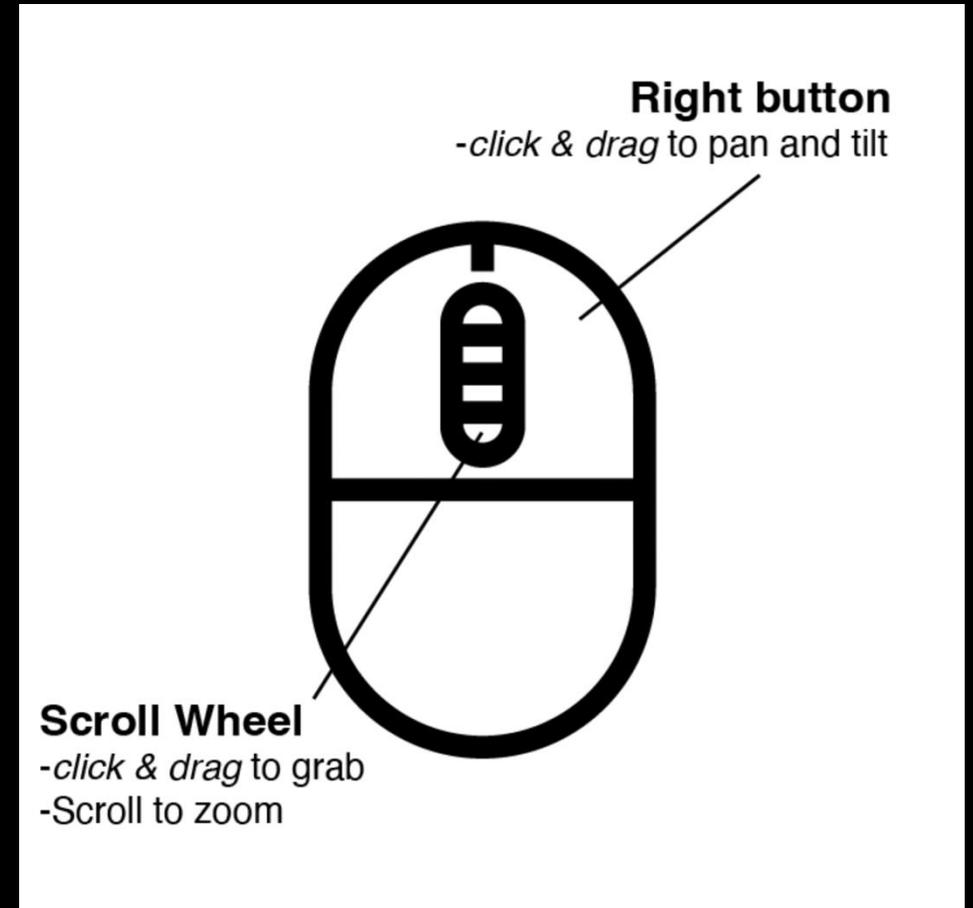
---

## 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 cone 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 curly one of the 3 arrows to rotate an object, you can also type in an angle once it is highlighted.

## Ruler

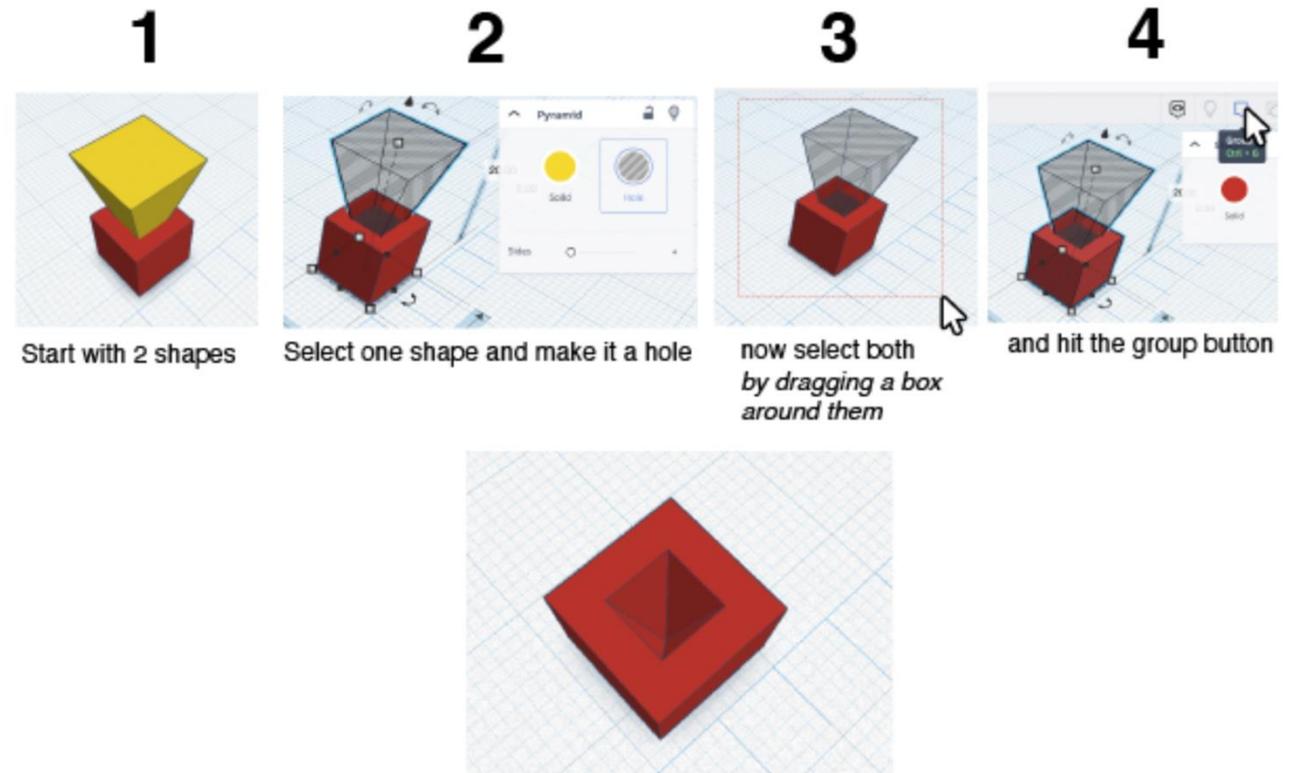
- Drag the ruler out onto the workplace to resize or arrange shapes using typed in dimensions

# Make a complex shape

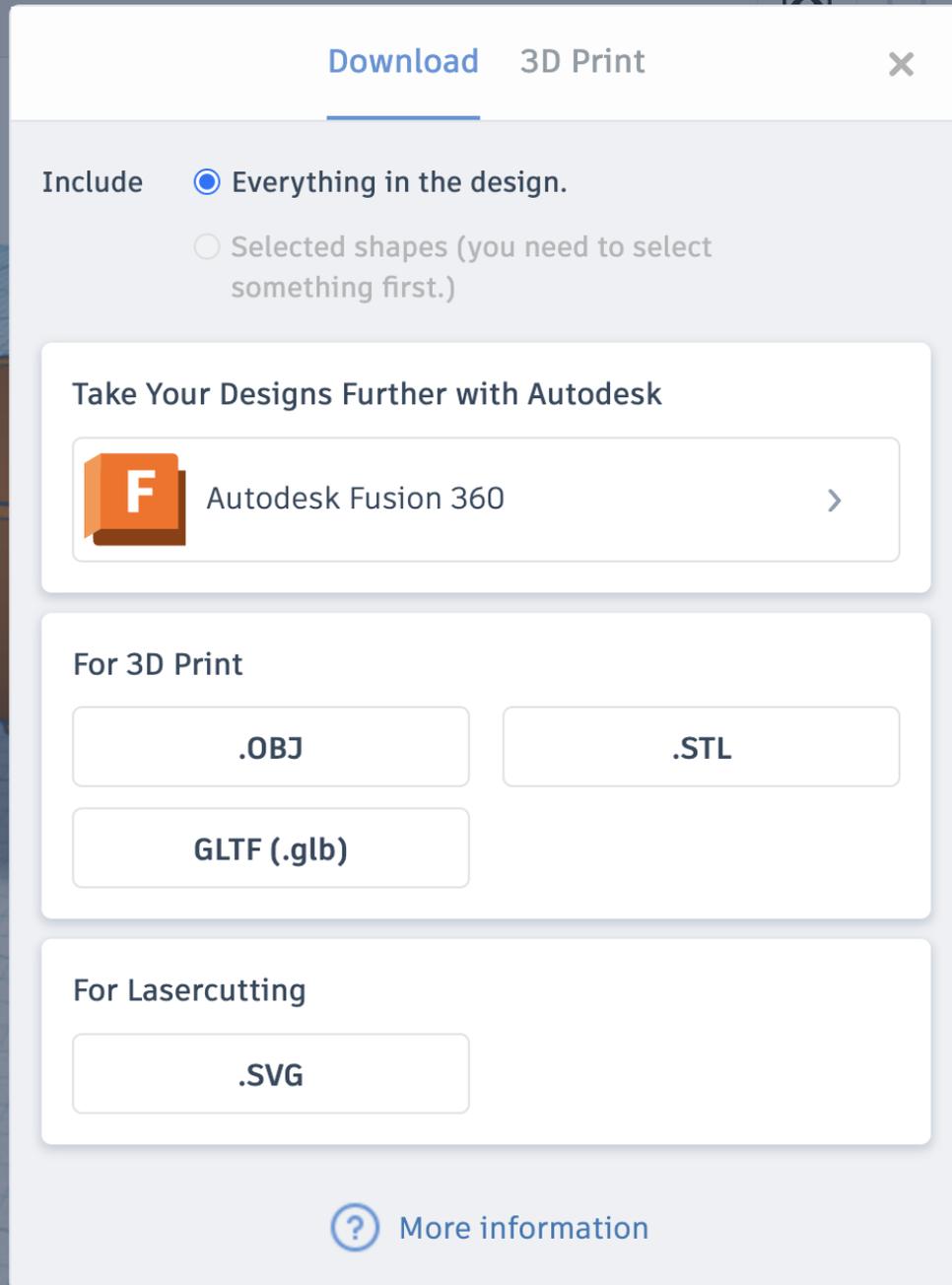
## Grouping solids and holes

- 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 complex 3D shape in 4 steps



# 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.

# 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.

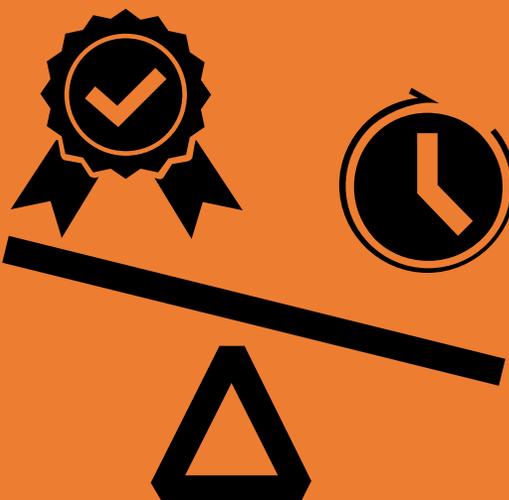


These are some of the main types of settings that can be controlled in a slicer software -

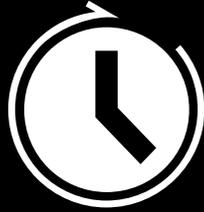
- Print Settings: quality and speed.
- Filament Settings: type of filament etc.
- Printer settings: the model of printer etc.
- Supports: added parts that support overhanging or bridge structures when printing.
- Infill: the internal structure (density/fullness) of a 3D printed part.

# Slicing with PrusaSlicer

---



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



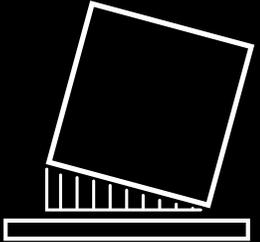
Does it need to be a quick print?

Or do you have time for high quality outcome?



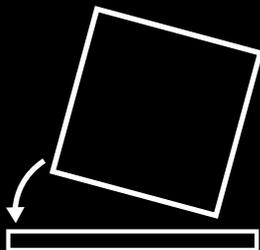
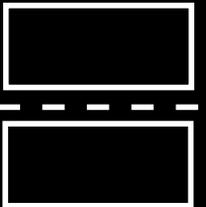
What percentage infill do you need?

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?

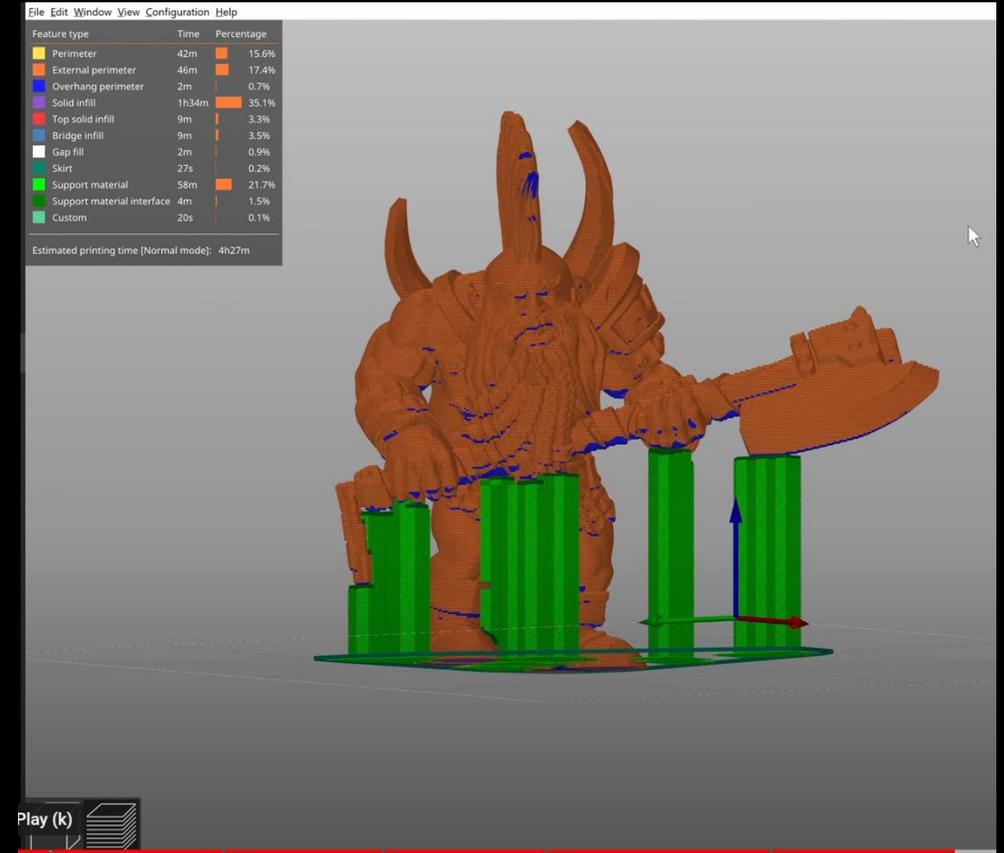
# Further info on Supports

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 are 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.

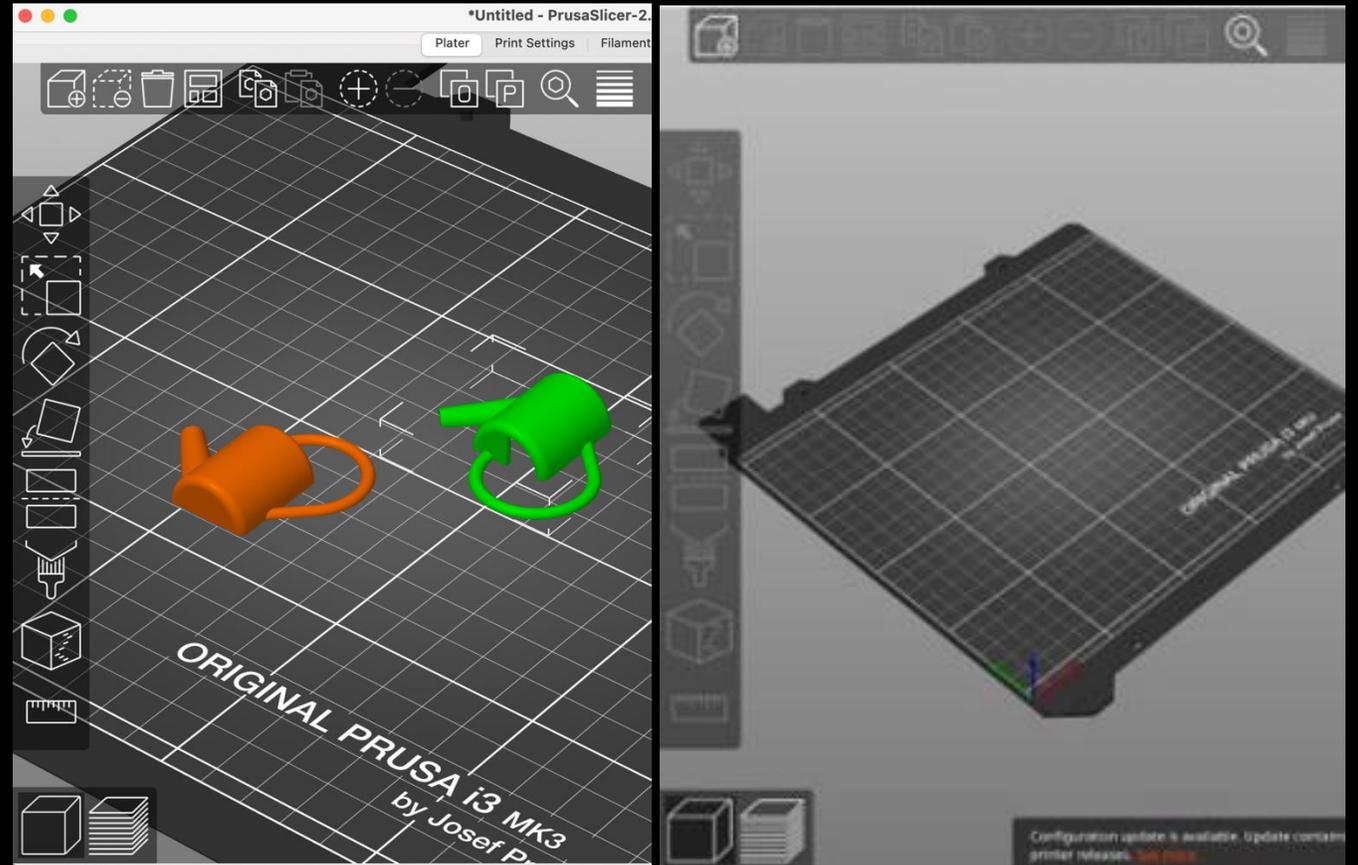


Source [Jakub Kočí](#)



# Slicing with PrusaSlicer

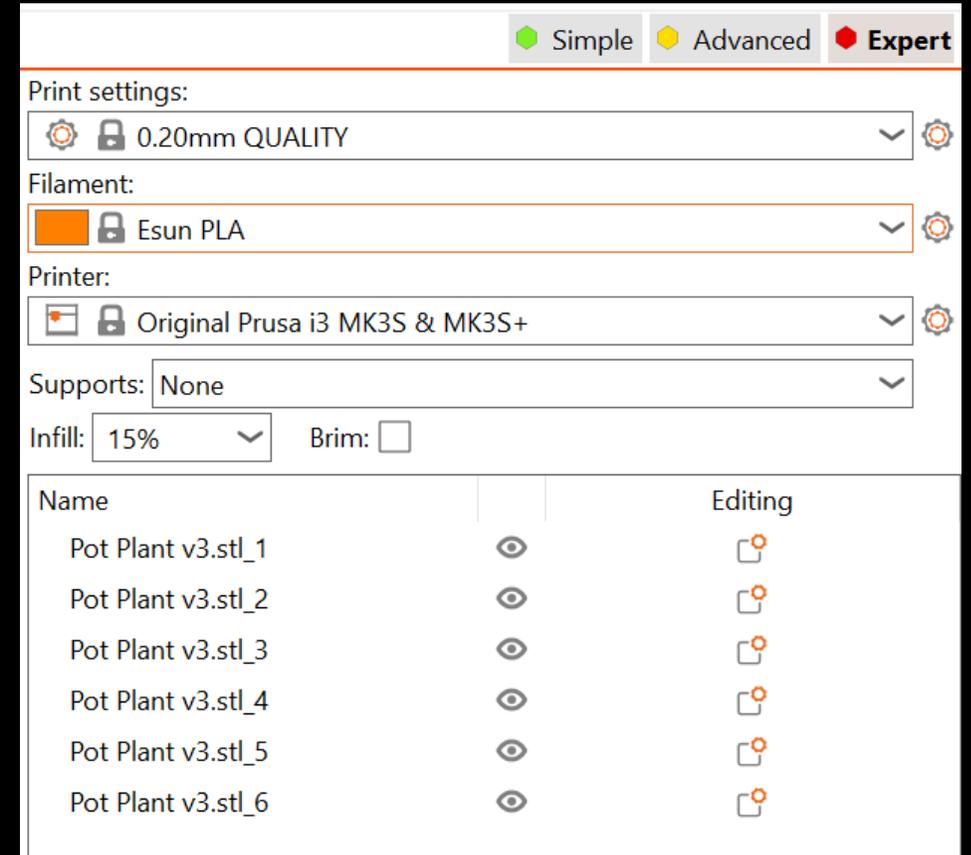
- Open the **PrusaSlicer** application. If you see the pop-up 'Configuration Update' is available, click to install the update before proceeding further.
- Use the **Add** button to import your model/s into PrusaSlicer.
- **Delete /Delete All** buttons to remove models.
- Use **Move, scale, rotate, Place 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.

The screenshot displays the PrusaSlicer interface with a sliced model of a container. The main window is titled '\*Untitled - PrusaSlicer-2.7.1 based on Slic3r'. The interface includes a 'Legend' panel on the left, a central 3D view of the sliced model on a grid, and a right-hand panel with various settings.

**Info Panel (Top Left):**

- Size: 44.74 x 39.49 x 10.00 Volume: 1697.95
- Facets: 40862 (3 shells)
- ⚠ 30 open edges
- Buttons: Slice now

**Legend Panel (Left):**

| Feature type               | Time | Percentage | Used filament |
|----------------------------|------|------------|---------------|
| Perimeter                  | 8m   | 16.7%      | 0.48 m 1.42 g |
| External perimeter         | 14m  | 29.6%      | 0.52 m 1.56 g |
| Overhang perimeter         | 38s  | 1.3%       | 0.02 m 0.07 g |
| Solid infill               | 5m   | 10.8%      | 0.33 m 0.98 g |
| Top solid infill           | 26s  | 0.9%       | 0.01 m 0.04 g |
| Bridge infill              | 1m   | 2.3%       | 0.03 m 0.10 g |
| Skirt/Brim                 | 25s  | 0.9%       | 0.02 m 0.06 g |
| Support material           | 10m  | 21.8%      | 0.46 m 1.38 g |
| Support material interface | 2m   | 4.7%       | 0.08 m 0.23 g |
| Custom                     | 6s   | 0.2%       | 0.02 m 0.06 g |

Estimated printing times:  
First layer: 4m  
Total: 47m

**Right Panel Settings:**

- Print settings: 0.15mm QUALITY (modified)
- Filament: Esun PLA
- Printer: Original Prusa i3 MK3S & MK3S+
- Supports: Everywhere
- Infill: 15% Brim:
- Object manipulation: X, Y, Z coordinates (144.1, 125.07, 5 mm), Rotate (0, 0, 0 °), Scale factors (100, 100, 100 %), Size [World] (44.74, 39.49, 10 mm)
- Info: Size: 44.74 x 39.49 x 10.00 Volume: 1697.95, Facets: 40862 (3 shells), ⚠ 30 open edges
- Sliced Info: Export G-code

# 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 always 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](mailto:appliedcreativity@slq.qld.gov.au)

