

# Standard Operating Procedure for 3D printing using:



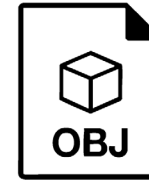
**Prusa Core one**



**PrusaSlicer**

## What you need to bring:

- A laptop with Bambu Studio slicing software installed:  
<https://prusaslicer.net/>
- Your CAD model exported as an STL, 3MF, or OBJ file.  
(Note: OBJ files may lose their scale. Check size before printing).



## What you need know:

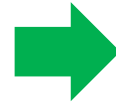
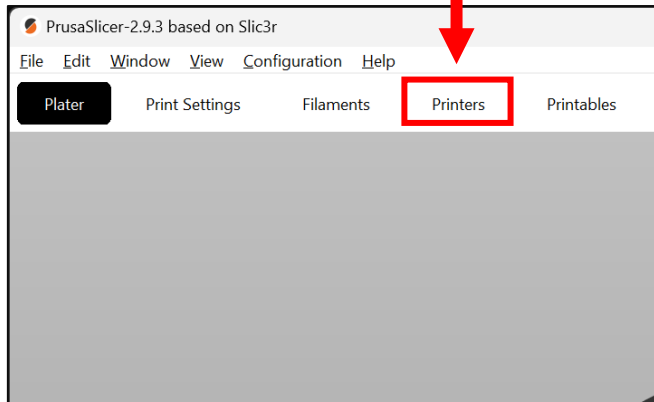
- The 3D Printer Farm consists of communal access machines for James Watt School of Engineering student work.
- You must understand how to set up and operate the 3D printers safely and effectively before use.
- You must adhere to the Standard Operating Procedure that follows, and the read the 3D Printer Farm Policy.
- If you are interested in using a larger machine or a material other than PLA, speak to the Technician present, or contact [cadgraphics@glasgow.ac.uk](mailto:cadgraphics@glasgow.ac.uk). See the [3D Printing page](#) for further details on other machines.
- If you are interested in using the Bambu or Ultimaker machines, see their Standard Operating Procedure.

[3D Printing in the  
School of Engineering](#)



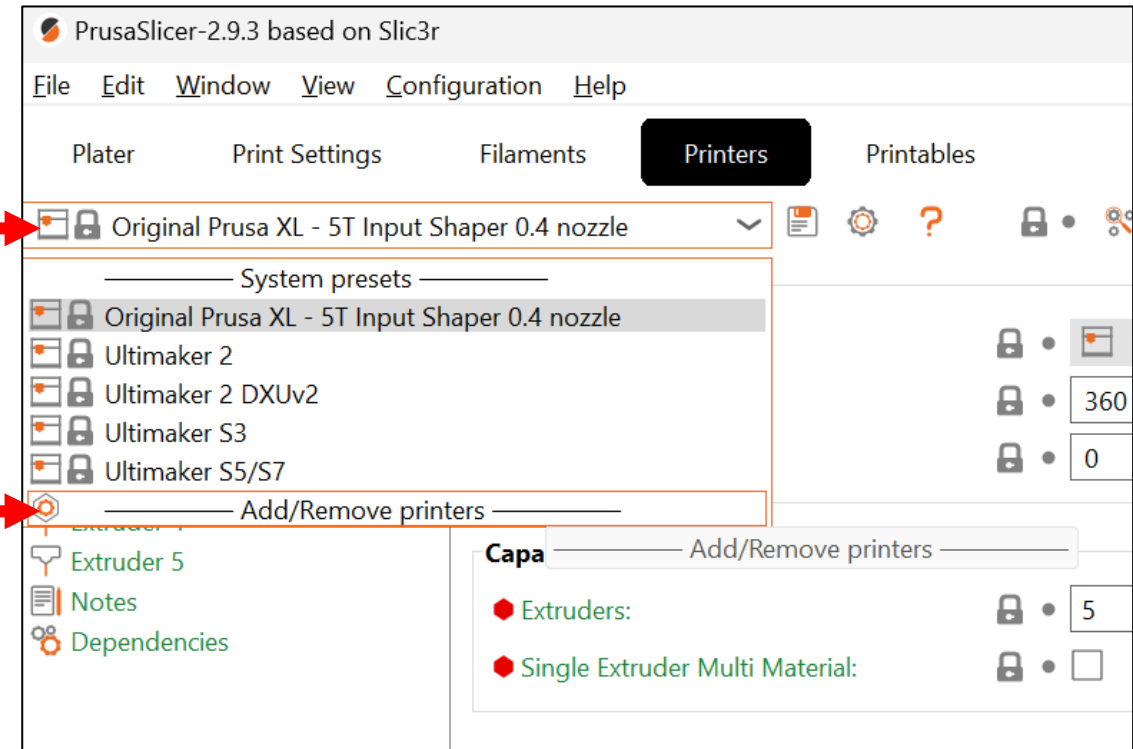
# PrusaSlicer: Add printer

- Click “Printers” from the top ribbon



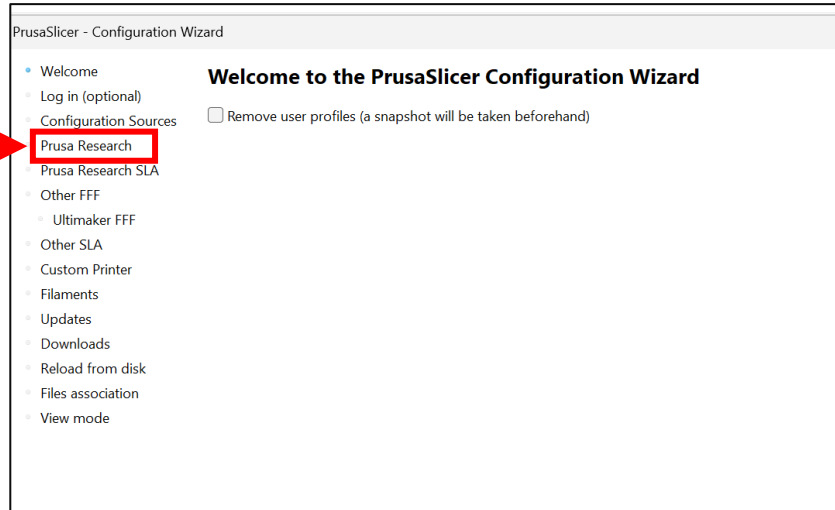
1. Click the drop down

2. Choose “Add/Remove printers”

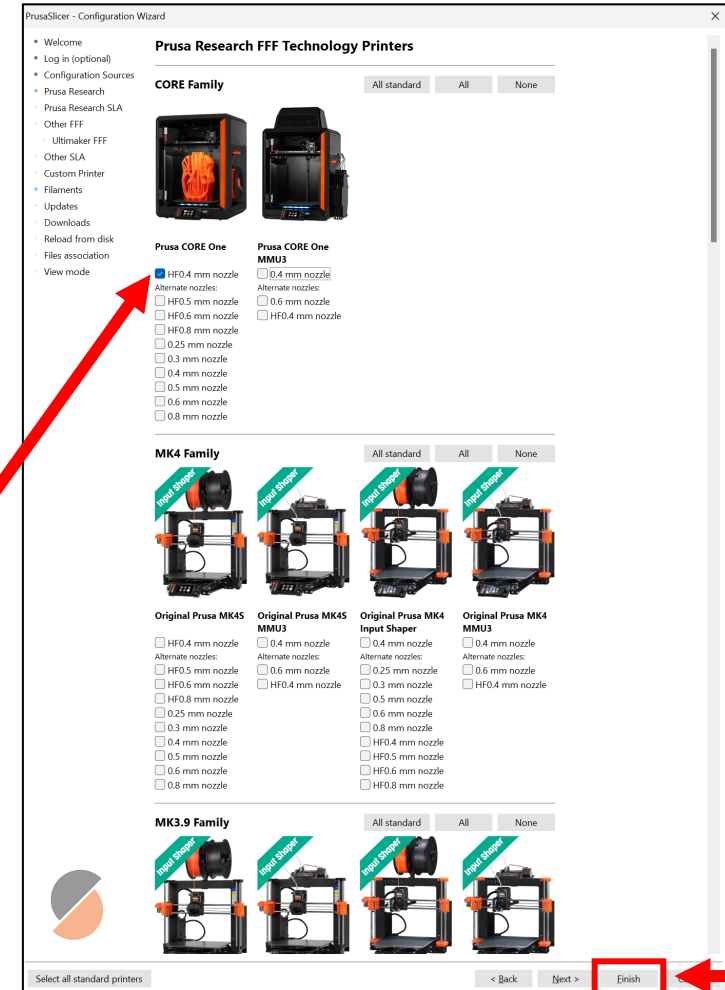


# PrusaSlicer: Add printer

- Click "Prusa Research"



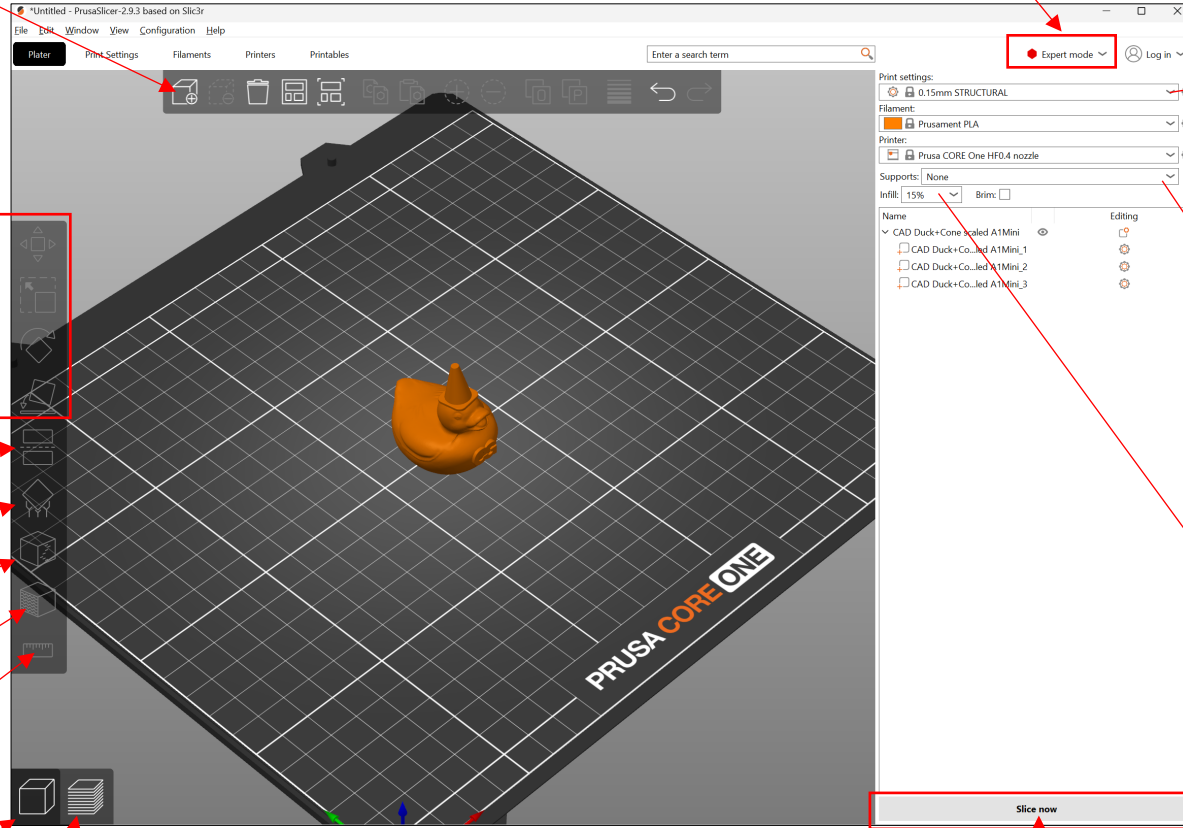
1. Check "HF0.4mm nozzle" under "Prusa CORE One" printer



2. Click "Finish"

# PrusaSlicer: Prepare Setup

- Add new model  
(Start here or drag and drop the file directly into the viewport)

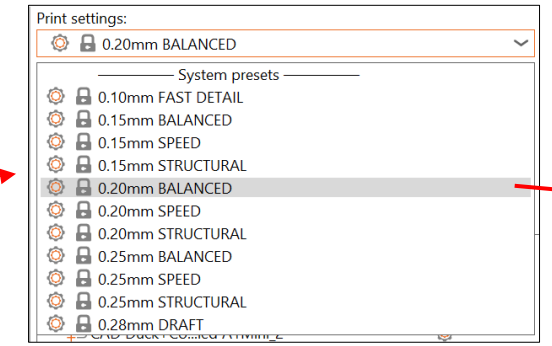


- Move / Rotate / Place tools
- Cut model tool
- Paint on supports
- Paint seams tool
- Paint fuzzy skin
- Measuring tool
- Edit mode

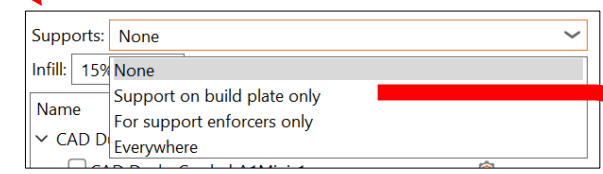
• Preview mode

- Change modes for more control over settings

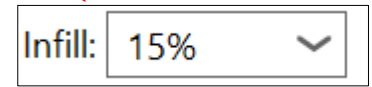
## Default print settings:



- Choose "0.20mm BALANCED" for the majority of prints



- Choose "Support on build plate only" (Everywhere is useful for trickier prints or use the "Paint on supports" tool for more control)



- For most parts, 15% Infill is plenty of strength (Try 10% for faster prints)

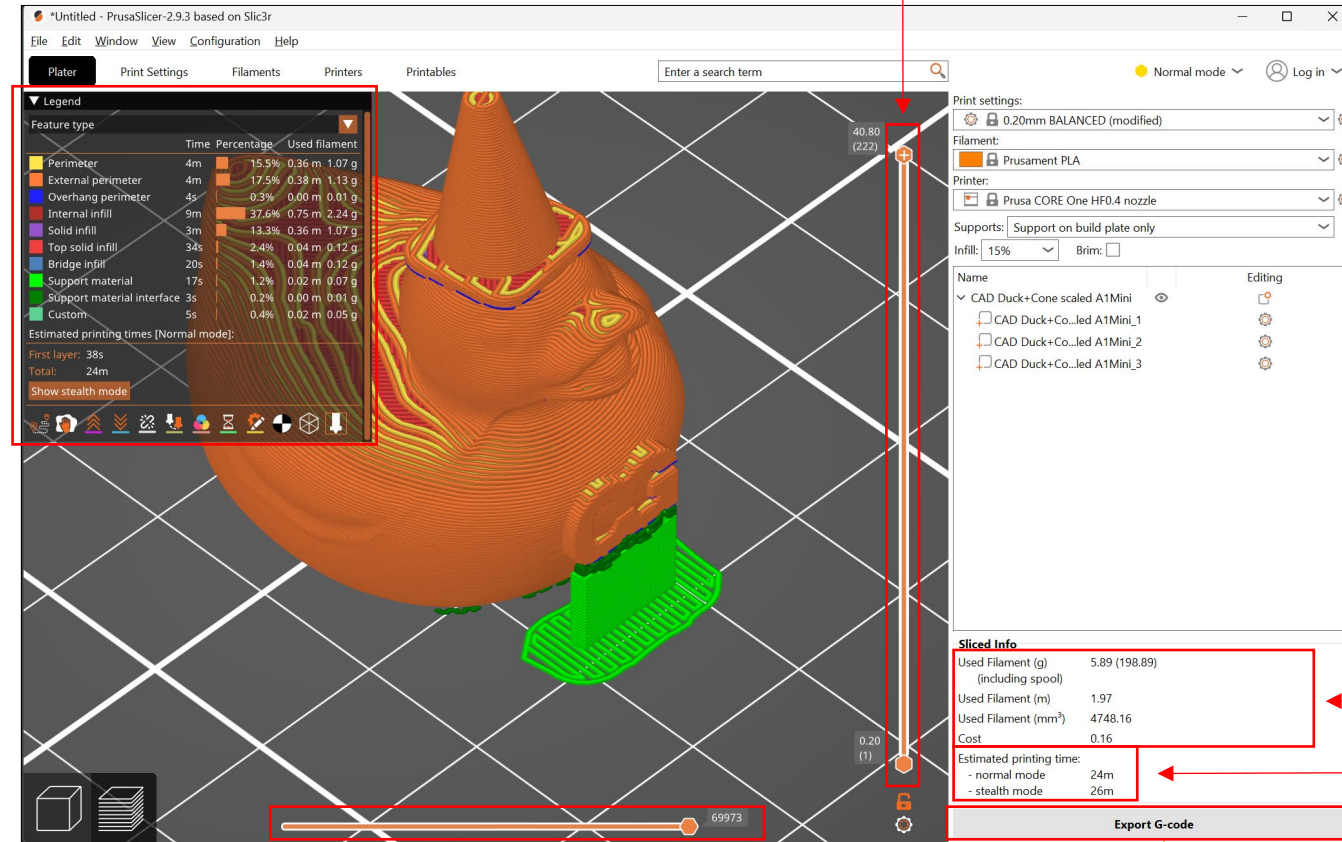
- Press "Slice now" to check print times and material use (This will automatically put you in preview mode)

# PrusaSlicer: Preview mode

- Move slider up and down to preview different layers

- Detailed print estimates

- In preview mode you can check how long the print will take and how much material you will use
- Change a setting in “Edit mode” and re-slice to see what difference it made
- In this mode, you can simulate the 3D printing process by using the sliders (Useful for problem solving tricky prints)



- Material usage

- Print time

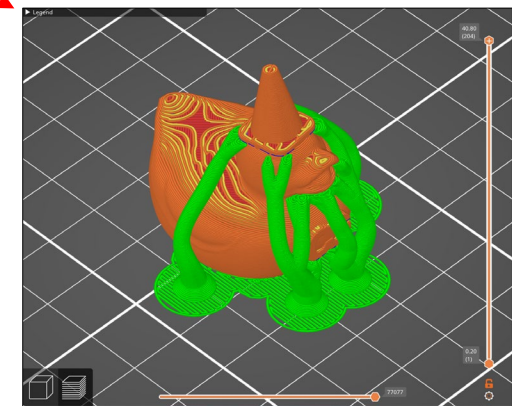
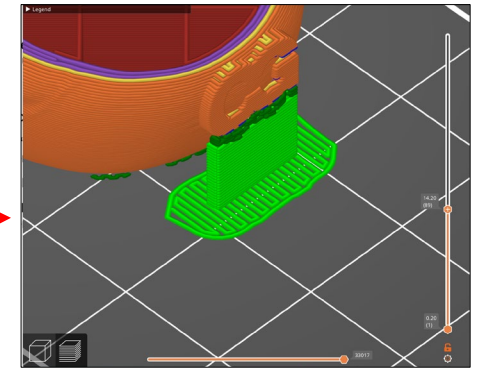
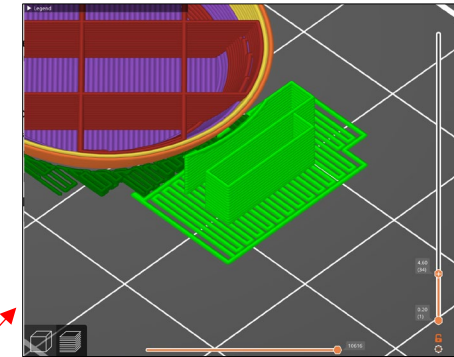
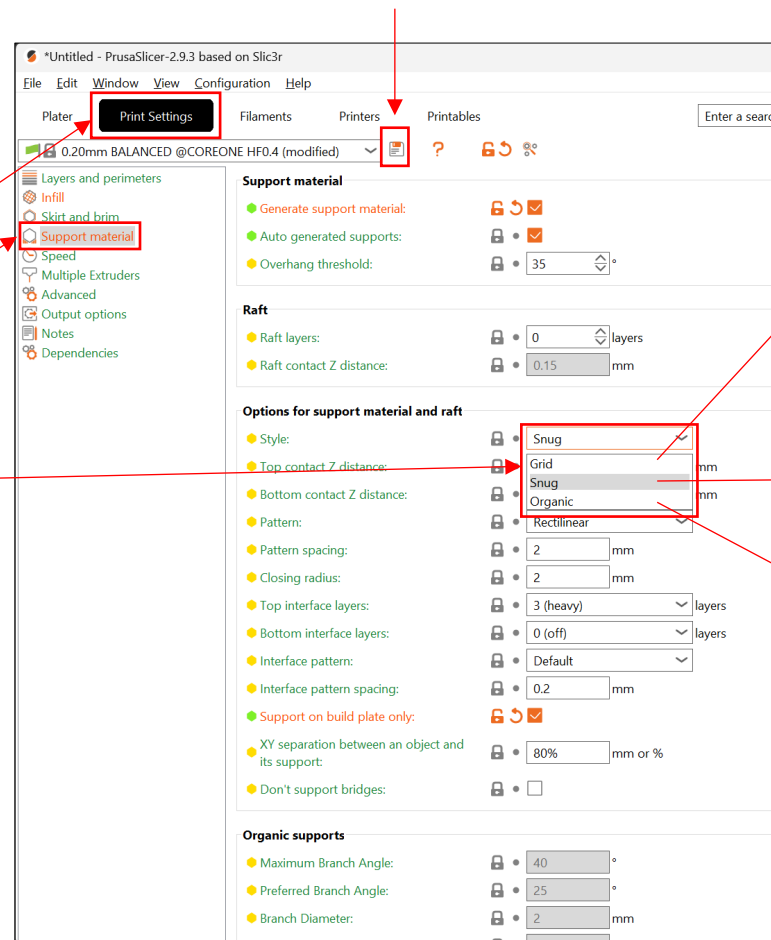
- Export code when ready to print

- Move slider left and right to preview how the printer will move during this layer

# PrusaSlicer: Support settings

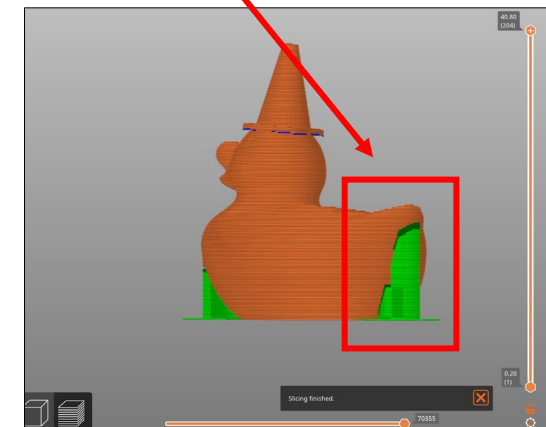
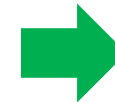
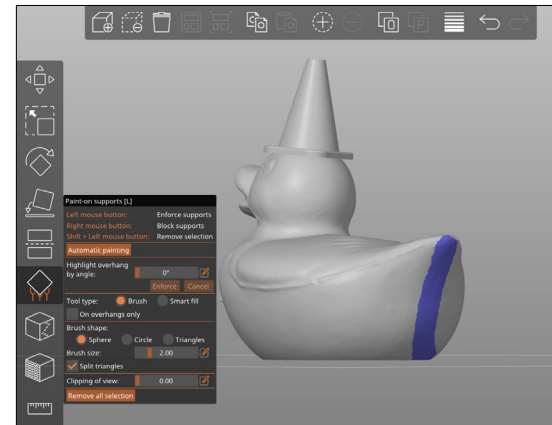
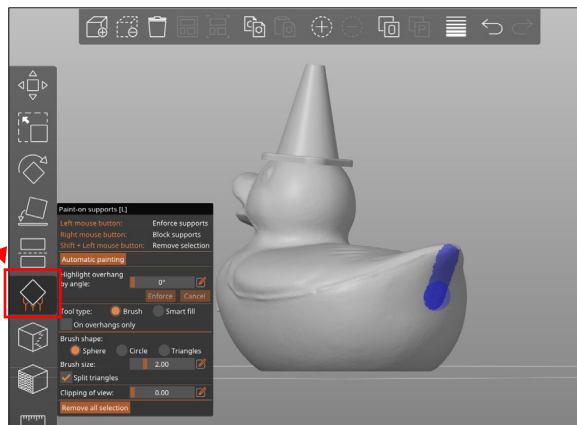
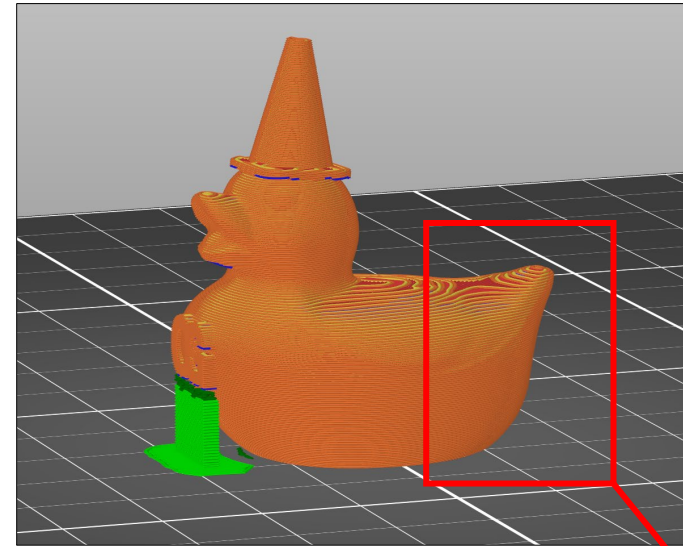
- Sometimes you'll want a different style of supports depending on your model
- For most prints, "Snug" is the easiest to remove after printing
- Choose "Print settings" from the top ribbon
- Then click "Support material"
- Choose the style from the drop-down menu
- NOTE: You will need to press "Slice" from the "Plater" window every time you change a setting to preview / generate the code
- You can change more settings to suit your print AND save them as slicing profile (Save icon at the top)

- Save slicing profile



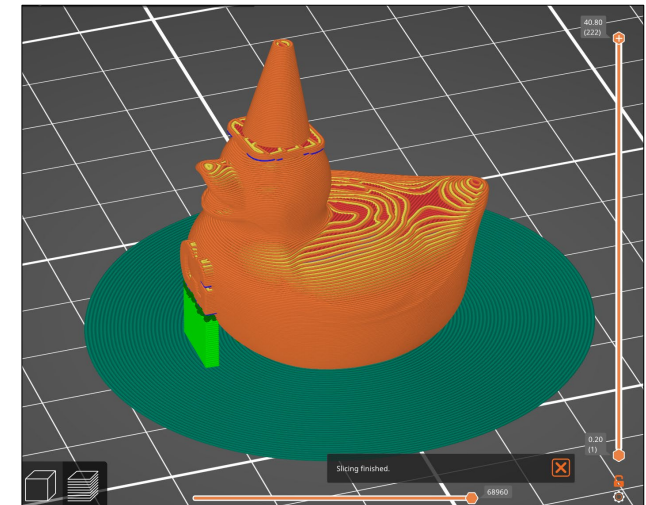
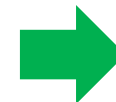
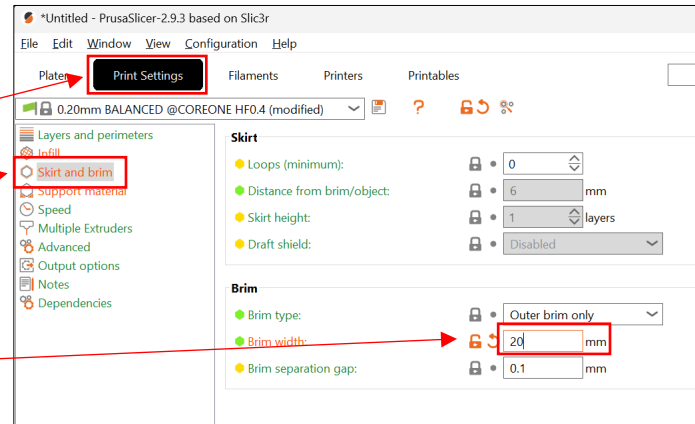
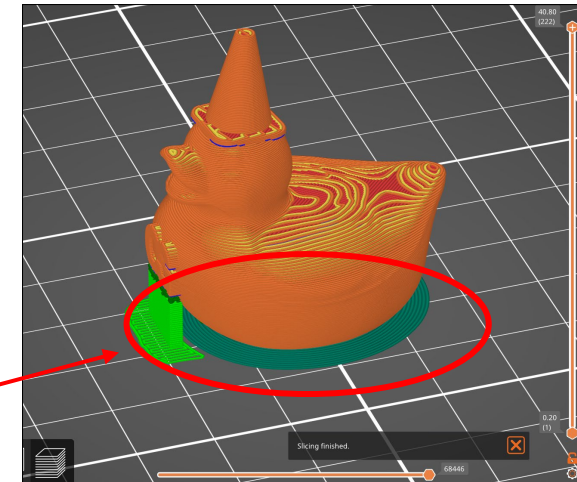
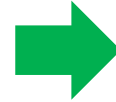
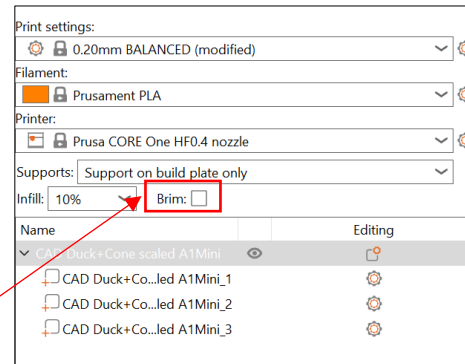
# PrusaSlicer: Support settings

- Sometimes the slicer may have a difficult time auto generating supports for you
- To overcome this, you can “Paint” where you want supports to generate
- In “Edit mode” click the “Paint on supports” tool and paint on the model to the build plate then click “Slice”



# PrusaSlicer: Support settings

- Sometimes your model may have difficulty sticking to the bed
- DO NOT use glue sticks on this printer! Instead, use these settings
- From the “Printer” window in “Edit mode” click “Brim” then “Slice”
- This adds extra material or a “Brim” around the model to offer a larger footprint which helps adhere to the textured print bed
- To alter these settings, go to “Print settings” from the top ribbon
- Choose “Skirt and brim”
- Change the “Brim width” then slice the model from the “Printer” window

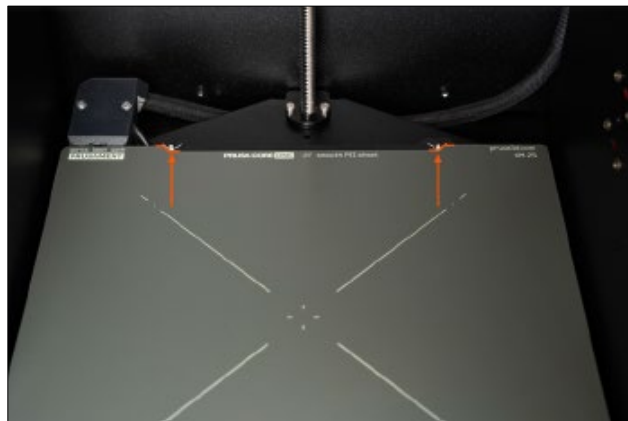


## Control panel:



1. Rotary Knob: Turn to navigate and press to confirm selections
2. Touchscreen: Tap and swipe to navigate and select items in the menu
3. Reset Button: Instantly restarts the printer. Useful when an immediate stop is needed

## Print bed:



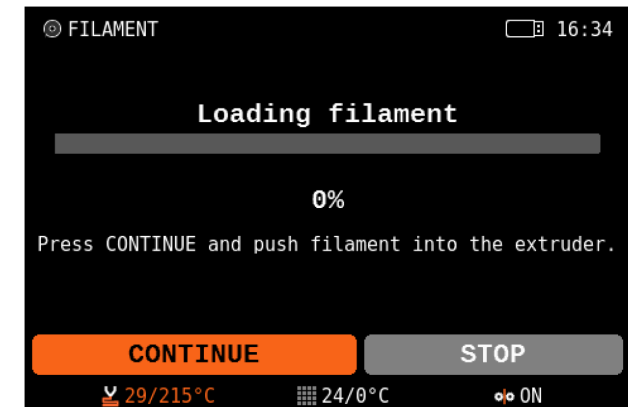
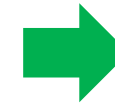
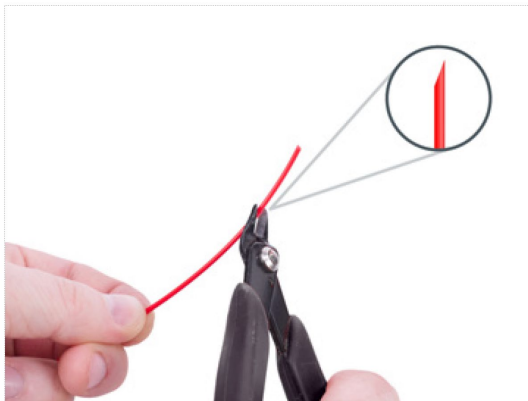
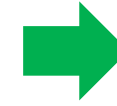
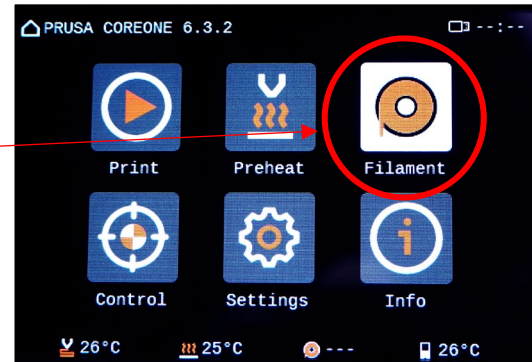
- To remove the print bed, lift it up by grabbing the front edge and remove from the printer
- To replace the bed, hold onto the front edge and line up the back edge cut out with the two pegs at the back of print area. Next, slowly lower the print bed to the magnetic bed.



**BEWARE OF PINCHING AND HOT COMPONENTS!**  
If in doubt, ask a technician for assistance

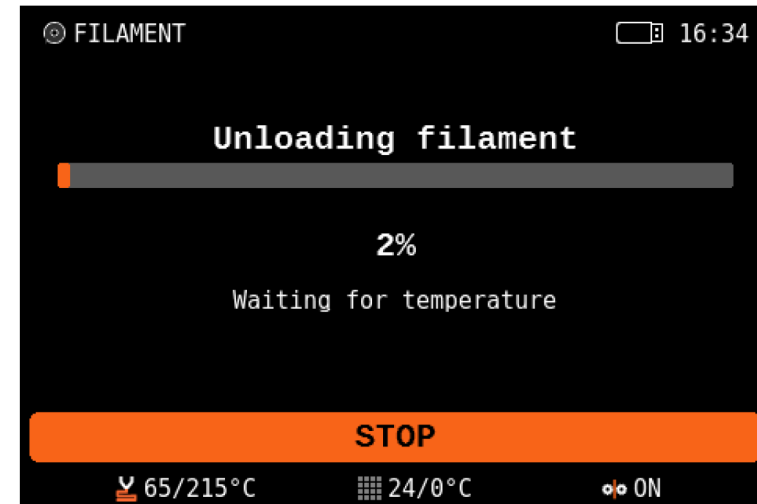
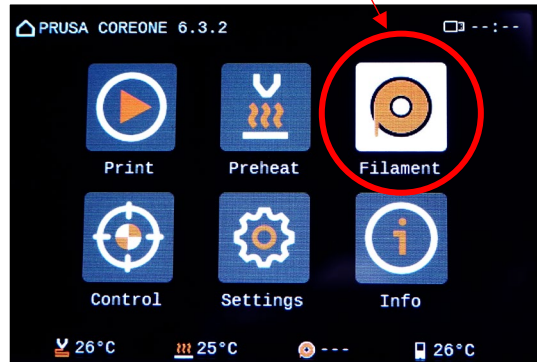
## Loading filament:

1. Select “Filament” on the control panel
2. Choose “Load Filament”
3. Select “PLA”
4. Cut the filament on an angle to create a narrow point
5. Feed the filament into the PTFE tube all the way to the nozzle and press “Continue” on the control panel
6. The printer will take the filament and begin purging material and will ask you to press “Continue” when this happens



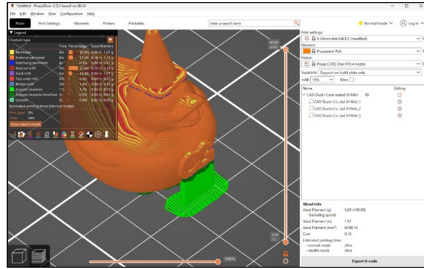
## Unloading filament:

1. Select “Filament” on the control panel
2. Choose “Unload Filament”
3. The printer will heat up and tell you when to pull the material out



# Overall workflow:

1



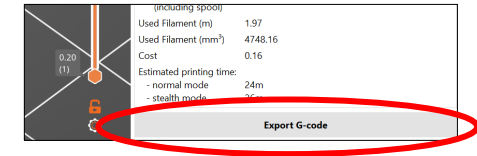
Load your model and experiment with settings until you are satisfied with your print estimates

2



Remove the USB from the Printer and plug it into your machine

3



Export your file onto the USB

4



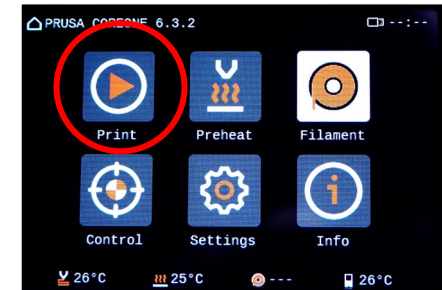
Plug the USB into the control panel of the printer

5



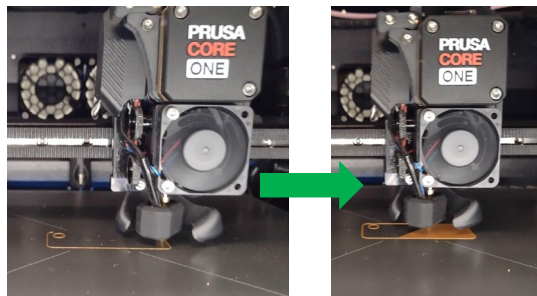
Ensure the printer is loaded with enough PLA material for your print. Check this against your print estimates

6



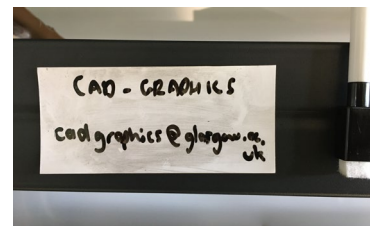
Press print on the control panel, navigate to your file and print

7



Wait for the printer to print the first layer. DO NOT WALK AWAY! Make sure material is sticking to the print bed before leaving it

8



Write your email on the whiteboard label near the printer



Do not interact with the inside of the machine or a printing model while printing is in progress. Pause or stop the job first.

# Checklist: Before you start

- Printing a large box or plate? → Consider redesigning for laser cutting and submit your DXF file to
- Does your file take >24 hours to print? → It is not suitable for these printers. Adjust infill and layer height settings to reduce time, or submit your STL/3MF file to be printed on our industrial machines
- Do your settings match the printer, material and nozzle specified in the settings?
- Have you checked that the spool has enough filament for your print?
- Do not use more than two printers at one time if your models take more than 2 hours to print.
- Ensure the magnetic build plate is laid flat and even on the platen.
- Wait for the first layer of filament to print before leaving. Put your details on the labels provided.

## Service Request System

[Technical Services  
Request System](#)



# Overall workflow:



Do not interact with the inside of the machine or a printing model while printing is in progress. Wait for the job to finish.

9



5 min

When the print has finished, allow the print to cool for 5 minutes

10



Remove the build plate and flex it to remove your part. Do not remove your model with the plate still in the machine.

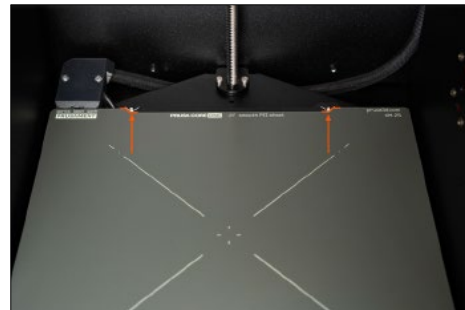
11



**Do not use metal scrapers on this print bed.** If you need to use a scraper, use a plastic one

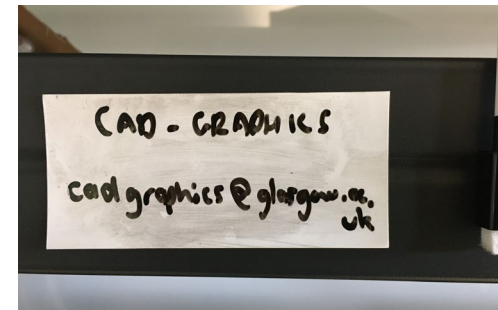


12



Return the build plate to the printer, ensuring the back edge cut out aligns with the two pegs

13



Clean the whiteboard, removing your details.

**The printer and surrounding area should be left in the same condition as you would like to find it.**

# Checklist: When you are finished

- Leave the machine and work area in the condition you would like to find it.
- Take the magnetic build plate out of the machine when removing your components. Do not remove printed objects from the plate when it is in the machine. This disrupts the printer's calibration.
- Remove all material from the build plate, and use the **plastic scrapers** provided if necessary. Do not leave build plates covered in material.
- Place the build plate back in the machine, flat and even on the platen.
- Put any tools used back in the boxes. Put any remaining waste material in the bin using the dustpan and brushes provided.
- Do not store private materials at the 3D Printer Farm. Unattended materials may be deemed fair use. Remove private spools from the facility after use.

## Service Request System

[Technical Services  
Request System](#)



## 3D Printer Standard Operating Procedures, Videos and guidance

[3D Printing in the  
School of Engineering](#)



## For enquiries, training, or troubleshooting

1. Speak to the Technician present

2. Contact:

[cadgraphics@glasgow.ac.uk](mailto:cadgraphics@glasgow.ac.uk)

- Prusa Core One Knowledge Base: <https://help.prusa3d.com/product/core-one>
- Print quality guide: <https://www.simplify3d.com/resources/print-quality-troubleshooting/>
- Prusa material table: <https://help.prusa3d.com/filament-material-guide>