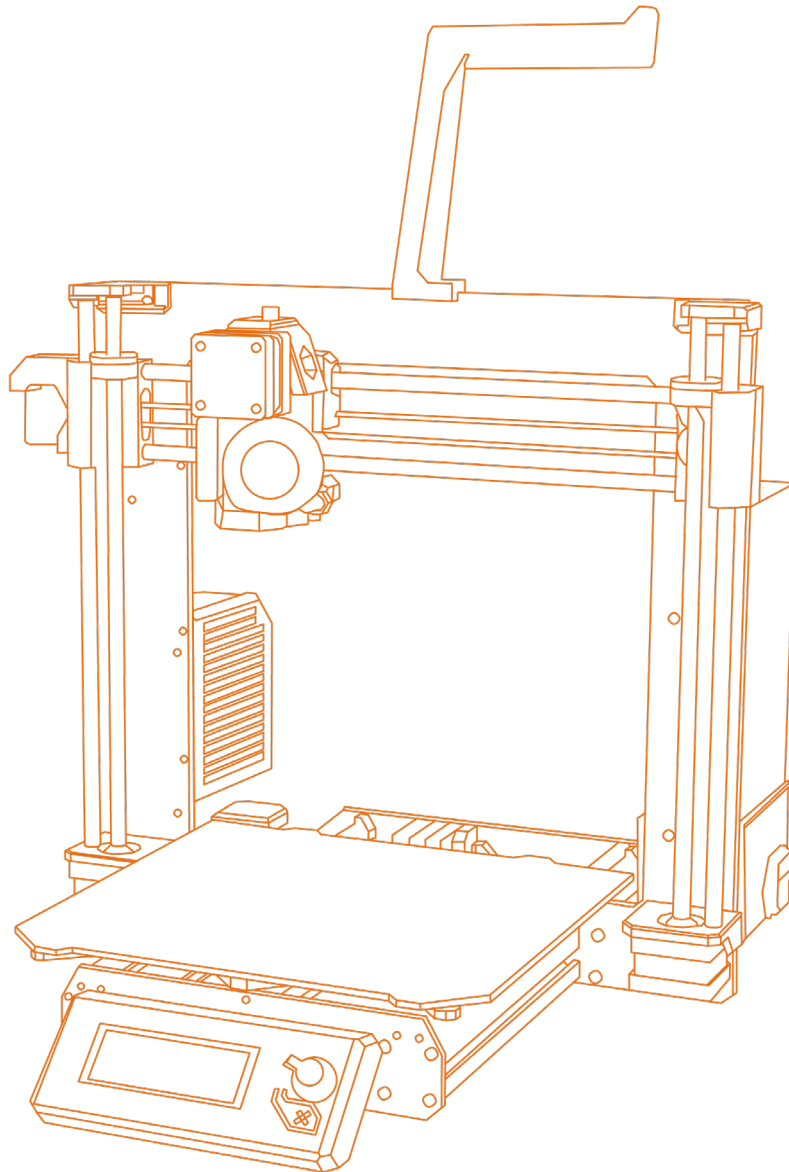


HOW TO SET UP AN .STL FILE FOR 3D PRINTING



# 3D

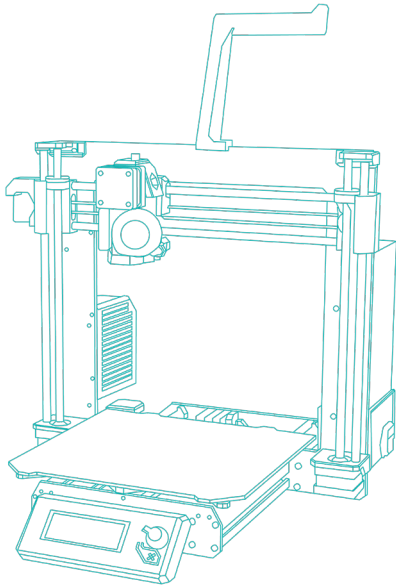
PRINTING GUIDE

## INTRODUCTION

In the Fabrication Laboratory we have 3 types of 3D printer available for use.

We have both plastic and plaster printers, but they work in the same way, both are additive processes that slice a 3-Dimensional CAD Object into layers before printing it layer-by-layer.

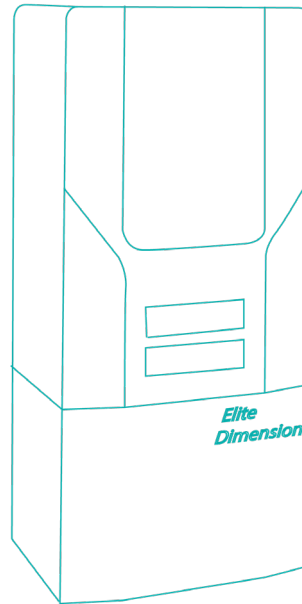
### PRUSA I3 MK3S



PLA Plastic Filament

Bed Size 250 x 210 x 210 mm

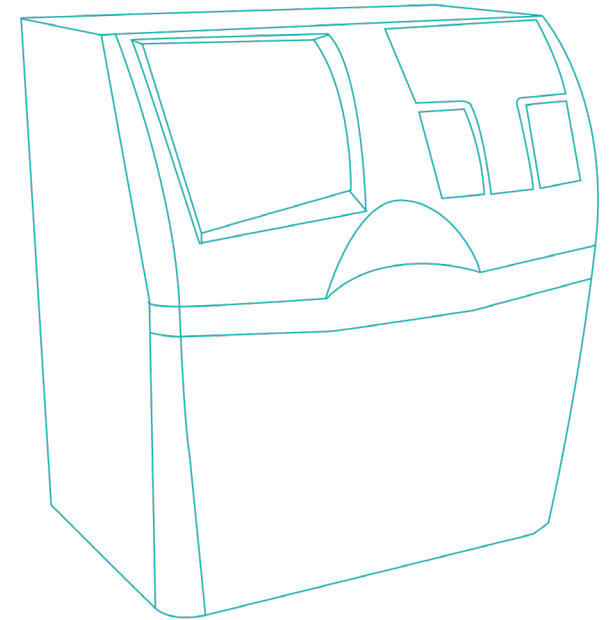
### DIMENSIONS ELITE



ABS Plastic Filament

Bed Size 203 x 203 x 305 mm

### ZCORP 250 & 450



Plaster of Paris

Bed Size 236 x 185 x 127 mm

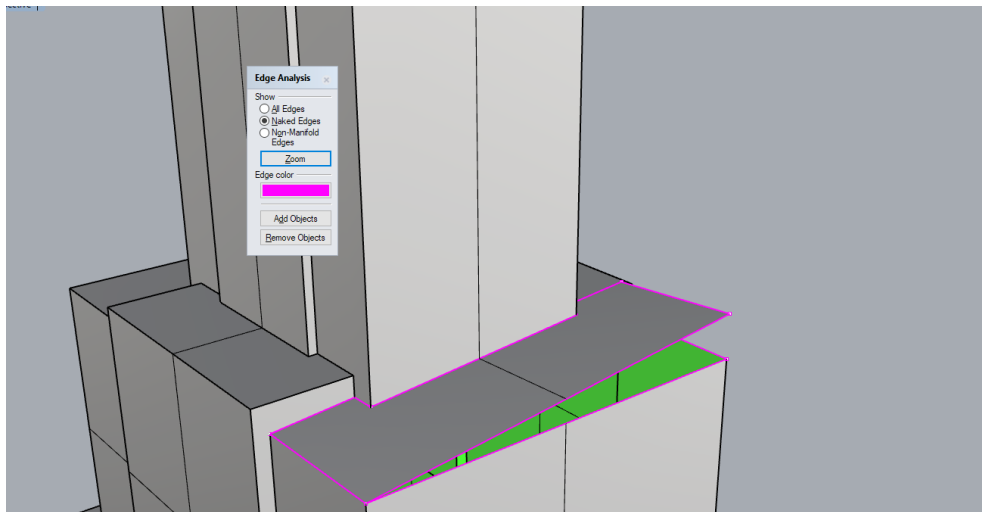
## HOW TO..

## FILE PREPARATION: EDGES

When creating a 3D Print file for one of the machines, it is important to remember that in order for the machine to be able to process what is on the 'inside' and 'outside' of your object, the object needs to be fully 'watertight'.

This is done by making sure your object is completely solid, without any gaps in its surfaces or intersecting surfaces. To ensure that your object is completely watertight, and has no gaps or issues, you must run the command 'ShowEdges', this will check for 'Naked Edges' and 'Non-Manifold Edges'. This will highlight the problem edges in rhino automatically. These need to be fixed before you export your object as an .stl file.

### NAKED EDGES

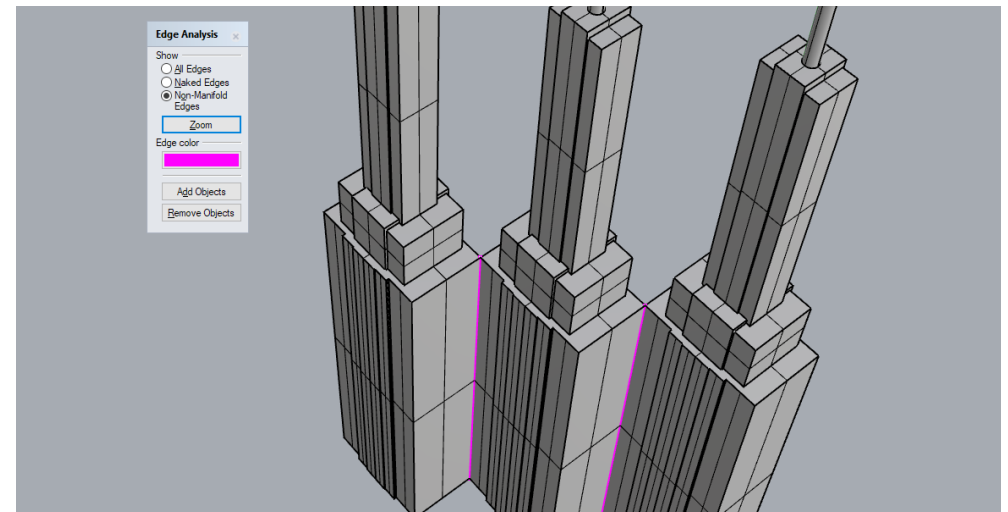


**Naked Edges** are a result of an imperfect seam between edges. This could either be a **hole** in the model or **misalignment** between two surfaces. To fix these, one surface will need **rebuilding**, relative to the other surfaces around its edges. This can be fixed by **bridging** the gaps, or **rebuilding** surfaces.

- + Explode your object and delete any problem surfaces.  
Then, using 'EdgeSrf', you can build a surface that touches the edges selected using this.
- + Alternatively, start from scratch using a 'PolyLine', then run either 'PlanarSrf' or 'Patch' to create a surface with your new boundary.

### HOW TO..

### NON-MANIFOLD EDGES



**Non-Manifold Edges** are edges that have **no density**. It means that although two solid object may share a edge, there is nothing physically joining them. Thinking back to the 'watertight' analogy, water cannot pass between those objects. Non manifold Edges are:

- + In **contact** with more 2 or more surfaces,
- + There is an **extra** surface in the model which is unnecessary
- + Two or more objects **share** a single edge which has been joined.

Ensuring that none of the situations listed above occur, you should judge which way is best to resolve any Non-Manifold edges.

## FILE PREPARATION: SURFACE DIRECTION

Everything in rhino has a **direction**. Lines have a start and an end, surfaces have a front and a back, solids have an inside and an outside. Sometimes when you're dealing with more complicated shapes and objects, these directions can become mixed and cause some issues when 3D Printing. The direction of your surfaces are called **Surface Normals**.

- + To check your surface normals are facing the right way, key in '**Dir**' (for direction), if you click, you will be able to **flip** the direction of all objects simultaneously. If this doesn't solve any issues, you will need to **explode** your object and **Individually flip** the surfaces causing you problems.
- + To help you see the direction of your faces, you can turn on "**Shaded backface**", which can be found by typing in '**DisplayProperties**', and by looking into the **Object Settings** subheading.
- + You **cannot** flip the direction of closed polysurfaces

In this example with shaded backfaces turned on, the green surface needed to be flipped, as the direction of the front face (indicated by the white arrows) is facing inwards, it should have looked like the corrected version, on the right.

## WORKING WITH A MESH

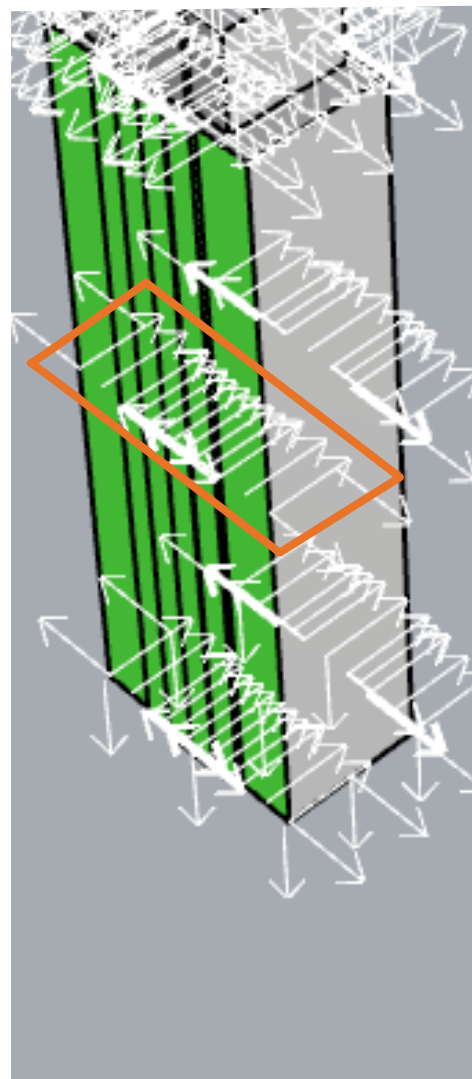
If you have generated your mesh using software other than rhino, it is worth checking the quality of your mesh in rhino, as **it is easy for a poorly made mesh to affect the quality of your print**.

In Rhino, you can check a mesh using '**\_Check**' if you find that you have multiple Surface Normal directions, the '**UnifyMeshNormals**' command should help.

## HOW TO..

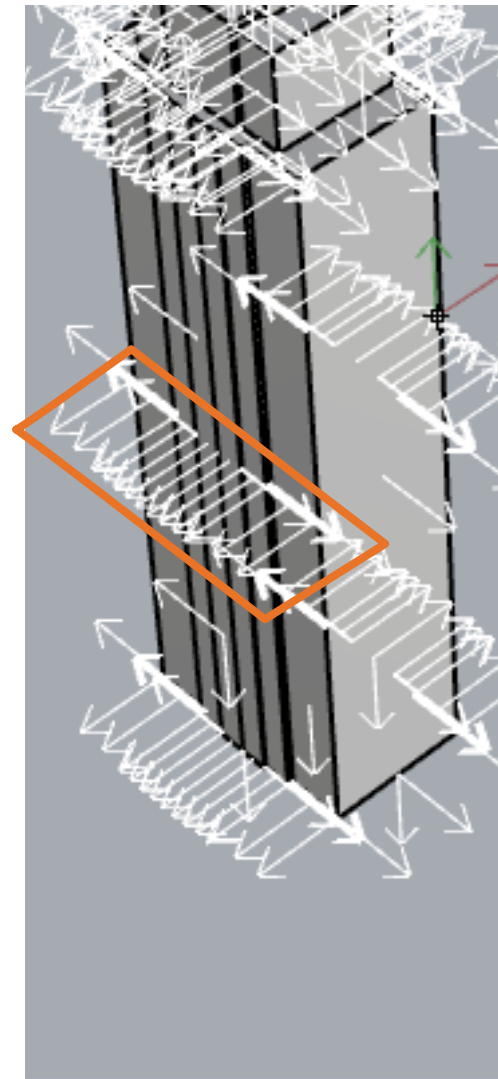
### INCORRECT EXAMPLE

NOTE how the directional arrows on the green face point towards the centre of the object.



### CORRECT EXAMPLE

Your surface normals should be facing away from the solid mass of your object.



## FILE PREPARATION: EXPORTING YOUR MESH

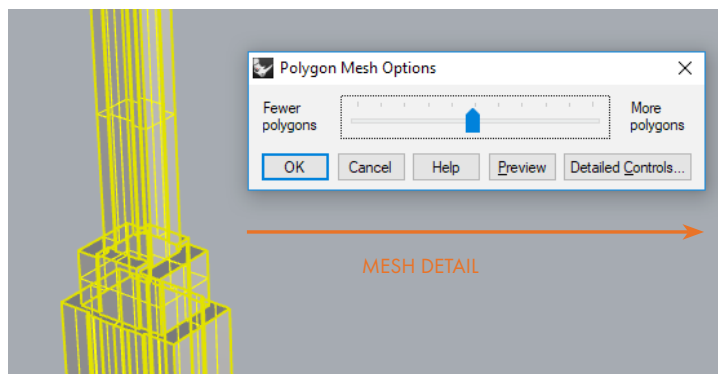
When using rhino you model with **NURBS** Surfaces, an **.stl** file is a **MESH** file type which is what is needed for 3D Printing, getting from one to the other translates free-form NURBS into triangulated meshes, which can result in imperfections during this translation.

The 3D Printers in the lab recognise **stereolithography** files, or **.stl** files for short.

Once you have created your object for printing you must export it as this file type **UNLESS** you are printing in colour on the ZCorp Powder printers, which will require a **.vrmf** file.

To create a mesh from your rhino NURBS, select your object and use the '**Mesh**' command.

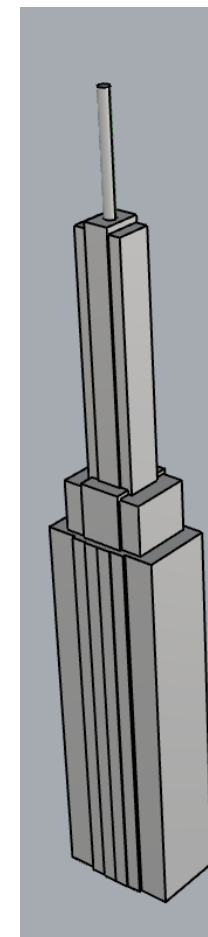
You will be given options for the number of polygons that your mesh will be made from,  
more polygons = better translation but bigger file sizes.



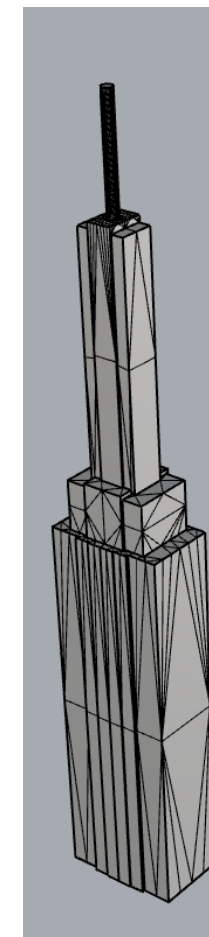
Once you have your mesh, **export** it as an **.stl** file from Rhino. If you have multiple parts or components, each **MUST** be saved as a **separate file**, as groups of objects **cannot** separately be edited later on.

To save your object, select it, use the '**Export Selected**' command, and then on the filetype drop-down, select the **stereolithography** option (**.stl**)

**NURBS**



**MESH**



## Prusa

Additional Steps are **required** for the Prusas, so **continue** onto the next page to find out how to use these machines.

## DIMENSION AND ZCORP

Once exported, you are able to **submit** a file to print on these machines, Make sure that the file name is correctly formatted, i.e;

'date'\_'your name'\_'machine'\_'Filament colour'  
10-12-2018\_Jane-Doe\_Abs\_Black

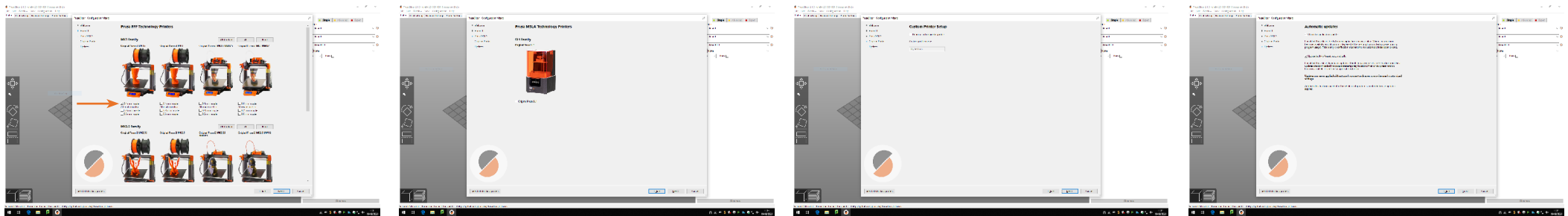
## HOW TO..

## PRINTING WITH A PRUSA: SETUP

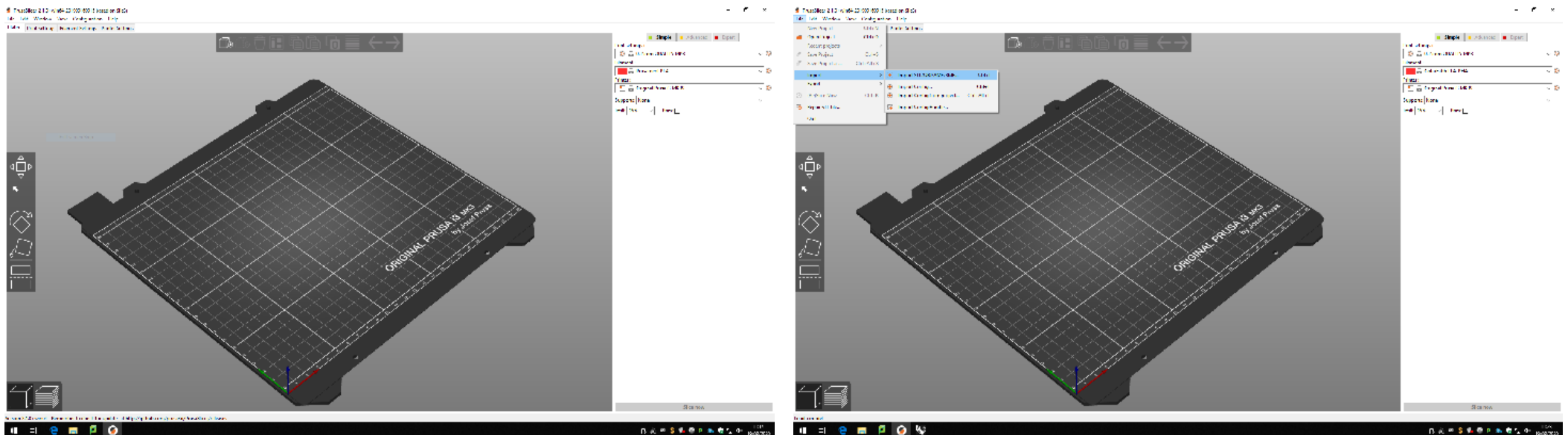
To print on the Prusas, you must process your .stl file through downloadable freeware called 'Prusa Slicer'.

In here you are able to rotate and arrange your pieces for printing, it is always recommended you rescale in rhino, to avoid any parts becoming too small to print correctly.

Prusa Slicer is **free to download** from Prusa's website, or you can use any of the **CAM Lab computers** to process it.



PRUSA MK3S 0.4MM NOZZLE

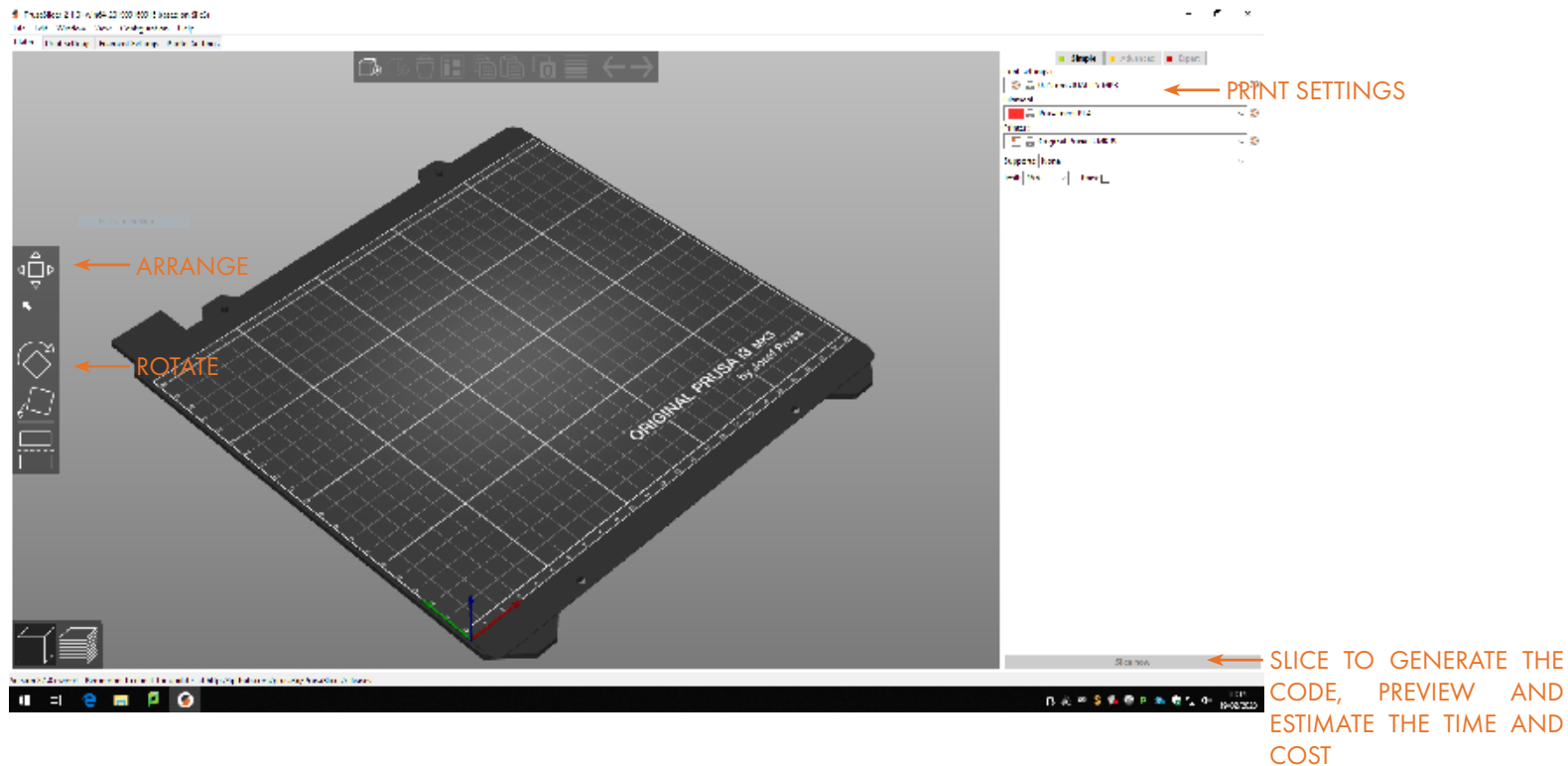


HOW TO..

## PRINTING WITH A PRUSA: SETUP

To print on the Prusas, you must process your .stl file through downloadable freeware called 'Prusa Slicer'. In here you are able to rotate and arrange your pieces for printing, it is always recommended you rescale in rhino, to avoid any parts becoming too small to print correctly.

Prusa Slicer is **free to download** from Prusa's website, or you can use any of the **CAM Lab computers** to process it.



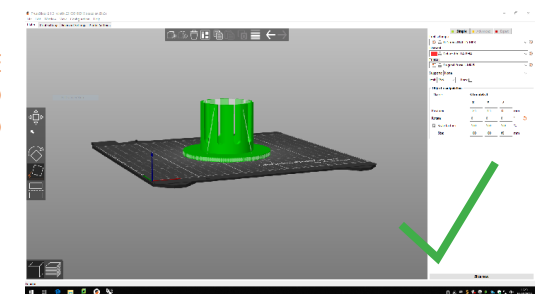
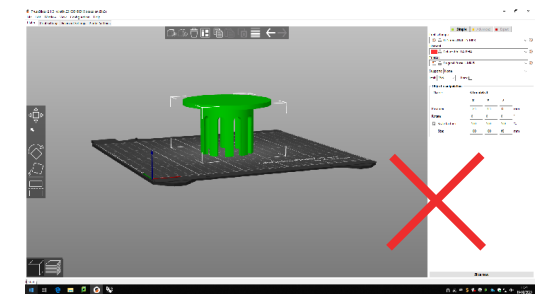
## NOTES

Prusas will ONLY run jobs when the Laser Lab is open.  
They do not run overnight.

Make sure that the bottom of your object lies AS FLAT AS POSSIBLE on the print bed.

Due to the way Prusas print SUPPORT, fine details and delicate parts are not always possible on this machine.

Avoid multiple tall objects in one job, as they have a tendency to fall over, this can be potentially avoided by printing with a 'brim'.

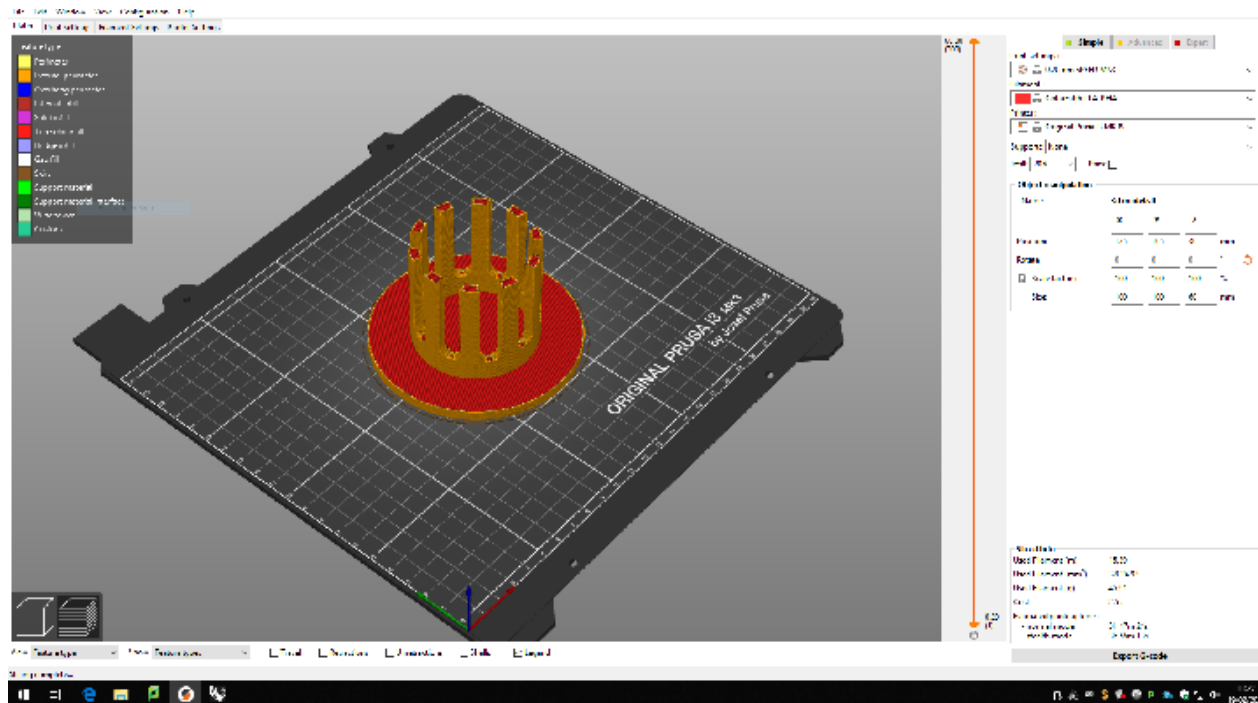




## PRINTING WITH Prusa: SETTINGS

Once you are happy with the position, scale and orientation of your objects, It is time to set up the **settings** for your print job.

To do this, click on **"Print Settings"**, then go into **"Custom Settings"**. This will bring up a window in which you can change just about everything to do with your job.



## KEY SETTINGS

### Print Settings

These are the default presets. The distance (0.2mm for example) is the distance between each printed layer. If your model has small detail, you should use a smaller layer height - but this **greatly** effects print time.

### Filament

This is the material you print with. This will be ClorFab PLA-PHA.

### Printer

If you set up the Prusa slicer correctly this should appear as Original Prusa i3 MK3S.

### Supports

The Prusas prints using molten plastic, which means as it prints, it **needs to be supported** - or else you will get **strings** in your model or **gaps** in your walls. If you have overhangs, it might be necessary to print with supports, bear in mind these need to be removed, usually **with pliers**, so the supports must be accessible.

### Infill

This controls how solid your object will be,

**0% Infill = Hollow Object**, **100% Infill = Solid Object**.

this dramatically changes the print time, so is worth tweaking depending on your model, remember that this becomes the **internal support** for your model.

### Brim

This is used for printing objects that are tall but with a small footprint. It increases the amount of material that adheres to the bed.

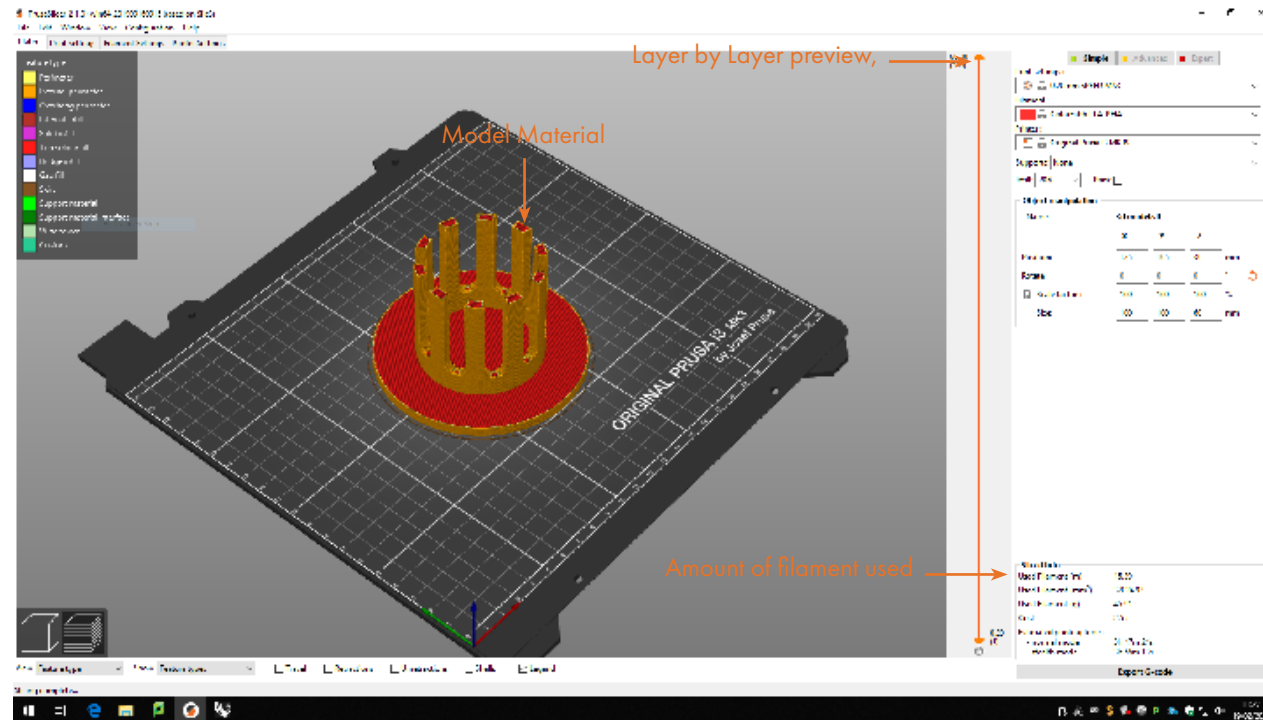


## PRINTING WITH PRUSA: PRINTING

When happy with the setup of your print, preview it by clicking on the “Slice” button on the right hand side of the screen.

This will generate a preview of your job, allowing you to review the print layer by layer, see how the supports and raft look as well as tell you exactly **how long** it will take to print.

You are able to zoom in and see how the details of your model look, you can easily change any of the settings **before** you export by going back to the settings panel and updating whatever settings need to be changed.



## FINAL STEP: EXPORTING AND PAYING FOR THE PRINT

If your job **looks as you expect** you are ready to export.

Take note of the amount of filament used in meters and pay before you start. Place the receipt in front of the printer.

Export to an SD card (we sell them very cheaply in the shop)

Choose a machine, if you want to change the filament you may.

Clean the bed before printing with isopropyl alcohol spray and blue roll.

Insert the SD card and print.