# Setting up your design for printing

* Download or export your **.stl** file from your design
* Open the Prusa Slicer software.
* Select the **Advanced** menu setting o
* Import the **.stl** file by using the Add symbol or selecting **File/Import** **STL/OBJ/3MF/AMF**
* Makes sure you only import one file! If you use the **Add** tool this will be obvious but using **File/Import** you can sometimes import 2 files by mistake ontop of each other which can damage the printer or worse, make it overheat dangerously. You can check in the File Name list underneath the print setting windows. You’ll need the **Advanced** menu settings to see that.
* Define your print settings

These are really important to make sure you have a successful print! Different filament and different models need different settings!

We’ve been using 0.75mm PLA to being with as it’s non-toxic (but not food safe). So although you could safely make a drinking straw holder, you couldn’t make a drinking straw.

Example settings that we have used sucessfully are:

**Print settings:**

Pretty-PLA between 0 – 0.2 mm:

**Filament:**

Prusa PLA

**Printer:**

Original Prusa I M3 MK3S

Select these settings using the drop down menus. You can add other settings by selecting the

* Select **Slice Now** and wait for the Slicer software to slice your model into G-code the printer understands
* Note the estimated and print time.

Generally you want the print to complete by 4pm to allow time for the bed and printhead to cool down and take the print

* Next choose **Export G-code**, and save the exported **.gcode** to the SD card ready to print! You’ll find that it prepends your file with the printers name, MK3
* Remove the SD card from the computer once saved.

# Printing Checklist

* With the printer off, clean the bed/ steel heated plate with an appropriate cleaner, for PLA use a piece of kitchen towel and cleaning alcohol. Carefully spray alcohol onto the cloth away from the printer and your face! For other materials please ask!
* Turn printer on at the wall
* Wait for bed to warm.
* Insert the SD card, the metal pins facing you and into the machine, into the slot on the left.

Have you printed this before? If you have and you know what to expect, then great, but you still need to keep an eye on progress and check that the print is not failing or you could damage the printer or cause it to dangerously overheat. You need to think about whether your design has any overhangs or anything that might require support.

Many of the models from Thingiverse are more than likely ‘ready’ but the rule is:

**IF YOU START A PRINT IT IS YOUR RESPONSIBILITY TO LOOK AFTER IT!**

**Questions**

Is there enough filament? Check what Prusa setting say after you’ve sliced your model

Does the material loaded in the same as your settings?

Have you cleaned the steel plate?

* Press the orange rotary knob to access the menu system
* Dial/Turn the orange knob to “**Print from SD**” and select by pressing the middle of the knob
* Dial down to your file ending in **.gcod**e and select it
* Now the printer will level the bed and start printing. Watch is start for first 10minutes or so and stay nearby to check the print is secure to the bed and printing as expected
* Keep checking!
* Done!

**Removing a print**

* With the printer off, remove the heated steel sheet Plate

The plate is magnetic so place your fingers either side of the front of the plate and tilt the front of the panel up then pull toward you gently. Gently bend and twist the plate to releas your print or use a spatula.

**Changing material**