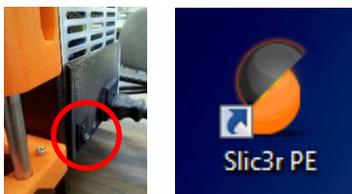


HOW TO USE THE PRUSA 3D PRINTER

Note: If you have any issues with this printer (filament loading/unloading, printing errors, etc.) do NOT attempt to fix the problem yourself. Please ask lab staff for assistance to avoid printer damage.

Prepare the printing software...

1. Turn on the printer.
2. Start Prusa > Slic3r program on the desktop.



Choose filament loading method...

1. If your object is **single color**, choose "Prusa i3 MK3 (Manual load, Single color object)" from the Printer list on the right.

Printer: Prusa i3 MK3 (Manual load, Single color object) ▼

2. If your object is **multi-color**, choose "Prusa i3 MK3 (Auto load, Multi-color object)" instead.

Load object in program...

1. Click the [Add] button under the [Plater] tab in the top-left corner, then locate the STL file that you want to print.



- a. If your object is **single color**, click the STL file, then click [Open].
- b. If your object is **multi-color** and has multiple STL files, hold-down [Shift] on the keyboard, click each file, release [Shift], then click [Open].

2. To rotate or resize object, right-click object, then click Rotate or Scale from the menu that appears.
3. Specify a print quality mode (MK3 Fast, Detail, Optimal, or Ultra Detail) from the Print Settings list on the right. **MK3 Fast is recommended; other modes print slowly.**

Print settings: 0.20mm FAST MK3 ▼

4. If supports are needed, choose "Everywhere" from the Support list on the right.

Preparing filament for a single-color object...

1. The active filament color is shown as F1-F5 on the orange panel on the printer. This corresponds with the numbers on the spool trays. Check the related spool tray to see if it is the color you want. **If none of the spool trays have the color that you want, ask a lab staff person to help you.**



2. Push the knob on the panel to display the menu.

3. Turn the knob to highlight "Preheat", then push the knob to select. On the next screen, choose "ABS".



4. Wait for the nozzle to reach 255 degrees.



5. If you like the current filament choice, push the orange button again, then choose "Load to nozzle" from the menu. Then choose "Load filament X..." where "x" is the active spool tray from Step 1.

6. To choose a **different** color spool, push the orange knob, then turn it to highlight "Unload filament". After the current filament unloads, push the orange button again, then choose "Load to nozzle" from the menu. Then choose "Load filament X..." where "x" is the spool tray that you wish to use.

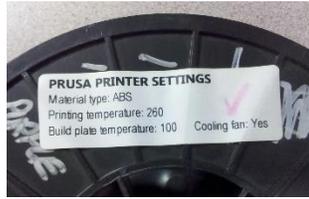
Preparing filament for a multi-color object...

1. You must prepare the printer by unloading the current filament from the extruder.
2. Push the knob on the panel to display the menu.
3. Turn the knob to highlight "Preheat", then push the knob to select. On the next screen, choose "ABS".
4. Wait for the nozzle to reach 255 degrees.
5. Push the orange knob, then turn it to highlight "Unload filament".



Change filament settings...

1. On the filament spool's label, note the Material Type, Printing Temp, and Build Plate Temp.



2. In Slic3r, choose the correct material type from the Filament drop-down list(s) on the right. (usually "Generic ABS" or "Makerbot ABS"). If you are printing a multi-color object, choose the **same** filament type for all five Filaments.

Filament: 

3. On the [Filament Settings] tab at the top, choose the filament type from Step 2 from the down-down list below the tabs. Enter the spool's Printing Temperature in the Extruder: "First layer" and "Other layers" fields and Build Plate Temperature in both "Bed" fields.

Temperature °C			
Extruder:	First layer: <input type="text" value="255"/>	Other layers: <input type="text" value="255"/>	
Bed:	First layer: <input type="text" value="100"/>	Other layers: <input type="text" value="110"/>	

4. Enable Cooling > "Keep fan always on" on the left if the spool label states it.

Preparing your object for multi-color printing...

1. You can assign filament colors (1-5) to different parts of your object(s). Right-click a part, then choose Settings from the menu that appears. Choose 1-5 in the Extruder field for the filament spool color that you want to use for this part.

Object Settings

Extruders

Extruder:

Wipe into this object's infill:

Wipe into this object:

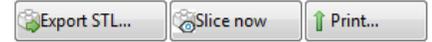
2. Enable Settings > "Wipe into this object's infill".
3. Click [OK], then repeat Steps 1 & 2 for assigning different spools to the other parts/objects.

Note: The onscreen colors will not match the actual filament colors on spools 1-5 on the desk. This is OK.

Start printing...

1. To ensure that your object sticks to the build plate, clean the plate with rubbing alcohol and a rag. **Do not use glue stick with this printer.**

2. Click the [Plater] tab at the top, then [Slice now] in the lower-right.



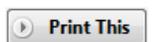
3. After slicing has finished (there is a progress bar in the lower-right corner), click [Print...] also in the lower-right.



4. Click the [Controller] tab at the top.
5. Click [Connect to printer], then wait a few moments for the Slic3r to contact the printer.



6. In the Queue section in the middle of the screen, click the [Print This] button on the right.



7. The printer begins heating the build plate and nozzle. When they are fully heated, printing will begin.

After printing...

8. **Do not use a scraper nor knife to remove your part!** These will damage the build plate.



9. Wait until bed cools to room temperature. **This is important!**
10. The build plate is magnetic and removable. Lift its edges to remove.
11. **Gently** flex the plate and your part will pop-off.