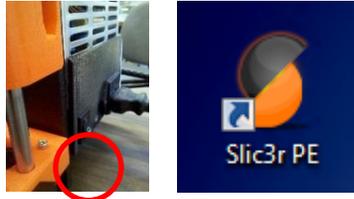# HOW TO USE THE PRUSA 3D PRINTER

Prepare the printing software…

1. Turn on the printer.

2. Start Prusa > Slic3r program on the desktop.

Choose filament loading method...

1. 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) ▼

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.

File Plater Object Window View Configuration Help
Plater Controller Print Settings Filament Settings Printer Settings

2. Click the STL file, then click [Open].

3. To rotate or resize object, right-click object, then click Rotate or Scale from the menu that appears.

4. 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 ▼

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

Change filament (if needed)…

1. On the orange panel on the printer, push the knob to display the menu.

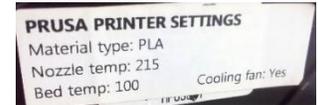2. Turn the knob to highlight "Preheat ABS", then push the knob to select.

3. Wait for the nozzle to reach 255 degrees.

4. Push the orange knob, then turn it to highlight "Unload filament".

5. When the printer beeps, pull-out the filament quickly from the top of the extruder.

To load different filament…

1. Locate a spool with a "Prusa" label on it. **Do not use a spool that does not have this label!**

2. Place the spool on the top rack.

3. On the orange panel on the printer, push the knob to display the menu.

4. Turn the knob to highlight "Preheat ABS", then push the knob to select.

5. Wait for the nozzle to reach 255 degrees.

6. Trim filament with scissors so that it has a sharp edge.

7. Insert the filament into the top of the extruder. The extruder will load it automatically, eventually coming out the nozzle below.

8. When the extruding stops, choose "Yes" on the panel to confirm that the filament has loaded.

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").

Filament: 🔒 Generic ABS ▼

Flip over →

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.
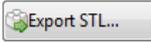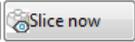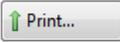
Temperature °C			
Extruder:	First layer: 255	Other layers:	255
Bed:	First layer: 100	Other layers:	110

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

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.
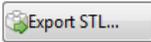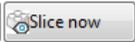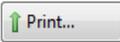
4. Click the [Controller] tab at the top.

Prusa i3 MK3
Port: COM11
Speed: 115200

Connect to printer

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.

Print This

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

After printing…

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

2. Wait until bed cools to room temperature. **This is important!**

3. The build plate is magnetic and removable. Lift its edges to remove.

4. **Gently** flex the plate and your part will pop-off.

The University of Akron
Wayne College 3D Lab