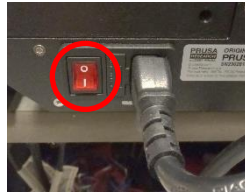# HOW TO USE THE PRUSA XL PRINTER

WARNING: Keep your hands/head away from top-front of printer! Toolheads move fast and may strike you.
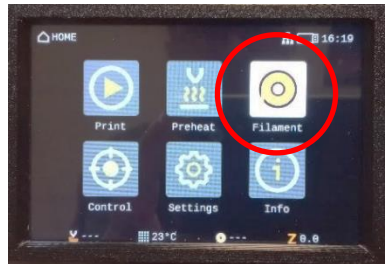
Prepare printer…

1. Turn on printer (switch behind printer, flip down).

Change filament (if you want different colors)…

1. On printer's front panel, turn wheel to highlight Filament, then push wheel.

2. Choose Change Filament, then Pick Tool 1.

3. Wait for toolhead to heat-up ("Waiting for temperature" screen).

4. At "100% - Was filament unload successful?" screen:
   a. On left side of printer, rotate filament spool clockwise until filament pulls out of white tube. **Don't let filament leave your hand; it will tangle on spool.**
   b. **Important: Trim filament end with cutters.**
   c. Choose "Yes" on panel.

5. Place desired filament spool on printer's holder.

6. Choose filament type you are using (typically PLA or ABS). Type is indicated on the spool's label.
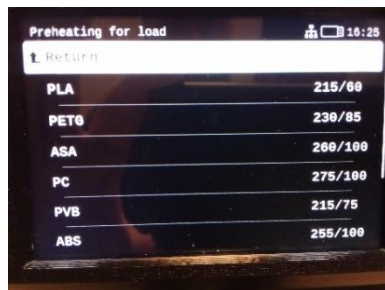
7. At "Loading filament" screen, insert filament into feeder, pushing it through long white tube, then into toolhead. Printer will beep when it's pushed far enough.
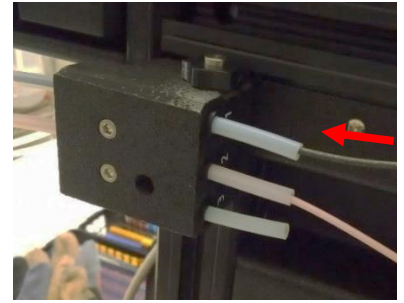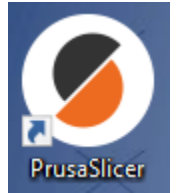
8. At "Is color correct" screen, choose "Yes".

9. Optional: repeat above steps to change filament for Toolhead 2.

Load object in program…

1. Start PrusaSlicer program on desktop (in 3D Lab/Prusa folder).

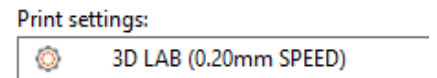2. Click the Add button in center toolbar, then locate STL file that you want to print.

3. Click your STL file, then click [Open].
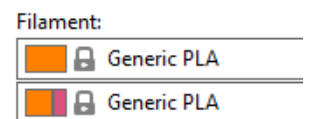
Adjust printing settings…

1. To rotate or resize object, click object, then click Rotate or Scale buttons on left side of screen.

2. Choose "3D LAB (0.20mm SPEED)" in Print Settings field on right side of window.

3. In Filament field, choose filament types loaded into toolheads 1 and 2 (typically Generic PLA, Generic PLA Silk, and Generic ABS).
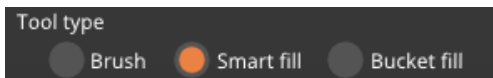
Multi-color prints (optional)…

If different colors are loaded in each toolhead, you can "paint" your object with multiple colors.

1. Click your object.

2. Click "Multimaterial painting" icon on left side of screen.

3. Enable "Smart fill" in window that appears.

4. Left-click areas of your object to paint with first toolhead color, right-click to paint with second toolhead color.
   a. Note: Onscreen colors may not match actual colors loaded in toolheads, this is normal.
   b. Note: Leaving "Multimaterial painting" mode displays your object as single color, but it will show multi-color in Preview mode.
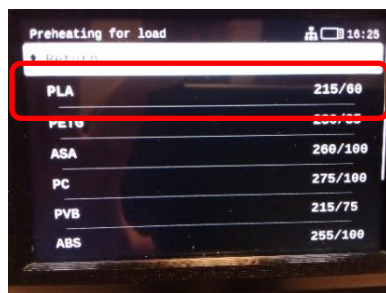
Dissolvable supports (optional)…

This printer can use filament that dissolves in water, useful if your object requires supports.

1. From printer's control panel…
   a. Load PLA filament in Toolhead 1.
   b. Load PrimaSelect PVA+ filament in Toolhead
2. Tell printer you are loading "PLA" filament.
3. From PrusaSlicer program…
   a. Choose PrimaSelect PVA+ for second toolhead's Filament field.
   b. Choose "3D LAB (0.20mm SOLUBLE INTERFACE)" in Print Settings field on right side of window.
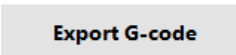4. When printing finished, manually remove supports then soak object in water for hours to remove remaining.
5. **Important! When finished printing, unload dissolvable filament and store in airtight bag. This filament quickly absorbs air moisture will ruin itself.**

Save "gcode" output file…

1. Click "Export G-code" button in lower-right corner.

2. Save gcode file to computer or flash drive.

Start printing…

1. To ensure that your object sticks to build plate, clean plate with rubbing alcohol and rag. **Do not use glue stick nor water with this printer. Only clean plate when it's cool.**

2. If you saved to flash drive:
   a. Insert flash drive on right edge of printer control panel.
   b. Your object should appear. Push knob to confirm.
   c. Compare filament types from file matches types loaded in toolheads.
   d. Choose "Print" to begin printing.
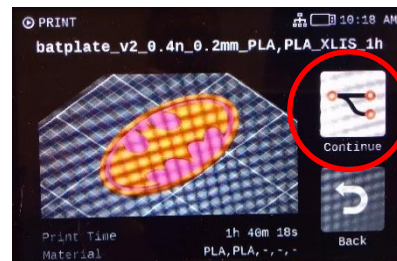
3. If you saved to computer…
   a. Run "Pronterface" program on computer's desktop (in 3D Lab > Prusa folder).
   b. Click the [Connect] button in the toolbar at top.
   c. Click the [Load file] button at top, then locate g-code file that you saved.
   d. Click the [Print] button to start the printing process.
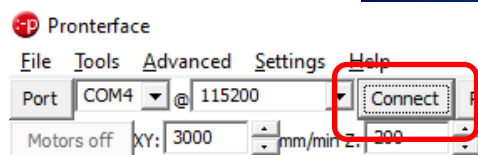
4. The printer begins heating the build plate and nozzle. When fully heated, printing will begin.

After printing…

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

2. Wait until bed cools to room temperature. **This is important, otherwise your part will warp!**

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

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