

## Digital Media Fab Lab – How To: Basic File Preparation in PrusaSlicer & Prusa i3 Machine Set-Up

1. Open PrusaSlicer
2. Import an .STL file (File < Import < Import STL/OBJ/AMF/3MF)
3. Using the ADVANCED tab, specify:
  - *Print Settings*. Choose print resolution. The lower the mm number = the thinner the layer height = the finer the resolution = the longer the print time
  - *Filament*. Select Prusa PLA or specify if printing with a filament blend
  - *Printer*. Select Original Prusa i3 MK2S
  - *Supports*. 3D models with over-hangs of 45° or more from the build plate will likely require supports for a successful print. If the design is tall and thin and/or has very little flat surface area at the bottom of the design then a 'Brim' will help keep the print steady as it progresses. Check the box next to 'Brim' to activate this option.
  - *Infill*. For test prints/drafts choose 5% or 10%. For functional parts/toys/small connections choose 20% to 45%. Rarely is it necessary to go beyond these infills when prototyping.
  - The design can be positioned, rotated, and scaled using the left-side tool panel in the software. Keep the largest and flattest side of your design oriented flat on the plate for the most success. Certain circumstances may require a different orientation. In that instance, custom supports may be necessary to achieve the desirable outcome.
4. Click the 'Slice Now' button to generate the tool path for printing. On the right-side panel, view the estimated cost, grams of filament, and print time. Use the vertical orange slider bar to view how the object will be constructed. Check to ensure that infill is within each component, if needed.
5. Click the 'Export G-code' button and save the file to the 3D printer's SD card.
6. Eject the SD card and place it in the Prusa machine.
7. Turn on the 3D printer. Flip the toggle switch on the right side of the machine to |.
8. Make sure the bed of the machine is free of other prints, filament, dirt, dust, and oils. If necessary clean the print bed with a microfiber towel damp with water.
9. Load filament into the machine. Using the dial and LCD screen, Preheat the machine for PLA. If necessary unload filament, and then load a different filament. Questions about this step in the process can be directed to the lab monitor.
10. Optional – Using a glue stick, apply a thin layer of glue to the print bed where the print will be positioned. The glue will help the bottom of the print remain secure on the print bed during printing.
11. Press the dial located near the LCD screen and turn it to highlight 'Print from SD'. Press the dial to select.
12. Find the file name and press the button to select it. The machine will home, preheat, and begin the print.
13. Observe the first 3 to 10 layers of the print to make sure it is succeeding. Stay during the remaining print time or leave and return to pick it up later.