Assembled MendelMax 2.0 Electronics Guide

Edition 0.1

http://www.makerstoolworks.com/

support@makerstoolworks.com
Freenode IRC: #makerstoolworks

Thank you for purchasing the MendelMax 2.0 kit. This is a living document and we encourage your comments to improve the experience for everyone.

Commenting is enabled in the insert menu.

1st highlight the area of comment, then click *Insert* -> *Comment*

Electronics used are both 120/240 VAC and 24 VDC. Additionally, this guide is a WORK IN PROGRESS, making it even more dangerous. This makes for both a fire hazard and a shock hazard. As this is sold as a kit, it is your responsibility to ensure it is safe. The safety of this machine depends on careful assembly and operation. We are not liable for any injuries or damages that result from following this guide. By continuing with this build, you hereby release and hold harmless Maker's Tool Works from any and all liability.

This document contains the following sections:

- Getting Started
- Installing the drivers and software
- Your First Slice and Prints: Fan and endstop mounts
- Connecting to the printer in Pronterface
- Uploading firmware

Getting Started

Now that reassembly is complete it is time to complete the software side of the setup.

Installing the Drivers and Software

Downloading the necessary files

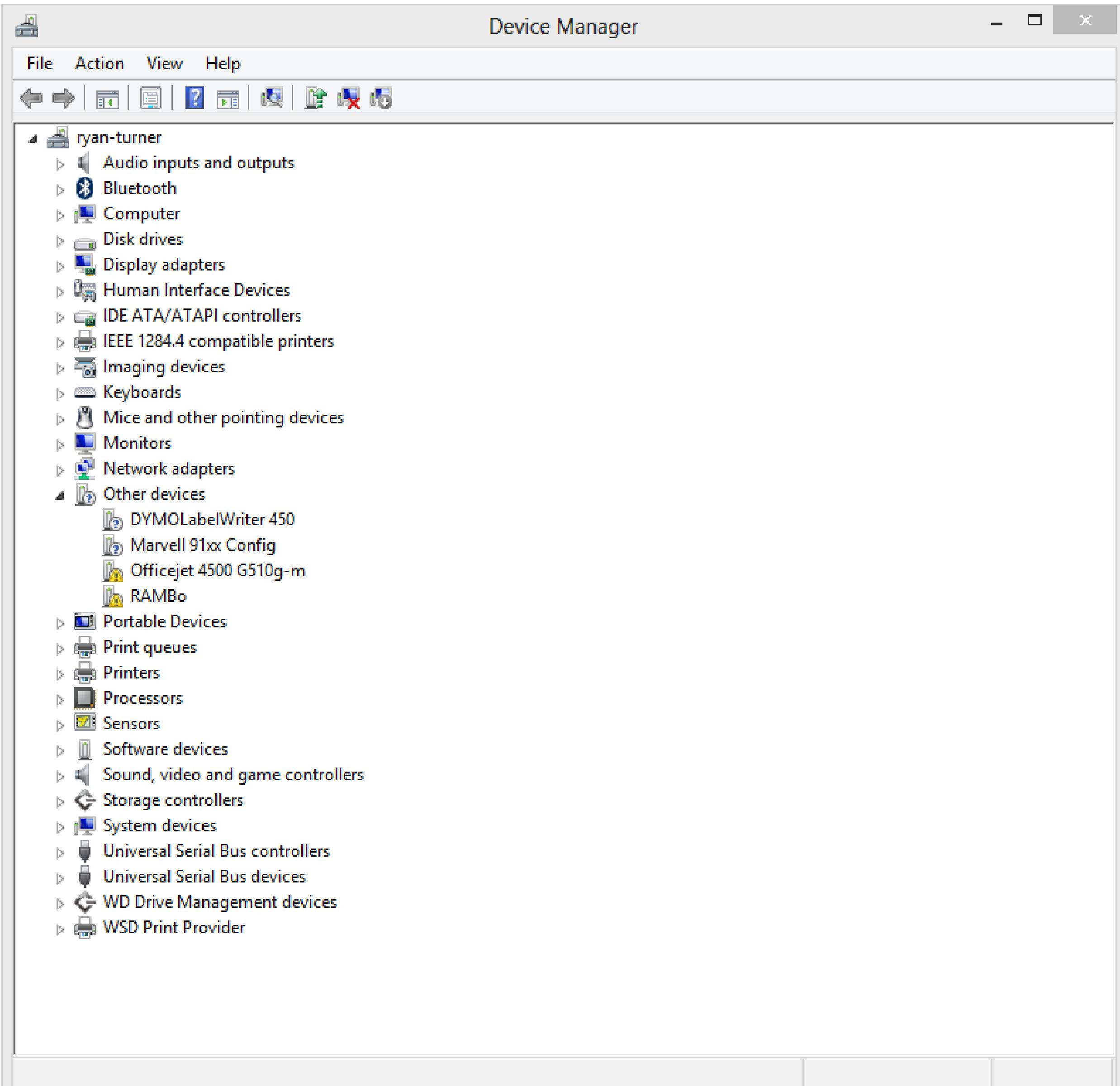
1. Access the Download Center and download the platform-appropriate versions of the following files:
    - Software: Arduino, Printrun, and Slic3r
    - Drivers (only necessary for Windows at this time), RAMBo.
    - Firmware, we have already installed the appropriate version for RAMBo.
        - Make sure to download the “Endstops” version of the firmware.
2. Unzip all files.
3. Get acclimated with all of the different software:
    - Arduino is an application used to modify, compile, and upload the firmware to the RAMBo print controller.
        - Using the GLCD on RAMBo requires a special arduino build
    - Marlin is the firmware that is running on the print controller, interpreting instructions and powering the different parts of the printer.
    - Printrun, specifically the Pronterface application within it, is the application for communicating with the printer. It takes manual instructions as well as the toolpaths that Slic3r makes and sends them to the printer.
    - Slic3r is for taking 3D models and converting them into toolpaths (gcode) for the printer to understand.

Getting Arduino IDE configured to upload firmware

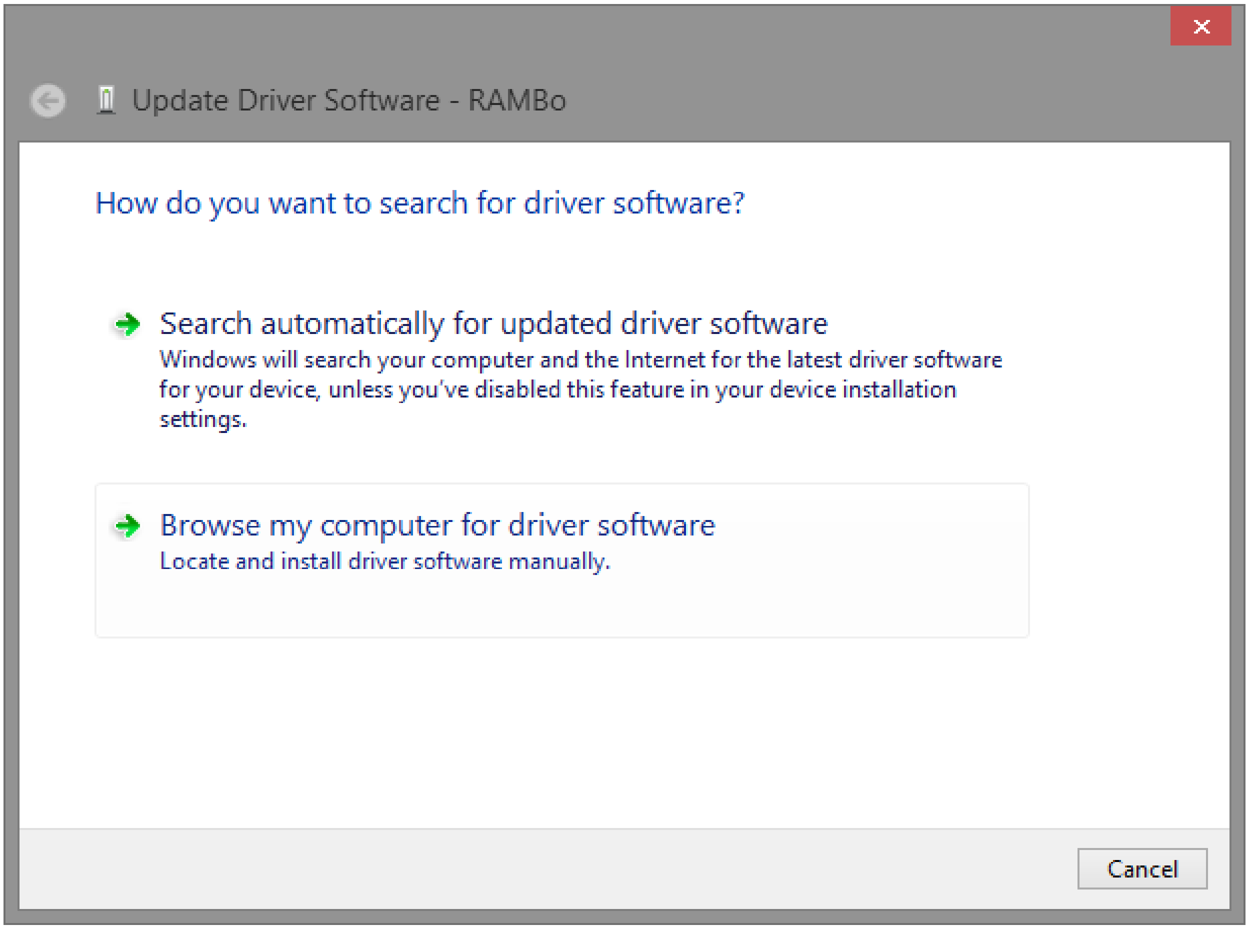
1. If using Windows, it is necessary to first install the RAMBo driver that you previously downloaded.

Installing RAMBo Driver

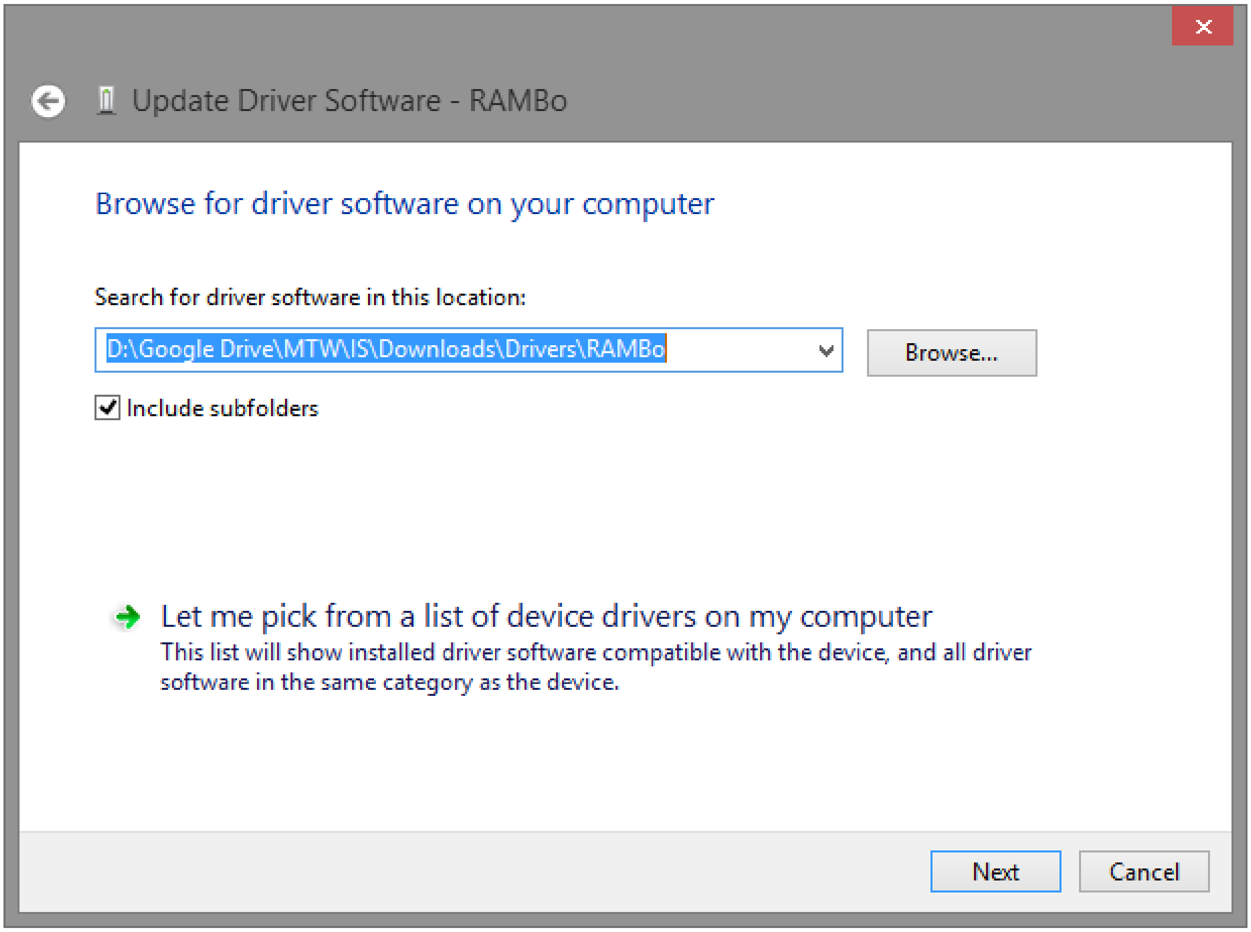
1. Attach the RAMBo to the computer via the USB cable.
2. Open Device Manager
3. Under `Other devices`, look for `RAMBo`



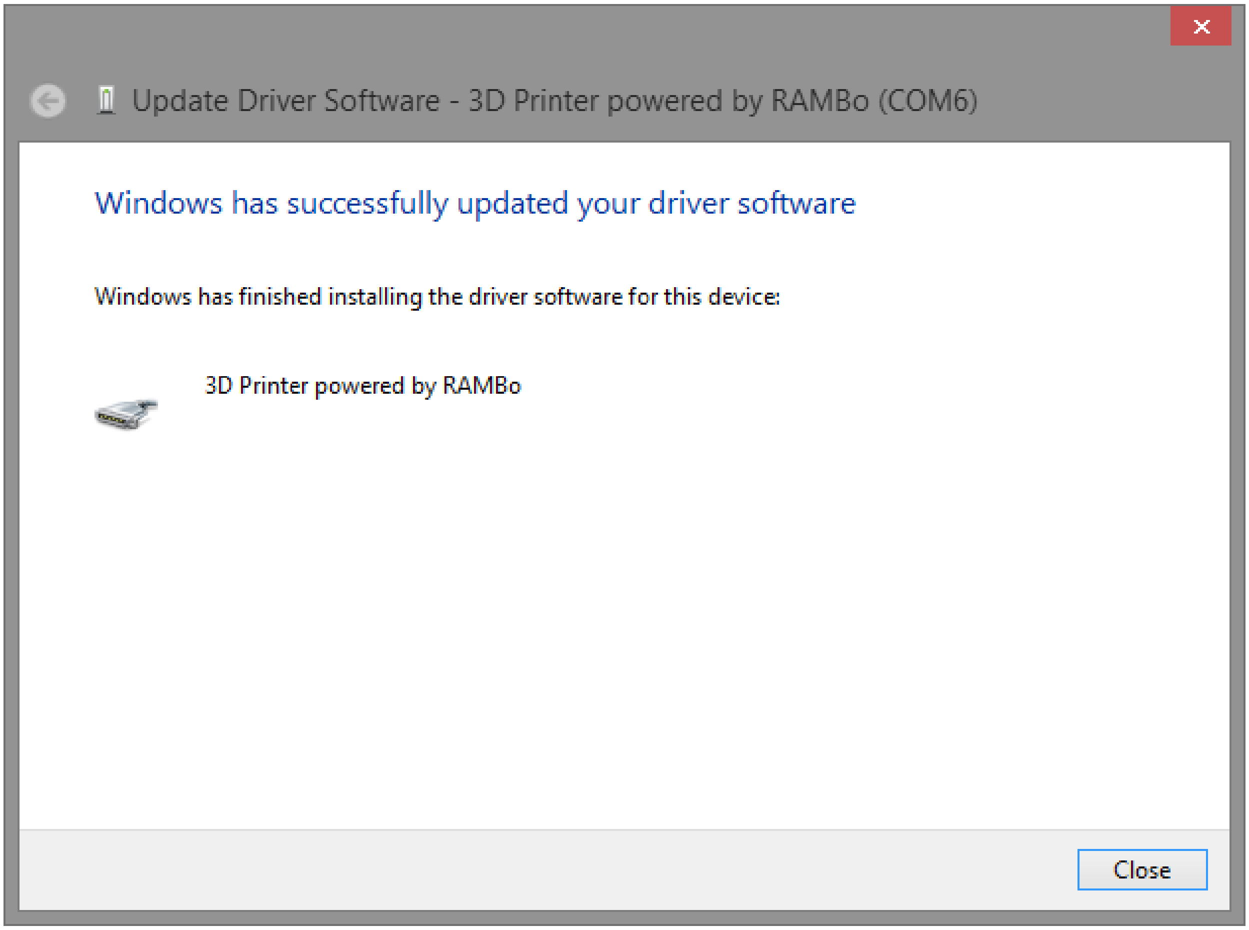
4. Right click `RAMBo` and select `Update Driver Software`



5. Click `Browse my computer for driver software`, then click the `Browse...` button and browse to the directory where you have unzipped the RAMBo drivers, click `OK`, and then click `Next`



6. Taking note of the COM port listed in the title of the window, click close



Your First Slice and Prints: MendelMax Keychain

As noted before, the workflow for using your 3D printer is converting a model to machine code using a slicer, then controlling the actual print using conttol software like Printrun aka Pronterface.

For a more in-depth guide on Slic3r please check out the new [Slic3r Manual (PDF)]

Slicing a Model

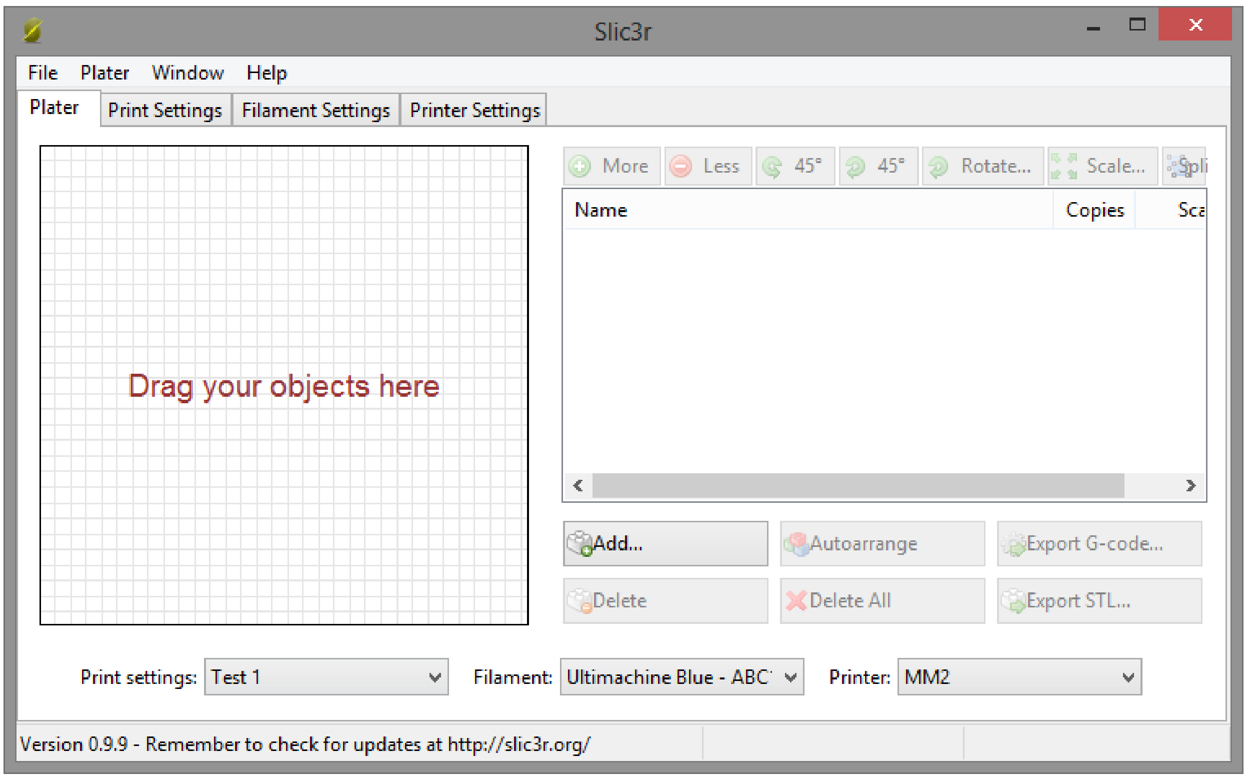
The 3D model file types STL and the newer AMF. The model file is fed into a slicer program that converts the file into Gcode used by the firmware to print the object.

Popular Slicing software:
Slic3r - http://slic3r.org/
Kisslicer - http://kisslicer.com/
Cura - http://wiki.ultimaker.com/Cura

We will use Slic3r in this initial setup. Feel free to experiment with other slicers.

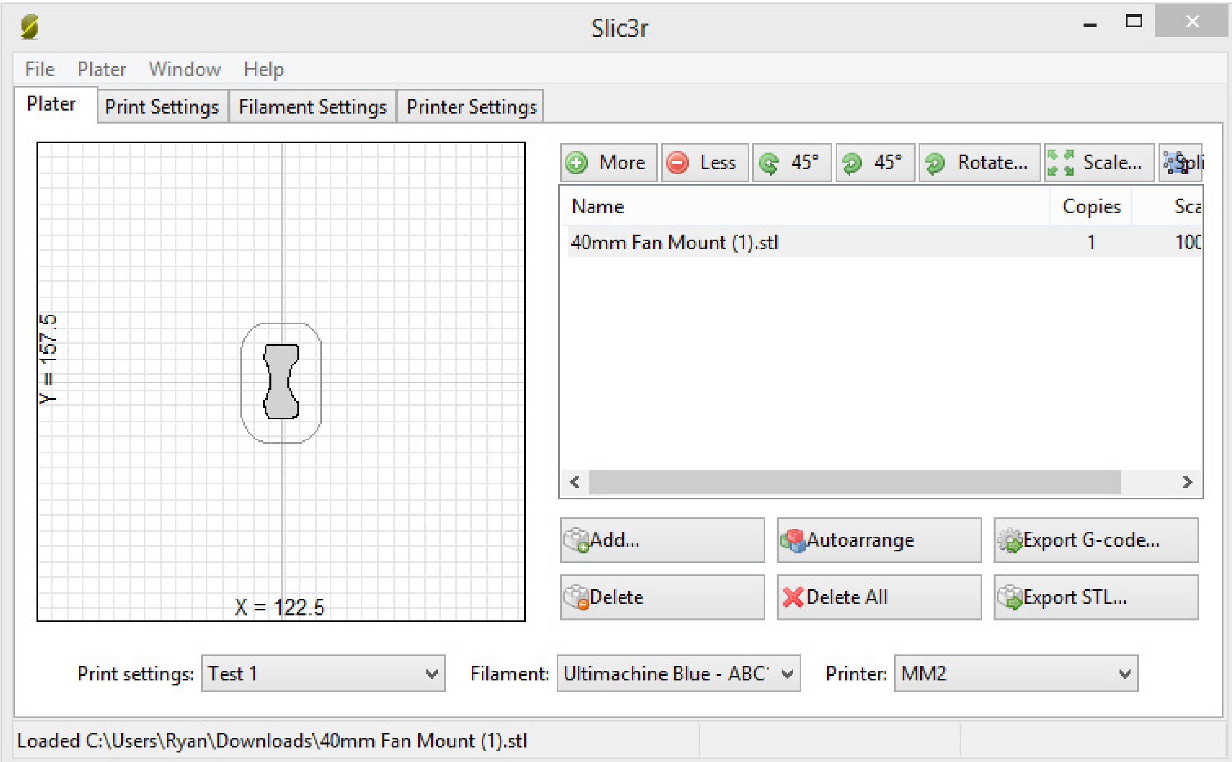
Getting the model, configuring Slic3r, and slicing the fan mount

1. First, download the models to print: they are available in our Download Center under the link “Browse STLs on Dropbox”; download the following models:
    - Mendelmax Keychain.STL
2. Open Slic3r and ignore the first-time setup wizard; if prompted, set Slic3r to “Expert Mode”.
3. Your printer was sent with a config file you can use by clicking `File -> Load Config...` Loading this allows you to skip the rest of the setup below.



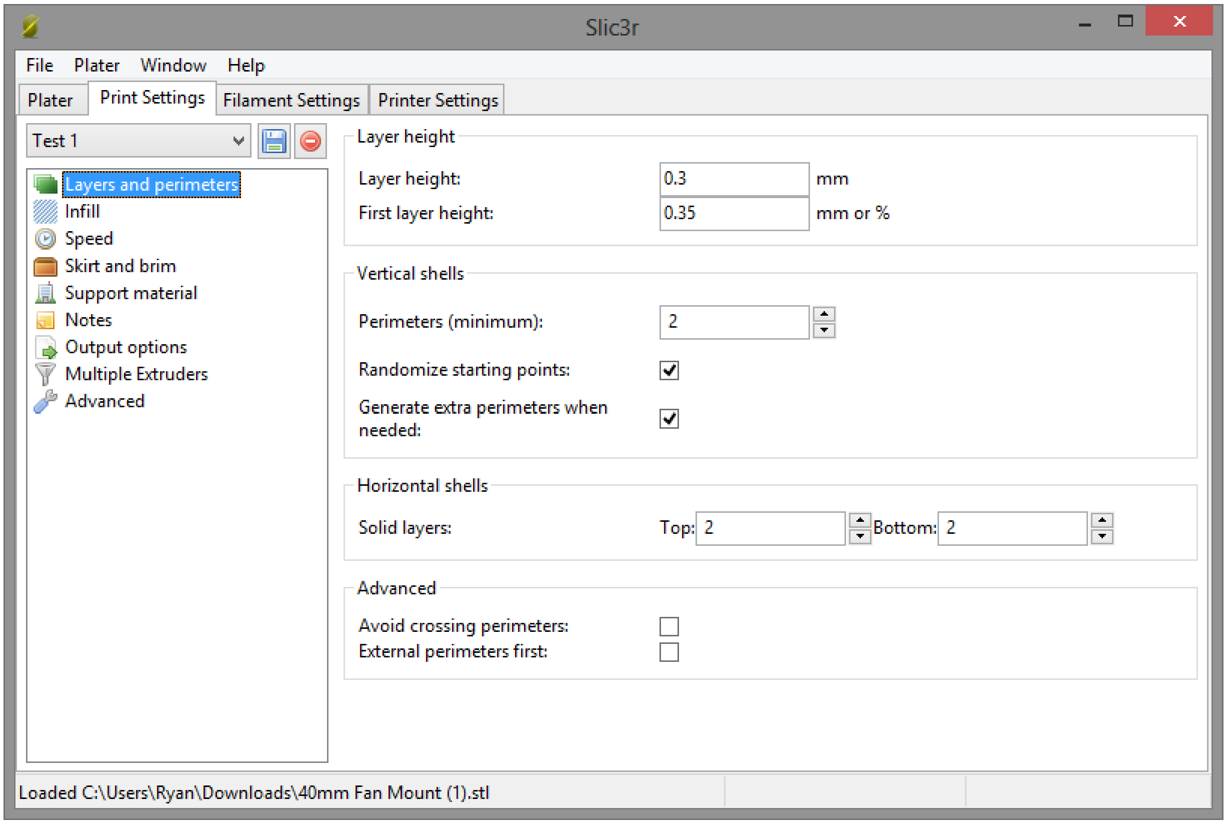
4. Click `Add...` or drag and drop the MendelMax Keychain.stl file to the Slic3r main

window, and the shape should appear in the left graph with an entry in the right column “MendelMax Keychain.stl”

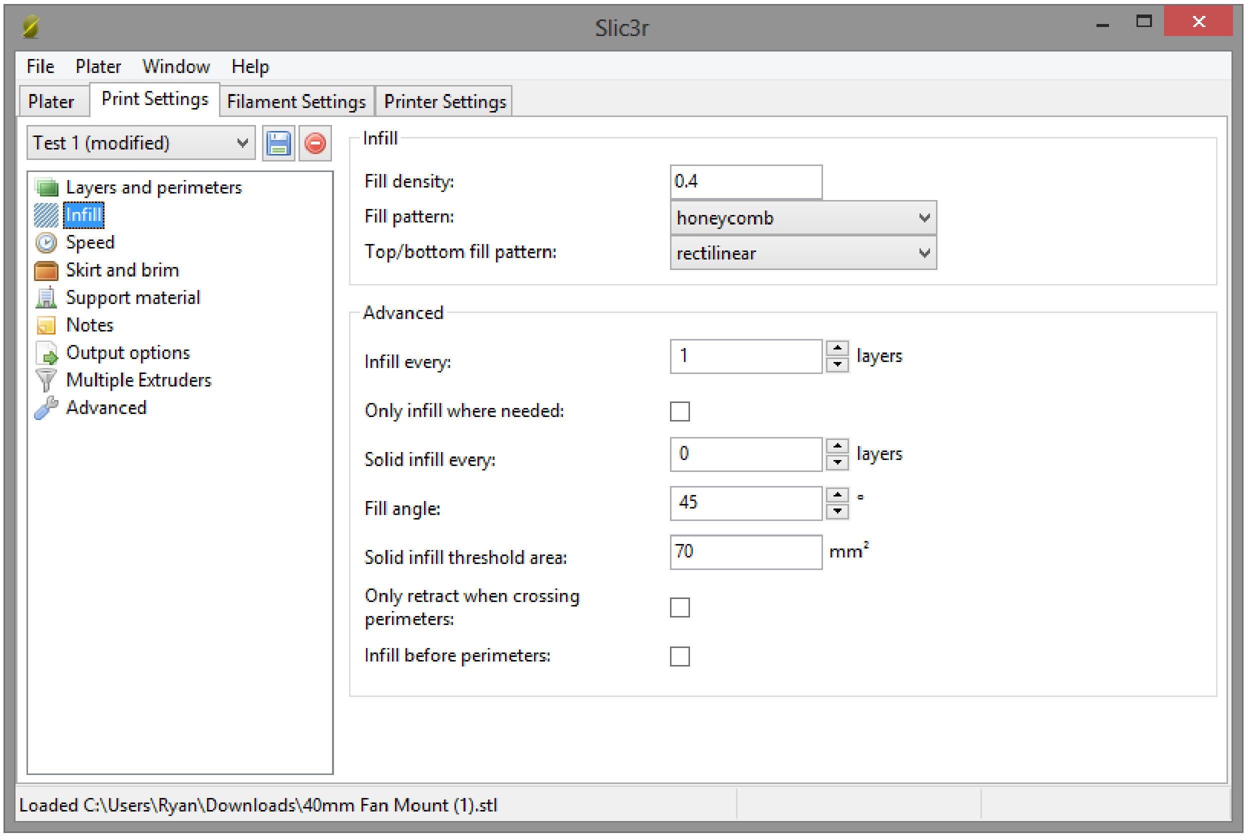


5. Adjust the settings in the following windows to match the pictures below. Make sure that before you change tabs you press the blue floppy-disk icon to save the configuration. This will serve as a starting point for your future prints. Notice that some of the settings are things that should be specific to the object you’re printing, like infill density and speeds.
    - ⚠ **Note:** You can hover over a setting to get more details about it.

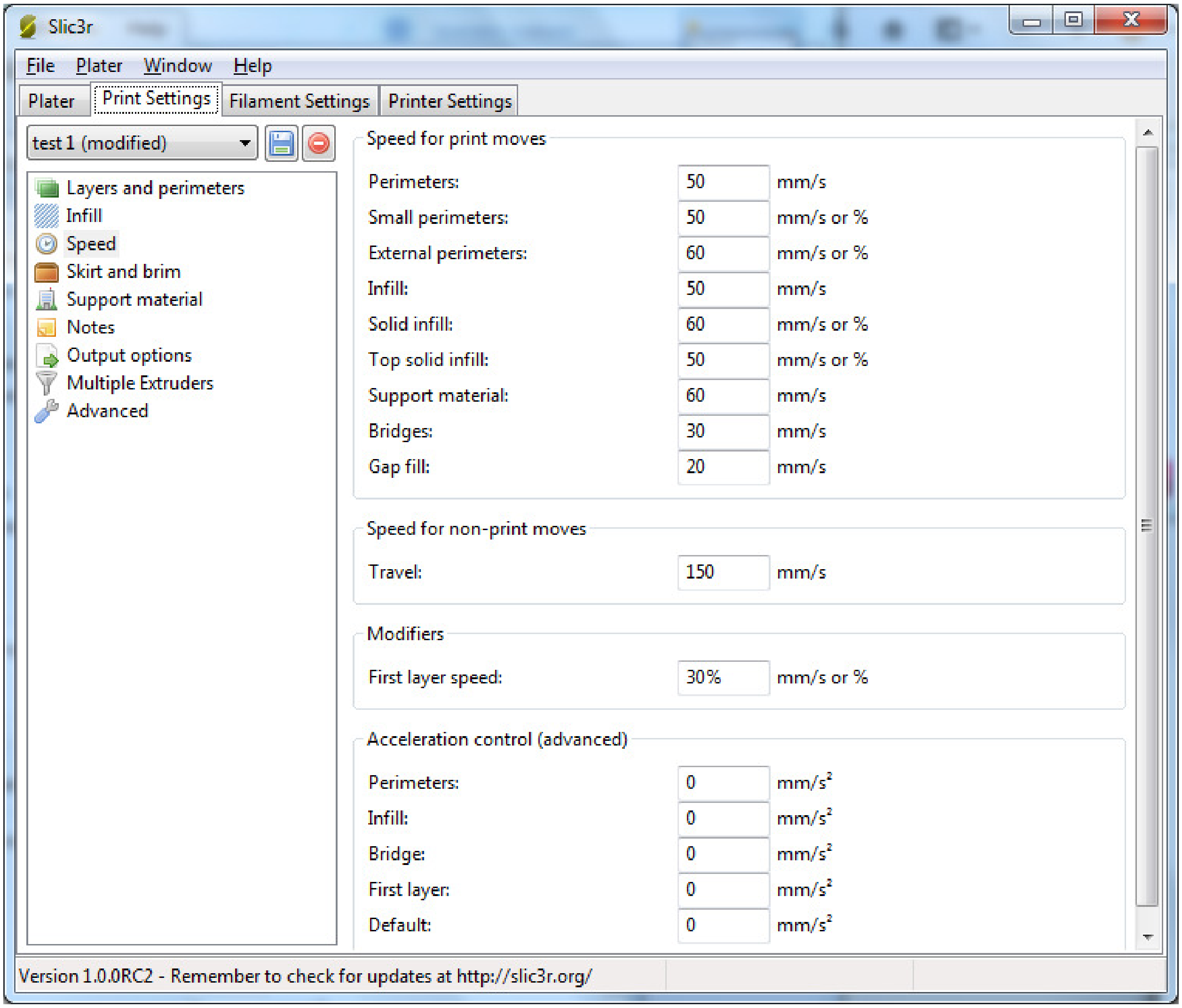
Print Settings



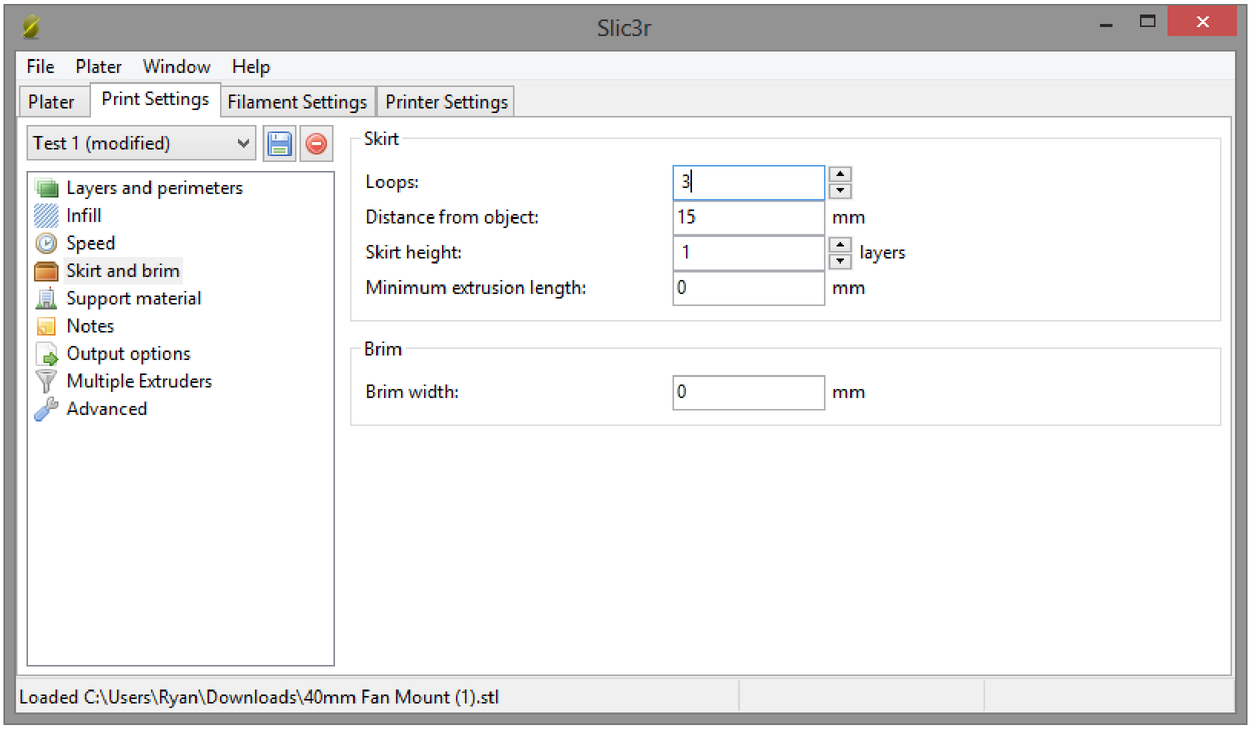
- Your layer height should always be below 80% of your nozzle diameter, and we provide a 0.4mm nozzle orifice, meaning that you want to never use more than a 0.32 mm layer height. The exception to this is in the 1st layer, however, as this helps your print stick easier. Perimeters (the X and Y oriented solid edges of an object) and solid layers (the Z solid edges of an object) are good at 2-3.



- Infill parameters define the “solidity” of the object; fill density can be between 0 (hollow) and 1 (solid). Typically 0.3 (30%) is a nice balance between strength and print time. The other parameters are beyond the scope of this documentation.

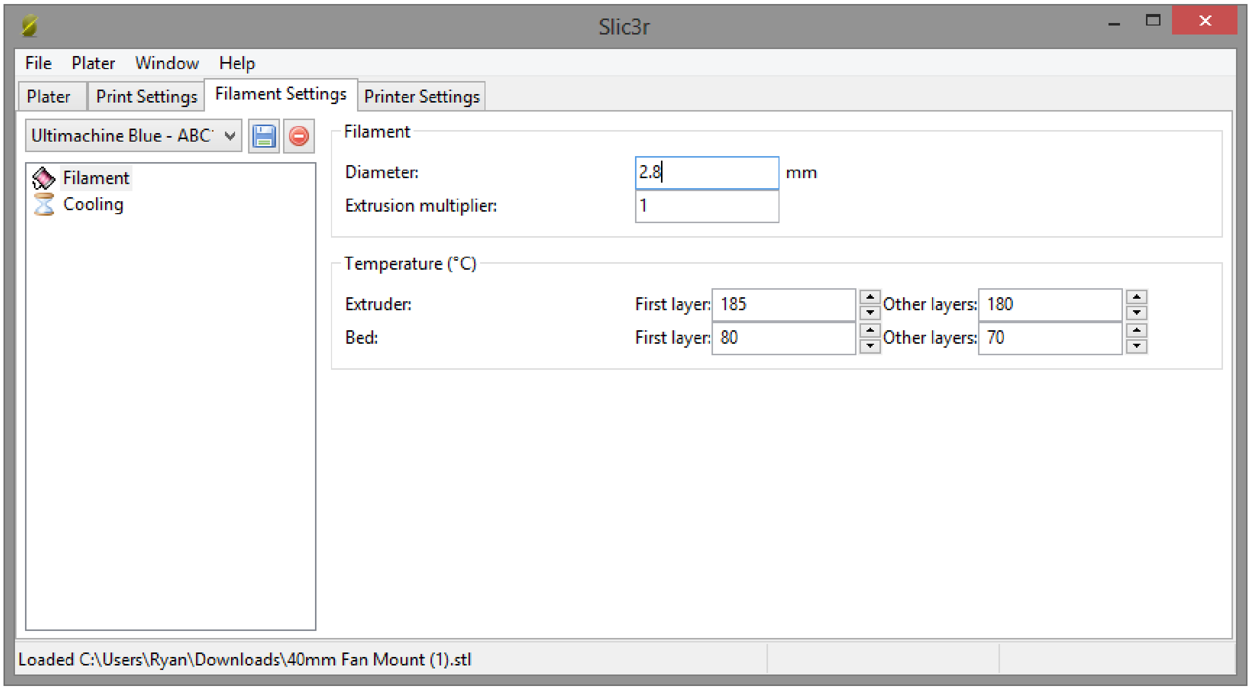


- Speed is also another parameter that inversely affects your print time and quality of the print. You can begin with default settings and adjust them faster later. A slow perimeter speed with faster infill speed is usually ideal.
- Some faster speed numbers can be seen here.

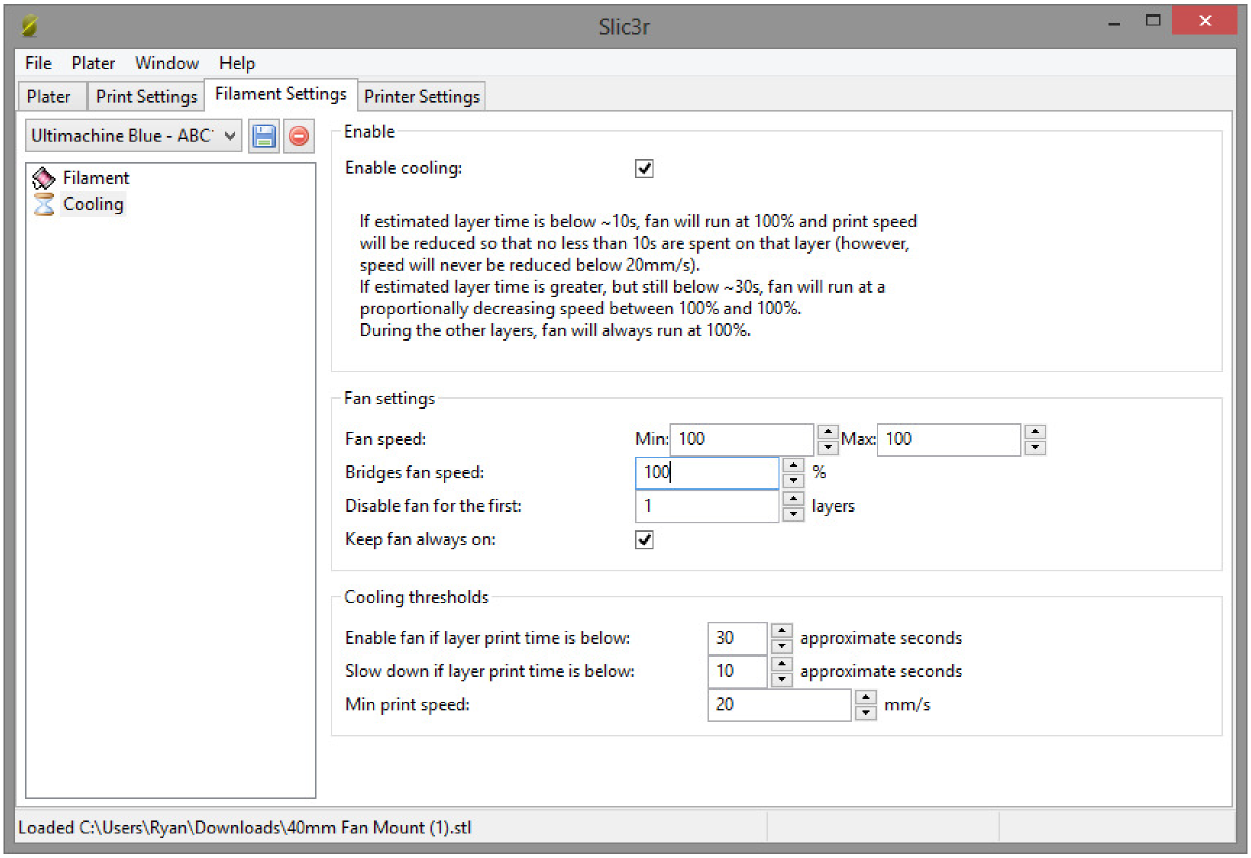


- Skirt is a loop printed around the object a certain distance away from the object; it is useful to get your hotend primed and the first layer height set perfectly before the actual object begins printing. For this first print, have it loop around the printer 3 times, 15 mm away from the object. This will give you plenty of time to adjust the first layer height.

Filament Settings

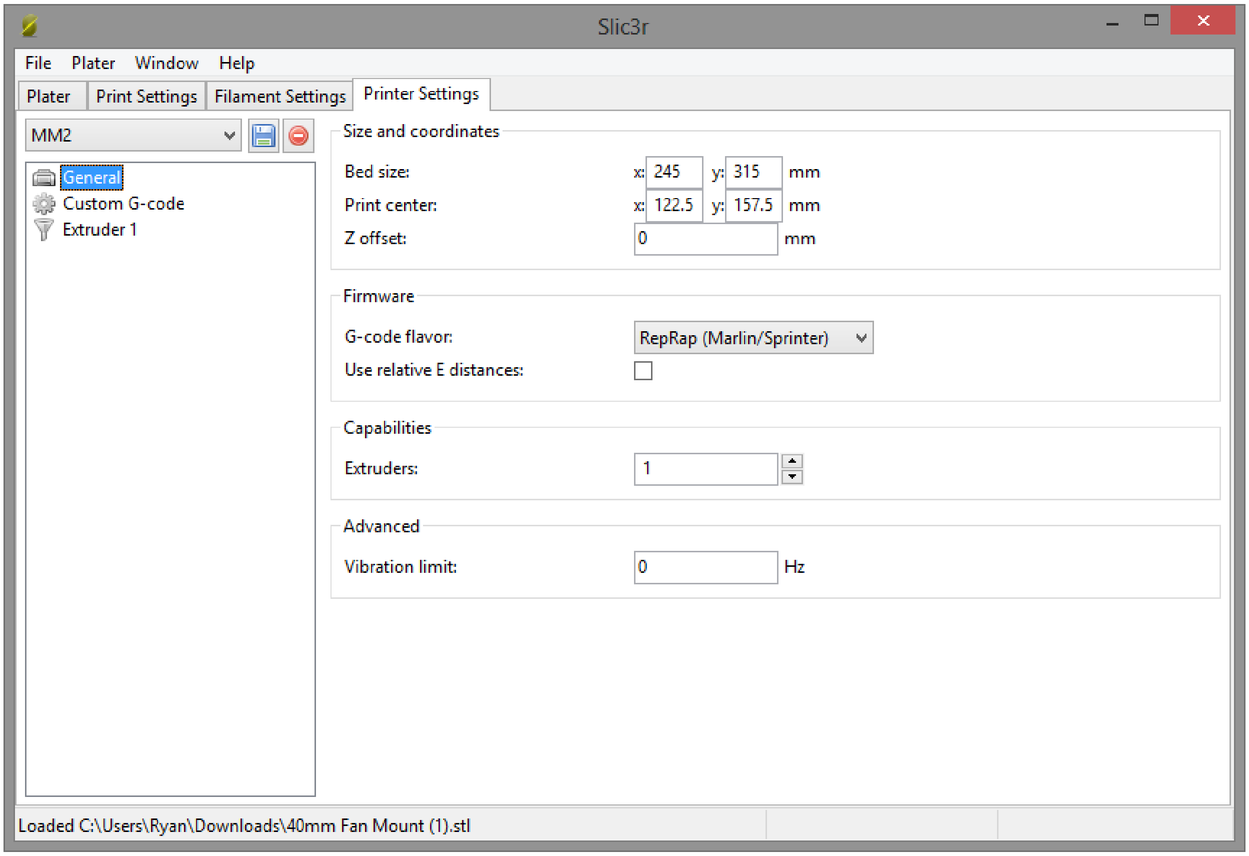


- Slic3r has to know the diameter of the filament that you are printing with; generally, 3mm filament is about 2.8 mm.
  - If you have calipers, measure the diameter on a few points of your filament and set these values to the average of that. Every time you change filament, you should measure it and put the diameter value in here, or your printer may print things with too much or too little plastic (making it hollow or overflowing).
- The temperature for your print is also set here; the temperatures shown here work well for PLA; ABS generally prints well at 215 c on the extruder and 105 c on the bed.
  - Acetone cleaning of the Kapton tape on the heater bed is essential. Also, with ABS, a higher temp, say 235C for the first few layers, then drop to 220C or so will give "good stick"

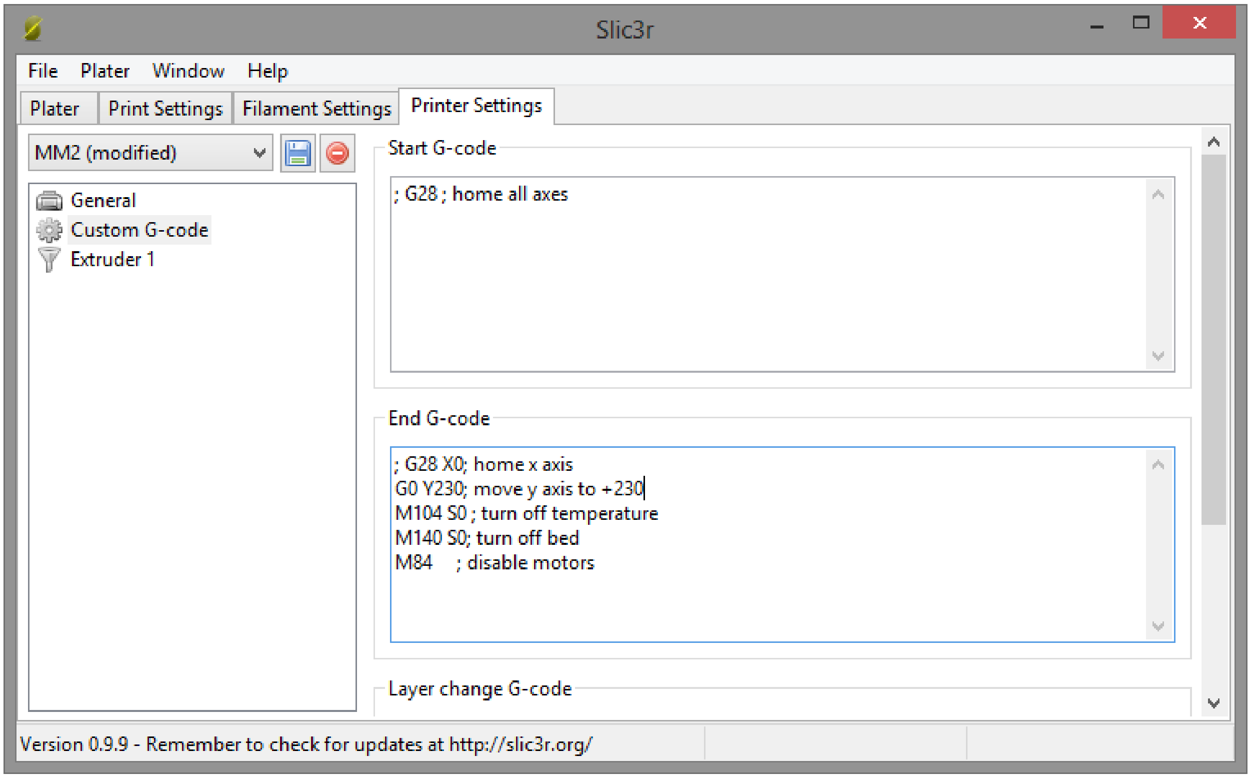


- Cooling is also specified on the filament settings tab, as different types of filament require different cooling requirements. If using PLA, setting the fan to run at 100% the entire time is fine; if using ABS, set the fan to minimum 0% and maximum 40%. Notice that the fan will be disabled for the first layer to make adhesion easier. For the rest of the cooling parameters, read the block of text above that describes the behavior.

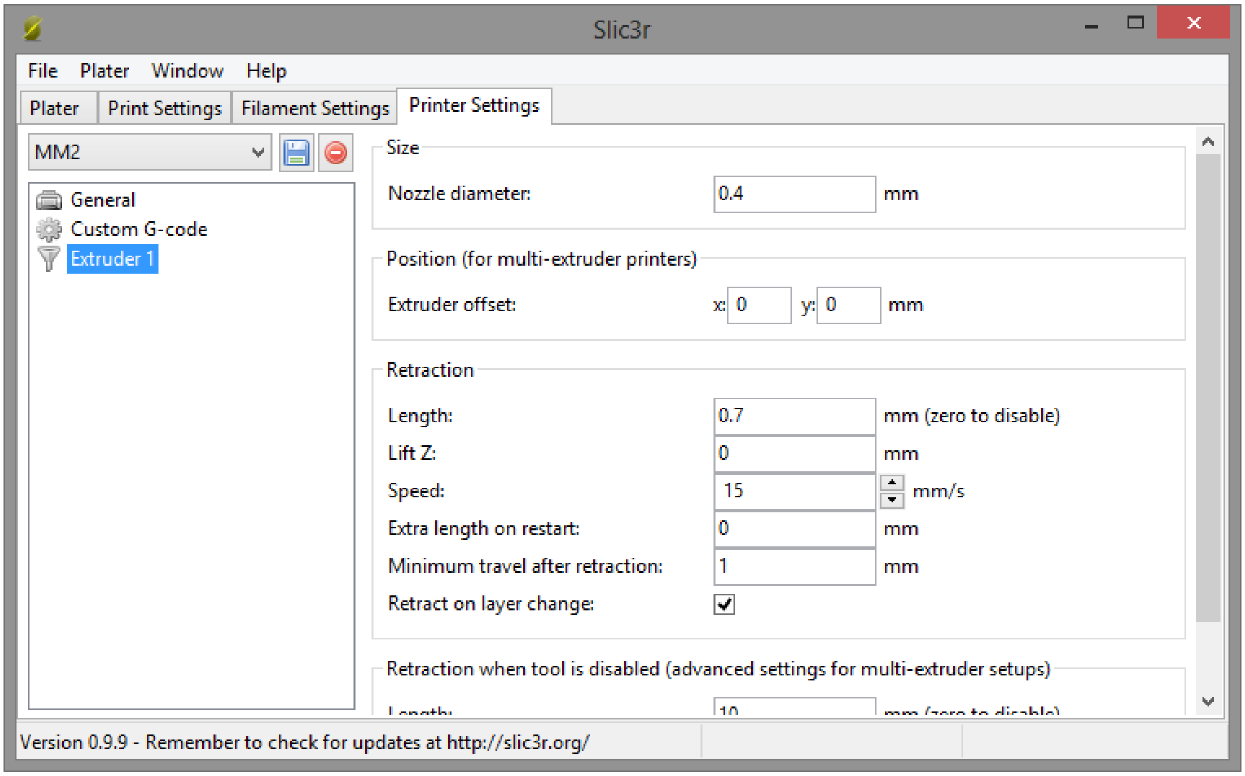
Printer Settings



- We need to tell Slic3r how large the printer area is for it to center prints; thats what this page does for us

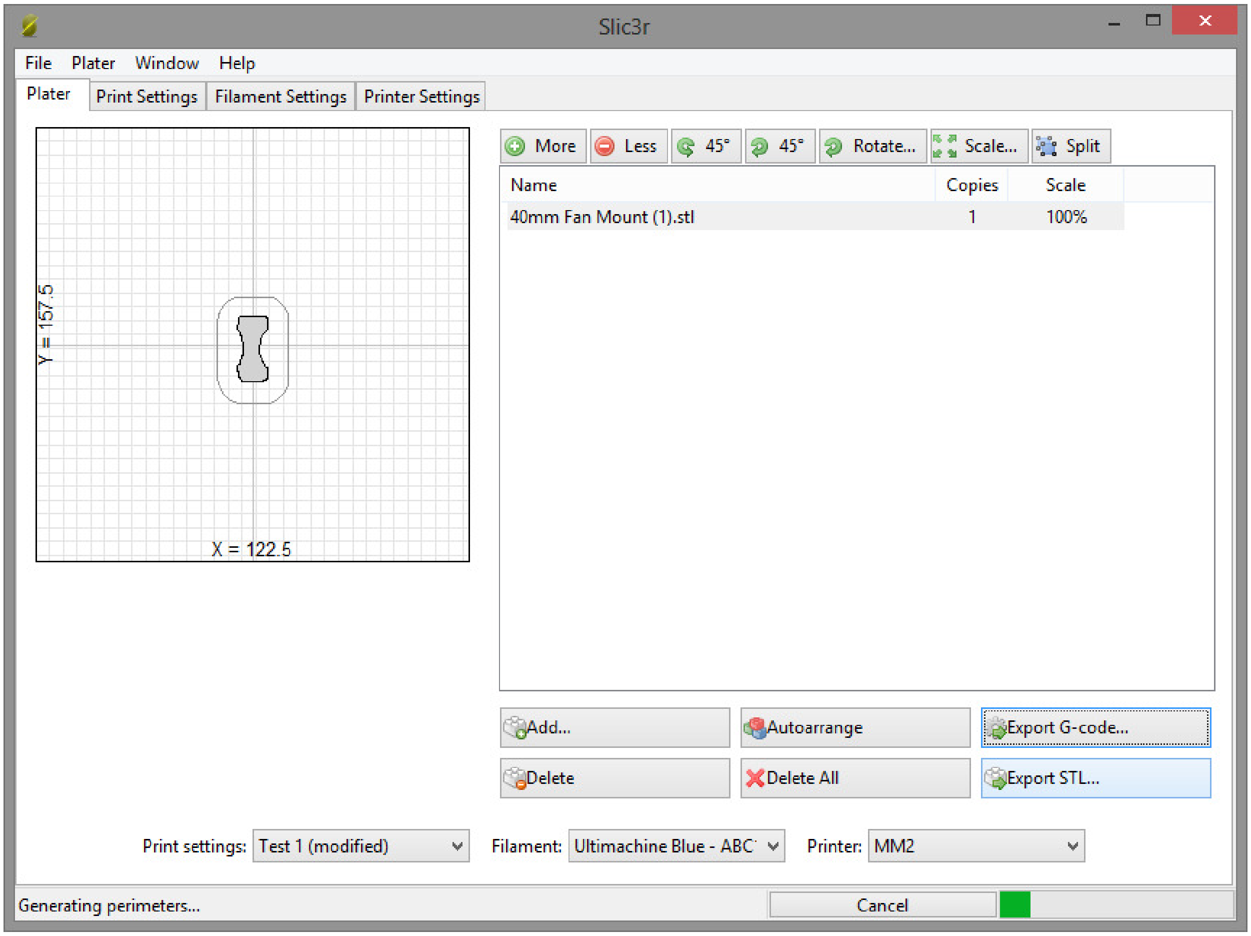


- These gcodes will be added before and after any generated toolpaths by Slic3r; this allows you to tell the printer to turn off the heater, fan, and motors. ⚠**Note:** Make sure to comment out both lines that include “G28” at the beginning, as these are the homing commands, and your printer cannot home at this point



- Set the nozzle diameter to the correct value. We ship a 0.4mm orifice.

6. With these settings saved, go back to the “Plater” tab and press the button labeled “Export G-code”; it will prompt you for where to save the created .gcode file, which is the set of instructions for how to print the models you have placed in Slic3r with the configurations set.



- In the future, you will only have to adjust these configuration changes when you wish to tweak a setting (layer height, extrusion temperature, speeds, etc); otherwise, you'll simply drag the objects onto the window and press "Export G-Code".

Controlling the Printer

There are many good host programs for operating and controlling your printer. Currently the most popular software is PrintRun http://reprap.org/wiki/Printrun

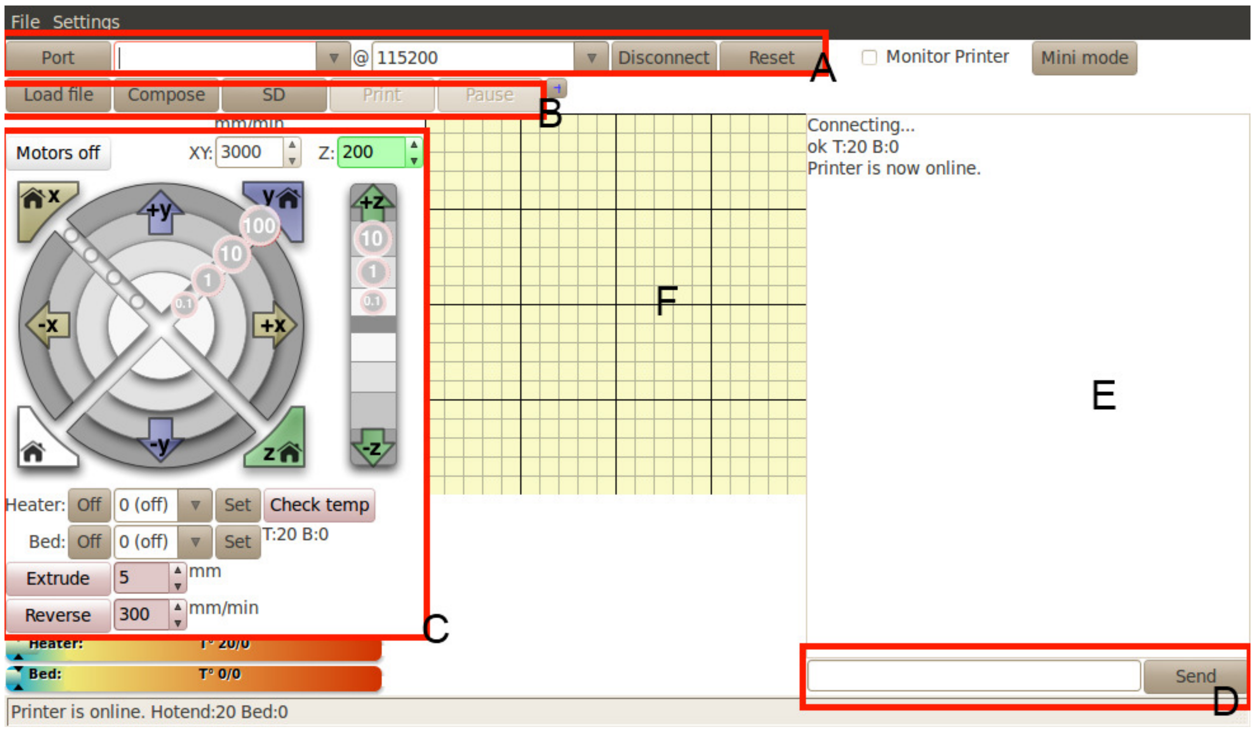
You can download a ready to run copy that doesn't require setting up python at: http://koti.kapsi.fi/~kliment/printrun/

Another newer control software is Repetier Host. Feel free to try it.

Connecting to the printer in Pronterface

The main Pronterface UI

Let's start by getting acclimated; remember that Printrun/Pronterface is the main tool that you use to control your printer.



A) Connection control, B) File control, C) Manual Control, D) Gcode Input, E) Printer Output, F) Part Preview

a. *Connection Control*, this is where to select the printers com port and baud rate speed (for ours, we use 250,000). Once these 2 settings are correct you can click `Connect` to establish control. You can also `Disconnect` and `Reset` the printer here.
   - **Note:** Most Linux systems require 115200 set here and in the firmware's configuration.h.

b. *File Control*, this allows you to Load and file to print. Once it is loaded you can begin a `Print` or `Pause`, or `Restart` it.
   Other options include `Compose` to build a platter of several parts, and `SD` to begin printing a file from an inserted SD memory card.

c. *Manual Control*, allows for manual movement and temperature control of the printer. It offers a range of different motions from .1mm to 100mm. You can also set the hotend or bed temperature. Lastly you can operate the extruder at different feedrates.

d. *Gcode Input*, allows you to manually send Gcode, the basic control language, to the printer. A list of the Gcode commands are available here: http://reprap.org/wiki/Gcode

e. *Printer Output* is where the status of the printer is displayed. Everything from connection/disconnection messages, commands you send to the printer, and error states.

f. *Part Preview* shows a simple representation of the toolpaths (movements, layer-by-layer) that are loaded to be printed. You can click on this area and get a preview that allows you to move through the layers by pressing shift and scrolling.

  - In the settings menu you can adjust the size of this preview area to match the printers build area.

Verify that everything works

1. Connect your printer to the computer using the USB cable.
2. Plug the power cord into your printer's IEC plug.
3. Open Printrun/Pronterface and connect to the printer using the *Connection Control*.
   - Remember that all of our firmware use a 250,000 baud rate, and that the COM port you need to connect to is the one you saw earlier in Device Manager.
     - Linux and Mac users will connect to a port like: `/dev/tty.usbmodem`
     - Linux users should use 115200 baud rate.
4. Slowly position by hand the X carriage in the middle of the X axis and the Y axis bed in the middle of the frame. Your Z Axis should be at about 20-40 mm above the bed.
5. In the Gcode Input enter and send the command `M119`, verify everything say `Open`
6. Using you hand Hold down the X endstop switch and send M119 again, The message should show X as `TRIGGERED`.
7. Next use you hand Hold down the Y endstop switch and send M119 again, The message should now show Y as `TRIGGERED`.
8. In Pronterface press the `Home X` and `Home Y` buttons.

9. In Pronterface, use the manual controls to move the X axis +10 mm.
10. Use the manual controls to move the Y axis +10 mm.
    - ⚠**Note:** Making the bed move in a positive direction actually moves the bed forward; this is counter intuitive. It is moving in relation to a stationary nozzle.
11. Use the manual controls to move the Z axis up 10 mm. Ensure that it is actually moving up and not down.
12. Send the Gcode command `G92 Z70` this tell the axis it is higher then it actually is so we can move below 0. This is a helpful trick if you haven't homed the axis.
13. Manually move the Z Axis down until the nozzle is just above the bed.
14. In the Gcode Input enter and send the command `M119`, verify the Z says `TRIGGERED`. The Z endstop in the back right should also have 2 lights on.
15. If the nozzle is against the bed and the Z still says `OPEN`, you need to adjust the screw holding the Z endstop magnet until the light comes on.
16. Check the "Monitor Printer" box for the temperature graph to come alive; check to see that both of your thermistors (for the hotend and heated bed) are reporting reasonable room-temperature values (typically, a comfortable room is 19-23c).
17. Set the heater to a reasonable extrusion temperature for your filament (PLA is 195c, ABS is 235c) degrees by typing in the number and pressing "Set"; watch to see the heater thermistor temperature change on the graph; be certain not to touch the hotend while it is heated.
18. Once the thermistor has reached the desired temperature, set the extrusion length to 15 mm and speed to 50 mm/min.
19. Press the Extrude button; the hob on the extruder stepper should move clockwise and the filament should be pulled down into the hotend.
20. Continue to press Extrude until plastic starts to spit out of the nozzle; you are currently priming it.
21. Avoiding touching the hotend, clear the extruded plastic off of the printer.
    - Tweezers are helpful to have for this task.
22. Set the Bed to a reasonable temperature for your material (80c for PLA straight to glass, 105c for ABS straight to glass) by typing in the number and pressing `Set`; watch to see the bed thermistor temperature change on the graph; avoid touching the heated bed when above 60c.

Starting the print

1. In pronterface, press the "Load File" button and browse to the .gcode file that you saved in Slic3r earlier.

2. Wait for the file to load; once it has loaded, an image will be displayed in the part preview window
3. Make sure that your printer thinks it is at 0, 0, 0 by sending the command M114 and verifying; if not, return to the previous section to set it at home.
4. Be ready to unplug power from the printer incase something goes wrong, press the "Print" button to begin the print
    - On RAMBo, you may consider hitting the Reset button instead.
    - Keep in mind that on the first layer, your fan should not turn on (as defined in your Slic3r cooling settings); this improves adhesion between the plastic and the glass
5. As the skirt prints around your object initially, manually adjust the height of the first layer to ensure that plastic is flowing smoothly and adhering to the bed; do this by turning the Z axis couplers by hand (make sure to move both in sync to retain the leveling); turning them clockwise raises the Z axis, and turning them counter-clockwise lowers the Z axis
6. In about 15 minutes or so, the print should complete; wait for the bed to cool down to about 30 degrees and then remove the fan mount and skirt from the glass; congratulations on your first print!

Congratulations on completing your first print on the MendelMax 2.0 3D printer!

We recommend that users now calibrate their printer in Marlin and tweak the settings used in Slic3r. Here is our recommended tuning guide.

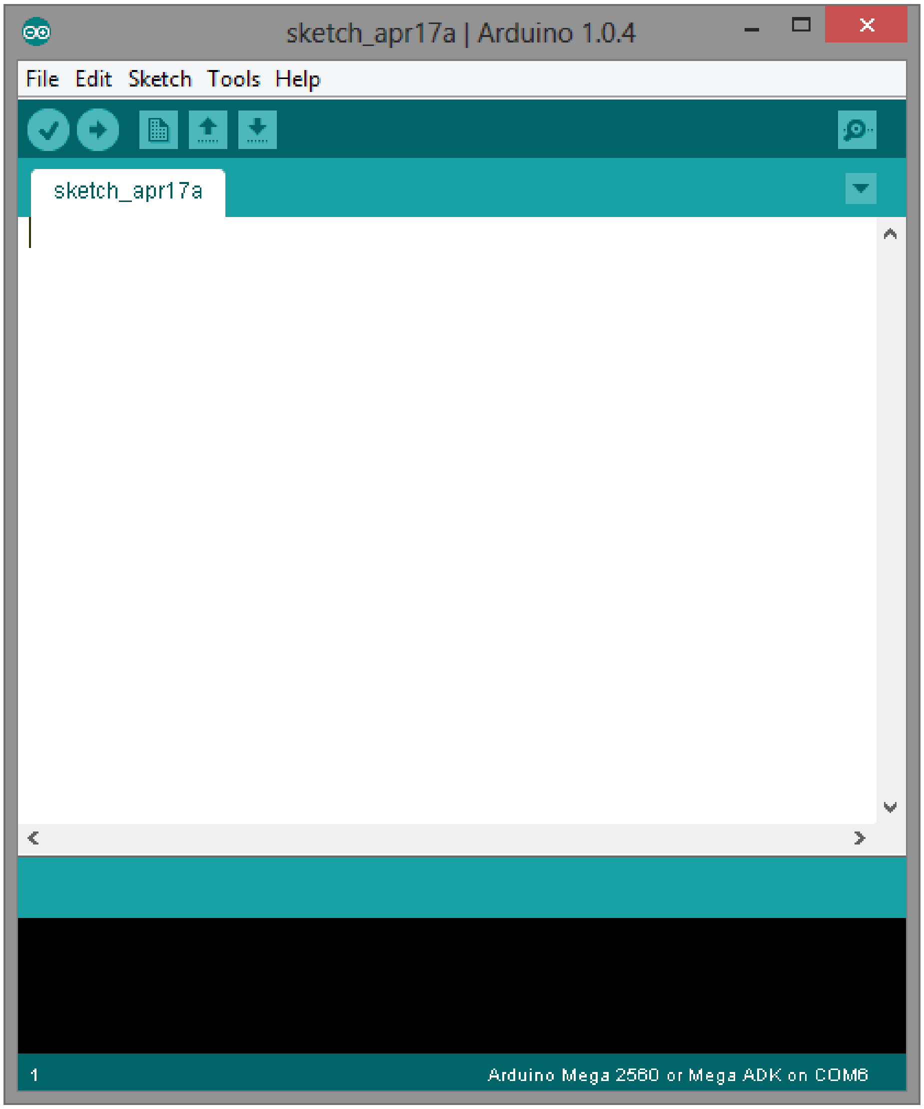
Additional Information

The following section should not be required on assembled units but is provided as a reference in case you need to perform them.

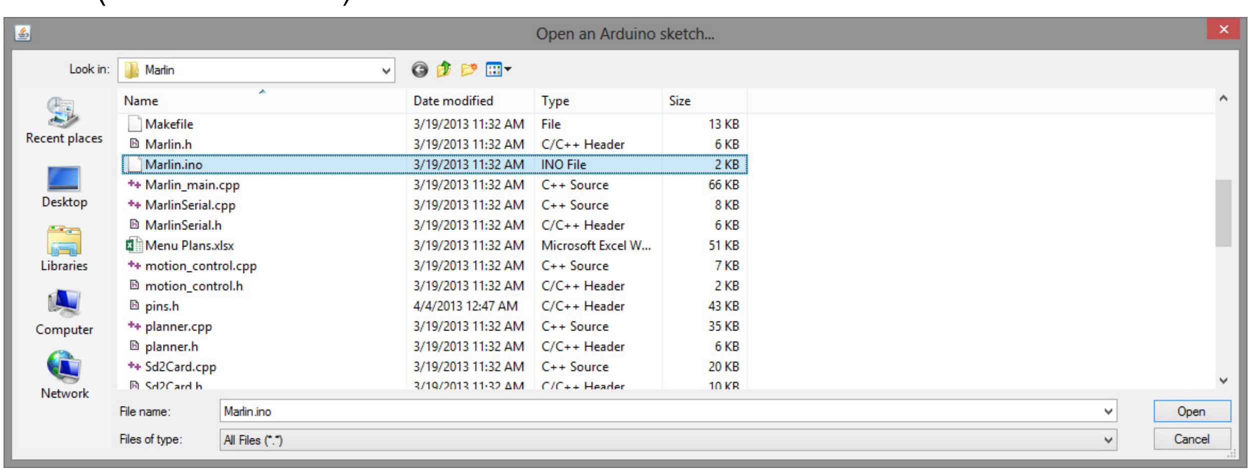
Uploading the Marlin firmware using Arduino to your print controller

The assembled printer already has the firmware uploaded and tuned, so these steps are not required right now. It will be helpful to read it for future reference.

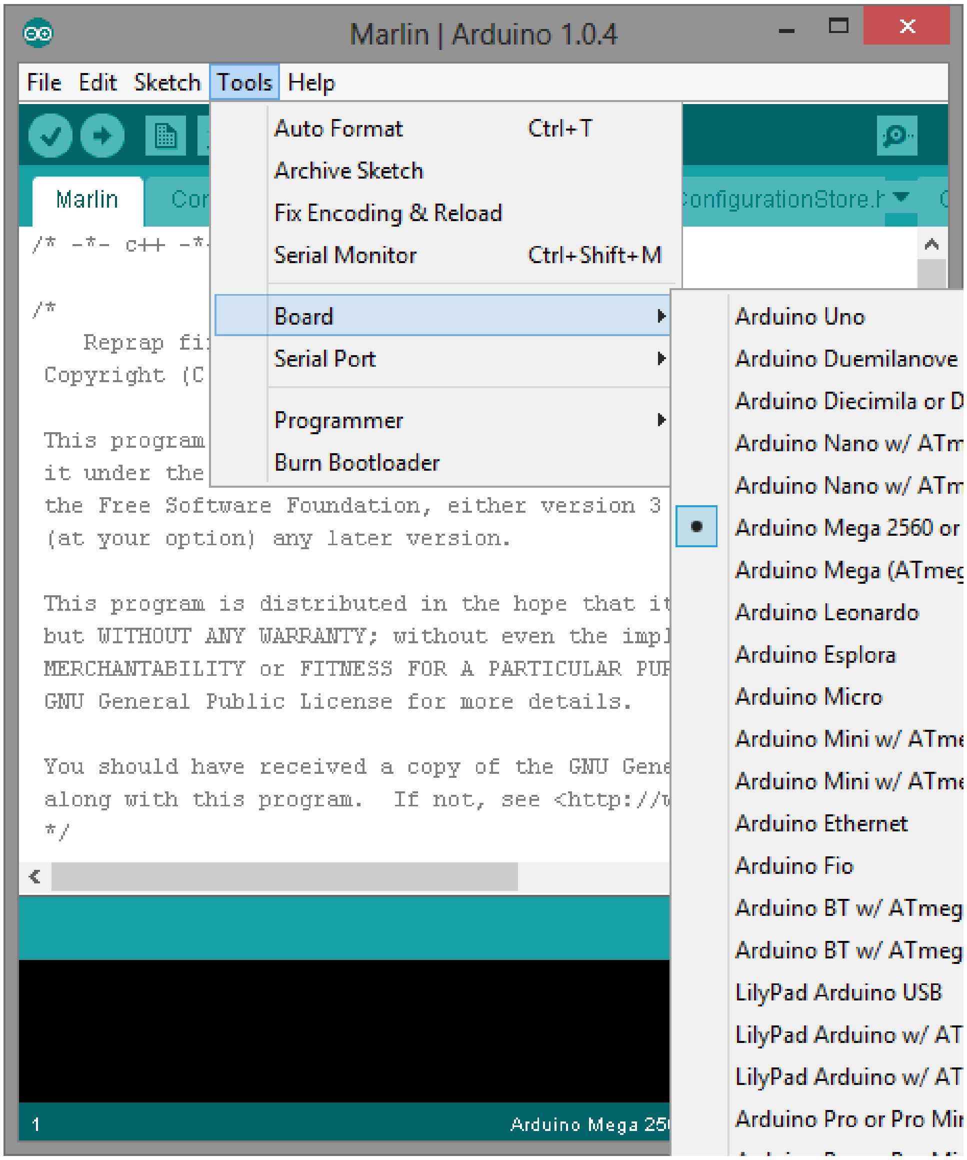
1. Open the previously-downloaded Arduino application.



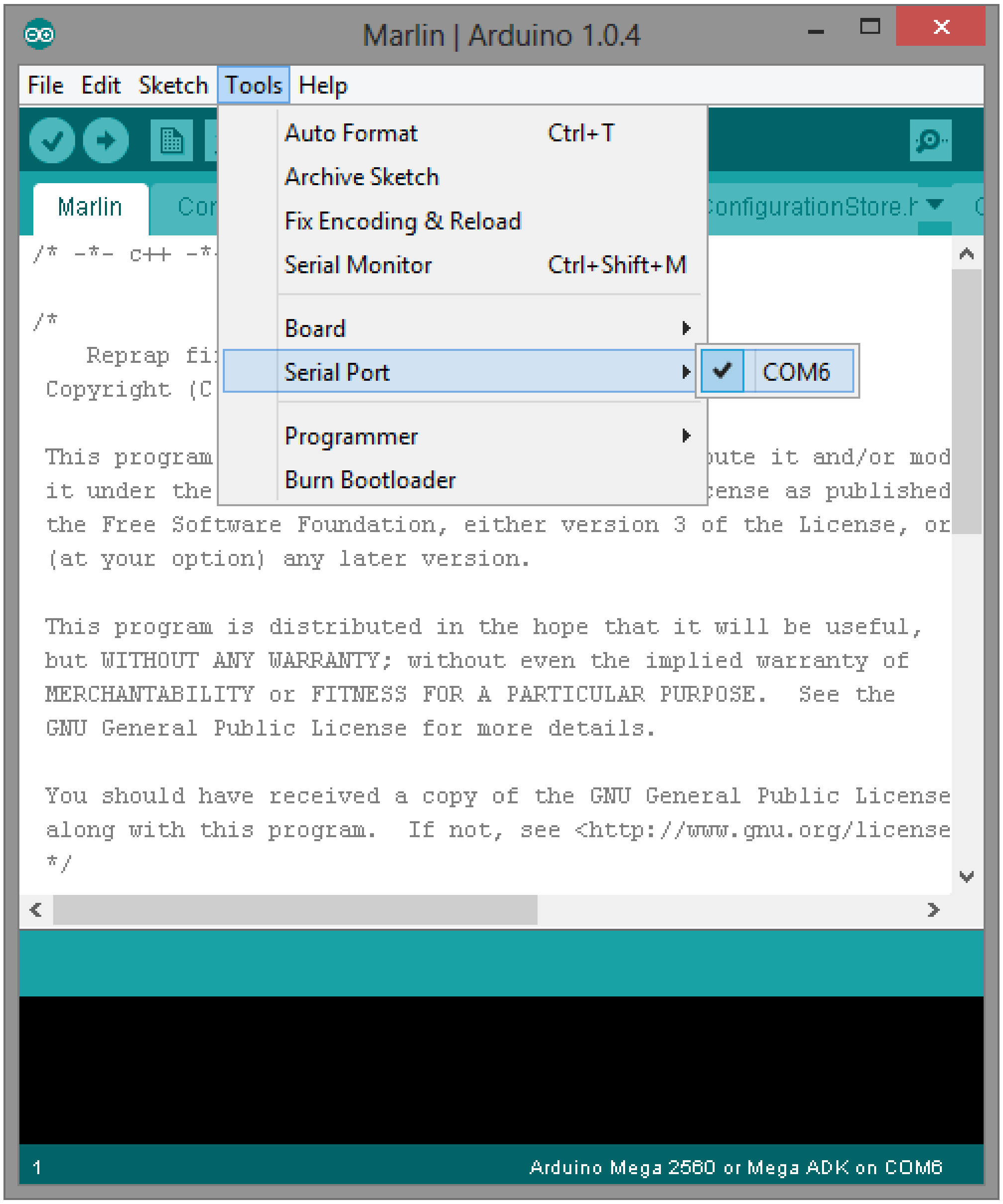
2. Select File: Open. Browse into the directory that you extracted your firmware, then in the Marlin folder, then selecting the file Marlin.pde or Marlin.ino (whichever exists)



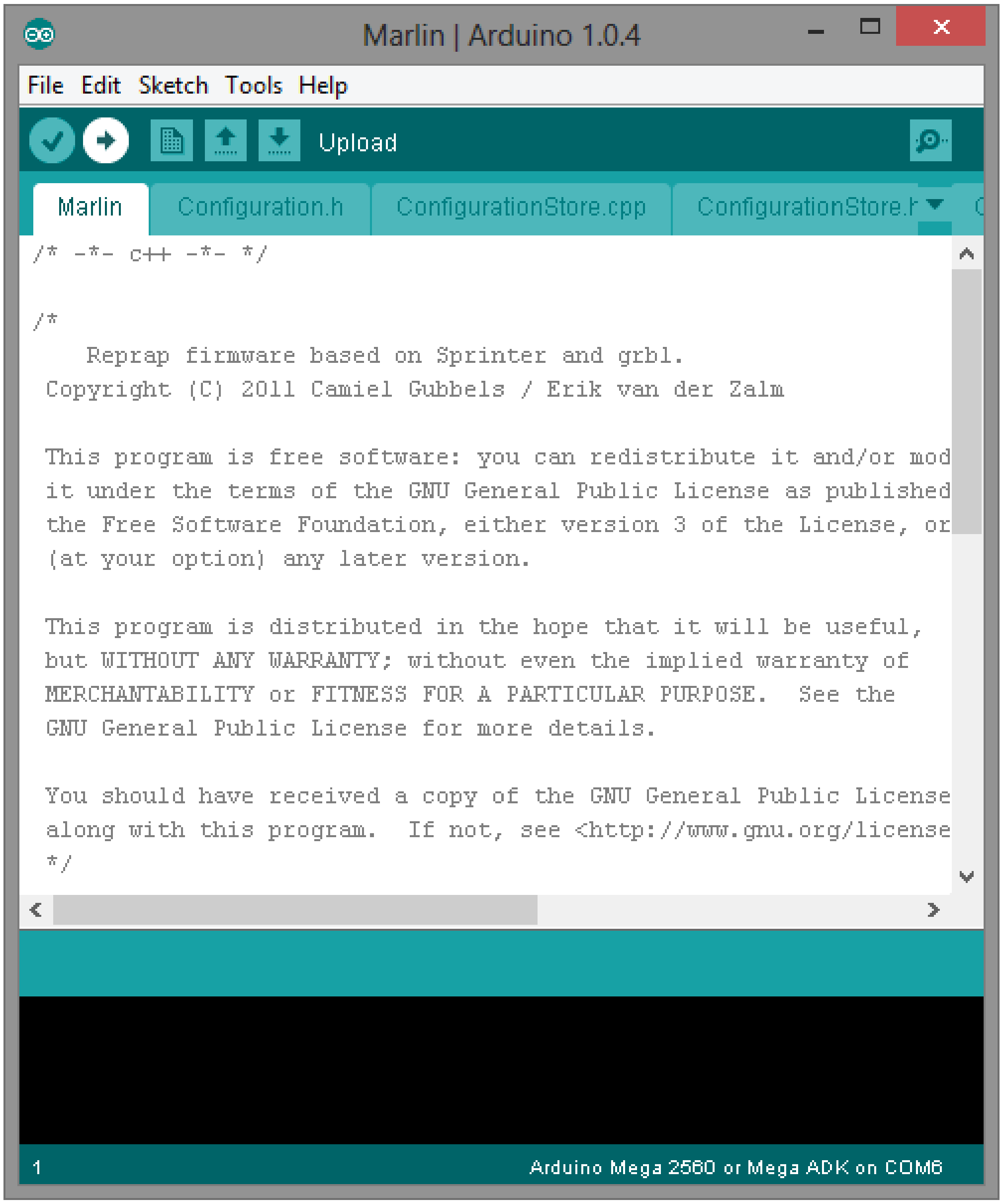
3. Select Tools: Board: Arduino Mega 2560



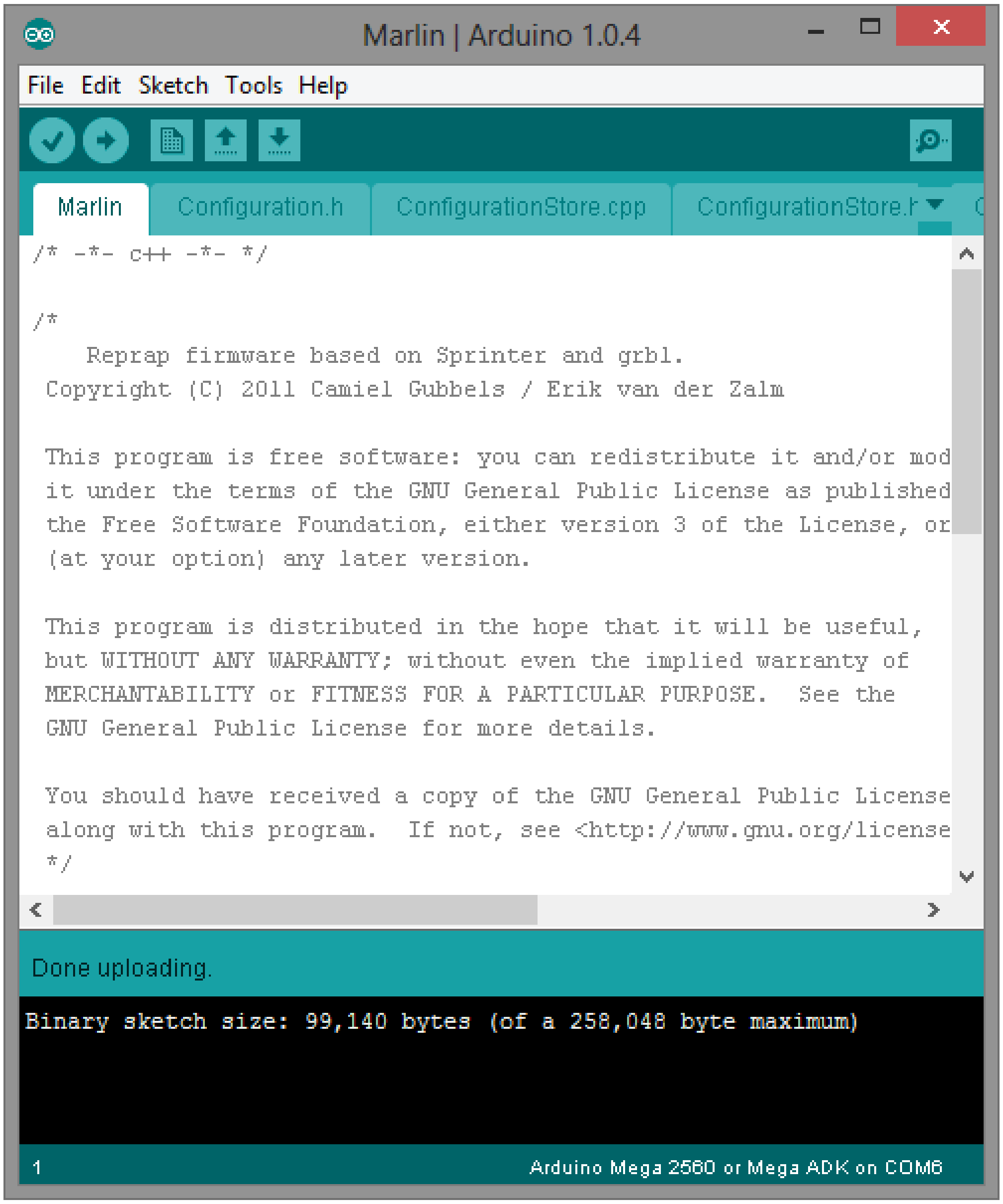
4. Select Tools: Serial Port: and then the port that you noted earlier when installing drivers



5. If you are running linux you need to change the firmware to `#define BAUDRATE 115200` in the configuration.h file.
6. Perform any modifications to the firmware and save them.
7. Press the Upload button



8. Check to see if the upload succeeded; if not, troubleshoot the board type, com port, and firmware



Wiring a PSU Switch

For switching the PSU on and off, you can use a SPST switch; we do not offer documentation at this time as to how to utilize this; if you choose to connect a switch, be certain to consult an electrician. A mounting bracket is available in our download center that mounts to the side of the bottom frame with M5-10 screws and M5 t-nuts.