Beginner's Guide to Operating the
Prusa i3 MK2.5 3-Dimensional Four-Color Printer

Kayla Pelloni and Adriana Noyes

Northern Vermont University - Johnson

May 17, 2019

Table of Contents

Starting up the Printer

1. Make sure the Prusa printer is plugged into the wall outlet.
2. Turn on the hood over the printer by finding the box labeled "Sentry Air Systems", and turn the black dial all the way to the right.
3. Turn on the printer using the power switch located on the right side of the printer. This will cause the panel on the front of the printer to light up blue and display a list of statistics about the printer.

Getting to Know the Info Screen

Once the printer is on, the screen will display the Info Screen, a list of statistics to refer to during the printing process.
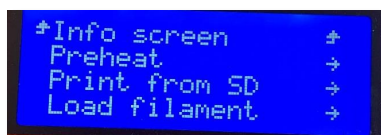Here is what the Info Screen looks like:

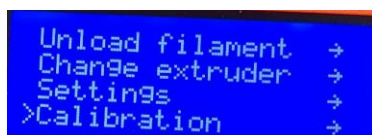Here is what these numbers mean, respectively:

1. Two temperatures are displayed here: the current nozzle temperature, and the temperature required to begin printing. The value depends on what type of filament is being used. Refer to the filament packaging to determine the nozzle temperature (often referred to as extrusion temperature.)
2. Two more temperatures are listed here: The current heatbed temperature, and the temperature required to begin printing. Use the recommended heat bed setting in the pre-sets in the **Settings** > **Preheat** menu. Please refer to the "Preheating" setting in this manual for more information.
3. This shows the percent completion of the print.
4. This shows that the printer is ready to print, and what it is printing.
5. This is the z-axis position.
6. This is the printing speed.
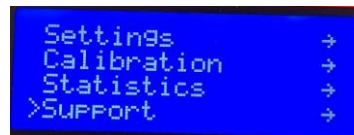7. This shows the elapsed time during the print.

Menu

To access the Menu, press the orange knob on the front of the printer. Here is what the screen will display (turn the orange knob clockwise to see more options, and press the knob to select an option):
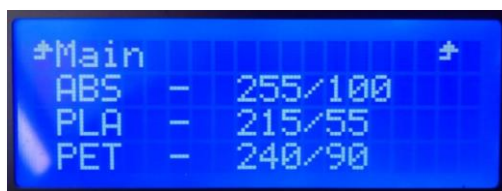

Preheating

It is always necessary to preheat the nozzle before beginning printing processes. This make cleaning the nozzle easier, and will ensure that the nozzle is hot enough to melt the filament. Here is how to pre-heat the nozzle:

1. Press the orange knob to access the Menu.
2. Select the **Preheat** option in the menu:

3. Scroll down, and select the option that best fits the temperature of the filament being used. Refer to the spool or packaging to find the extrusion temperature of the filament. For example, the black PLA filament spool states that the extrusion temperature for this type of filament is between the range of 180°C-210°C. The heatbed temperature for PLA is around 55°C. Therefore, the PLA – 215/55 setting would be the best option as this is Prusa's recommended nozzle and bed temperature setting.


4. When an option is selected, the printer will begin to preheat to the selected settings.

    HELPFUL HINT: The nozzle and heat bed temperatures can be manually changed before or during the printing process by going to **Menu** > **Settings** > **Temperature**, and selecting the mechanism to adjust the temperature.

Self-test

The self-test ensures that everything is working properly with the printer.

1. Go to the Menu, and select the **Calibration** option:

    ```
    Unload filament   →
    Change extruder   →
    Settings          →
    >Calibration      →
    ```
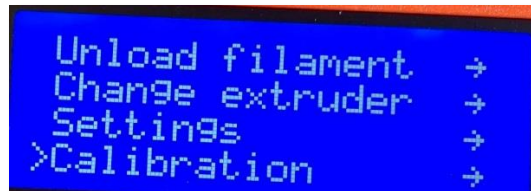
2. Select the **Self-test** option on the Calibration menu:

    ```
    Main              ↕
    Auto home
    >Selftest
    Calibrate XYZ
    ```

3. The Printer will read "Self-test Start". Just wait a few seconds.
4. A new screen will appear asking if the Left Hot end fan is spinning; If the small fan on the left side of the printing mechanism is spinning, select **Spinning**. If no, hit the orange "X" under the knob to stop the self-test.

    ```
    Fan test
    Front print fan?
     Spinning
    >Not spinning
    ```

5. Another screen will appear asking if the Front Print fan is spinning. If the small fan on front of the printing mechanism is spinning, select **Spinning**. If no, hit the orange "X" under the knob to stop the self-test.

    ```
    Fan test
    Left hotend fan?
     Spinning
    >Not spinning
    ```

6. The screen will now read "Checking Endstops". Just wait as the printer runs through these processes; each should read "OK" after each process if there are no issues.


7. If all goes well, the printer will automatically return to the Info screen, and display "Self-test OK" at the bottom of the screen. Congratulations! You have successfully ran a self-test.
8. In the event that an error occurs, an error number will be displayed. Please refer to the Prusa User Manual for errors, or try to use the troubleshooting steps provided in this manual.

Calibration

Calibration is done in order to make sure that the nozzle is exactly where the printer thinks it is when printing.

1. Preheat the nozzle to the appropriate nozzle temperature, as listed in the first section of this manual. (Ex. PLA: 215°C/55°C)
   See the section titled "Preheating the Nozzle" for more guidance.
2. Select the **Calibration** option in the Menu.
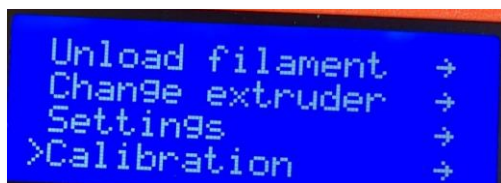
3. Select **Calibrate XYZ** option in the Calibration Menu.

4. Using the dial on the front of the printer, rotate the orange knob in either direction **SLOWLY** until the black stopper reaches the top. When the printer makes a grinding noise, **STOP**. Press the orange knob to continue.
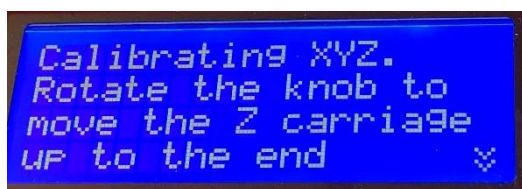
5. At highest Z-Axis point, it is easier to clean the nozzle out. Using the small needle-like mechanism, insert the tip of the needle in the small metal nozzle end located here:
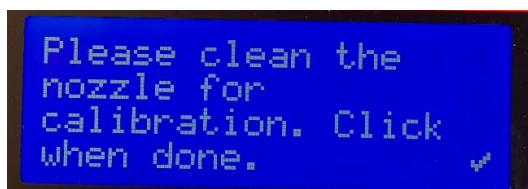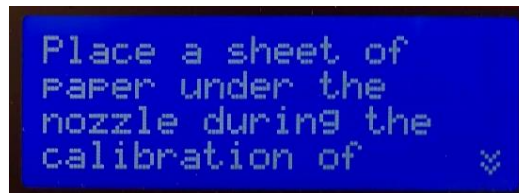

The needle should be inserted all the way down to the handle to ensure nozzle is clean. Press the orange knob to continue.

6. If the nozzle is at the highest Z-axis point, which it should be, press "Yes" to the next prompt:
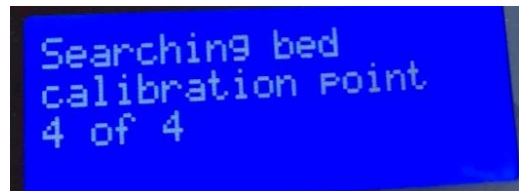
```
Are left and right
Z carriages all up?
 Yes
>No
```

7. Place a sheet of paper on the heatbed, and press the orange knob to continue.
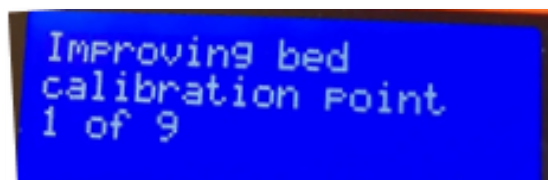
```
Place a sheet of
paper under the
nozzle during the
calibration of       ⌄
```

8. The screen should now read "Searching bed calibration point". Allow the printer to run through all four processes without interuption. Please ensure that the paper does not slide around too much on the heatbed during this proccess; it must stay under the nozzle. If all goes well, you will see this screen:
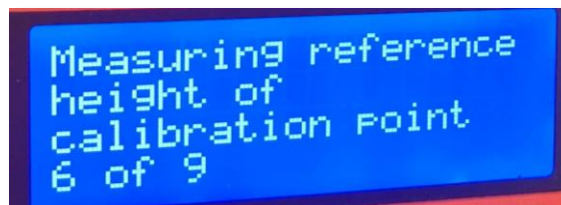
```
Searching bed
calibration point
4 of 4
```

The printer will automatically advance to the next process, which will look like this:

```
Improving bed
calibration point
1 of 9
```
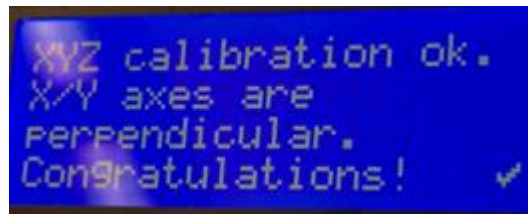
Allow the printer to run through all nine calibration points. The printer will automatically continue to the next process, which looks like this:

```
Measuring reference
height of
calibration point
6 of 9
```

Here is the screen you will see next, if there are no errors or interuptions:

Congratulations! XYZ axes have been calibrated successfully.

Loading and Unloading Filaments

One of the most common problems when working with the Prusa is that the filament occasionally breaks during or before a printing process. This issue can be resolved by reloading the filament. It is best to reload all four filament extruders before beginning the printing process.
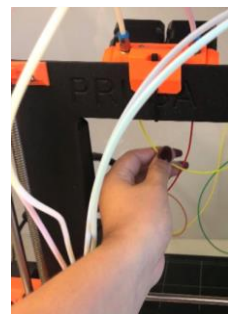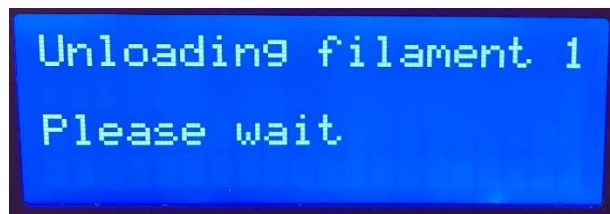
Here is how to **unload all** filament:

Note: The nozzle <u>MUST</u> be preheated before beginning this process. Please refer to the "Preheating" setting in this manual for more information.

1. Select the **Unload Filament** option in the menu.
2. Select **Unload All**

3. The printer will automatically begin to unload the filament, one extruder at a time. During this time, reach back and pull gently on the filament to guide it out of the extruder gears, until it is fully removed. Be careful not to burn yourself on the hot nozzle or heatbed during this process. Repeat this process for all four extruders; the printer will automatically advance to each extruder.


4. After all four filament are pulled from the gears, the unloading process is finished.

Here is how to **load all** filaments:

1. Each filament must be cut at a 45° angle before loading. Using a heavy duty pair of scissors, tilt the scissors at a 45° angle to the filament, and cut, thus making an angled and somewhat sharp point. Repeat for all four filaments.
2. Select the **Load Filament** option in the menu.
3. Select **Load All**
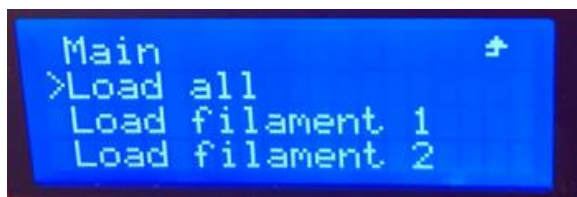
4. Insert the angled filament tip into the extruder gears, as shown below. The gear will grind against the filament, causing a small grinding sound. Press the orange knob.
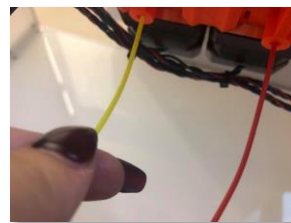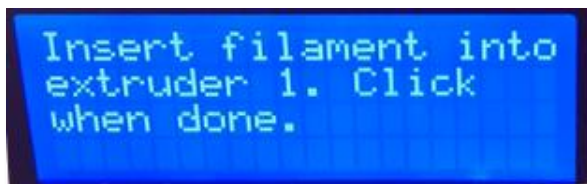
5. The printer will automatically begin to load the filament. During this time, you should reach back and guider the filament up into the extruder gears, until it is fully loaded. Be careful not to burn yourself on the hot nozzle or heatbed during this process. The color of the filament should reach all the way to the nozzle end. If it does not, press the "x" under the knob, unload the filament and reload it again. Repeat this process for all four extruders; the printer will automatically advance to each extruder.
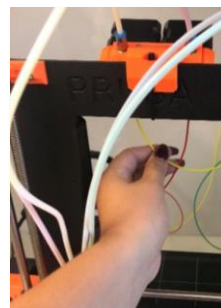
6. After all four filaments are fully inserted, the loading process is finished.

How to Upload a G-Code to the Printer SD Card

1. First, search online for a template to print. The easiest way is to search for 3D printing templates for the Prusa MK 2.5. Once the file of an object to print has been located, download the file to the **Data Drive (E:)** > **PrusaDocuments** > **PrintingTemplates** folder. The files that are supported with the Slic3r Prusa Edition application are .stl, .obj, .amf, .xml and .prusa files. It is highly recommended to use files that are designed for Prusa printers. Feel free to use the templates already in the **PrintingTemplates** folder.

2. Once the file has been downloaded, remove the SD card located on the left side of the printing screen, and insert it into the SD card slot located on top of the computer.
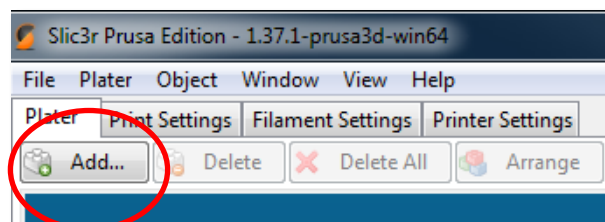

3. Once the SD card is inserted, there will be a new drive under **Computer** reading **PRUSACARD (H:)**.
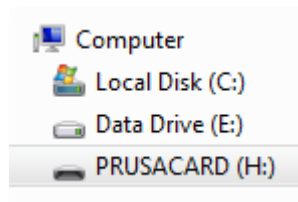
4. Open the **Prusa3D Slic3r MK2** application located on the desktop.

5. Click the **Add…** button located in the top left corner of the screen.

6. This will bring up a window of all the files on the computer that are compatible with this software (.stl, .obj, .amf, .xml and .prusa files). Select the file downloaded in step 1. This should bring up a picture of the downloaded 3D model on the plane.

   Note: the block shown in the corner of the plane is the waste block the printer will use to alternate colors throughout the print; single color prints do not have waste blocks

7. Now select **Export G-Code**, browse to **PRUSACARD (H:)**, and select **Export**. Make sure to name the file something recognizable on the Prusa screen (less than 20 characters)
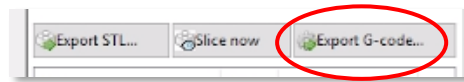
   The g-code file for your 3-D model is now uploaded to the Prusa SD card.

8. Now browse to the **PRUSACARD (H:)**, right click, and **Eject** the card from the computer. Insert the SD card back into the SD slot on the Prusa printer.

9. The 3-D model will now be accessible on the SD from the printer.

How to Print From the SD Card

Note: It is only after running through the **Calibration** (page 5), **Self-Test** (page 3), and **Pre-heating** (page 2), and **Load all Filaments** (page 8) processes described in this manual that one can begin to print a model. Please ensure that <u>all</u> these steps have been completed prior to printing a model.

In this tutorial, we will be using the MK3 PLA Prusa Logo model, a preset code tested by Prusa developers. This code will produce a small, dual-colored Prusa logo, as pictured below:

1. On the main menu, select **Print from SD**

2. Select the file named **MK3_PLA_Prusa_200um_20M.gcode**.

3. The printer will automatically begin running processes to prepare for printing. The print is preprogramed to pre-heating the nozzle and heatbed to the proper PLA temperatures along with repositioning the nozzle. The screen will display printing statistics with a status of the print at the bottom.

    At this point, the printer will complete all printing processes automatically until the model is complete. If the filament is not looking smooth, and is displaying blobbing, stringing, or does not adhere to the heatbed, **Pause Print** and adjust settings of the print manually to resolve these issues (refer to **Adjusting Minor Settings During a Print**), or **Stop Print** and refer to the **Troubleshooting** guide for more support.

4. Once the printer has fully completed printing the object, there will be a prompt to unload filaments. Select the option to **Unload All** filaments used in the print. This will allow the printer to extrude all excess filament from the nozzle before unloading the filament.

5. Wait until the bed has cooled, and use the scraping tool to carefully remove the object and all excess glue or filament from the heatbed.

6. Spray Windex™ cleaner only on the heatbed, let set for 30 seconds, and wipe excess with a paper towel.

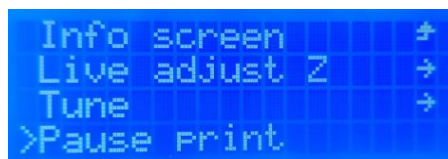## Adjusting Minor Settings During a Print

Adhesion Adjustments

If there are issues with the filament not adhering to the bed, try using glue (any washable glue stick) to increase adhesion. If this does not work, refer to the **Troubleshooting** section for adhesion solutions.

Stringing or Blobbing Adjustments
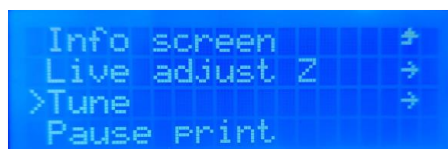
If stringing or blobbing is a minor issue, adjust the settings of the print using the **Tune** menu. Here is how to access the menu:

1. First, pause the print by pressing the orange knob, scrolling down and selecting **Pause Print** on the menu.


2. Press the orange knob again to open the options menu while the printer is finishing movements. Scroll down and select **Tune.**

This will display a menu of settings that can be adjusted during the print.
Select an option, and turn the knob to increase or decrease temperature, flow, or speed of a feature. Press the orange knob after dialing in the correct value to change this setting. The printer will automatically return to the info screen and begin adjusting.
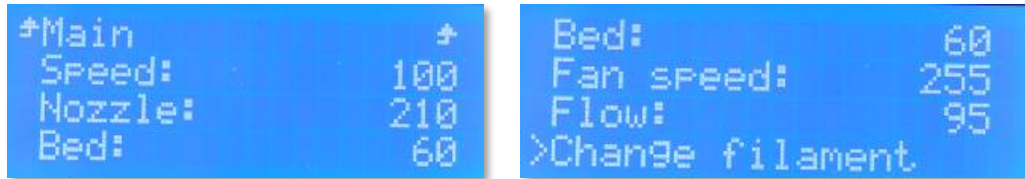
3. Watch the status of the changed setting on the info screen. Refer to the **Getting to Know the Info Screen** (page 1) to see what these values represent.
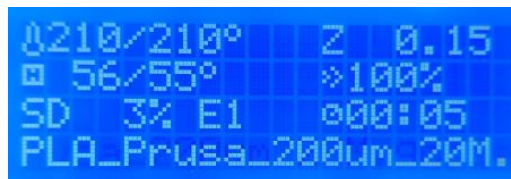
4. When the changed setting reaches the desired amount, press the orange knob to access the menu, scroll down , and select **Resume Print**.

   If at any point the printing process needs to be stopped, please use the **Stop Print** option in the options menu. This will provide steps for unloading the filament used in the print, which is a required step after every print. Make sure to select **Unload All Filaments** when prompted to do so.

   If there are still problems with filament blobbing, stringing, or not adhering, refer to the **Troubleshooting** section of this manual for more support.

Troubleshooting

Here are some common errors in the printing process, as well as some solutions to take to reduce these errors:

Stringing and Blobbing

Adjust Extrusion Rate

Extrusion Rate (Flow) is the value which determines how much filament is coming out of the nozzle.
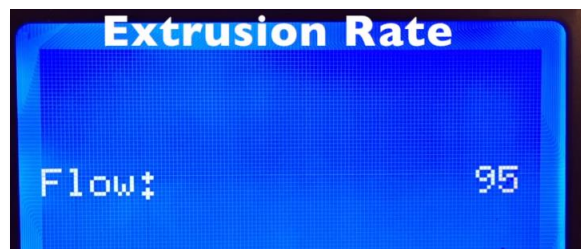
the extrusion should match what the printer is expecting, otherwise there will be over-extrusion, which can lead to blobbing and stringing:

Under-extrusion can lead to weak prints with gapping:

Generally, with any filament, the higher the temperature is raised the stringier it will get. If the prints are really stringy, try dropping the temperature.

Here's a print with identical print settings, except the temperature was dropped ten degrees in the second print:

Notice how the stringiness has been reduced. Be careful not to print too cold though; this will produce weak prints and nozzle jamming.

Solution: Try Running Extrusion Rate Calibration Using the Calibration Steps Model

Extrusion rate is one of the most important factors to a great print. It allows the model to have strength and surface function. This means that many print issues can come from this one setting being wrong.

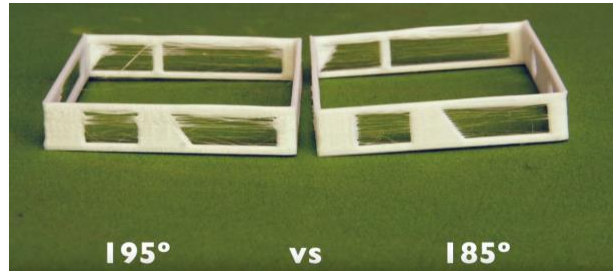Extrusion rate is specific to each filament so when working with a new filament it is important to set the extrusion rate accordingly. In order to do this take a few diameter measurements of the filament to get the average diameter. Using the Extrusion Rate Model found on the SD card will allow for easily setting the extrusion rate correctly.

Run the model and set the infill percentage to 95% to the outer walls and three top layers. Run the model at a medium seed and set the layer height to 0.2 millimeters. Ignore the first 5 layers of the model and look as the infill begins to run. Small gaps should appear between the infill lines. If no gaps are present, the extrusion rate is too high. Drop the extrusion rate on the printer and allow a few more layers to run before looking at the infill again. If the printer doesn't allow for changing the extrusion rate mid-print, stop the print and change the extrusion rate in slicer and repeat the previous steps.

The top layer of the print should be smooth without any gaps, if gaps are present in the top layer then the extrusion rate is too low. The perfect extrusion rate will have a smooth, gapless top layer with tiny gaps in the infill. All exterior edges should be smooth as well.

Retraction

Retraction is the amount that the printer will rewind filament after it finished a section of printing. There are usually two values which need to be adjusted in Sli3er: the retraction length and retraction speed. Both values will affect the end result, however retraction length is the most important value to adjust first, since the low values produce excessive stringing while the high values can cause jamming.

Solution: Try adjusting retraction in the nozzle using the Ice Biome Model.

Begin by printing the ice biome model, letting it run long enough to observe the stringing between the points. If stringing begins, stop the model and increase the retraction. Continue this process until the stringing stops.
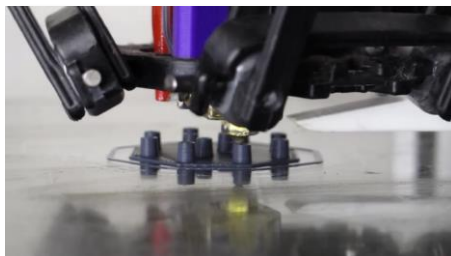
If there is no improvement lowering the retraction, try setting the retraction speed in printer settings to 20 millimeters a second for PLA. For most filaments 60 millimeters a second is the ideal speed so increase the speed until there is improvement in the stringing.
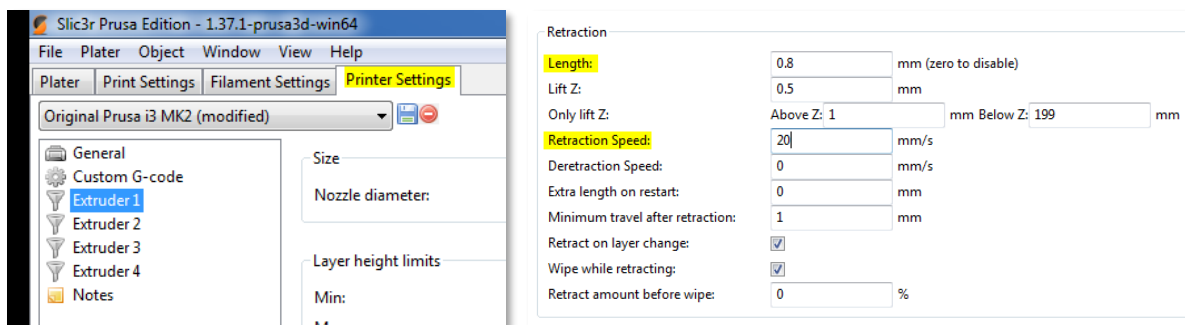
Here is how to adjust retraction in Slic3r:

1. To adjust retraction, open the desired object file in the **Prusa3D Slic3r MK2** application located on the desktop.
2. In **Slic3r**, select **Printer Settings** > **Extruder 1** (It is necessary to repeat this process for all extruders being used)

Quick Tips for Stringing

- Lower the extrusion temperature to 190
- Increase retraction to 0.6 mm
- Lower the retraction speed. Set the extruder speed to 25mm/second. Set the print speed to slow, around 50%. Set the fans to 100% with no z-lift.

If there is still no improvement in the straining retraction may not be the issue. Try replacing the 1` end (refer to the Prusa User Manual for replacing printer parts, such as hot end).

For more troubleshooting help, visit Prusa's Quick Troubleshooting guide here:
https://www.prusaprinters.org/how-to-fix-the-most-common-3d-printing-errors/

Quick Tips

- Hold down orange knob on printer for a faster way to adjust z-axis position
- The nozzle and heat bed temperatures can be manually changed before or during the printing process by going to **Menu** > **Settings** > **Temperature**, and selecting the mechanism to adjust or set.
- Online support can be your best resource to resolving issues with this printer.
- Refer to Prusa's Quick Troubleshooting guide here: https://www.prusaprinters.org/how-to-fix-the-most-common-3d-printing-errors/
- For any further questions, or for tips and tricks visit https://help.prusa3d.com/ it contains all the necessary information for using the Perusa printer.

Contributions

Kayla Pelloni is the primary contributor to the Beginner's Guide to Operating Prusa i3 MK2.5 3-Dimensional Printer, developed in Spring and Fall semesters of 2018. Kayla constructed and formatted the sections of the manual focusing on "Starting up the Printer", "Loading and Unloading Filaments", "How to Upload a G-Code to the Printer SD Card", and "How to Print From the SD Card". Kayla also edited the "Troubleshooting" section, and is responsible for the format of the overall document.

Adriana Noyes is the secondary contributor to the Beginner's Guide to Operating Prusa i3 MK2.5 3-Dimensional Printer, developed in Spring 2018. Adriana collected photographs to match the procedural steps in the manual. Adriana is the primary editor of the section on troubleshooting. Adriana Noyes made FINAL edits to paper.

Kayla Pelloni and Adriana Noyes worked collectively during the Spring 2018 semester to assess the overall issues with the Prusa printer, and printed several small 3-dimensional frog models during the process.

Kayla Pelloni and Adriana Noyes attended a workshop on Turning Drone Data Into Information at University of Vermont in Spring 2018. This workshop was part of the Geological Society of America's 53rd annual meeting. This workshop focused on reviewing drone platforms, drone sensors, and turning drone data into information that can be used for geospatial analysis.

Kayla Pelloni constructed and presented a poster presentation for the Spring 2018 Student Symposium at Northern Vermont University- Johnson. This presentation summed up the overall troubleshooting process during the 2018 semester as well as future projections, and included a small frog model that was printed successfully.

Adriana Noyes produced a multicolor print of a penguin at the end of the Spring 2018 semester, after several months of failed attempts of producing a multi-colored model.

Kayla Pelloni carried out a successful print of a flexible Tyrannosaurus Rex in the Fall 2018 semester. This project was successful after several attempts. The purpose of this project was to assess the issues associated with a basic print, and adjust basic settings of the printer to reduce aesthetic imperfections such as stringing and blobbing. This project was the first flexible 3-dimensional model to be produced from the Prusa printer at Northern Vermont University-Johnson

Kayla Pelloni operated a DJI Phantom Pro Drone and captured a collection of images of the Northern Vermont University- Johnson campus in the Fall 2018 semester. The images were then imported into Agisoft Photoscan to be stitched together to construct a basic 3-dimensional model of the campus. While the importation and development of the model was successful,

the model could not be printed on a small scale with the Prusa 3-dimensional printer due to gaps in the model as a result of flight issues.

Adriana Noyes carried out the upgrade of the Prusa MK2.5 to MK3 during the Fall 2018 and Spring 2019 semesters. This process includes using the Prusa printer to print small parts, which are then used to add new features to the printer.