FT-5 R2

Setup Manual v1.0

Folger Technologies LLC

www.folgertech.com

Table of Contents

Preface

Congratulations! If you have made it this far, then you have successfully built your Folger Tech FT-5 R2 3D Printer Kit. Before you move any further, triple check your wiring to avoid damaging components.

Make sure all components have been fully assembled, and check the X and Y axis movement by manually moving them by hand, feeling for any irregular friction. Make sure all bolts are tight, all wires are connected, and any dust or debris from the assembly process is cleaned off.

Assuming everything checks out, you can continue on with this setup and calibration guide.

Firmware

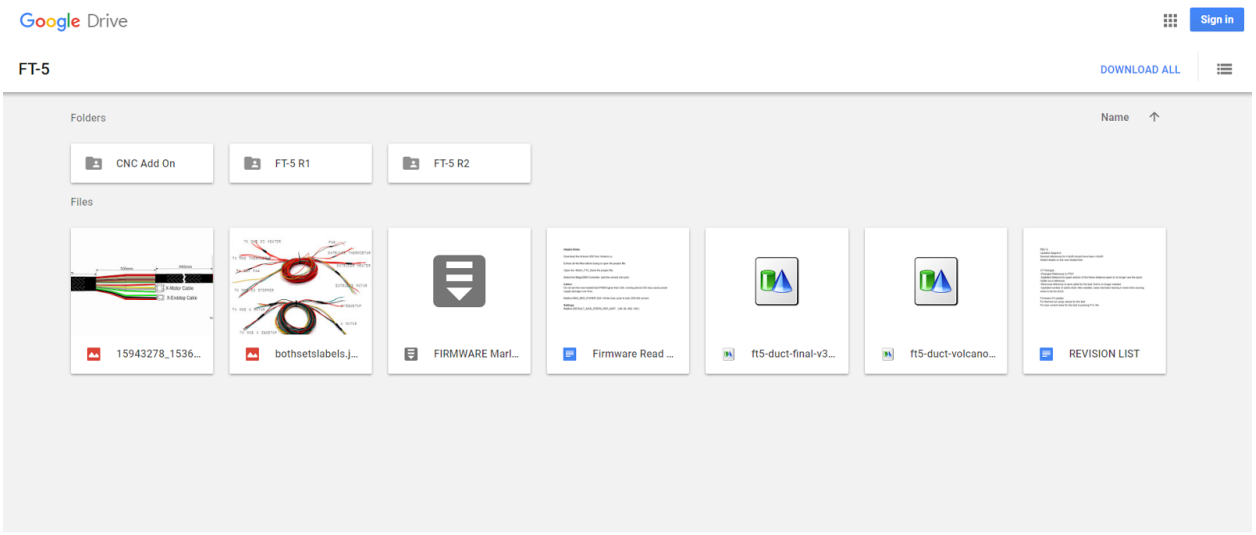
Controlling every being is a brain, and influencing the brain, is knowledge. The MKS Gen 1.4 board is the brain of the FT-5 R2, but out of the box, it has no knowledge. It doesn't know it's purpose in life, and will sit as a useless circuit board until it gains the information needed to operate as a 3D printer. You are now the teacher, and it is time to teach the board how to do its magic.

Step 1 - Locate the firmware

Like all of our kits, we use the open source firmware 'Marlin'. While you are more than welcome with experimenting and trying other firmwares (when compatible with the board), the stock firmware is setup and calibrated to be easy to get the FT-5 R2 running. Fine adjustment may be needed later on, but it will get the ball rolling.

The firmware is located on the Google Drive page for the printer. This link will always be on the printer's store page on the Folger Tech website. Currently, the link is: https://drive.google.com/drive/folders/0B9b1NbuMK524aGhPQXRLTW9mams

This drive contains the guides, firmwares, and other resources for both the FT-5 R1 and R2.



On the drive should be the firmware ZIP file. As we upgrade the firmware to newer versions of Marlin, the name may change, but currently it is:
FIRMWARE Marlin_v1_1_2_ R1R2ft5_stock.zip

It will always be a zip file, and should be labeled 'Firmware'.

Click and download, save to somewhere you will remember.

Step 2 - Arduino IDE, and opening the firmware

If you click on your brand new zip file, you will need to extract it to a new folder somewhere on the computer. I recommend whatever folder you put the Zip file in.

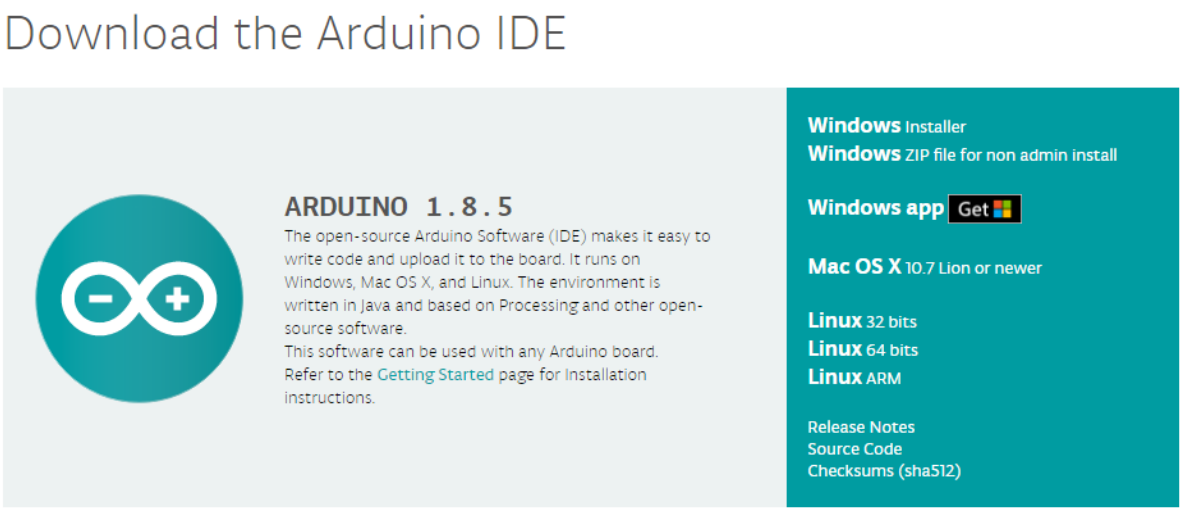
Once you do this, you should have a new folder. Navigate in until you find the '**Marlin_ft5_stock'** folder. Inside this folder lies the firmware.

Inside you will see all the separate components of the firmware. It can be fairly daunting seeing all of those files without knowing what they do. Don't let this scare you, as there is an easy solution, ARDUINO IDE!

The Marlin firmware is Arduino based. In order to easily open and manipulate the firmware, we need to use the Arduino IDE software.

https://www.arduino.cc/en/Main/Software

Navigate to the above link and download the newest version for your operating system. In rare cases, Arduino will update something within their own ecosystem, which will render the firmware un-uploadable. If that were to happen, you can download older versions of Arduino from that page as well.

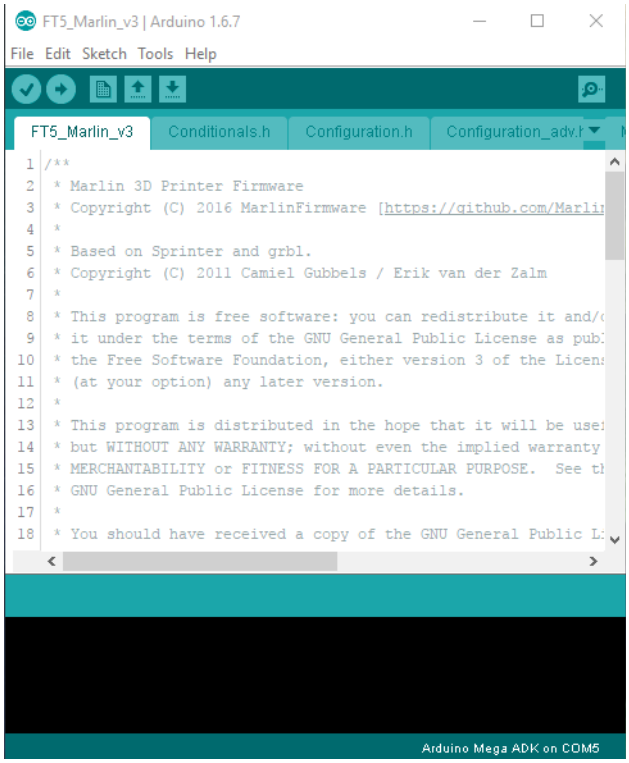


Install the software, and open it.

Go to **File>Open**, then navigate to the Firmware folder you just downloaded.

Open **Marlin_ft5_stock.ino** from within the folder. You should see multiple tabs open up.

While you *shouldn't* need to do any firmware modifications to get the printer running, the majority of changes you may need to make is on
the **Configuration.h** tab. We will discuss this more later on.



Step 3 - Connecting the Board and Uploading the Firmware

While having the firmware on your computer is great and all, it won't do you any good unless it is on the board.

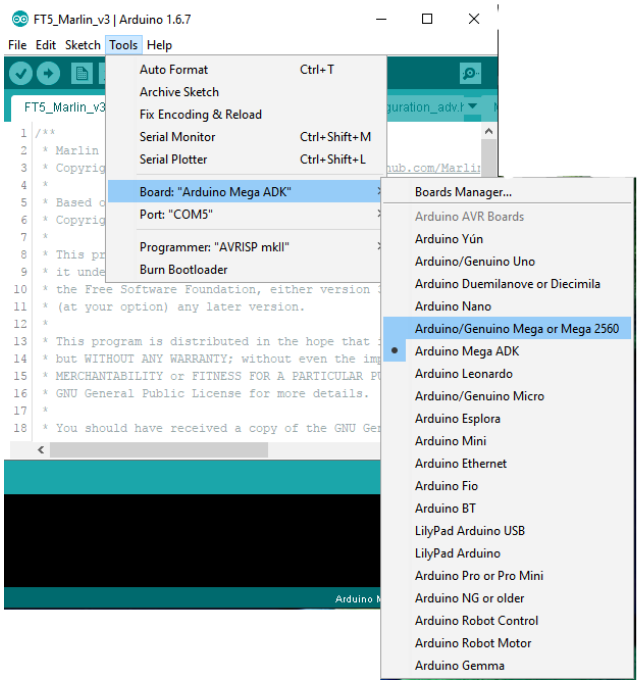
Take the USB A to USB A cable, and connect the printer to the computer. It is best to do this part with the PSU power OFF, although you should be able to upload the firmware regardless of the power status.

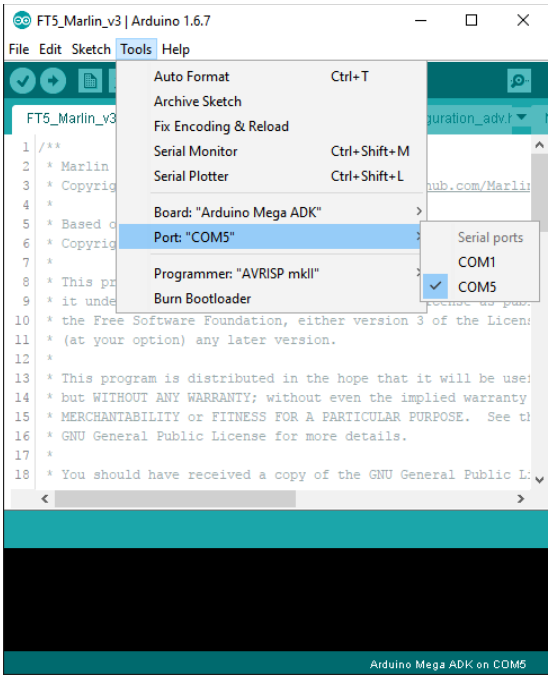
Upon connection, you should hear/see a connection notification on your computer. The board will install some basic drivers for a minute or two. If you do not get any sort of connection notification, there could be an issue with the wiring of the board, which is preventing connection.

Assuming you have a good connection, go back into Arduino IDE.

Go to **Tools> Board** then pick either
Arduino/Genuino Mega or Mega 2560
or
Arduino Mega ADK

The latter seems to be more reliable, but for some reason, some boards prefer one over the other. If you get any errors during the upload process, try the other board selection.

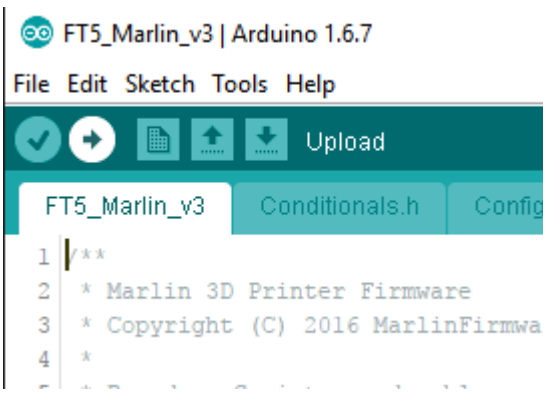




Next go to **Tools>Port**, and select the COM port for your MKS board. Normally, Arduino IDE will automatically select the board when it detects the board. COM1 is almost NEVER the correct option, so do not pick that.

If COM1 is your only option, or you do not see any Ports listed, restart the IDE software. If that does not work, then try a different USB port on the computer. USB 3 ports seem to sometimes have issue, so use USB 2 ports if possible. Using USB hubs or adapters can also cause issues.

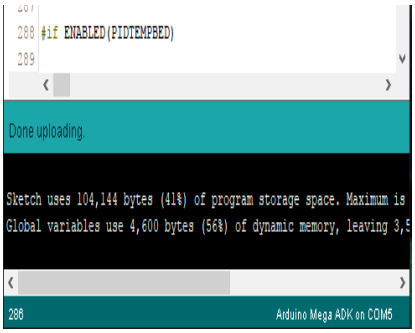
Once you have the Board and Port selected, you are ready to upload.



The small check arrow under the File tab is the verify button. If you make changes to the firmware, use this button to verify that your code is still correct in terms of syntax.

The small right arrow is upload. Clicking this will verify the code, then upload the code to the connected board.

Click the upload button to begin the upload process. You will see various information and code appear towards the bottom of the window.



When the code is finished uploading to the board, you will see “Done uploading”.

Congratulations, your board now knows what it needs to be a 3D printer. The next part is setting up the software that we will use to test, calibrate, and do some first prints.

Slicer Setup and Calibration

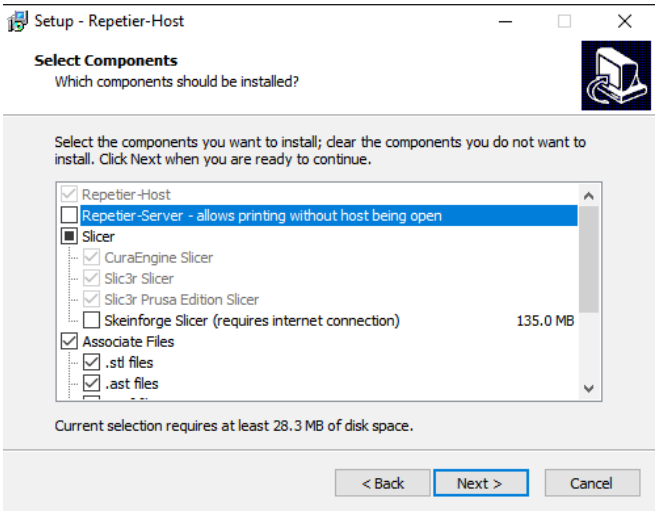
A 3D printer, by its own nature, is an awesome desktop decoration, but that is surely not why you bought it. Your FT-5 R2 should now be fully assembled, and have the proper firmware loaded onto it. But that is only half the battle. The next step is to set up the slicer software, and do some basic calibration.

Step 1 - Repetier Setup

For this guide, we'll be using the software Repetier-Host. There are a large number of slicers in the 3D Printing community, many free and fantastic, but we will be using Repetier due to it's simple design, and easy to use machine controls with integrated slicer. The newer versions of the software Cura now has an FT5 preset which you may find useful, but the overall amount of options and settings may confuse a new user. As you learn, feel free to try out any slicers you can find, as they should all be compatible.

Head over to the link below and download the latest version. Your version may not look exactly like what you see in this guide as they update the software, or depending on the OS you are currently on, but the general setup is basically the same regardless.

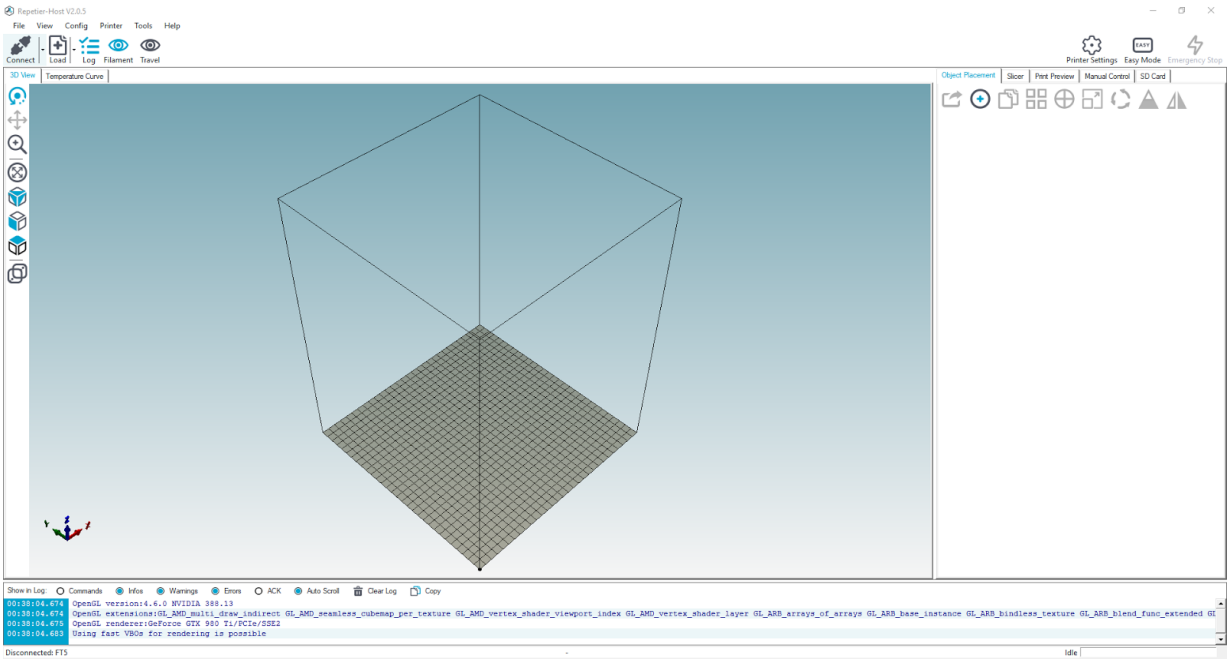
https://www.repetier.com/download-now/



Go through the standard installation process, with one exception. On the page seen here, unselect Repetier-Server. Unless you *know* that you will be using it in your application (server based printing), it is not needed, and it can sometimes cause connection issues with the board and the Arduino Software.

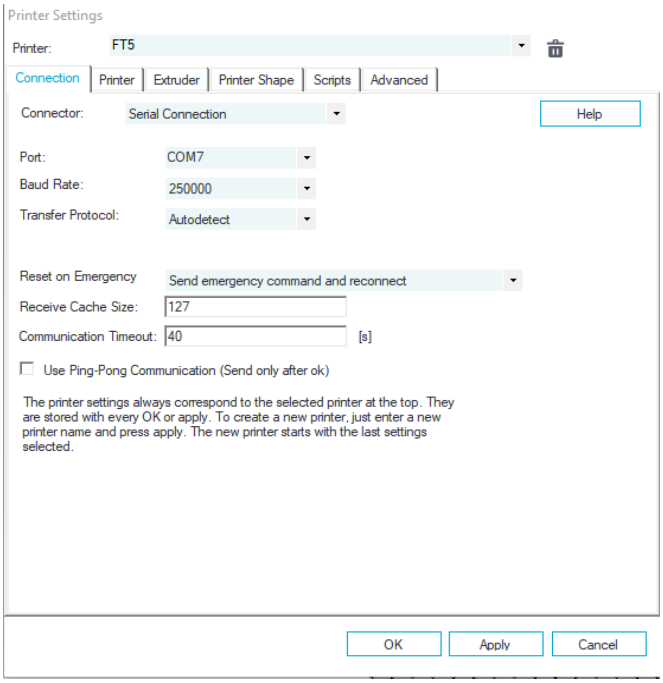
Once it is installed, open up Repetier-Host.

You should be greeted with a screen that looks similar to this:

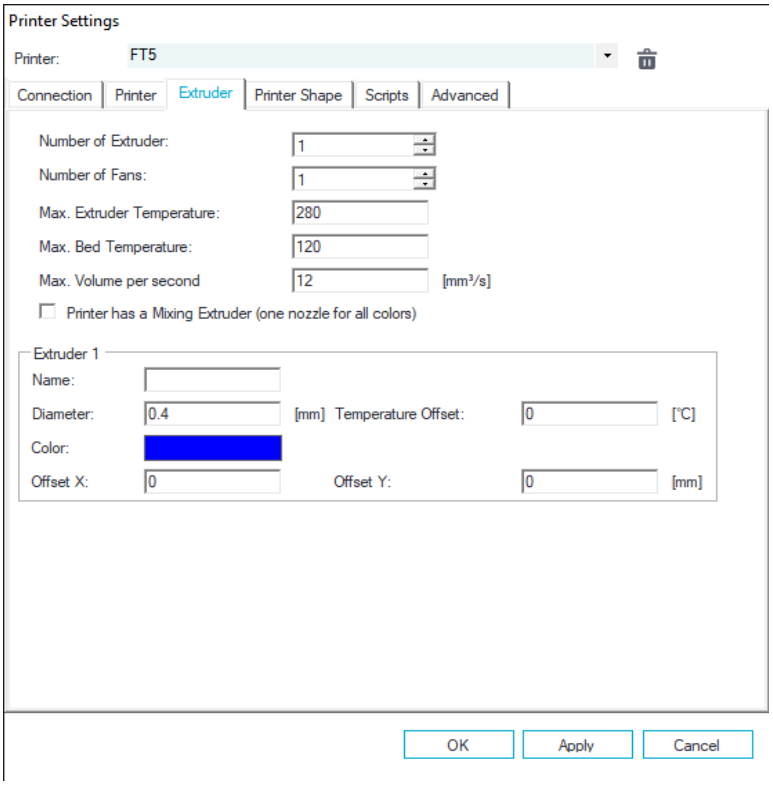


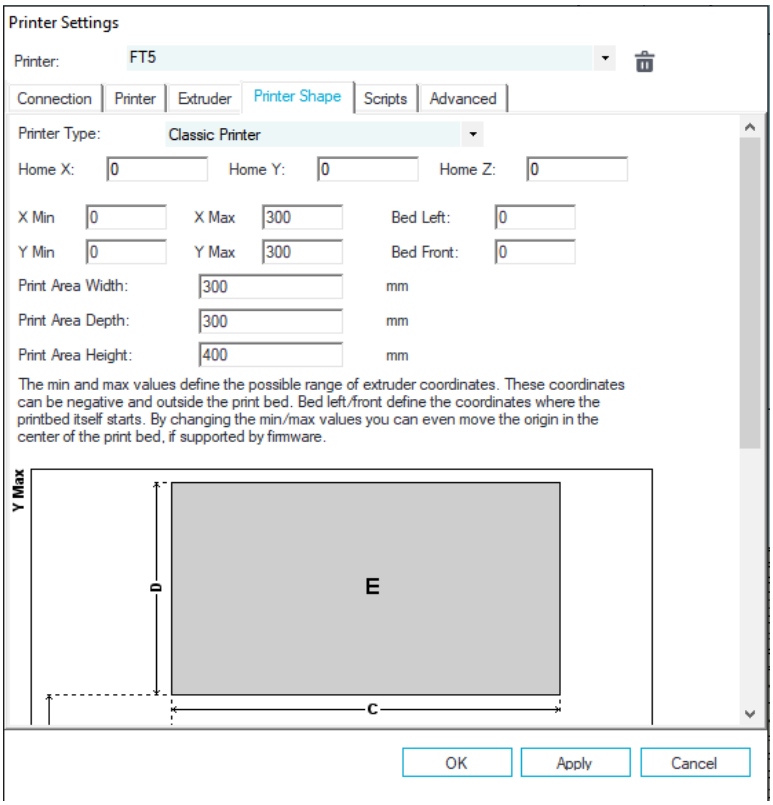
The shape you see in the middle is your build area. By default, this will not be correct for the FT-5.

Up at the top bar, go to Config > Printer Settings, and copy all of the settings in the following screenshots.



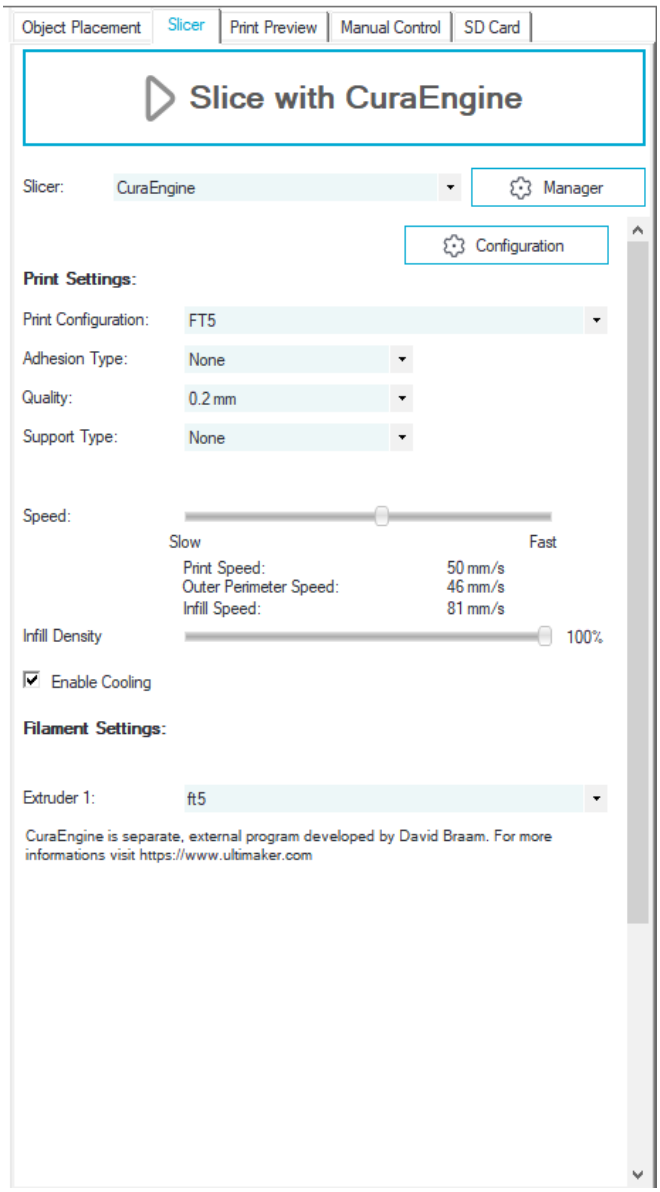
Note that the PORT setting on this page will probably not use the same COM number as what you see here. This varies for everyone, but it should be the same COM port as you used with the Arduino IDE Software.





The remainder of the tabs can be left as they are. Click Apply and OK, and you should now see the correct shape on the screen.

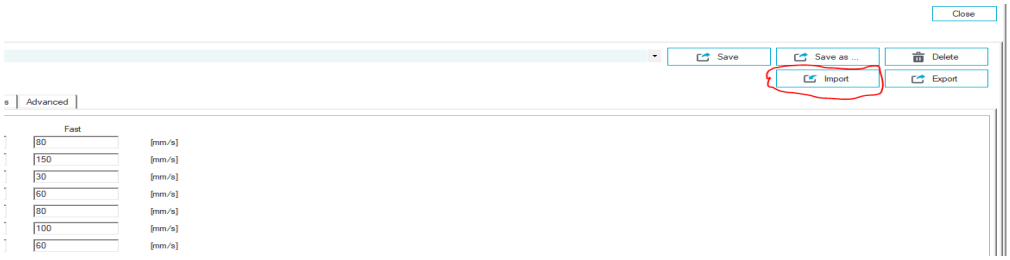
Step 2 - Slicer (CuraEngine) Setup



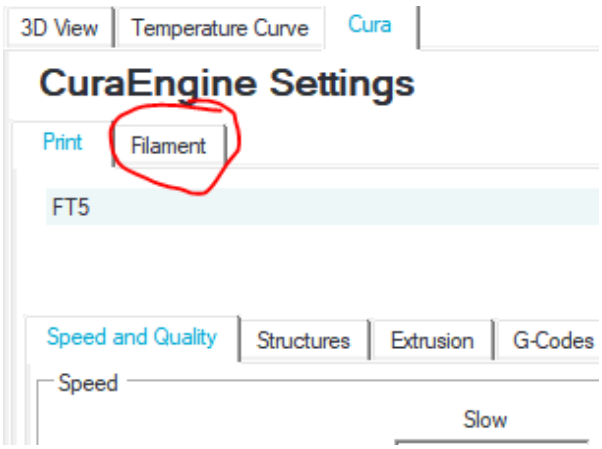
The tabs on the right of the screen are where the majority of your time in Repetier-Host will be spent. Click on the Slicer tab. By default, the Slicer setting will probably be on Slic3r. While it is a decent slicing algorithm, CuraEngine seems to produce better prints, so select it instead.

Click on configuration. If you are new to 3D printing don't worry, as we will make it easier for you with all of the confusing settings. Those with experience can adjust these settings as they see fit.

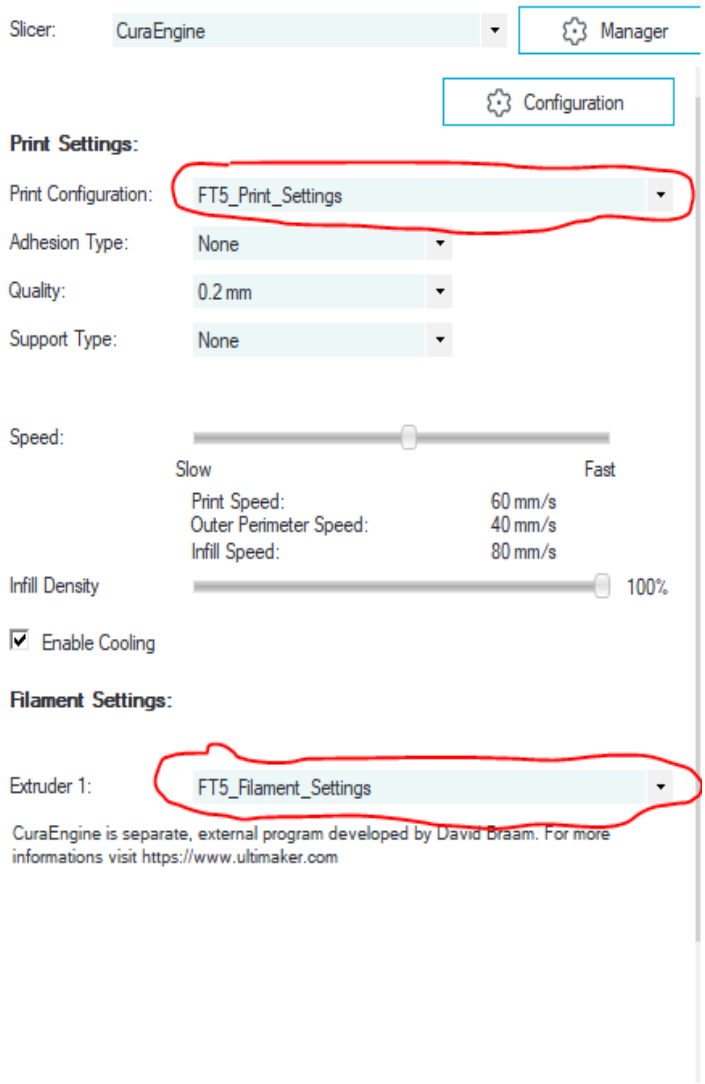
Included in a zip file on the Google Drive will be two preset files. These will be basic settings to get the FT5 going. It is highly recommended that you experiment with settings, research and learn about the settings and what they do, but for now, these presets should get you printing.



Click on import, and import the .RCP file on the Print Settings tab



Move to the Filament tab, and import the .RCF file.



Back over in the Slicer tab, Make sure you select the presets in the Print Configuration and Extruder 1 drop down menus. This will enable the settings to be used when slicing.

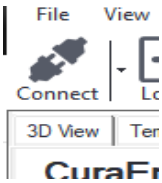
Quality is defaulted to 0.2mm, which is a decent starting point for testing.

The filament settings are set for 1.75mm PLA, at 200c extruder, and 60c heated bed. You can change the temperatures in the Filament tab in the configuration, just be sure to SAVE AS a new name (Add the type of filament to the name, or something descriptive to know what those settings are for).

The speed slider will change the overall speeds between the Slow and Fast settings in the Print settings. The presets have been set fairly conservatively. I recommend a slower setting for your first few prints.

Once these are done, Repetier-Host is now set up, and it is time to connect and calibrate the FT-5.

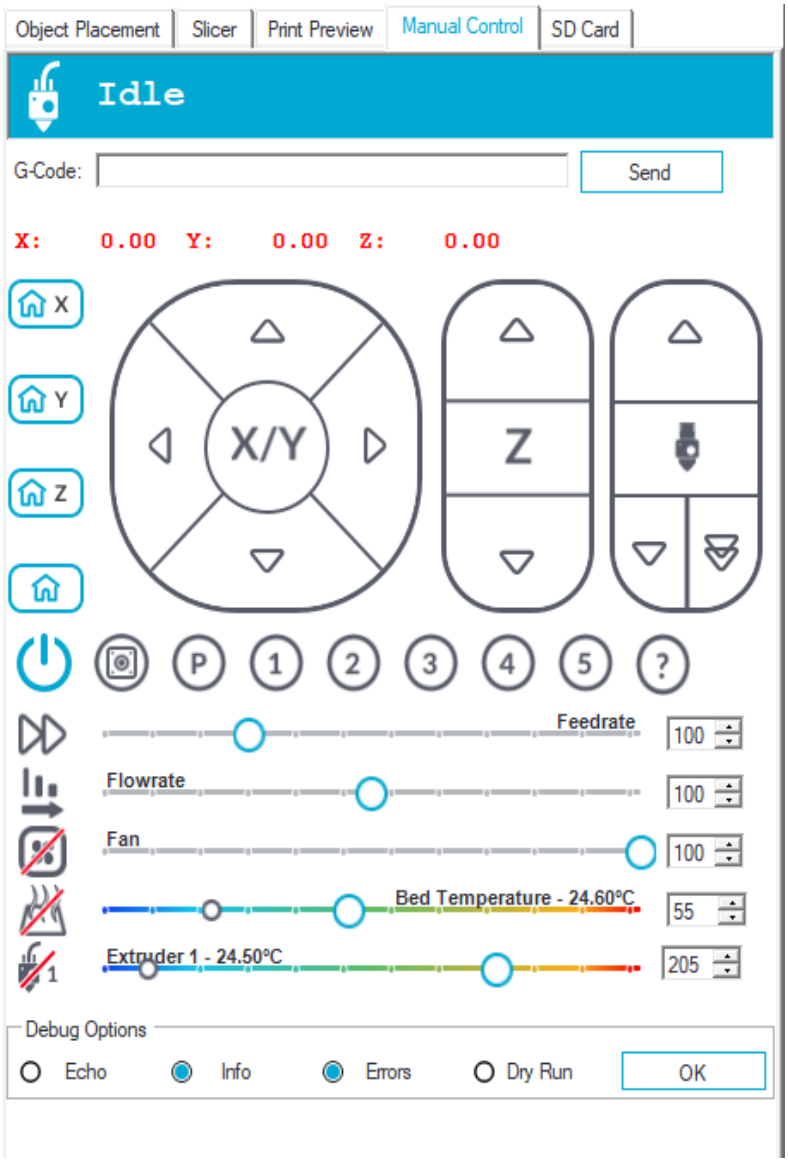
Step 3 - Machine Connection



Your FT-5 should already be connected via USB from the firmware upload process. Make sure you have selected the correct COM port in the settings. In the top left corner, click Connect.

If all goes well, the stars align, and a little luck falls the right way, the button should change to Disconnect.

In the right side panel, go to the Manual Control Tab. It should say Idle in the Status Bar.



The G-Code line is where you can manually send G-Code.

The four House icons on the left are homing commands, individual axes, and Home All respectively.

The XY arrows move the X and Y. Know that due to the orientation of the homed position of the physical printer, moving the printer with the arrows will likely move the carriage opposite of the direction the arrow is pointing. This is normal. Hovering over the arrows will show you different movement distances.

The Z up arrow increases the Z height, so the bed will lower when commanded up and vice versa.

Extruder down arrows will extrude the filament, ONLY WHEN THE HOTEND IS UP TO FULL TEMP.

At this point you should see a room temperature reading for the bed and extruder.

Step 4 - Machine Calibration

Stepper Driver Tuning

At this point, you can turn on the power supply switch. If the USB connection disconnects when the PSU is turned on, check your wiring.

The first step before chucking in some plastic and making a mess...err...creating some awesome things, is to tune the stepper drivers. While this process can be written out, it is much easier to follow this video.
https://www.youtube.com/watch?v=XU6lgFeZ7ZQ

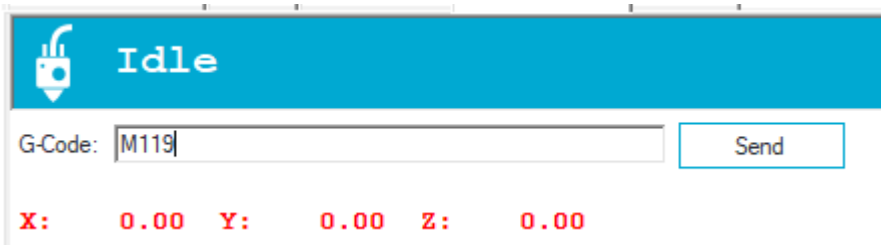
Skim over it, find the important parts, give it a like :)

The main information you need to know other than the process detailed in that video, is to tune the X, Z and E stepper drivers between 0.7-0.9v, and the Y steppers drivers between 1.2-1.4v.

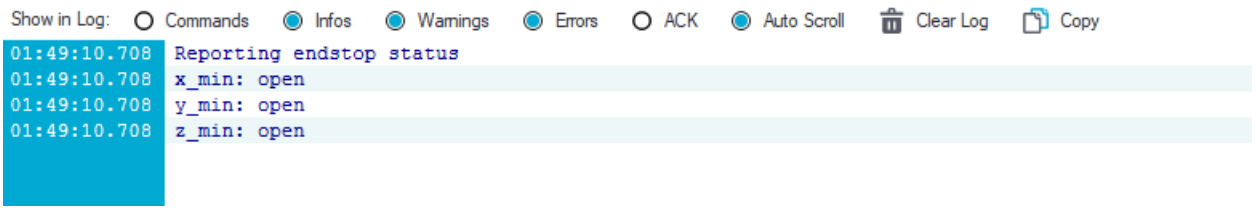
Using the method shown in the video, and the voltages detailed here, your motors should now be getting all the power they need to operate.

Endstop Calibration

Make sure the extruder carriage is not on the X or Y endstops.



In the G-Code box, send command M119



In the console along the bottom of the screen, you should see an endstop status report. If no endstops are triggered physically, all endstops should report as OPEN.

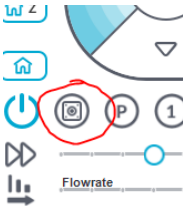
Click on the X HOME and the X should home itself to the endstop. Repeat with the Y axis.

If at anytime an axis begins to move uncontrollably in the wrong direction, or is not stopping when it should, do not be afraid to hit the EMERGENCY STOP button in the top right corner of the screen, or turn off the power switch.

For the Z axis, the endstop position is vital to the machines operation. It is best to position the Z endstop in a spot where you know the bed will not be in contact with the extruder nozzle when the Z is homed to the endstop.

DO NOT TOUCH the endstop circuit board with anything conductive as that can short out the board and damage the controller.

Click the Z HOME button and the Z should home to the endstop. Adjust the Z endstop up to the point that nozzle and bed are separated by a thickness of a sheet of paper, this will take a bit of adjusting and time.



Once the endstop is in the correct position while the X and Y are homed, you can use the DISABLE STEPPERS button to disable the lock on the motors. This will allow you to move the X and Y around by hand.

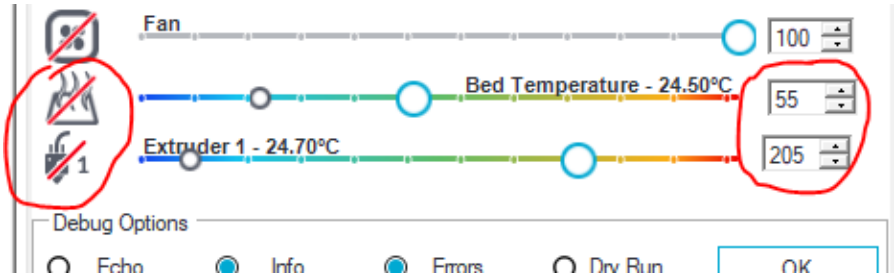
Heat Bed Leveling

You know the Z axis is homed at the XY home, but the rest of the bed may need a bit of adjustment. Move the X and Y around by hand, and adjust the bed up or down using the eight adjustment screws around its perimeter. This may take a bit of time.

When you can move the X and Y around the bed, and the nozzle is approximately the same distance from the bed at all points on the bed, then the bed is sufficiently level compared to the nozzle.

Heater Testing

Note that these temperatures are hot! Do Not Touch the bed or extruder while they are heated, and wait for them to fully cool before touching!



On the right, you can set the temperature, in C, that you want to hit.

On the Left, is the activate icons. Red line means they are off.

Set the Extruder to 200, and click its icon. The extruder should get to 200c fairly quickly and easily. Once it does, click the icon again to disable it.

Set the Bed temperature to 80c, and enable it. The bed should get to that temperature easily and quickly (<5 minutes depending on room temperature).

If the bed has issues getting to temperature, it could be a wiring issue, poor connection, low voltage, or too low of room temperature.

Once it gets to full temperature, disable the bed.

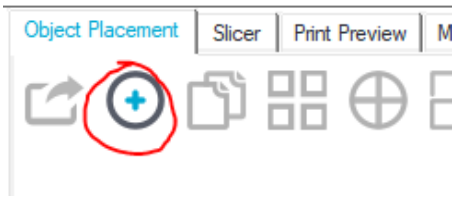
And that's about it… On to your first print!

First Print

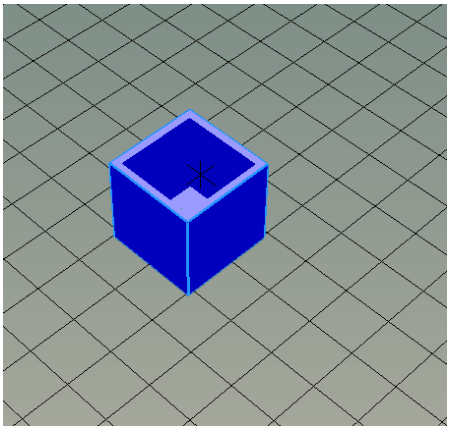
Step 1 - Prepare File

For the sake of simplicity, use the test cube linked below as a first test print.
https://www.thingiverse.com/thing:1008046

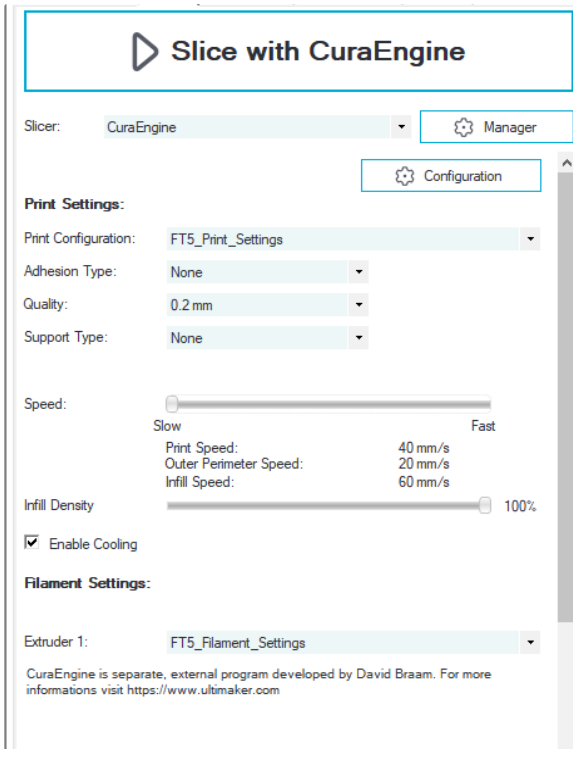
Download the STL file to a location you can remember (a STL or 3D Printing folder is recommended).



On the right panel, click on the Object Placement. Use the Add button to add in the STL file you just downloaded.

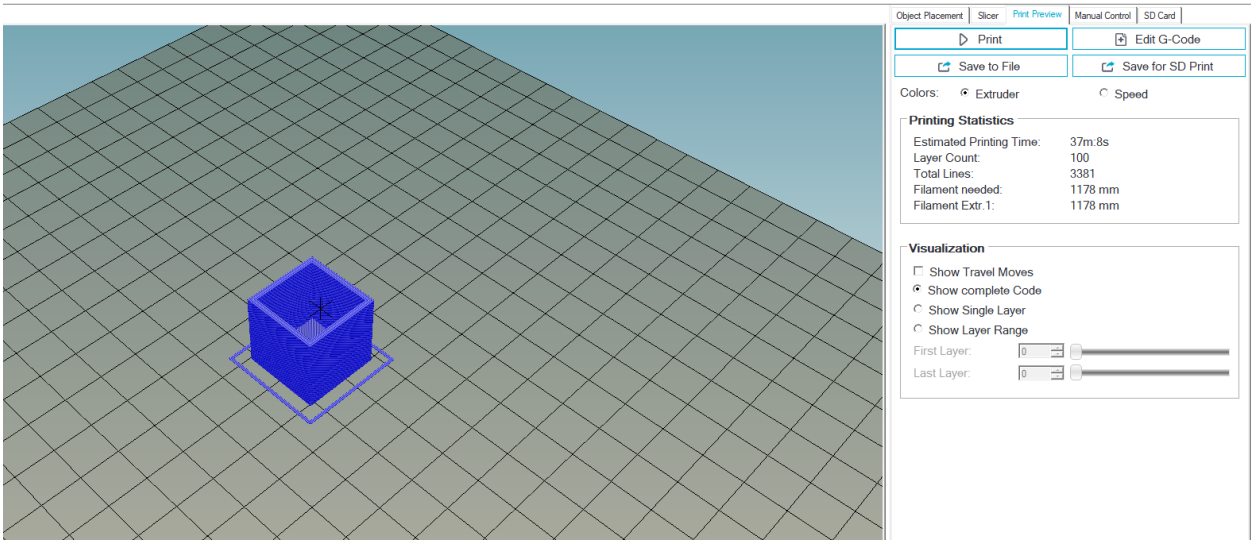


The object now magically appears in the middle of the area. You can use the controls to move, rotate, multiply and scale the object.



Once satisfied with the object placement, move to the slicer tab. Make sure the presets (or your modified settings) are selected, and lower the speed to the slow setting. Adhesion type is not really needed for this model, and no support is needed. Infill is not really important for this model, but you can use 50-100% with not much noticeable difference.

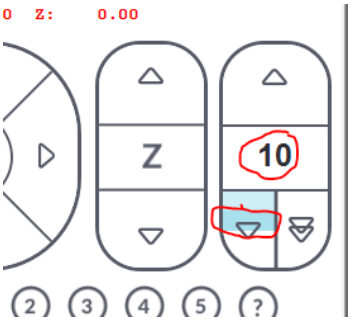
Once the settings are satisfactory, hit SLICE!



Once the slicing is finished (could take some time depending on the complexity of the model and the power of your computer), you will be taken to the Print Preview tab. This will show you estimated times and filament amounts, as well as allow you to look at the model line-by-line to make sure parts will be printed properly. This is very handy for more advanced objects.

Navigate to the Manual Control tab and set the heaters to the temperatures set in the filament settings (200c for extruder, and 60c for the heated bed via the presets), and enable both heaters.

Once the extruder is up to temp, feed in the filament into the top of the extruder. It is easier if you cut a sharp point on the filament end with some cutters. You should be able to feed the filament all the way into the hot end by hand, and begin pushing some filament out.



Once filament is coming out, use the single arrow down extruder button to extrude 10mm of filament. Some filament should extrude out.

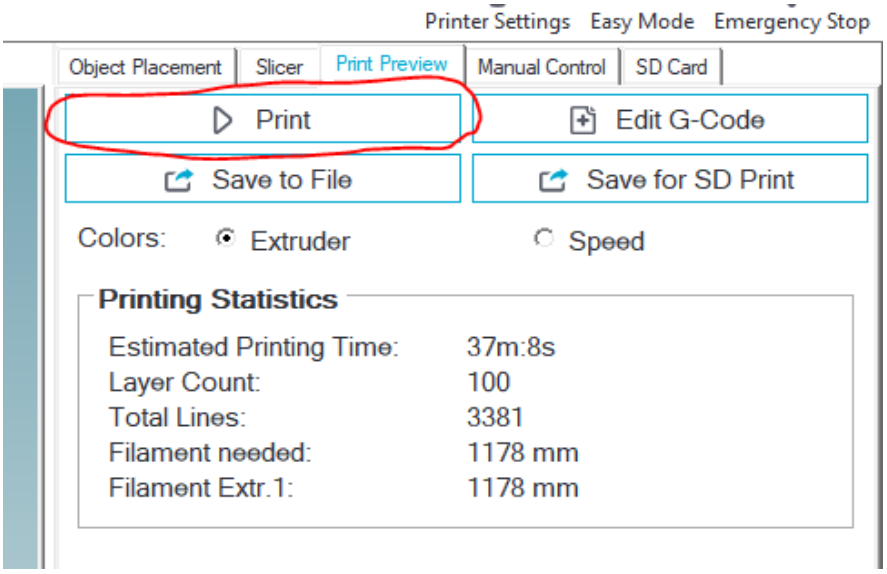
If everything has been successful up to this point, you are ready to print.

Step 2 - Hit Print!

Head back to the Print Preview tab. At this point, both heaters should be up to temperature. Make sure your filament spool holder is unimpeded, the bed is clear of any obstructions, and you are ready to see if the last many hours you have invested in the construction of the FT-5 will pay off on the first try. Remember the location of the Emergency Stop button on Repetier, and do not be afraid of using it or powering off the machine if something is going wrong.

If you are using PLA, then you can print onto the bare bed, but some blue painters tape or a piece of glass as the bed are usually a better print surface. If you use glass, you will need to recalibrate the Z endstop position.

Hit the Print Button.



The printer should first home all of the axes, then move to the middle of the bed, and start printing.

Keep an eye on the first layer of the print, noting the distance of the nozzle from the bed. If the filament is coming out, but not sticking, then either the filament is too far from the bed, and you will either need to adjust the Z endstop up (slightly), or adjust the first layer height in the CuraEngine Configuration Filament settings. It is defaulted to 0.3mm.

If you do not see any filament coming out, and the nozzle looks very close to the bed, or is scraping the bed, then it is too close. You will need to adjust the Z endstop down a small amount. Once you fine tune the Z endstop position, you shouldn't have any sticking issues.

Any adjustment to the Z endstop will require first stopping the print, adjusting the endstop, and restarting the print.

If the filament is sticking to the bed, just watch the print as it goes and look for any abnormalities, odd noises, artifacts in the prints, skips, or any other types of failures.

If everything goes right, you should have a perfect calibration cube when the print is finished.

If that is the case, ***congratulations on completing your first print!***

At this point, you now know enough to start printing other objects. You are now only limited by your imagination.

HAPPY PRINTING!

Community ran resources (unofficial)

Facebook

Forum

Reddit

Thingiverse

Credits

Written and illustrated by Dustin Corbin.
Edited and formatted by Chris Sorrows.