MAKERGEAR

Beginner's Guide to Getting Started with the M2 V4 Rev. E

Congratulations on your purchase! You've made a *fantastic* choice for your 3D printing needs!

If you're new to the world of 3D printing, there is a lot to take in at first, but there are also a lot of resources to help you if you get stuck. This first page tells you how to get to that help. To make the resources immediately available, two of the first things you'll want to do are register your **M2**, and join the forum.

To Register Your Printer:

Send an email with your **Name**, **Address** and the **Serial Number** from the printer to:

Registration@makergear.com

(The serial number is located above the bar code, on a sticker on the back of the machine.)

To Join the MakerGear Users Forum:

http://forum.makergear.com/

The forum is your best place for getting a quick answer. It has a lot of users who've been around the block a few times. (It's a safe bet that you're not the first person to encounter whatever problem you run into.)

Or, if you have no issues to resolve, feel free to post there and just introduce yourself….it's a friendly bunch, and every one of us had to get started the same way. (And you might learn a trick or two.)

(Usual forum rules apply – keep it friendly, no bashing, hate-speech, inflammatory rhetoric, actively promoting other printers, etc. MakerGear has not expressly forbidden it, but you'll run the risk of ticking off a lot of people who think the ***M2*** *is the "bee's knees", and I'll tell you up front, they'll defend it. Final discretion rests with the moderators. If they cause a ruckus, or have the potential to do so, you can expect your posts to start disappearing, so keep it polite.)*

To Contact MakerGear Technical Support:

You can also contact MakerGear directly for support, although your first resource should probably be the forum, since you'll likely get a quicker response from the larger group.

MakerGear tech support is via email….and they are generally very quick to respond.
If you need to contact MG support for assistance with your machine, follow the instructions at the link below:

http://www.makergear.com/pages/support-1

Contents

Tools List

Although you *can* get by without them, the tools below just make it easier to use a 3D printer.

1. **Hairspray/Washable Glue Stick** – Either of these can be used to provide extra adhesion on top of the yellow polyimide tape for sticking the prints to the plate.

2. **Sprue (Wire) Cutters/Tweezer Nose Pliers** - For removing support and cleaning up your prints.

3. **Burr Grinders** (Amazon) - Are also great for removing junk if you have a Dremel tool.

4. **Fine Grade Sand Papers** - For smoothing your finished prints.

5. **Eye Protection** - **Always wear safety glasses or goggles when you are cleaning up a print.** *(It's very easy for a snipped bit to fly up and hit you in the eye, even if you wear regular glasses.)*

6. **Thin Bladed Spatula** - For prying the edge of the completed print off the glass once it has cooled **completely**. *("Cricut" makes a good one.)* (Amazon)

7. **Gloves** - Useful when prying a print off the glass. *(You can cut yourself when the print lets go.)*

8. **Spare Hot End/Nozzle/Glass Plate** – Handy to have because when you need it, it's *immediately*.

9. **Metric Feeler Gauge** set – advanced users can vary the Start Height with them for precise control.

10. **Calipers** – Useful for general calibration purposes and fine-tuning prints.

11. **Optional Bed Surface -** As you become more comfortable with your printer, you might want to try one or more of the bed surfacing options below:

    PEI surface:
    Pros: Releases easily when cool, holds beautifully under heat without adhesive, stays perfectly flat if it is applied correctly, and is easy to use for beginners, inexpensive.
    Cons: Has to be applied to the borosilicate glass with sheet tape. *(Think screen protectors.)*

    Zebra plate:
    Pros: Heavy flexible plate with copper inserts on both sides. Replaces the glass, no application needed. Holds beautifully without adhesive for certain filaments. Excellent release for large prints due to the flexible nature.
    Cons: Can warp over time, or at higher heat. Not the best choice for beginners due to the flex. Cannot be used for tall ABS prints.

    MIC6 tooling plate:
    Pros: Aluminum replacement plate - distributes heat all the way to the edges of the plate for better adhesion on large area prints.
    Cons: Does require adhesive. Plate has to be ordered, cut to size, and the faces bead-blasted (if desired) - you can't buy the correct size off the shelf. There may be extra costs for custom sizing.

*More info on these surface options is in the **Modifications** section of the forum. Any time you change the print surface or plate, you will need to re-set the print **Starting Height** for the bed, using either the **QuickStart App**, or the **Z-Adjust App**.*

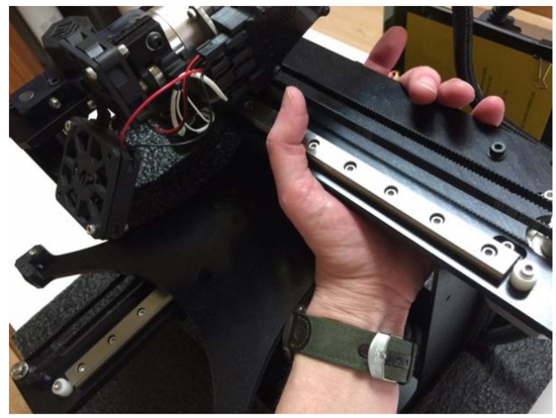
The Quick Start Guide
Single Extruder M2 V4 Rev.E

1. Setting up the Machine:

Choose a solid, stable, desk, table, or workbench to set up your machine.
(These things vibrate when they print.)

Un-Boxing Tips:

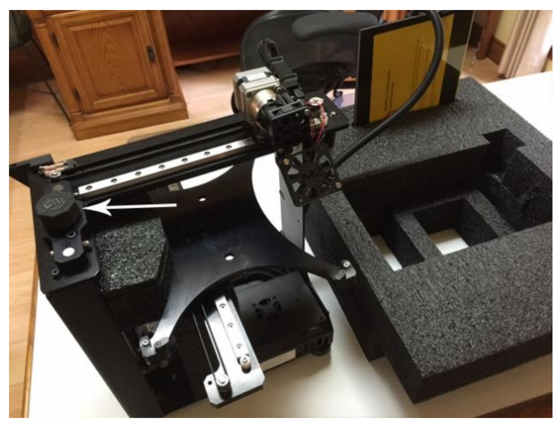
1. Open the box while it's on the floor, and remove the long brown side box first; it makes it easier to remove the M2.



2. Take the small and larger top pieces of stabilizing foam off by lifting carefully up, while the machine is still in the box.

3. Lift the M2 straight up out of the box.
The bottom foam will lift out with it.

4. Remove the bottom foam from the M2. Raise the spider a little by turning the top (Z-axis) knob to the right to make it easier to remove the foam under the Y-axis rail. Then lower it a little to remove the foam to the left of the spider.



5. Place the attached heated bed plate (HBP) on the spider, glass side up, wires to the rear. Make sure the HBP is centered on the little black screws in the four corners. Remove the binder clip and instructions under the plate. Clip the plate down with the attached corner clips.

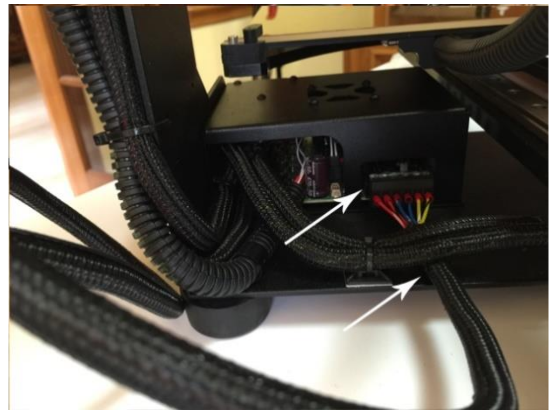
6. You want to turn the clips parallel with the long edges of the plate to avoid bumping the screws on the underside of the X-rail on full bed prints. Do not remove the yellow polyimide tape on the glass.

Open the brown side box and locate the **MakerGear User's Guide**. Check to make sure that everything listed in the guide was included before assembly. (p.4) Finish unpacking the parts.

(Take a moment to read through the entire guide for descriptions of your machine, and Maintenance Instructions, before proceeding with the Set Up.)

Set Up Tips:

1. **Check that the correct voltage level** for your country has been set on the power supply box. www.worldstandards.eu/electricity/plug-voltage-by-country



2. Insert the Power Supply connector into the 6-Pin slot in the rear of the machine. Thread the cable for the power supply underneath the existing wires.

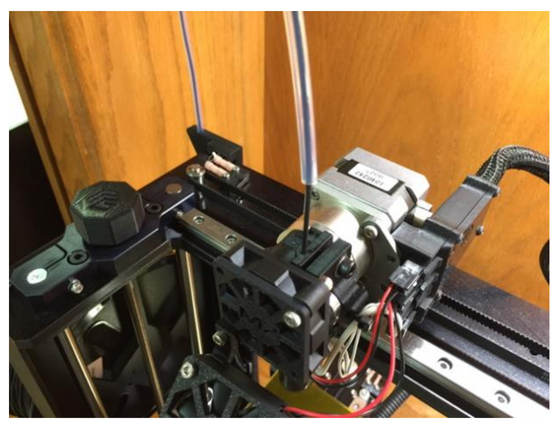
3. Plug in the Power Supply and turn it on. Connect the USB cable.

4. Attach the spool holder with the longer edge up. Put the filament on the holder with the spool unwinding to the rear. Cut a flat edge on the filament end.

5. Thread the filament up through the guide and into the clear filament guide tube, then into the hole on top of the Filament Drive.

6. Now that the machine is prepped you will be using the new **MakerGear QuickStart App** to:

   1. Establish a connection to the **M2**.
   2. Set the **Starting Z Height**, check (and adjust) the bed Level.
   3. Load the filament, heat the bed and extruder.
   4. Once you have set the **Starting Height** and run through the **Leveling Process** – you can also print your first print(s) in PLA.



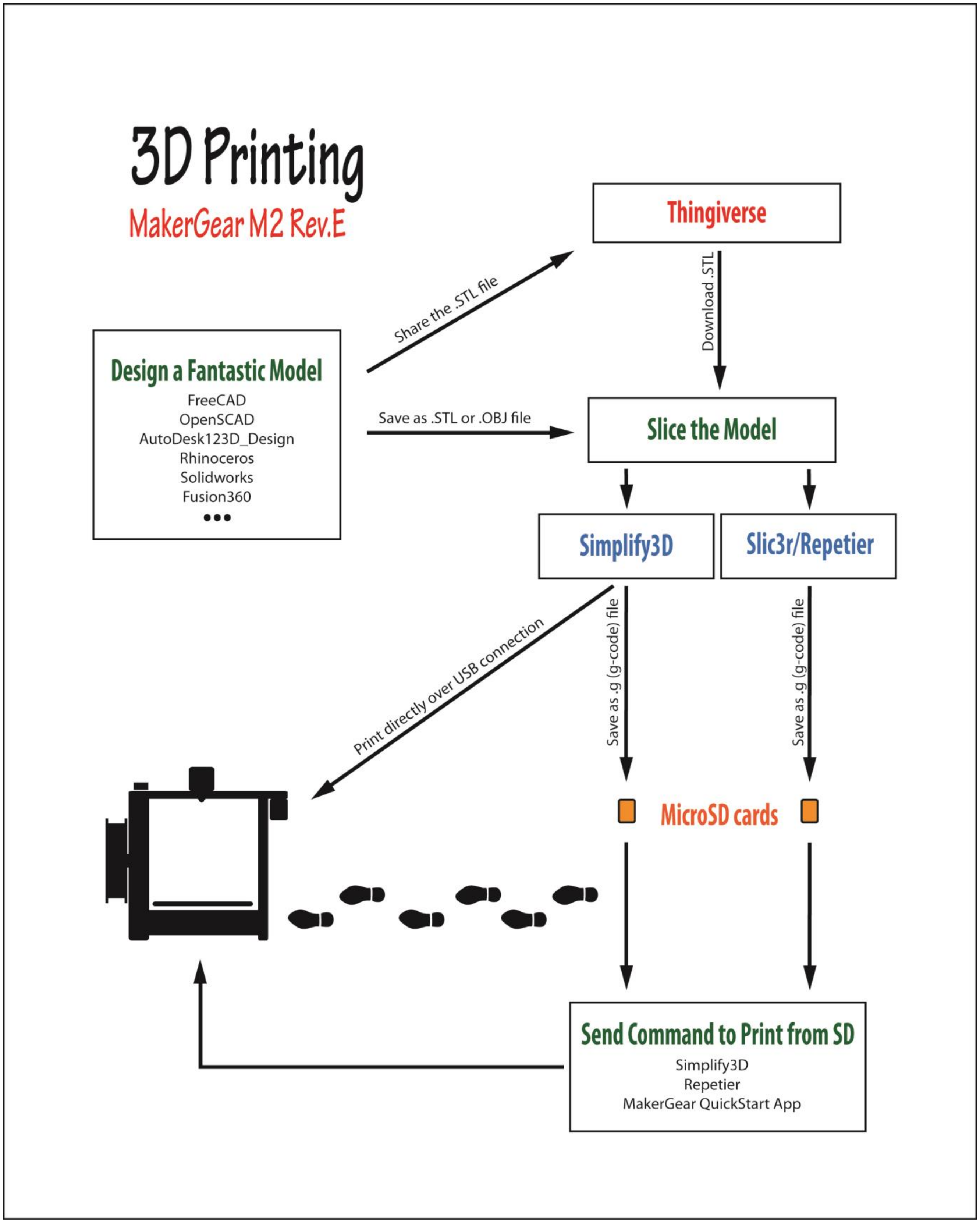
Download the **MakerGear QuickStart Application** for your computer here:

http://www.makergear.com/pages/m2-quick-start-app

After you have printed one (or more) of the samples on the Micro SD card, let the bed cool, and then remove the print from the bed. (***Carefully*** pry up one corner on the print once the bed cools down and it should pop right off.)

2. Moving on to Printing Other Files:

The 3D Printing Process….



The 3D printing process works by taking a 3D model saved in .STL or .OBJ format, and "slicing" it in another program, which converts it into a series of thousands of "**G-Code**" commands that tell the printer when to turn on the fans and the nozzle, how much to extrude, how far to travel before turning, when to stop printing, etc. These commands can be saved in a g-code file (**.g)**, and taken to the printer for printing. Some slicer programs can also send the commands directly to the printer over a USB connection.

The M2 can be used with the (recommended) **Simplify3D** slicer software, or the free open source option of **Slic3r** through a **Repetier** or **PrintRun** interface. For specific usage instructions on any of these 3rd party programs, you'll find the links below helpful, and there are users on the forum familiar with both methods.

Simplify 3D:
Purchase and install: https://www.simplify3d.com/
Setup Guide: https://www.simplify3d.com/support/hardware-setup-guides/makergear/
Tutorials & Guides: https://www.simplify3d.com/support/
http://jinschoi.github.io/simplify3d-docs/

Slic3r:
Download and install: http://slic3r.org/
(note: If you are using ***Slic3r****, load the correct config.ini file for the* ***M2 RevE*** *from your SD card.)*
Tutorials & Guides: http://manual.slic3r.org/

Printrun/Pronterface:
Download and install: http://koti.kapsi.fi/~kliment/printrun/
Setup Instructions: http://makergear.wikidot.com/m2-getting-started

Repetier-Host:
Download, install, guides: http://www.repetier.com/

Also see the guides below if needed:

Appendix A: Starting a Print with Simplify 3D
Appendix B: Starting a Print with Repetier/Slic3r

3. Improving Your Print Quality:

As you become more comfortable with printing in plastic, you will discover that a lot of factors can affect the quality of the final print.

The **M2 Rev.E** printer uses a four point spider, bottom Z-Stop orientation, semi-automatic leveling, and the **MakerGear QuickStart Application** to automatically set a **fixed** *(non-varying)* first layer gap for you, which is maintained no matter what the temperatures are that you are using to print.

(Note: For previous model M2 owners – the new 4-point spider and bottom Z-stop orientation ***are*** *required to support the firmware necessary to affix the gap at a specific size. MakerGear will be offering these as an upgrade as soon as they have an adequate supply. Check the forum once in a while.)*

The **fixed** first layer gap reduces a great deal of opportunity for user error.

Much of what remains deals with the different properties of the filaments themselves. Each plastic reacts differently to factors like cooling, printing and retraction speeds, temperatures, acceleration, etc. Some of them warp madly when printing overhangs, and some cling like a spider-web to everything, including the nozzle, and pull out into annoying threads when making travel movements.

Get comfortable with your results printing one kind of filament **before** moving on to the next.

What You Need to Know:

1. When you switch to printing a different type of filament *(ex: from PLA to PETG)* you need to research the properties of the new filament before trying to print it, and you need to understand the properties of the filament you are switching from, because they are vastly different. (*In all cases*.)

   The forum has a specific section on different **Filaments**, and the successes/problems that people ran into when printing them on the M2. Use the forum as a starting point for new filaments.

   Insta's "Filaments that Work" thread lists settings for several popular filament types: http://forum.makergear.com/viewtopic.php?f=11&t=1951

2. There's usually a fairly broad range of suggested temperatures for the extruder and bed that apply to specific types of filaments. If the manufacturer offers a range, start with the lower *(not the lowest)* end of the range, and work your way up only if you are having trouble keeping the print stuck to the plate throughout the print, or if your layers aren't bonding. *(Lower printing temps have less effect on the physical characteristics of the plastic, and printing too hot can cause heat soak issues on tiny details.)*

3. Do not exceed recommended print temperatures for a filament. *(It can bake the filament onto the nozzle….eventually reducing the inside diameter, and clogging the nozzle.)*
   Check what you have set up in your print profile before starting a print.

4. Don't automatically expect the same results when switching from one kind of filament to another. *(You might need to tweak the settings a bit to optimize them for the new manufacturer, or for a different color.)*

5. When you switch from one **type** of filament (PLA/ABS/PETG/Flex/PC etc.) to another, you will need to adjust the **Filament Drive Screw Tension**. *(Not doing so is a quick trip to a filament jam.)*

Appendix C: Setting the Filament Drive Screw Tension

6. If you switch back and forth between different types of filaments in the same nozzle – **always** use **Cleaning Filament** in between. The **eSun** brand is widely used. It also jams very easily due to its flexible nature – watch the Filament Drive tension. *(It keeps lower temp PLA remnants in the nozzle from being baked into place when you switch to PETG.)*

7. **PLA** is the fastest printing filament – in general, the more flexible it is, the *slower* you have to print it. *(Printing flexible filaments at too high a speed will also jam the hotend.)*

8. Set your **Extrusion Width** to be wider than your nozzle **hole** size, but less than the OD of the flat face on that nozzle. (This gives the machine correct control over placement of the thread.) (Ex: For a **0.35 mm** nozzle, the correct range is from **0.35 mm to 0.50 mm** – so use **0.40 mm**)

9. Set your **Layer Height** according to the following formula :

   Extrusion Width/Layer Height is greater than or equal to **1.8**

   In our example, with a layer height of **0.20 mm** the formula reads 0.40/0.20 = 2.0 which is greater than 1.8. You could also use 0.50/0.25 = 2.0, or 0.36/0.20 = 1.8. The reason for this is to give a *slightly* squashed layer to give better adhesion, and layer bonding. Any ratio that comes out **below 1.8** is going to start causing adhesion and bonding problems.

   (And if math isn't your thing - just set your **layer height** to be **half of your extrusion width**.)

10. Many of the filaments that we print are hydrophilic *(they pick up moisture from the air)* and that makes them get stiff over time and start to break. You'll want to store these in a sealed container or bucket of some kind when you aren't using them, with a desiccant to extend their life.

11. When you change spools, use the **Jog Controls** or **Machine Control Panel** to retract the old filament out before loading the new color. *(100 mm should do it.)* Don't just cut the filament over the filament drive and try to chase it through with another color – they tend to jam if you don't do it a certain way.

    Note: You have to heat the nozzle up to printing temp for that filament before you can use the Jog Controls to retract it. *There's a safety switch built into the firmware. It does* ***not*** *mean the extruder motor has died if it won't retract or extrude.*

12. **Filament Tangles** can be a *real* problem if you let go of the end when loading or unloading the filament. Not necessarily at that moment, but later on, when you're not expecting it. Don't let go of the end of the filament. In addition, dust left on the filament from manufacturing can cause clogs to form in the nozzle, so you'll want to run a **filament dust catcher**. The link below describes both issues:

    http://forum.makergear.com/viewtopic.php?f=11&t=2772&p=17361&hilit=STAPLES#p17361

13. And last, but not least….Just because you got it from **Thingiverse**, it does ***not*** automatically follow that it is a good file for 3D printing. People are generally eager to share what they create, but not all of them follow rules of good 3D design. *(Anything* ***Voronoi*** *comes to mind.)* Look for actual printed results, not 3D renders, so you can see if a design is even feasible without a dual extruder and dissolving support filament. Read the printing instructions. Not everyone tests their prints before posting the files.

Quick Start Guide
Dual Extruder M2 V4 Rev.E

Getting started with the dual extruder system:

Setting up the machine for the dual is very similar to the instructions for the single extruder, so you should read through the instructions for that *first*. Everything works the same, with a sprinkle of exceptions, which I will go over below.

But first……..If this dual extruder is your first 3D printer, I'm going to offer some free advice:

*************   Learn to use **one** extruder before you try to print with two   ********************

When you print with two hot nozzles simultaneously, you are going to have a lot of issues with oozing filament from the inactive nozzle, and will need to print a shield around your object to catch the drips. The slicer software can handle this for you, but you will still see all of the potential problems with your prints that are normal with this kind of manufacturing process, and if you can't see what's going on, you can't stop the print before you waste a lot of time and filament.

The Differences:

1. Obviously, you have two extruders, not one.
   According to the firmware conventions for the **M2 Rev.E**:

   The **left hot end** is **Tool 0**.
   The **right hot end** is **Tool 1**.

   You will use those labels when you set the temperatures and conditions for each extruder in your slicer software. And the difference when using two extruders in a dual is that you have to specify **which tool** you want to work with when you adjust the settings, or use the **Jog Controls**.

2. When you set the **Starting Height**, you will do so using the **left nozzle (Tool 0)**. While you are setting these, you can raise the right nozzle **slightly** in the clamp, so that it does not contact the feeler gauge before the left nozzle does.

   *Only the **left nozzle (Tool 0)** can span the entire bed. The **right nozzle (Tool 1)** cannot reach all the way to the edge of the plate on the left. (It's something to keep in mind when you are deciding which filament you want to extrude out of each nozzle later.)*

3. After you have checked the level and set the **Starting Height** using the **Left Nozzle (Tool 0)**, there is an additional step that you have to take with a dual. You need to set the height of the right nozzle.

Adjusting the Right Nozzle Gap

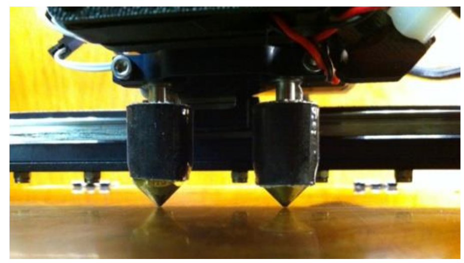
1. Raise the right nozzle slightly in the clamp, without letting it turn. *(The heater block needs to stay perpendicular to the X-rail)*.

2. Clean any drips off of the nozzles if you have extruded through them. *(Your tweezers are good for this while the nozzle is cooling off.)* You'll need clean flat nozzle surfaces to do this.

3. With the extruders and bed ***cold***, slowly raise the plate up until it actually **touches** the **left nozzle. (Nozzle 0.)**

4. Tape the **Z-Knob** into place so that it does not allow the bed to move.

5. Loosen the screw that holds the **right Hot End (1)** in place, and press it down until the right nozzle also contacts the plate.



6. Check the gap from eye level. *(There should be none.)*

7. Tighten the screw.

Starter Profiles for Simplify 3D Rev.E Dual machines

The **Dual Starting Profiles (FFF files)** that have been posted in the forum that work for earlier versions of the **M2** will not work on your machine. The **Rev.E** machines use a bottom-set **Z-Stop** orientation, had firmware changes, require different startup scripts, etc. A separate set of Dual Starting Profiles is posted on the forum specifically designed for use with the **Rev.E** machines.

These are to be used as a starting point only. Expect to tweak them a bit for your ambient conditions.

<u>Tips for Printing PLA</u>

In order to get the best results with a filament, it helps to know a little something about the characteristics of that filament when printing. I'm going to list some tips for printing with **PLA**, but they are *<u>not</u>* going to work the same with other filaments. Each has its own likes and dislikes when printing – you'll need to do some research on the others as you switch to different filaments.

PLA is fairly stiff, and it warps up as it cools. (You might need to use adhesive.)

PLA does not ooze much, you can get by with a **Retraction** setting of **1.0 mm**.
You generally don't need to **Wipe** or **Coast**.

PLA is *<u>terrible</u>* on **Overhangs**; it sags and droops, or warps up without support.

Cooling it off quickly reduces that effect.
(You will frequently see curled droopy overhangs on the sides of a print that face away from the bed fan.)

Either print with **Support** underneath it, hit it with a **LOT** of cooling, or <u>preferably both</u>. I actually added a small desk fan to print rear overhangs with **PLA**, but there is another trick that you can do that works even better. If you remove the **fan guard** on the **<u>bed</u>** fan, it increases the cooling power exponentially. Just be very careful to not let the filament get drawn into the fan when you are loading it – the curl in filament tends to point it right *at* the fan. Also remember to make an adjustment to your printing temp and bed temp if you remove the fan guard – it cools things off by about 10 degrees. If you are printing something with an overhang, choose an **Outline Direction** of **Inside-Out**, otherwise use **Outside-In**.

You can also orient your print on the bed so that the worst overhang areas point directly towards the fan.

You can reduce your **First Layer Speed** to around 60% or so to help with bed adhesion if you need to, and give the filament time to stick to the warm bed.

It's always a good idea to print a **Skirt** around the object to be printed, 2-3 outlines, 1 layer, at a distance of from 2 to 4 mm from your print. It gets the nozzle well primed and can show gap problems before the print starts, giving you time to stop it before you waste a lot of plastic.

Use a 1 layer **Brim** or a **Raft** if you have a tall object with a very small footprint, that otherwise might have trouble sticking to the plate. (Rafts are a pain to remove though, so not recommended unless you *absolutely* need it.)

Temperatures: Start out on the low side of the temperature range with **PLA**. Many people print it successfully with 190°C extruder temp, I use about 205°C but I have the fan guard removed and that cools things a lot. Bed adhesion was good for me at bed temp of 50°C with the borosilicate and tape. If the filament is not sticking or bonding, you can increase the extruder temp up to 220°C safely with **PLA**. Keep in mind though, that when printing small isolated parts, heat soak can become a problem, and you'll see more oozing and surface blob issues at the higher end of the temperature range.

Bridging: **PLA** is ***fantastic*** for **Bridging** (printing over open air between **two level edges**); you generally don't have to make any adjustments to it. I once accidentally bridged a 140 mm gap with **PLA** and it didn't sag a bit. *(Would have been great if I had actually* **wanted** *a bridge there.)*

One final note – it's optional, but you can slow down the prints and get a much better print. With 3D printing, slower **Printing Speed** always results in better quality, and the default settings in **S3D** are **high**. I keep my **Default Printing Speed** for **PLA** at **4200mm/min** (fairly slow). The advantage that it gives is less ringing around surface corners and holes, and a much nicer surface finish. Conversely, you don't want to slow down the **Travel Speed** if you have a filament that oozes badly.

A sprinkle of notes on printing other filaments:

Quick and dirty:

PLA likes to be cooled off quickly. Use lots of fan.
ABS *doesn't*, and will crack and split on tall prints unless an enclosure is used to keep the heat in.

PLA is a low temp filament….it does not make good coasters and coffee mugs.

PLA is essentially odorless when printing, as is **PETG**. **ABS** is not, it is petroleum based, and not safe for food contact.

PETG is a good (*non-splitting*) alternative to **ABS**, but it can be tricky to get set up – see the forum for a detailed write-up on printing it. (Keep in mind when reading descriptions of what other people have done with filaments, for things like adjusting the **Z-Offset**, your machine does not require this step. You have a fixed **Starting Height** and a fixed gap, and you can't change them easily through manipulation of the slicer software.)

PETG, **ABS** and **PC** are good choices for printing spare parts for the machine. (**PLA** is not. It can melt.) There is shrinkage associated with all filaments, some more than others. *(Be prepared to ream out the filament holes in the printed replacement drives with a long 2.0 mm drill bit.)*

FLEX filaments require special treatment due to their tendency to buckle inside the clear guide tube.

Oozing or **Drooling** filaments can usually be improved by lowering the printing temp.

Troubleshooting Tips:

1. Bed Adhesion Issues.

To get a successful print, the first layer has to stick **everywhere** through **the entire print**. But as plastic cools, it shrinks and warps, pulling away from the bed. So you need to come up with something that keeps a print stuck firmly to the plate while it is being printed, but then releases the print when you want to remove it.

There are different methods of keeping the plastic stuck to the plate, and they are different for different filaments, but in general, the use of hairspray or gluestick is common, unless you have purchased a specialty surface that holds without adhesive.

The adhesive needs to be applied fairly thickly, and most beginners tend to naturally skimp on it a little at first.

Just try increasing the adhesive until you get it to hold.

In general, heating the plate slightly helps with bed adhesion, but there's no need to overdo it. With a borosilicate glass plate, and an adhesive of some sort, **PLA** sticks fine in the 40°-50°C range.

Removing the print should be done once the glass plate cools completely – and sometimes it is necessary to freeze the print off of the plate if it is stuck too securely. Let the borosilicate plate cool (off the bed) for a few minutes, then put it into the freezer with the print still attached. After about 15 minutes to an hour (depending on the size of the print), the print will usually pop off without damaging the glass. What is happening is the print is freezing and contracting faster than the glass, breaking the seal. *(Trying to pry the print off while warm can pop a piece out of the glass, so let it cool.)* And the borosilicate is designed to handle temperature swings.

The use of full sheet Kapton or polyimide tape provides a bit of additional protection for the glass. *(It's rare, but sometimes with a large surface area print and high infill, you can still have a piece of glass pop off, even if it's completely and correctly cooled. The forces on the glass are a lot higher than you would expect from something like hairspray.)* You can apply your adhesive of choice directly to the polyimide tape, just like the glass.

Some of the issues you will see with failed bed adhesion are:

a. The corners of your print will start to pull away from the bed – This causes the print to warp up and eventually the nozzle might actually hit the print – knocking the print loose, causing misalignment issues in the print , or even sticking the nozzle into the print.
*(You do **not** want that to happen….It's a **huge** mess to clean up.)*

b. Support structures might get knocked over - and if you don't notice it, you'll wind up with an air-printing situation later. *(Bird's nest - **big** mess also.)*

c. The first layer isn't sticking in certain spots - This can either be because the gap is too large between the nozzle and the bed for the first layer, or because you don't have enough adhesive in that spot. Try refreshing the adhesive, and if that doesn't fix the problem with the next print, adjust your **Starting Height** using the **QuickStart App** or the **Z-Adjust App**.

2. Incorrect Filament Drive Screw Tension.

You want the tension on the filament drive screw to be **just** enough to catch the filament and guide it through to the nozzle….a *lot* of problems are caused by having that screw set too tightly. Once again, different filaments have different properties, and they need a tension adjustment, but not much of one.

PLA is a relatively stiff filament – you want to set the tension so that the "bite-marks" in the extruded end are about 10% to 15% of the diameter. It likes a very loose tension. **PETG** is a softer filament that deforms more easily – you tighten the tension by about a quarter of a turn to switch to **PETG** from **PLA**. Calibrate and reset the tension when you switch between different kinds of filament.

Some of the issues you will see with an incorrect Filament Drive Screw tension are:

a. Jammed and stripped filament – nozzle stops extruding accompanied by thunking or clicking sounds as the stepper motor skips and strips the filament. Once the filament is stripped the thunking stops, but nothing comes out and several layers may skip before you notice it. Stop the print. *(The thunking sounds will not hurt the machine.)* Retract the filament – see if it is stripped, adjust the tension, and redo your print. If the filament is not stripped, it might be jammed. Remove the fans and check around the extruder gear to see if there is a wad of warped filament wrapped up inside.

b. It's very common to think that you have a clogged nozzle or hotend - when really all you have done is stripped the filament. Always check for stripping first.

3. Clogged Nozzle.

The easiest way to get a clogged nozzle is to switch back and forth between **PLA**, which is a low temperature plastic and one of the higher temp plastics like **ABS** or **PETG** without cleaning the remnants of the last filament out first. The higher temps for the **PETG** or **ABS** will cook the leftover **PLA** into carbon on the inside of the nozzle, and eventually it's going to completely clog.

If the remnant is **PETG** or **ABS** in the nozzle, it will not melt completely at the lower **PLA** temperatures, so again, you've got nothing extruding.

Clogs can be cleaned out, but it requires either several hours of soaking in a solvent, or the *careful* use of a propane blow torch if you happen to have one on hand.

To avoid this situation, we do a couple of things. One is to run a **filament dust wiper** on the filament before it goes into the clear plastic guide tube. There are several nice ones on **Thingiverse** that you can print quickly as one of your first projects. Or, you can just wrap a tissue inside a binder clip around the filament before it goes into the guide tube – it works just as well. That catches any dust that gets deposited on the filament during processing and keeps it out of the nozzle.

And for switching between types of filament, you ***always*** want to use some **eSun Cleaner Filament** before switching to a different filament.

(Note: It's really a bear to use, but it's what is available. Be very careful with the rate of extrusion when using it – it has lousy diameter consistency, tends to strip out very easily, and jams like the dickens, so watch the tension.)

What you will see with a clogged nozzle:

a. Nothing comes out of the nozzle, or the diameter of the extruded thread is greatly reduced. – Watch the videos below and check the forum for instructions for safely removing your hot end and disassembling it to clear the clog.

Remove the V4 Hotend: https://www.youtube.com/watch?v=OdducxLlZyk
Remove a V4 nozzle: https://www.youtube.com/watch?v=tyoxU0RbVS0

How to cold clean a clogged V4 style nozzle:
http://forum.makergear.com/viewtopic.php?f=3&t=2942

(Note: Jammed or stripped filament also causes no extrusion, and is the more likely candidate. Check for that first.)

4. Bed Leveling problem.

After using the machine for a while, you might start to see problems that indicate the level needs to be adjusted.

Some of the issues you will see with an un-level bed are:

a. When laying down the first layer, you will see an *extreme* difference - between the printing on the right side of the bed versus printing on the left side, or between the front and the back. And you will see this on all your prints that are large enough to cover a wide enough area.

(For example: the deposited thread in the upper left quadrant might be smashed into a thin transparent ribbon, with the first layer not sticking in the opposite quadrant, and the thread being okay everywhere else.)

Fix the problem by re-leveling the bed using the **MakerGear QuickStart App.**

4. And the final problem – Ugly Prints.

This is more of an aesthetic issue; it isn't something that is going to completely wreck a print. It just means you haven't got the correct settings for that filament yet. Correcting this requires making adjustments to the **Simplify3D** or **Slic3r** settings to make a particular filament do what you want it to.

Best way to do that is:

a. Print some and make adjustments, one at a time, until it does what you want it to. That takes time and patience, but it really ***does*** help you to learn what works and what doesn't.
And once you learn what does what, it makes it much quicker to learn to print a different kind of filament that you've never used before.

b. Use someone else's settings as a starting point (FFF files- Simplify3D). I generally don't like to use someone else's settings file - it doesn't really save much time, because whatever that person has done – there's every likelihood that they are using a different nozzle than you have, or a different print surface that might require slightly different temperatures, or they might have a totally different Z-Home setup, and need to use **Z-Offsets** instead of **Starting Heights**. You still need to walk through every setting and check it.

But for certain things like retraction amounts, coasting, wipe, & bridging settings – it can be helpful to know what other people have had success with. The **Filaments** section of the forum has a lot of useful information on printing just about everything, and that's a good place to start.

Finally: An excellent overall **Troubleshooting** resource is now available on the **Simplify3D** site here:

https://www.simplify3d.com/support/print-quality-troubleshooting/

The Forum is also an excellent place to get input from the experts on your prints. Pictures of the problem will help them to diagnose the issue. *(Try to keep them at or under 800 ppi, or they take forever to load.)*

And be sure to point out that you are using a **Rev.E machine** when you ask for help….the fixes are different.

Appendix A
How to get started with Simplify 3D

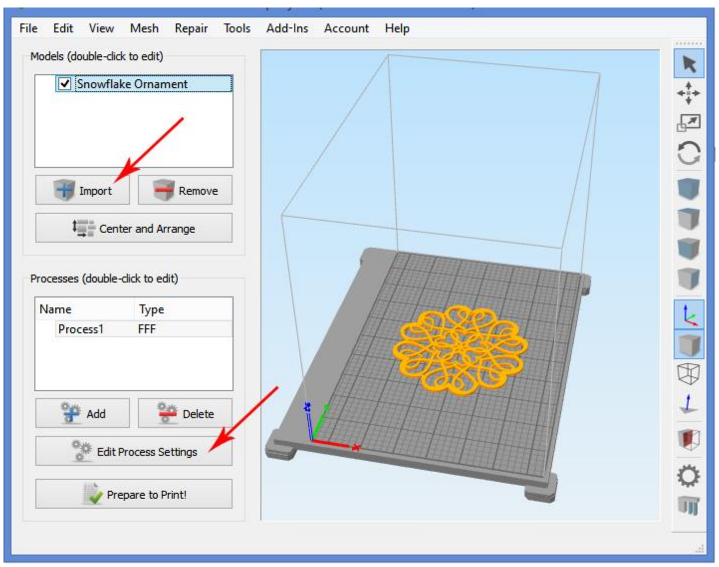
1. After leveling the bed and setting the **Starting Height** with the MakerGear **QuickStart App**, download a file (Thingiverse) or save your own **.STL** format file from your design software.
2. Install and Register the **Simplify 3D** software
3. Turn on the **M2**
4. Open **S3D**. After a moment the software should recognize the printer and connect.

*(If you open **S3D** first, and then turn the printer on, you need to connect manually in the **Machine Control Panel**.)*

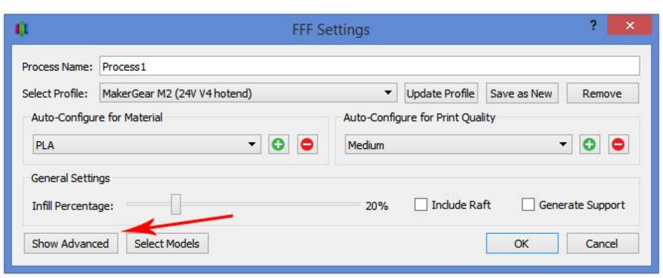
5. Click on **Help** > **Configuration Assistant.**

For the single M2 Rev.E – choose **MakerGear M2 (24V V4 hotend)**
For the dual M2 Rev.E – choose **MakerGear M2 Dual**

1. Import the Model

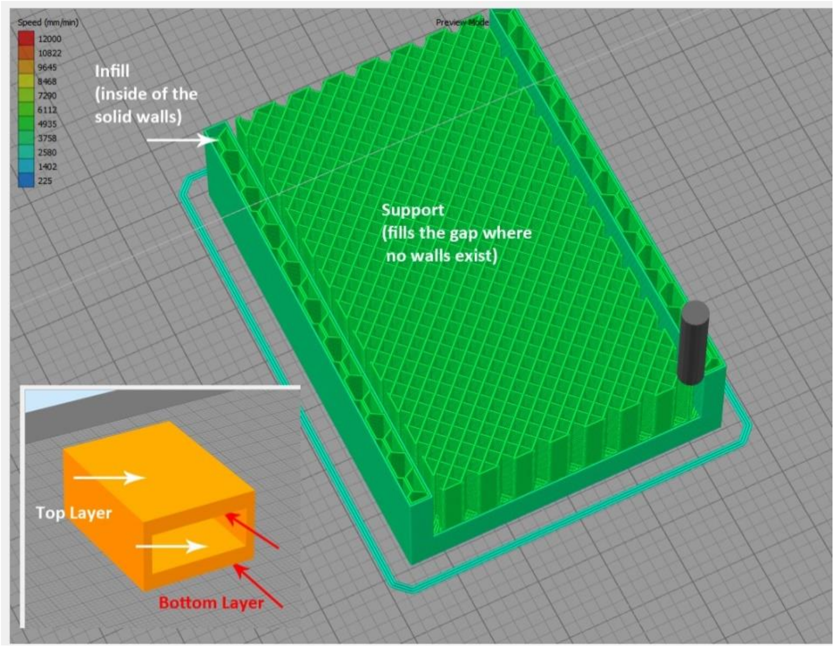


2. Edit the **Process Settings** if desired. (Click on the **Advanced** tab for more options)



The configuration assistant preloads a quick and dirty set of default values for **PLA** – if you want to print something else, you can auto-configure for Material. (Start with a Medium print quality.) You can also change the **Infill** percentage with the slider, and automatically have the software generate support for the model if it is needed by checking **Generate Support**.

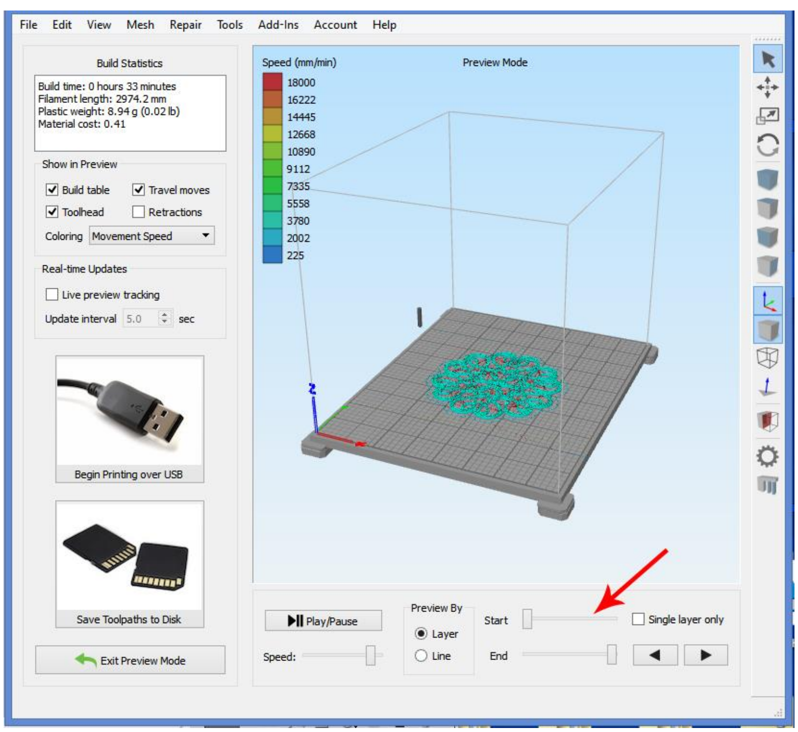
FYI: The difference between Infill and Support:



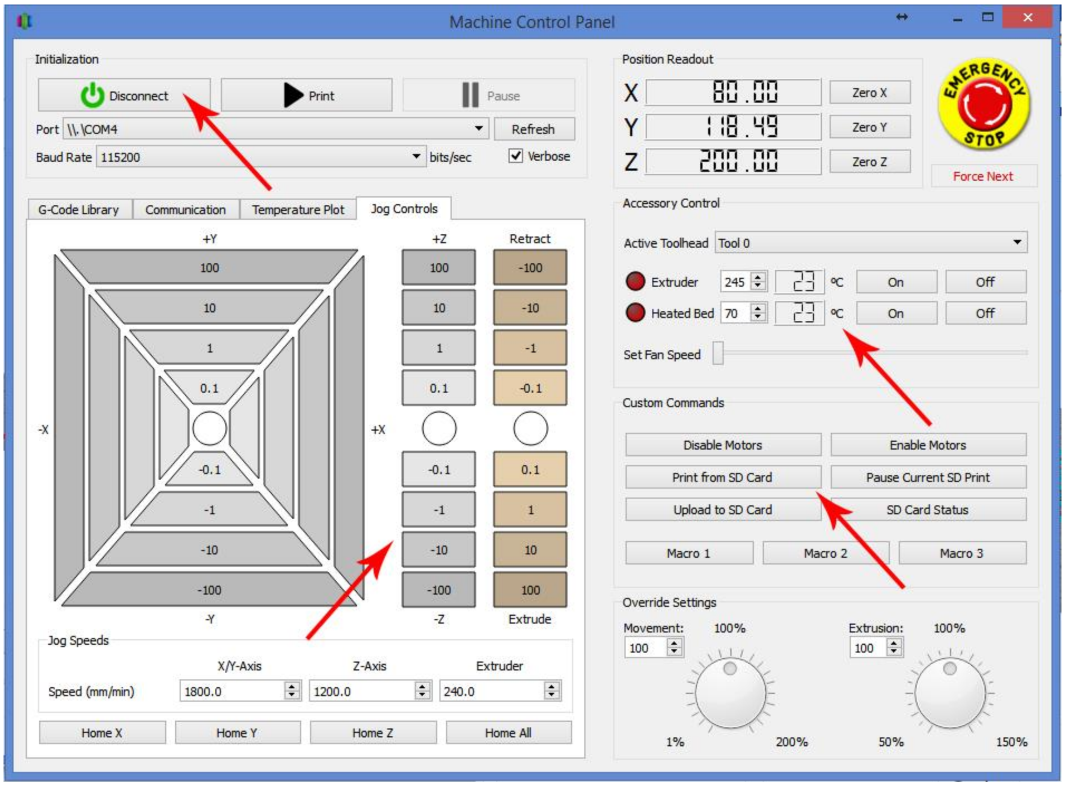
3. Prepare to Print

Click the **Prepare to Print** button for a preview of the print. Use the scrubbers at the bottom to step through the print to see how it is going to go down. Make any changes to support or orientation by backing out of the **Preview Mode** and go back to the previous screen. When ready, you can either **Begin Printing over USB**, or save the **G-Code** file to a **micro SD card**. (**Toolpaths.**)

When you save the file, use all capitals, 8 letters max, no spaces, and give the file an extension of "**.g**"



If you save the **G-code** file to the **SD card**, take it to the printer and then open the **Machine Control Panel** (the little gear icon on the right side) to start the print from the SD card. Once the print has started, you can disconnect the USB cable to the computer and use it somewhere else if desired, the print will continue without being connected. (It's recommended to do lengthy prints from the **SD card**.)



There are other useful tools in the **Machine Control Panel** as shown….**Jog Controls**, the **Temperature Controls** for preheating, and the **Connect/Disconnect** button. *(Obviously for those to work, a connection has to be established with the machine over USB.)*

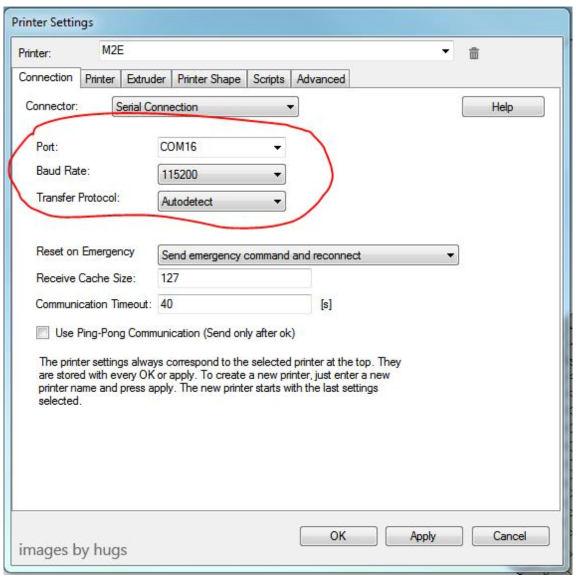
Simplify3D is a very powerful tool – be sure to reference the tutorials in the links section of the **Beginner's Guide**.

Appendix B
Getting Started with Repetier/Slic3r

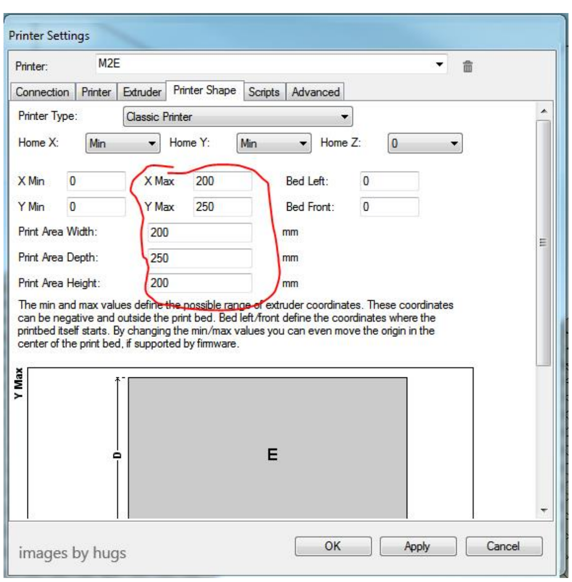
1. Level the bed and setting the **Starting Height** with the MakerGear **QuickStart App.**
2. Download and install **Repetier-Host.** (It comes bundled with **Slic3r** and **Cura**.)
3. Turn on the **M2.**
4. Insert the **MicroSD card** that was supplied with your **M2** into your computer card reader, and copy the **config.ini** file that came on it to your computer.

1. Establish a Connection with the M2

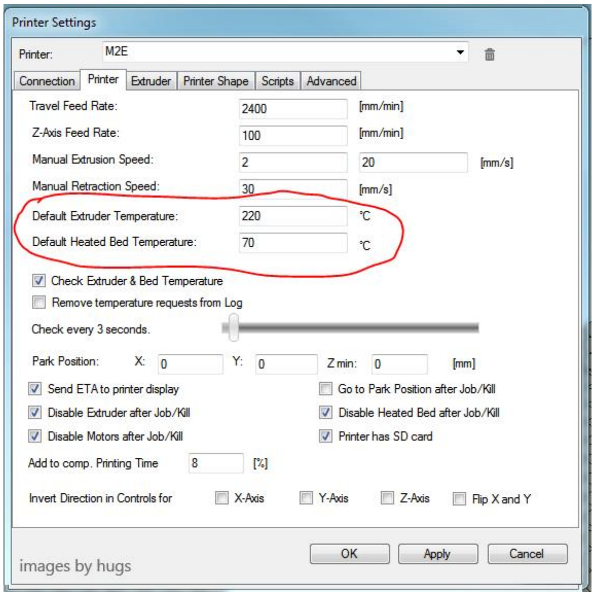
Click on the **Config** button in **Repetier** and choose **Printer Settings**.
Select your **COM Port** from the drop down list and set the **Baud Rate** to **115200**.



2. Define Your Basic Printer Setup

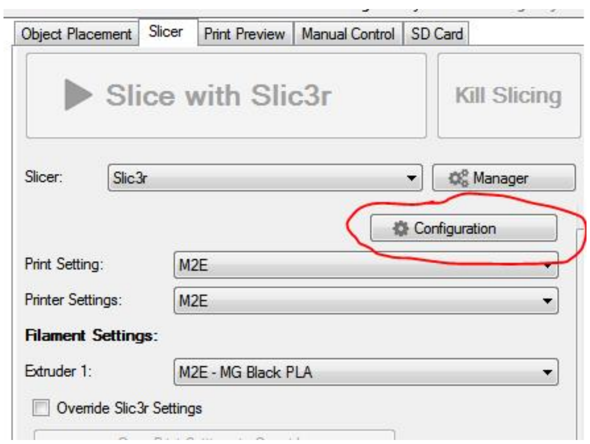


3. Set the default Temps for your Filament



4. Import the Slic3r configuration file

Choose the Slicer tab on the main page, and select **Slic3r** in the dropdown box.
Click the **Configuration** button.
Click **File** > **Load Configuration Bundle** and browse to the **config.ini** file you copied from the card.



Make sure that the desired settings loaded, and continue.

5. Basic Workflow for working with Slic3r/Repetier.

1. Place a model **(.STL)** on the bed. *(Object Placement Tab)*
   Click the plus icon to add an object.
   Right click and drag to shift the objects.
   Rotate and Scale if desired.

2. Slice the model. *(Slicer Tab.)*
   Click "**Slice with Slic3r**".
   The file will process and the tab will switch to **Print Preview**.

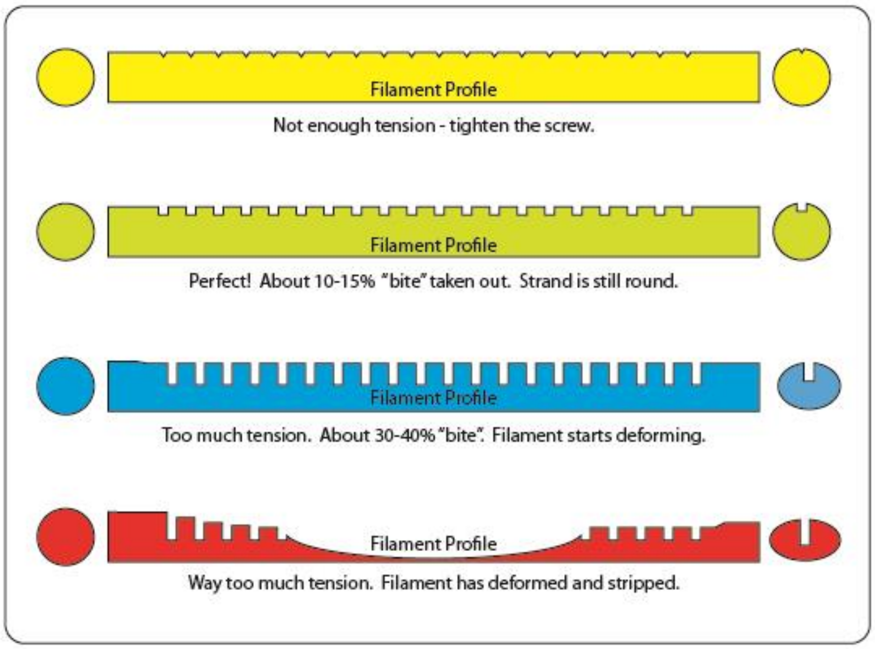
3. Print. (Manual Control Tab)
   If you're happy with how it's going to print, click the "**Connect**" button on the top left.
   Preheat the bed. (Toggle the icon next to "Bed Temperature" so there isn't a line through it.)
   When the bed reaches 60C you can start heating the extruder . (Toggle the icon next to "Extruder 1" so there isn't a line through it).
   When both temps are up, click the "**Print**" or "**Start Print**" button.

   (Process courtesy of Hugs.)

Appendix C:
Set the Tension in the Filament Drive Screws

To forestall extrusion problems, the last step is testing the extrusion and setting the correct tension in the Filament Drive Screw. *(If you have a dual you will set it for both drives.)* The tension should ideally be just firm enough to catch the filament and guide it through, without smashing it in any way, since that will cause problems with jamming and stripping the filament.

Insert filament into the drive, heat that nozzle up to the correct temperature for that filament, and use the jog controls in your slicer to extrude about 100 mm of filament, in batches of 10-20 mm at a time. Make sure the filament is feeding into the drive. (*After about 60 mm have been extruded, you should start to see the filament coming out of the nozzle).* Once that happens, retract the filament completely and examine the end that was pushed into the gear.



Look at the diagram to determine how to adjust the tension on the screw.

Repeat the above steps, adjusting by 1/8th turn of the screw at a time, until your bite marks match the green strand in the diagram. *(Then perform the same process for the other filament drive with a dual).* You can mark the correct tension with a spot of paint in the 12:00 position, so that it's easy to set in the future.

*(Always adjust the tension when switching between different **types** of filament.)*