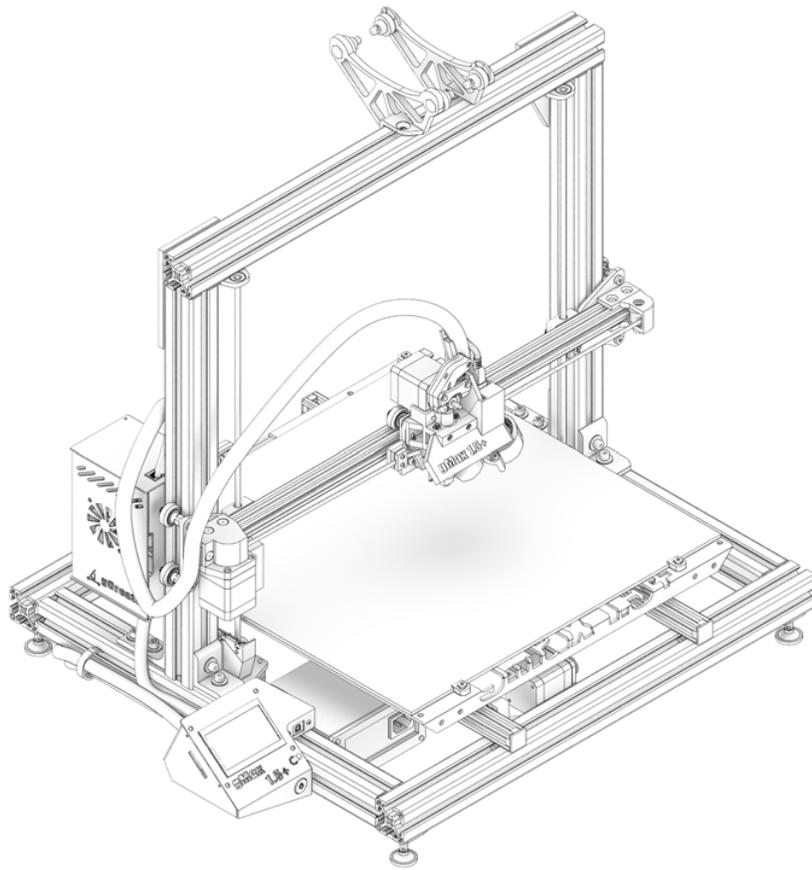


gMax 1.5+ 3D Printer

Getting Started

v170911 (Marlin 1.1.1)



Please review this guide to set up and learn about your gMax Printer. Even if you are familiar with 3d printing, please review this manual.

This guide should be used after you have received the printer and followed the unboxing instructions and can be used in conjunction with our youtube Quick Start video.

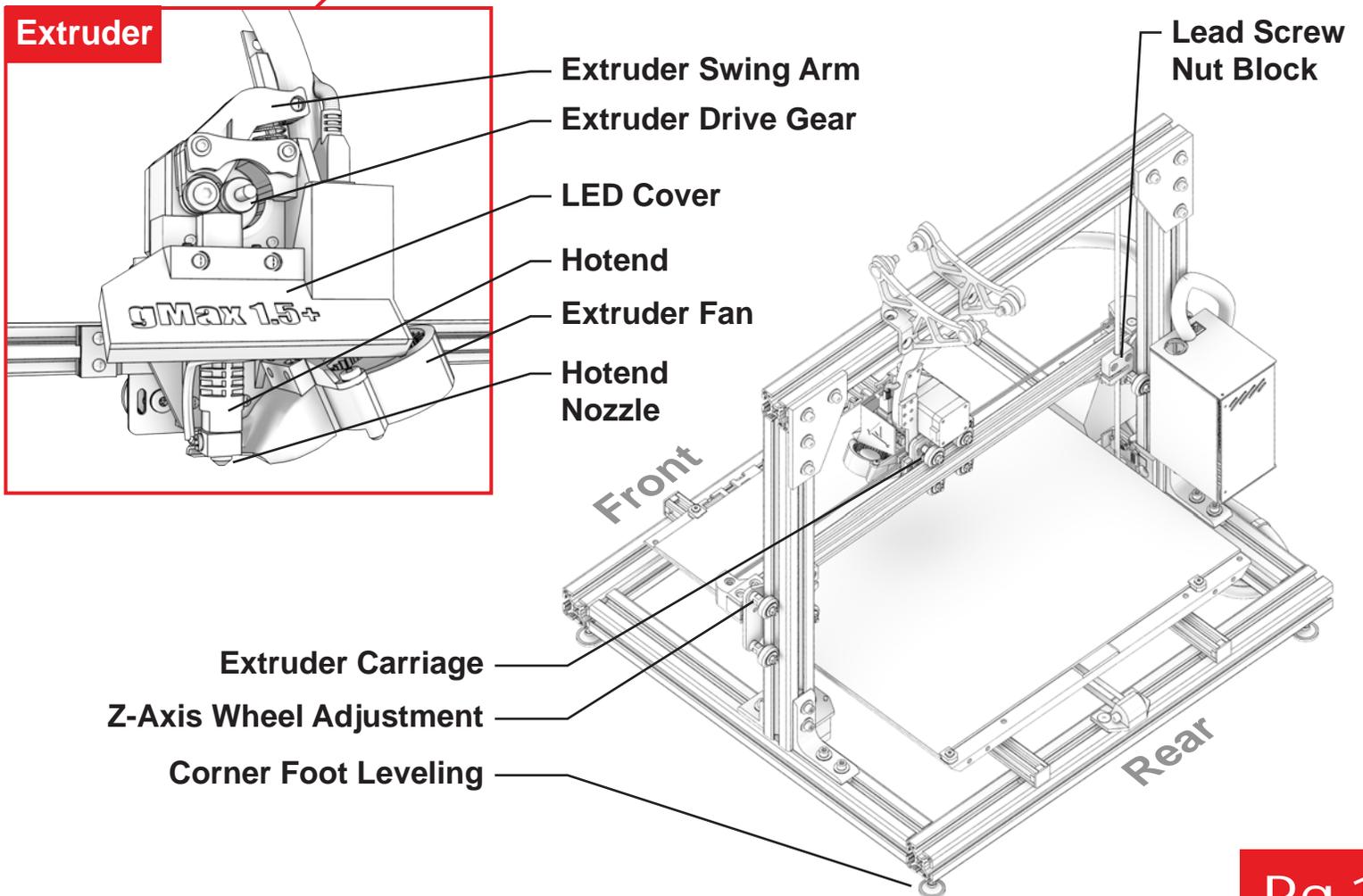
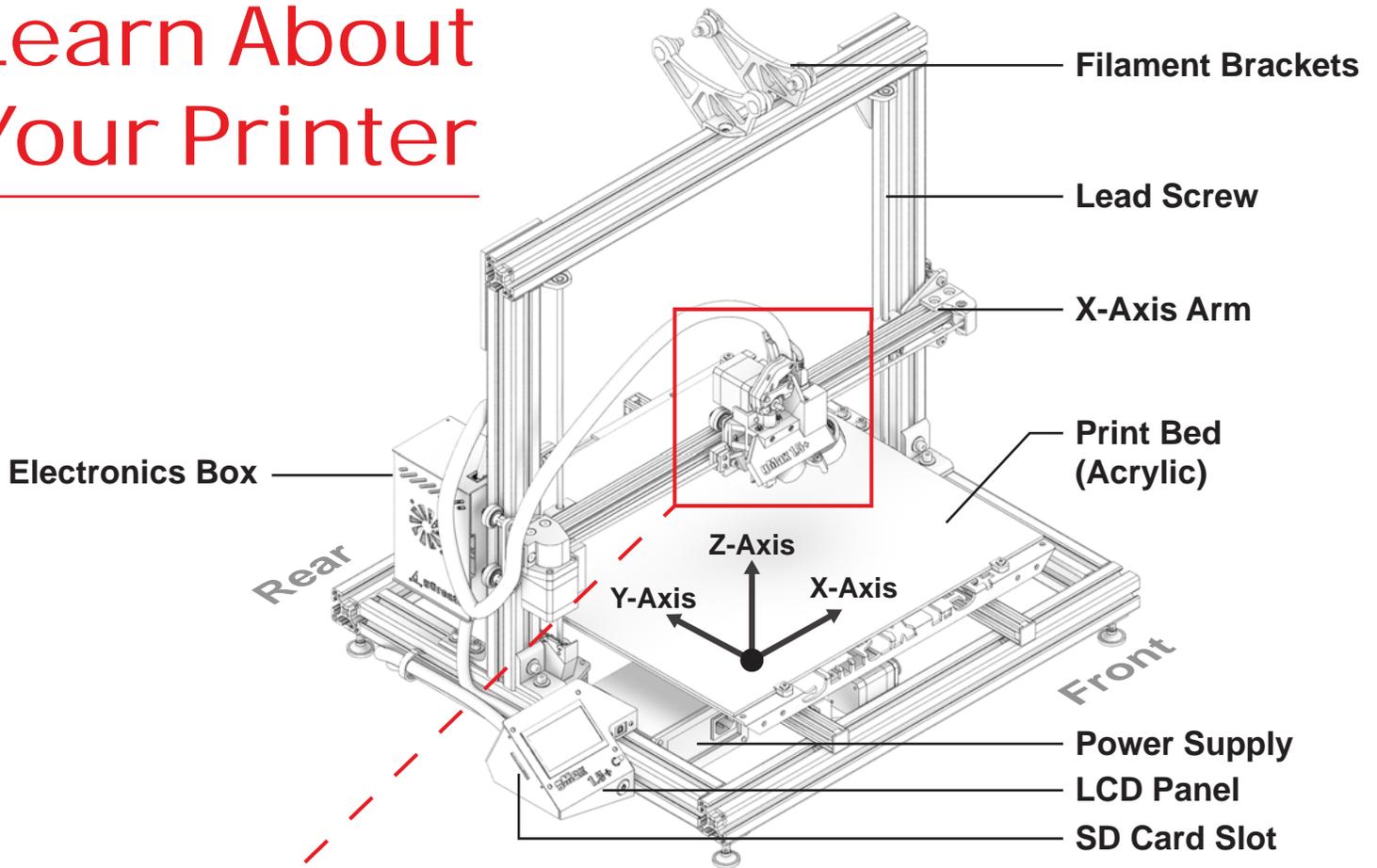
You can find the video at [youtube.com/gCreate3d](https://www.youtube.com/gCreate3d)

Important Note

If you are printing through a USB cable connected to the computer, please **remove** the SD card from the LCD screen before printing.

Not doing so may result in unwanted results after the bed leveling routine.

Learn About Your Printer



Machine Specifications (For Reference)

Max Hotend Temperature (Do not set above)	243C (J-Head) 300C (E3D All-Metal)
Layer Height Range	0.080mm - 0.740mm (Nozzle Dependant)
Filament Diameter	1.75mm
Firmware	Marlin 1.1.1 (gCreate Modified)
Electronics	Ramps v1.4, ATmega 2560
Dual Extruder Offset (If Applicable)	X-Offset: 33.90mm Y-Offset: 0mm
Power	120V - 240 V (If your power supply says full rage, you can plug in 240v)



Note

For best results, set your layer heights in **40 micron** (0.040 mm) increments.
For example 0.16mm, 0.20mm, 0.24mm etc.



Warning

- Set your max temperature **5 degrees below** the numbers above to prevent a **"MAXTEMP"** error.
- This machine includes sensitive wiring and hardware. It should be kept in a well ventillated and dry environment. Humidity can have asverse or negative affects on the product and should be avoided.
- This printer includes parts which can exceed 350C and caution should be taken. Do not let pets or children near the product without supervision.
- Always keep a working fire extinguisher near the printer and have a working smoke alarm near the printer.
- Safety is very important and should be taken into consideration.

Open Accessories Box



gMax Accessories

1. Open the accessory box and remove all the parts. You need the **Filament spool brackets** and the **SD Card** in the next steps.
2. Each printer also comes with a scraper for removing parts from the build plate, a power cord, and an extra bag of hardware for future upgrades.



Filament Brackets



SD Card

Install Filament Brackets

1. Take the filament brackets and secure them to the top rail using the **hex wrench set** provided.
2. Open the supplied spool of filament and place on the filament spool brackets on top of the printer. Adjust the filament brackets to match width of spool and ensure the brackets aren't too tight against the spool. **It must be secure yet unwind very easily.**

Note

Ensure the filament unspools from the bottom and not from the top to prevent the spool from falling off during a print.

Should the filament get tangled, this will prevent the spool from falling.

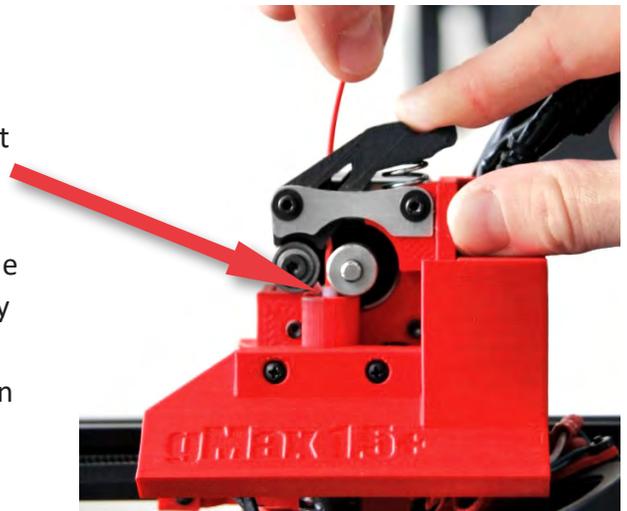


Insert Filament In Extruder



1. Remove the supplied spool of filament from it's box for the initial print, **cut off the first 24"** and **throw it away**.
2. Every time you insert filament, cut end of filament at an angle to produce a clean end and **straighten 4"** of the end of the filament **by gently bending it**. This will make it much easier to insert into the extruder.

3. Pinch the extruder arm above the spring and insert filament in the hole at the top.
4. Slide filament in until you see it enter the top of the clear tube just under the metal drive gear. You may have to twist the filament as you push it in. The filament should slide in easily all the way down about 4" total.

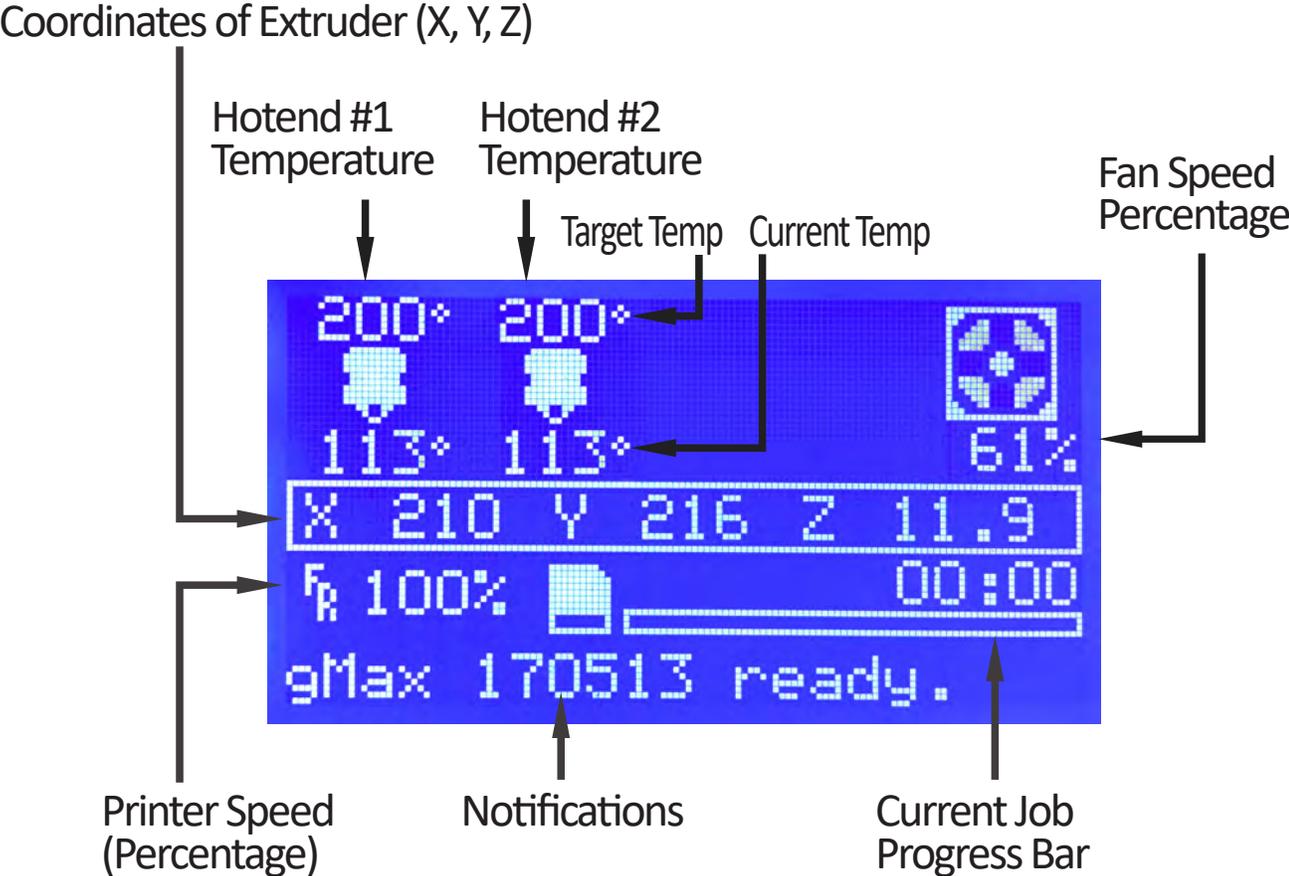
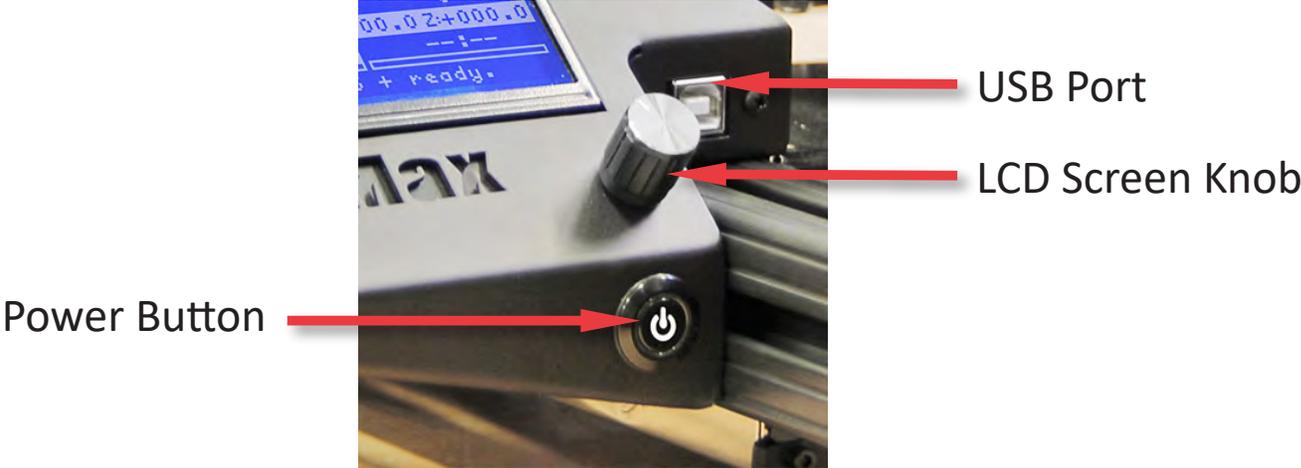


Plug In and Turn On



1. Plug in the printer to a surge protected power strip.
2. Make sure power switch is on.

LCD Screen Breakdown



gMax Initial Run (Required)

1. You **must run the supplied gcode file** to test your gMax printer after transport.
2. Slide the SD card (**upside-down**) into the left side of the LCD screen.
3. On the LCD screen, the bottom line should now say **“Card Inserted”**. If not, try removing the SD card and reinserting it and make sure the **“lock”** is in the off position on the side of the card.
4. Push in the LCD screen knob (**this is your accept button**) and scroll down to **“Print from SD”** and push in the knob again.
5. Under **“gCreate Files”** Select **“gMax Initial Setup”**. Click **to start test**, then follow the instructions on the LCD screen and **click the knob when prompted between tests**. The test file will now perform the following proce-



- X, Y, Z Axis test (slow moves, then fast moves)
- Bed Probe test (center and full grid)

If the bed probe doesnt drop/raise, the extruder only moves up ,or the probe is blinking during this test, contact support.

- Extrusion test (with heat)
- Fan test then cooldown



Note

The gMax “Initial Run Test File” should take about 15 minutes to complete and will wait for you to “Click to proceed” between tests.

If the printer fails during any of these tests contact support for solutions.

Your First Print

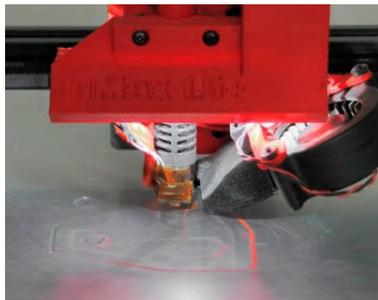
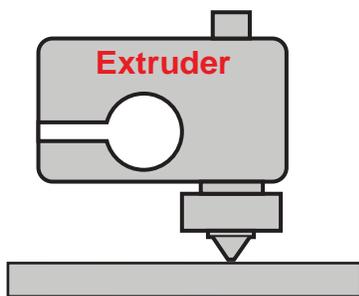
1. This first print will confirm that your gMax is running the same as when it left the manufacturing facility. **This is a necessary and useful print.** The result is a printed calibration object used in the future if you need to calibrate your extruder height.
2. Under “gCreate Files” Select “gMax First Print”.
3. As the print begins, click the knob twice quickly to access “Babystep Z”. You can turn the knob to raise or lower the 1st layer
4. **You should use the babystep function at the start of each print to ensure the perfect 1st layer.** Babystep moves the z-axis in very small increments which you can see as you turn. Keep note of the number and adjust the 1st layer as needed to achieve an even flow of filament referring to the diagram below.



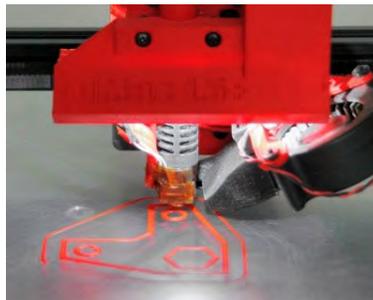
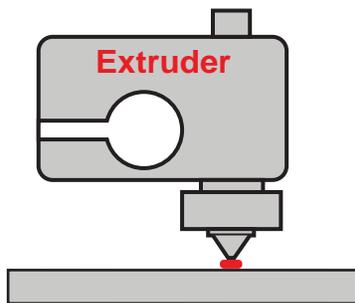
Note

When Babysteping up or down:

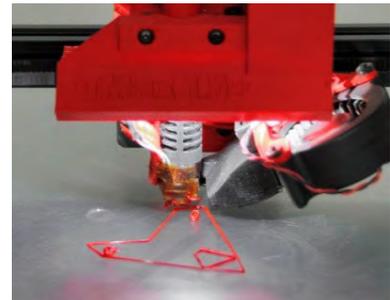
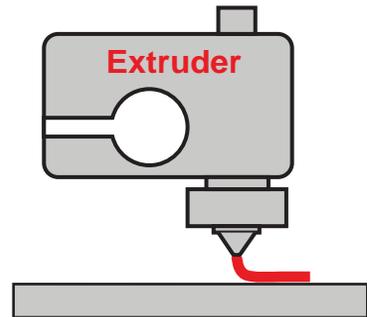
Turn **CLOCKWISE** to raise
and **COUNTERCLOCKWISE** to lower



Extruder is too low
and digging into bed



Extruder is perfect



Extruder is too high
and printing in air

Removing the Print. Strength vs Finesse

1. After the print finishes you can immediately remove it if the bed is **acrylic**. If using a **heated bed**, unplug the bed (or lower the temperature) and let it cool several minutes before removing.
2. If the first layer printed correctly you should see no curling at the edges and the print should be easy to remove with the supplied scraper.
3. **USING CAUTION**, take the supplied scraper and gently tap at the edges of the print. Look for corners of the model and tap them with the scraper flat on the bed. After several taps the print should pop off the bed. If the print has a large surface area on the print bed it may be harder to remove and you may have to slide the scraper below the model.



Warning

The scraper is very sharp. **NEVER** place your hand behind the area you are scraping. Taking a few extra seconds to remove the model is much better than a trip to the hospital.

Adjusting the Bed Level Sensor Offset

1. The bed level sensor should sit 1.9mm above the hotend tip. Adjusting this offset will affect how high or low the nozzle prints the first layer and it may need to be adjusted over time.
2. To adjust the bltouch offset, you first need a distance to add or subtract. If you use Babystep-Z when starting a print, a distance will appear as you turn the knob. Additionally you can “**Prepare > Auto Home**” the printer. After auto homing lower the nozzle using “**Prepare > Move Axis > Move Z > Move 0.1mm**” until the nozzle just barely touches the bed and take note of the z-axis height.
3. Using either of these two methods, will result in a distance that must be added to the bltouch offset. Go to “**Control > Motion > Z Offset**”.
4. Use the current number for the z offset and add the number you recorded earlier.
For example $(-1.9) + (-0.12) = -2.02$
5. To save these settings go to “**Control > Store Memory**”. Now the height will be saved for the start of the next print. As always, use babystep z at the start of a print (**Push the knob twice to access the function at the start of a print**).

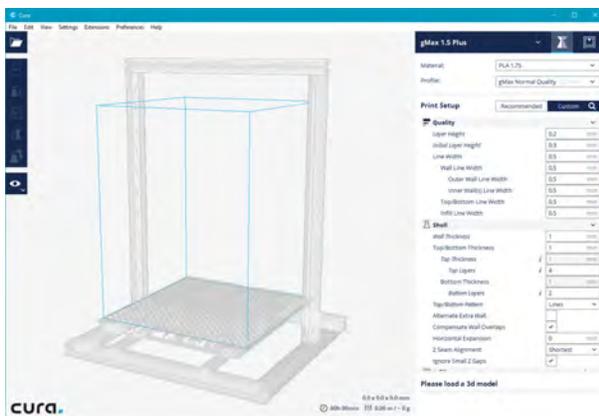
Slicing

Slicing is one of the most important parts of the 3d printing pipeline. The proper settings can result in a great print however many settings are usually model-specific. Taking the time to learn the main settings can result in a better printing experience.

1. For slicing, we recommend **Cura**, **Simplify3d**, or **Slic3r**. View our youtube channel [youtube.com/gcreate3d](https://www.youtube.com/gcreate3d) , forum.gcreate.com or the slicing program websites for more information.

Use the configuration files on the SD card or download them directly from our forum for the latest versions.

Note: Simplify3D may have older versions of our files which do not have the bed leveling code.



Cura (Free)



Simplify3D

Note

Slicing Tips:

- Typically there are several settings you will change depending on your 3d model and filament type. The majority of print quality issues can be fixed by changing slicing settings.

Use our slicing configuration files with the correct start gcode. You must include the following start gcode to use the bed leveling function:

```
G28 X0 Y0 ; Home X and Y axis  
G28 ; Home all axis  
G29 ; Run bed leveling sequence
```

- Print speeds, layer height, exterior wall thickness (or perimeter count), infill density, support material settings, hotend temperature and fan cooling are the typical settings to modify. Refer to the slicing program websites for many useful tutorials and slicing tips.

- Using high quality filament to reduce print quality issues.

Tips While You Print

1. Always keep an eye on your print and check on it often . If any part of the print fails this may destroy the rest of the print and possibly damage the printer.

2. You can adjust the speed of your print in real time by simply turning the knob on the LCD screen. This is very useful if you need to slow down the printer at a difficult area or you notice the print isn't cooling enough.



3. You can use the Tune menu to adjust fan speed, hotend temperature and filament flow during a print. Note **these settings are not permanent** and they will revert to standard settings when the printer reaches a new gcode command or is turned off.

4. The fan will not operate below 30% (or 76 from the LCD menu) since the power is too low and you should not set your fan over 85% or it may cool the hotend.

5. • You can easily change filament during a print. Simply go to **“Tune > Change Filament”**. This command can also be sent by your host software or you can put it in your gcode file manually using M600 where you want the change to happen.

- When activated the print will pause, the extruder will raise and move out of the way and it will retract the filament. After 45 seconds the extruder will turn off for safety.

- Then slide in your new filament and push the knob to resume.

6. Always plug your printer and heated bed into a UPC power backup or surge protector.

7. Sanding the acrylic bed is recommended after every print or before an important print. Use the supplied sanding sponge and wet with water to reduce dust. When done, wipe the acrylic clean with a paper towel.



few

8. When the nozzle is hot, carefully use a paper towel or needle nose pliers, to remove any excess plastic which may have built up. Be careful not to touch the very hot nozzle.

9. Do not use any lubricant on the lead screws, instead clean them off periodically with acetone.

Bed Leveling

IMPORTANT: In order to properly use the auto bed leveling function, you must use our slicing configuration files. These files have special start gcode functions to run the bed leveling operation. **Bed leveling directly from the LCD menu is not saved and is only for testing purposes.**

1. For proper bed leveling, ensure your slicing program has the correct “start gcode” with the following lines of code:

G28 X0 Y0 ; Home X and Y axis

G28 ; Home all axis

G29 ; Run bed leveling sequence

2. Depending on your slicing software, the “start gcode” section may be in different areas. Using our config files ensures proper setup.



Note

Download the latest slicing configuration files from our forum.

Leveling X-Axis Arm

Leveling your x-axis arm will allow the x-axis carriage to raise and lower smoothly and prevent binding of the lead screws. The arm should be level as described below however the bed leveling feature of the printer will automatically account for bed leveling.

1. To level the X-Axis arm go to “**Prepare > Auto Home**”. When it is done, turn off the printer. Then use the calibration object from the previous print to manually level the x-axis arm by turning the lead screws, by hand, until the object barely slides under each side of the x-axis arm.

