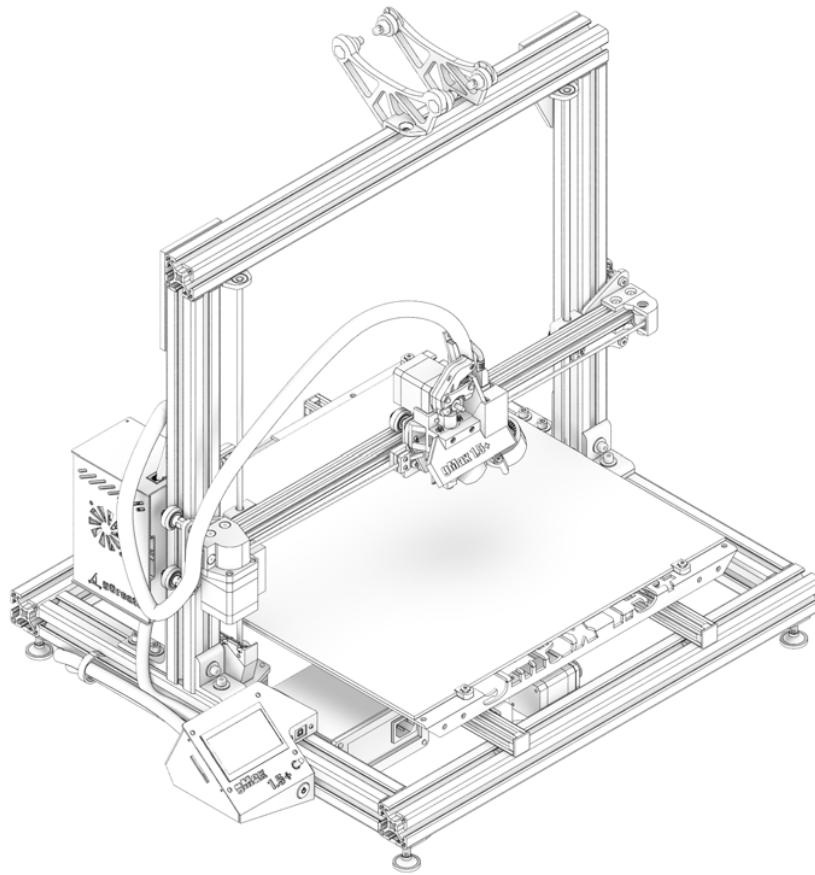


gMax 1.5+ 3D Printer

Getting Started

v181216 (Marlin 1.1.9 Firmware)

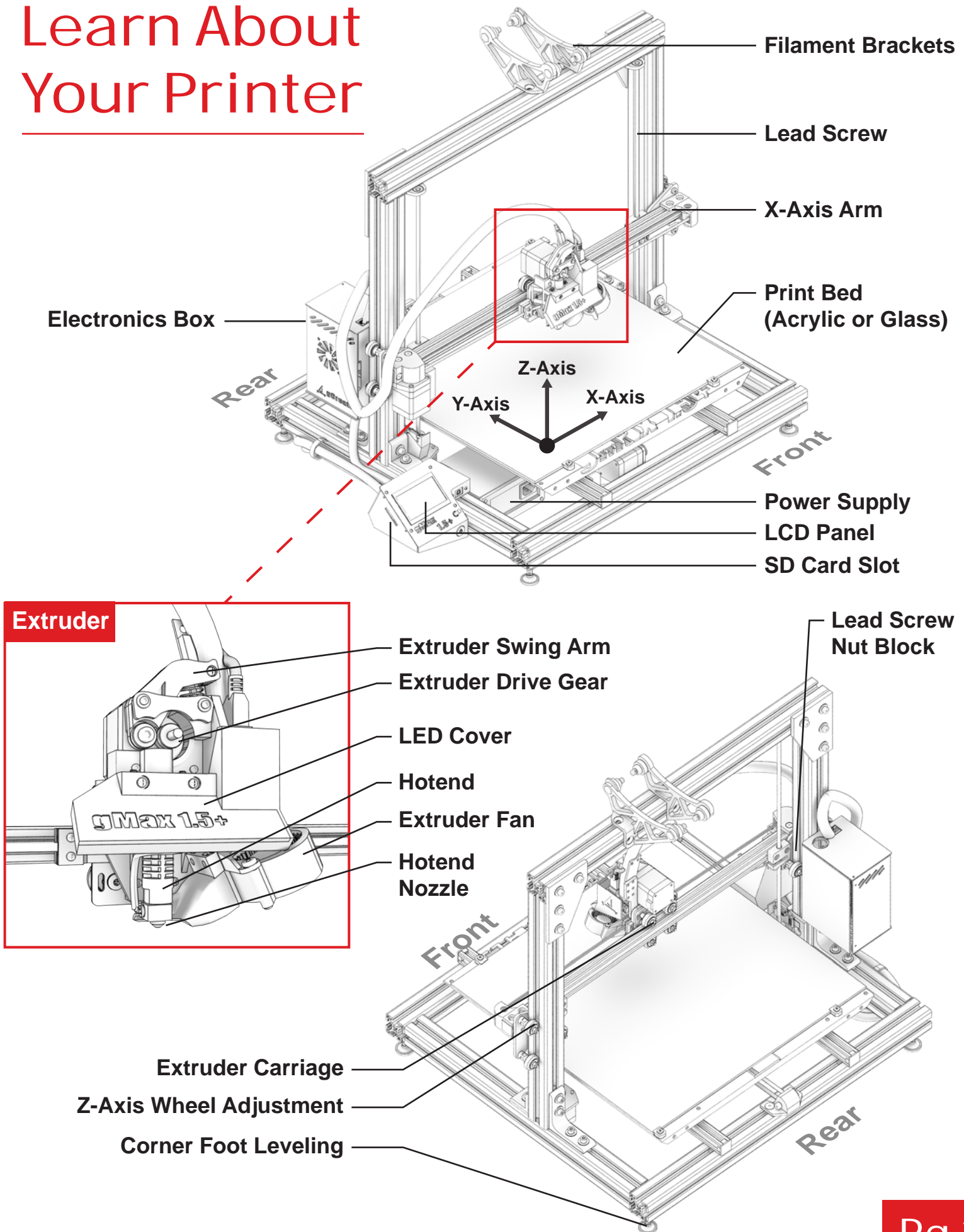


Even if you are familiar with 3d printing, please review this guide to set up and learn about your gMax Printer.

This guide should be used after you have received the printer and followed the unboxing instructions.

Check out [youtube.com/gCreate3d](https://www.youtube.com/gCreate3d)

Learn About Your Printer



Machine Specifications (For Reference)

Max Hotend Temperature (Do not set above)	300C (E3D All-Metal)
Layer Height Range	0.080mm - 0.740mm (Nozzle Dependent)
Filament Diameter	1.75mm
Firmware	Marlin 1.1.9 (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 range, 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 maximum firmware limit to prevent a "**MAXTEMP**" error.
- This machine includes sensitive wiring and hardware. It should be kept in a well ventilated and **dry environment**. Humidity can have adverse or negative affects on the printer and should be avoided.
- This printer includes parts which can exceed 300C 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 always 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 comes with a scraper for removing parts from the build plate, a power cord, and an extra bag of hardware for future upgrades.



Filament Spool Bracket



SD Card

Install Filament Bracket

1. Slide the filament spool bracket on to the top rail and tugthen the knob to secure it.
2. Open the supplied spool of filament and place the center hub on the filament bracket arm. For new spools make sure the filament doesnt unwind and wrap around the bracket.

Note

When printing tall objects, make sure the filament bracket is not in the way of the extruder or they will collide.

The filament bracket was designed to be placed anywhere along the top of the rail in various positions.

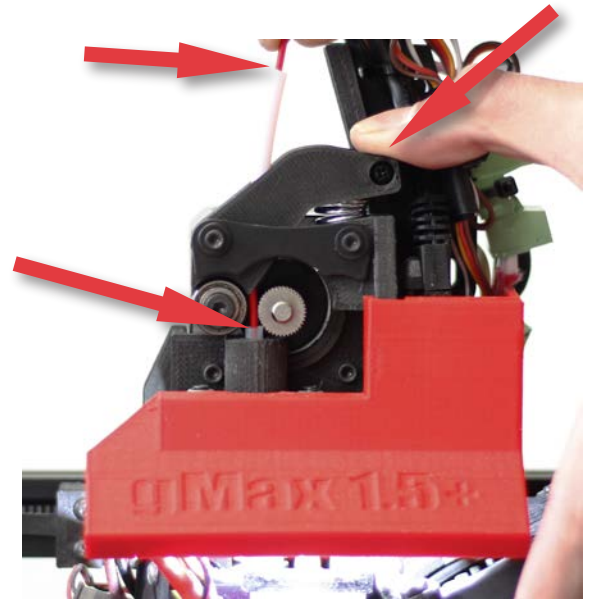


Insert Filament In Extruder



1. We recommend cutting off the first 24" of a new spool of filament.
2. Every time you insert filament, cut end of filament at a 45 degrees angle and **straighten 4" of the end of the filament by bending it.**

3. Pinch the extruder arm above the spring and insert filament in the white PTFE tube.
4. Slide filament in until you see it enter the top of the tube just under the metal drive gear. The filament should slide in about **4 inches more.**

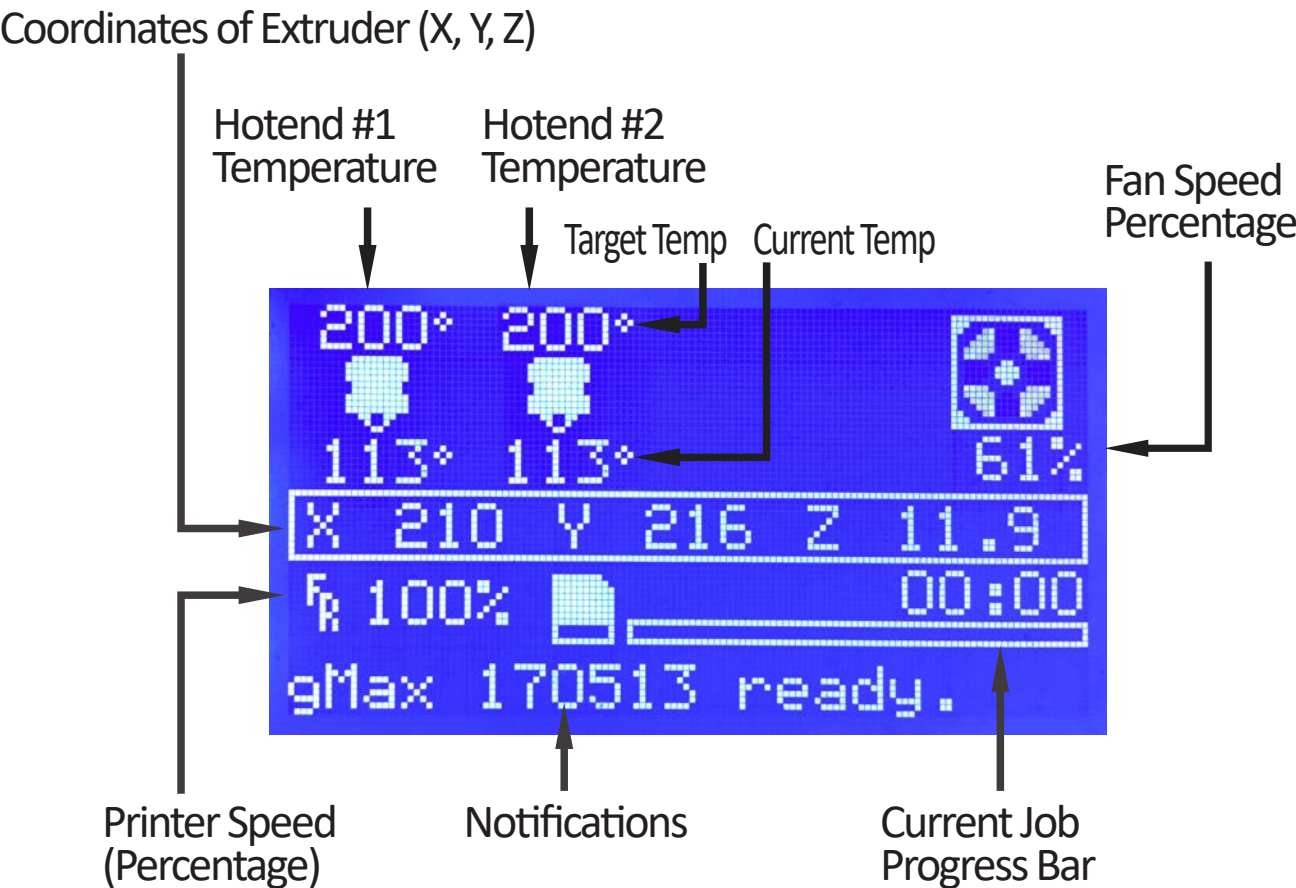
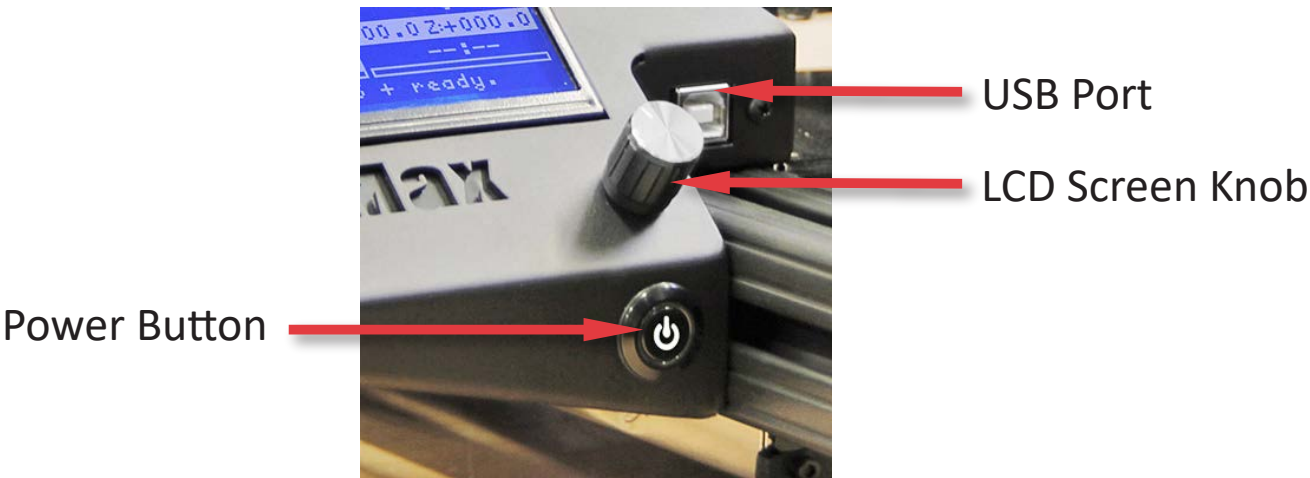


Plug In and Turn On



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 perform several procedures:



- X, Y, Z Axis test (slow moves, then fast moves)
- Bed Probe test (center and full grid)
- 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 or the bed probe doesn't drop/raise contact support for solutions.

Your First Print (Required)

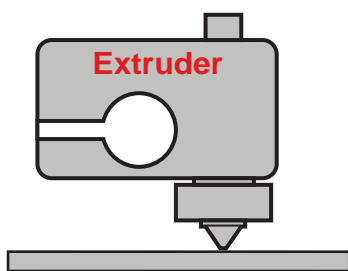
1. After conducting your Required Initial Run, this first print will confirm that your gMax is running the same as when it left the manufacturing facility. **This is a necessary and useful print.** Make sure to run it with the supplied filament and on a cold acrylic bed.
2. The gMax is equipped with an auto bed level sensor. To run bed leveling to “**gMax Tools > Level Bed**” and it will automatically save to memory when completed. **Run this program if your bed needs to be re-leveled.**
2. Next, under “**gCreate Files**” Select “**gMax First Print**”.
3. As the printing begins, **click the knob twice** quickly to access “**Live Z Adjust**”. You can turn the knob to raise or lower the first layer if needed.
4. **Use the Live Z Adjust function at the start of each print to ensure the perfect first layer.** Babystep moves the z-axis in small increments as you turn the knob to achieve the right first layer height. Refer to the diagram below.
5. To save the new height, go to “**Control > Store Settings**” any time before restarting the printer



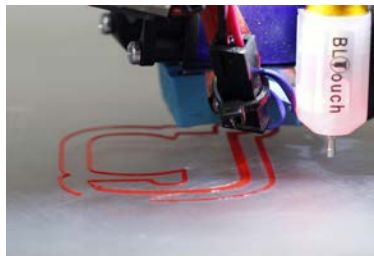
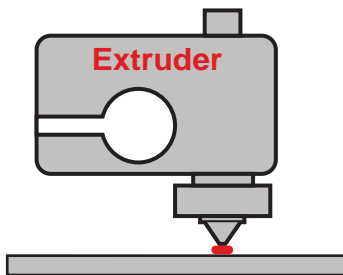
Note

As your print starts, double click the knob to enter Babystep mode.

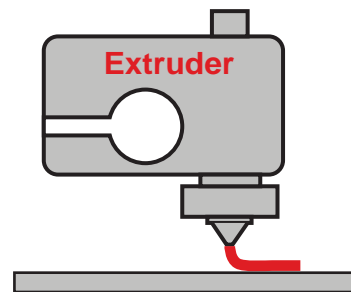
Turn **CLOCKWISE** to raise extruder
Turn **COUNTERCLOCKWISE** to lower



Extruder is too low
and digging into bed



Extruder is perfect

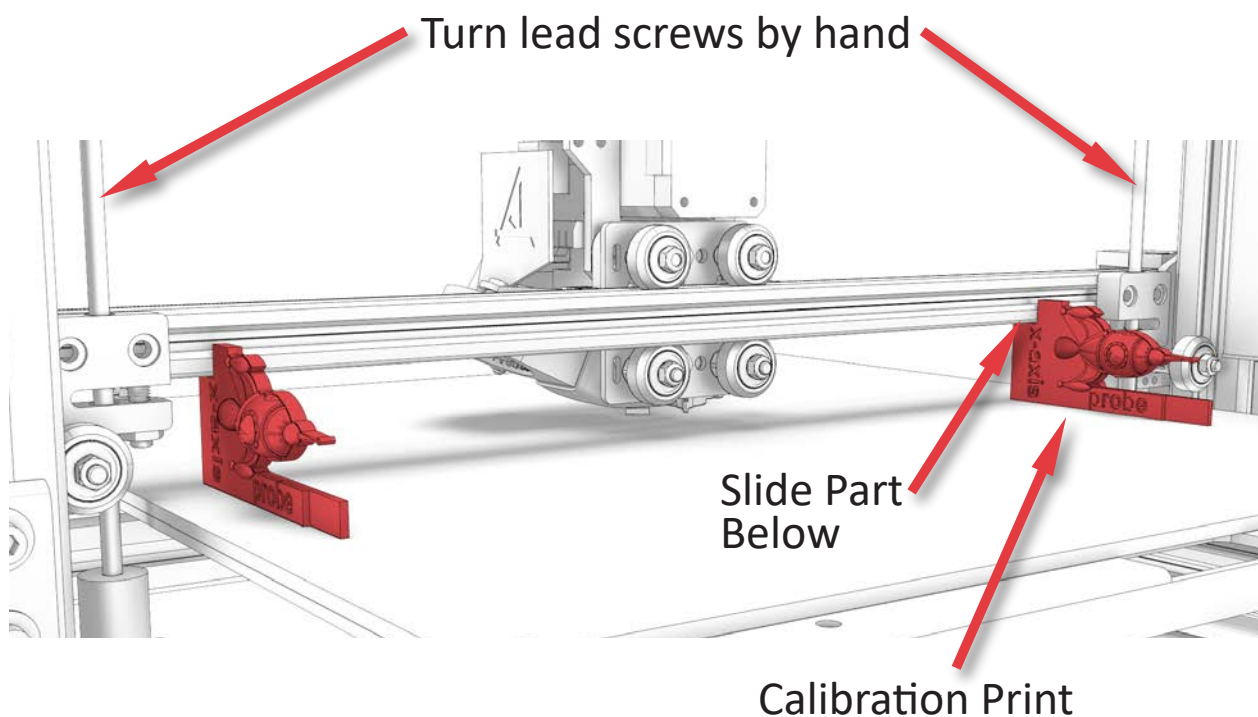


Extruder is too high
and printing in air

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 must be re-run after any adjustments**.

1. To level the X-Axis arm go to “**Prepare > Auto Home**”.
2. When it is done, turn off the printer and 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.
3. When complete, re-run bed leveling by going to “**gmax Tools > Save Bed Leveling**” .



Removing the Print. Strength vs Finesse

1. If the bed is **acrylic**, after the print finishes you can immediately remove it.
If you are using a **heated bed**, turn it off 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 scraper.
3. **USING CAUTION**, take the scraper and gently tap at the edges of the print. Look for corners of the model and tap them. After several taps the print should pop off the bed. If the model 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.

Bed Leveling

IMPORTANT: For bed leveling, you must run the routine from “gMax Tools > Level Bed” to level the bed and automatically save to memory.

It is important the x-axis stays level **relative to the bed**. If it changes you must re-run the bed leveling. We recommend running the bed leveling command before any large prints or after several prints.

1. The printer will automatically use the latest saved bed leveling matrix.
2. Depending on your slicing software make sure your “start gcode” has the following code. You **MUST** Remove any G29 commands!

G28 X0 Y0 <-- Remove

G29 <-- Remove

G28 ; Home all



Note

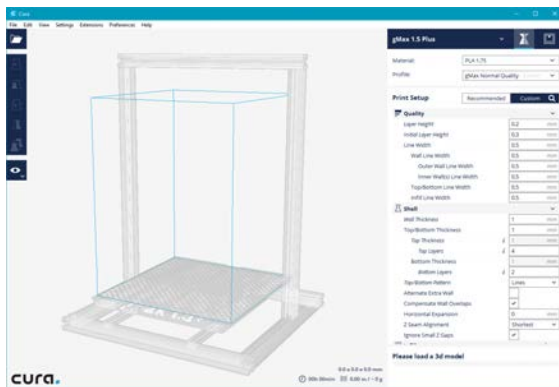
Always download the latest slicing configuration files from our forum.

3. When using Marlin 1.1.9, any babystep adjustments are automatically applied to the bed level sensor offset. If you would like to save it “Control > Store Settings” any time before your printer is restarted. The offset will now be saved for the start of the next print.

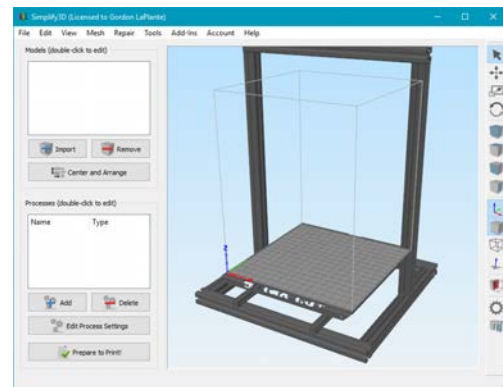
Slicing

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

1. For slicing, we recommend **Cura** or **Simplify3D**. View our forum at forum.gcreate.com, our YouTube channel youtube.com/gcreate or the respective slicing program websites for more information.
2. **Simplify3D may have older gMax configurations built-in.** Use the configuration files on the SD card or download them directly from our forum for the latest versions.



Cura (Free)



Simplify3D



Note

Slicing Tips

- Depending on your 3d model and filament type you should often change several main settings.

The majority of print quality issues can be fixed by changing slicing settings.

Using Marlin firmware 1.1.8 and above, the bed leveling has changed.

Make sure remove any G29 commands in your start and ending gcode.

Always download the latest configuration files from us to ensure you have the correct settings.

- 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 Cura and Simplify3d websites and forum.gcreate.com for many useful tutorials.
- Always use high-quality filament to reduce quality issues. Filament affects the print quality.

Useful Tips While You Print

1. You can adjust the speed of your print in real time by turning the LCD screen knob. Slowing the speed can be used to improve the print at a difficult area or if the print isn't cooling enough.

2. 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.



3. The cooling fan will not operate below 30% (or 76 from the LCD menu) since the power is too low. You should not set your fan over 85% or it may cool the hotend.
4. You can change filament during a print. Go to “**Tune > Change Filament**”. It can also be sent by your host software or you can put it in your gcode file manually by putting M600 where you want the change to happen in the gcode.

When activated the print will pause, the extruder will raise/move out of the way and it will retract the filament. After 45 seconds the extruder will turn off for safety. Insert your new filament and push the knob to resume.

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

6. Sanding the acrylic bed is recommended after every few prints. Use the sanding sponge and **wet it with water** to reduce dust. When done, wipe the acrylic clean with a paper towel.



7. When the nozzle is hot, carefully use needle nose pliers to remove any excess plastic which may have built up.
8. Do not use any lubricant on the lead screws, instead clean them off periodically with acetone.
9. Check your print often. If any part of the print fails, it may destroy the rest of the print and possibly damage the printer.
10. We recommend using the **cold acrylic bed whenever printing with PLA** for improved adhesion and enhanced cooling.