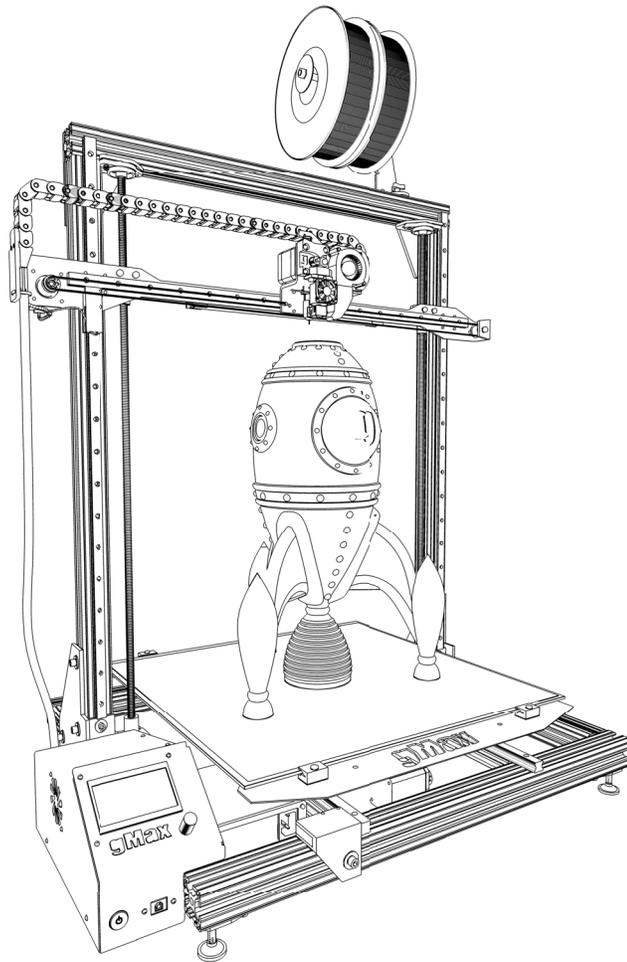


gMax 2 3D Printer

Getting Started

v190730 (Marlin 1.1.9 Firmware)

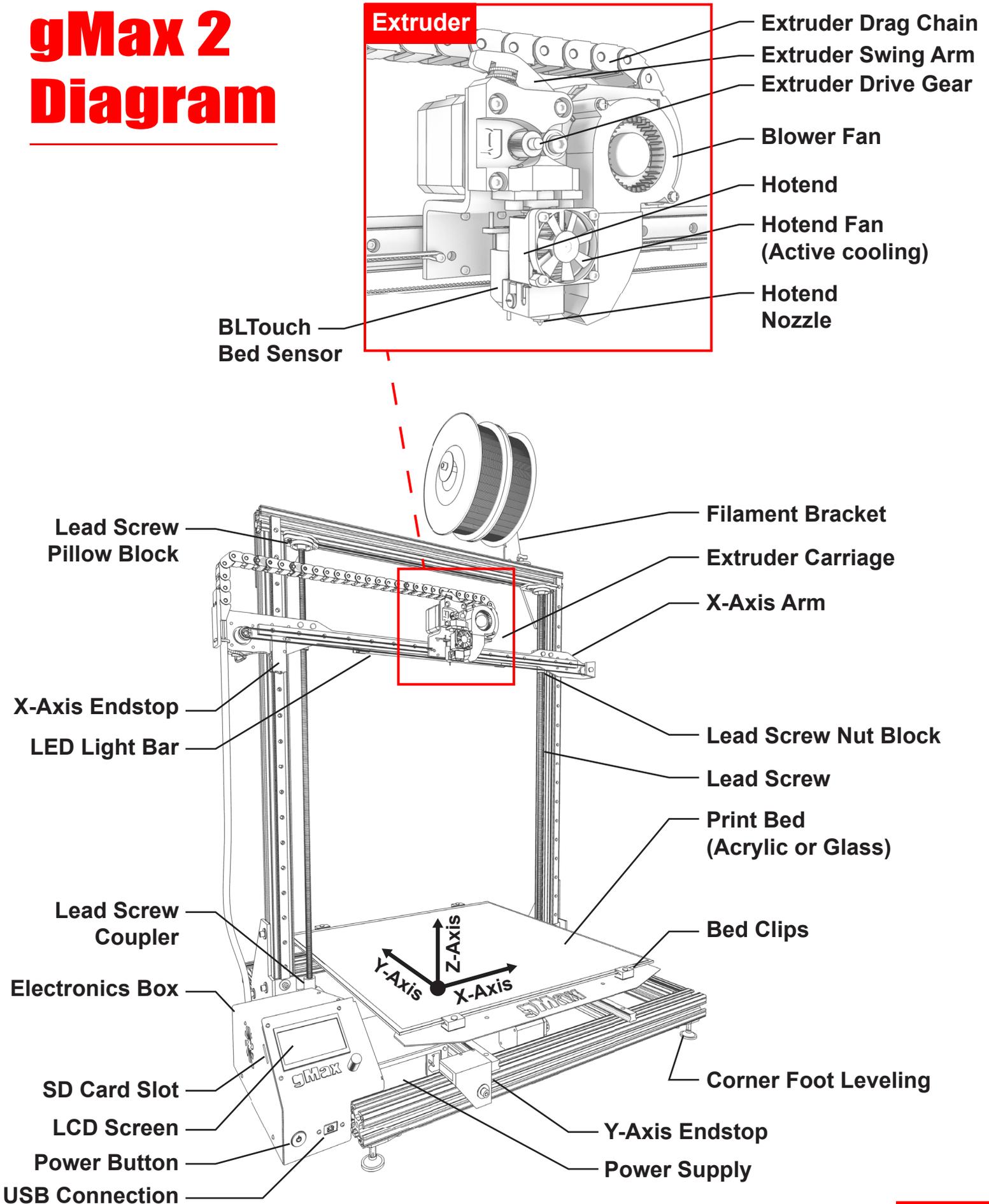


Even if you are familiar with 3d printing, please review this guide to set up and learn about your gMax Printer.

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)

gMax 2 Diagram



Machine Specifications (For Reference)

Max Hotend Temperature (Do not set above)	300C (E3D All-Metal)
Layer Height Range (Suggested)	0.080mm - 0.740mm (Nozzle Dependent)
Filament Diameter	1.75mm
Firmware	Marlin 1.1.9 (gCreate Modified)
Electronics	Ramps v1.4, ATmega 2560
Power	120V - 240 V (Full range)



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. Let the hotend cool down 20 minutes before touching it.
- Always keep a **working fire extinguisher** and have a working smoke alarm near the printer. Never place flammable objects near the printer including liquid chemicals that can release flammable vapors.
- Never open the electronics case when the printer is powered on.
- Any modifications or attempted repairs, not explicitly directed by gCreate, that cause damage are not covered under the Warranty

Printer Accessories

Open the accessory box and remove all the parts. Each printer comes with a scraper for removing parts from the build plate, a power cord, sanding sponge, hex keys, nozzle cleaning brush and an extra bag of hardware for future upgrades.



Print Scraper



Sanding Sponge



Filament Spool Bracket



SD Card



Filament Cutters



Additional Nozzles



Brass Nozzle Cleaning Brush



2.5mm Driver



Spare Hardware

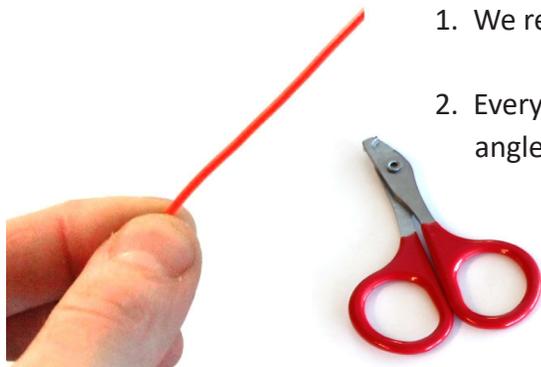


Hex Key Set



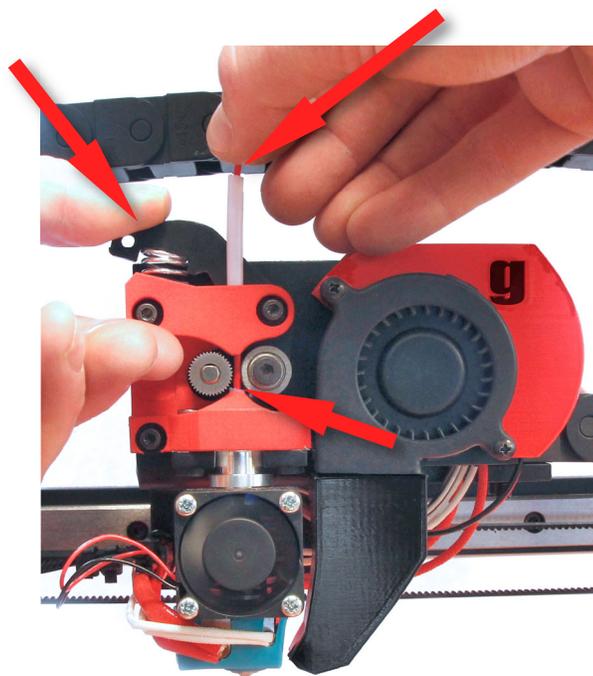
Power Cord

Insert Filament In Extruder



1. We recommend cutting off the first 12" of a new spool of filament.
2. Every time you insert filament, cut end of filament at a 45 degree 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.

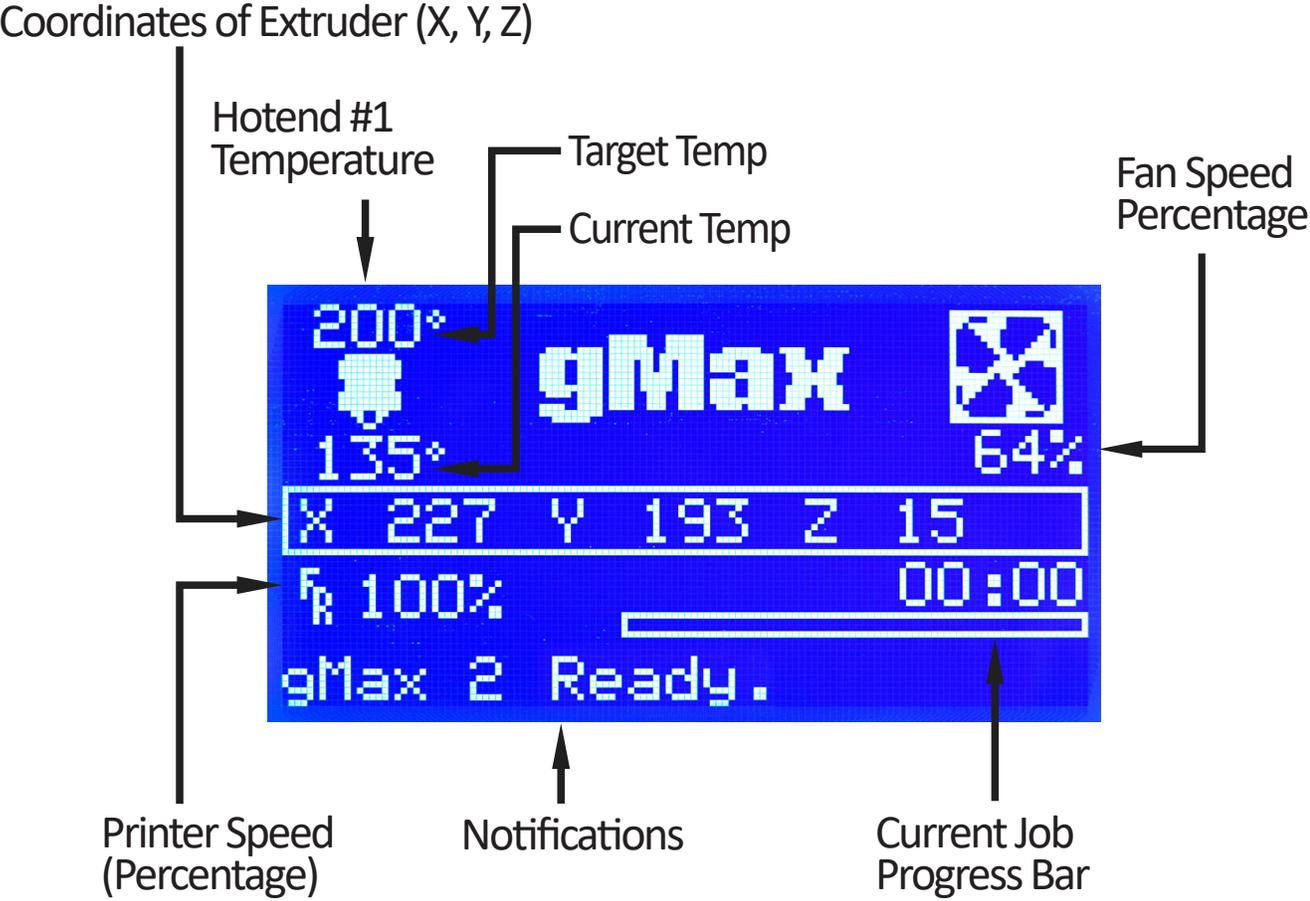
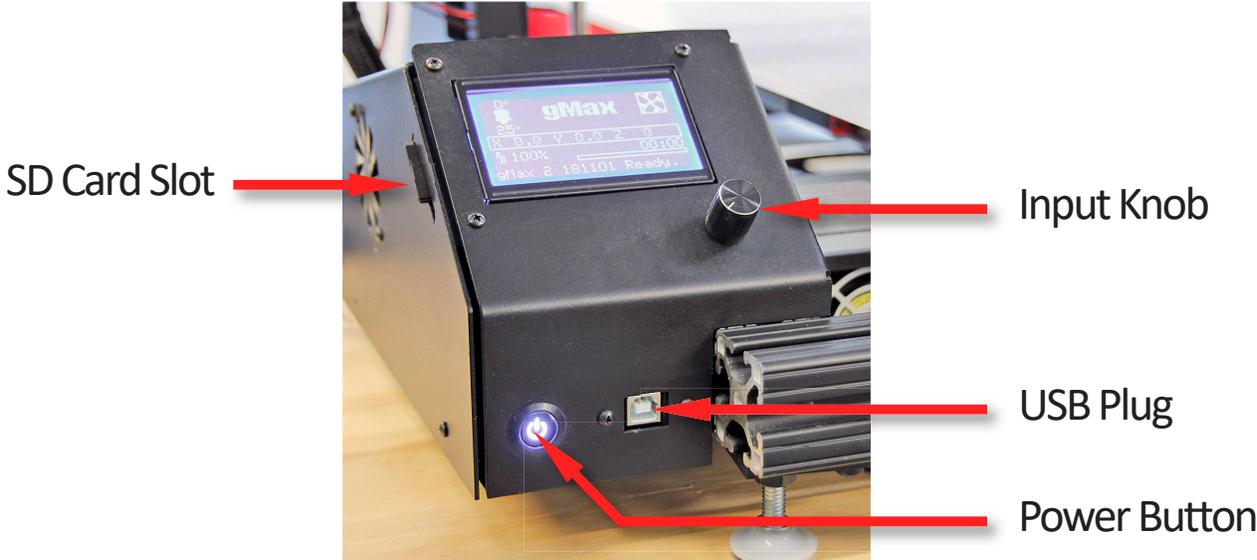


Plug In and Turn On



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 functions:
 - 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



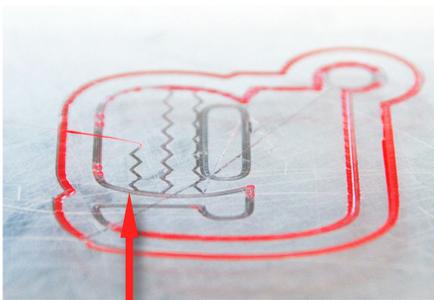
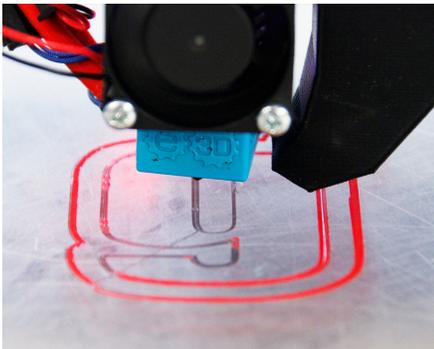
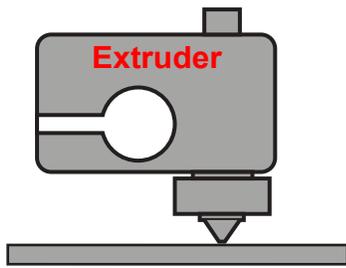
Note

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.

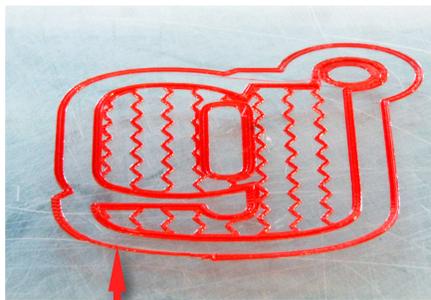
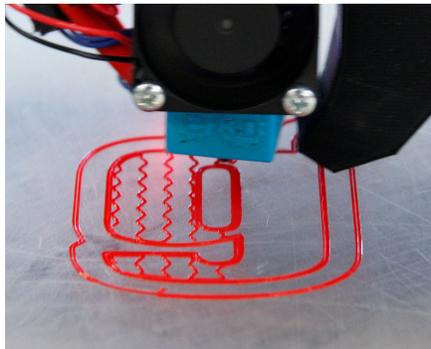
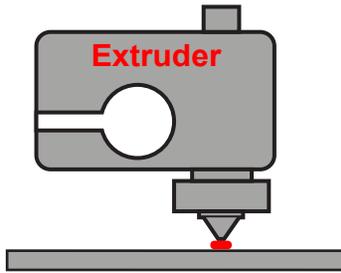
1st Layer Cheat Sheet

Extruder is too low



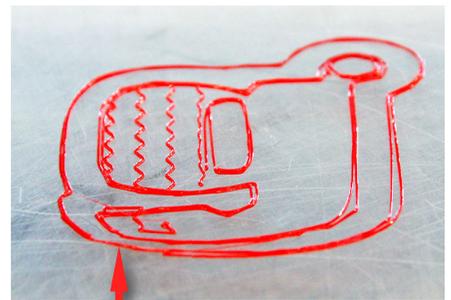
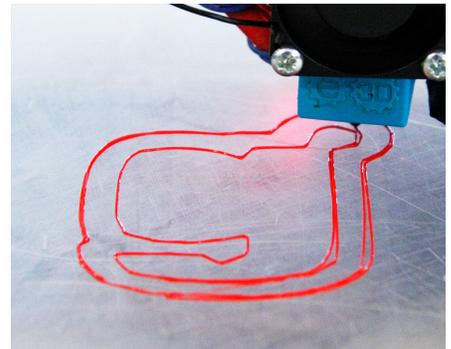
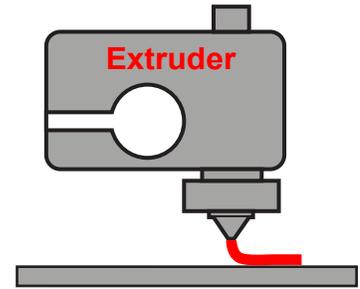
Too low and nozzle is digging in the bed causing damage.

Extruder is perfect



Correct height. Sticks to bed just enough.

Extruder is too high



Too high and not adhering to the bed



Lower
(Counter clockwise)

Raise
(Clockwise)



Note

As your print starts, double click the knob to enter "Live Z Adjust" mode.

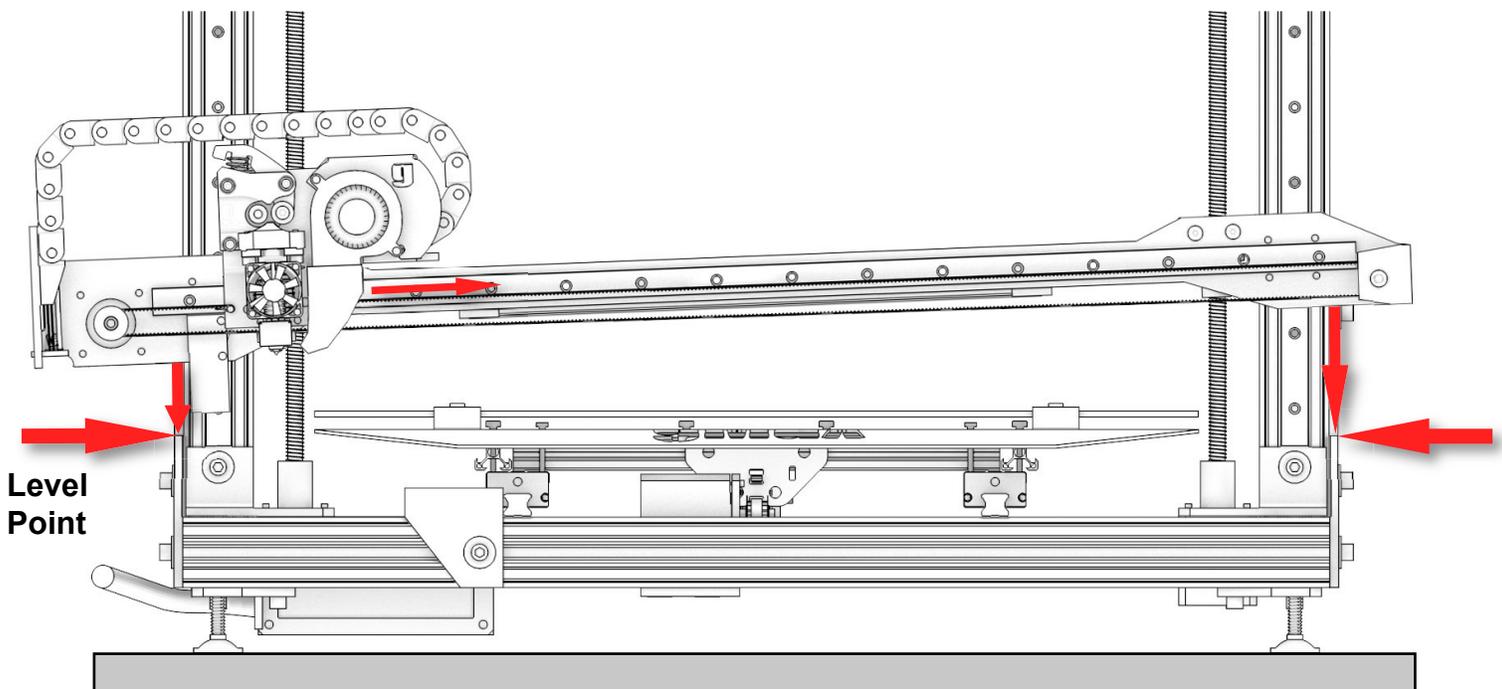
Turn **CLOCKWISE** to raise extruder
Turn **COUNTER CLOCKWISE** to lower.

You may have to turn up to **0.100** or **0.200** to see a change.

Leveling X-Axis Arm

The gMax 2 has a new method for leveling the X-Axis arm. The X-axis arm should be leveled using this method prior to any bed leveling.

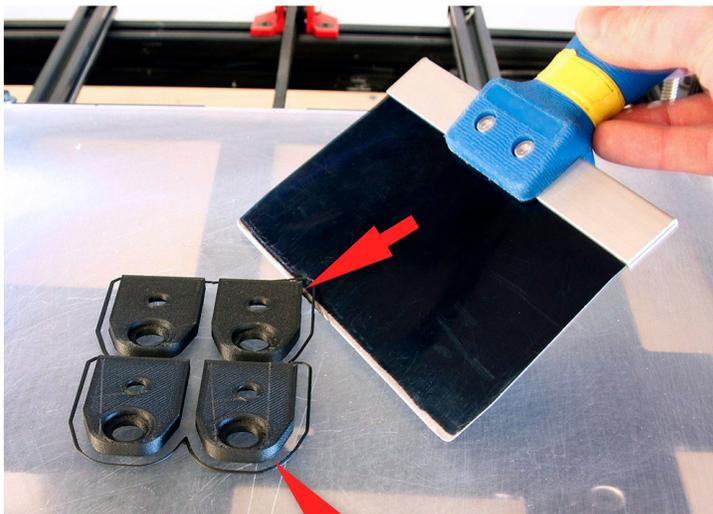
1. To level the X-Axis arm go to “**gMax Tools > Level X-Axis**”.
2. The printer will home to the center, and the extruder will move to the right side.
3. The printer will then lower until the extruder hotend is below the bed and the X-axis arm will bump against the metal side plates. This will ensure the printer always has the same baseline when starting a print and the bed leveling will handle the rest.



4. If the arm doesn't look level, run the routine again.
5. Run bed leveling. Go to “**Prepare > Level Bed**”.

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.
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.



Tap Edges

Warning

The scraper is very sharp. **NEVER** place your hand behind the area you are scraping and take caution not to dig into any build surfaces.

1. Sanding the acrylic bed is recommended after every few prints. Sanding will improve the surface finish and help achieve great adhesion.

Use the sanding sponge and **wet it with water** to reduce dust. When done, wipe the acrylic clean with a paper towel.

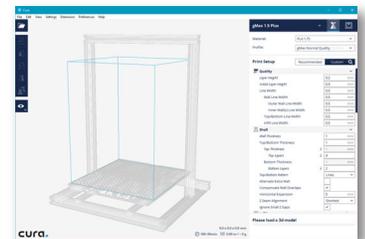
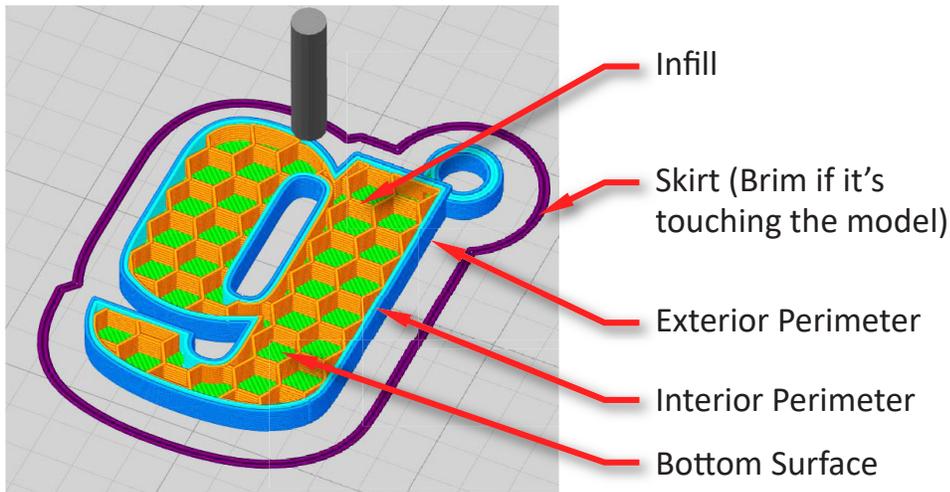


Slicing

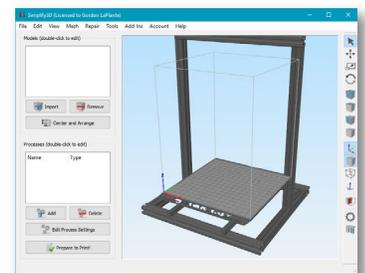
Slicing is one of the most important parts of the 3d printing process. The proper settings can result in a great print however many settings are usually model-specific. Taking the time to learn the best settings will result in a better printing experience.

1. For slicing, we recommend **Cura** or **Simplify3D**. View our forum at forum.gcreate.com, our YouTube channel youtube.com/gcreate3d 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.

Basic Anatomy of Slicing



Cura (Free)



Simplify3D

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.**

- 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 issues and improve your results.



Note

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

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