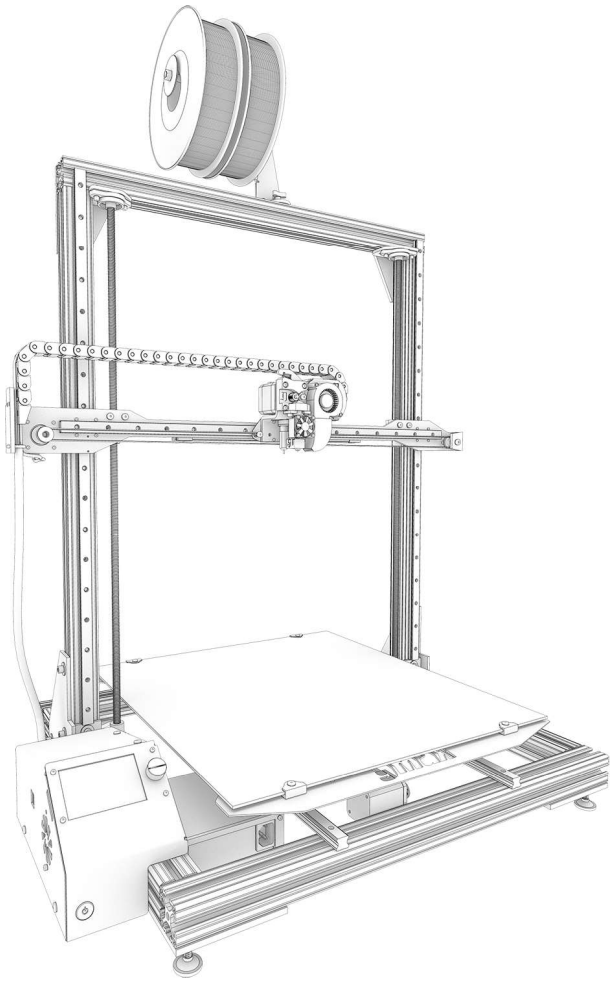
gMax 2 PRO
Getting Started

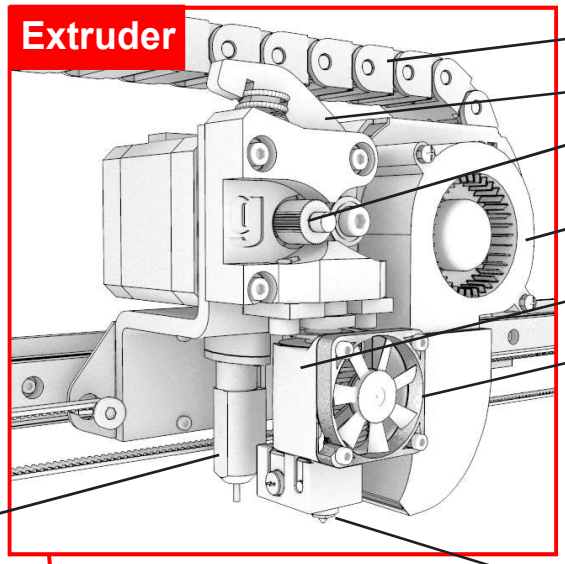
v230317 Single and Dual Extruders (Marlin 2.1.1 Firmware)

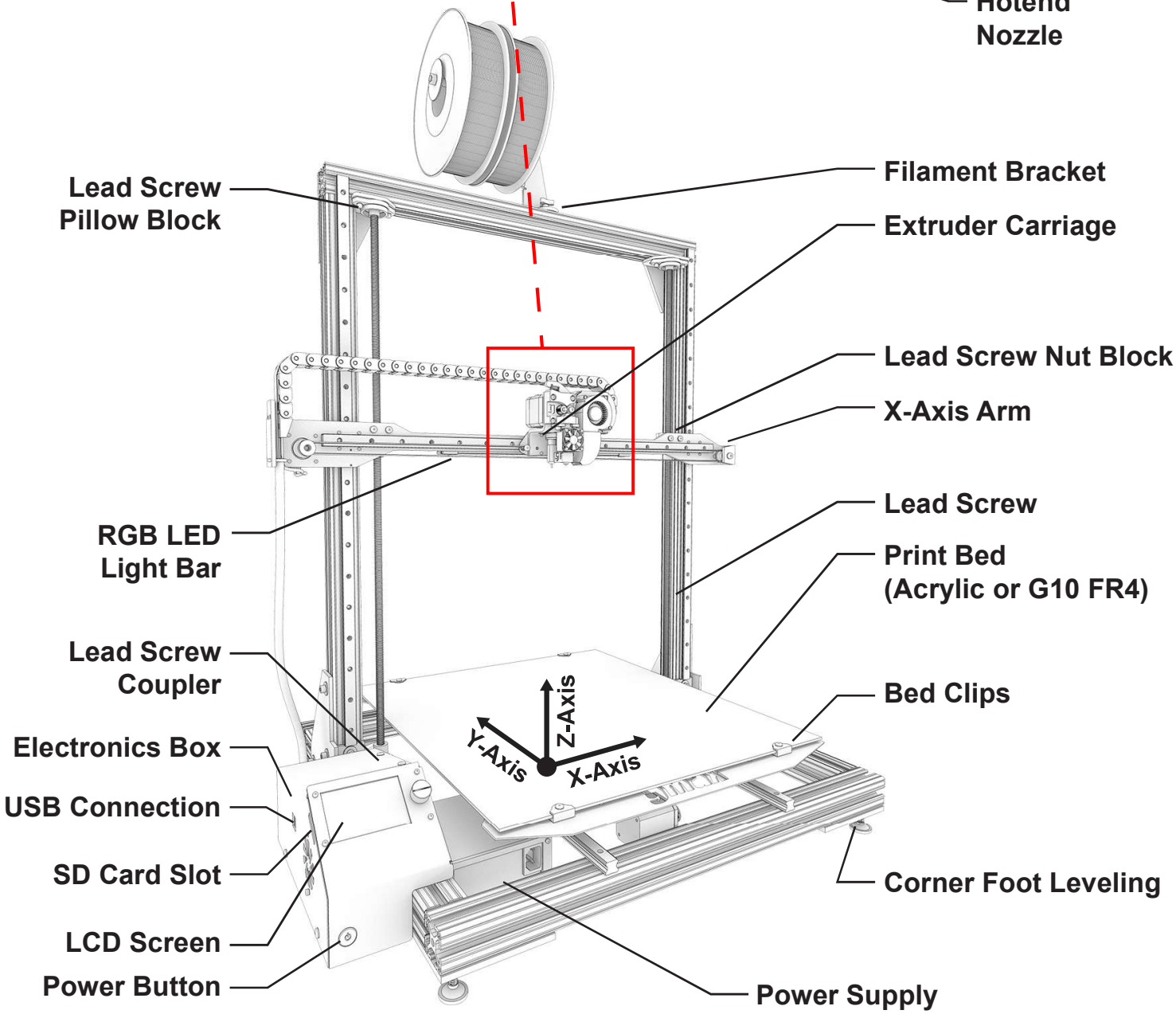


Even if you are familiar with 3d printing, please review this guide to set up and learn about your gMax Printer since all printers are different.

This guide should be used after you have received the printer and followed the unboxing instructions. This manual is for Single and Dual extruder printers

gMax 2 PRO Diagram





Machine Specifications (For Reference)

Max Hotend Temperature (Do not set above)	245C - 400C (Hotend dependent)
Layer Height Range (Suggested)	0.080mm - 0.740mm (Nozzle Dependent)
Filament Diameter	1.75mm
Firmware	Marlin 2.1.1 (gCreate Modified)
Electronics	SKR v2, STM32 32-bit (Arm Cortex M4) SKR v3, STM32H743VI (Arm Cortex M7)
Power	120V - 240 V (Selectable)
Heated Bed	120C MAX TEMP

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 350C and caution should be taken. Do not let pets or children near the product without supervision. Let the hotend cool down 10 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
- Plug your printer and heated bed into a UPC power backup and surge protector.

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, hex keys, nozzle cleaning brush and an extra bag of hardware for future upgrades.

Filament Spool Bracket

Print Scraper

SD Card

Filament Cutters

Additional Nozzles

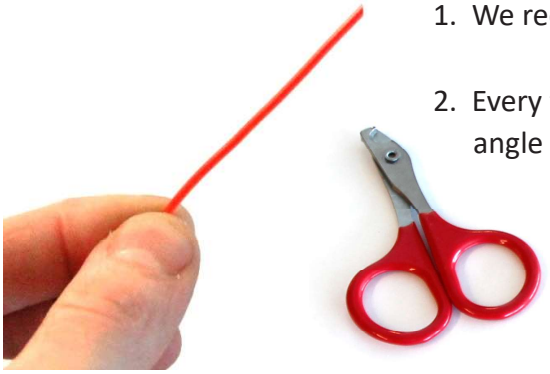
Brass Nozzle
Cleaning Brush

Spare Hardware

Hex Key Set

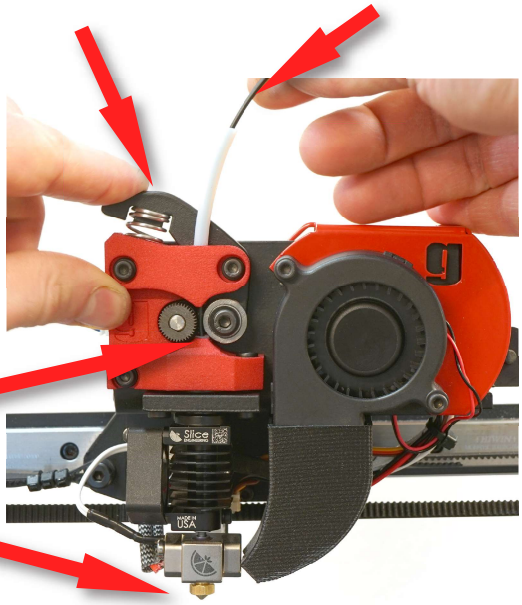
Power Cord

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.

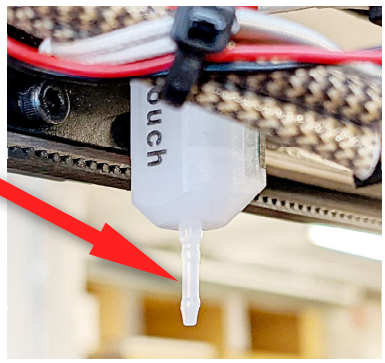
4. Slide filament in until you see it enter the top of the tube just under the metal drive gear. The filament should slide in about **4 inches total** to reach the nozzle.

Bed Leveling Probe

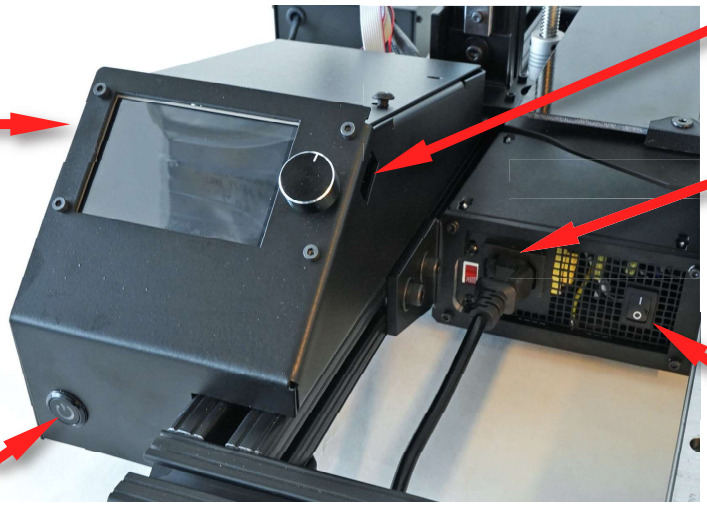
1. With the printer off, gently pull down the plastic pin on the bltouch probe on the extruder. Inspect it to make sure it isn’t bent or broken. It should extend about 0.375” (9.5mm) from the body.

2. Starting the printer with the pin down performs a “soft reset” and can sometimes resolve an issue if the probe doesn’t deploy.

3. When you turn on the printer, the probe should move up and down then stay stowed away.



Plug In and Turn On



SD Card Slot

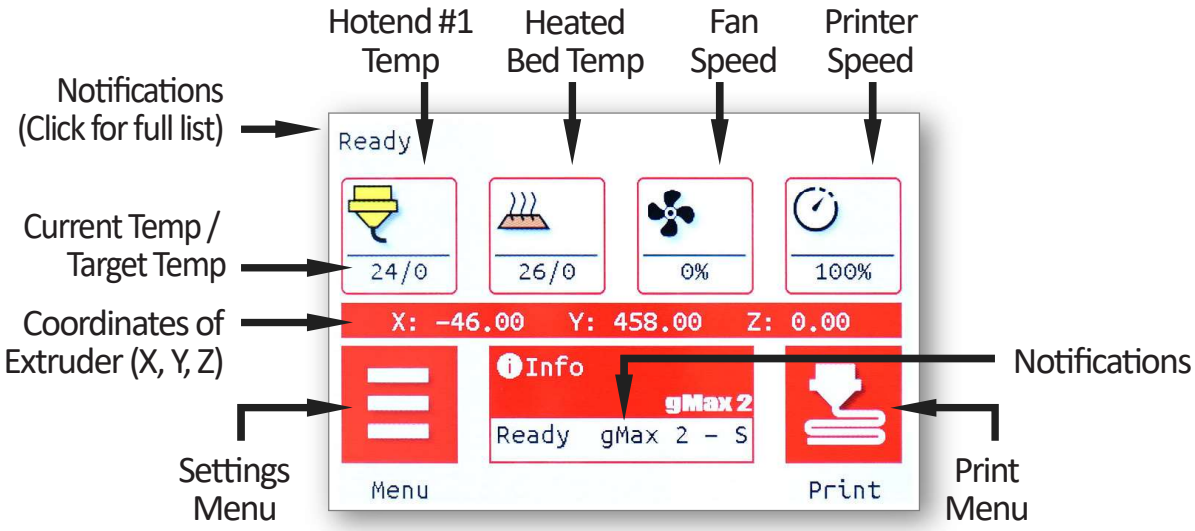
Power Button

USB Slot (use 90 degree adapter or small usb stick)

1. Plug in the printer to a **surge protected** power strip. A UPC power supply is recommended.

2. Make sure power switch is on.

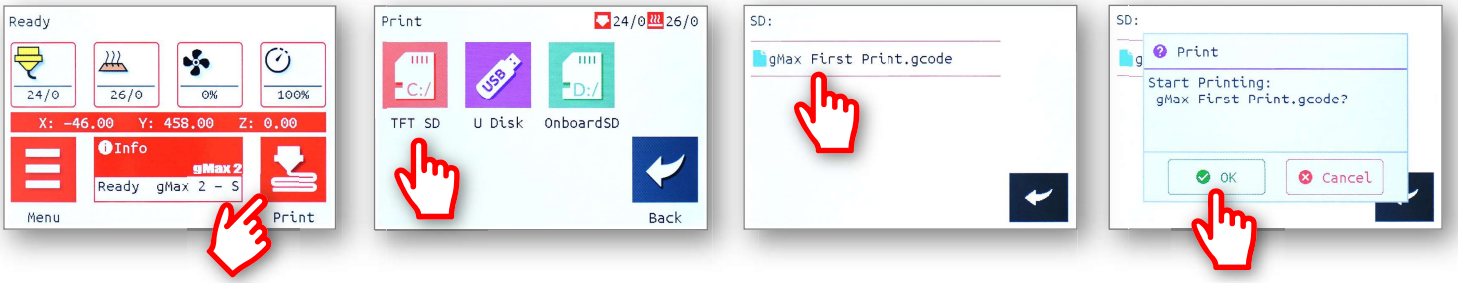
LCD Screen Breakdown



gMax Initial Run (Required)

1. You **must run the initial setup** file to test your gMax printer after transport. Slide the SD card (**upside-down**) into the left side of the LCD screen.

2. On the LCD screen, the notification area will 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.



3. Select "**Print > TFT SD > Initial Setup.gcode**".

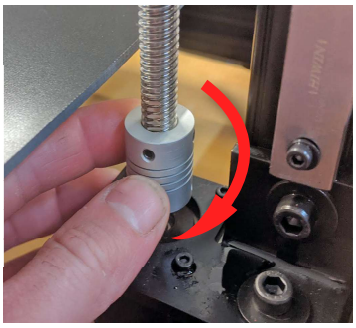
The test file will perform several functions:

- Bed Probe Test
- X, Y, Z Axis Motor test (slow and fast)
- Multiple Homing Test
- Extrusion test (without heat)
- Extruder Heat Up
- Fan test

Note

The gMax "Initial Run Test File" should take about **15 minutes** to complete. If the printer fails during any of these tests or the bed probe doesn't drop/raise check the ribbon cables or contact support.

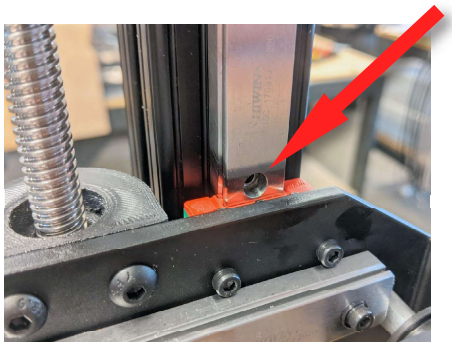
Manually Level X-Axis Arm



The gMax 2 PRO x-axis arm should be manually leveled prior to bed leveling especially after it has been shipped otherwise z-axis may bind as it raises.

1. Turn the couplers that connect the z-axis motors to the lead screws. Clockwise will raise the x-axis arm and counter clockwise will lower it.

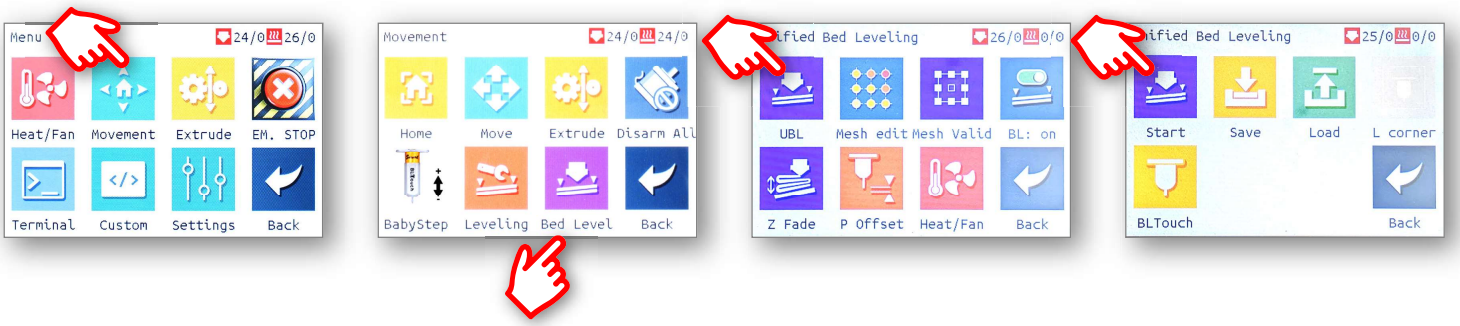
2. Line up the top of the arm on both sides to the nearest bolt hole on the linear rails.



Bed Leveling

The gMax 2 Pro uses a bltouch probe and Unified Bed Leveling (UBL) to create a 3d mesh of the bed. The mesh is saved to memory and helps ensure the extruder follows the surface of the bed. We recommend running the bed leveling command before any large prints. Also, ***never put G29 in your start gcode****.*

1. Always **preheat the bed to your desired first layer temperature** prior to leveling the bed. Go to "**Heat/Fan > Preheat**" then select your temperature. To run the bed leveling routine go to "**Menu > Movement > Bed Level > UBL > Start**".



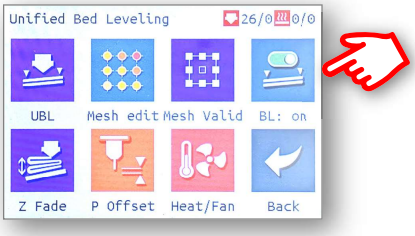
2. When the leveling routine finishes click "**Ok**" and save it to slot 0.

3. Click "**Ok**" to load the mesh before a print.

3. You can review the mesh saved to memory, by selecting "**Mesh Edit**".

Note

Make sure Bed Leveling is on after the leveling has completed



Tune the Hotend MPC Heating

Use MPC tuning to tune your hotend heating cycle to match the environment. It is recommended to do this if your environment changes.

1. Go to "**Menu > Settings > Machine > Tuning > MPC > Start**".

2. Let the tuning finish.

Your First Print (Required)

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 and it must be run with the supplied filament.**

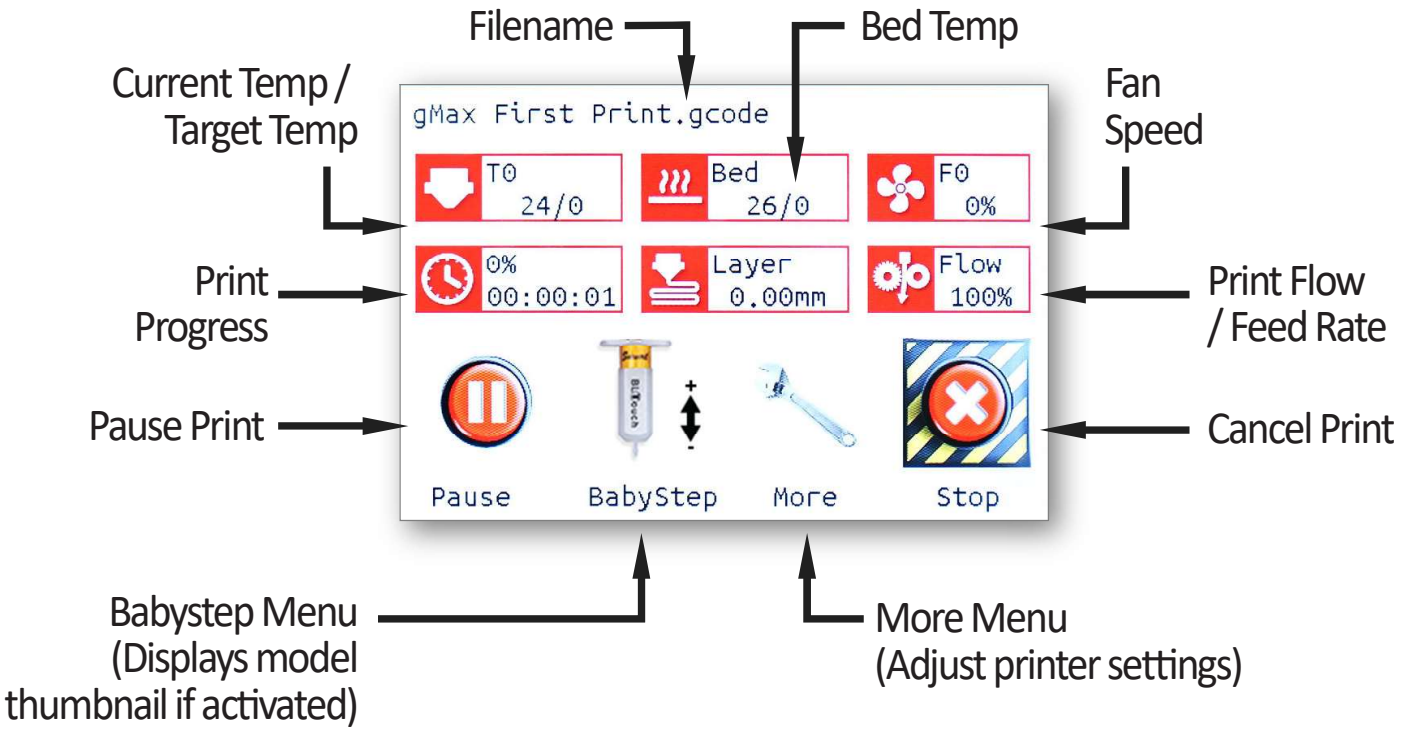
2. Use the supplied alcohol pads to clean the bed. Any oil on the bed will reduce adhesion.

2. Go to "**Print > TFT SD >Your First Print**". As the printing begins, you **will** need to adjust the first layer height using the Babystepping feature. Refer to the next page for more information.

Note

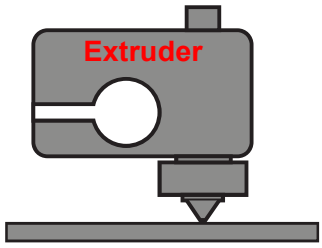
The printer bed probe is intentionally set high when leaving our facility. You will have to babystep the printer on your first print.

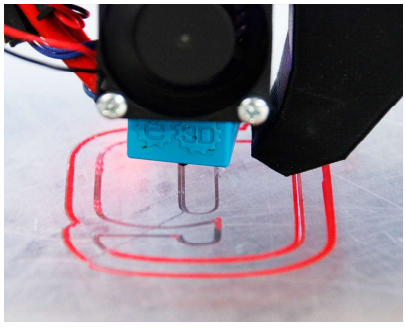
Printing Status Screen

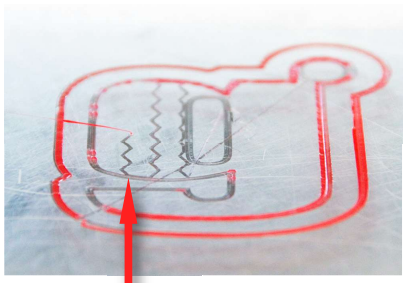


Babystepping Cheat Sheet

Extruder is too low

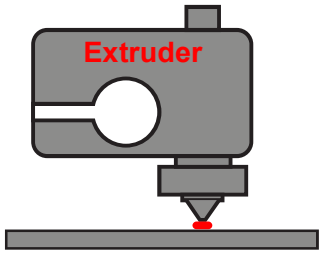


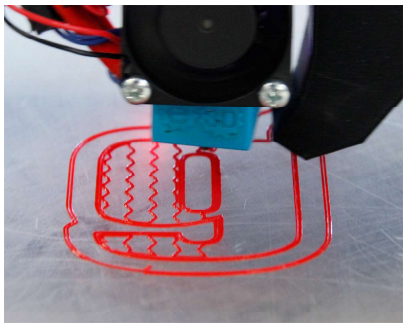


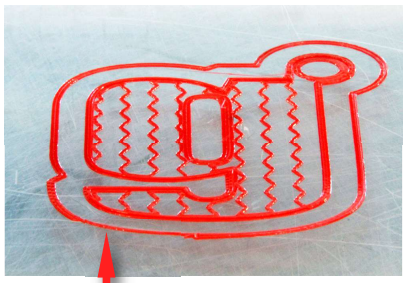


Too low and nozzle is digging in the bed causing damage.

Extruder is perfect

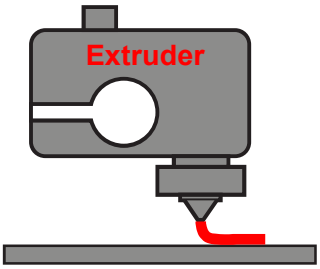


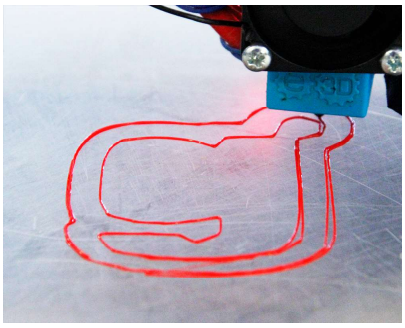


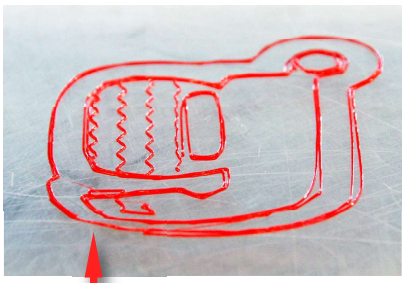


Correct height. Sticks to bed just enough.

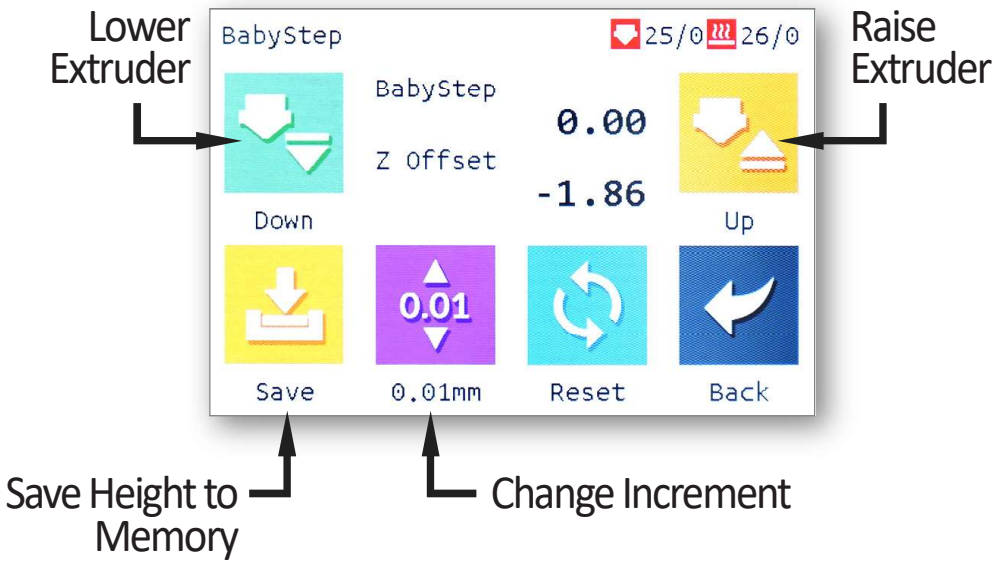
Extruder is too high







Too high and not adhering to the bed



Note

As your print starts, click “Babystep”.

You may have to raise or lower up to **0.50 mm** depending on settings and if your nozzle has been changed. It will take a few seconds to see any change in height

Removing the Print. Strength vs Finesse

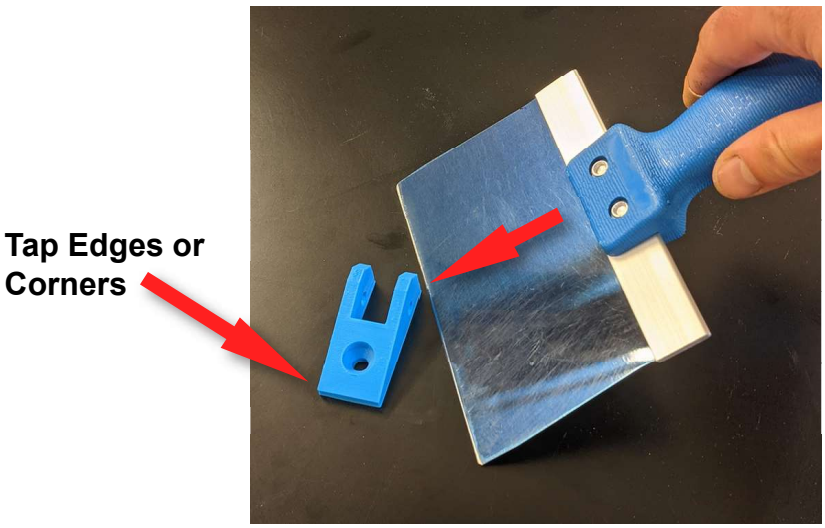
1. If the bed is **acrylic**, after the print finishes you can immediately remove it.
   If you are using a **heated bed**, turn it off and **let it cool several minutes before removing** or you risk damaging the print surface or warping your print.

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.

Warning

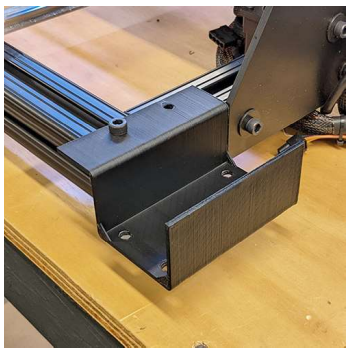
The scraper is very sharp. **NEVER** place your hand behind the area you are scraping and take caution not to dig into any build surfaces.



Additional Resources

1. The SD card supplied with your printer has sample files, user guides and wiring diagrams for the gMax. Feel free to explore the files and perform test prints with our settings. The sample files on the card can be a great starting point to use for your prints.

3. You can find a very useful SSR bracket on the SD card which makes for a great test print. This bracket holds the SSR box on the left rear of the printer.. We have included a drop in tnut and M5 bolt to secure the bracket.



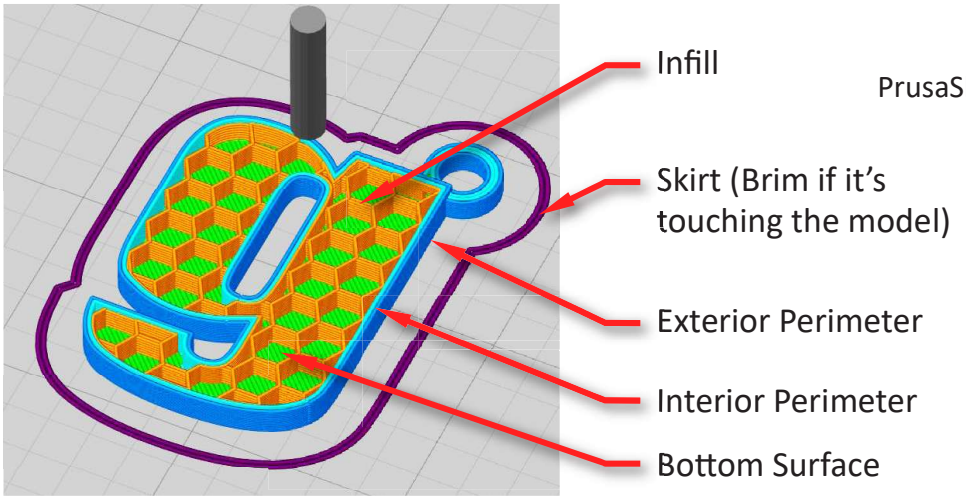
Slicing

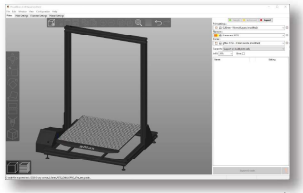
Slicing is one of the most important parts of the 3d printing process. The proper settings can result in a great print however many settings are usually model-specific. Taking the time to learn the best settings will result in a better printing experience.

1. For slicing, we highly recommend **PrusaSlicer.** You can also try **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.

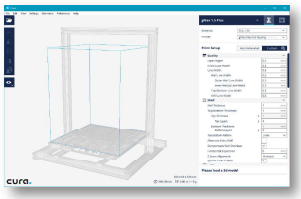
2. **The slicing programs may have older gMax configurations built-in.** Always use the configuration files on the SD card or download them directly from our forum for the latest versions.

Basic Anatomy of Slicing

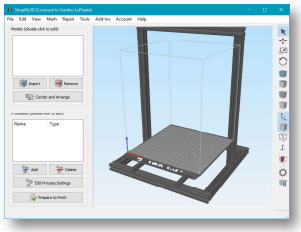




PrusaSlicer (Recommended and Free)



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 our articles and tutorials for more information.

Note

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

• Always use **high-quality** filament to reduce issues and improve your results. Also, several PLA filaments require higher temperatures around 215C if you experience clogging.

Useful Tips While You Print

1. The SD card contains useful guides and sample files. Review the User Guide for information on maintenance, error codes and troubleshooting tips.

2. You can adjust fan speed, hotend temperature and filament flow during a print. Note **these settings are not permanent** and they will change when the printer reaches a new gcode command (overriding the settings) or the print is canceled.

3. The cooling fan will not operate below 30% since the power is too low.

4. You can change filament during a print by just pausing a print or using the runout sensor. You can also trigger it manually by putting M600 in your gcode where you want the change to happen.

5. **Do not** use any lubricant on the lead screws, instead clean them off periodically with acetone or rubbing alcohol. The POM nuts are self lubricating and additional lubricant will cause dust to accumulate and get sticky.

6. Check your print often. If any part of the print fails, it may destroy the rest of the print or damage the printer.

7. **Do not** use acetone or harsh solvents to clean acrylic of the G10FR4 print bed. Instead use **isopropyl alcohol** or **glass cleaner**. If the bed had any oil on it, plastic won't stick. You can use high hold hairspray, glue stick or additional adhesives on the heated bed for certain brands of PLA or other filaments to get better adhesion.

X-Axis Twist Wizard

Marlin 2.0.9.3 *and later includes an X-Axis Twist Wizard to compensate for any twist across the x-axis. It is useful to compensate if one side of the bed is higher/lower after bed leveling. If the printer has probing errors during bed leveling, Reset the EEPROM and run the X-twist after leveling.*

1. Currently the wizard must be run from "**Classic Marlin**" mode. Press anywhere on the screen for two seconds and select "**Marlin Mode**".

2. To run the wizard go to "**Configuration > Advanced Settings > Probe Offset > X-Twist Wizard".** Follow the instructions on the screen. Use a thin slip of paper below the nozzle to adjust the height.

Note

The x-axis twist values will save to memory and only needs to be re-run if you reset the printer to factory defaults or install new firmware.

Temperature Chart

Use this chart as a starting point for recommended temperatures. ***Note*** *these temperatures will vary per filament manufacturer, environmental changes and per 3d model.*

		PLA	TPU	PETG	Nylon
Heated G10 FR4 Bed	Bed:	73°C	50°C	80°C	100°C
	Filament:	220°C	195°C	235°C	235°C
Heated G10 FR4 Bed (w/ Hairspray)	Bed:	55°C	50°C	70°C	95°C
	Filament:	210°C	195°C	210°C	210°C
Cold Acrylic	Bed:	0°C	0°C	N/A	N/A
	Filament:	208°C	195°C	N/A	N/A

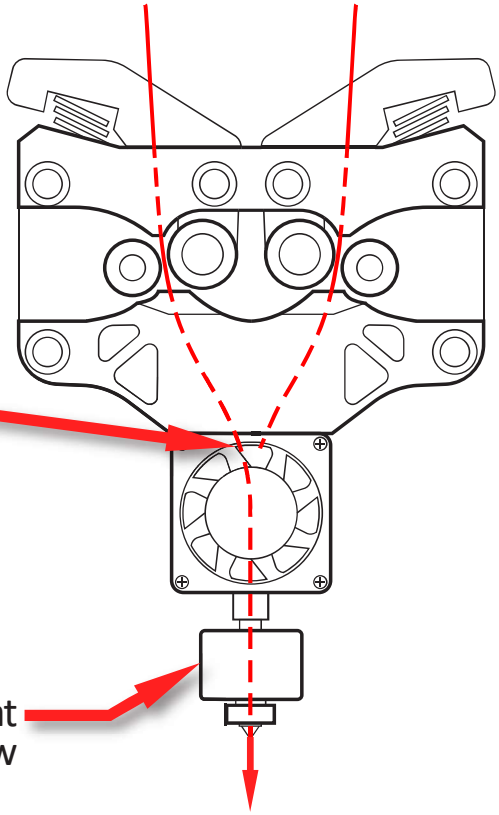
Dual Extruder Printers - 2in1 Dual

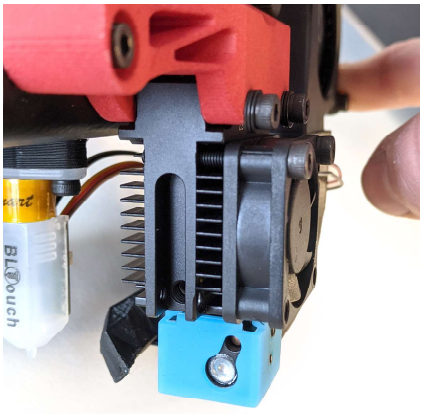
The 2in1 dual extruder uses similar settings as the single extruder and does not require independent adjustments of the hotend height. Make sure to update your slicing settings.

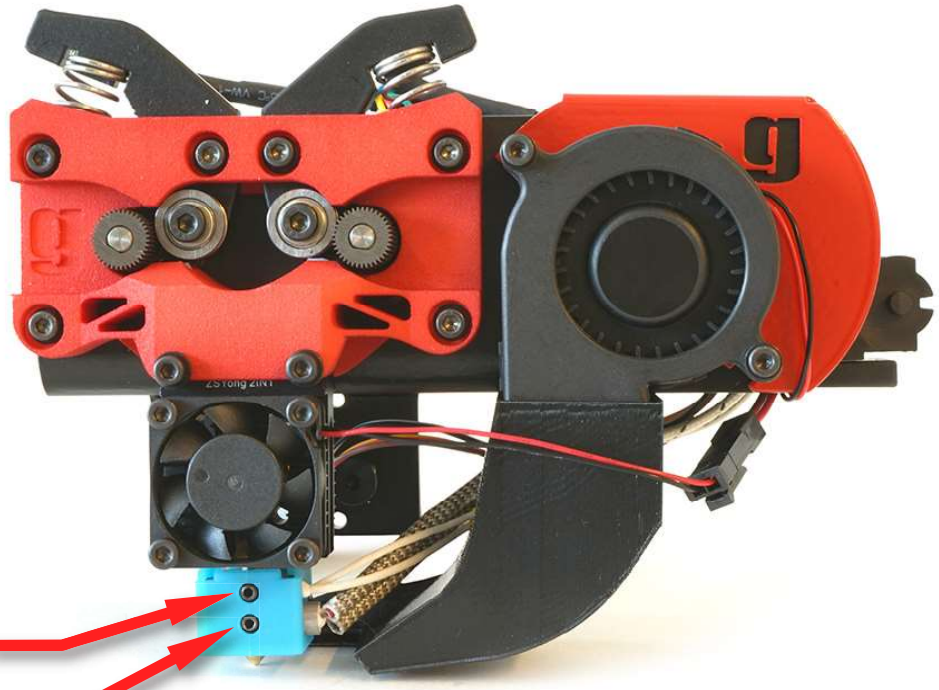
1. Make sure when using the 2in1 dual extruder to insert the left (extruder #1) filament first until it comes out the hotend.

2. Then insert the right (extruder #2) filament until you feel when it hits the first filament and pull back about 5mm.

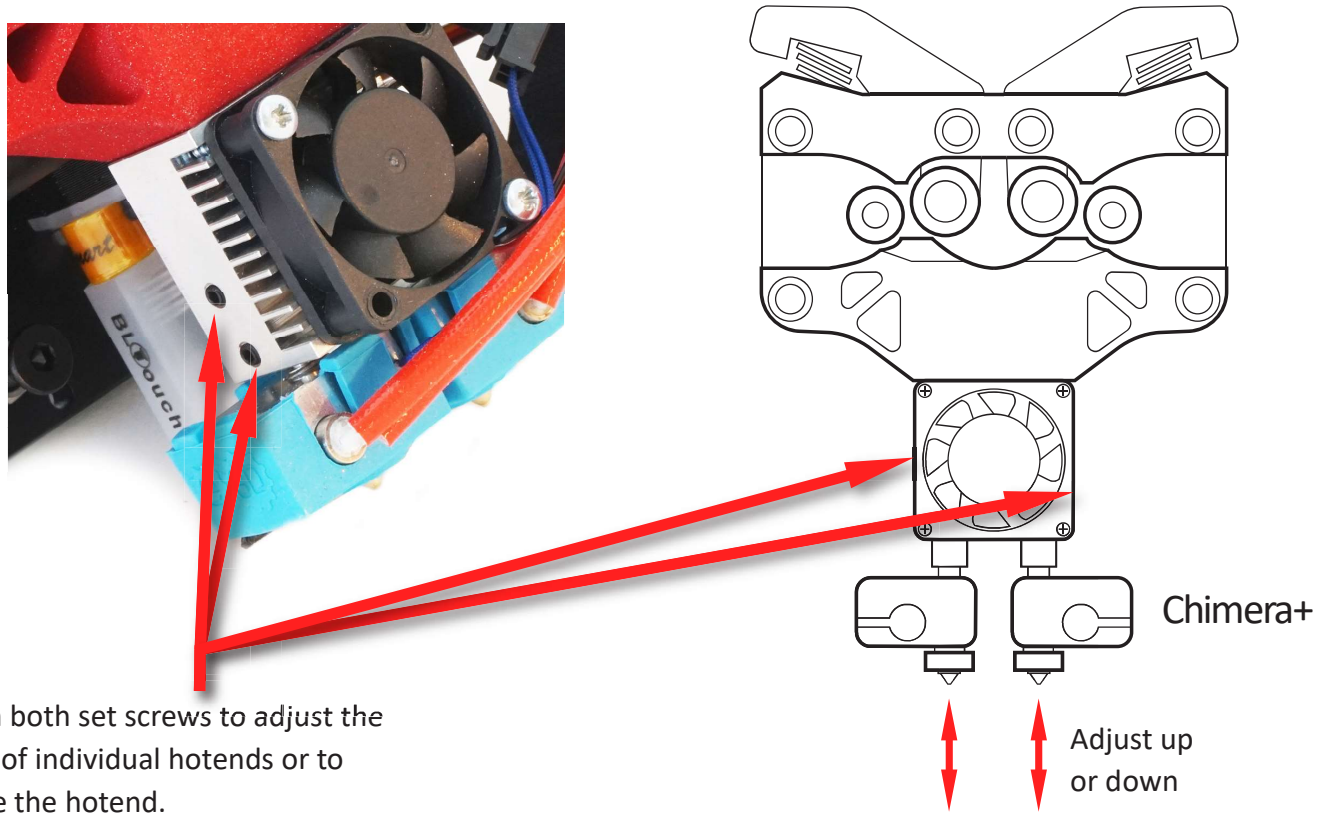
3. Babystep and bed level the printer just like a single extruder.







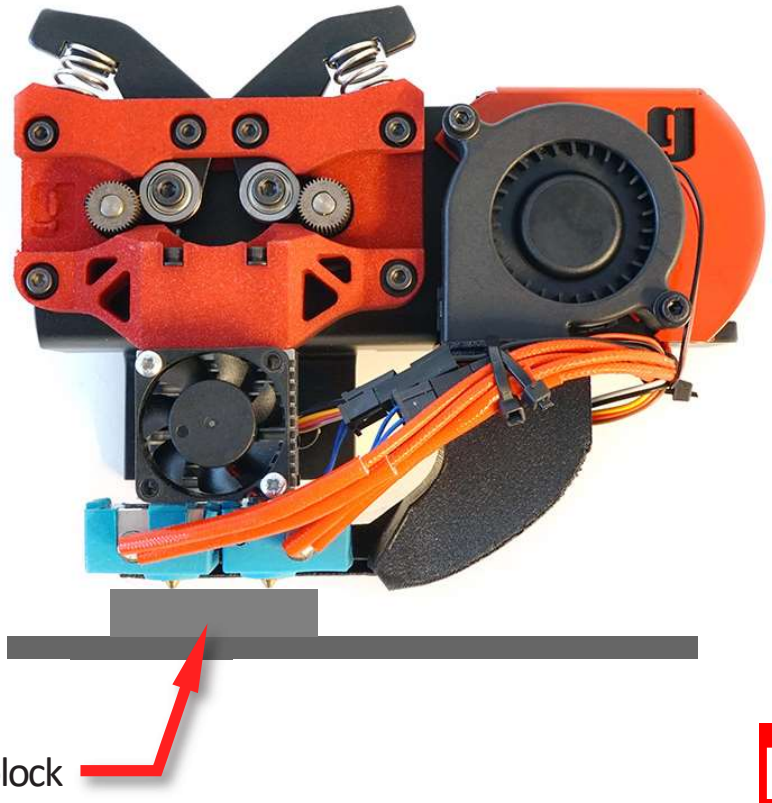
Dual Extruder Printers - Chimera



1. Loosen both set screws to adjust the height of individual hotends or to replace the hotend.

2. Slide aluminum block under nozzle to help adjust height. To raise or lower the extruder go to “Prepare > Move Axis > Move Z > Move 0.1 or 0.01” from the LCD menu.

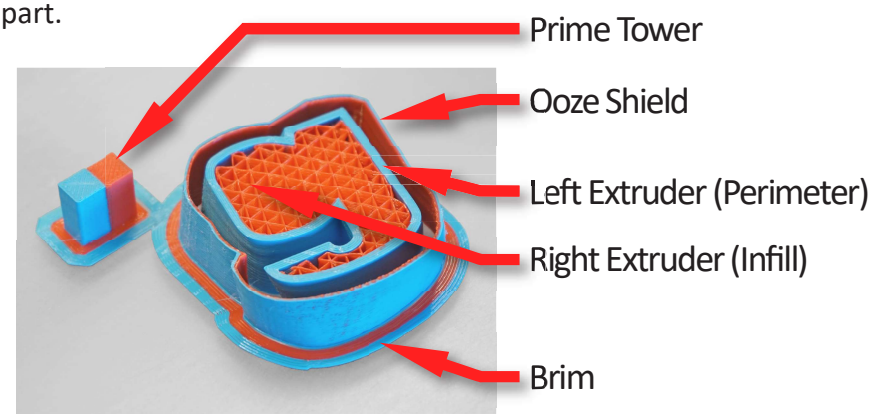
3. Tighten set screws when complete.



Dual-Extrusion Printing Tips (Dual Chimera)

1. Use a "**Brim**" and an "**Ooze Shield**". The Ooze Shield will help clean the inactive nozzle during a print to avoid wiping the plastic on the part.

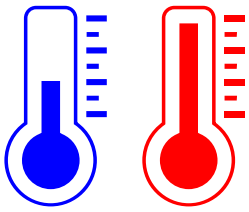
Settings:
Skirt/Brim offset: 0
Skirt/Brim Layers: 2
Ooze Shield: On
Ooze Shield Extruder: Both



2. Use a "Prime Tower". The Prime tower will help material to begin flowing when switching between nozzles.

Cool Inactive Nozzle

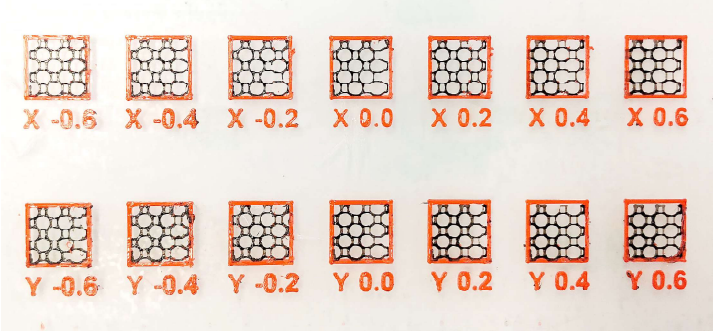
Cool the inactive nozzle to avoid that nozzle leaking during a print. In Cura, search for "**Standby Temperature**" and in PrusaSlicer search for "**Ooze Prevention**".



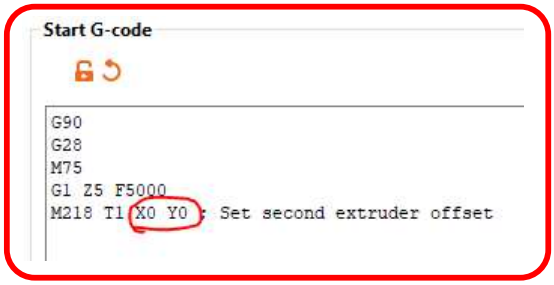
2nd Extruder XY Offset Calibration

You may have to adjust the 2nd extruder offset if you change nozzles or replace parts. The firmware has an offset already built-in so the start gcode offset in the slicing program is set at X0 Y0 and can be adjusted as needed.

On the SD card that came with the printer run the file named "**gCreate Sample Files > Dual Chimera Extruder Files > chimera_dual_offset_calibration_210915.gcode**"

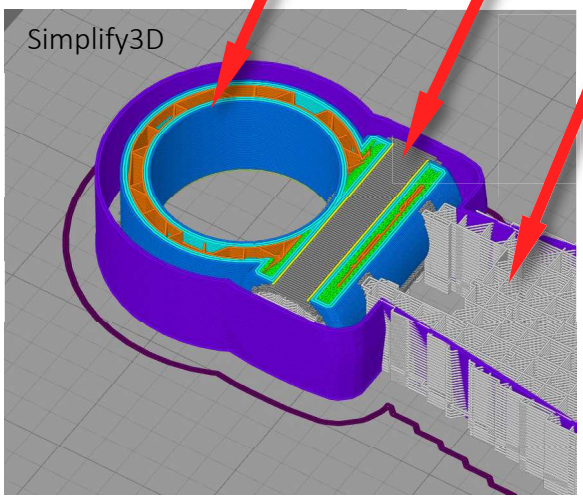


In your slicing software look for the start gcode section and make sure to change the **M218** command to match the new numbers and save the profile.



Water Soluable (PVA) Support Material

You can use the gMax 2 PRO dual extruder to print your part out of one material and the support material out of water soluable PLA material. This support can then be dissolved away in water for a cleaner surface finish.



1. Follow the previous example settings for Dual Extrusion printing.

2. Make sure to increase the interferance layer density of the PVA support material to 100% and the distance from the part to zero. Below are additional settings to keep an eye out for.

PrusaSlicer 2.4 Support Settings

Options for support material and raft

Style:	Snug	
* Top contact Z distance:	0 (soluble)	mm
* Bottom contact Z distance:	Same as top	mm
* Pattern:	Rectilinear grid	
With sheath around the support:	☐	
* Pattern spacing:	3	mm
Pattern angle:	0	°
* Closing radius:	2	mm
* Top interface layers:	5	layers
* Bottom interface layers:	Same as top	layers
Interface pattern:	Rectilinear	
Interface pattern spacing:	0	mm
Interface loops:	☐	
Support on build plate only:	☐	
XY separation between an object and its support:	0	mm or %
Don't support bridges:	☐	
Synchronize with object layers:	☐	

Cura Support Settings

Support

Generate Support	✓	
* *Support Extruder*	Extruder 2	
* Support Infill Extruder	Extruder 2	
* First Layer Support Extruder	Extruder 2	
* Support Interface Extruder	Extruder 2	
* Support Placement	Everywhere	
Support Overhang Angle	50	°
Support Pattern	Zig Zag	
Support Density	50	%
Initial Layer Support Line Distance	0.8	mm
Support Brim Width	8.0	mm
Support Brim Line Count	20	
* *Support Z Distance*	0	mm
* *Support X/Y Distance*	0	mm
Support Horizontal Expansion	0	mm
Support Infill Layer Thickness	0.2	mm
Gradual Support Infill Steps	0	
Enable Support Interface	☐	
* *Enable Support Roof*	✓	
* *Enable Support Floor*	✓	