

gMax Printer 1.5+ Quick Start Guide

v1.0 – 8.20.15

Summary:

This guide should be used after you have received the printer and followed the unboxing instructions. This manual can be used as a “Quick Start” guide to using the printer as well as a refresher on using the printer in the future.

To jump straight to bed leveling check out page 8

Machine Specifications (For Reference)

1. Max Hotend Temperature:	243C <i>(Do not set your max temperature above 235C to prevent a “MAXTEMP” error)</i>
2. Layer Height:	0.080mm (Min) - 0.36mm (Max) IMPORTANT! <i>For best results you must set your layer heights in 40 micron increments. Ex 0.08mm, 0.12mm, 0.16mm etc...</i>
3. Filament Diameter:	1.75mm
4. Firmware:	Marlin (Slightly Modified)
5. Electronics:	Ramps v1.4, Arduino, 1/16” Microstepping

Plug In and Power up

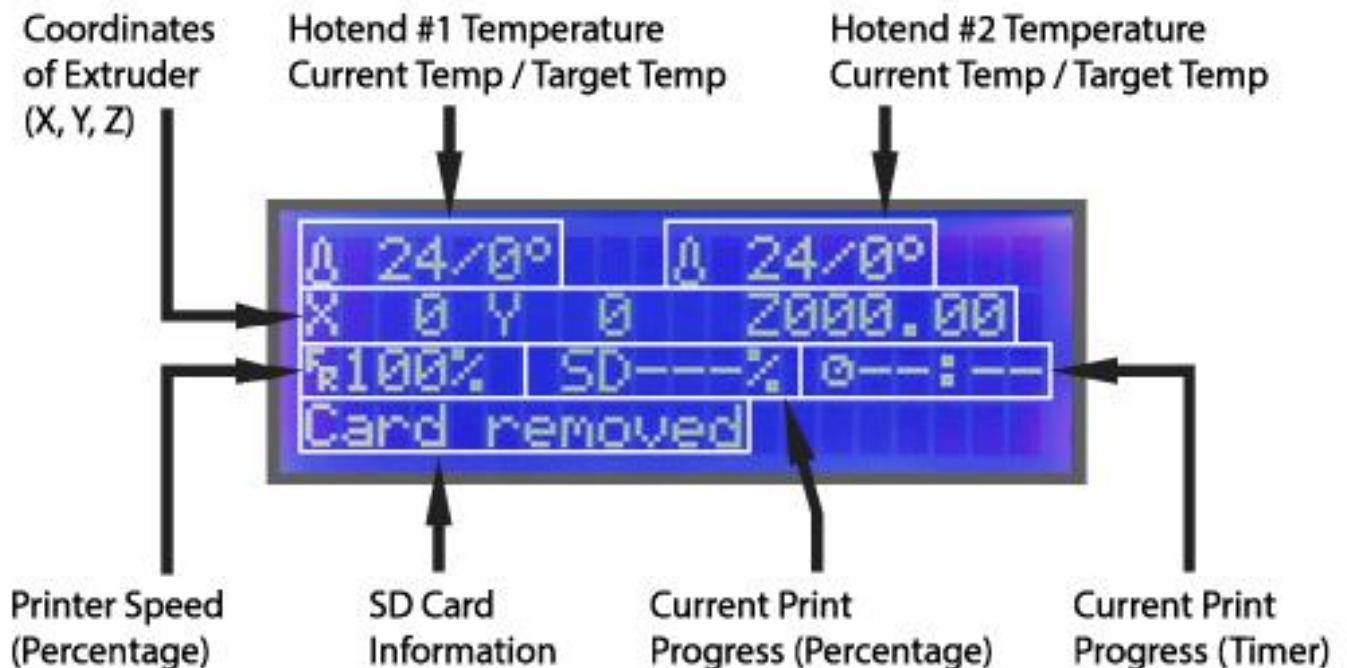
1. Remove the power cord from the accessories box and use the cord to plug the power supply into the wall.

Note: The power supply is a computer ATX power supply and it will have many additional wires coming out of the back. These wires are currently not used and will be secured.

2. Press the power button located beneath the LCD screen to turn on the printer.
3. The printer LCD screen should immediately turn on and display text. Below is a diagram of what each portion of the LCD screen represents. (Note: This is for dual extruders and single extruder setups will not have a “Hotend #2” reading):

Note: If your LCD screen displays the message “MINTEMP” or the Hotend “Current temp” reads 0 degrees, do not proceed and refer to the forum or contact us.

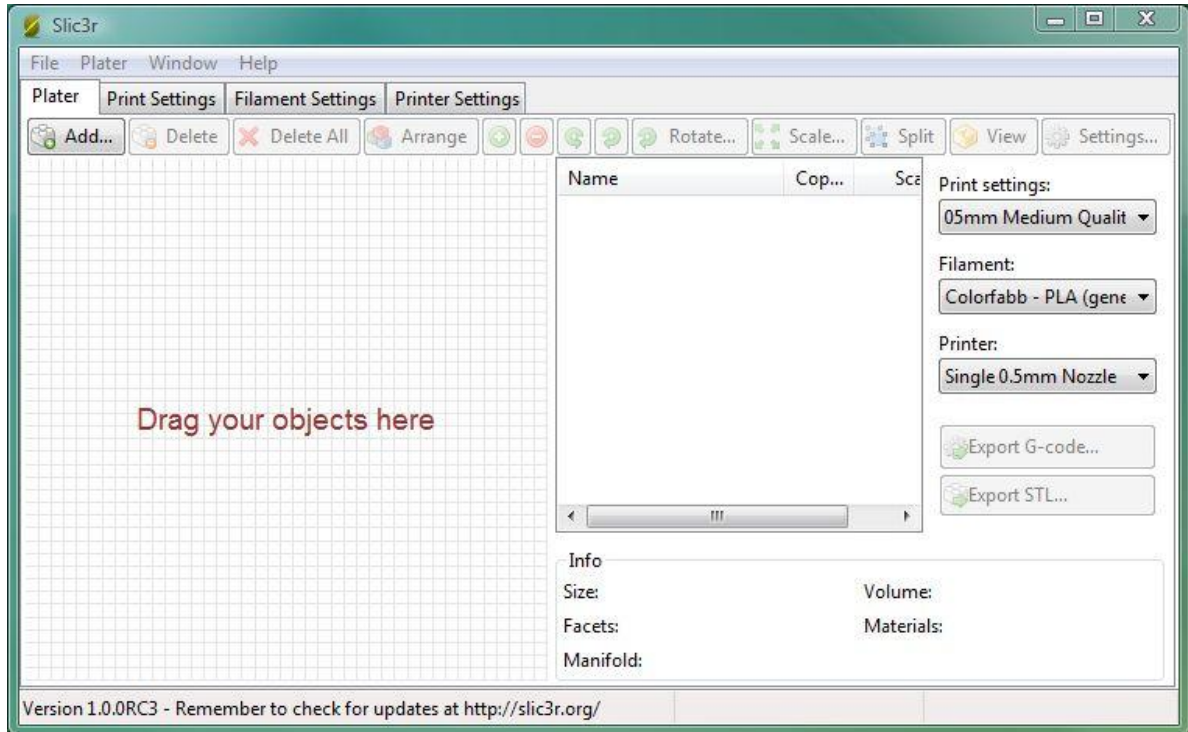
Slicing



1. We must take the 3d model of your choosing and “slice” it into many layers. It is in the slicing program where we define the 3d print characteristics such as infill density, walls, layer thickness and cooling parameters. This is where the magic happens. Each layer is made up of a

path for the extruder to follow. Refer to the manual or the gCreate YouTube channel for information and tutorials on slicing.

Note: We recommend slic3r for its ease of use and constant updates.



2. Slicing your model will result in the creation of a gcode file.

Note: A gcode file is a text file filled with commands used to run the printer. It can be opened with a simple text editor such as Microsoft Notepad and edited if you know gcode. A gcode file is not the same as a 3d model or an .stl file.

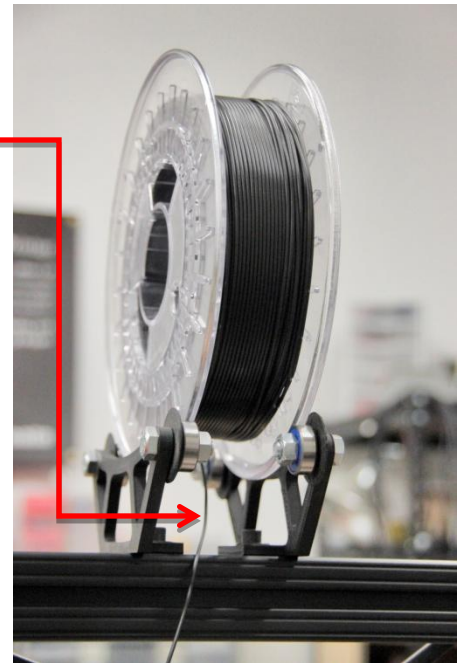
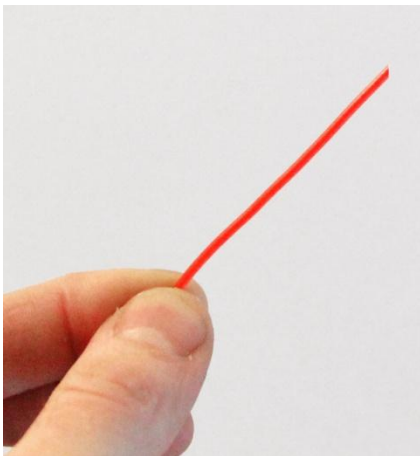
3. Insert the SD card, we supplied you, into the computer and load your gcode file on the card.
4. Properly eject the SD card from the computer.

Insert Filament Into Extruder

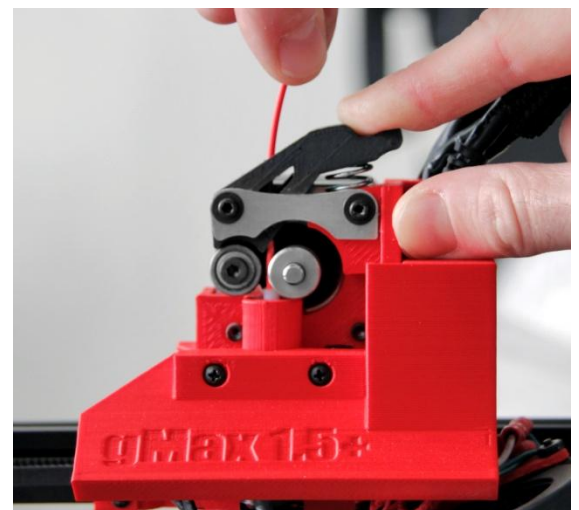
1. Open a spool of filament and place on filament spool brackets on top of printer. **Ensure the filament unspools from the bottom and not from the top to prevent the spool from falling off during a print.**

Should the filament get tangled, this will prevent the spool from falling and it also allows the filament to unwind smoothly.

2. Adjust the filament brackets to match width of spool.
3. Every time you insert filament, cut end of filament at an angle to produce a clean end. It is recommended to straighten 4" of the end of the filament by gently bending it. This will make it much easier to insert into the extruder.



Insert the filament as shown here

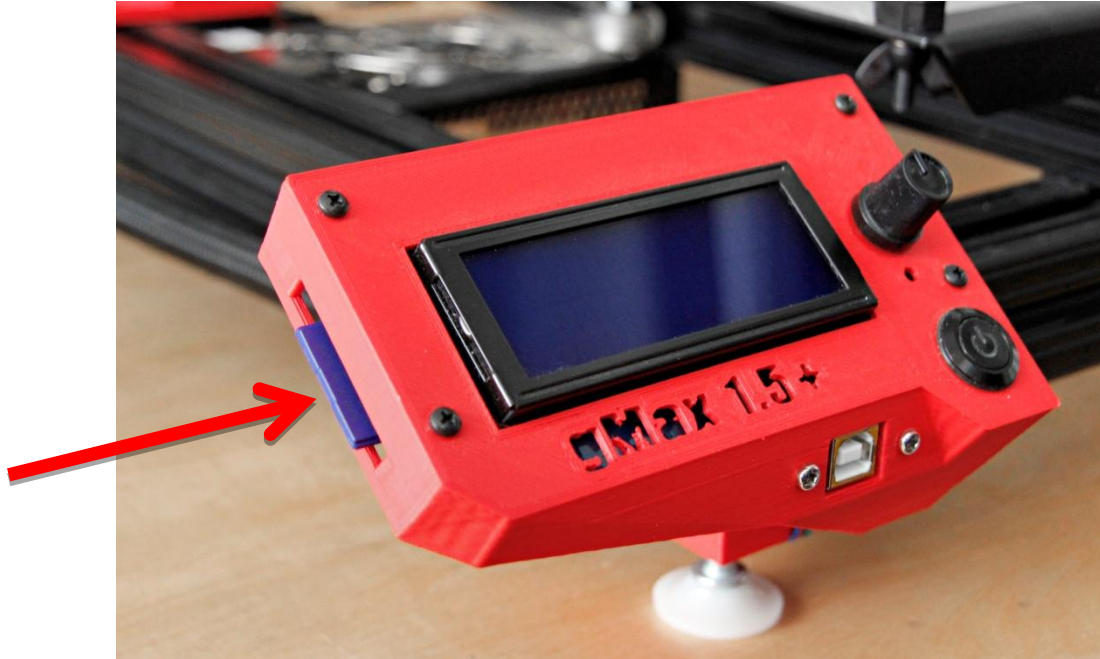


4. Pinch the extruder arm above the spring and insert filament into to hole at top.
5. Slide filament in until you see it enter the top of the clear tube just under the metal drive gear. You may have to twist the filament as you push it in to get it to enter properly. The filament should slide in very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

Note: When changing filament colors/types, you may need to purge the old material out of the nozzle. When the nozzle is hot and while pinching the extruder arm, push material through the nozzle to purge the old material.

Choose a File to Print

1. Slide the SD card (**upside-down**) into the left side of the LCD screen. It slides in just below the circuit board and it should push-click into place. You should also push-click when removing it.



2. Using the LCD screen, the bottom line should now say “Card Inserted”. If not, try removing the SD card and reinserting it.
3. Push in the LCD screen knob (this is your accept button) and scroll down to “Print from SD” and push in the knob again.
4. Select the file you wish to print and push the knob.

Your first Print

1. First, you will see the extruder move to the left until it hits the left (**x-axis**) endstop.
2. Secondly, the bed will move forward until it hits the front (**y-axis**) endstop.
3. Finally the z axis will lower until it hits the left (**z-axis**) endstop attached to the aluminum frame.

Note: If the printer stops at any of these steps or misses the endstop, refer to the troubleshooting portion of the forum for help or contact us.

4. Once the extruder stops, this is considered the printer's "Home" position and it will "re-home" every time you start a print. You can also home the printer using "**Prepare > Auto Home**" from the menu.
5. If you are using the latest slic3r configuration files, you will then see the z-axis lift 10mm and stop while the hotend starts to heat up. This will prevent the hotend from melting the acrylic bed as it heats up. If you are using Simplify3D the head will not lift so it's a good idea to always raise the extruder before starting a print to prevent damage to the bed.
6. The hotend will then continue heat up, which you can see on the LCD screen, until the current temperature reaches the target temperature. If it doesn't heat up refer to the troubleshooting area of the forum or contact us.
7. The heat will reach the target temperature after about 20-30 seconds.

Printing Starts...

1. The extruder will then move to the center of the bed and start extruding plastic. As the print begins, you may have to slightly adjust the height of the extruder during the first layer (Refer to the chart on the next page).

You can do this by using the LCD screen. Quickly go to "**Tune > Babystep Z**" then turn the knob. This will raise or lower the extruder by very small amounts so you can fine tune the first layer height. Clockwise raises the extruder and counter-clockwise will lower it.

2. If you notice the x-axis doesn't seem level and one side is higher or lower you can fix this by manually turning one of the z-axis lead screws. Clockwise will raise that side of the x-axis, or counterclockwise will lower it.

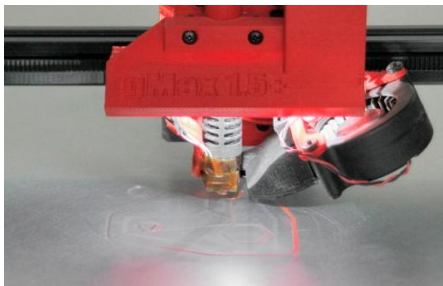
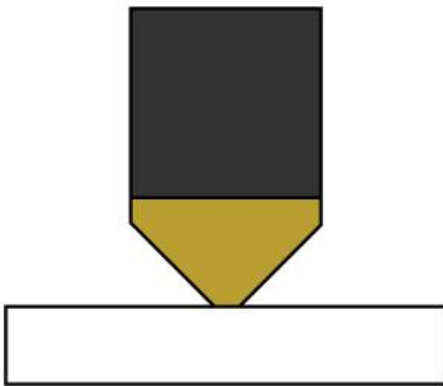
Note: Using a z-lift (slic3r setting) is not the best way to adjust your z-axis height. You should adjust it manually and adjust the z-axis endstop set screw to guarantee future prints will be level.

3. The goal is to make sure the plastic is extruding flat on the bed. If you can't see any plastic or the motor is skipping (a ticking sound), it may be too close to the bed in which case you should raise the extruder. Look closely at the nozzle tip as you turn the motors to see the height changes.

Note: You may need to adjust the first layer as you continue to use your printer to ensure the print starts out perfectly flat but you should always adjust the z-axis leveling screw, before starting the next print to ensure it is flat.



First Layer Print Diagram

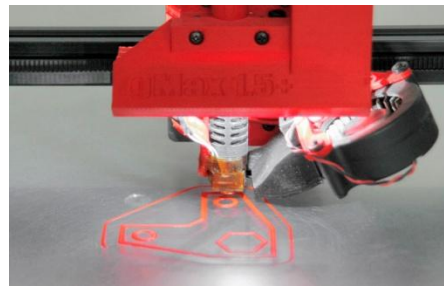
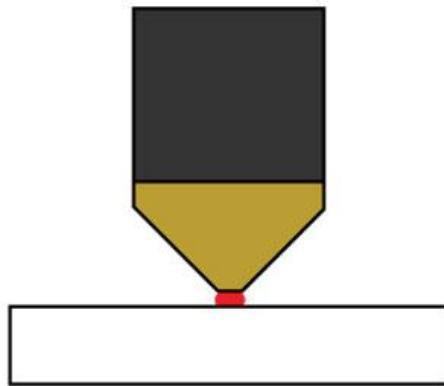


Nozzle is too low

Nozzle is dragging on print bed and not allowing the filament to be pushed out. If this print continues the bed may become damaged.

Solution

Raise the nozzle (outlined above) and make sure to turn the z-axis set screw (outlined below) to set the new z-axis zero height.



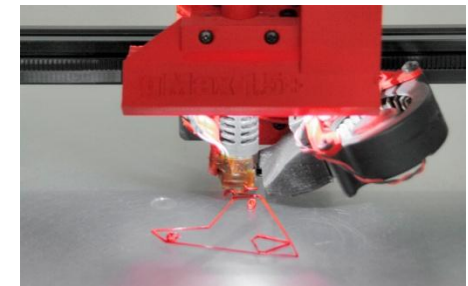
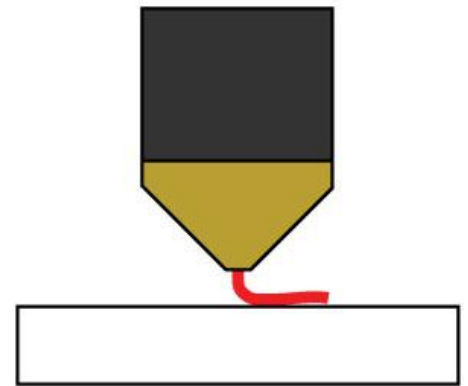
Nozzle is just right

Nozzle is slightly above print bed allowing filament to be evenly spread onto the surface. The filament should be a clean continuous line throughout the whole print.

Solution

Nothing.

Keep an eye on the print and let it finish.



Nozzle is too high

Nozzle is too far from print bed causing the filament to not adhere and instead it will extrude into the air. You will either see a blob form at the tip or stringing in the air.

Solution

Lower the nozzle (outlined above) and make sure to turn the z-axis set screw (outlined below) to set the new minimum z-axis height.

Next Steps

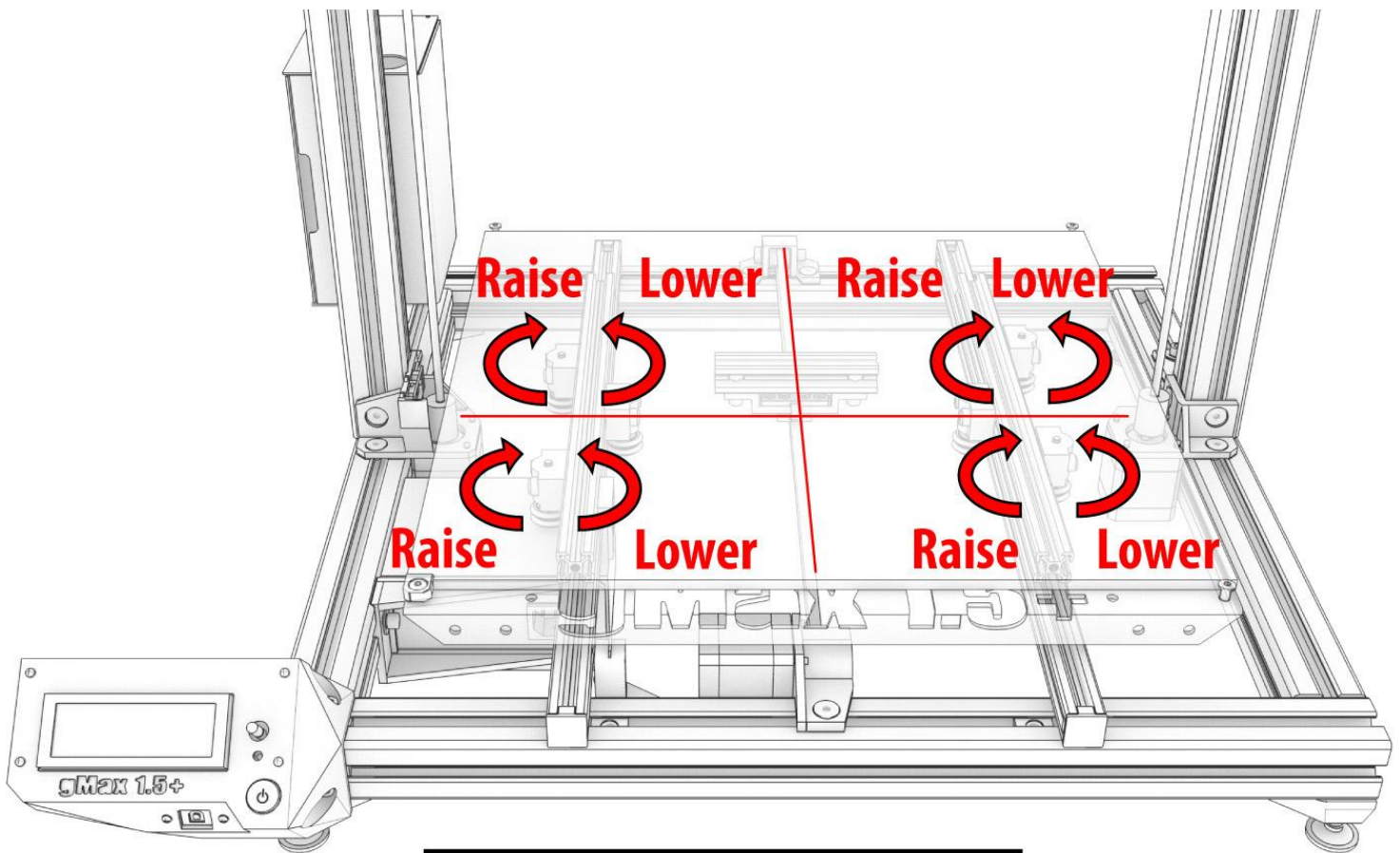
Make sure to keep an eye on your print until it finishes. Every file you print has its own characteristics and sometimes it will surprise you what happens. Also the beginning of the print is critical to the final results and you should make sure it adheres correctly.

Note : As print starts, keep an eye on the filament spool to ensure it does not bind or wrap around the brackets. This is especially true for new spools of filament.

Leveling The Bed

If you notice your bed may not be level, use the guide below to adjust the height of each v-slot wheel which will raise or lower the respective corner.

To level bed, turn each v-slot wheel, under the bed, using an 8mm open wrench or 4mm hex key. Use the large bed leveling print to determine if each corner is high or low and adjust accordingly. Make sure to adjust one corner at a time.



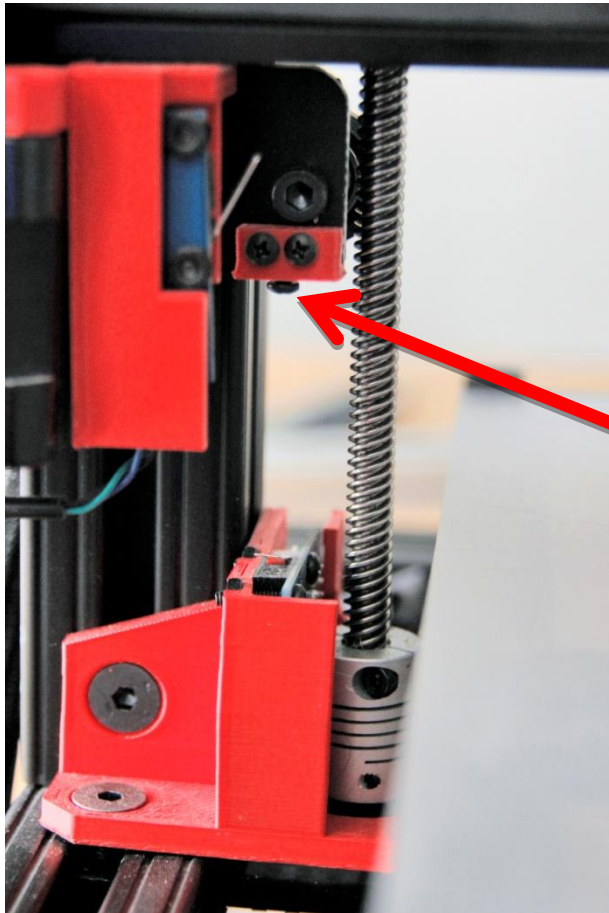
For Each Corner:

Turn Clockwise to Raise Corner

Turn Counter-Clockwise to Lower Corner

Adjust Minimum Z-Axis Height

You should also adjust the z-axis endstop screw to set the new zero point if you ever have to adjust the 1st layer height. The zero point is the lowest point the extruder will go and this will be the height in which the print will start. This should be done after adjusting your first print.



Setting the z-axis screw should be done any time you have to manually adjust the 1st layer height. Raising or lowering this screw will set the z-axis height for your next print.

Note: To access the set screw you must raise the extruder by 5"-6". This can be done by using the LCD screen and going to "Prepare > Move Axis > Move 1mm > Move Z". Then simply turn the knob.

Tips During Your Print

You have some time now so why not read these tips.

1. Always keep an eye on your print and check on it often especially if it's the first time running the file. 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 flow during a print. Note these settings are not permanent and they will be overridden when the printer reaches a new gcode command.
4. The fan speed is on a scale from 0-255 with 255 being equal to 100%. Also the fan will not operate below 80 since the power is too low.
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 using M600. When clicked the print will pause, the extruder will raise and move out of the way and it will retract the filament. Then slide in your new filament and push the knob.

Note: You can use this method to add multiple colors to your print by changing the colors at certain layers. You can also add the M600 command at any point in your gcode file to trigger the filament change command at specific spots in the print.

Removing the print. David vs. Goliath

After the print finishes you can immediately pop it off (unless it's very small in which case you may want to let it cool for a few minutes).

If the first layer printed correctly you should see no curling at the edges and the print should be easy to remove.

USING EXTREME CAUTION take the supplied scraper and gently tap at the edges of the print.

We have found this tapping action is much safer and much easier to remove a model rather than pure brute force and it is also better for the print bed. 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.

Remember the scraper is very sharp and 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.

Once the print is removed, inspect it to see how it printed using the current settings. You can record what happened with each print and over time it will become intuitive.

If the print has any stringing or globs stuck to the side you can carefully use a blade to remove the plastic or in some cases it just flakes off.

Note: A heat source, such as a lighter, works great to heat the plastic to fix any problem areas and to smooth very sharp edges. Use caution when using the lighter because smoke particulate may stain the model. Remember you just want the heat from the flame not the flame itself to touch the model. This method will also remove any slight stringing.