



MAKERGEAR

[Beginner's Guide to Getting Started with the M2](#)

Congratulations on your purchase! You've made an excellent choice for your 3D printing needs! 😊

If you're new to the world of 3D printing, there is a lot to take in at first, but there are also a lot of resources to help you if you get stuck. This first page tells you how to get to that help. To make the resources immediately available, two of the first things you'll want to do are register your **M2**, and join the forum.

To Register Your Printer:

Send an email with your **Name**, **Address** and the **Serial Number** from the printer to:

Registration@makergear.com

(The serial number is located above the bar code, on a sticker on the back of the machine.)

To Join the MakerGear Users Forum:

<http://forum.makergear.com/>

The forum is your best place for getting a quick answer. It has a lot of users who've been around the block a few times. (It's a safe bet that you're not the first person to encounter whatever problem you run into.) 😊

Or, if you have no issues to resolve, feel free to post there and just introduce yourself....it's a friendly bunch, and every one of us had to get started the same way.

*(Usual forum rules apply – keep it friendly, no bashing, hate-speech, inflammatory rhetoric, actively promoting other printers, etc. MakerGear has not expressly forbidden it, but you'll run the risk of ticking off a lot of people who think the **M2** is the “bee's knees”, and I'll tell you up front, they'll defend it. Final discretion rests with the moderators. If they cause a ruckus, or have the potential to do so, you can expect your posts to start disappearing, so keep it polite.)*

To Contact MakerGear Technical Support:

You can also contact MakerGear directly for support, although your first resource should probably be the forum, since you'll likely get a quicker response from the larger group.

MakerGear tech support is via email....and they are generally very quick to respond.

If you need to contact MG support for assistance with your machine, follow the instructions at the link below:

<http://www.makergear.com/pages/support-1>

Contents

❖ Tools List	04
❖ Quick Start Guide –Single Extruder	
• 1. Setting up the Machine/Installing Driver	05
• 2. Checking the Level	05
• 3. Adjusting the Level	06
• 4. Preparing the Plate	08
• 5. Establishing Communication with Your Slicer Software	09
• 6. Checking and Setting the Z-Endstop	10
▪ Checking the Z-Endstop Gap	
▪ Cold Setting the Z-Endstop	
▪ Hot Setting the Z-Endstop (all nozzle styles)	
▪ Hot Setting the Z-Endstop (V4 style nozzle)	
• 7. Preparing to Print	14
• 8. Printing	15
• 9. Calibrations	16
▪ Calibrating the Z-Offset	
▪ Calibrating the Tension on the Filament Drive Screw	
▪ Calibrating the Extrusion Width	
❖ Quick Start Guide – Dual Extruder M2	
• Getting Started with the Dual Extruder System	17
• Adjusting the Right Nozzle Gap	18
• Starter S3D Profiles for the Dual	19
❖ Tips for Printing PLA	20
❖ Troubleshooting Tips	22

Tools List

Although you *can* get by without them, the tools below just make it easier to use a 3D printer.

1. **Metric Feeler Gauges** (Amazon) - They are invaluable in setting the Z-stop.
2. **Calipers** - Needed for calibrating the extrusion widths and heights.
3. **Hairspray/Washable Glue Stick/Wide Blue Painter's Tape** - Any of these can be used to stick the prints to the plate and you might prefer one method over the other.
4. **Sprue (Wire) Cutters/Tweezer Nose Pliers** - For removing support and cleaning up your prints.
5. **Burr Grinders** (Amazon) - Are also great for removing junk if you have a Dremel tool.
6. **Fine Grade Sand Papers** - For smoothing your finished prints.
7. **Eye Protection** - **Always wear safety glasses or goggles when you are cleaning up a print.**
(It's very easy for a snapped bit to fly up and hit you in the eye, even if you wear regular glasses.)
8. **Thin Bladed Spatula/ Honed Putty Knife/Single Edge Razor Blade with holder** - For prying the print off the glass. *("Cricut" makes a good one.)* (Amazon)
9. **Gloves** - Useful when prying a print off the glass. *(You can cut yourself when the print lets go.)*
10. **Painter's Masking Tape** - Used to hold the Z-knob in place while leveling and setting the Z-stop.
(There are printable options for Z-knob holders on Thingiverse as well.)
11. **Spare Hot End, Nozzle and Plate** – Handy to have because when you need them, it's *immediately*.
12. **Optional Bed Surface** - As opposed to the hairspray and spatula route, you might want to try one of the bed surfacing options below:

PEI surface - Pros: Releases easily when cool, holds beautifully under heat without adhesive, stays perfectly flat if it is applied correctly, and is easy to use for beginners. **Cons:** Has to be applied to the borosilicate glass with sheet tape. *(Think screen protectors.)*

Zebra plate - Pros: Heavy flexible plate with copper inserts on both sides. Replaces the glass, no application needed. Holds beautifully without adhesive for certain filaments. Excellent release for large prints due to the flexible nature. **Cons:** Can warp over time, or at higher heat. Not the best choice for beginners due to flex. Cannot be used for tall ABS prints.

MIC6 tooling plate - Pros: Aluminum replacement plate - distributes heat all the way to the edges of the plate for better adhesion. **Cons:** Does require adhesive. Plate has to be ordered, cut to size, and the faces bead-blasted (if desired) - you can't buy the correct size off the shelf. There may be extra costs for custom sizing.

*(There is more information on what the surfaces do, how to apply them, and what makes them better for certain prints in the **Modifications** section of the forum.)*

The Quick Start Guide

Single Extruder M2

1. Setting up the Machine:

Choose a solid, stable, desk, table, or workbench to set up your machine.
(These things vibrate when they print.)

Because the **M2** changes slightly as improvements are incorporated into the base model:
Carefully read through all of the current instructions provided by MakerGear (included in your box) before assembling.

You can also direct download your RAMBo **Windows Driver** before assembly, to have it ready. It is available at the page linked here:

<http://makergear.wikidot.com/m2-firmware>

Assembly is easy, and generally all that is required is to clip a few ties, plug in a connector or two, tighten a few screws to attach the spool holder, and load your driver (if you use Windows). Keep in mind, the User Manual and Wiki might not get updated as often as the instructions that came in your box. It's a rapidly evolving field, and MakerGear is continually testing and incorporating the latest improvements. *(If they don't agree, follow the current instructions you received in the box.)*

There is an excellent step-by-step demo for the basic setup process here:

M2 Initial Setup: <https://www.youtube.com/watch?v=vCKDJIZKubA>

2. Checking the Level:

If you purchased a pre-assembled (non-auto-leveling) machine, it has already been leveled by experts, and the packaging protects it very well during transport. It is highly unlikely that you will have to adjust the level, but it's not a bad idea to check it once you have assembled the machine, before you have extruded through the nozzle. *(If you have extruded, clean any drips off of the nozzle with the tweezers while it is cooling. You need a clean nozzle surface for this test.)*

You need:

Masking Tape

Sheet of copy paper

Pull up a chair in front of the machine. With the **power off** to the machine, **center the nozzle over the bed**. With your hands underneath the **Y-axis rail**, (not the spider), **slowly** lift the bed until it is a couple of inches under the nozzle.

Sit in front of the machine, get comfortable, and **position your eyes level with the bottom of the nozzle**.



Turn the **Z-axis knob** to finish bringing the bed up until it almost touches the nozzle.
(You should still be able to see a small gap at eye level, about the thickness of the sheet of paper.)

When the nozzle is close enough, use the masking tape to tape the Z-knob into place, to keep it from turning. (It will hold the bed at that position until the tape is removed.)

Without taking your eyes off of the gap between the nozzle and the bed, (using your hand) **slowly** move the print head all the way to the left, all the way to the right, and then back to the center along the X axis.

If the bed is un-level from left to right, you will see the size of that gap change. (**Stop** if the nozzle hits the bed, you might need to lower the bed a fraction to run your test. Re-center the nozzle, and re-tape the Z-knob.)

Perform the same test for the front-to-back direction by again, centering the print head, and then **slowly** moving the bed forward and backward along the Y axis (by hand). Keep your eyes on the gap at eye level while you do it.

Note: After you have eyed the gap, check it by sliding the sheet of copy paper between the nozzle and bed at various points along the axes to check the resistance. The paper should slide in and out with the same slight resistance at all points.

If the size of the gap doesn't change by more than 0.1 mm (roughly the thickness of the sheet of paper) from the center to either side of the bed, it's in good shape. If the gap changes by more than 0.2 mm (two sheets of paper) it's going to need to be adjusted.

To help keep your plate level for a long time:

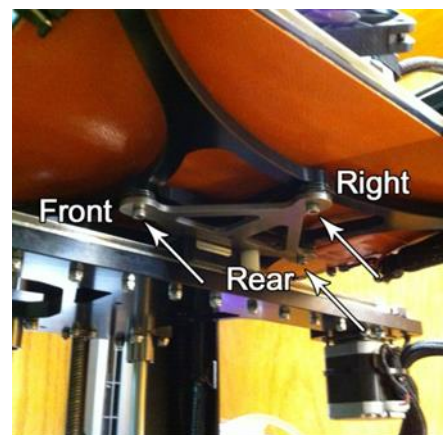
1. Do not lift the bed using the spider or the bed plate; always lift from underneath the Y-axis rail.
2. Don't try to manually lift the bed with the stepper motors engaged. (It won't budge.)
3. Don't pry stubborn prints off of the plate while it's still on the bed. Remove the plate first.

Note: Because glass is rarely perfect, you might see some isolated low spots or bulges in the glass as you move the print head and the bed. If the bulge or low spot is small enough, it shouldn't affect your printing – what you are looking for is an overall lean in one direction over another.

3. Adjusting the Level:

It's very easy to quickly make minor adjustments to the bed level by turning the three adjustment screws on the triangular plate (located under the bed) that the spider sits on, if you understand how turning each of the screws affects the plate.

The diagram on the next page is very handy for explaining it, and it's a good idea to print out a copy and keep it near the printer for reference.



Bed Leveling Cheat Sheet

Rear Adjustment Screw

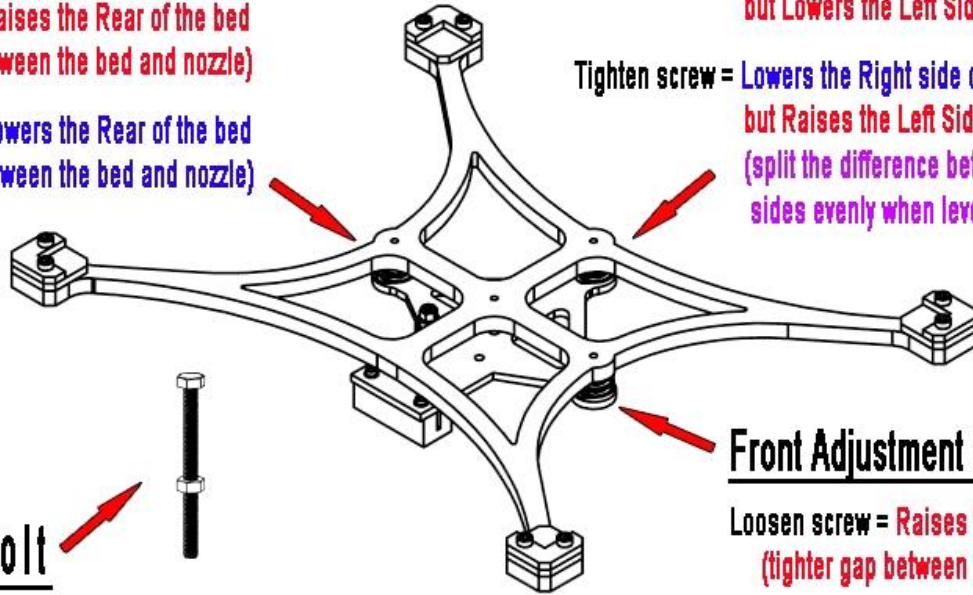
Loosen screw = Raises the Rear of the bed
(tighter gap between the bed and nozzle)

Tighten screw = Lowers the Rear of the bed
(Larger gap between the bed and nozzle)

Right Adjustment Screw

Loosen screw = Raises the Right side of the bed,
but Lowers the Left Side of Bed.

Tighten screw = Lowers the Right side of the bed,
but Raises the Left Side of Bed.
(split the difference between the
sides evenly when leveling.)



Z-Stop Bolt

Loosen Bolt = Moves the entire Platform/Bed away
from the nozzle.

Tighten Bolt = Raises the Platform/Bed closer
to the nozzle.

Front Adjustment Screw

Loosen screw = Raises the Front of the bed
(tighter gap between the bed and nozzle)

Tighten screw = Lowers the Front of the bed
(Larger gap between the bed and nozzle)

Diagram by sthone

Use the **Front Adjustment Screw** and **Rear Adjustment Screw** to adjust the size of the gap along the Y axis. (Front to back.)

For example: Say the front of the bed is too low relative to the rear of the bed, (*the gap is larger at the front of the machine than at the rear*), you would position the nozzle at the area of the largest gap, watch it at eye level, and slightly loosen the **Front Adjustment Screw** to bring the plate up until the gap at the front is the same size as the gap at the rear of the plate. Check the gap (all the way forward and back) again. You never want to make big adjustments to the screws, use tiny increments and check the results, then adjust more if needed.

To adjust the gap from left to right on the plate, (along the X axis), you use only the **Right Adjustment Screw**. That screw causes the plate to rotate relative to an axis that runs between the other two screws, so if you tighten the gap on the right side of the bed, the gap grows larger on the left side of the plate.

To correctly use this screw, position the nozzle directly over the widest part of the gap, refer to the diagram for the correct direction to turn the screw, keep your eye on it and either *slightly* tighten or loosen the **Right Adjustment Screw** to reduce that gap **by half**. Check the gap all the way again, choose the next highest gap, and adjust once more if needed. Again, **only close half of the gap**.

For more information:

A more detailed description (with photos) of **Setting the Z-Stop and Bed Leveling** is available on the MakerGear Forum: <http://forum.makergear.com/viewtopic.php?f=3&t=2783>

And a video of the more expert leveling process is available at the link below:
<https://www.youtube.com/watch?v=R53wWwktH0s>

Note: The diagram shown above is a simplified view of the leveling procedure, designed to take care of minor adjustments before they become major ones, and it assumes starting with a basically level bed. If you have a large variance at any of the corners, or have let the bed get severely out of level, you will need to read the more detailed descriptions and watch the video for a better understanding of the leveling forces at work. (There are actually 3 axes that the plate rotates through, and major adjustments would need to be handled according to the video instructions.)

Remove the tape from the Z-knob, and **gently** lower the bed (*supporting it*), when you are done.

4. Preparing Your Plate:

If you are planning to use the Polyimide (Kapton) tape on your glass plate, remove it from the bed clips and apply the tape at this time. (The tape is used to improve print adhesion for certain filaments, and can be used in conjunction with other adhesives if needed. It can also level out a slightly low spot in the glass, and it provides protection from possible chipping of the glass when removing prints.) Same goes for blue Painter's Tape, if you intend to try it in lieu of adhesive. Evenly apply 3 strips of tape along the long edge of the plate, (with no overlap), before the next test. If you have purchased a separate surface (such as PEI material) you will also want to apply it before performing the next check.

*Note: And this is important....Any time you change the height of the print surface relative to the nozzle, you will need to re-set the **Z-Endstop**. If you set the Z-stop initially using painter's tape, and then change out the tape with fresh painter's tape, that is not usually enough to warrant re-setting the Z-stop.*

*But if you switch from painter's tape to Kapton tape, that **is**. Switching to a different plate, or applying a different surface to your existing plate, is going to change the gap between the nozzle and the print surface. Keep that in mind if you use multiple plates for different things on your machine.*

The use of adhesives alone is not enough to affect the gap.

5. Establishing Communication with your Slicer Software:

At this point, if you have not already done so, you will want to establish communication with your slicer software or machine control software, since you will be using the controls to heat the nozzle and bed for the next test. (Setting the **Z-Endstop**.)

Basically, the way the 3D printing process works is that a 3D model (created in any appropriate 3D modeling program) is “sliced” in another program, which can look at that model and automatically convert it into a series of thousands of “**G-Code**” commands that tell the printer when to turn on the fans and the nozzle, how much to extrude, how far to travel before turning, when to stop printing, etc. This slicer software can be used to either send instructions to the printer over USB, or by saving a G-code file with the instructions to an SD card, which is then inserted into the card reader on the printer. (*In an assembled **M2**, the SD card is shipped inside the card reader on the machine.*)

The M2 can be used with the (recommended) **Simplify3D** slicer software, or the free open source option of **Slic3r** through a **Printrun** (aka: Pronterface) or **Repetier** interface. MakerGear has nothing to do with the development of any of these 3rd party programs, so you’ll need to access the individual web sites for specific usage instructions. (*There are users on the MakerGear forum familiar with both methods.*)

Simplify 3D:

Purchase and install: <https://www.simplify3d.com/>

Setup Guide: <https://www.simplify3d.com/support/hardware-setup-guides/makergear/>

Tutorials & Guides: <https://www.simplify3d.com/support/>
<http://jinschoi.github.io/simplify3d-docs/>

Printrun/Pronterface:

Download and install: <http://koti.kapsi.fi/~kliment/printrun/>

Setup Instructions: <http://makergear.wikidot.com/m2-getting-started>

Slic3r:

Download and install: <http://slic3r.org/>

Tutorials & Guides: <http://manual.slic3r.org/>

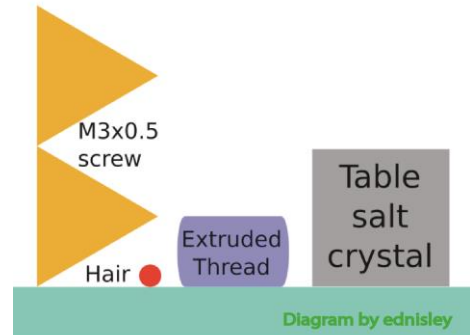
After you have downloaded and installed the software, (*following the setup instructions at the above links*), turn on the **M2**, establish a connection with the software, and practice using the **Jog Controls /Machine Control Panel** to heat the nozzle and bed, (**215°C/60°C**), move the plate up and down (by a few millimeters), and move the print head left and right (by a few millimeters). At this time, **avoid** using the **Home Z** button on the Jog Controls. (You will be checking/setting that in the next step.)

(It’s also not time yet to load the filament, since if you extrude through the nozzle, you’re going to have to heat it up, retract the filament, and clean the nozzle off before continuing.)

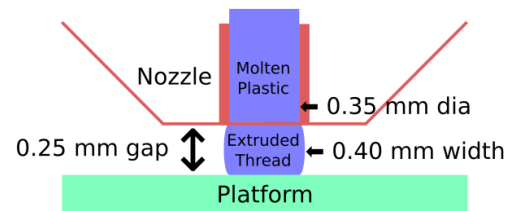
Once you get comfortable with basic manipulation of the controls, continue to the next step.

6. Checking and Setting the Z-(End)Stop: (It's *all* about the gap!)

In 3D printing, **the most critical setting** that you have to get absolutely right is the size of the gap between the tip of the nozzle and the print surface when you lay down your first layer. If that gap is too large, you'll be squirting plastic into thin air, your filament won't stick to the plate, and you might experience poor bonding. If it's too small, you'll be smearing plastic across the plate, squishing it out the sides, and potentially jamming the nozzle.



The **Z-EndStop** is set once before you begin printing, to lock the **Z-Home** value, so that the gap between the nozzle and the surface of the plate comes out to just at or below one layer height (**0.20 - 0.25 mm**) while the bed and nozzle are at the correct temperature for whatever filament you are printing. *(Additional fine tuning is required of course, since you will be printing a lot of different filaments at different temperatures.)* But you want to get as close as you can to that ideal with the **Z-Home**.



On a pre-assembled **M2**, the **Z-stop** is set once for you, using a Kapton-coated glass plate, and the cold-set method.

(If the plates now come with the Kapton tape pre-applied, then the test was probably performed with your plate. You will just need to check the Z-stop gap to make sure the bolt didn't shift during shipping, and to make sure the bolt is tightened.)

The gap can be approximated by using a standard business card, while the machine is cold.

For more accuracy you want to set the gap by heating the bed and nozzle to printing temps and using a **Metric Feeler Gauge (0.15 mm to 0.20 mm)**.

Either way, after you have checked or reset the **Z-stop**, you will fine-tune the gap using a control in your slicing software called the **Z-Offset**, which you will determine later in the final Calibrations step.

Checking the Z-EndStop Gap:

(Do not press down on the bed while checking the gap.)

Quickest method:

Home Z while the **M2** is turned on but not heated, **center** the print head, and slide a business card in and out of the gap between the nozzle and the bed. It should just fit under the nozzle without binding, while perfectly flat.

Better method:

Heat the nozzle to **215°C** and the bed to **60°C**, **center** the print head, **Home Z**, and slide a folded sheet of copy paper in and out. The folded paper should just slide in and out without binding.

Best method:

Heat the nozzle to **215°C** and the bed to **60°C**, **center** the print head, **Home Z**, and use a set of metric feeler gauges to determine what the gap actually is, by sliding various gauges into the gap and seeing which one fits perfectly. You want the gap to fall into the **0.15 mm to 0.20 mm** range.

Once you've checked the gap, use one of the methods below to adjust it, if needed.

Cold Setting the Z-Endstop: (All Nozzle Styles)

MakerGear has a video demonstrating the cold-set calibration process:

MakerGear M2 Z-Endstop Calibration: <https://www.youtube.com/watch?v=j0Cz6jPPECM>

Hot Setting the Z-Endstop: (All Nozzle Styles)

This first method works for all nozzle styles.

(It is a *lot* easier to set the gap for the V4 style nozzle (*shipping standard on machines as of 1/1/16*) so if you have that nozzle, and a set of metric feeler gauges, you can skip to the next section.)

You will need:

A **0.15 - 0.20 mm Metric Feeler Gauge**; or one sheet of copy paper, folded (approx. 0.20 mm when doubled)

Masking Tape - to keep the Z-knob from turning

A *narrow* 7-mm wrench or a pair of needle nose pliers

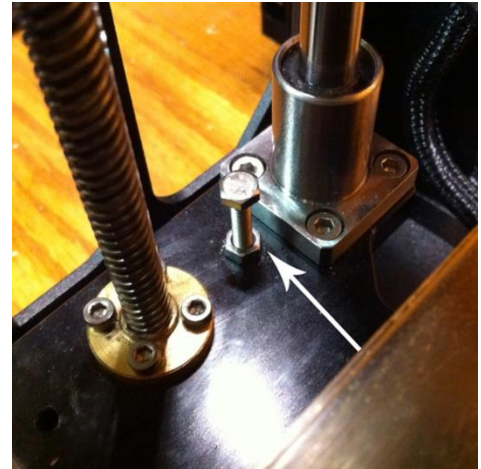
1. Position the nozzle in the **center** of the bed.
2. Make sure the heated bed plate is seated flat between the rubber corners.
Clip the prepared glass plate to the bed.
(If you have extruded through the nozzle, heat the nozzle, retract the filament completely, and clean off the nozzle tip with tweezers. You need a clean flat tip on the nozzle before proceeding.)
3. Set the bed temperature to **60°C** and the extruder temperature to **215°C**, using the **Jog Controls** or **Machine Control Panel**.

4. Remove the spool of filament from the holder - it will be in the way.

5. Loosen the nut on the **Z-stop bolt** (counterclockwise) with a **7 mm wrench**, or a pair of **needle nose pliers**.
(It's very hard to reach with your fingers.)

6. Lift the bed (from under the Y axis rail) until the bed is a couple of mm away from the tip of the nozzle.

7. Use the **Z-knob** to bring the bed up to a distance of **0.15 – 0.20 mm** from the nozzle, using the metric feeler gauge* to set the distance. The gauge should slide in and out with a slight resistance, but not stick.
(Do **not** touch the nozzle.)

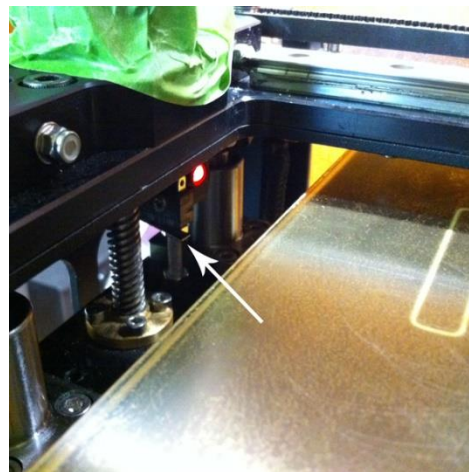
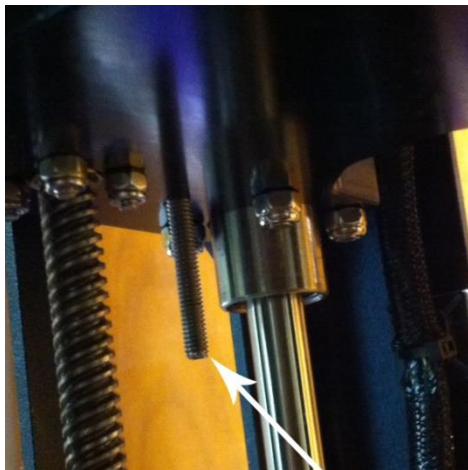


*Alternatively, you can use a folded sheet of copy paper, which measures approximately 0.20 mm.

8. Tape the Z-Knob into place to keep the bed from moving.

9. Locate the loosened **Z-stop bolt** under the bed.

Turn the bolt from under the bed until it contacts and **just** triggers the **LED light switch** above the plate, with the bed in the current taped position.



10. Remove the tape and gently lower the bed about halfway down. Tape it into place again. While holding the bolt stable so that it doesn't turn, tighten the nut to fix the bolt into position.

11. Remove the tape and **Home Z** to check the gap.
(Make sure the gauge still slides in and out without compressing the springs in the spider.)

Hot Setting the Z-EndStop: (V4 Style Nozzle)

1. Position the nozzle in the **center** of the bed.
Make sure the heated bed plate is seated flat between the rubber corners.
Clip the prepared glass plate to the bed.

2. Set the bed temperature to **60°C** and the extruder temperature to **215°C**, using the **Jog Controls** or **Machine Control Panel**.
(Retract any filament and clean any drips off of the nozzle with the tweezers – you need a clean, flat surface on the nozzle.)

3. Swing the fan up out of the way so that you can reach the screw that secures the hot end in the clamp.

Raise the bed to about a half inch underneath the nozzle.
Tape the **Z-knob** to keep the bed from moving.

4. **Slightly** loosen the screw that holds the **V4 hot end** in the clamp, using the provided M3 allen driver. (Not enough to drop the hot end out; you just want to be able to shift it in the clamp.)

5. Use the driver to press the hot end up *slightly* in the holder, and tighten the screw just enough to keep it from falling out of the clamp.

(Do not let the hot end rotate in the mount, and be careful- it's hot.)

6. Remove the tape from the **Z-knob**.

Home Z using the **Jog Controls**.

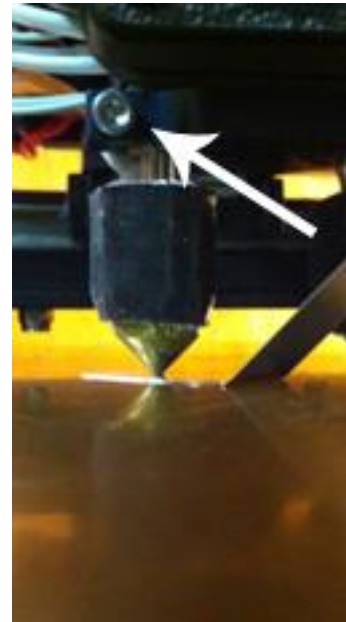
Place the (**0.15 mm to 0.20 mm**) **feeler gauge** underneath the raised nozzle.

7. Loosen the screw holding the hot-end again, and tap the top of the heater block with the M3 driver to cause it to drop down onto the feeler gauge.

Tighten the screw to secure the hot end.

Swing the fan back down into position.

8. Lower the bed and **Home Z** again to check the gap with the feeler gauge.



7. Preparing to Print:

You will do this each time you prepare to print a model.

1. Cut a flat end on the end of the filament from the spool and load the filament onto the spool holder with the spool unwinding towards the back of the machine. Take the cut end up through the filament guide, through the guide tube, and into the 2 mm hole on top of the filament drive.

(Do not let go of the end of the filament until you have fed it through the filament guide.)

Video on the process here:

MakerGear M2 Filament Feeding: <https://www.youtube.com/watch?v=Ni7xN4616No>

2. Attach your **filament dust wiper***. The filament dust wiper prevents clogs from bits of dust that wind up on the spools during manufacturing, and it's a good idea to always use one.

**(A small clip with a folded tissue inside, loosely wrapped around the filament before it goes into the filament guide. A wide variety of printable alternatives are also available on Thingiverse.)*

3. Prepare your plate with whichever adhesive you choose to use. (Hairspray, gluestick, none.)

If you choose to use hairspray or gluestick, you need to apply a **medium heavy**, even base coat while the plate is cold, and let it dry completely before printing on it. It's not necessary to wash the base coat off each time you use it, you can just refresh the base coat with a spritz before new prints, and replace the base coat when it gets uneven. If you choose to use no adhesive, the plate has to be *perfectly clean*, (*as in...wash it with soap between uses, and don't touch it with oily fingers*), and your first layer gap has to be spot-on. Clip the dried plate to the bed.

4. Heat the nozzle to the temp necessary for your filament on the **Jog Controls** or **Machine Control Panel**.

(You must heat the nozzle up to temperature before you can extrude.)

There is a block in the firmware that stops extrusion if the nozzle temperature is below 180°C.)

5. Extrude **100 mm** or so to clear the nozzle and prime it. Preferably in **20 mm** batches, not all at once.

(While you can get away with extruding 200 mm of PLA through in one batch, if you try it with some of the softer filaments, you're eventually going to jam your nozzle, so get into the habit of extruding in small batches to prime and clear your nozzle.)

6. Adjust any appropriate settings in **Simplify3D** or **Slic3r** for whichever filament you are printing.

There are generic starting settings for **PLA** and **ABS** pre-loaded into **S3D**. Tested successful settings for various other filaments are discussed in the **Filaments** section of the forum and many are also listed here:

Insta's Filaments that Work: <http://www.forum.makergear.com/viewtopic.php?p=11&t=1951>

These settings are a good place to start. Remember that you are likely going to have to adjust them a bit, since one kind of PLA does not print *exactly* the same as another kind from a different manufacturer. But for things like retraction rates and amounts, bridging settings, and fan speed settings, they give an excellent jumping off point. (*Fine tuning as you go along will result in excellent prints.*)

Note: Since PLA is the easiest filament to print, (and a good one to start with), a brief discussion of its characteristics is provided in a separate section before the Troubleshooting Tips.

8. Printing:

1. Import your model.
2. Slice it using your chosen slicer software.
3. Pre-heat the bed using the **Jog Controls**. (*Saves time*).
4. Send the model to be printed. The software will heat the nozzle, move it into position, and start extruding.

*Note: The **M2** uses a pre-configured starting script in **Simplify3D** that moves the print head forward, back, then off to the right of the bed, extrudes 20 mm or so of filament to prime the nozzle, and then wipes itself off on the plate before starting the print. This is normal. The extruded thread can be captured with the tweezers as it falls and removed, or left to fall beside the plate and picked off later.*

If you pre-heat the nozzle as well, you might have to deal with oozing filament from the nozzle over the plate. Use the tweezers, not your fingers, to capture the drips before the print starts. (It's hot, and you'll leave oils on the plate.)

Make sure that you take the time to run the few calibrations on the next page as your first prints. It makes for fewer errors in your prints down the road.

9. Calibrations:

There are a few very important calibrations that you will want to perform before you hit the ground printing.

1. Calibrate Your Z-Offset:

This simple but ***critical*** calibration will save you a lot of headaches. The file you will want to use and instructions are found at the link below. Do this once for each *type* of filament. (**PLA, ABS, PETG** etc.)

Calibrate the Z-Offset: <http://forum.makergear.com/viewtopic.php?f=3&t=3124>

2. Calibrate the Tension on your Filament Drive Screw:

Do this once for each *type* of filament. (**PLA, ABS, PETG, Flex**, etc.)

Having the incorrect tension on the filament drive screw *also* causes a lot of printing problems.

Heat the nozzle, feed the filament in **and retract it completely** using the Jog controls in **Simplify3D**, then examine the end, and refer to the guide here:

Setting the Filament Drive Screw Tension:

<http://www.forum.makergear.com/viewtopic.php?f=3&t=2694>

3. Check the Calibration of Your Extrusion Width:

The default dimensions of 1.75 mm filament diameter and multiplier of 0.9 give a surprisingly accurate result as far as filament extruded width goes, but if you want to fine tune it (for instance, if X-Y plane dimensional accuracy is critical in your design), the process for doing that is below. Check the width of the extruded wall with an average 1.75 mm filament diameter, and adjust the 0.9 multiplier according to the instructions.

Calibrate the Extrusion Width: <http://forum.makergear.com/viewtopic.php?f=3&t=1964>

That's it....go make something! 😊

Quick Start Guide Dual Extruder M2

Getting started with the dual extruder system:

Setting up the machine for the dual is very similar to the instructions for the single extruder, and for the most part, you can follow the getting started guide for the single extruder with a couple of exceptions, which I will go over below.

But first.....If this dual extruder is your first 3D printer, I'm going to offer some free advice: 😊

***** Learn to use one extruder before you try to print with two *****

When you print with two hot nozzles simultaneously, you are going to have a lot of issues with oozing filament from the inactive nozzle, and will need to print a shield around your object to catch the drips. The slicer software can handle this for you, but you will still see all of the potential problems with your prints that are normal with this kind of manufacturing process, and if you can't see what's going on, you can't stop the print before you waste a lot of time and filament.

The Differences:

1. Obviously, you have two extruders, not one. According to the conventions for the **M2**:

The **left hot end** is **Tool 0**.

The **right hot end** is **Tool 1**.

You will use those labels when you set the temperatures and conditions for each extruder in your slicer software. And the difference when using two extruders in a dual is that **you have to specify which tool** you want to work with **when you adjust the settings**, or **use the Jog Controls**.

2. When you check the **Level** and set the **Z-(End)Stop** for the machine, you will set these using the **left nozzle (Tool 0)**. *(While you are setting these, you can raise the right nozzle **slightly** in the clamp, so that it does not contact the feeler gauge before the left nozzle does.)*

*Only the **left nozzle (Tool 0)** can span the entire bed. The **right nozzle (Tool 1)** cannot reach all the way to the edge of the plate on the left. (It's something to keep in mind when you are deciding which filament you want to extrude out of each nozzle later.)*

3. The **Dual M2** comes with the **V4** style nozzles. *(When you set the **Z-stop**, you can use the second listed Hot-Set method, which does not require fiddling with the Z-stop bolt and nut.*
4. After you have checked the level and set the **Z-Stop** using the **Left Nozzle (Tool 0)**, there is an additional step that you have to take with a dual before you can set the **Z-Offset** in the **Calibrations** step. If you have a Dual setup, you will be forced to set **one gap** that has to work perfectly for **both** nozzles. Since those might be extruding at different temperatures, (different heat expansion), you will have to adjust how you set the gap by changing how far down you seat each nozzle in the clamps.

Adjusting the Right Nozzle Gap

Set the height for the right side nozzle depending on how you plan to use it.

Different Colors of the Same Filament

If you plan to use the machine to print filaments that print within the same temperature range, (ex: *dual color printing with PLA*), do the following:

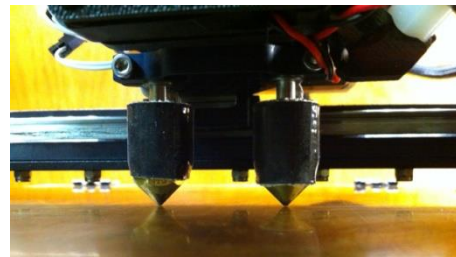
1. Clean any drips off of the nozzles if you have extruded through them. (*Your tweezers are good for this while the nozzle is cooling off.*) You'll need a clean flat nozzle surface for this.
2. With the extruders and bed **cold**, slowly raise the plate up **past** the Z-EndStop, until it actually **touches** the **left nozzle. (Nozzle 0.)**

Tape the Z-Knob into place so that it does not allow the bed to move.

3. Loosen the screw that holds the **right Hot End (1)** in place, and press it down with the M3 hex driver until the right nozzle also contacts the plate.

Do not let the Hot End rotate in the clamp.

4. (*Check the gap from eye level. There should be none.*)
Tighten the screw.



Different Filaments in each Extruder

If you want to use the left extruder to print one kind of filament, and the right extruder to print a different kind of filament with a widely different temperature range, (ex: left extruder **PLA**, right extruder **PETG**), you need to set the height of the right nozzle while everything is heated up to the correct temperatures, in order to maintain the relative height of the gap between the plate and each nozzle. (*Not doing so will have you constantly adjusting Z-Offsets, or worse, will cause gaps, failed bonding, and poor print quality when both are used at the same time.*)

1. Remove the plate from the bed if you have any kind of applied surface on it, and turn it over so that the bare glass is facing up. (*You do **not** want to hit plastic or tape with a hot nozzle.*)
Clip it in place.
2. Heat the bed up to **60°C**. (*You don't need to take it all the way up to **PETG** or **ABS** temps.*)
3. Loosen the screw that holds the **right Hot End (1)** in place, and raise it slightly in the seat before lightly tightening. (*Do not let the Hot End rotate in the holder.*)
4. Heat the extruders to the correct temperatures for what you will be printing out of each nozzle.
5. Retract any loaded filaments completely and clean off the tips of the nozzles.

(When you choose which filament goes where, remember that the right extruder (Hot End 1) cannot reach all the way to the left side of the plate, and for dual prints, you are limited to approx. 170 mm along the X-axis.)

6. When everything is at temperature, carefully raise the plate up **past** the Z-End Stop, until the bare glass actually **touches** the **left nozzle (Nozzle 0)**.

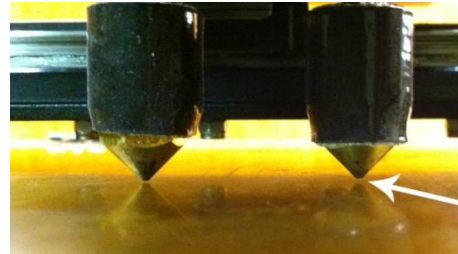
Tape the **Z-Knob** into place so that it does not allow the bed to move.

7. Carefully loosen the screw that holds the **right Hot End (1)** in place, and press it down with the M3 hex driver until the right nozzle also contacts the glass.

(Check the gap from eye level. There should be none.)

Tighten the screw.

(If you let the plate and nozzles cool in place you will see a larger gap under the hotter nozzle, which is what you want.)



By doing this, you have adjusted the relative gap on each nozzle to account for the expansion it will undergo at its respective filament printing temperature.

Now you can set one **Z-Offset** that will apply equally to both nozzles.

Starter S3D Profiles for the Dual

Because setting up the profiles for a dual extruder in **Simplify3D** is significantly more complicated than setting up for a single extruder, you might want to get started using the shared dual profiles at the link below.

(Read the discussion there first, please – you will need to make some adjustments to make the profiles work for you. It's still going to save you a lot of time setting up.)

Simplify3D Dual FFF Profiles for the M2: <http://forum.makergear.com/viewtopic.php?f=13&t=2984>

Additional handy resources for using the **Dual M2** with **Simplify3D** are linked below:

Dual Extrusion Wizard | Simplify3D:

<https://www.youtube.com/watch?v=qPufaJhyqME&list=PLogLRK5xzNdieu445p2O7wODJpQBVf-Ss>

And more discussions and modifications are available on the forum in the **M2 Dual Extruder** section.

Tips for Printing PLA

In order to get the best results with a filament, it helps to know a little something about the characteristics of that filament when printing. I'm going to list some tips for printing with **PLA**, but they are not going to work the same with other filaments. Each has its own likes and dislikes when printing – you'll need to do some research on the others as you switch to different filaments.

PLA is fairly stiff, and it warps up as it cools. (Stick it down well. Use adhesive on your plate.)

PLA does not ooze much, you can get by with a **Retraction** setting of **1.0 mm**. You generally don't need to **Wipe** or **Coast**.

Set your **Extrusion Width** to be wider than your nozzle **hole** size, but less than the O.D. of the flat face on that nozzle, to give the machine correct control over placement.

(Example: For a **0.35 mm** nozzle, the correct range is from **0.035 mm to 0.50 mm** – so use **0.40 mm**)

Set your **Layer Height** according to the following formula :

Extrusion Width/Layer Height is greater than or equal to **1.8**

In our example, with a layer height of **0.20 mm** the formula reads $0.40/0.20 = 2.0$ which is greater than 1.8.

You could also use $0.50/0.25 = 2.0$, or $0.36/0.20 = 1.8$. The reason for this is to give a *slightly* squashed layer to give better adhesion, and layer bonding. Any ratio that comes out **below 1.8** is going to start causing adhesion and bonding problems.

(And if math isn't your thing - just set your **layer height** to be **half of your extrusion width**.) 😊

PLA is *terrible* on **Overhangs**; it sags and droops, or warps up without support.

Cooling it off quickly reduces that effect.

(You will frequently see curled droopy overhangs on the sides of a print that face away from the bed fan.)

Either print with **Support** underneath it, hit it with a **LOT** of cooling, or preferably both. I actually added a small desk fan to print rear overhangs with **PLA**, but there is another trick that you can do that works even better. If you remove the **fan guard** on the **bed** fan, it increases the cooling power exponentially. Just be very careful to not let the filament get drawn into the fan when you are loading it – the curl in filament tends to point it right *at* the fan. Also remember to make an adjustment to your printing temp and bed temp if you remove the fan guard – it cools things off by about 10 degrees. If you are printing something with an overhang, choose an **Outline Direction** of **Inside-Out**, otherwise use **Outside-In**.

You can also orient your print on the bed so that the worst overhang areas point directly towards the fan.

You can reduce your **First Layer Speed** to around 60% or so to help with bed adhesion if you need to, and give the filament time to stick to the warm bed.

It's always a good idea to print a **Skirt** around the object to be printed, 2-3 outlines, 1 layer, at a distance of from 2 to 4 mm from your print. It gets the nozzle well primed and can show gap problems before the print starts, giving you time to stop it before you waste a lot of plastic.

Use a 1 layer **Brim** or a **Raft** if you have a tall object with a very small footprint, that otherwise might have trouble sticking to the plate. (Rafts are a pain to remove though, so not recommended unless you *absolutely* need it.)

Temperatures: Start out on the low side of the temperature range with **PLA**. Many people print it successfully with 190°C extruder temp, I use about 205°C but I have the fan guard removed and that cools things a lot. Bed adhesion was good for me at bed temp of 40°C (50°C without fan guard) with the borosilicate and Kapton tape. If the filament is not sticking or bonding, you can increase the extruder temp up to 220°C safely with **PLA**. Keep in mind though, that when printing small isolated parts, heat soak can become a problem, and you'll see more oozing and surface blob issues at the higher end of the temperature range.

Bridging: **PLA** is *fantastic* for **Bridging** (printing over open air between **two level edges**); you generally don't have to make any adjustments to it. I once accidentally bridged a 140 mm gap with **PLA** and it didn't sag a bit. (*Would have been great if I had actually wanted a bridge there.*) 🤪

One final note – it's optional, but you can slow down the prints and get a much better print. With 3D printing, slower **Printing Speed** always results in better quality, and the default settings in **S3D** are **high**. I keep my **Default Printing Speed** for **PLA** at **4200mm/min** (fairly slow). The advantage that it gives is less ringing around surface corners and holes, and a much nicer surface finish. Conversely, you don't want to slow down the **Travel Speed** if you have a filament that oozes badly.

A sprinkle of notes on printing other filaments:

Quick and dirty:

PLA likes to be cooled off quickly.

ABS *doesn't*, and will crack and split on tall prints unless an enclosure is used to keep the heat in.

PETG is a good (*non-splitting*) alternative to **ABS**, but it can be tricky to get set up – see the forum for a detailed write-up on printing it.

PLA is the fastest printing filament – in general, the more flexible it is, the *slower* you have to print it. Make sure you adjust the printing speed and the **Filament Drive Screw Tension** when you switch between filaments.

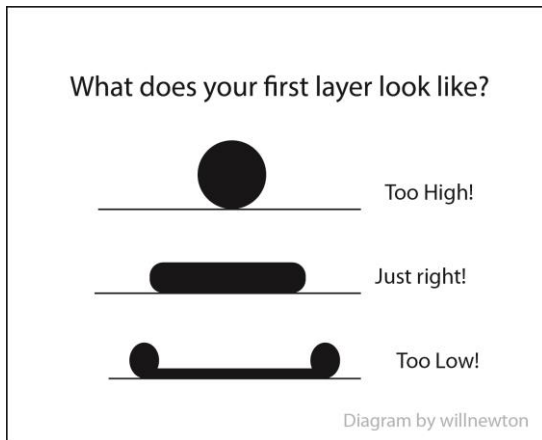
If you switch back and forth between different kinds of filaments in the same nozzle – always use **cleaning filament** in between. (See the **Troubleshooting Tips – Clogged Nozzle** for more info.)

And many of the filaments that we print are hydrophilic (they pick up moisture from the air) and that makes them get stiff over time and start to break. You'll want to store these in a sealed container of some kind when you aren't using them, with a desiccant to extend their life.

Troubleshooting Tips:

Most of the problems that beginners see stem from only a handful of issues. Many of these can be spotted (with practice) by watching how the “critical-first-layer” is laid down.

(You’ll quickly become an expert at spotting potential trouble just by watching that first layer go down.)



The diagram shows what the profile of an extruded thread looks like based on the nozzle position relative to the bed.

1. Incorrect Gap between the Nozzle and Bed. (Setting the Z-EndStop, and the Z-Offset Calibration).

The correct gap has to fall within a very small range, and using a business card to set the gap when the nozzle and bed are cold makes it almost impossible to get right the first try. *(Testing has demonstrated that you can lose up to 40% of your gap when you subsequently heat the bed and nozzle, so if you’re not setting it while hot, you’re guessing.)*

Most beginners prefer to have the gap be a little larger to start with in the beginning (**0.20 mm**). It’s perfectly okay to use a thicker gauge and then fine-tune with the **Z-Offset** to move the bed closer to the nozzle when you start your prints.

Some of the issues that you will see with an incorrectly set Z-Stop gap are:

a. First layer smashed completely flat to a thin transparent ribbon – it means the gap is dangerously small, reset your Z-stop gap (**The Z-Endstop Calibration**) so that the nozzle has more room to print the first layer.

b. First layer not sticking – the gap might be too large. If the filament isn’t sticking anywhere, your gap might need to be closed in a bit by adjusting the Z-stop gap (**The Z-EndStop Calibration**). *(Or it might be an adhesion issue, discussed below.)*

- c. **First layer is squashed enough to create a sharp skirt on the calibration square, and the first layer is shorter than the other layers, but otherwise the print is sticking everywhere** – fine tune the gap using the **Z-Offset Calibration** and enter the values into the **Z-Offset** slot in the **G-Code** tab of **Simplify3D** or **Slic3r**.
- d. **First layer is sticking well in most places, but not in a couple of spots** – there might be a low spot in your glass plate. (It's not necessarily a leveling problem, glass is rarely perfectly even.) This can be adjusted using the **Z-Offset Calibration** as well, to average it out.
- e. **Rounded discrete threads on the first layer that do not bond to each other** – it means the gap is too large, use the **Z-Offset** to bring the plate in closer for the first layer.

2. Bed Adhesion Issues.

To get a successful print, the first layer has to stick **everywhere** through **the entire print**. But as plastic cools, it shrinks and warps, pulling away from the bed. So you need to come up with something that keeps a print stuck firmly to the plate while it is being printed, but then releases the print when you want to remove it.

There are different methods of keeping the plastic stuck to the plate, and they are different for different filaments, but in general, the use of hairspray or gluestick is common, unless you have purchased a specialty surface that holds without adhesive.

The adhesive needs to be applied fairly thickly, and most beginners tend to naturally skimp on it a little at first.

Just try increasing the adhesive until you get it to hold.

In general, heating the plate slightly helps with bed adhesion, but there's no need to overdo it. With a borosilicate glass plate, and an adhesive of some sort, **PLA** sticks fine in the 40°-50°C range.

Removing the print should be done once the glass plate cools completely – and sometimes it is necessary to freeze the print off of the plate if it is stuck too securely. Let the borosilicate plate cool (off the bed) for a few minutes, then put it into the freezer with the print still attached. After about 15 minutes to an hour (depending on the size of the print), the print will usually pop off without damaging the glass. What is happening is the print is freezing and contracting faster than the glass, breaking the seal. (*Trying to pry the print off while warm can pop a piece out of the glass, so let it cool.*) And the borosilicate is designed to handle temperature swings.

The use of full sheet Kapton tape provides a bit of additional protection for the glass, and I do recommend it. (*It's rare, but sometimes with a large surface area print and high infill, you can still have a piece of glass pop off, even if it's completely and correctly cooled. The forces on the glass are a lot higher than you would expect from something like hairspray.*) You can apply your adhesive of choice directly to the Kapton tape, just like the glass.

Some of the issues you will see with failed bed adhesion are:

- a. **The corners of your print will start to pull away from the bed** – this causes the print to warp up and eventually the nozzle might actually hit the print – knocking the print loose, causing misalignment issues, or even sticking the nozzle into the print. *(You do **not** want that to happen....It's a **huge** mess to clean up.)*
- b. **Support structures might get knocked over** - and if you don't notice it, you'll wind up with an air-printing situation later. *(Bird's nest - **big** mess also.)*
- c. **The first layer isn't sticking in certain spots** - this can either be because the gap is too large as outlined above, or because you don't have enough adhesive in that spot. Try refreshing the adhesive, and if that doesn't fix the problem with the next print, adjust your **Z-Offset** numbers.

3. Incorrect Filament Drive Screw Tension.

You want the tension on the filament drive screw to be **just** enough to catch the filament and guide it through to the nozzle....a *lot* of problems are caused by having that screw set too tightly. Once again, different filaments have different properties, and they need a tension adjustment, but not much of one.

PLA is a relatively stiff filament – you want to set the tension so that the “bite-marks” in the extruded end are about 10% to 15% of the diameter. It likes a very loose tension. **PETG** is a softer filament that deforms more easily – you tighten the tension by about a quarter of a turn to switch to **PETG** from **PLA**. Calibrate and reset the tension when you switch between different kinds of filament.

Some of the issues you will see with incorrect Filament Drive Screw tension are:

- a. **Jammed and stripped filament** – nozzle stops extruding accompanied by thunking sounds as the stepper motor skips and strips the filament. Once the filament is stripped the thunking stops, but nothing comes out and several layers may skip before you notice it. Stop the print. *(The thunking sounds will not hurt the machine.)* Retract the filament – see if it is stripped, adjust the tension, and redo your print. If the filament is not stripped, it might be jammed. Remove the fans and check around the extruder gear to see if there is a wad of warped filament wrapped up inside.
- b. **It's very common to think that you have a clogged nozzle or hotend** - when really all you have done is stripped the filament. Always check for stripping first.

4. Clogged Nozzle.

The easiest way to get a clogged nozzle is to switch back and forth between **PLA**, which is a low temperature plastic and one of the higher temp plastics like **ABS** or **PETG** without cleaning the remnants of the last filament out first. The higher temps for the **PETG** or **ABS** will cook the leftover **PLA** into carbon on the inside of the nozzle, and eventually it's going to completely clog.

If the remnant is **PETG** or **ABS** in the nozzle, it will not melt completely at the lower **PLA** temperatures, so again, you've got nothing extruding.

Clogs can be cleaned out, but it requires either several hours of soaking in a solvent, or the *careful* use of a propane blow torch if you happen to have one on hand.

To avoid this situation, we do a couple of things. One is to run a **filament dust wiper** on the filament before it goes into the clear plastic guide tube. There are several nice ones on **Thingiverse** that you can print quickly as one of your first projects. Or, you can just wrap a tissue inside a binder clip around the filament before it goes into the guide tube – it works just as well. That catches any dust that gets deposited on the filament during processing and keeps it out of the nozzle.

And for switching between types of filament, you *always* want to use some **eSun Cleaner Filament** before switching to a different filament.

(Note: It's really a bear to use, but it's what is available. Be very careful with the rate of extrusion when using it – it has lousy diameter consistency, tends to strip out very easily, and jams like the dickens, so watch the tension.)

What you will see with a clogged nozzle:

- a. Nothing comes out of the nozzle, or the diameter of the extruded thread is greatly reduced.** – Check the forum for instructions for safely removing your hot end and disassembling it to clear the clog.

V4 style: <http://forum.makergear.com/viewtopic.php?f=3&t=2942>

V3B style: <http://makergear.wikidot.com/m2-hotend-assembly>
<http://forum.makergear.com/viewtopic.php?f=3&t=2074>

(Note: Jammed or stripped filament also causes no extrusion, and is the more likely candidate. Check for that first.)

5. Bed Leveling problem.

After using the machine for a while, you might start to see problems that indicate the level needs to be adjusted.

Some of the issues you will see with an un-level bed are:

- a. **When laying down the first layer, you will see an *extreme* difference** - between the printing on the right side of the bed versus printing on the left side, or between the front and the back. And you will see this on all your prints that are large enough to cover a wide enough area.

(For example: the deposited thread in the upper left quadrant might be smashed into a thin transparent ribbon, with the first layer not sticking in the opposite quadrant, and the thread being okay everywhere else.)

Fix the problem by re-leveling the bed.

6. And the final problem – Ugly Prints.

This is more of an aesthetic issue; it isn't something that is going to completely wreck a print. It just means you haven't got the correct settings for that filament yet. Correcting this requires making adjustments to the **S3D** or **Slic3r** settings to make a particular filament do what you want it to, and that requires a basic understanding of how that filament prints.

Best way to do that is:

- a. **Print some and make adjustments** until it does what you want it to. That takes time and patience, but it really **does** help you to learn what works and what doesn't. And once you learn what does what, it makes it much quicker to learn to print a different kind of filament that you've never used before.
- b. **Use someone else's settings as a starting point (FFF files- Simplify3D)**. I generally don't like to use someone else's settings file - it doesn't really save much time, because whatever that person has done – it's a guarantee that their setup is not the same as yours. You will still have to calibrate after you load their **FFF** files, adjust the **Z-offset** to your number, change the temps to match your nozzle, etc.

But for certain things like retraction amounts, coasting, wipe, & bridging settings – it can be helpful to know what other people have had success with. The **Filaments** section of the forum has a lot of useful information on printing just about everything, and that's a good place to start.

Finally: An excellent overall **Troubleshooting** resource is now available on the **S3D** site here:

<https://www.simplify3d.com/support/print-quality-troubleshooting/>