Monoprice Mini Delta V1 Quick Start

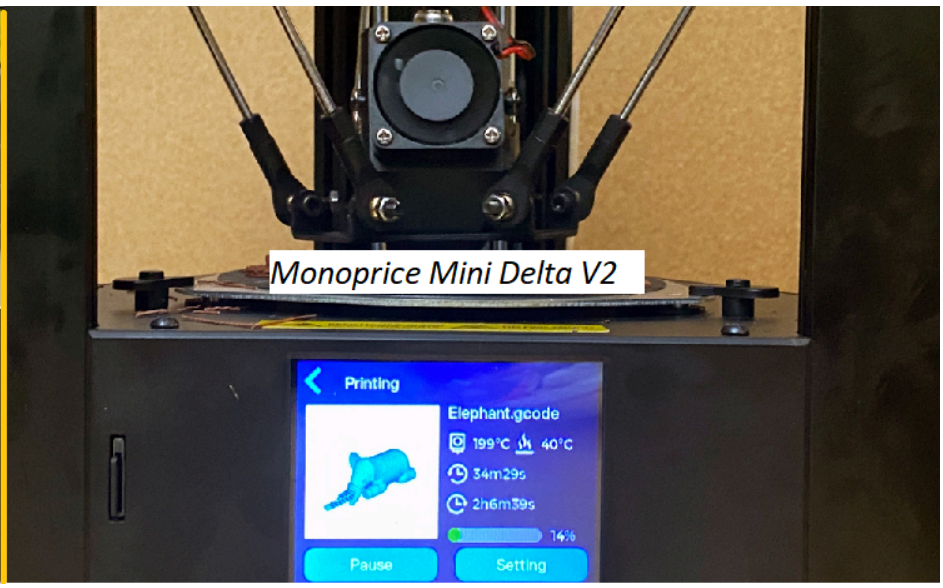
Printing your first Cat and Beyond
A.K.A MPMD for Dummies

NOTE: THIS GUIDE IS *NOT* FOR THE MONOPRICE MINI DELTA V2!!!!!!!!!!

This guide was developed for the original Monoprice Mini Delta (MPMD) and closely-related variants. It is not valid for the much newer Monoprice Mini Delta V2. See the image below to determine which MPMD you have - you can tell from the LCD screen.

Not much is known about Monoprice Mini Delta V2 at this time, but we do know that the firmware is very different and also that the manufacturer is different.

- Monoprice does not manufacture these printers in-house. The V1 was manufactured by Malyan while the V2 is being manufactured by WEEDO.
- The gcode suggestions contained within this document are only valid for the Monoprice Mini Delta V1.
- It still remains to be determined which hardware tweaks and parts are compatible.



Now, onto the startup guide for the Monoprice Mini Delta V1...

A quick note: If you're BRAND NEW to 3D printing, you might want to check out Kevin Taylor's 3D Printing Glossary, which has been included here at the end of the guide. You can refer back to this often as you get more and more advanced.

For easy access to the Google Drive's Outline feature, open the View Menu at the top and check "Show Document Outline" or press Ctrl-Alt-A and you'll get a handy shortcut list to different parts of this guide.

Printing your first Cat (or how to test the printer)

So you just got a brand new (or refurbished) Monoprice Mini Delta 3D printer. Whether this is your first 3D printer or just your first Delta style, this guide will get you started printing some simple objects and set you on the path to filling your house with cheap plastic trinkets and possibly useful additions.

Out of the box, you'll want to keep an eye on a few things. First, when the printer comes to you, it will probably have some foam protecting the hotend/nozzle and the arms that hold them. Should be obvious, but remove that and free the arms. You'll also find a small pack of tools, including a scraper, SD card and some Allen Keys. DON'T LOSE THESE! You'll need them later. There should also be a weird looking Metal Hook, that's for hanging your filament on the back of the printer (opposite the display).

First, you will want to make sure that all three bed hold-down clips are oriented properly and as secure as possible. Make sure that the fat ends of the bed clips touch the bed, as pictured, and push down on the clips HARD to minimize wobbling. You may have to push down on the bed clips periodically until you can get around to upgrading them (mentioned later).

Next, turn on the printer. The MPMD doesn't come with a power switch, so it is recommended to either use a power strip with an on-off switch or a smart plug so you don't have to keep unplugging the connector from the printer or the power supply.

It is not recommended to unplug the 12VDC connector that goes directly into the printer with power on. It may spark and ruin the contacts over time.

Once it's powered up, insert the included SD card and grab your first roll of PLA filament (1.75mm diameter only). Hang it on the included spool holder, with the loose end coming over the top.

- The trick to loading filament is to cut the end at a 45 degree angle with a sharp pair of scissors. Alternatively, cut it straight, then sharpen the cut end like a pencil (on some sandpaper glued to a little block of wood), but blunt and rounded.

(There's a small gap in the hotend just below the tube that the filament gets caught unless pointed.)

- Using the printer's front display and menu, select Preheat, and heat the nozzle to about 210c.
- Squeeze the extruder lever and gently feed the sharp filament into the open side.
- If it gets stuck, wiggle the white tube just above the nozzle fan and push some more.

- Once hot plastic starts coming out of the nozzle, stop pushing, release the lever and stop the preheat to turn off the heater. After a few seconds, remove the extruded plastic from the nozzle and clear the bed area (you can wipe the bed with alcohol wipes to keep it clean).

Now it's time for the MPMD rite of passage, the Lucky Cat. Everyone's favorite test file that comes preloaded. Using the front menu, select Print and choose the file Auto00.g to start the print. (Optionally for the Cat only, press the LED indicator/button next to the SD card slot). The screen will change to show heater temperatures. After the bed heats up, the nozzle will heat and the printer will run a quick Auto Leveling script then print your very first cat.

99% of printers will print the cat file just fine, remember, this file was DESIGNED to work with the printer. If it doesn't want to print right, skip to Basic Troubleshooting below. Otherwise, enjoy your new plastic cat and continue with this guide to learn how to print more advanced things!

3D Printing: A quick intro to the hobby and the MPMD

Welcome to the hobby of 3D Printing. Whatever your reason for buying this printer, from cheap plastic trinkets and decorations, to prototyping parts for manufacturing, to actually selling your own designs, I hope this guide will help get you started on the road to a successful 3D Printing experience.

Before you get too excited, let me remind you that this is still a hobby, and you're at the entry level here. The MPMD is a great "entry level" 3D printer, it was my first and might be your first too. 3D printing has come a long way in the last few years, you no longer need to build your printer, run wires, solder and load firmware to get a working machine. There are a lot of printers out there for a lot of purposes, materials and skill levels. More and more, they're coming ready to go, where you need minimal assembly and very little calibration to start printing. That said, you WILL still end up tweaking, modifying, fixing and even calibrating eventually.

Monoprice advertises this as a plug-and-play printer, and for maybe 50% of people it can be. With this guide, it's more like 80-90% and if you follow it all the way through, there's no reason it can't be 100%, but for a few printers, it will take a LOT of work to overcome quality control issues on this rebranded printer. I've had mine for a year, and even with the help of the main Facebook Group, it takes me a lot of work to keep this thing printing top quality. So I'm writing this guide to share a lot of what I've learned, be a reference for the experienced ones, and hopefully save the new users in the group some trouble.

I include the list below not to scare you away, but just to share some of the stories from the 10% who have troubles out of the box and what kinds of problems you may need to overcome in the future. I plan to cover all of these eventually, it's just a slow process to fix all the little details of a complicated machine like this. (Credit to Dennis Brown over on the FB group for a lot of the engineering related details and fixes).

- The MPMD ships with firmware that prints about 2% small (varies by machine), which helps compensate for their bed sticker being too small to print the 110mm spec. For most decorative prints, this won't matter, but for precise parts, you'll need to go through this ENTIRE guide to correct for the dimensional accuracy (Not written yet).
- The mechanics ship with arms that follow a path that looks like a domed wavy washer instead of flat across the bed surface. Delta style printing requires a LOT of internal math to keep the arms even, but there are things you can do to help them along (again, listed below).
- The firmware settings try to compensate with "auto leveling" by subtracting an inverted dome shape from the positions. However the compensation is not exact and the wavy washer shape is still there.
- The beds are tilted on some machines, compared to the vertical rails. Again, software and calibration will handle some of this, but it's not perfect if your machine is "warped", so depends on how accurate you need it.
- The auto bed level removes the tilt from the bed, but can not compensate for anything else as shipped. There are users who are working on a 3rd party firmware to help compensate and improve the auto level routines.
- The Bed hold down clips do not allow for proper auto bed level to compensate for these things on bigger prints as the bed tends to bounce, so parts larger than 50mm diameter may have trouble printing. For now, make sure they're firmly pushed down. Better replacement designs are covered later in the MPMD Calibration Roadmap.
- The extruder should be calibrated for each new filament or the prints may come out blobby or skimpy. Partially due to the "Bowden" design of the printer and partially due to inconsistencies in various filaments.
- The heating element ships with poor firmware settings and does not hold temperature very well. There's some basic corrections for this in my profile, but the bed still won't get hot enough to easily print ABS without further modification.

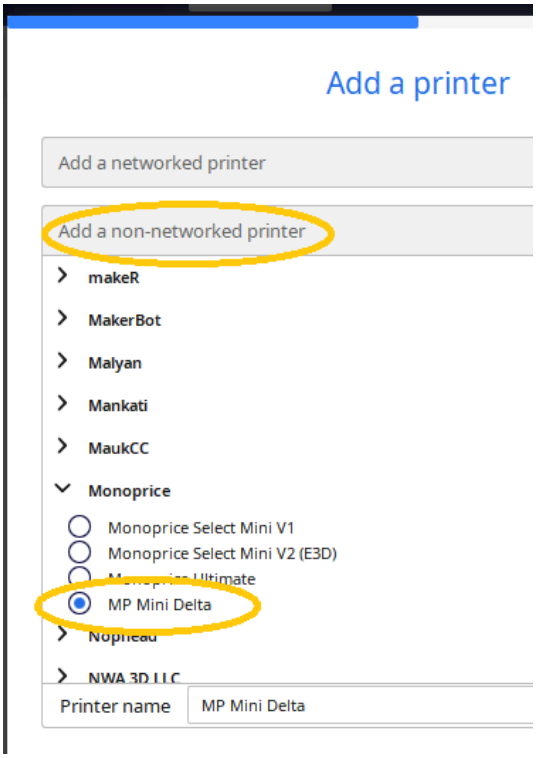
Basically, a lot of the claims that Monoprice makes for this printer aren't being met by the factory (this is rebranded from Malyan) and the user community has worked hard to turn this cheap machine into a workhorse, so keep reading for details and compiled knowledge on how to do the same.

Setting up your Slicer

Now that your cat is printed, or even while it's printing, you'll need slicing software to convert 3D files (STL files) to the gcode that a printer understands. There are a few options for free slicing software but the most common is Cura.

- **To download Cura**, head to https://ultimaker.com/en/products/ultimaker-cura-software and download the latest version for your computer (last version successfully tested is 4.6.2). Don't use version 4.7.X.
- You may also want to **download Brian's profiles** (follow one of the links, below).
    - Brian's Drive: bit.ly/curampmd
    - Alternate Link: https://www.mpminidelta.com/slicers/cura

3/29/2021: Brian's profiles have not been updated since Cura 3.6. Many users report that they now get good results just using the built-in Cura profiles with optimal layer heights.

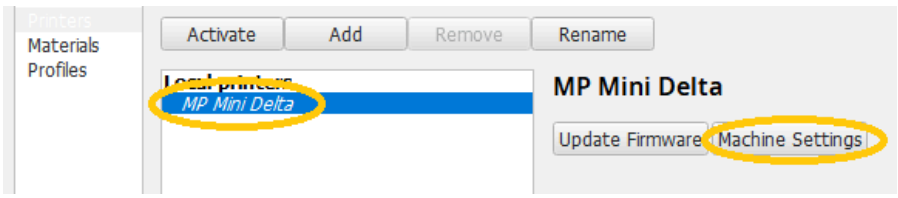


Thanks to the combined effort of several users in the Facebook Group, the "MP Mini Delta" printer definition is now included in Cura as of version 4.5 (optional video guide). If you want to run an older version of Cura, click here for the old version of the MPMD 101 Cura setup instructions.

If you are opening Cura for the first time and you follow the prompts, it will eventually ask you to "Add a printer." Here, you will want to look under "Add a non-networked printer". Scroll down to "Monoprice", and then select the "MP Mini Delta" before continuing. If you're not running Cura for the first time, then you can simply add a printer using Cura's menu.

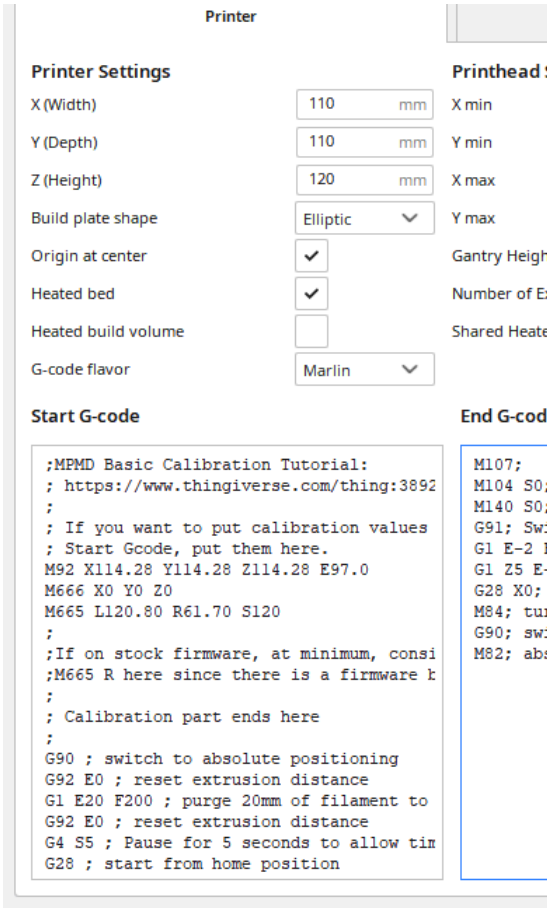
Before you run off to start printing, you should consider performing some basic calibration steps (see link) and importing Brian's profiles into Cura. You do not have to do all of the calibration steps on the previously linked page, but at a minimum, **it is strongly recommended** that you correct your motor steps per mm, M92 XYZ values. This will help a bit with your dimensional accuracy and also set you up to print at your **magic layer height increments of 0.07, 0.14, 0.21, 0.28**, etc.

- In Cura, go to Settings->Printer->Manage Printers…
- Now, with "MP Mini Delta" selected, click on "Machine Settings".



- We want to focus on the "Start G-code", so expand the window so that you can see it better.
    - This is code that will run at the start of every print.
    - The g-code contained here is read line-by-line, starting over on each new line.

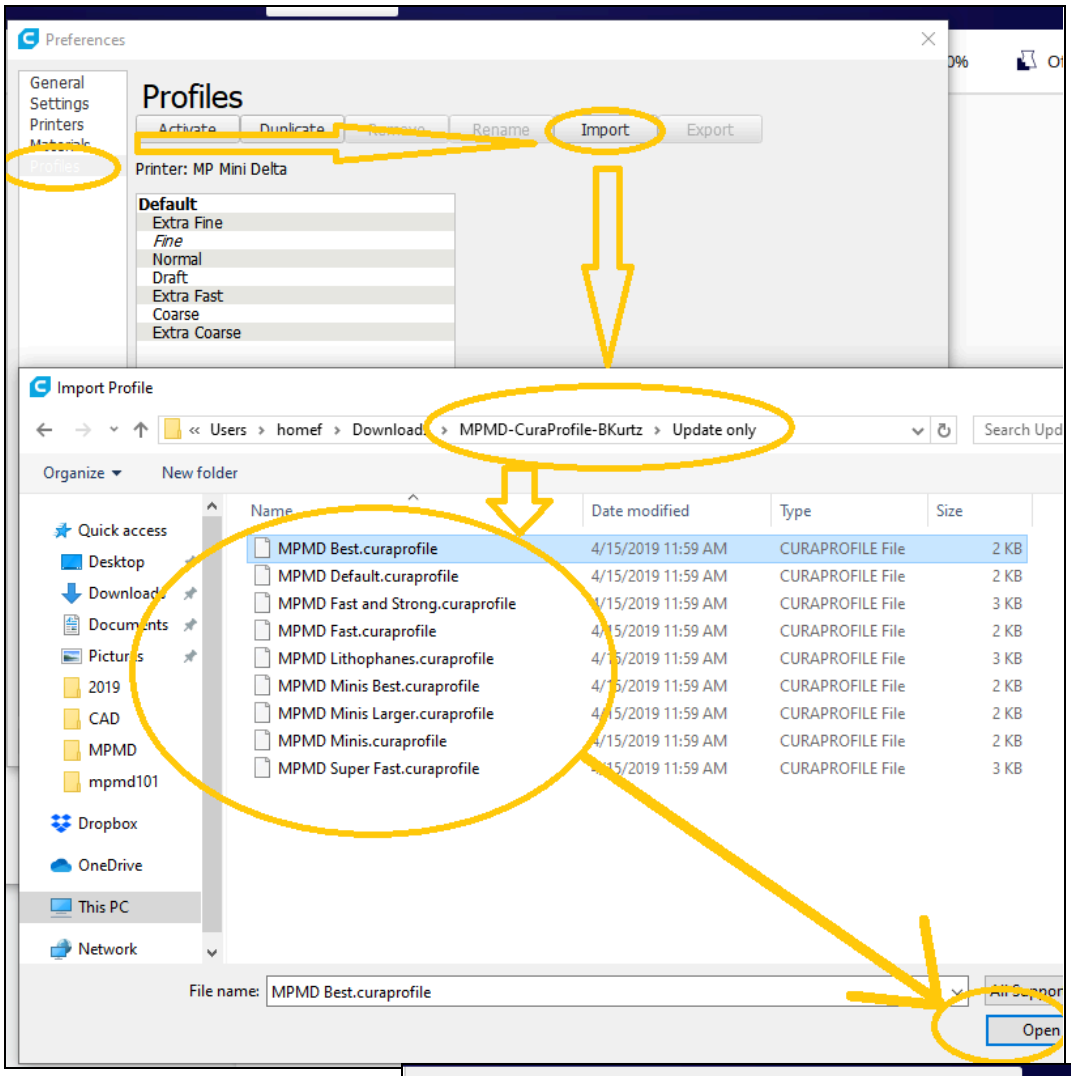
- Everything after the semicolon on a line is ignored. I.E., for G-Code, a ";" is used for commenting.
- For this guide, I am going to recommend that you copy-paste some of the configuration values from the basic calibration page into your Start G-Code, but advanced users may wish to save the values to EEPROM instead.
- Steps to set your M92 values:
  - Check your firmware version (displayed when the printer boots) and make a note of the first two digits (typically V37-V45).
    - Side Note: It is not recommended that you flash new firmware without a good reason. Certain firmware versions are intended to ship with specific Monoprice Mini Delta mainboards (not all of them are the same!!!).
  - Firmware Versions 37-41 (1/8 microstepping drivers)
    - M92 X57.14 Y57.14 Z57.14 E48.50
  - Firmware Versions 43 and 44 (1/16 microstepping drivers)
    - M92 X114.28 Y114.28 Z114.28 E97.0
  - Firmware Version 45?
    - Even though this firmware version *should* correspond to the 1/8 microstepping values, user reports indicate that this is not always the case. Please follow the steps outlined on the calibration page or this video to determine your correct values. Setting the incorrect values could cause the printhead to float ~3 inches in the air or slam into the build plate.
  - Copy-paste the appropriate M92 line into your Start Gcode. Make sure there are no comments in front of your code. An example is shown in the screenshots. See the video guide if you need more help.



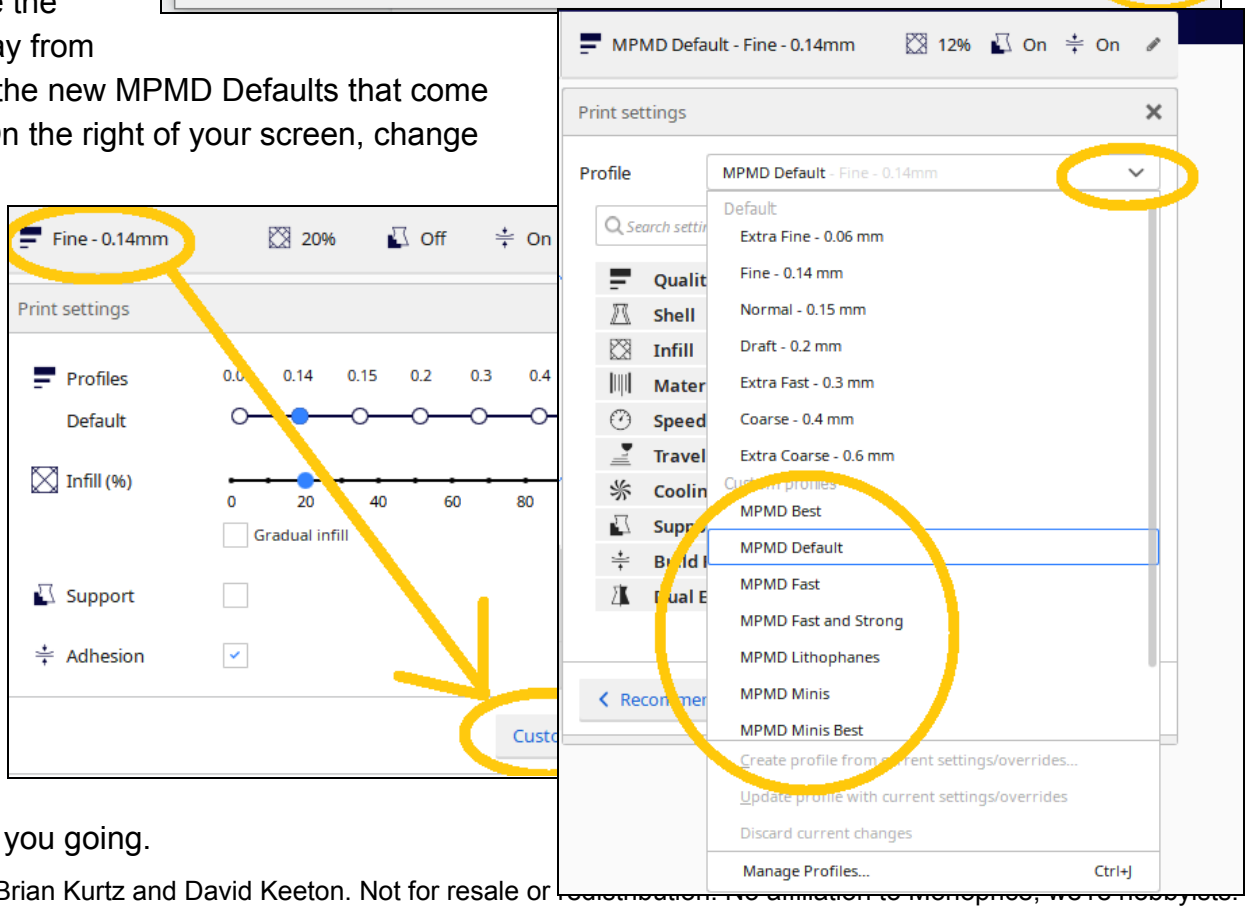
- While you are in the Start Code, make note of the line that says "G29 P2 Z0.28" which is your auto-leveling sequence. This is where you will set the z-offset if the filament won't stick later.
  - The number after "P" controls your auto-leveling sequence pattern.
    - With a stock bed with stock bed clips, you should watch patterns above P2 carefully, because the other sequences may tap outside of the stock bed sticker (not good) and/or make the bed wobble too much (also not good).
- The number after "Z" is where you will set your z-offset
  - This effectively tells the nozzle how high/low to start relative to where the printer thinks the bed is located. Every printer will need a different number here, but 0.28 seems to be a good starting point for most. More details on fine-tuning this parameter can be found under "Basic Troubleshooting."

- Close Machine Settings.

- Next, we want to import Brian's profiles, which you downloaded previously, into Cura.
  - In Cura, go to Preferences-> Configure Cura…-> Profiles
  - Click on "Import" and then navigate to the downloaded "Update Only" folder ("New Install" was for Cura versions 3.6-4.4.1 and can be ignored now).
  - Import all of the profiles, one-by-one.



The last step before you're ready to print all the free models you can find online is to change the printing profile away from Cura's defaults to the new MPMD Defaults that come with the profiles. On the right of your screen, change over to the Custom Settings tab. Now you'll see a dropdown for Profiles. Select MPMD Default for 90% of your printing. Eventually you can experiment with the other MPMD profiles that adjust layer height and print speed, but Default should get you going.



Quick Explanation of Brian's Profiles:

MPMD Default: 0.14mm layers, used for every-day printing at pretty good quality.
MPMD Best: 0.07mm layers for extremely fine detail (takes awhile).
MPMD Fast: 0.21mm layers for prototyping or models with less detail
MPMD Super Fast: 0.28mm layers for extra fast but very rough models
MPMD Fast and Strong: 0.28mm layers and thicker walls/more infill

MPMD Lithophanes: http://3dp.rocks/lithophane/ used for turning pictures into 3D Prints.
Use Outer Curve and Positive Image (under Image Settings) for best results. Tweak Model settings.

MPMD Minis: 0.07mm layers and extremely solid, for D&D Miniatures and similarly small/detailed figures
MPMD Minis Best: 0.05mm layers for extreme detail. Orientation matters, experiment!
MPMD Minis Larger: 0.07mm layers but less infill for larger monsters and small terrain.

Slicing free files

The internet is full of free 3D files for printing, but 98% of them will be a format called STL. You can't load an STL directly to your printer, so you need to "slice" it first and convert it to gcode. Now that you have a working profile from above, grab your favorite files from any one of the sites below and open with Cura.

Places to get 3D Files

www.thingiverse.com www.myminifactory.com www.shapeways.com www.yeggi.com
Just a small sampling, Thingiverse is the most common, but any site that offers STLs will work.

Prepare a file for printing (Needs to be updated for the redesigned 4.0 interface)

Once you have an STL downloaded, open it in Cura. It should appear on the virtual printer bed after a few seconds. Right click on the model and select "Center" if necessary to move it onto the bed. On the left you'll find controls to resize (scale), move or rotate the model as needed. On the right, you'll find a large collection of settings, some hidden for now. The only ones to worry about for now are the following:

- Bed Adhesion: Raft will be the most common for now, it lays down a few base layers before the model to help it stick to the bed and even out any remaining issues with leveling.
- Support: 3D printers can't print angles past 45 degrees very well, they don't like printing in thin air. Support is used to prop up overhangs and free floating sections. Turn this on if your model has sharp angles that leave it free floating or things like Arms that come out from the side.

To see your rafts and support, as well as a preview of your model, hit Process in the bottom right. Then at the top-right of the preview window select Layer View and change the color settings to Line Type. Raft and Supports default to Blue, the little guide window will show you the other colors for various parts.

Now that you've sliced the file (processed) you can save it to a file. Cura defaults to adding MMD to the beginning of the file in case you have more than one type of printer, and the extension is .gcode which is actually just a text file. You can open it in notepad, or similar text editor if you want to see, but there's a lot of machine code commands, so don't edit until you know what you're looking at.

Save the file somewhere you can find it, then copy to the printer's SD card, or save directly to the SD card if it's in your computer. Once your new gcode is on the card, you can put that back in the printer and print the new file just like the cat.

Please note that using my profile, you'll get a slightly different start of the print from the cat as I'm using a mesh bed leveling command and a purge to make sure the nozzle is ready to print.

Note about the SD Card:

The MPMD is a little picky about which SD cards and format it will read, but the one that it comes with is prone to failure over time. So for now it's best to use the one that came with your printer and look for a backup . See the Wiki for more details: https://www.mpminidelta.com/parts/microsd_card

General Maintenance

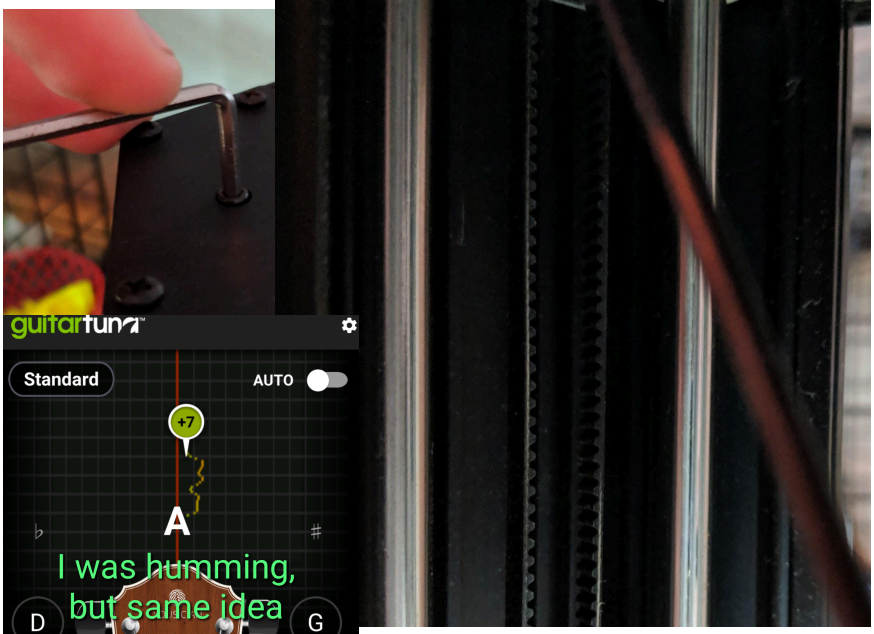
Tuning the Belts

One of the main differences between a Delta and most other printers is the way they calculate where the nozzle is. Since you can't set the XYZ positions independently, it is very important that all 3 towers be identical. Building them the same is (debatably) easy, but for the brain of the printer to know where to move the nozzle, the 3 belts all need to be the same length AND the same tension. Since tension is very hard to judge and takes a special tool, some of the brains over on the Facebook Group have determined that you can "tune" the belts like a guitar string, which confirms that they're the same tension and the printer will move evenly.

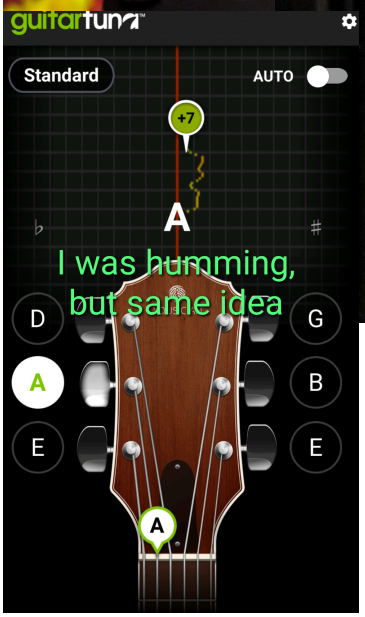
Note that over tensioning the belts will strain the pulleys and motor, while undertensioning them will cause slippage and layer shifts, so this is a VERY IMPORTANT step early on (they can be wrong from the factory), or more likely just from everyday use, the set screws may slip or the belts may stretch slightly. Try tuning the tension to the first audible note on the guitar tuner (usually A2, but sometimes different, like E2).

Start by finding yourself a tuner. If you happen to have a good instrument tuner in your house, great. Since most of us don't have one just lying around, but we do have a smartphone, grab that and go download Guitar Tuna:
Android: https://play.google.com/store/apps/details?id=com.ovelin.guitartuna&hl=en_US
iOS: https://itunes.apple.com/us/app/guitartuna-guitar-bass-tuner/id527588389?mt=8
You can pick your favorite if you don't like this one, it just happens to be one of the better ones the group found.

You'll need to confirm which end of the belt is **not** connected to the arms. **(This advice has changed recently)** With the **power off** you can gently move the arms up and down while watching or feeling the belt. The side that moves in the **OPPOSITE** direction as you move the arms is the one you want. In my experience, it's the **RIGHT** side when looking from the "inside" of the printer.



Gently push each carriage as high as it can go till it touches the idler pulley at the top. Now at the top of the tower, you'll find the "set screw" adjustments for the belts and you'll need the large Allen Key that came with your printer. Loosen this screw a little to start.



You'll need a VERY quiet room as the phone tuners are sensitive. Place your phone near the printer (inside, on top, the table right next to it, they all work). Set your phone to Standard Guitar and select the A note. With the arms at the top again, strum the belt near the middle and see if you can make a note the phone will hear. If using another app, your goal is A2 (110hz). Adjust the set screw till you get the right tone, repeat for the other two towers. For more details, check out the original Facebook post and/or this video.

Cleaning the Bed

One of the most critical factors to getting things to stick to the bed is having it clean. The printer will often leave behind extra plastic, your fingers will leave behind grease and dirt, dust will build up, all these can contribute to prints breaking away from the bed or never sticking in the first place.

Lots of people will use different bed surfaces, but for most of them, the same suggestions apply. On a fairly regular basis, every few prints or even every print, you should do the following:

- Check that there's no extra plastic on the bed from previous prints. Use the plastic scraper or a metal putty knife to clear excess skirts, brims, dots, or whatever. Avoid sharp knives as they may damage the bed.
- Inspect the bed for damage. If there's nicks or bumps or portions missing from a difficult to remove print, it might be time to think about a new surface. Check Basic Mods below for my recommendation.
- Clean the bed of dirt and grease. The easiest way is a bit of rubbing alcohol and a clean cloth, or more likely those little alcohol wipes you find at the pharmacy. I bought a huge box of them online, they work great.

Lots of people use glass, and that means re-applying a glue or hairspray every print, but cleaning off excess and re-applying is a great habit every few prints anyway. And for those who come from other printers and like Blue Tape, making sure to replace it when it's damaged is key, but if you leave it on for a few prints, it can't hurt to wipe it down too. Again, making sure your bed is clean is one of the biggest keys to a good first layer adhesion.

Lubrication of the moving parts

As with any complex machine, you have a lot of moving parts and those parts need to move freely. One of the better ways to insure this is to lubricate the joints and moving parts every once in awhile. Go grab yourself some Super Lube synthetic oil, or something similar, and I like an applicator pen. You won't need to do this very often, so order them now and you'll be fine till they arrive, or if you already have something very close, that's fine too.

Per Dennis Brown on Facebook "I put a drop on each rail above each linear bearing (where the carriage rides on the rails) when the hot end is near the build plate. I also put a drop on each of the 12 ball joints on the ends of the 6 arms." I would do this every month or two, or if you hear something binding up when the printer is moving.

Keeping your Filament Fresh

Most 3D printing filament absorbs moisture over time. Nylon is the worst, but even the PLA that most of us use will get saturated if left in a humid environment. If you have more than one roll that's not vacuum sealed, you should invest in a way to keep the rolls you're not using dry. Get some kind of "weather tight" container. I like the big rubbermaid ones, but anything that will fit a few rolls. Especially if you're in a humid area, or a hot summer, keep the rolls you're not printing with in a sealed container, preferably with some Silica Desiccant (the little gel packs that always say "don't eat" you find in new shoes, etc). You can buy it in bulk off Amazon, it'll suck the moisture out of a sealed area and keep your filament fresh. You can even get rechargeable dehumidifiers filled with the stuff now. If you're not printing with a specific roll, keep it sealed away, you'll thank me later. If you're not planning to print for a day or two (or longer), take the roll off the printer and store it too.

Some Basic Mods and Spare Parts

Video Link Covering David's Tier 1 Upgrade List: https://youtu.be/XXr8FAVFZIk

Hopefully by now, you'll be happily churning out lots of little plastic toys/parts and they're looking good. There's a few things you'll want to print for your machine and a few basic maintenance tips/tricks to keep things running smooth. Most of the info below was compiled from the Facebook Group and my own experience.

Buy some spare parts, there are certain things that break eventually. Matthew Upp has kindly assembled a website that sells parts for the Mini Delta and other Monoprice Printers at Gigdigit.com and I recommend picking up at least the following: Bowden Connector for MPMD This **WILL** break eventually, so either have spares or print one of the upgrades found in this Thingiverse Collection (make sure you read the Thingiverse Collection description for important information). Some of these options require no additional hardware, while others may require one of the more standard Bowden couplings.

While you're at Gigdigit, check out his range of spares and bookmark it for future breakages. If they're in stock, it can't hurt to buy some Extra Insulation and Spare Nozzles too.

New Feet for your Printer: There's not a lot of space under the printer for airflow, so Dennis Brown, our resident engineer has recommended Rubber Feet upgrades. Or, since you have a fancy new printer, you can print your own for free! https://www.thingiverse.com/thing:2471644

A bed sticker replacement: Some people coming from other printers will use blue tape or glass, but I learned to LOVE my PEI cover. Dennis once again walks us through its installation here. You can buy a large sheet from Amazon and cut 116mm rounds out (you'll get 5). Once you rip your factory sticker removing a tricky print, you'll be happy you have a replacement on hand.

While you're on Amazon, get some good Digital Calipers, a pair of Flush Cutters, some Alcohol Wipes for cleaning your bed (smaller boxes available, but you'll use them). If you don't already have some needle nose pliers, a sharp craft knife, superglue of some kind, I'd recommend picking up some of those as well.

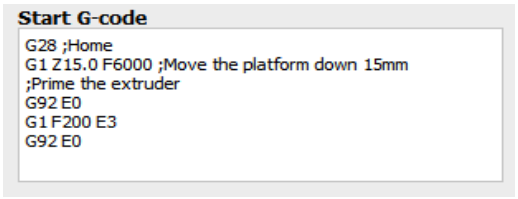
One of the simplest printable mods you can get is the Tower Dust Shield that snaps into your towers and keeps rogue plastic from getting into the base of the printer. Print 3 in your favorite color!

Basic Troubleshooting

Your Profile didn't load properly (or I don't see the right Start G-Code).

There's finally a proper version of my profile, so if you're having issues (especially if you were using an old version) redownload bit.ly/curampmd and start over by wiping out any previous profiles to clear the error, then follow the new and updated instructions above. Or go to the latest version of Cura (anything 4.5 or beyond) which now has the MPMD included by default thanks to David Keaton (you can still use my print profiles).

My Printer Won't Connect to USB



Cura is very bad at USB printing. If you want to print over USB from a computer, you're better off with Repetier Host or Pronterface. 90% of people find it easier to load files via the SD card (web interface is slow). Eventually, you can invest in an Raspberry Pi and load OctoPrint as an all in one printer management interface. Any of the methods listed above you can still slice files with Cura and send the Gcode with a different program.

Special Note: If you're running Windows 7, you'll need These Drivers to use Repetier or Pronterface.

Printer stops printing or won't start at all

Despite the above advice about printing from the SD card, the one that comes with the printer is often the cause of some very random errors. Here's a great guide from Matthew Upp on the Wiki: https://www.mpminidelta.com/troubleshooting/not_printing

My print looks weird/doesn't look right

Possibly the best visual guide I've ever seen is here: https://www.simplify3d.com/support/print-quality-troubleshooting/

While not every solution is identical for the Mini Delta, this will at least get you started and at least you will understand more what to ask when you come back to the Facebook Group.

Here's another similar guide: https://support.3dverkstan.se/article/23-a-visual-ultimaker-troubleshooting-guide

Why does my printer heat up higher than what I set?

Cura will automatically default to 5c hotter than your main printing temperature for the first layer (both the bed and the nozzle). This is to help with the first layer sticking. The temperature should lower back to “normal” after the first layer is printed. To adjust this, use the search bar in Cura to find “Initial Layer” settings.

The Red Light by my SD card is on (or why does it turn on?)

The status light by the SD card takes some getting used to. It will be solid blue if you’re using USB, and should flash red (or the blue turns purple) when a Bed Switch is pushed during leveling. It’s normal for that to flash every time the nozzle taps the bed during the auto level sequence. It will occasionally flash during printing if the nozzle hits the print due to overhangs curling up or over extrusion, so that’s another place to look.

If the light remains red/purple for an extended period of time, especially before a print, then most likely you have a stuck switch and will have problems when you try to start a new print. Loosen the bed clips by twisting them 90 degrees and gently lift the bed. Check if anything is stuck under the bed, and manually click the 3 black buttons you find there to see which one is stuck. Sometimes just pressing it a few times will clear it up, you can also apply a bit of rubbing alcohol and then click it to see if you can loosen whatever got it stuck. Once clear, replace the bed and put the clips back, Fat side inward, to resume normal operation.

What does the Speed on the display mean (and why is it always 1.0)?

A common misunderstanding is the “speed” on the front display when printing. Your printer’s speed is actually set by the slicer in mm/sec and is anywhere between 10mm/s and 120mm/s (60 is my default). The 1.0 on the display is more of a Multiplier. If Cura is set to 60mm/s and you change the display to 0.5, the printer will slow down to 30mm/s or ½ of the slicer settings. Similarly if you change it to 1.2, your printer will speed up to about 72mm/s. Just be aware that at least once, I changed this, let a print finish, and the next print remembered the old speed even though it showed 1.0, I had to re-select 1.0 to get it to resume normal speeds.

My bed never gets hot enough or takes forever to heat up

Whether you’re trying to print ABS or just want the bed hotter than 60c, you’ll quickly discover that despite Monoprice’s advertisements, the printer won’t easily get beyond 50-55c on the bed. This is mainly cause the 5a power supply isn’t enough to run the bed and heater at the same time, so the printer will alternate between them when running. This is also why the bed will heat up FIRST when you start a new print.

Some people have found ways around this, including but not limited to Enclosures, playing with the base fan, user made firmware and bigger power supplies, but these are all advanced mods. For your average PLA prints, you won’t need much more than 45-50c on the bed, so setting it a bit lower will actually speed up your preheat and get you printing faster. If you absolutely NEED it hotter, look for more advanced guides coming soon-ish.

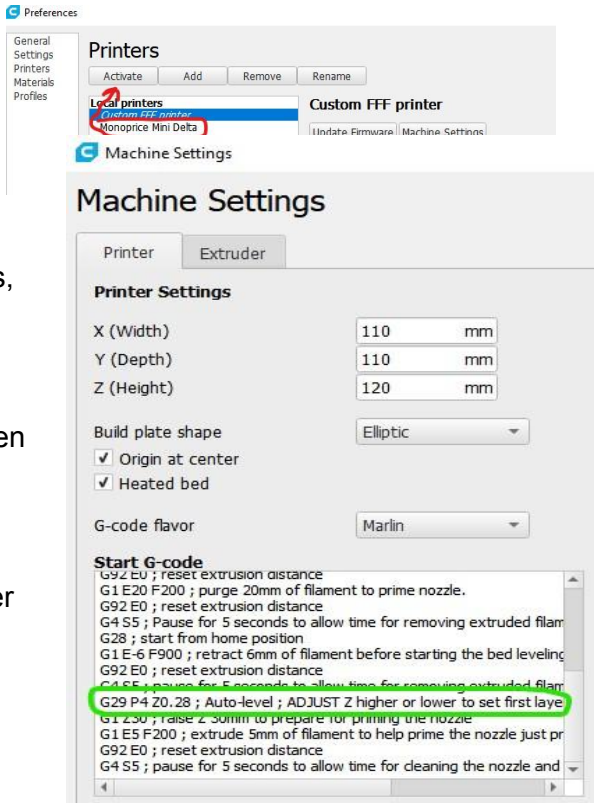
My print won’t stick to the bed/Nozzle Scrapes the bed and nothing comes out.

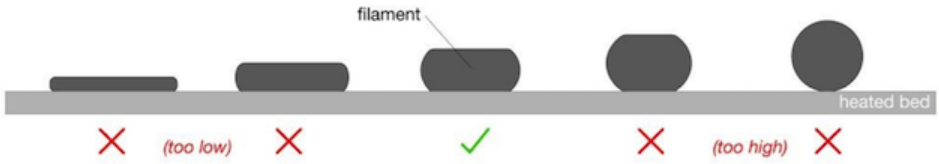
This is one that is different from most other printers, so it bears a special section. Getting a proper first layer is KEY to good prints. There’s a single setting to help adjust your first layer (mentioned above) and it’s part of your bed leveling code.

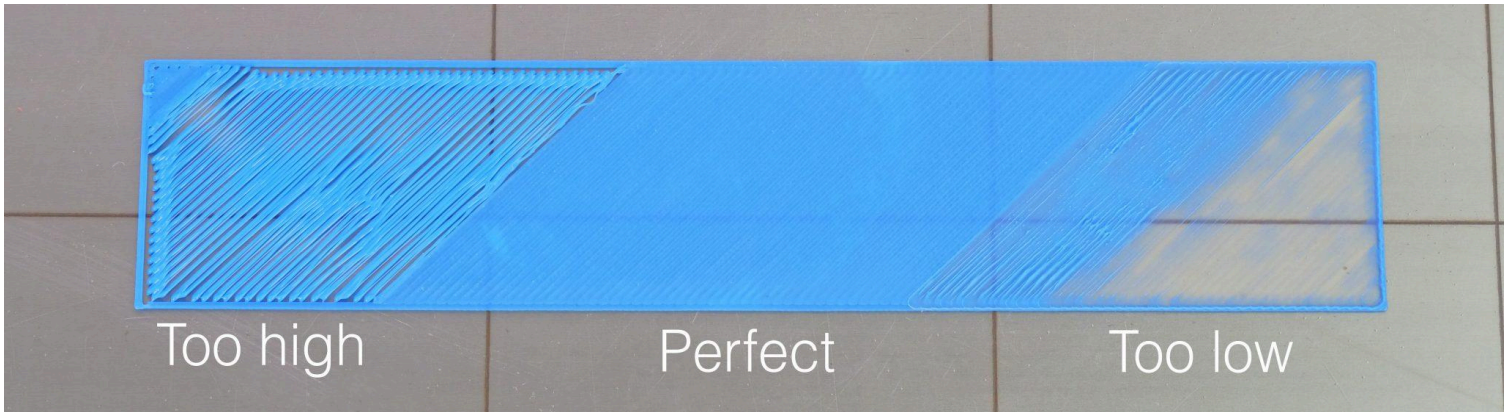
- Under Settings - Printers - Manage Printers, make sure your Mini Delta is Active, then click Machine Settings and look for the box labeled Start GCode.
- Scroll down until you see the line that begins G29. There are two values, usually P2 if using the default Cura def, could be a C-0.8 or similar, then there’s a Z value.

The Z value is the key here. If your nozzle starts too close to the bed and either scrapes it or no filament comes out for the first two layers or so, then Z needs to be larger (Z0.24 becomes Z0.26 or Z0.28) until the first layer comes out right.

If your nozzle is too far from the bed and your print isn’t sticking well, you need to make Z smaller (Z0.4 becomes Z0.38 or Z0.36) until the first layer is flat and even.







0.02 adjustments are a good place to start. Just adjust the code, re-slice and reprint to test each adjustment.

Do I need to update my Firmware?

The Mini Delta's Firmware is not meant to be updated, and it's often not worth the time or effort. Unfortunately, it is closed source, so we don't know what each version does, and due to differences in the Stepper Drivers across releases, it can actually cause problems. Monoprice also doesn't support Users upgrading their firmware. The most common Firmwares are V37 (from the initial batches and Kickstarter), V41 (First major release), V43 and V44 (New 1/16 step drivers, so not intercompatible) and V45 (Meant for older machines, but no discernable differences from V41). You can see your firmware number when the printer first boots as Vxx followed by some other numbers.

My first layer is too high/low at certain spots on the bed and/or auto leveling doesn't work.

Some people get lucky with their printer coming from the factory, and then, there's everyone else. You can tune your belts, experiment with various G29 auto-leveling patterns, and/or z-offset values, but you may find that those calibrations can only take you so far. The process to improve bed leveling might take just a few more, or possibly many steps (and at least one or two hardware upgrades) to get a good first layer across the entire advertised 110 mm diameter circle. Tutorials to fix this exist, but they may not be quite as easy to follow as MPMD 101. See the MPMD Calibration Roadmap and FAQ and follow the recommended steps IN ORDER.

Additional Calibrations/Upgrades

At this point, you should be well on your way to having fun 3D printing! However, if you want to take your print quality and reliability to the next level, more tweaking, calibrating, and upgrading via external guides will be required. Unfortunately, many of these guides are not as new-user-friendly as MPMD 101. Maybe one day, someone can make simplified guides. For now, the MPMD Calibration Roadmap and FAQ document is a good index to many of the tried-and-true guides/upgrades utilized by many in the Facebook group. It also contains additional Troubleshooting and Frequently Asked Questions.

Terminology/Glossary

The following list of terms has been borrowed (and tweaked a little) from Kevin Taylor's lovely post on the Facebook Group. I've included it here to help clarify some of the things you might see in this guide and on other posts around the internet regarding 3D printing.

Basic 3D Printing Terms as They Relate to the Mini Delta

These are not in alphabetical order, but are in the order that seems to make the most sense to me as far as each term laying the foundation to understand the next term in the list.

Measurements

All measurements and settings are in metric units (millimeters and Celsius) because that's what the majority of the planet uses. The US, Liberia and Myanmar are the only three countries still mostly standardized on Imperial measurements (inches, Fahrenheit). You can easily find converters online if you need them.

Firmware

This is the software that's baked into the printer to interpret print commands and operate the various motors and heaters. The Firmware version is displayed on the LCD as it's booting up. Make a note what version you have. It'll come into play later.

Nozzle, Hot End, Print Head

Generically speaking, this is the part the heated plastic filament comes out of. There are subtle differences between each of the three terms that are outside the purview of basic terms.

Extrusion

The act of the printer squirting out the heated plastic filament from the nozzle.

Extruder

This is the feeder mechanism that the filament goes into. The Extruder is responsible for pushing the filament through the nozzle.

Print Bed, Printing Surface, Bed

This is the platform that the heated plastic is extruded onto. In most cases, the bed is heated to help the plastic stick to the bed surface.

Bed Adhesion

Getting the plastic to stick to the print bed. This is trickier than you'd think and there are many different methods for helping the printer stick the part down to the bed. Some combination of glass, blue painters tape, glue stick, or PEI material are used by most people to achieve good bed adhesion. Proper settings are also required.

STL File

This is a file that contains the 3D drawing you're wanting the printer to print. STL is a common file type that is supported by many different 3D drawing programs.

Slicer

The application that takes the STL you downloaded from Thingiverse.com or any one of several other 3D printing sites, and converts it into a format that the printer can interpret to print the object on the print bed. Cura and Simplify3D are two of the most common, but certainly not the only slicers used. Cura is free, Simplify3D is not.

GCODE

This is the term for the commands issued to the printer from the "sliced" STL file that instructs the printer how to move the print head, how much filament to extrude and when, and how hot the nozzle and bed need to be. These commands are largely the same across multiple printer styles and brands.

Sending GCODE to the Printer

While not precisely a term, it is something that trips up most beginners. The MPMD comes with Repetier Host on the SD Card. Installing this on a computer and then hooking the computer up to the printer with the USB cable will allow you to manually send GCODE to the printer one line at a time. This is necessary to complete most of the recommended calibrations of the printer. Repetier Host isn't the only way to accomplish this, but it is included with the printer and doesn't require much searching around to get it working. How to get Repetier Host working will be handled in separate post (or later in the 101 guide). While Cura can do the same thing, it's not recommended due to the buggy way it handles directly connecting to the printer. Your mileage may vary.

L and R Values

L stands for the length of the 6 rods that connect to the print head. R is the radius (measurement from the exact center of a circle to the outside edge) of the print bed. Adjustments to these values are key to getting prints that are the same size as the dimensions for the object in the STL and/or GCODE file. They also come into play when getting the print head to move in a level manner. These values can be sent to the printer using the following GCODE: M665 Rxx.xx Lxxx.xx where X is the value of L and R. Example: M665 L123.0 R65.5

Leveling the Bed Part 1

This term is a bit of a misnomer. You're not adjusting the print bed itself, you're adjusting the way the print head moves across the surface of the bed to get it to move in as flat of a manner as possible in relation to the surface of the print bed. Getting this adjustment right goes a long way to making sure you get successful prints of good quality that stick to the print bed. Dennis Brown has a really good write up on making these adjustments complete with a spreadsheet that does all of the calculations for you. However, before jumping into that, there are other calibrations/upgrades you need to perform, detailed in this MPMD Calibration Roadmap: bit.ly/mpmdfaq

Levelling the Bed part 2

The printer itself tries to calculate the surface of the print bed using a pre-programmed series of taps onto the print bed. The GCODE command that tells it how to make that measurement is G29. There are several different methods you can instruct the printer to use. These are the Pattern or P Values. The commonly used ones are P3, P4, P5, and P6. The most effective one, for my printer with a glass bed that is, seems to be P5. What will work for you really depends on several different factors. Once you have gone through Dennis's calibrations, he recommends using the P5 or P6 pattern. You'll need to do some test prints to determine which one is going to work the best for your printer. The command to send to the printer for that is G29 P5. You can also include a Z offset. Z is the height value for the print head, or how far from the print bed it should start printing the first layer. Brian Kurtz, who's profiles I highly recommended you use, recommends starting with Z0.28 and making adjustments in 0.02 increments up or down to get a good first layer. Putting that all together, the GCODE is G29 P5 Z0.28

Startup GCODE

You put all of the things above together in the Startup GCODE. In Cura, this is set under Settings, Printer, Manage Printer, Machine Settings. On my computer, I have to use the mouse and grab onto a corner or side to make the window big enough to see the code. If you're using Brian's profiles, the majority of this is already set for you*.

*Some things to be aware of when looking at your Startup GCODE:

M92 - Depending on which version of firmware and its related hardware you have, this line can have two different settings. If you have V41 or V45 firmware, you have 1/8step Stepper motors, therefore it should be

M92 X57.14 Y57.14 Z57.14

For V43 and V44 Firmware you have 1/16 stepper motors and should use this:

M92 X114.28 Y114.28 Z114.28

Semicolon ;

This tells the printer to ignore anything AFTER the ; on that line only. It's a great way to turn things on and off in your start code as well as add notes to a line as reminder as to why that line is there.

Examples:

Adding Comments to your GCODE:

M29 P5 Z0.28 ;Runs the leveling routine

M666 X0 Y-0.34 Z0.43; Numbers from calibration spreadsheet on 2-14-2018

Prevent a command from being sent:

; M665 L123.0 R63.5

M500

Saves your settings to the printer. Even this can differ based on firmware as to which settings will actually stick through a power cycle. More on this in Dennis' alignment guides and eventually the 101 guide

Optional: Buy me a Coffee

This section is COMPLETELY OPTIONAL. Most of you know my name from the Facebook Group, but for those who got this link from someone else, I'm Brian Kurtz. I'm a nerd, techie, and Theatrical Lighting Guy for a living. I wrote this guide out of a love for the 3D printing hobby, and would gladly help people all day for free if I could. But like most of us, I have a full time job, so writing this guide is my shortcut to help as many people as possible and make teaching much faster.

I've spent months, almost a year at this point, absorbing all the knowledge of the Facebook group for the MPMD and modding/tweaking my printer, profiles, settings and process. My printer is barely recognizable, controlled by OctoPrint and about as level as I've seen any MPMD, but I remember the frustration of trying to get my prints to stick and my supports to come off clean. Compiling all this knowledge and work into profiles/guides for the group was just a fun project for me, as this is a hobby I thoroughly enjoy and a wonderful community around it.

I'm still learning too, and there's definitely more info and updated profiles to come. But I'll reiterate this… The guide you just read and the profiles it linked to are now and will ALWAYS be COMPLETELY FREE, so share them and use them to improve your own experience and others, modify them, experiment, learn, even improve them (though if you do, send me the tweaks for future releases, lol).

The biggest reward is to see the wonderful things people can print with this little machine, even considering all the potential issues that certain MPMDs come with from the factory, so the free and easy way to reward my time is to share this guide and its profiles with those who need help so they can get their printer up and running. The short link to this guide is **bit.ly/MPMD101** and the profiles are **bit.ly/curampmd** so just copy those and send them to people who need some help and advice!

With all that said, I know a lot of people prefer to show their thanks in more than just words, so ONLY IF YOU WANT TO, I've included donation links below to send me a few bucks. There's no minimum, and don't break the bank if you CHOOSE to donate/tip, but you can choose an amount you feel is appropriate for my knowledge/time/settings. I will keep updating this manual and my profiles as time allows, and they'll always be free.

PayPal: paypal.me/BSK42 or red@brianskurtz.com

Google Pay/Wallet: brianskurtz@gmail.com

Venmo: @RedFliesBy

Facebook: Can send through Messenger

Please make a note that your donation is related to this guide or the MPMD so I don't get confused. If I'm missing a big payment service, let me know…

David's Comments

Brian Kurtz started and maintained this document on his own for quite a long time. He eventually gave edit access to this document to me, David Keeton. You probably also know me from the Facebook Group, Reddit, Calibration Roadmap, or maybe even my YouTube channel. Up until recently, I kept my updates relegated to adding a couple of minor troubleshooting steps and adding/updating links throughout the document. More recently, I worked with others in the group to add the Monoprice Mini Delta printer definition to Cura 4.5 and beyond. This called for a much larger update in MPMD 101 than what I've typically done in the past, but I did save the old Cura installation instructions into a backup document. This is still mostly Brian's document, so by all means, buy the man a coffee!

The people in the MPMD Facebook Group are pretty great. New users join everyday, and even though things have gotten much, MUCH easier thanks to the combined efforts of many, the questions keep shifting. If you see the opportunity, please take the time to answer some of these questions, even if it is with a simple link to MPMD 101, the Calibration Roadmap, the Basic Calibration/Troubleshooting page, or something from the YouTube Playlist. Cheers!