

# Introduction to FDM 3D Printing

FDM (Fused Deposition Modeling), alternatively referred to as FFF (Fused Filament Fabrication) is a method of creating 3D objects out of thermoplastics by laying down a single layer at a time out of a hot end and nozzle, which heat the plastic filament to near its glass transition temperature (a state of matter past solid, but not quite liquid, more of a viscous, rubbery state). Different materials require different temperatures and settings to get good prints, but those are the very basic concepts of FDM Printing.

## General Workflow



As with most things in a makerspace, the work process generally begins with either solving a problem, or creating something for a specific purpose which is not easily obtainable otherwise. For the purpose of this class, we will be skipping over both ideation and 3D Modeling (as important as they both are, they deserve their own class). As such, we then begin with:

## Material Considerations

Different materials have different physical characteristics, as well as different ideal print settings and environments. There are a bevy of materials available (too many to list every single one here), but for general use FDM printing on our machines at MakerLab, the most important ones to cover include:

- PLA (Polylactic Acid): The most widely used filament base. Highly recommended for new users and users looking for a no fuss experience.
- ABS (Acrylonitrile Butadiene Styrene): Printed parts can withstand higher temperatures and are generally more impact resistant. Quite susceptible to temperature and printer environment while printing, as well as issues with warping while printing. Not recommended to print without an enclosure.
- PET (Polyethylene), or PETG (Polyethylene Glycol): Printed parts have a greater temperature resistance, but not as much as ABS, and are more impact resistant than PLA (in this case because they're more flexible and less rigid). Good UV resistance for outdoor parts. Tends to print slower and hotter than PLA.
- TPU (Thermoplastic Polyurethane): Flexible when printed (depending on rated shore hardness as well as print settings). Good impact resistance because of the flexibility of the material. Tends to print better on Direct Drive Printers than Bowden.
- PP (Polypropylene): Parts are tough and flexible, similar to PETG but in some cases can outperform it. More difficult to print due to warping and adhesion issues.
- Nylon: Very impact resistant, with better heat resistance than PLA. Very susceptible to moisture absorption as well as the printer environment (Not recommended to print without an enclosure).
- PC (Polycarbonate) or Carbon Filled: Extremely rigid and strong prints. Good temperature tolerance. Requires a hardened nozzle to print.
- Wood/Metal filled: Usually added to PLA. Adds unique characteristics, but can degrade the nozzle quickly when used.



# SLICING

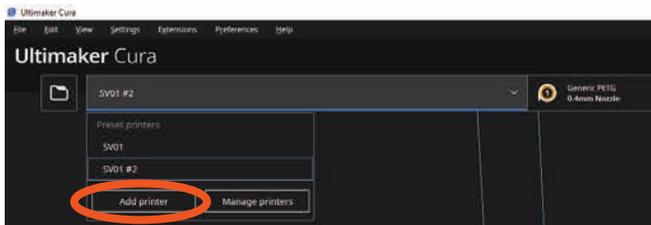
For this class, we will be learning how to slice a file using Cura. There are multiple other slicers out there (PrusaSlicer, Slic3r, IdeaMaker, Simplify3D, Repetier to name a few), many of which have their own advantages and disadvantages. For our machines, Cura is a nice, one-software-fits-all solution. We'll be using Cura 5.1.1 and above, to take advantage of Cura's new Arachne slicer engine, which brings some nice quality of life upgrades to things.

The job of a Slicer is to take a pre-made 3D model, and turn it into a set of GCode instructions which tell the printer how to behave. While it is possible to write GCode, using a Slicer makes the process much easier and more intuitive.

**File Types:** .STL, .OBJ, and .3MF are the most common file types you will see for 3D Printing. Any will work fine in Cura, as long as the original 3D model doesn't contain any geometry errors or is not manifold (these can sometimes be repaired in Cura if you run into an issue). It's also common to see .STEP files, which contain a set of instructions which preserve the editability of the 3D model. These four file types are the ones you're most likely to run into.

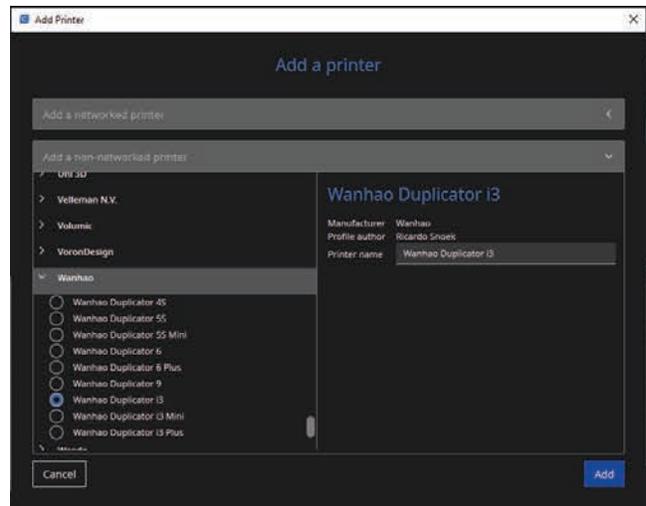
## Setting Cura Up for a Specific Printer

While the design stations at MakerLab will have Cura pre-configured for our 3D Printers, you can also download and install it on a personal machine if you'd like. We'll walk through the basic setup steps for our FDM Printers below. Regardless of the printer, the process will begin by clicking the 'printers' menu in the upper left of Cura, and going to 'Add printer'. In the pop-up menu, you will select 'add a non-networked printer'.



### Wanhao Duplicator i3

The Wanhao is the easiest to setup, as Cura has a profile already made for it. Simply scroll down to 'Wanhao', and select the Duplicator i3. Now proceed to Add the printer, and you'll be set up in no time.



### Lulzbot Taz 6

The Taz 6 will require a little bit of custom Gcode in it's printer profile to work properly with Cura, but don't worry, it's a simple copy-paste to get it up and running. To get started, you'll want to select 'Custom' -> 'Custom FFF printer' in the above non-networked printer menu. In the next menu, you'll want to change the following parameters:

Build plate:

X: 280mm

Y: 280mm

Z: 250mm

and make sure to tick 'Heated bed' to 'yes'.

The back of this page has an image of all the correct Taz6 settings, as well as the GCode.

# Machine Settings

## Custom FFF printer

Printer	Extruder 1		
<b>Printer Settings</b>	<b>Printhead Settings</b>		
X (Width)	280.0 mm	X min	-20 mm
Y (Depth)	280.0 mm	Y min	-10 mm
Z (Height)	250.0 mm	X max	10 mm
Build plate shape	Rectangular	Y max	10 mm
Origin at center	<input type="checkbox"/>	Gantry Height	250.0 mm
Heated bed	<input checked="" type="checkbox"/>	Number of Extruders	1
Heated build volume	<input type="checkbox"/>	Apply Extruder offsets to GCode	<input checked="" type="checkbox"/>
G-code flavor	Marlin		
<b>Start G-code</b>	<b>End G-code</b>		

```
M73 P0 ; clear GLCD progress bar
M75 ; start GLCD timer
G26 ; clear potential 'probe fail' condition
M107 ; disable fans
G90 ; absolute positioning
M420 S0 ; disable previous leveling matrix
M140 S{material_bed_temperature_layer_0} ; start bed heating up
M104 S{material_standby_temperature} ; soften filament
G28 ; Home all axis
M117 Heating... ; LCD status message
M109 R{material_standby_temperature} ; wait for temp
M82 ; set extruder to absolute mode
G92 E0 ; set extruder position to 0
G1 E-10 F100 ; retract filament 10mm
M104 S{material_standby_temperature} ; set to wipe temp
;M106 ; turn on fans to speed cooling
M109 R{material_standby_temperature} ; wait for to reach temp
M107 ; turn off fan
;M206 X0 Y0 Z0 ; uncomment to adjust wipe position (+X - nozzle moves
left)(+Y - nozzle moves forward)(+Z - nozzle moves down)
G12 ; wiping sequence
M106 S255 ; turn on fan to blow away fuzzies
G4 S5 ; wait 5 seconds
M107 ; turn off fan
;M206 X0 Y4 Z0 ; resetting stock nozzle position ### CAUTION: changing this line
can affect print quality ###
M109 R{material_standby_temperature} ; wait for extruder to reach probe temp
G1 X-10 Y293 F4000 ; move above first probe point
M204 S100 ; set probing acceleration
G29 ; start auto-leveling sequence
M420 S1 ; activate bed level matrix
M204 S500 ; restore standard acceleration
M104 S{material_print_temperature_layer_0} ; set extruder to print temp
;M425 Z ; use measured Z backlash for compensation
;M425 Z F0 ; turn off measured Z backlash compensation. (if activated in the
quality settings, this command will automatically be ignored)
G1 X0 Y0 Z15 F5000 ; move up off last probe point
G4 S1 ; pause
M117 Heating... ; progress indicator message on LCD
M109 R{material_print_temperature_layer_0} ; wait for extruder to reach printing
temp
M190 R{material_bed_temperature_layer_0} ; wait for bed to reach printing temp
G1 Z2 E0 F75 ; prime tiny bit of filament into the nozzle
M82 ; set extruder to absolute mode
M400 ; wait for moves to finish
M117 TAZ 6 Printing... ; progress indicator message on LCD
```

```
M400 ; wait for moves to finish
M140 S45 ; start bed cooling
M104 S0 ; disable hotend
M107 ; disable fans
G91 ; relative positioning
G1 E-1 F300 ; filament retraction to release pressure
G1 Z20 E-5 X-20 Y-20 F3000 ; lift up and retract even more filament
G1 E6 ; re-prime extruder
M117 Cooling please wait ; progress indicator message on LCD
G90 ; absolute positioning
G1 Y0 F3000 ; move to cooling position
M190 R45 ; wait for bed to cool off
G1 Y280 F3000 ; present finished print
M140 S0 ; cool down
M77 ; stop GLCD timer
M84 ; disable steppers
G90 ; absolute positioning
M117 Print complete ; progress indicator message
```

If you don't want to type all of these settings in, you can find a link to copy/paste at:

<https://community.ultimaker.com/topic/41188-cura-51-and-lulzbot-taz-6/>

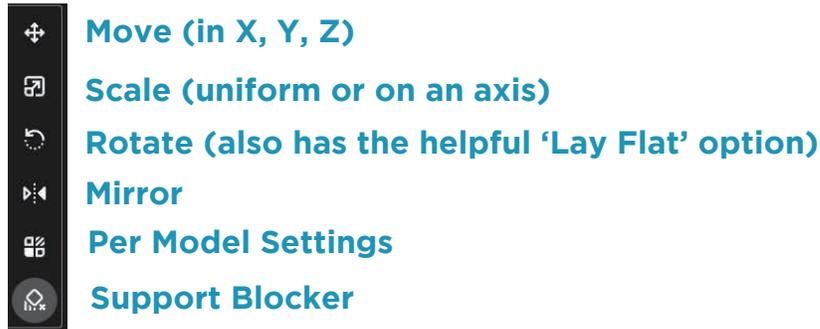
\*with no space between '-51-' and 'and'

Once this is all copied in, you can proceed to Add the printer, and the Taz will be set up in your version of Cura! It's a little bit of a process, but you'll only need to do it once!



First, you'll want to open your file. This can be done from the File menu, or at the top left corner. With the file open, you can select which printer you'd like to slice it for, select what type of filament you're planning on using (if your specific brand/type of filament is not in the drop-down list, you can use the far right drop-down to set specific settings for it).

With your model open, you'll see a toolbar on the left for orientation and advanced support settings. The bottom left has a toolbar to help keep you oriented in the 3D workspace.



### Camera Orientation Pre-Sets

Many filament brands and types have pre-made profiles available either in Cura, or in the Cura marketplace. This is a good place to start looking, especially if you're new to 3D Printing. If something does not have a ready made material profile, it's generally a good idea to check the manufacturer's website, Amazon or other marketplace reviews, and/or do a google search for \_\_\_\_\_ filament best settings, to see if someone else has already done some work for you.

## PRINT SETTINGS

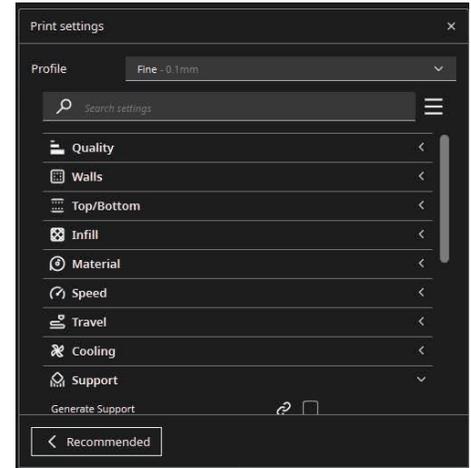
Cura has a vast number of settings which can be tweaked in this menu. Having so many settings can feel overwhelming if you're new to 3D Printing, so you may want to go into the top menu of Cura to Preferences -> Configure Cura -> Settings, and in the drop down menu in this pop out select 'Basic'. This will limit the displayed Print Settings to only the most basic ones for getting started. For this class we'll focus mainly on these settings, with a few exceptions as they apply.

**Quality Layer Height** will set the height of each layer. The Taz 6 uses a .5mm nozzle, and the Wanhao uses a .4mm one (most printers today use .4 by default, but nozzle sizes can always be swapped). You do not want to set this setting any higher than the physical nozzle size on the machine you're using, and in most cases will want to limit it to roughly 3/4 of the actual nozzle size. A good rule of thumb for both machines is to use .2mm in this setting for faster prints with an acceptable quality. .1mm will print slower, but have more fine details. Generally you will want to stay between .2mm and .1mm until you're a little more comfortable with printing.

**Walls** Typically, **Wall Thickness** will be auto-calculated based on your other settings. If you are having issues with over- or under- extrusion which are not due to print temperature, changing this value can help compensate in some cases (many PETGs are a good example, where setting wall thickness a little higher as well as lowering **Travel Speed** can help produce a good result).

**Wall Line Count** is how many walls a printed part has to its' exterior surfaces. Cura tends to default this value to 2, we recommend using at least 3 walls in most cases (Doing so helps with part structural integrity and will help any overhangs to better adhere). Setting this value higher will generally produce tougher parts, but will add a little bit to print time.

**Horizontal Expansion** is set to 0 by default. This setting can also help with under extrusion



### Print Settings

if you find yourself experiencing that issue. Setting this value higher in .2mm increments can help fill gaps created by under extrusion.

**Top/Bottom** These settings are auto-generated by other values you set in Cura, and you typically want to leave them that way. Changing these is best accomplished by changing the **Top Thickness, Bottom Thickness, or Top/Bottom Thickness**, and letting Cura calculate the other values. If either the top or bottom of your 3D Print is having issues with the filament going together, increasing these values can help give your print a little more thermal mass in these finicky areas to help achieve better prints.

**Infill** will set both the amount of material deposited on the interior of your part, as well as the pattern it is laid down in. For most prints, we recommend using a Cubic **Infill Pattern**, and roughly 20% **Infill Density**. Increasing **Infill Density** can add more structural strength to a part, as well as helping to make things more watertight or liquid-tight, while decreasing this value can save on material use. Using too low of an infill can cause sagging and print issues, depending on the 3D model being printed. Feel free to play around with the **Infill Patterns**, but generally Cubic will give good results in both strength and part quality.

**Material** is generally the first place you want to look if you're having issues with a print. **Printing Temperature** is the temperature the nozzle will be heated to, and **Bed Temperature** sets the temperature of the print bed. Most filaments will list their recommended Print and Bed Temperatures on the roll, making it easy to figure these numbers out. If you're having issues and suspect that one or both of these are to blame, you can always refer to the material chart in this guide, or Google the filament in question to try to find best settings online. Printing a **Temperature Tower** (another Google search to find one you like) can also help you determine what the best temperature settings for a material are.

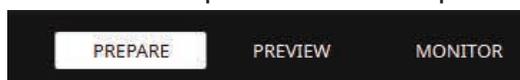
**Speed** in a general sense controls the speed at which the print head moves around. This is another setting which can help you with troubleshooting if you're having issues. Some materials (TPU, flexible filaments, and PETG, for example) tend to print much better at lower speeds.

**Travel** the basic settings here will control whether **Enable Retraction** is ticked, and whether **Z Hop** is ticked. **Enable Retraction** will tell the gears which feed the filament into the hotend to pull the filament back a bit when the print head is travelling without extruding. This helps prevent stringing (little 'strings' hanging off of a print), as well as creating a cleaner and more reliably uniform print surface on the part. **Z Hop** will lift the Z axis a minute amount while the print head is travelling without extruding, and can help in prints which use a lot of vertical supports or have tall pieces to them (generally tall prints are more prone to failure, **Z Hop** can help with this a bit).

**Cooling** controls whether the part cooling fan is turned on, and how fast it will spin at maximum **Fan Speed**. Cura's automatic settings for Cooling are generally pretty good, but if you're getting poor layer adhesion (visible lines between layers), lowering the **Fan Speed** by increments of 5-10% can help solve this, depending on the filament used.

**Support** will place support material under any overhanging parts which overhang by an angle greater than 45 degrees. In the part preview window, overhangs which may fail to print without support will be highlighted in red, letting you know if you should enable support or not.

**Build Plate Adhesion** controls the method used to adhere a printed part to the build surface. This can vary depending on material and the 3D model. Generally, **Skirt** will leave a thin line around the part to be sure the nozzle is laying down material well before it gets to the part itself, **Brim** will create a 1 layer thick surface touching the print to help it adhere to the bed and not release or warp, and **Raft** creates a 4 layer surface around and under the part, which can be helpful with more difficult the print materials to prevent warping and part failure.



**Prepare** is the workspace in which most Slicing work is done. Once you settings are in place, you can click **SLICE** in the bottom right corner to generate the GCode file. Once a file is **SLICED**, you can then use the **Preview** tab to get a layer-by-layer estimation of how the file will print. In this workspace, the right hand side of the window will let you scroll through layers to preview how they will lay down and is helpful to identify potential problems before they eat up hours of your time. Once a file is fully **SLICED**, you can either send it to the printer over a USB connection (Taz 6), or save it on a Micro SD Card to insert into the printer (Wanhao Duplicator i3).

# Machine Setup

Our 3D Printers operate on many of the same principles, but have some important distinctions between them which will affect how you get the machine ready to print your part. Be sure that you're using the same machine you sliced the file for in Cura, otherwise you will likely have a bad experience.

## Lulzbot Taz 6

The Taz 6 uses 2.85mm diameter filament, and has a larger build volume. To load filament

1. Power the machine on.
2. In the navigation menu, choose 'Change Filament'
3. Choose either 'Preheat PLA' or 'Preheat ABS', depending on the temperature needed by the filament you are using. Getting an exact temperature match isn't crucial here, so consult the Material Chart and pick which one more closely fits your filament.
4. The printer will home, begin heating, and unload any filament that is loaded. While it's doing this you can get your filament loaded on the spool holder and ready to go.
5. When prompted, disengage the filament guide on the print head, feed your filament in until you get a bit of resistance, then engage the filament guide once again. Sometimes the guide may need to be adjusted a bit, this is done with the two thumbscrews on the left hand side of it.
6. Confirm in the panel that the filament has been loaded. The machine will feed some through, let it finish then discard the excess filament it fed through.
7. With the filament loaded, you can either load your sliced file into the Cura install on the printer laptop, or attach your own laptop via USB with Cura set up for the Taz 6. Either way, once the filament is loaded, be sure your file is sliced, then hit 'Send' in the lower right corner of Cura. The printer will run through a homing and leveling process, then begin printing.
8. Watch the first few layers go down, as this can be a good indication if the print will have any issues. Making sure the first layer has good adhesion and doesn't have any weird issues is always a good idea. After confirming this, the print will proceed as needed, and finish close to the estimated time given in Cura.
9. To unload filament, select 'Change Filament' again, and repeat the above steps until the filament is ejected from the print head. Now you can power the machine off and put the filament away.

## Wanhao Duplicator i3

The Wanhao uses 1.75mm diameter filament, and has a bit smaller total build volume. To load filament:

1. Power the machine on (switch is one back of control unit).
2. In the navigation menu, choose 'Quick Settings' -> 'Preheat PLA' or 'Preheat ABS', depending on the filament you're using. Try to roughly match the nozzle temp to whichever material your filament is closest to.
3. Back out of the menu so you're looking at the main screen. Here you can monitor the nozzle and bed temperature. Wait for the nozzle to heat to either 215C or 240C, depending on what you chose above. While the nozzle is heating you can get your filament ready and on the spool holder.
4. Once the nozzle is within a few degrees of its target heat, insert the end of your filament roll into the PTFE tubing. Don't force it in, just put it in until you feel a little resistance.
5. In the navigation menu, select 'Extruder' -> 'Extruder Position'. Turn the dial clockwise to feed filament into the hot end. This interface can be a little finicky, but stick with it a few turns at a time until you get a steady flow of filament out of the nozzle. Remove any of this extra filament, and make sure the print bed is clean (Isopropyl Alcohol and a rag work great for this).
6. Back in the menu, choose 'Print File'. Be sure that the micro SD card is inserted in the machine with your sliced file on it BEFORE powering the machine on. If not, just shut the printer off and turn it back on with the micro SD in. In the 'Print File' menu, navigate to your sliced file (the newest file will populate at the bottom of the list, so just scroll down a bit to find it). Select your file, then wait for the printer to home and begin printing. If there's any material ooze from the nozzle while it homes, feel free to carefully remove it with a pair of tweezers.
7. Wait to confirm that your first layer has successfully layed down. A lot of print issues can be solved by making sure the first layer adheres properly. After this, let your print run until it is finished. Watching the first several layers can often give a good idea of if there are any issues to deal with.
8. To unload filament, pre-heat the nozzle, and use the 'Extruder Position' to back the filament out until it can be pulled free from the PTFE tubing.

# Machine Panel Options

All our machines can be tweaked and operated from the front panel. Some of these options are for advanced users and should be left as they are, and some are necessary to use the printer. Some of the more necessary options are:

## Lulzbot Taz 6

**Movement** will allow you to **Disable Steppers** in order to move the build plate manually, and **Move axis** if you need to lift the Z due to a failed or problematic print.

**Temperature** controls the **Nozzle** and **Bed Temp.** if you need to set either outside of loading in filament. **Cooldown** will cool down both.

**Configuration** contains more advanced settings, and should not be changed.

**Change Filament** will allow you to change the filament, with presets or a custom nozzle temperature.

**Print from SD** will let you print directly from the SD card if you need to (it's recommended to print from USB most of the time on the Taz).

## Wanhao Duplicator i3

**Quick Settings Preheat PLA/ABS** will pre-heat the nozzle to the necessary temperature.

**Cooldown** will cool down the nozzle and the bed. **Disable Stepper** will let you manually move the print bed, if you need better access to a part.

**Print File** is where you select the file to print.

**Position X Pos., Y Pos., and Z. Pos** will let you move the individual axes. **Z. Pos** is especially helpful if you have a failed print and need to get the print head out of the way.

**Extruder Extruder Position** is where you will control the loading and unloading of filament into the machine, after the hotend is heated.

**Fan Speed** this should be ideally left alone and controlled via Cura.

**SD Card** if you notice a print failure or need to restart a print, this menu will let you **Stop** or **Pause** the print.

**The final two main menu options are for more advanced users, and should not be changed.**

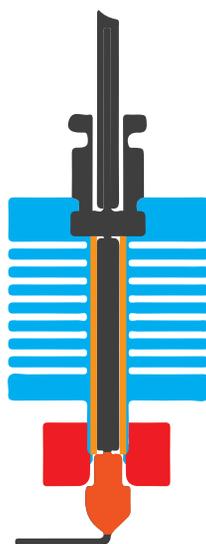
# Machine Operation

FDM Printers operate by applying heat specifically where it is required, and getting rid of it when and where it is not. Outside of other machine aspects (motors, control boards, motor drivers, etc.), this is the basic principle of printing. Much of this happens in the print head between the **extruder, hotend, heat break, heatsink, heat block, thermistor, and nozzle.**

**Hotend** is made up of several elements, all of which work together to apply heat directly to the filament to heat it to its glass transition point. These include:

**Heat Break** is the internal conductor of heat, which pre-heats the filament as it travels toward the Heat Block.

**Heatsink** is meant to rapidly remove heat from the hotend, so the heat is primarily directed toward the inner chamber where the filament passes through, and thermal runaway doesn't occur. One or more fans are mounted near the heat break to help remove excess heat.

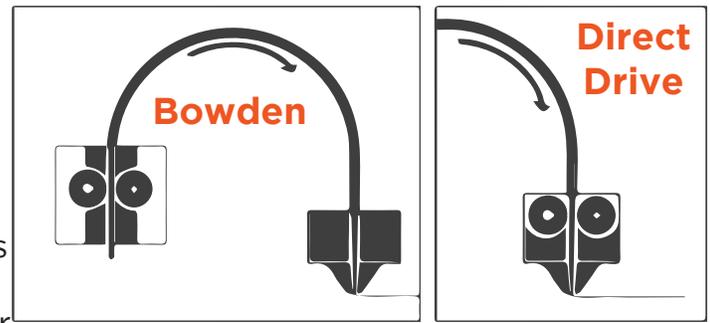


**Extruder** is what applies force to push the filament through the heated elements to the nozzle. There are two types (see next page).

**Heat Block** is the element which heats up via electrical resistance to provide heat to the rest of the hotend. This is also where the **Thermistor** is mounted, to keep accurate track of how much heat is present.

**Nozzle** is what the filament is ultimately fed through. Typical sizes range from .1 to .8mm openings, but for general use .4mm is extremely common.

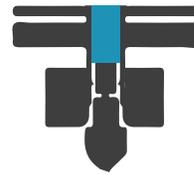
**Extruder Types:** The primary methods of forcing filament into a hotend are known as **Direct Drive** and **Bowden**. The primary differences between the two lie in where the force is applied to the filament in order to move it forwards and backwards (extrusion and retraction). **Direct Drive** systems put the applied force right next to the hotend. This gives a mechanical advantage with filaments such as TPU and to a degree PET/PETG, which can bind up easier in a **Bowden** system, but the trade-off is it tends to make the print head heavier, meaning it can't move quite as fast as a **Bowden** can. **Bowden** systems mount the gearing which applies the force somewhere on the printer frame, and it is fed through a line of PTFE tubing to reach the hotend.



**PTFE Lined vs. All-Metal:** A difference between printer extruders is their material composition. PTFE lined extruders have a small run of PTFE tubing which feeds the material into the hotend in a low-friction environment. They ease the travel of the material, but are more susceptible to the PTFE carbonizing over time, which then creates clogging issues if not replaced. All-Metal extruders solve this problem by using a tube made of one type of metal to direct material to the hotend, and generally need less maintenance. Bi-metal hotends are tubes made of two or more types of metal, and are generally engineered to apply as much heat as possible to a material, whether because the material requires it or in order to speed up printing time. In a PTFE lined setup, it is important that the PTFE contacts the nozzle as directly as possible - if there is a gap between the two it can create backpressure, as well as creating a spot which is likely to start oozing out of the top of the heater block, both of which can easily ruin a print and are a pain to repair/deal with.



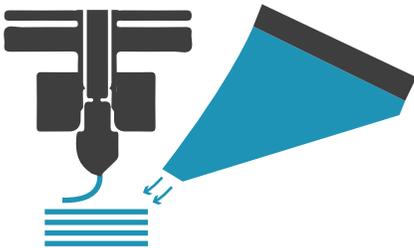
Little or no gap creates an ideal printing environment.



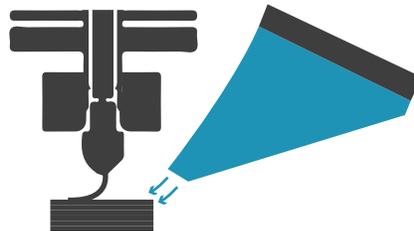
Significant gap can create back pressure and overflow issues out of the heat block.

**Nozzle Types:** The most common nozzles are made from brass or sometimes copper, both of which have good thermal conductivity, but are more susceptible to wearing out. Hardened steel or Tungsten carbide nozzles are the better choice for abrasive filaments (metal or wood filled or containing Carbon particles), but conduct heat less effectively, thus require more heat from the hotend in order to print effectively.

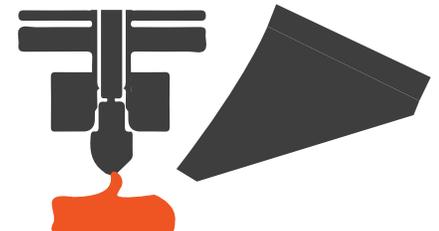
**Part Cooling:** In addition to the fan(s) which help to cool the Heat Break, most all FDM printers will also employ one or more cooling fans to cool the filament once it has exited the hotend and nozzle. Ideal part cooling will allow extruded filament to adhere properly to either the build plate or the previous layer, while eliminating the likelihood of warping, drooping, or just an overall globbed up mess of a print. It also is key in maintaining dimensional accuracy where it is needed in a part.



Too much cooling. Layers don't adhere properly and ringing/layer shift are more likely.



Ideal cooling. Good layer adhesion, and the intended part structure is maintained throughout the print.



Not enough cooling. Parts will tend to warp and deform if they don't just turn into a blob of goo.

Good part cooling will typically direct the cool air provided by a fan more towards the part itself than the nozzle. Some filament types will do better with less cooling as well (PETG for example).

# Troubleshooting

There are a lot of variables in 3D Printing, and figuring out which specific one(s) are causing problems can sometimes be a process. A few common problems, and potential solutions:

**Filament not feeding correctly through the nozzle, or there is a big pile of goop on the nozzle or hotend:** This is usually caused by a clog in either the nozzle or the hotend. Nozzles wear out over time, and any residue left behind by a lower temperature filament can potentially carbonize when heated to a higher temperature than it is rated for, both of which can cause a clog. Often a clog can be resolved by heating the nozzle up, and using one of the cleaning tips to gently work the clog out of the heated elements. If this helps get the filament flowing again, it's often a good idea to extrude a little filament to try to grab whatever was causing the clog, then let the hotend/nozzle start to cool down. As the nozzle is cooling, you can do a cold pull (manually forcing a little filament through at a reduced temperature) to help grab any other bits that were causing the issue. Alternatively, this can also be down to extrusion and retraction settings in Cura. If it doesn't seem to be a clog but you're still having extrusion issues, try raising your printing temperature and/or changing your Retraction in Cura (you'll need to enable Advanced or Expert settings for the latter to be visible).

**Elephant's Foot, or warping at the bottom of a print, causing it to lose adhesion with the bed:** Another fairly common issue, this usually comes down to either Print Temperature, Bed Temp., excessive cooling on the first few layers, or Horizontal Expansion/Speed. Essentially what is happening is that some parts of the print are cooling faster than others, at a rate which is causing noticeable warping. Printing a Temperature Tower can help you determine if it's an issue with nozzle temperature, if that's not it we recommend next decreasing your initial cooling, followed by increasing Speed incrementally and/or increasing Horizontal Expansion in small increments.

**Print not sticking to the bed, or releasing mid-print:** Different materials have different properties, and getting one material to correctly adhere to the print surface can be quite different from another. Sometimes using a little bit of hairspray, or a layer of painter's tape on the print bed can help with adhesion. A good tip for ABS is to make a slurry out of a little bit of the filament dissolved in Acetone, and paint a thin layer of that on the bed. Glue sticks can help in a similar manner. This can also be a matter of print bed temperature. There are multiple ways to improve adhesion, if you are having an issue a Google search of your specific filament can often help solve the problem.

**Poor layer adhesion:** This will be apparent if there are small gaps between your printed layers. This can usually be solved through increasing the print temperature in 5C increments, increasing your layer height in small increments, lowering your speed, or reducing your cooling. Increasing Horizontal Expansion can also help, and playing with your Top/Bottom settings can if the issue is more prominent toward the bottom or the top of your printed part.

**Stringing:** This is what we call it when you get a bunch of small 'strings' or 'fuzzies' coming off of a surface which is meant to be smooth. This usually has to do with temperature and extrusion, printing an extrusion tower or extrusion test is the first place to start (Google is your friend here).

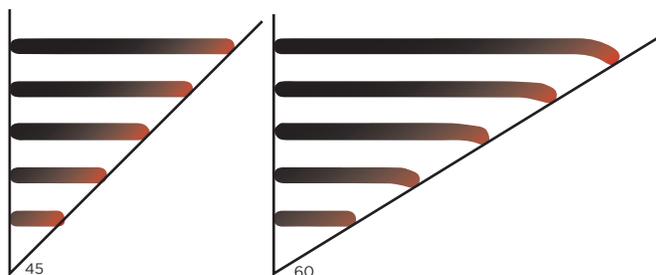
**ringing or layer shift:** Ringing is when the individual print lines of each layer are more pronounced than usual, and layer shift is the extreme end of this, when a layer will noticeably shift from where it should be. The primary culprit in both is usually Speed - when the print head is moving so fast that the motors may skip steps, or the GCode may lose track of correct head positioning. Sometimes fast head movement is ideal, but if you're having either of these issues, try reducing your speed, and potentially reducing your printing temp. a bit as well to compensate.

**'Bubbles' in print surface:** There are two main reasons you may see this occur - either the filament is being heated too much in the hotend, or it has absorbed too much moisture by being improperly stored. All polymer filaments are at least a bit hygroscopic - meaning they like to absorb moisture if it's available in the atmosphere they're stored in. You can usually tell if a filament has absorbed too much moisture, as it will become excessively brittle the more moisture it absorbs. Filament spools can sometimes be resuscitated by being run through a drybox - really just a dehydrator for filament. This problem will be much less prevalent if filament is properly stored when not in use - in an airtight container with desiccant pouches. Out of all the filament types, PLA, Nylon, PVA, and ABS are the most hygroscopic, while HIPS is among the least.

**Bed Leveling:** If you're having issues with your first layer not laying down properly, chances are it could be a calibration issue having to do with bed leveling. The Taz 6 will auto-level, but even auto-leveling beds still need some occasional hands-on to maintain that level. If you suspect one of our print beds is getting out of level, please let us know ASAP so we can address the issue promptly. Not all printers feature knobs on the print bed for manual bed levelling, but it is becoming a more common feature. The Wanhao does have leveling knobs located on the underside of the print bed. If you ever suspect the bed is out of level, you can check by running a leveling test print (a search on Google or on your 3D model repository of choice can help you find one, generally it's a single layer grid pattern which is layed down). This test print can show you if there are any areas of the bed where the filament is not properly adhering (whether it's not sticking down, or is not laying down a continuous bead of filament). It's normal for beds to sometimes have small, micro valleys in spots, these are more difficult to solve for with bed level and will require more of a Slicer-based solution. If one side or one corner of the bed appears out of level, that can be easily solved by bed leveling. To level the print bed, you want to get a sheet of regular paper, and move the nozzle from one bed corner to the next, after Homing the nozzle to it's set Z axis home height. At each corner, you want the paper to be able to slide between the nozzle tip and the bed without interference, but not have any further gap than this. It's a good idea to get a little interference, then back that off very slightly until the paper can slide cleanly between the two. Each corner will have a knob below it to adjust the height up/down. Doing this at each corner will suffice for setting the bed level. Many more modern printers can auto-level the bed with a mesh leveling process, we will update this guide as needed to cover that. Please also make sure there is no filament deposition on the nozzle prior to leveling, as this will throw the process off.

## Common Tips

**Bridging** is the term used to describe a strand of filament layed down between two points, without the need for support underneath it. Long bridges can be difficult to achieve, but with sufficient part cooling they're made easier. Ideal part cooling will blow cool air just below the printer nozzle, so that the layer being printed can sufficiently adhere to the one under it, but isn't so hot that it wants to sag or warp.



**Overhangs and Support** are directly related. An overhang is when a printed layer lies outside of the layer beneath it - a general rule of thumb for overhangs is that they're fine up to 45 degrees, and anything over 60 degrees definitely requires a support structure. The 15 degrees in between can vary from material to material, but Cura is fairly good at guessing this.

**Alloys and Material Science** It used to be the case that filaments could be divided among clear lines between different types of material, but with the popularity and ease-of-use of PLA, it has received a lot more attention in terms of R&D. What this means is that there are now multiple alloys and specific formulations of PLA which can perform near the thermal and toughness properties of other materials (ABS, PET, etc.). If you have a specific use case which requires a particular material, then obviously that material is best suited to your use, but if you're okay with approximate performance you can often get good results out of a PLA Pro or PLA +, which tend to be easier to print with. Higher temperature and more industrial materials like Nylon, PC, and Carbon filled still have their own unique benefits, so the playing field hasn't been entirely leveled.

**Bed Adhesion:** Some filament types tend to require specific aids in adhering to the print surface. These may include: painter's tape, kapton tape, hairspray, glue stick, a slurry made of the filament material (mostly ABS for that one). Other, more advanced solutions include PEI coated sheets, Buildtak or similar sheets, G10 Garolite, and other sheet goods which can be cut to a bed's specific size and held in place with binder clips while the print is running. Some materials can bind to specific build surfaces much more permanently than others (glass build plates like the one on the Taz 6 are more susceptible to this), and damage the print bed when removed. If you're unsure about a specific material, please google it before printing to see if it requires any special treatment for proper bed adhesion.

**SLACK** We've tried to build this guide to be as comprehensive as possible, while still maintaining enough brevity that it's not strictly a technical manual. With so many variables present in the world of 3D Printing, we heavily encourage new users to use our Slack workspace if they are having issues, or have any questions they'd like answered. We also don't always have all the answers, so we highly encourage people to utilize their search engine skills if they're having a particular problem.

# Thanks and Happy Printing!