

My First 3D Printer

July 2022

By Gary Reighn

Welcome to the 3D printing club! Here are some notes and thoughts from my experience that may help you get started.

Important Note: *These devices, especially the ones priced under \$500, are very much DIY. You should expect to spend a lot of time setting up, tinkering with, modifying, and adjusting the printer. These are not plug and play. If you are not comfortable with mechanical devices and/or do not have a lot of patience, do not take the plunge. Printing itself can take hours or days depending on the size of the piece being printed. However, once you get the hang of things you will be amazed at the things you can print!*

FDM Printer Recommendations (as of July 2022):

Note: This guide only covers Fused Deposition Modeling (FDM), or Fused Filament Fabrication (FFF) 3D printing, i.e. the use of a roll of plastic filament extruded by a heated nozzle to form a 3D object. Other technologies such as resin printing (SLA, MSLA, etc.) are not included in this guide as they function very differently from an FDM style printer.

NOTE: This is a vibrant and fast-moving area. New printers are being released almost monthly and sometimes at increasingly lower prices. Please do a search to look at some of the latest models before making a purchasing decision.

The printers mentioned here are all made by "Creality", a Chinese manufacturer of a wide range of 3D printers. Most of their models are recognized as some of the best printers for their prices. There are better quality printers (construction quality not quality of the prints made) but you will pay more. The two I would recommend at this time are the Ender 3 and the CR-10 Mini (keep in mind new models are always coming out (like the Ender 5) and prices are always changing).

The Ender 3 V2 is an excellent printer, probably the best for around \$250. There is also a Pro version and a 3X version for a little more with a couple of nice added features. Any are a good choice. The Ender 3 is referred to as a DIY kit. It has all the parts, and they are well labeled but it requires some assembly. Expect to take an hour or two to put it together. The CR-10 Mini on the other hand is mostly assembled and all you have to do is mount the main piece and connect the wires before getting started. The CR-10 Mini runs around \$330. Another difference between the two is the build area. Another newer and excellent printer is the Ender 5. Here are the specs:

Creality	X (Width)	Y (Length)	Z (Height)
Ender 3 V2	220 (8.6")	220 (8.6")	250 (9.8")
CR-10 Mini	300 (11.8")	220 (8.6")	300 (11.8")
Ender 5	220 (8.6")	220 (8.6")	300 (11.8")

I purchased a Creality Ender 2 (precursor to the Ender 3) printer in November of 2017. I also purchased the CR-10 Mini in Dec 2018. If you go with the Ender 3, there are a lot of similarities between that and the two I have, so my experience should translate to what you are using. However, there are some differences, and you can rely on the YouTube video tutorials and Ender 3 Facebook groups for help with those. I do not have any personal experience with the Ender 5 but from others telling it appears to be a top-notch unit with some advantages over the other two.

BEFORE TURNING IT ON, BE SURE TO SET THE VOLTAGE SWITCH TO THE PROPER SETTING FOR YOUR LOCATION (120V for US). Otherwise, you will blow out the printer.

Use for the Printer:

What you plan to do with the printer can affect which model you buy. However, a main difference between the models shown above is the size of the print you can build. If you plan to build big, get the Mini. They also offer other models of the CR-10 up to monstrous 400x400x500mm build sizes. Naturally, they are more money.

The other big issue with regards to the choice of printer is the plastic filament you plan to use. There are three common choices people print with today, PLA (PLA+ and other variations), PETG and ABS. There are others like Nylon, TPE but they are more exotic and for specialized uses and can be more difficult to print with an entry level printer.

3D Printer Filament	Easy to Use	Physical Properties		
		Strength	Flexibility	Durability
PLA	YES			
ABS				
PETG (PET, PETT)				

Note that only PLA shows Yes for ease of use. The chart below will help explain that.

	PLA	ABS	PETG
Temp Required to Print	185-210	210-250	210-240
Heated Bed Required	Optional	Yes	Yes
Heated Enclosure Required	No	Yes	Optional
Toxic Fumes	No	Yes	No

I do all my printing with PLA, a corn-based, biodegradable plastic. It is the easiest to print with and good for lots of things. It also doesn't require extensive ventilation when printing. If you require more strength (i.e. printing drones or something) then PETG or ABS are better choices. PETG is more flexible and durable than PLA and easier to print than ABS but has many of the qualities of ABS. A lot depends on what you will be printing and the environment that it will be used in. Google these filament types to read some articles on their strengths and weaknesses and pros and cons. Also check out YouTube videos on printing with each of these filament types. Your printer will come with a small sample of PLA.

Recommendation: When you buy your printer, also order from Amazon a 1kg roll of any color PLA filament you might want to use. The amount they give you with the printer will not go far, maybe not even finish one print in the case of the Ender 3. The CR-10 gives a little more but not much. **The filament size you must order is 1.75mm.** Avoid the fancy filaments for now like wood, carbon fiber, ABS, PETG, TPU, Nylon, etc. until you get the hang of the printer.

There are several places you can order your printer from. They all come originally from China. Some companies have warehouse in the US so they can ship quickly. Other ship them direct from China which can result in it taking a couple of weeks to get your printer. I have ordered mine from 3Dprintersbay.com and Aliexpress.com.

Software:

You will need several pieces of free software to operate the printer and get the most from it. These are listed below, and all are **free**. Be sure to set up accounts for the ones that require it to be notified of updates and such.

Slicer: This is the program that turns a 3D image file (.stl) into the code the printer will use to do the printing (g-code).

Cura 5.x from <https://www.Ultimaker.com> (latest version as of July 2022)

3D Design Program: The easiest one to use for novices is TinkerCAD. <https://www.Tinkercad.com>.

There are other ones out there so feel free to explore at some point. There are some free ones and others that are very expensive but powerful (and hard to learn!).

Site for 3D Files: There are several sites where you can get 3D files "ready to print". Some files are free, others have a small charge.

<https://www.thingiverse.com/> One of the biggest and best sites. Mostly free with optional tipping of the designer.

<https://www.myminifactory.com/>

<https://cults3d.com/>

<https://www.youmagine.com/>

<https://all3dp.com/1/free-stl-files-3d-printer-models-3d-print-files-stl-download/> - A list of other sites.

Help and Support:

I strongly recommend that you join and/or visit the following Groups on Facebook. They are very helpful when trying to solve printer specific issues with your printer.

[Creality Ender-3 3d printer user group](#)

[Creality Official Ender-3 3D Printer Group](#)

[Creality Ender-3 3D Printer Support & Modding Community](#)

[Creality Ender-3 3D Printer User Group](#)

[The Creality Ender-3 3D Printer Place](#)

[CR-10 MINI Users Group](#)

[Creality](#)

Other Support:

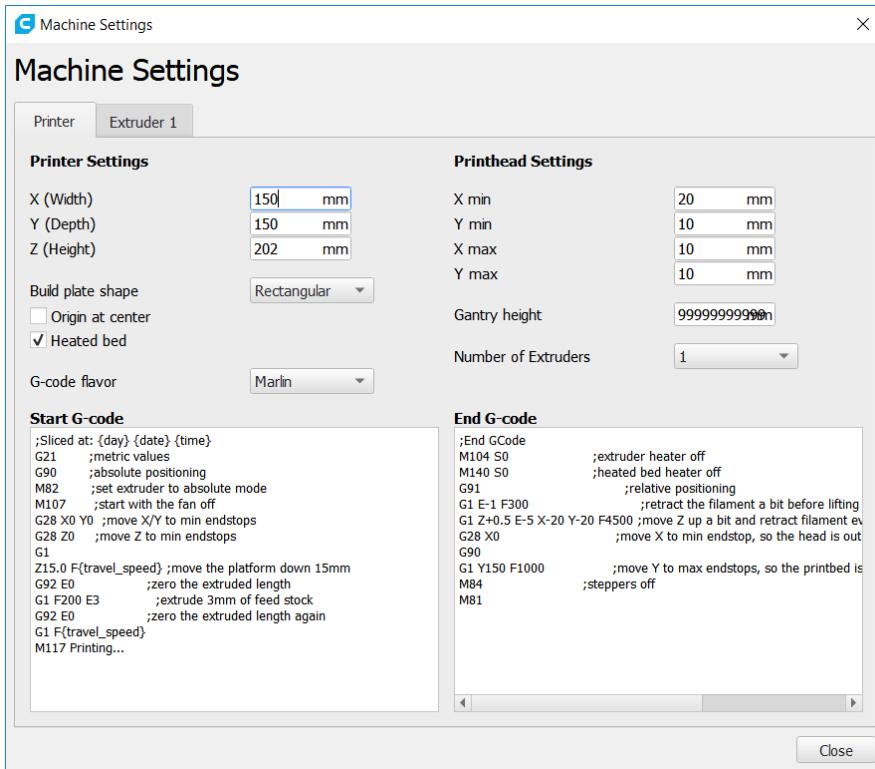
Get use to going to YouTube!! There are hundreds of videos about these printers on YouTube. Just do a search and you will find videos on building the printer, troubleshooting it, improving it, etc.

Getting started...

Using CURA:

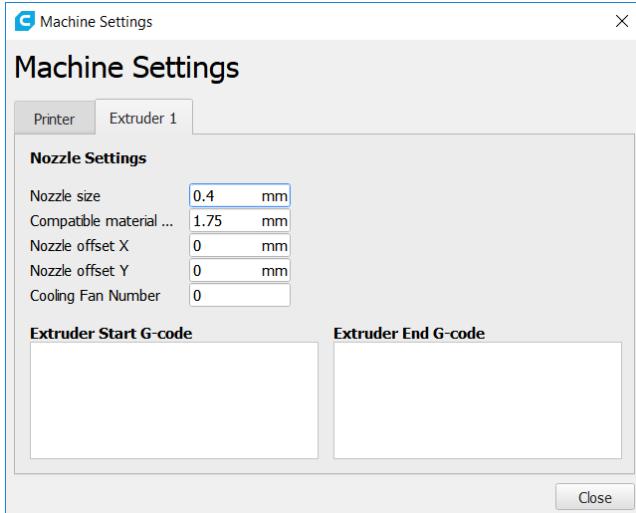
The first thing you must do is set up the printer profile. In Cura, go to Preferences, Configure Cura, Printer, Add and select Other. You should see a list that includes the Ender 3 or CR-10 (same as the mini). Select that.

Here are my machine settings. Yours should be similar except your bed size is larger than my Ender 2's.



The boxes labeled Start G-code and End G-code show the instructions that will take place every time you start and end a print. They do things like moving the print head to the home position and getting the extruder ready to start flowing filament. If your profile doesn't already include this code, check the respective Facebook group for good versions for your model.

Next, check the Extruder 1 settings. It is very important that they match your hardware.

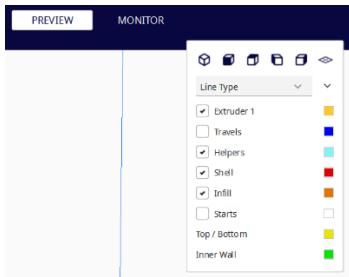


Make sure the nozzle size is the same as what you are using (The default nozzles that come with the printer are typically 0.4). Also make sure the filament size is 1.75mm. Always check this setting after an upgrade.

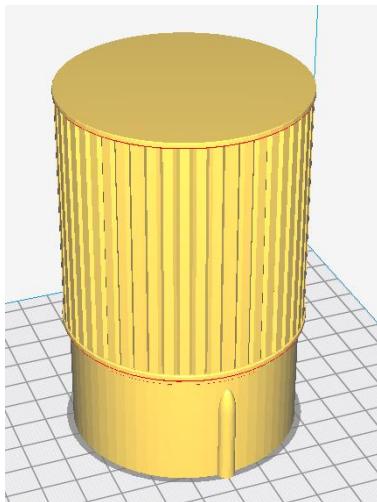
In Cura, select your filament type (again, typically you'll be using PLA). This will provide a default set of settings to get you started. There are literally hundreds of settings you can tweak. I haven't explored even half of them yet, but most are not required to be changed to get good prints.

On the Facebook groups for these printers, they sometimes have a bunch of useful files in the File section including some Cura profiles you can use to start with if you prefer. Check out some of those.

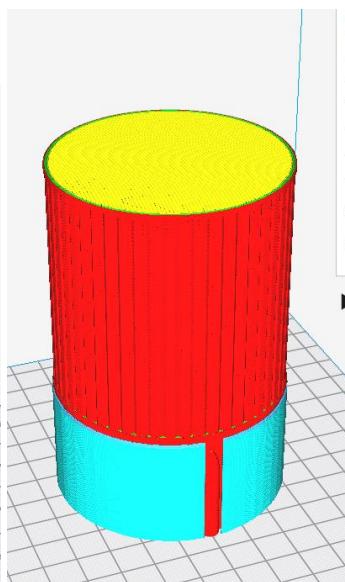
You can use the slider to the right of the build area to see exactly how your filament will be laid down layer by layer. If you click the little right arrow icon, it will make a movie of how the print head will move on each layer. This often can help you identify problems before spending all the time printing. All the different things that can print are shown in different colors on the right. You can select which ones will display on the model by checking the boxes. I recommend checking the ones I have checked below.



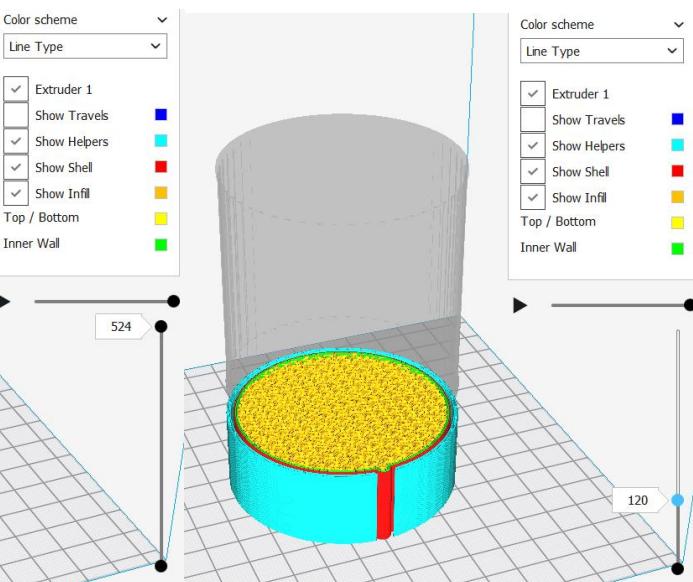
Model View:



Layer View (Slider at top of model)



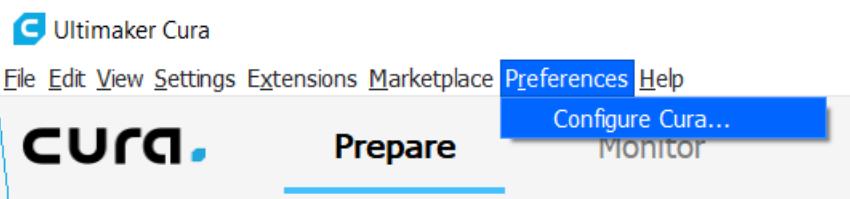
Layer View (Slider at layer 120)



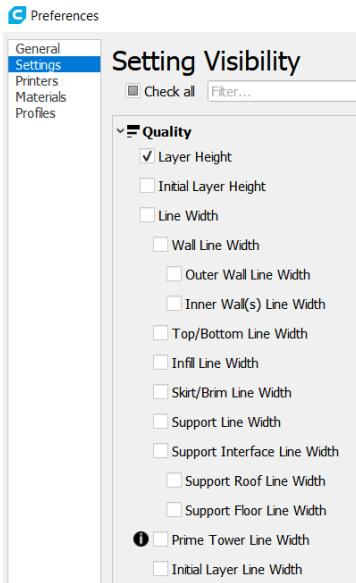
Making Options Visible in CURA

There are hundreds of options you can set in Cura to control all the possible features of your printer. However, most of them are not visible to change by default. To see them, you can do one of two things.

- 1) Select ‘Preferences’ from the main menu and then ‘Configure Cura’.



Click on ‘Settings’ to bring up the entire list of possible options. *Don’t panic!* Most of these you will never use or care about. But a few are important.



Note: Clicking a check box here only makes the option visible in the main Cura window. It does not turn that option on or set a value. You must do that from the main Cura window.

- 2) The second way to see and change them is to click on the settings icon (gear) in the main Cura window to the right of each major option section as shown by the red squares below.

Profile: My Draft - 0.2mm

Search...

Quality

Layer Height: 0.2 mm

Shell

Wall Thickness: 0.8 mm

Wall Line Count: 2

Top/Bottom Thickness: 0.8 mm

Top Thickness: 0.8 mm

Top Layers: 4

Bottom Thickness: 0.8 mm

Bottom Layers: 3

Z Seam Alignment: Sharpest Corner

Seam Corner Preference: Hide Seam

Enable Ironing:

Infill

Infill Density: 10 %

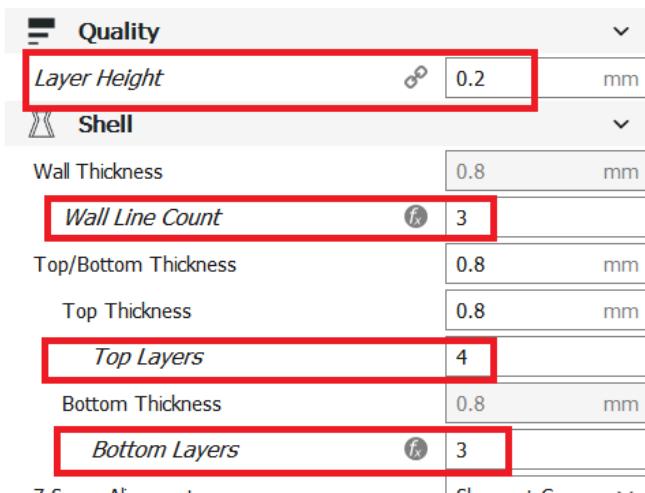
Clicking the setting icon brings up the detailed settings for that section. You can now turn on whatever options you wish to modify or control. You will see in the sections below some of the ones I make visible for the more common settings.

Be sure to save your profile when making these changes or you will lose them the next time you start Cura.

Quality - If you hover your mouse over the setting name it will give you a little description of what it does. Sometimes they are helpful, other times not. The most used layer height is 0.2mm. Most of my prints have been done at that setting. For producing finer details, you can reduce the layer height down to 0.16, 0.12, 0.1, even 0.05 (although I have not tried that last one yet!). However, reducing the layer height increases the print time because you are printing more layers for the piece. For example, 0.1 vs 0.2 will double the print time so only use that when printing parts that need lots of detail. When you get more familiar with the printer, you can also change the nozzle from 0.4 to other sizes for different results.

Wall line count tells how many layers thick the wall will be. The more layers, the stronger the wall and the part. Also, the more layers, the smoother the outer surface will be. 3-4 is a good number to start.

Top and bottom layers do just what they say, determine how many fully solid layers will be printed before it starts printing what's called the Infill. The more top and bottom, the smoother and stronger the top and bottom of your parts will be.



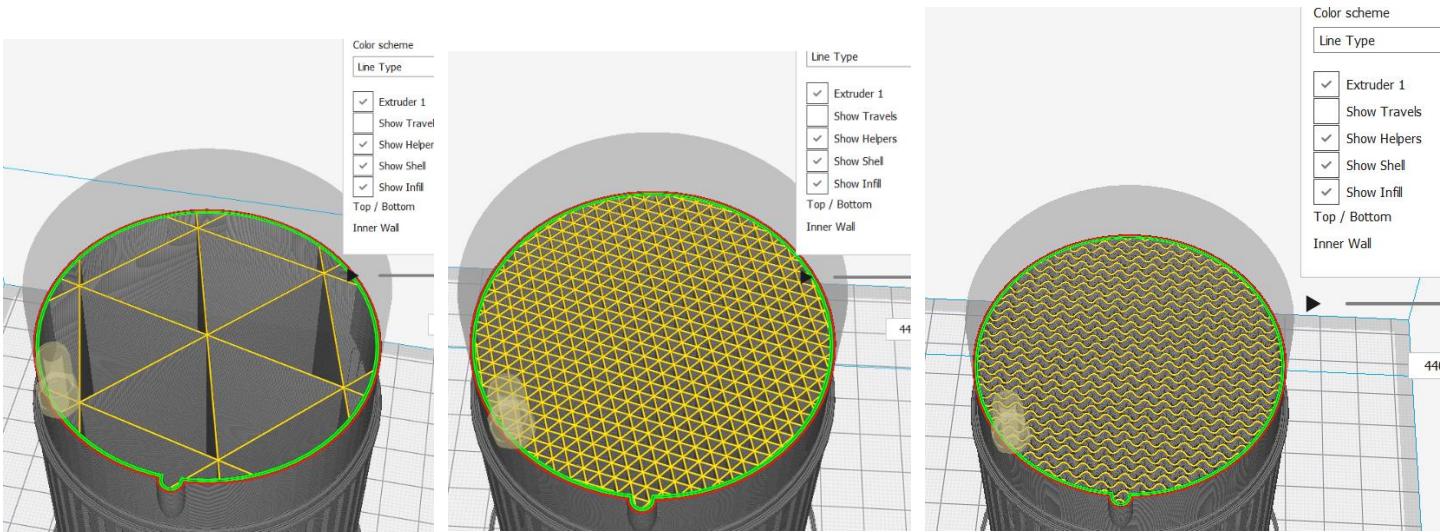
Infill - Infill is the material printed inside your structure. You could print a solid layer on every layer but that would take forever and doesn't really improve the strength. The % of infill can be selected to change the amount of material extruded on in the inside of your model. However, large models can get away with just 10-25% infill. If you are printing small parts and want them as solid as possible, you can choose 100%. Some people also say 0% is valid for some models. You can also select the infill pattern. A lot of people recommend Triangle for strength. A new one just came out with Cura 3.6 called Gyroid which I am playing with now.



Triangle infill at 5%:

Triangle infill at 50%

Gyroid infill at 25%



Note that the print time goes from 4hr 29 minutes to 13hr 51 minutes by changing the infill from 5% to 25!! The gyroid infill at 25% shows a build time of 9hrs 13 minutes. Choose your infill carefully as it does have a big effect on print time.

Also note that I have found the estimated print times shown by Cura and not always very accurate. Usually, it takes longer than what they calculate it would take.

Print Temperatures - Most standard PLA's will print well at between 190 and 210 degrees. Always refer to the information on the PLA you purchased as to the recommended print temperatures. It is believed best to start off a little hot for the first layer to help it stick better to the plate. Then you can ramp it down for subsequent layers using the settings below. I run my build plate at 45 Celsius. PLA doesn't really require much heat on the bed itself. The flow rate field below is the percentage of flow of the filament you are sending to the nozzle. If your printer is calibrated properly, 100% is the correct answer. There are good videos on how to calibrate your flow rate.

Material	
Printing Temperature	195
Printing Temperature Initial Layer	200
Build Plate Temperature	45
Flow	100

Retraction – When the print head moves over an area where it is not printing, the extruder retracts a few mm of filament to prevent it from stringing along between the two areas. If you get a lot of stringing on your print, you can tweak these settings to help prevent it.

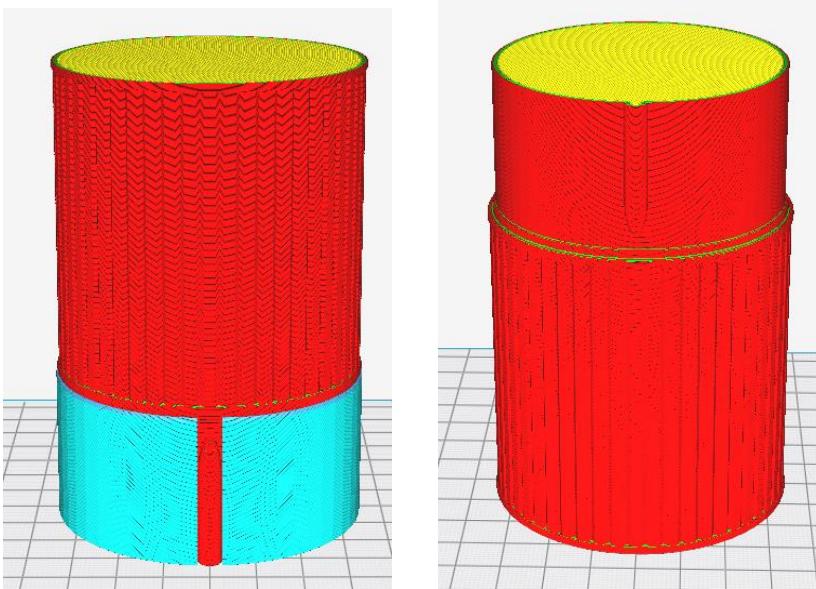
Enable Retraction	<input checked="" type="checkbox"/>
Retract at Layer Change	<input type="checkbox"/>
Retraction Distance	5
Retraction Speed	25

Print Speed - This is the standard print speed, 60mm/s. However, if you are printing something with lots of details, you can improve the quality by slowing the print speed down to 40 or 50. Don't forget it will affect the print time.

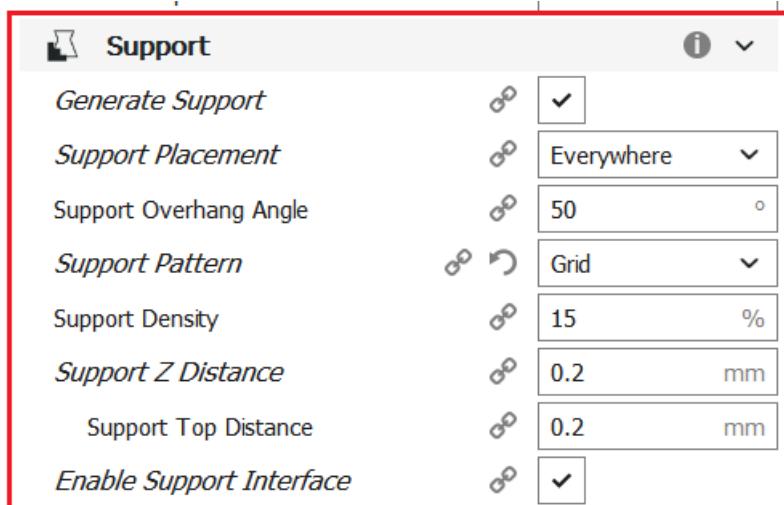
Speed	
Print Speed	60

Support - Support is one of the most important settings to play around with. These printers can't print in thin air so any part of a design that does not touch the print bed or another part of the model needs support. You can set it to only provide support to anything touching the build plate or to any part of the model that requires it. The overhang angle tells the software how sharp an angle can be printed before support is required. You can also select the support pattern although that isn't as critical. Support material shows up as 'Helpers' in Cura, usually with a teal color as shown below.

You should try to avoid using supports where possible. Many times, by changing the orientation of the model on the build plate, you can eliminate the need for support. Here is the model of a cylindrical piece with varying diameters. If I print it with the thinner end down, support is required to hold up the top part where it is hanging in midair. The teal section below is the support being added. If I flip it 180 degrees, I now don't need support as everything is supported by the model itself.



The support is designed to not totally connect to the model, so it can be peeled away after printing. This doesn't always come out as intended and there are several parameters in the support section of Cura that can help fix that. Try the different options and then use the layer view to see how things have changed.



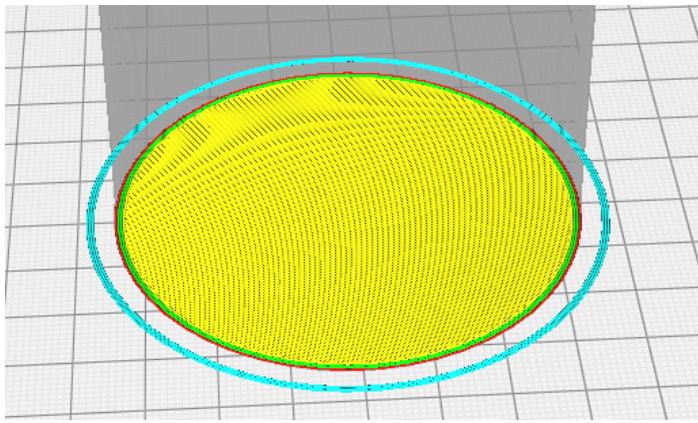
Build Plate Adhesion - There are several options here you can choose. Selecting 'None' means only the model will be printed. I typically use the next option, Skirt, which causes the printer to print a couple of lines around the space where your object will be printing. I use this to help make sure the bed is leveled and the filament is coming out properly before the actual print starts. I usually set the skirt line count to three lines, but you can set it to whatever number you want. The next setting is Brim. This will print a couple of lines around the model, touching it around the perimeter. This

is useful if you are printing a tall thin object as it allows you to put more filament on the build plate, so the model doesn't come off. However, since the brim touches the model, you will have to peel it off when the print is done and clean up any edges. Lastly there is Raft. This puts an entire structure under the model and then prints the model on top of it. It can be useful when for whatever reason getting the model to stick is problematic. However, it adds a lot of time to the print and if your adhesion is good, there is no need to use it.

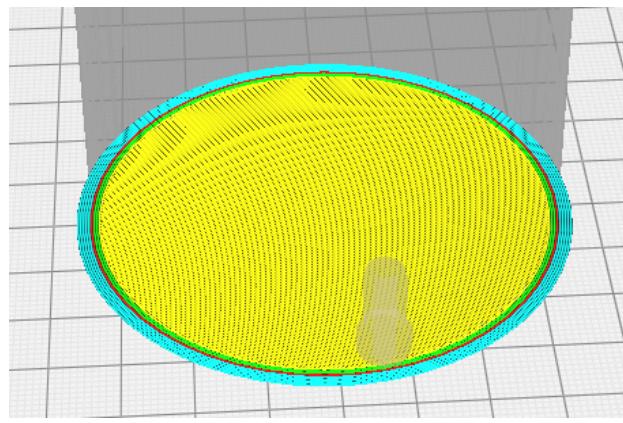


Here are what these look like.

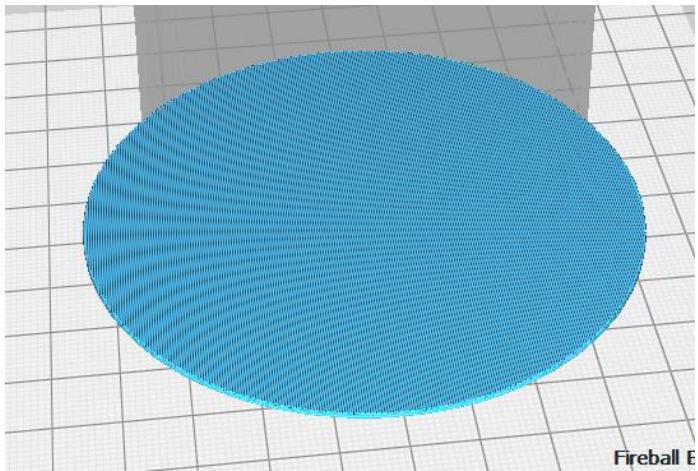
Skirt of three lines at layer 1:



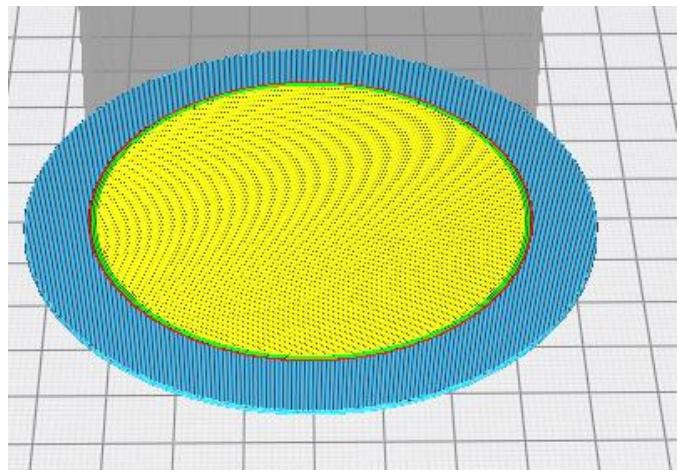
Brim of 5 lines at layer 1:



Raft of 10 lines at layer 1:



Raft at layer 5 where the model starts printing:



Bed Leveling and Nozzle Height

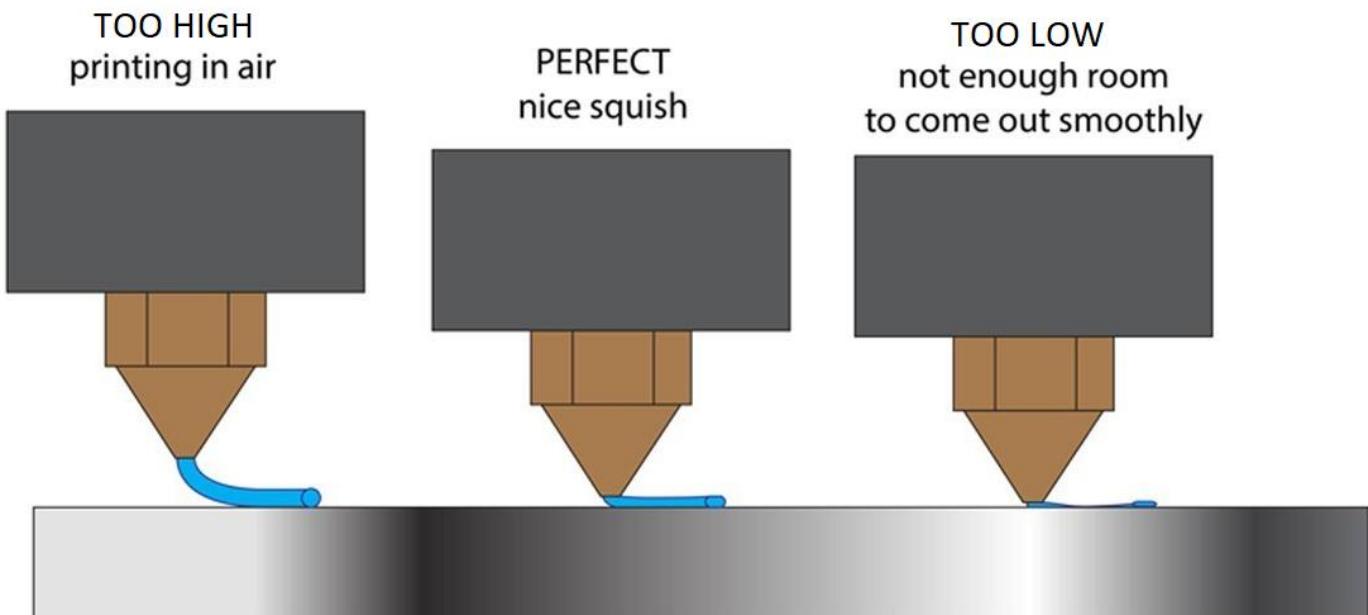
The two most important things you will need to do to get printing correctly are ensuring a level/flat bed and getting the correct height between the nozzle and the bed. The bed must be flat to a very exact degree for the first layer of filament to be properly laid down and adhere to the bed. If the print comes loose during printing, the whole print will fail.

The Ender 3 comes with a removable bed designed to help your prints stick to the bed. I do not have any experience with that, so you will be on your own. You may want to check the Facebook groups or YouTube for any advice on using that. The CR-10 Mini comes with a removable glass bed held on by these stationary clips.



(I prefer the use of a mirrored piece of glass cut to the size of the print bed and held on by stationary clips with a glue stick used for model adhesion, but this is up to you. Mirrored glass is preferred over plain glass as supposedly mirrored glass is made to more exacting surface level tolerances to ensure the mirror doesn't look like one in a funhouse! Mirrored glass can be found cheaply and easily at most big-box hardware stores. Look for mirror tiles for mounting on the wall. While there, pick up a cheap glass cutter; measure the bed, scribe the mirror tile with the cutter and snap the glass clean. You can use a sanding block to clean up any sharp edges. I then use a thin layer of "glue stick" glue on the glass where the print will lay. Having a perfectly level surface is the first step to successful prints.)

As for setting the height between the nozzle and the bed, I won't go into detail here as there are lots of videos on leveling a bed. Please look at them and follow their instructions. Here is a graphic which shows the results of different nozzle/bed heights and their impact on the first layer:



the GAP between your nozzle and build surface is too far! your filament is coming out in the air and not being squished into the bed, it will not get good adhesion, and probably will get dragged around corners and not lay down precisely.

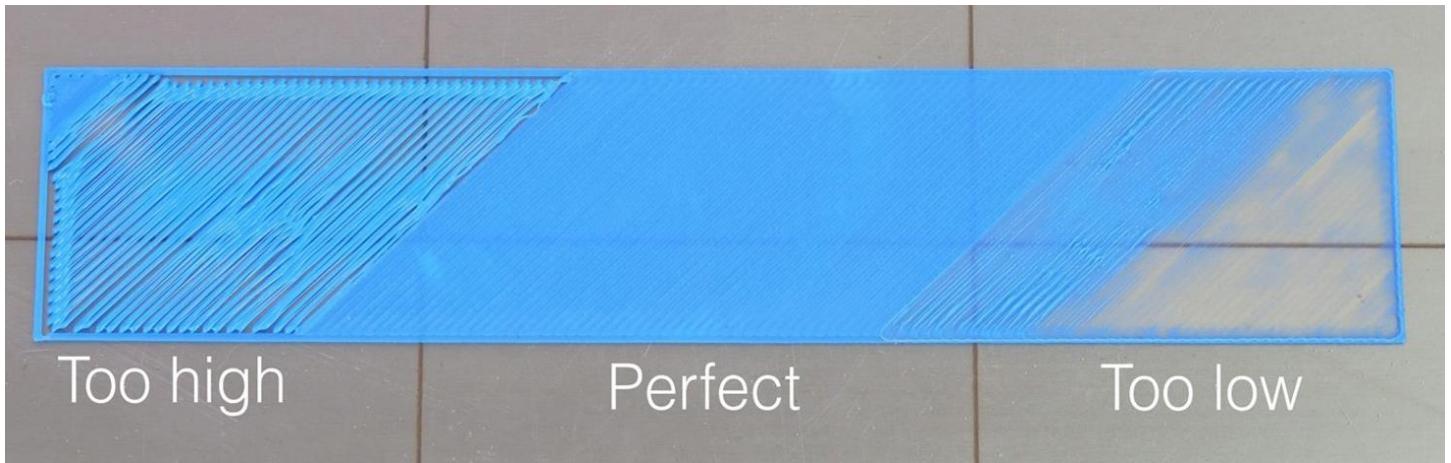
the GAP between your nozzle and build surface is just right! your filament is being compressed between the nozzle tip and build surface as it is coming out. Your first layer will lay down precisely, complete fills nicely, and stick. This will make or break all future layers!

the GAP between your nozzle and build surface is too close! your filament is being compressed too much between the nozzle tip and build surface as it is coming out. This will create too thin of a first layer, inconsistent first layers, may prevent the filament from coming out, and may even block up the nozzle!

If your nozzle is too high from the bed, the first layer will print like a round bead and be easy to dislodge with your hand. If the nozzle is too close to the bed, you may damage the bed surface, the filament may not come out at all, or it may come out sporadically and very thinly. Again, not good.

When the nozzle is just right, you should get a flattened oval type of appearance and the filament should be firmly attached to the print bed. I use a “Skirt” as described above to check my level when starting each print. If the skirt isn’t perfectly laid down, I cancel the print at that point, adjust the bed and start the print over again (after having removed the failed skirt from the bed!). You can also adjust the bed while the skirt is printing. This can be useful in dialing in the correct settings. Increase the number of lines in your skirt if you need more time to level the bed before the actual print starts. The same process works if you are using a Brim or Raft. Get those right before wasting filament on the actual model and you will have many more successful prints.

Here is another picture of the first layer. The Too high and Too low refer to the print head and how close it is to the print bed.



To be continued...

Appendix A – Raspberry OctoPi Print Server and Camera

I strongly recommend you obtain and configure a Raspberry Pi microcomputer to increase your enjoyment of using the printer. The Raspberry Pi will connect via Wi-Fi to your PC and allow you to load files, control, start, stop prints from your PC without the need for the SD card swapping. It also supports video cameras for monitoring printing while away and for taking cool time-lapse photos of your printing.

Here is the hardware you will need to obtain. All of it is available on Amazon as of this writing. You want to buy at least Raspberry Pi version 3 B+.

- CanaKit Raspberry Pi 3 B+ (or the latest version) Starter Kit (32 GB EVO+ Edition, Premium Black Case) - \$79.99
- Raspberry Pi Camera Module V2-8 Megapixel,1080p - \$24.95
- Adafruit Flex Cable for Raspberry Pi Camera - 2 meters [ADA2144] - \$8.99

You will need a USB cable to connect the Raspberry to your printer. The Raspberry has the standard big USB connector. Check your printer to see what size USB connector it has and get a cable if you don’t already have one. It’s usually a common USB connector.

Once you get the Raspberry, go to the Octoprint web site <https://octoprint.org/> go to the download page and follow their instruction on installing their version of the OS, “OctoPi”. It’s straight forward as long as you don’t make the mistake I did (don’t say yes to any Microsoft prompt to format the SD card!).

For the camera you can print a case and mount from Thingiverse as I did. The cable is to connect the camera to the Raspberry.