TRAILS 2.0 Slicing Guide

A video of these steps can be found here: https://www.youtube.com/watch?v=9tGCd-3UJZU

I) What is slicing and why do I need it?

Technology is great—when it works and you understand it! Unfortunately, these two things do not always happen together. 3D printing is a great example; often something does not work and you just do not know why.

Actual footage from 2023 TRAILS 2.0

Just keep doing your thing, Mr. Printer…

I do not share this to scare you off, but to let you know that everyone struggles with these kind of things. There are certainly many factors that make a technology viable, but one important issue when faced with failure is what you do next. Do you give up? Ask for help? Do something else for awhile? Sell the farm? Go all in on your next attempt?

Getting any part from concept to physical component is no easy feat. Despite the multiple steps, there are two important ones that need to happen in the software that you actually have quite a bit of control over. These are Computer Aided Design (CAD) and slicing.

CAD is the process of creating a digital 3D model. This guide does not go over how to do this well, but I will point out a few things. First, your CAD model must be a part that can physically exist. I can create something in CAD that is totally impossible to make in real life. Think about something you have seen in the movies—perhaps a superhero or fantasy franchise. Some of these things look cool on-screen but are impossible to physically produce (I am thinking of you, M. C. Escher!). Second, your CAD model often must be converted to 'dumb' geometry. This means that it does not contain equations, associativity, or proprietary data. After creating the CAD model, you would typically export it to a 'dumb' format such as ".stl", ".stp", ".obj", or ".3mf" (note that there are others). This then allows the slicer to do its work and prepare the model for the printer. There are exceptions to these two guides, but I will let this stand for now.

You may be wondering, "Why can't I just send the CAD model to the printer?" That is an excellent question, and in some rare cases you can (hence why I like Fusion 360 so much). Most of the time, however, it is like this: imagine you want a fancy cake, but you do not have the time, desire, or skill to make it yourself. So, you contact the town's best baker. You purchase all the ingredients (locally, of course!) and take them to the baker. Then you dump the pile of ingredients on his counter and walk away without providing any other information because you are in a hurry and the baker does not speak the same language as you. What is the likelihood that you will get exactly the cake you imagined? It is zero. In fact, you may not even get a cake! All of the ingredients may be present, but there are no instructions.

In the example above, you are the designer, the ingredients are the CAD model, and the baker is the 3D printer. All 3D printers need instructions, and this is where the slicer comes in. It translates the CAD data into something the 3D printer can understand. This language is called g-code.

<u>TL;DR: You need a slicer to translate CAD data into instructions that the 3D printer is able to understand.</u>

II) What do I need?

In order to do this successfully, you need a few physical items, some software, and a heaping helping of patience.

- i. Hardware:
  1. Computer
  2. 3-button mouse (wired or wireless)
  3. 3D printer
  4. Filament
  5. USB drive or SD card and reader (depending on your printer and computer)
  6. Electricity
- ii. Software
  1. CAD program (Fusion 360, Onshape, NX, etc.)
  2. Slicer (Cura, etc.)
  3. Text editor (not required but suggested if you want to dig deeper)
- iii. Patience
  1. Pray for it!

The computer should be capable of running both the CAD program and the slicer. You should be able to find information regarding specifics on the programs' websites. A 3-button mouse is a requirement for doing CAD or slicing work at any reasonable speed. Your 3D printer and filament should be compatible (more on this in a different guide). The printers we use in TRAILS 2.0 are able to print files that are stored on SD cards, but make sure of the requirements of your printer before making a purchase. I have used some that have to be connected to the computer the entire time (inconvenient!) and some that only work with USB drives (inconceivable!).

The first step is to export your file from the CAD package. This is typically done with a File→Save as or File→Export command. In either case, you should save the file in the STL format. Do not worry too much about the specific details of the translation; often the defaults will suffice. I strongly suggest, however, that you practice good file management by naming your files with logical names and using version numbers. For example, saving a file with a student's last name, the part, and the version would look something like this:

Lee_BigHook-v1.stl

I would advocate that you have students name their parts this way before sending them to you. Naming things this way will help prevent files from getting overwritten and will also help you know what you are printing when navigating the tiny screens typically found on desktop 3D printers. Few things are more frustrating in 3D printing than having a bunch of files named such as "Part.stl" or "Export.stl" and having to figure out which one is to be printed next.

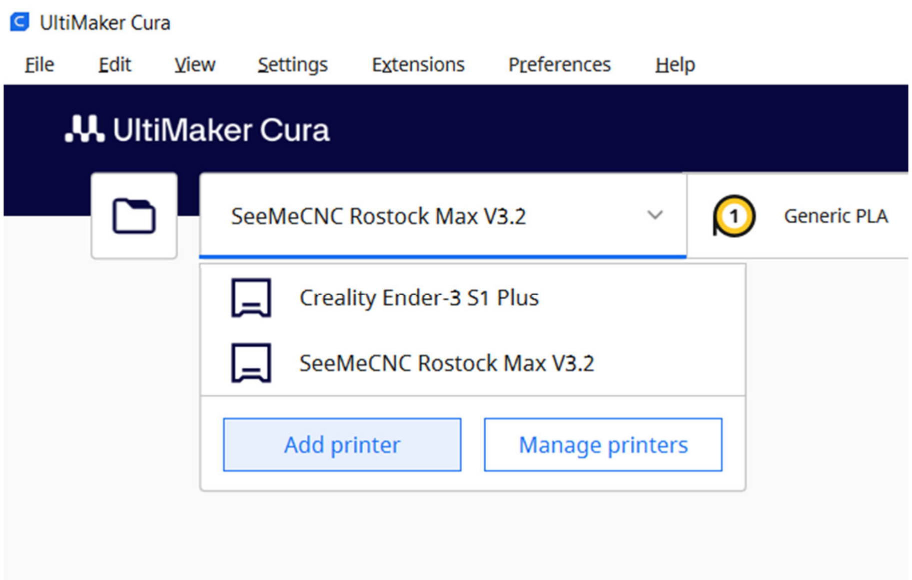
I suggest having students email their files to you, but they could also save them on an external drive (such as the aforementioned USB drive) and physically give it to you.

TL;DR: Have a computer, slicer software, and patience. Have students export their CAD files as STLs with descriptive, unique names (and version numbers!).

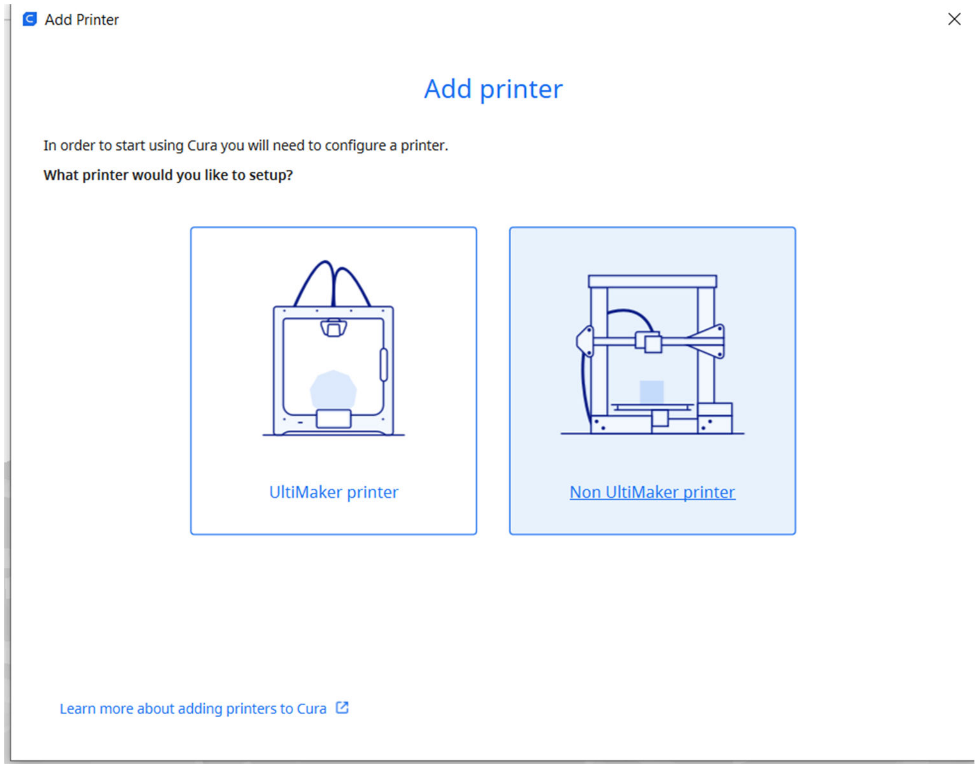
III) Slicing example

A. Now that you have the file(s), you need to open them in the slicer. As with CAD packages, there are too many possibilities to cover every one. This guide is based on Cura. In addition to being free, Cura is the basis for many companies' specific slicers. Regardless of which slicer you use, these steps should be helpful. Cura can be downloaded at: https://ultimaker.com/software/ultimaker-cura/

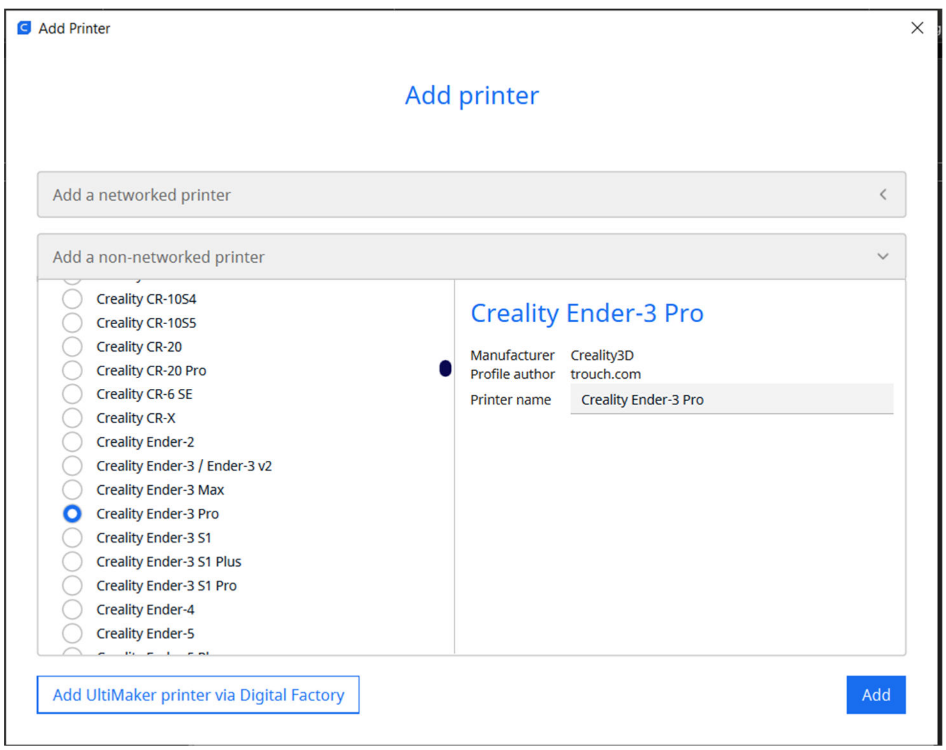
The first step is to ensure that the printer settings are correct. Fortunately, Cura comes with profile settings for the most popular printers and materials. We will be using the Creality Ender3 Pro and a 0.4 mm nozzle with PLA. There are, of course, many more options.



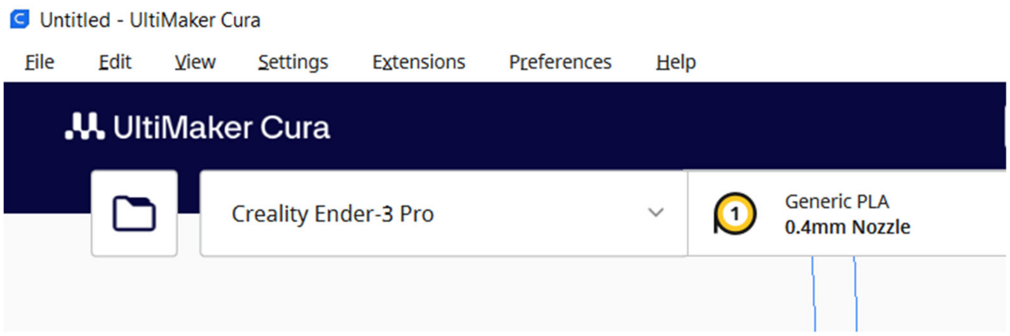
Click the printer drop-down and select "Add printer".



Click "Non UltiMaker printer".

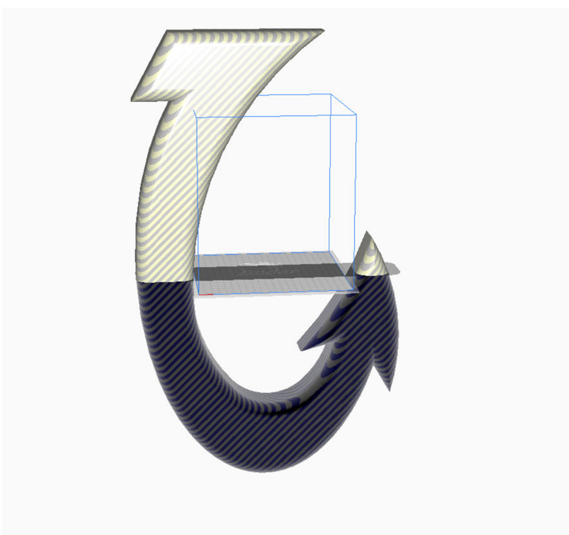


Expand the non-networked printer and the Creality3D sections and select the Ender-3 Pro. Then click "Add".

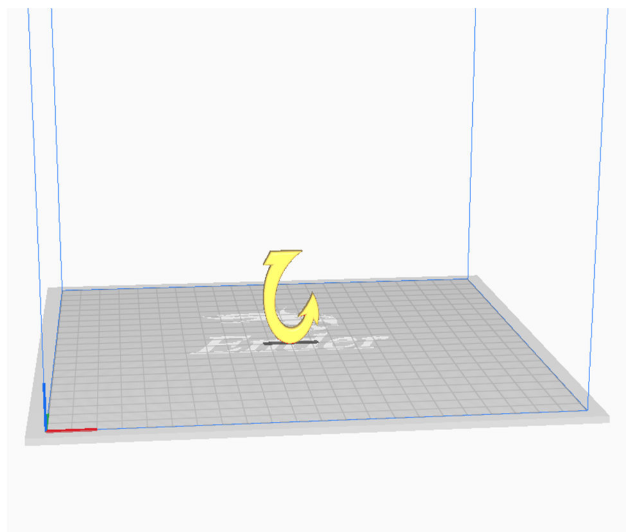


Notice that the printer, filament, and nozzle are correct.

B. The second step is to open the STL file and verify its printability and scale. Scale is one of the most common issues. As most slicers default to the unit of millimeters, I suggest having students design in mm. All slicers I have used are able to scale models, but in order to get what you want you have to know what unit the model was designed with. Often multiplying or dividing by 25.4 will yield the correct result, but I always check. Notice the two images below; the hook on the right will physically fit; the one on the left will not (notice the checked pattern).

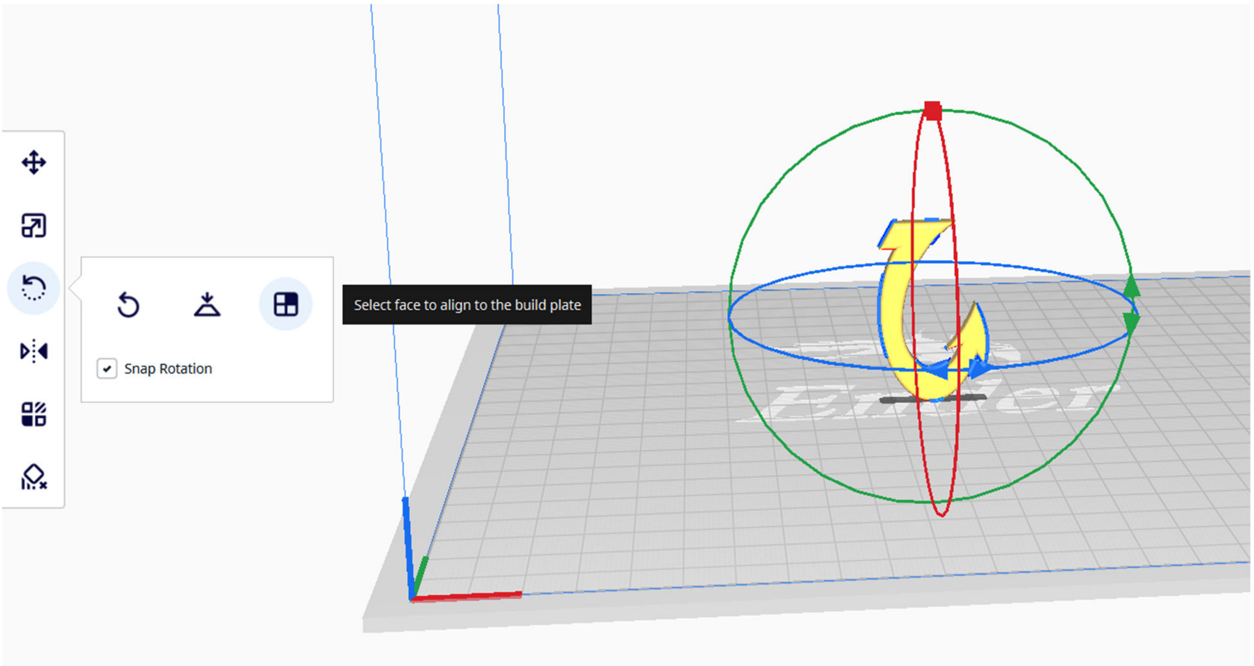


Too big!



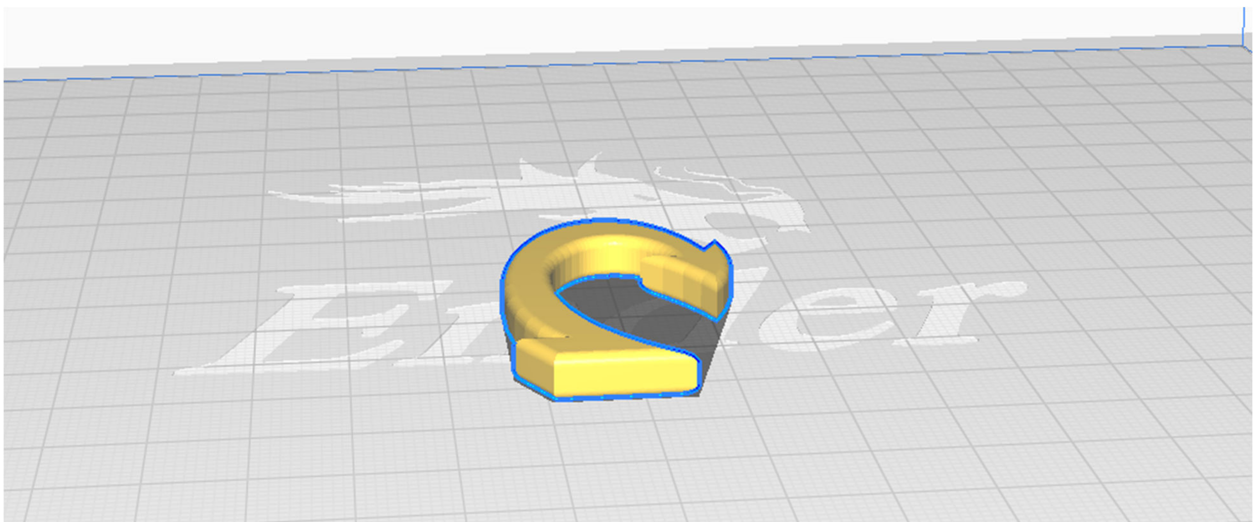
Just right!

C. The third thing to check is the orientation of the part. Build times are approximately proportional to the height of a part. For this and other reasons the height should be minimized. As this hook is thin in one dimension it is a simple task to orient it; other parts can be more challenging and may require thoughtful positioning to achieve efficient and successful prints. Click on the hook, click on the Rotate tool, and select "Select face to align to the build plate". This is a long-winded way of saying, "lay it down".



Too tall!

Now click on one of the long faces of the hook (front or back).

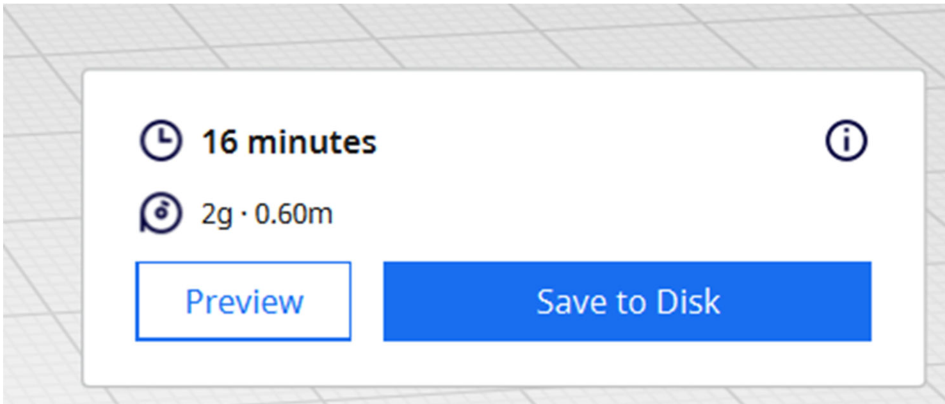


Now this looks good.

D. Next you need to create the instructions for the printer. Remember the baking and recipe example? We are now creating the instructions that tell the printer where to move, how to move there, and how much filament to extrude. This step is simple—just click “Slice”.

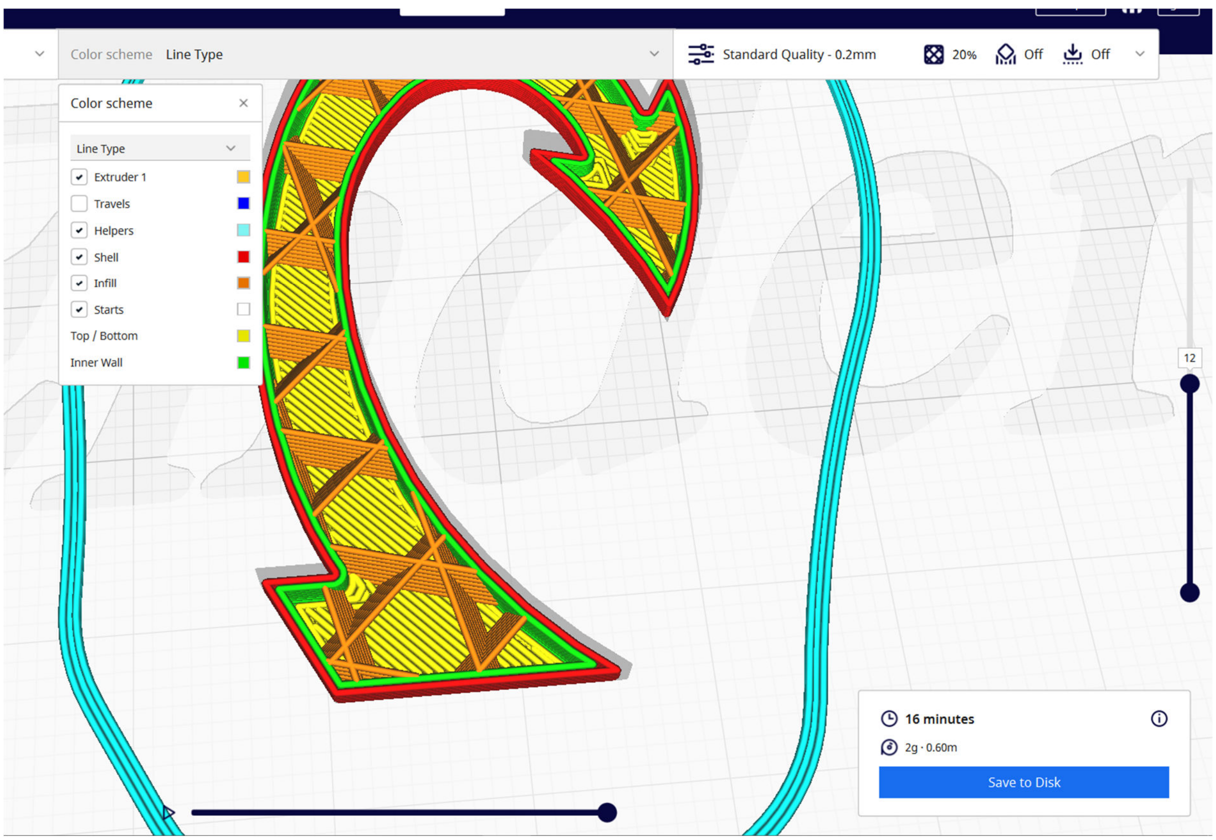
Slicing button

Once that is done, you have a few options. You could go ahead and save it to a disk (remember the SD card?) or you could preview. I suggest taking a look at the preview now.



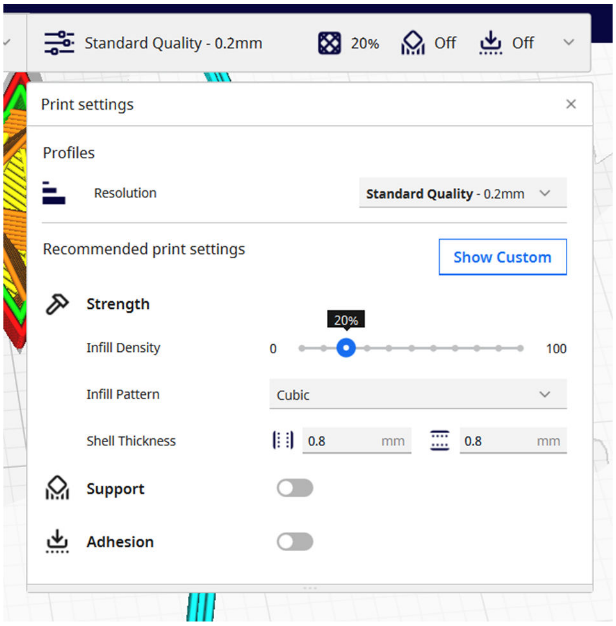
Time and material information and file options

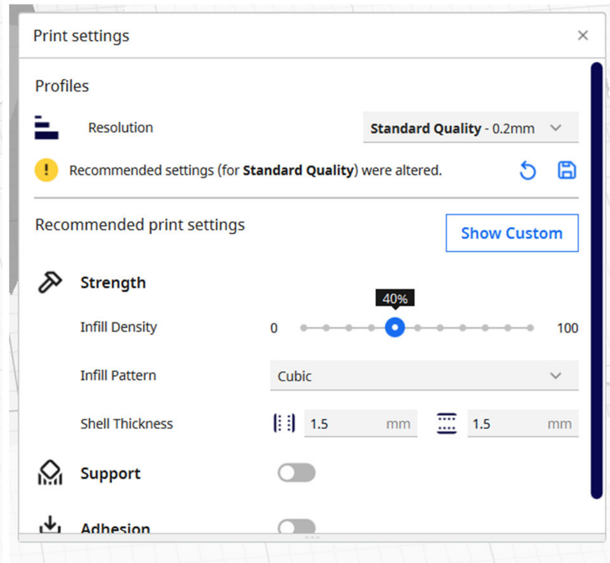
E. The Preview function (also a tab at the top center of Cura) allows you to see how the printer will move. It also allows you to see things like the infill pattern and wall thickness. Dragging the slider at the bottom moves the printer around on the current layer and dragging the slider at the right moves the printer between layers (virtually in both cases). If you drag the slider down about halfway (layer 12 in my case) and zoom in, you can see what the printer will do.



Preview—the part looks a bit thin

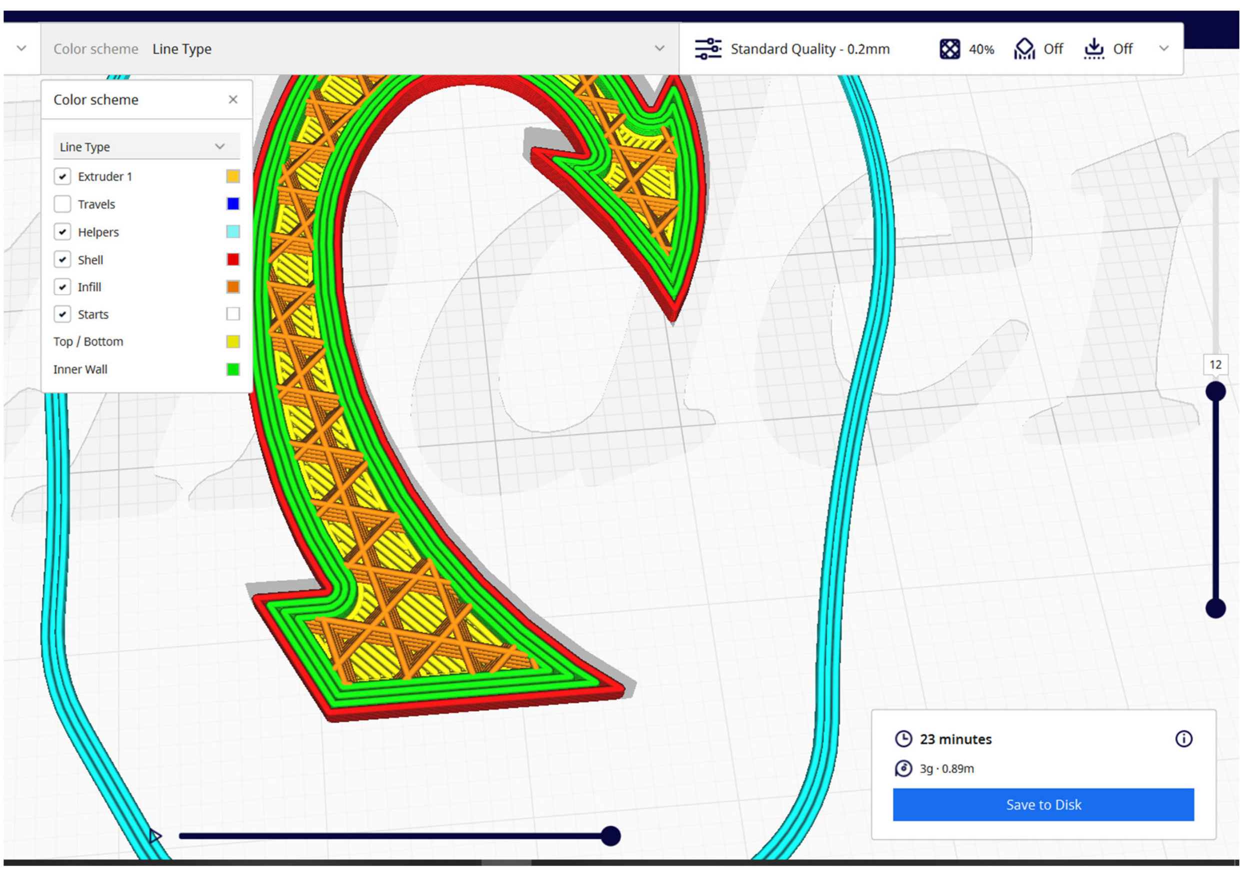
Notice the color coding on the extrusion path. These will not be printed in different colors, but it does give you an idea of what is going on. I think this looks like it will be too weak, so I am going to increase the infill density and the shell thickness.





Original settings *Adjusted infill and shell*

F. You will need to slice again. This looks like it will be stronger. Also notice that the time has increased. Be careful here—if this is the first time you have printed something, consider printing it with less infill or printing just a few layers of it to save time and material.



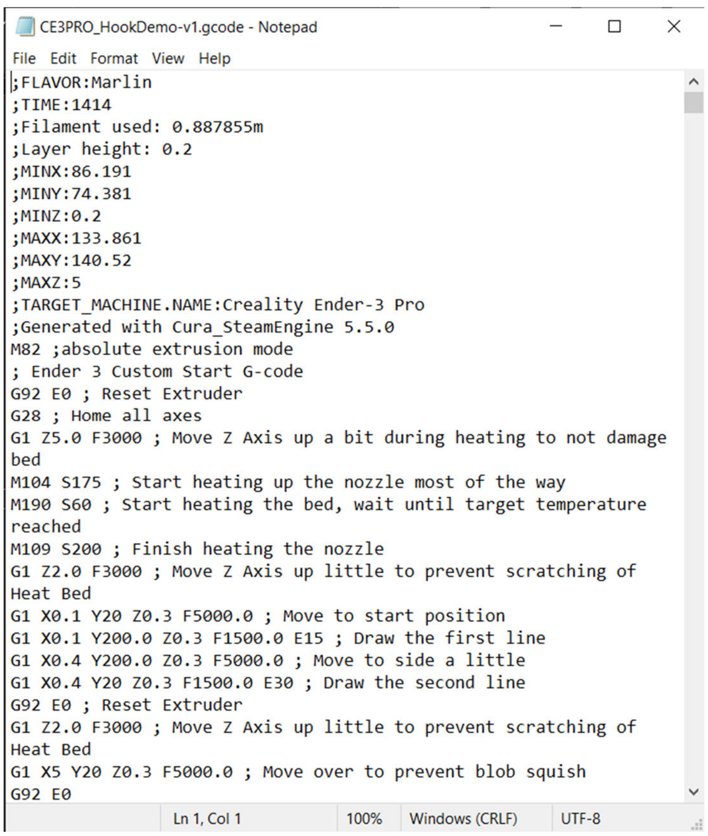
A more robust part.

G. Once you are finished, save the file to the location of your choice. Remember to use version numbers!

File name with version number

H. Now eject the disk (from the Cura menu), remove the SD card, insert it into the 3d printer, select your file, and print!

I. If you open the g-code file, you will see the instructions that are sent to the machine to tell it where and how to move. There are many resources on how to edit g-code. Feel free to do some digging and learning but be careful: it is possible to break your printer by giving it bad instructions (telling it to do things that it should not). Remember that robots are obedient to a fault.


CE3PRO_HookDemo-v1.gcode - Notepad

File Edit Format View Help

```
;FLAVOR:Marlin
;TIME:1414
;Filament used: 0.887855m
;Layer height: 0.2
;MINX:86.191
;MINY:74.381
;MINZ:0.2
;MAXX:133.861
;MAXY:140.52
;MAXZ:5
;TARGET_MACHINE.NAME:Creality Ender-3 Pro
;Generated with Cura_SteamEngine 5.5.0
M82 ;absolute extrusion mode
; Ender 3 Custom Start G-code
G92 E0 ; Reset Extruder
G28 ; Home all axes
G1 Z5.0 F3000 ; Move Z Axis up a bit during heating to not damage
bed
M104 S175 ; Start heating up the nozzle most of the way
M190 S60 ; Start heating the bed, wait until target temperature
reached
M109 S200 ; Finish heating the nozzle
G1 Z2.0 F3000 ; Move Z Axis up little to prevent scratching of
Heat Bed
G1 X0.1 Y20 Z0.3 F5000.0 ; Move to start position
G1 X0.1 Y200.0 Z0.3 F1500.0 E15 ; Draw the first line
G1 X0.4 Y200.0 Z0.3 F5000.0 ; Move to side a little
G1 X0.4 Y20 Z0.3 F1500.0 E30 ; Draw the second line
G92 E0 ; Reset Extruder
G1 Z2.0 F3000 ; Move Z Axis up little to prevent scratching of
Heat Bed
G1 X5 Y20 Z0.3 F5000.0 ; Move over to prevent blob squish
G92 E0
```

Ln 1, Col 1 | 100% | Windows (CRLF) | UTF-8

G-code file

<u>TL; DR: Seriously? This third part should really be read!</u>