3D PRINTING WORKSHOP

SLQ Wiki Fabrication Lab 2026/02/25 15:39

3D PRINTING WORKSHOP

In this space you will find the instructions to install Cura, the software used in the 3D printing workshop currently touring the Isaac Region.

For more information on 3D printing on the SLQ Wiki click here.

Downloading & Installing CURA 3.6.0

You can print the

Cura Installation Instructions

as PDF here.

1. Visit Github to download Cura.
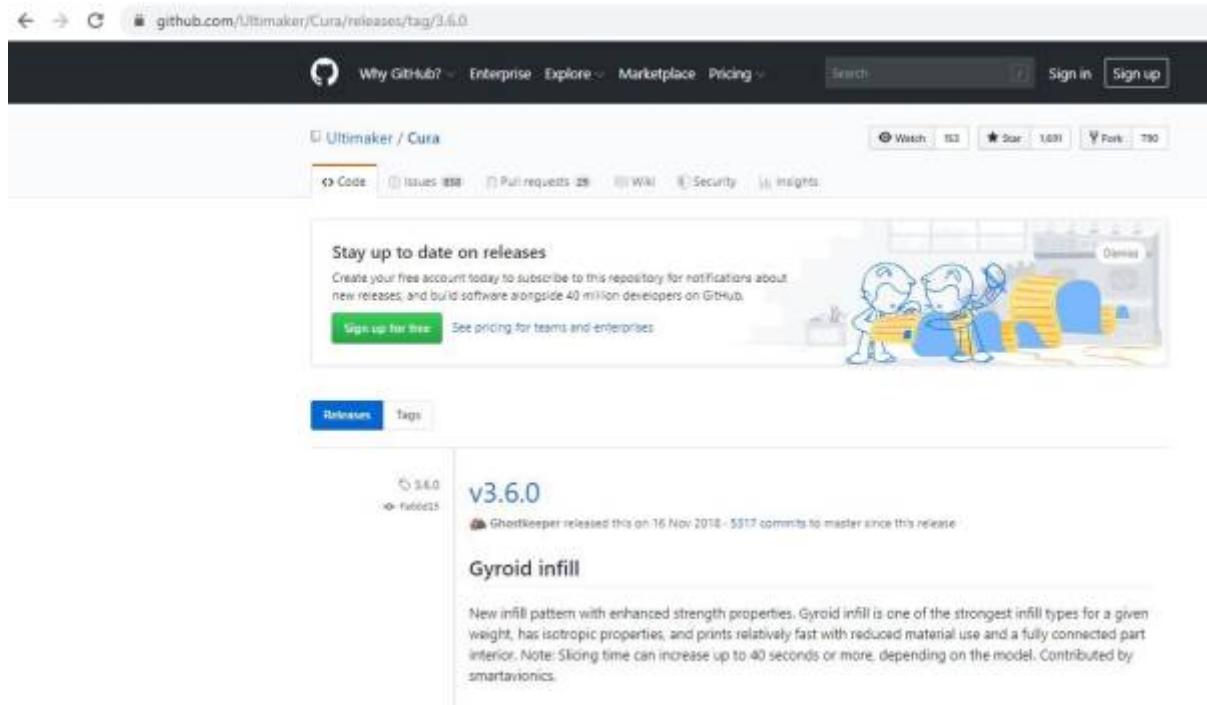
2. Scroll down to the bottom of the page until you find the link to download the application (highlighted file in below picture).
3. Download the highlighted file & install onto your computer.

- Add a new post processing script that pauses the print at a certain height that works with RepRap printers. Contributed by Kriechi.
- Fix updates to the print monitor temperatures while preheating. Contributed by fieldOfView.
- Fixed a bug where material cost is not shown unless weight is changed.
- Fixed bugs crashing the CuraEngine when TreeSupport is enabled.
- Fixed a bug where Ultimaker Cura would upload the wrong firmware after switching printers in the UI.
- Fixed a bug where the layer view was missing if the first layer was empty.
- Fixed a bug where erroneous combing movements were taking place.
- Fixed a bug where the initial layer temperature is set correctly for the first object but then never again.
- Fixed a bug where clicking the fx icon didn't respond.

Install will be as per a standard program installation for Windows, until the steps below:

1. During the install, a message box will pop up & ask you to "choose Components". Select the options shown in the picture below & click "Install".
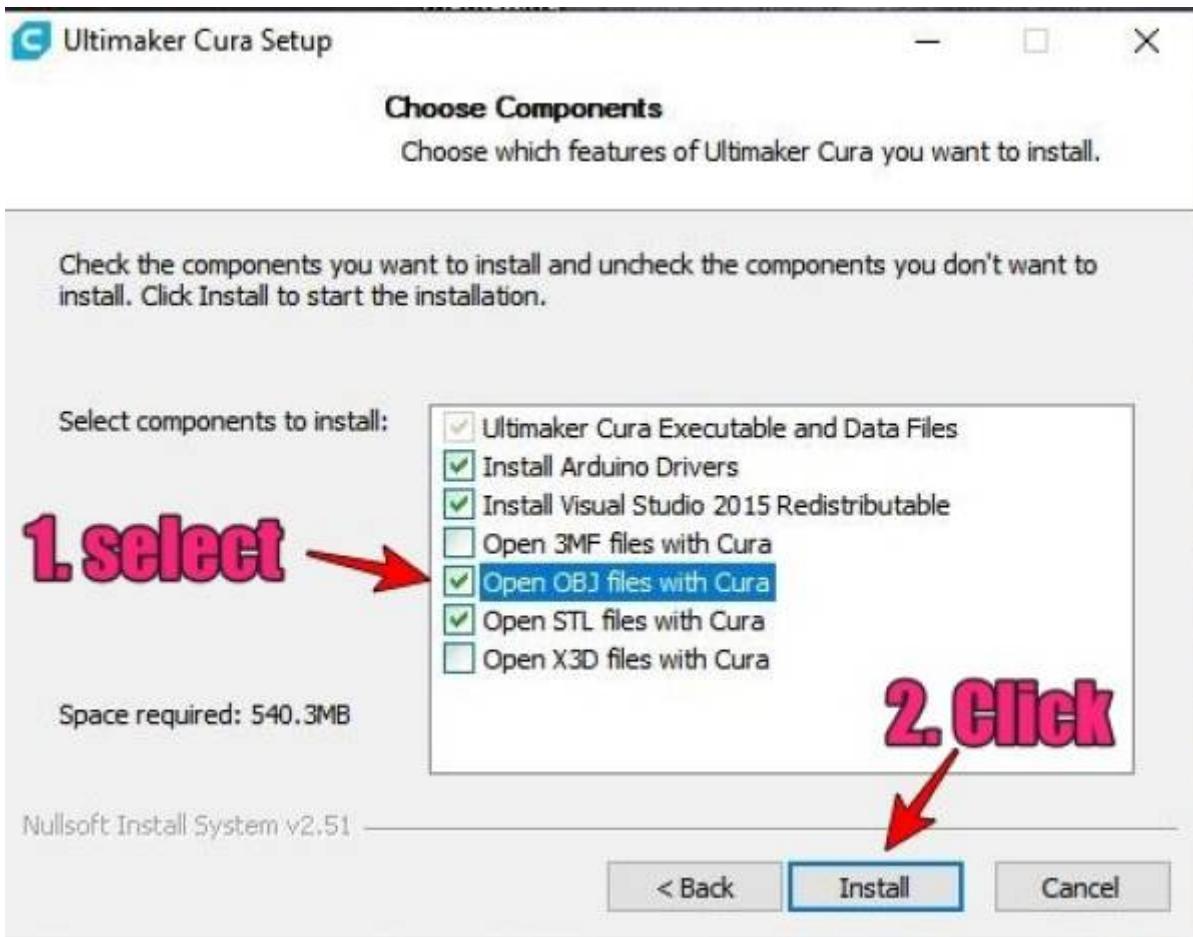
2. Once install is completed, Click "Finish" & CURA should open (if not, open it yourself).
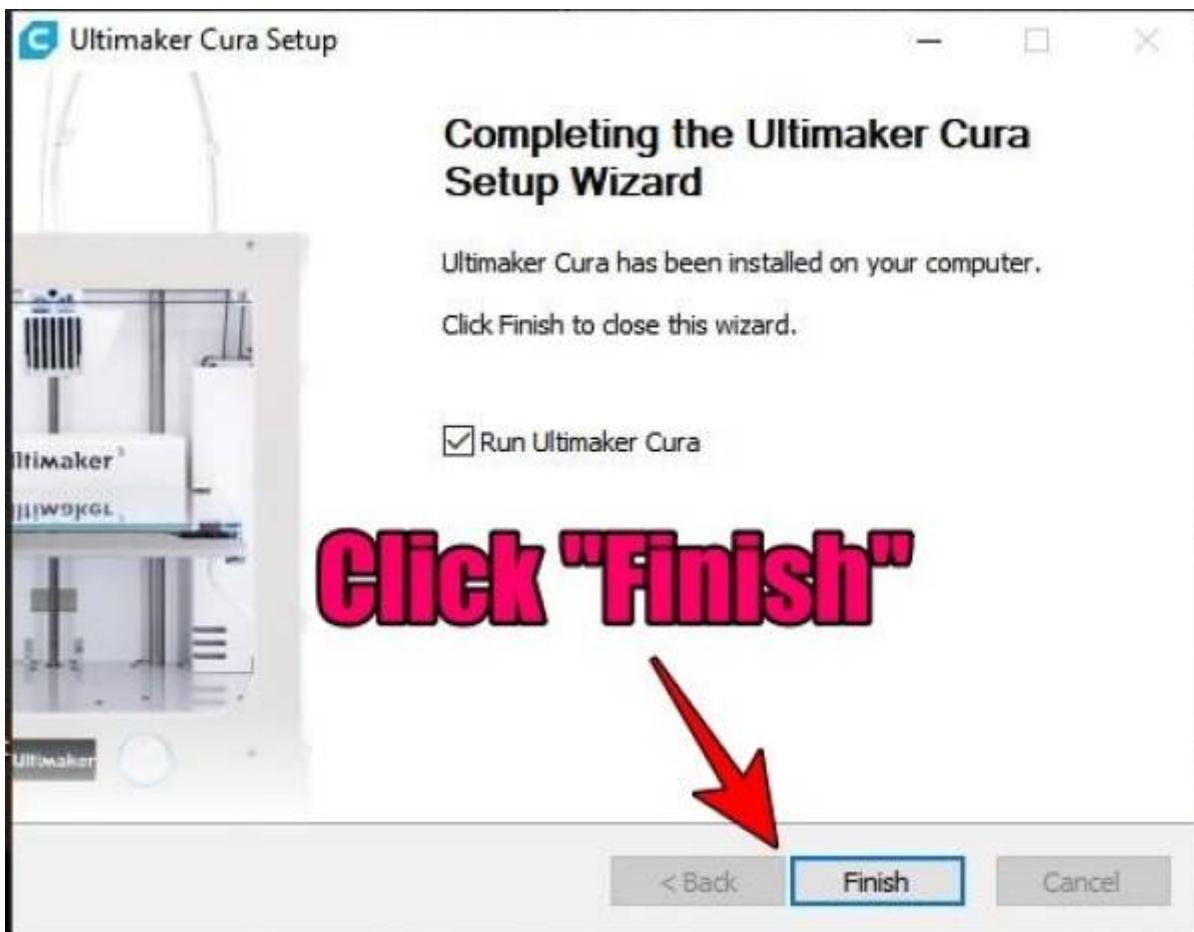
3. When CURA opens, a window will ask you to read & agree to the terms of use. Read the terms in the pop-up window & *click only if you agree*.

4. If you agree to the terms, another window will open & ask you to select your printer. We'll be selecting a printer from "**other**". Click "**other**" & a drop down selection list will appear.
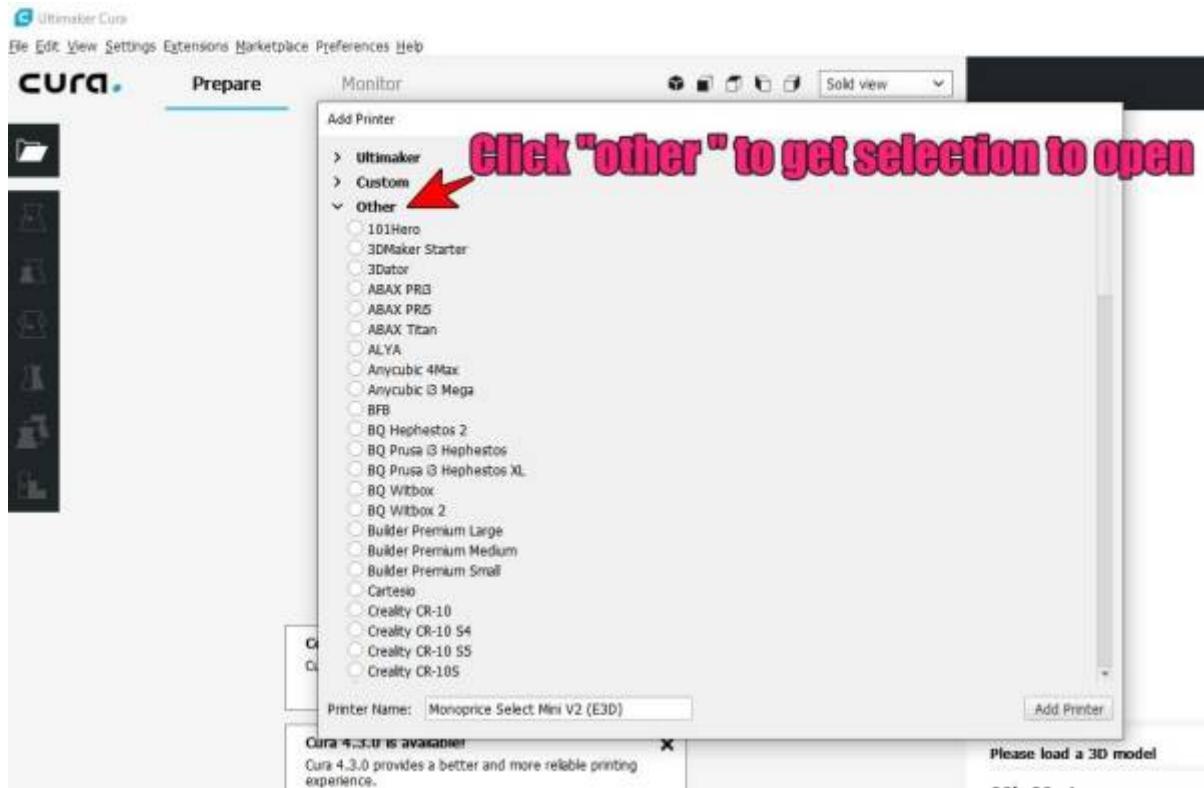
5. Scroll down & select "**Monoprice Mini V2 (E3D)**". Once selected, click "**Add Printer**".
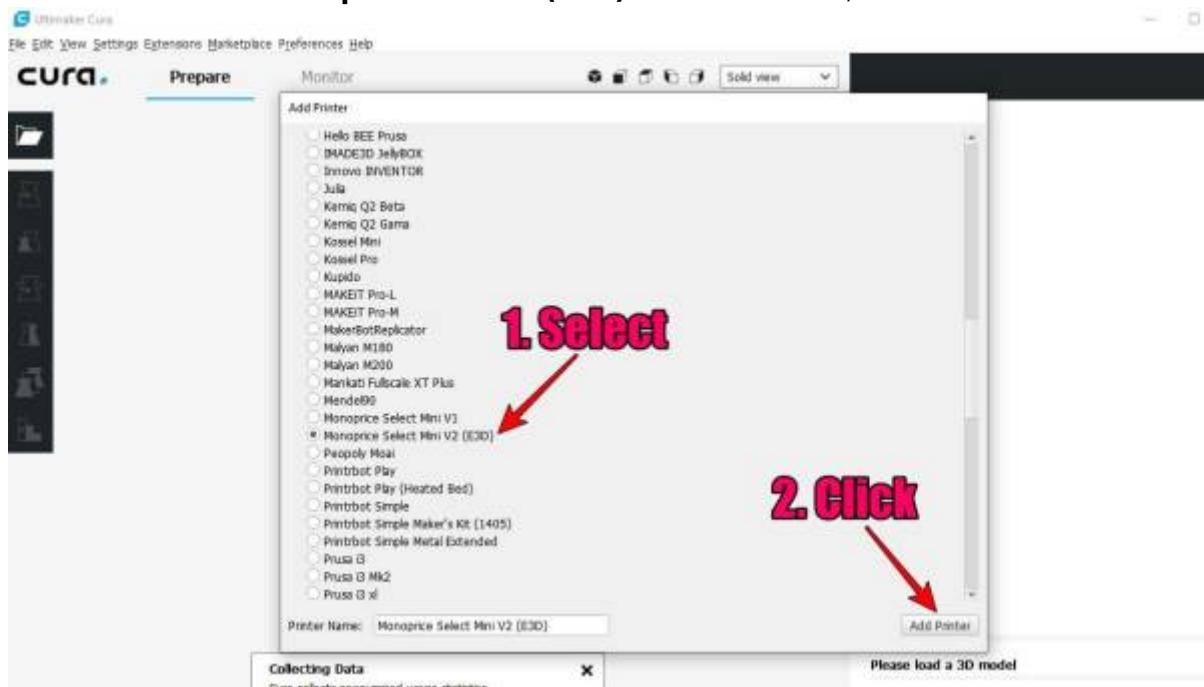
6. A new window will open, called "Machine Settings". **Do not alter any text, simply click "Finish"**.
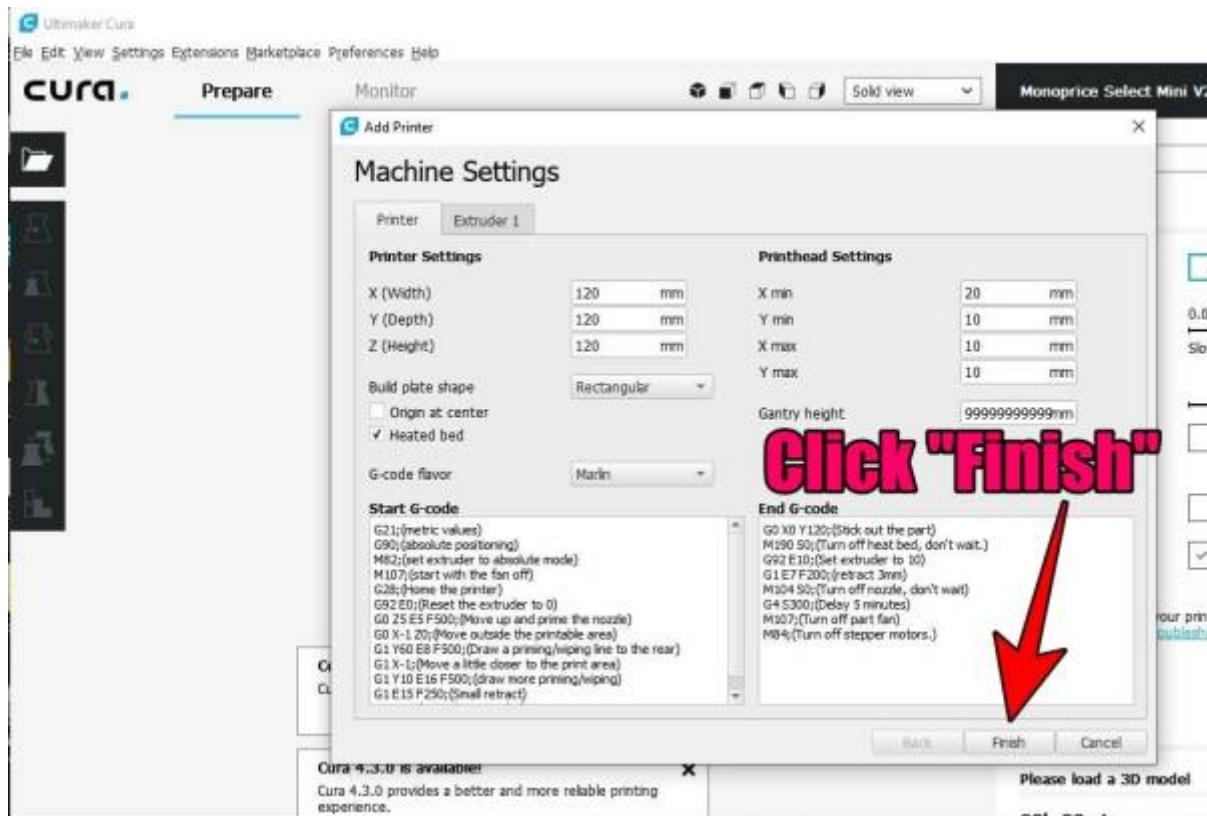
In the next step, we'll be downloading & installing a "plugin" from the CURA marketplace (free).

1. Open CURA, if it isn't already.
2. Click on "Marketplace", as shown in the pic below:
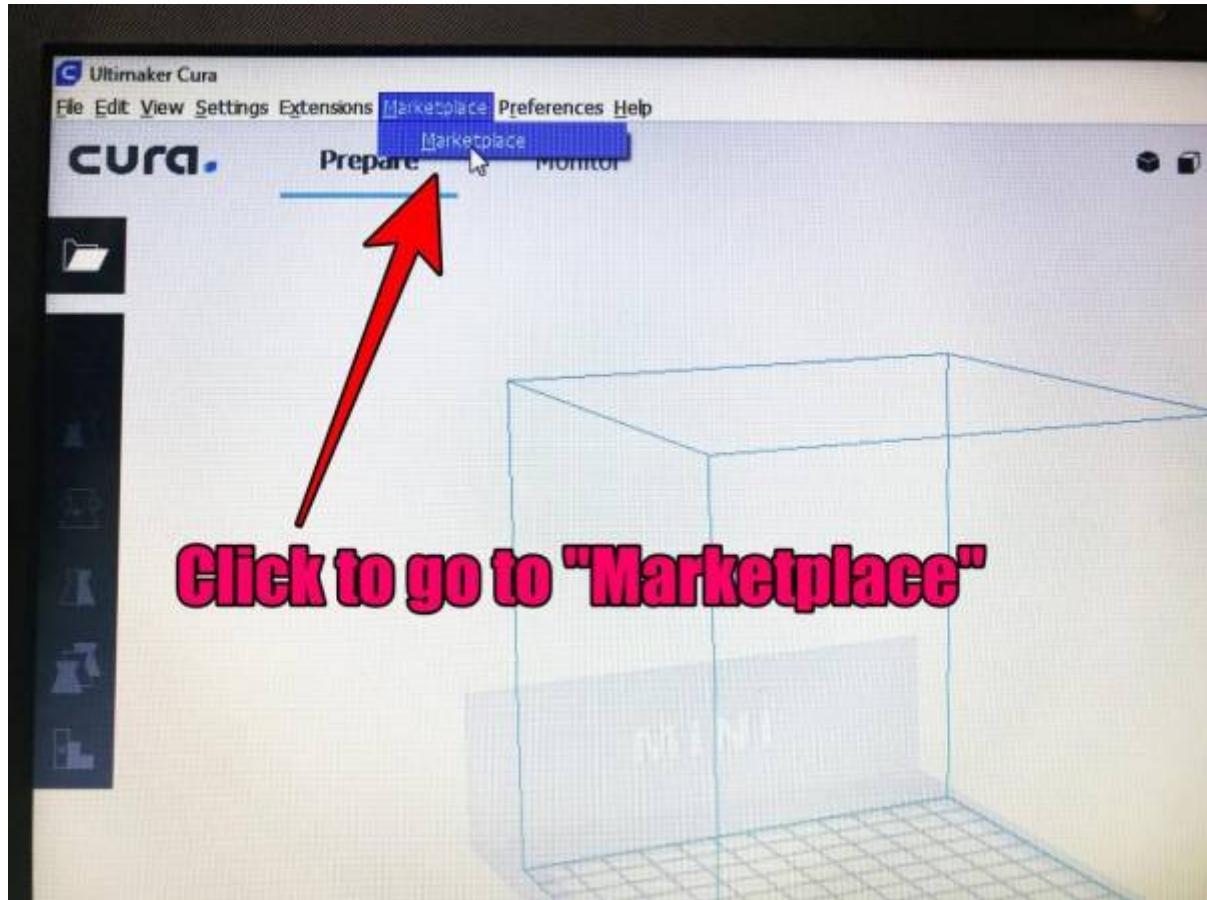
3. Double click on "Cura Settings Guide" to open it.
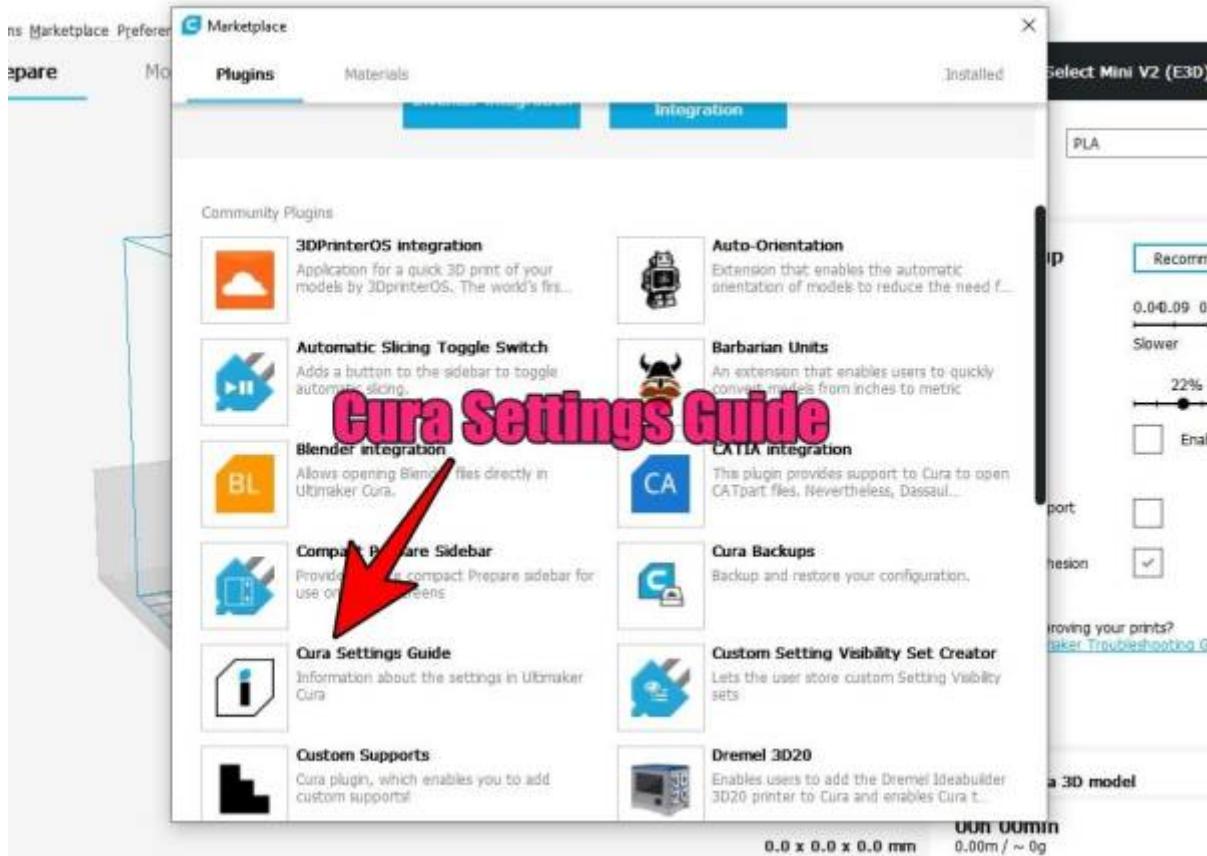
4. Click "install" to start the installation.
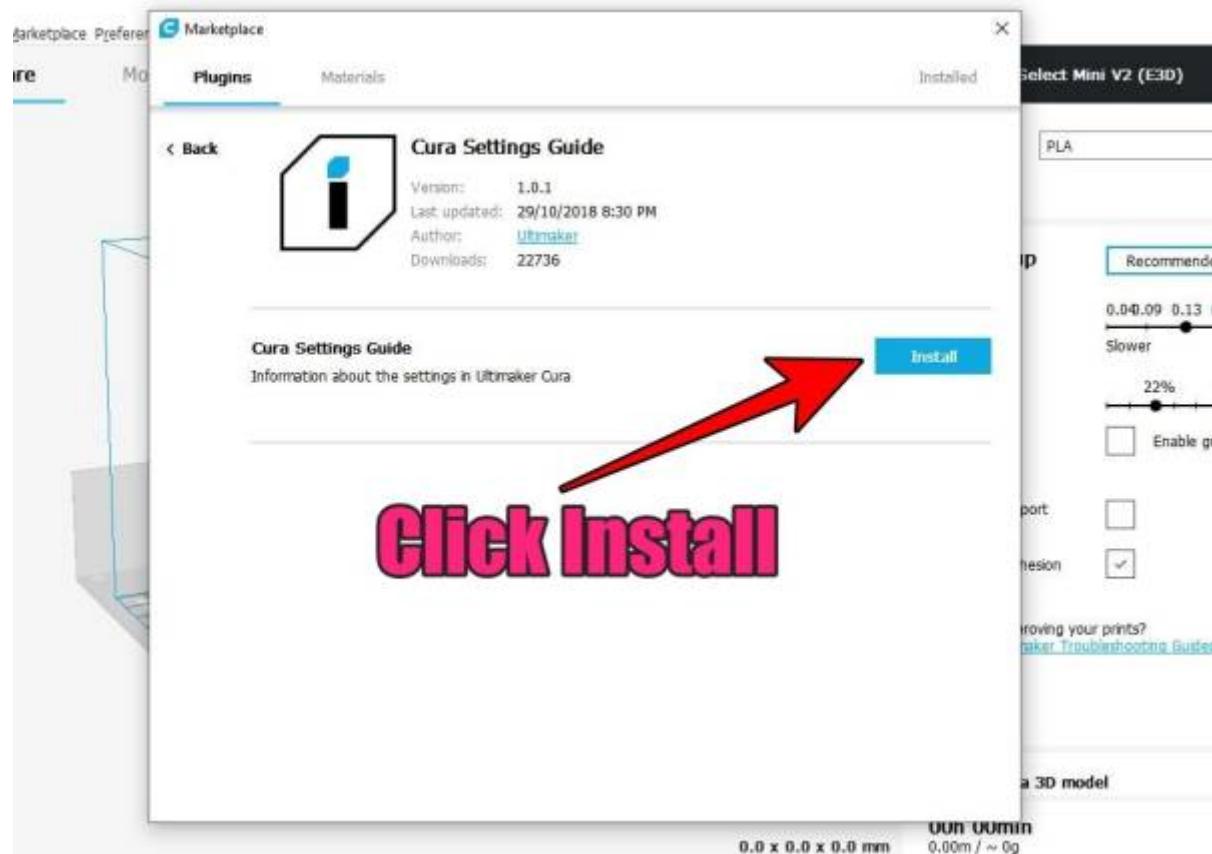
5. Click the "Quit CURA" button to complete the install.
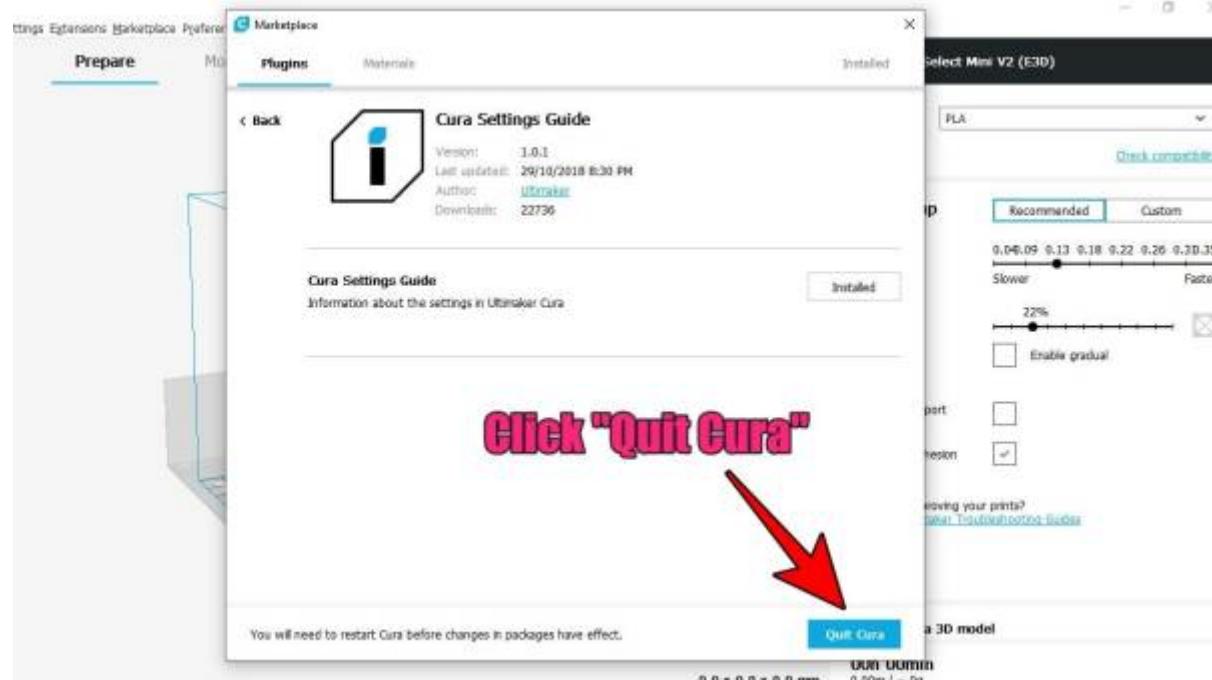
Cura will close & when re-started, the plugin will be installed.

6. Re-start CURA to make sure all is working. You should get a window like that in the picture below.
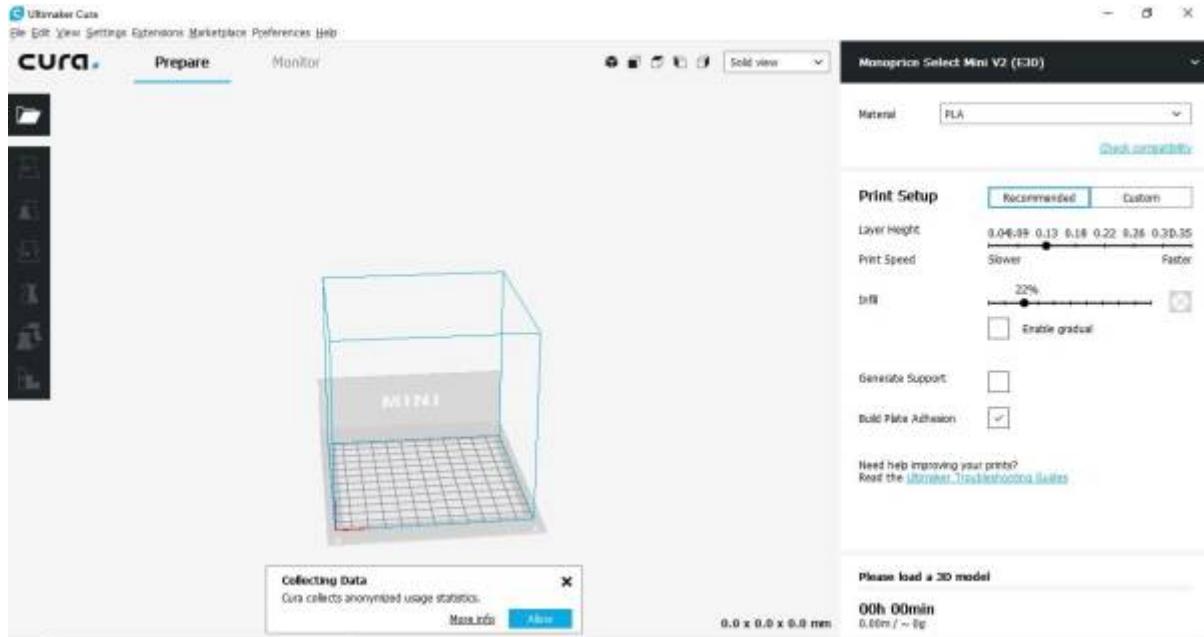
If CURA looks as it does in the above picture, all has gone well. You are now ready to attend the "Basic 3D Printing" Course.

Workshops in Isaac

- 22/08/2020 | 9.30am - 2.30pm | Carmila
- 23/08/2020 | 9.30am - 2.30pm | Dysart
- 05/09/2020 | 9.30am - 2.30pm | Clermont
- 06/09/2020 | 9.30am - 2.30pm | Middlemount
- 19/09/2020 | 9.30am - 2.30pm | Glenden
- 20/09/2020 | 9.30am - 2.30pm | Nebo
- 17/10/2020 | 9.30am - 2.30pm | Moranbah

Workshop Facilitator

Royce from 3D Print CQ will facilitate the workshops.

I'm Royce, a trade qualified auto-electrician with over 30 years experience & a passion for electronics & design. This experience coupled with a passion for excellence is clearly evident when I design and print my products.

I first became interested in 3D printing when I found a website called "Thingiverse". I found a really unique design to print, but had no way to do so….

So I bought a 3D printer.

My love of 3D printing grew from then on. I've always liked to make things on my lathe or mill, but it's slow work. With a 3D printer, I can create complex designs & print them in far less time.

I found that there was always a product I needed, but couldn't find anything on the market, so began designing my own products.

Workshop Material

- Workshop Notes provided by 3D Print CQ

BED LEVELING

BY BILLIE RUBEN

Bed leveling is, quite literally, the foundation of any successful print, and one of the biggest hurdles newbies must overcome in order to avoid the dreaded plastic spaghetti. But never fear! I've put together this handy guide to help you! ♥

INCORRECT BED LEVELING CAN CAUSE:

POOR ADHESION
Making first layers hard to lay down

FAILED PRINTS
Resulting in plastic spaghetti

UGLY FIRST LAYERS
That have gaps or weird patterns

WARPING
Ruining the accuracy of your prints

It occurs when the nozzle is not aligned properly to the bed. This is more accurately called 'tramming' (as nothing is being leveled to the ground), but 'leveling' is the common verbiage.
As beds are usually leveled at 3 or 4 points, you may see any combination of the below in your first layer.

X MUCH TOO CLOSE
Filament extrudes thin or not at all in some places, often bulging out in others. Extruder motor might slip/click.

X A LITTLE TOO CLOSE
Filament bulges out of the sides of the nozzle which squishes into other lines, sometimes causing a pattern.
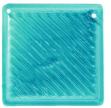
✓ PERFECT
Filament is flattened, adheres well to nearby lines, no gaps or peaks between lines.

X A LITTLE TOO FAR
Filament appears rounded, connection between lines is loose, gaps may appear.

X MUCH TOO FAR
Filament is totally rounded, little to no connection between lines, doesn't stick to bed.

HOW TO FIX IT: LIVE LEVELING

1 ROUGH LEVEL, VISUALLY
Move nozzle to lower left corner of bed, adjust leveling knobs until nozzle and bed touch, then turn it back just a bit little until a gap barely appears. Repeat for each knob on your bed.

2 SLICE A LEVELING HELPER
In your slicer, resize a rectangular prism so the base is almost as large as your print bed, and set the bottom layer pattern to concentric.
First layer should look like a square spiral.

3 LIVE LEVEL
Print it. As it's laying down the first layer adjust the leveling knobs so that the lines it's extruding look like the perfect example above, then cancel the print.

4 MAINTAIN IT
Your bed is now level. To maintain the level, print all future models with a skirt (or brim) and watch it lay the brim down, adjusting leveling as required as it prints the skirt.

FURTHER HELP:

If the above instructions aren't working for you, it could be one of these issues;
• If the middle of your bed is too high or low; your bed is warped, seek a replacement or add borosilicate atop it.
• Your speed may be too high. Seek a slicer settings profile from a trusted source (or use the slicer defaults).
• Your bed surface may need to be cleaned, or you may need to add some glue to the surface (esp. if cold).
• Your temperature may be wrong (bed usually should be around 60°C and nozzle 200 °C for PLA).
• Your z (up/down) axis end stop may be too high or low, try to move it, or print an adapter.
• The axis your nozzle travels on may not be level, seek instructions specific to your machine.
To solve these issues you might need some further help from other printer folk. I help run the largest 3D printing communities on **Discord (discord.gg/B4tp8MH)** and **reddit (reddit.com/r/3Dprinting).** We'd be happy to help you there!

I release full-resolution copies of these guides for free so you can print your own, but if you'd like to support me making more, consider purchasing a print from my store on **RedBubble (redbubble.com/people/BillieRuben/shop)**
If you'd like more of these guides, I'll be posting them to my **Twitter @BillieRubenMake.** Happy Printing! ♥ **Billie**

MODEL-MAKING PROGRAMS FOR 3D PRINTING

BY BILLIE RUBEN

Printing pre-made models can be fun,
but the real power of 3D printing comes in making your own designs, tailored exactly to your needs and desires.
This guide will help you choose the best software for you to use in making 3D models for your printed projects! ❤

WHAT DO YOU WANT TO MAKE?

ACCURATE OBJECTS	ORGANIC SHAPES	INORGANIC SHAPES	COPY OF A REAL THING
functional/ fitted parts, dimensioned objects, tools, products.	soft-surfaces, people, animals, characters, terrain, clothing.	hard/ flat surfaces, controlled curves, non-functional robots, vehicles, buildings.	people, buildings, objects, anything from real-life that would be hard to model.
You want… **PARAMETRIC DESIGN**	You want… **SCULPTING**	You want… **HARD-SURFACE MODELLING**	You want… **PHOTOGRAMMETRY OR SCANNING**
Which uses measurements, calculations, drawings, etc, to build an object that is easy to iterate upon.	Which emulates a digital block of clay that you can push, pull, smooth, inflate, pinch, etc.	Where you add and remove simple shapes (such as cubes, spheres, cylinders), to form complex objects.	Where you take many photographs or scans and combine them through software to form a model.
Start with… **Fusion 360**	Start with… **SculptGL**	Start with… **TinkerCAD**	Start with… **AliceVision**
✓ Huge community ✓ Various tools suited to iterative design ✓ Free for hobbyists	✓ Kid/school-Friendly ✓ Uncomplicated UI ✓ Great model repair ✓ Free and Open Source	✓ Kid/school-friendly ✓ Intuitive UI ✓ Editing existing STLs, ✓ Good model repair	✓ No extra equipment ✓ Works using photos ✓ Simple interface ✓ Free and Open Source
Or try… **FreeCAD**	Advance to… **Blender**	Advance to… **Blender**	Advance to… **Laser Scanning**
✓ Python scripting ✓ Free and Open Source	✓ Specialised tools ✓ Free and Open Source	✓ Specialised tools ✓ Free and Open Source	✓ Much greater detail ✓ High upfront costs

TUTORIALS and MORE OPTIONS

The above list just contains the more popular choices and the ones I find newbies learn quickest and easiest. There are, however, many options and it's important to find the program that suits you.
To that end, a comprehensive, community-curated list of programs can be found on the /r/3DPrinting Subreddit wiki:

reddit.com/r/3Dprinting/wiki/MakingModels

It also includes links to tutorials for each of the programs spoken about here (and others!).
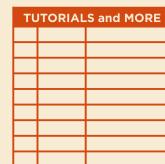
FURTHER HELP:

If you get stuck, I help run the largest 3D printing communities on **Discord (discord.gg/B4tp8MH)** and **reddit (reddit.com/r/3Dprinting).** We'd be happy to help you there!

I release full-resolution copies of these guides for free so you can print your own, but if you'd like to support me making more, consider purchasing a print from my store on **RedBubble (redbubble.com/people/BillieRuben/shop)**
If you'd like more of these guides, I'll be posting them to my **Twitter @BillieRubenMake**
Happy Printing! ❤ **Billie**

# CAD DESIGN TIPS FOR 3D PRINTING

BY BILLIE RUBEN

Like all production processes, 3D printing has constraints which are to be considered during the design phase in order to make high-quality, functional, and beautiful objects. I love seeing people make their own designs to fulfil their specific needs and desires, so I've collected many of these considerations to help you hit the ground running! ❤
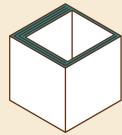
Make walls a multiple of your extrusion line width for a smooth slice. If it was 0.4mm use 0.8, 1.2, 1.6, etc.
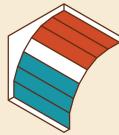
Filament must be laid upon existing material, so avoid steep overhangs to reduce the need for support.
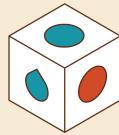
Vertical holes are fine, but horizontal ones should be tear-drop shaped to mitigate steep overhangs.

Pointed arches are better than round ones as they eliminate steep overhangs.
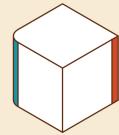
Vertical edge fillets increase quality by reducing inertia during harsh directional changes.
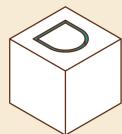
Roughly 0.3mm clearance should be added between fitted parts using offset face at end of modelling process.
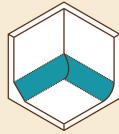
Adding a fillet or chamfer between a wall and base strengthens the join by adding more interface.
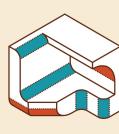
Fillets don't work well from below, due to harsh overhangs. But they can look great in other areas.
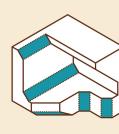
Equal chamfers always work (even from below) as their overhang remains at a printable 45°.
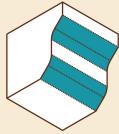
Combining fillets and chamfers mitigates the issues of fillets alone and smooths the chamfer.
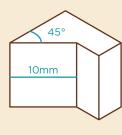
Using parameters and constraints allows you to easily edit and iterate upon your designs.
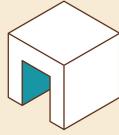
Printers can bridge gaps between bodies quite easily Distance varies, but most can easily handle 2cm+.
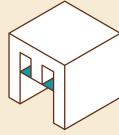
A thin, sacrificial bridging layer can reduce the need for support material. It is cut away after printing.
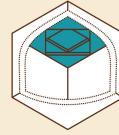
Triangles can be staggered under a large roof, to enable larger distances to be bridged.

Sacrificial, perpendicular ribs can be added to support overhangs during printing.
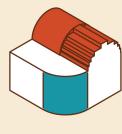
Curves look good with an axis in the Z direction, but due to the layering process can look very poor in the X/Y.
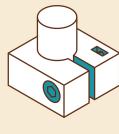
A slit, bolt and trapped nut can be added to holes to allow them to be tightened around another part.
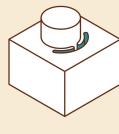
Compliance can be added to parts to enable flex, which enables push-fitting parts.
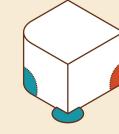
Reduce the risk of a print warping up from the bed by rounding out or adding mouse ears to corners.
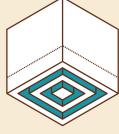
Concentric slits can be cut from the base of a model to about 10mm up to prevent warping.

Use software that makes manifold objects (without tiny gaps or reversed faces), to avoid slicing errors.
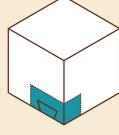
Complicated or fitted parts of an overall print can be isolated and printed to test for fit.
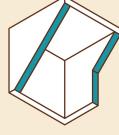
Diagonal ribs can be added to support/enable a roof to bridge between them. Can be beneficial inside a model.

Text looks best when indented into a vertical surface. It reduces overhangs and has better resolution.
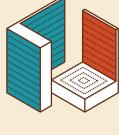
Due to the planar layering of most 3D printers, print orientation has a significant impact on strength.

FURTHER HELP:

If you get stuck, I help run the largest 3D printing communities on **Discord (discord.gg/B4tp8MH)** and **reddit (reddit.com/r/3Dprinting).** We'd be happy to help you there!

I release full-resolution copies of these guides for free so you can print your own, but if you'd like to support me making more, consider purchasing a print from my store on **RedBubble (redbubble.com/people/BillieRuben/shop)**

If you'd like more of these guides, I'll be posting them to my **Twitter @BillieRubenMake**
Happy Printing!   ❤ **Billie**

iki