

Cura Slicer Quick Start Guide

Cura is “slicer”: a program that converts 3D models into instructions for 3D printers by “slicing” the models into thin horizontal slices. Without Cura, you would have no good way of telling your JellyBOX what to print!

Naming Note: Cura is an open-source slicer. Since version 3.04, “Cura” has been renamed to “Ultimaker Cura” to signify Ultimaker’s role in chaperoning the development of this gem. Cura IMADE3D Edition is a customized Cura with the latest and greatest slicing settings for your JellyBOX.

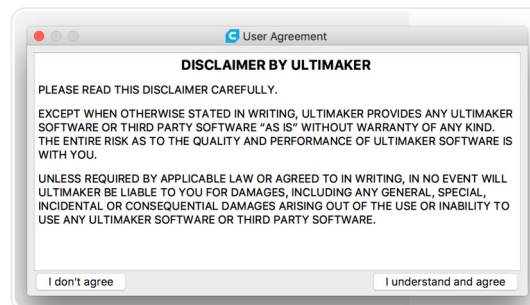
Table of Contents

1. [Initial Setup](#)
2. [Cura Interface](#)
3. [Your First Slicing Job](#)
4. [Pro Tip: Use our Simplify-Ed Theme to Get Pleasant Colors](#)
5. [Cold vs. Heated Bed](#)
6. [What’s Next?](#)

Initial Setup

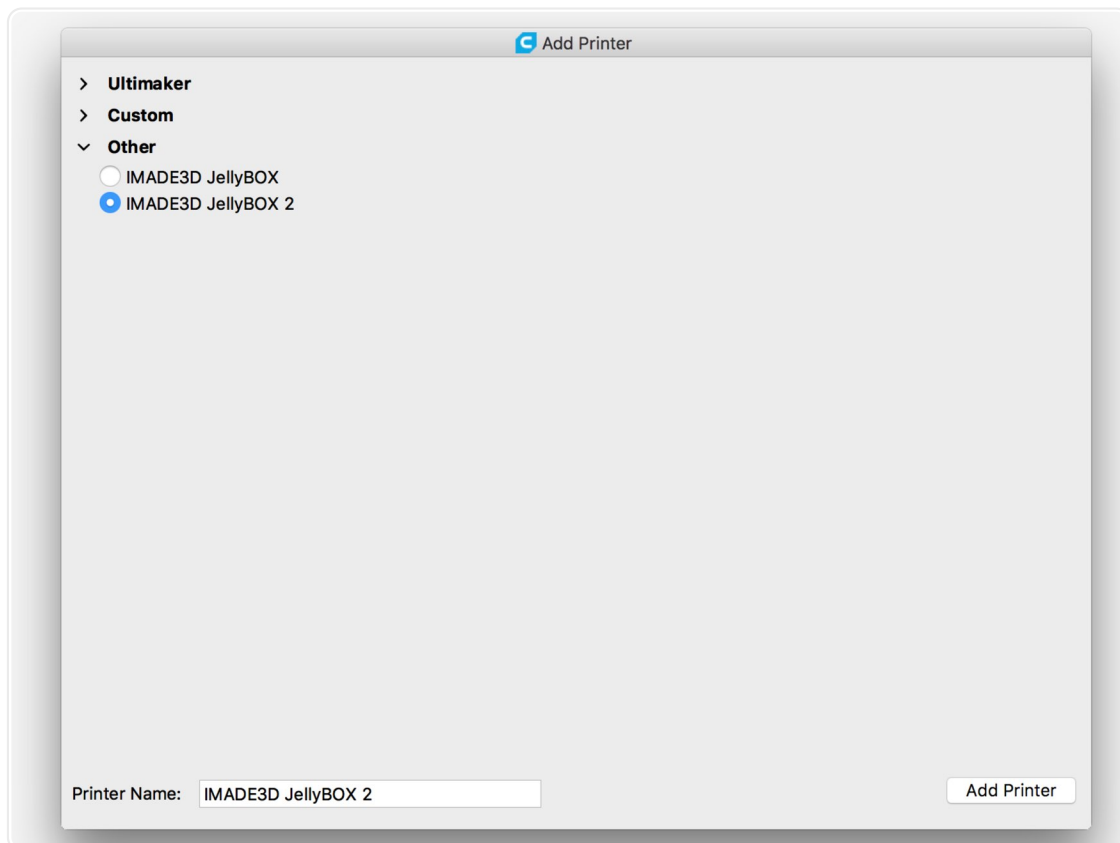
Cura on your computer may look a bit different depending on your system and Cura version.

When you open Cura for the first time, you may have to agree to the User Agreement.



The **Add a new printer** wizard will start automatically if this is your first time. Select either JellyBOX or JellyBOX 2.

(Later, you can add additional machines by going to Settings > Printer)



You also have the option to allow or disable Collection of *anonymous* usage data that is used to help develop future Cura versions. We recommend *allow*.

Collecting Data ✕

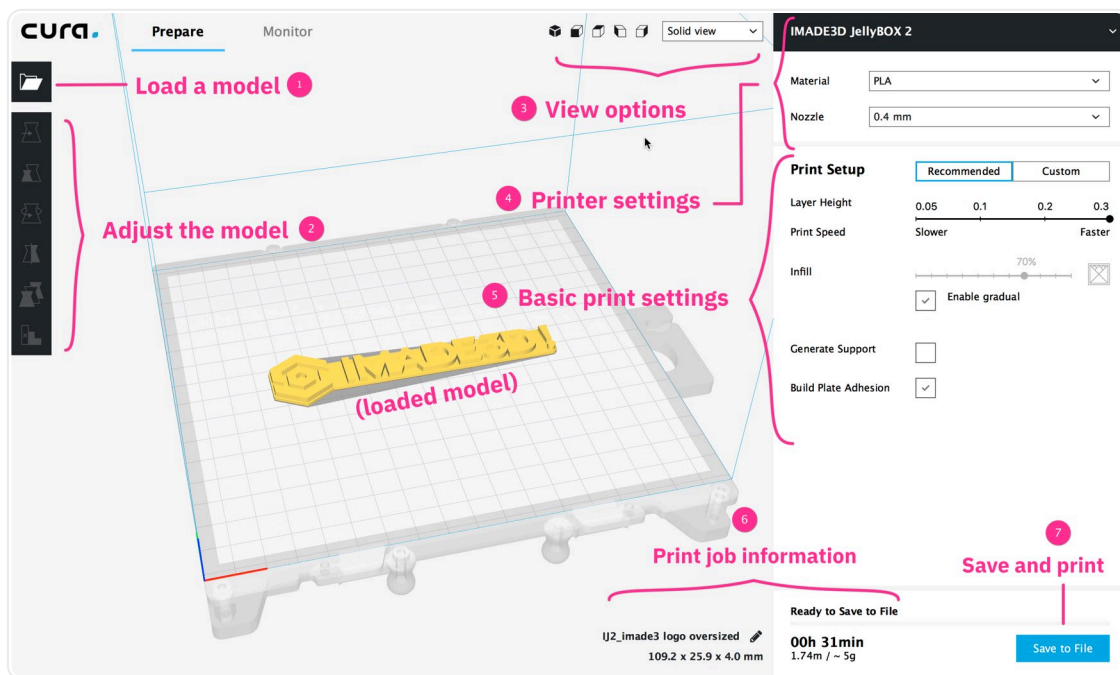
Cura collects anonymized usage statistics.

[Allow](#)

[Disable](#)

Cura Interface

After you add your first printer, you see the main Cura screen. Here's the main sections



1. Model loading button
2. Model adjustment tools
3. View options
4. Printer settings
5. Basic print settings
6. Print job information
7. Save and print

Moving Around

It's essential that you can see your model from various angles. Practice this and you'll feel like a master!

- **shift click** and drag to **pan** (=move)
- **right click** and drag to **rotate**
- **scroll** to **zoom**

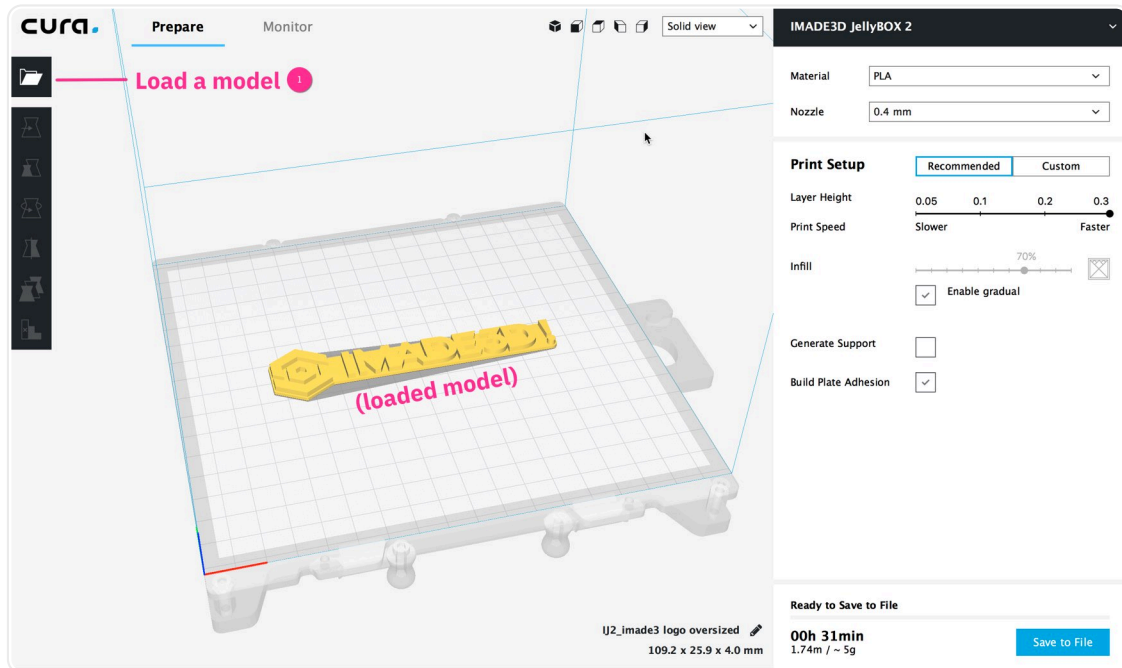
Full report:

Action	Mouse	No-scroll mouse	Touchpad
Click (to select...)	Left button	Left button	Click
Pan = Move	Push scroll	Shift + right button	Shift + click
Rotate	Right button	Right button	Control + click
Zoom	Scroll	Space + vertical move	Scroll (two fingers) OR Space + vertical move

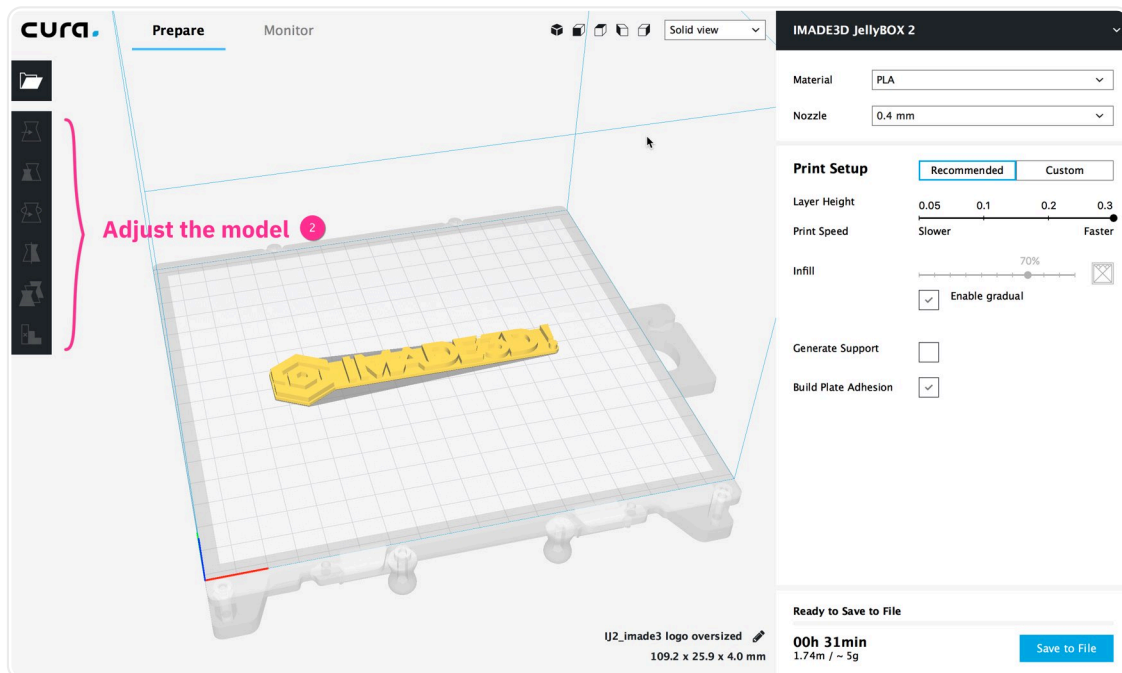
You may supplement manual manipulation with **preset views** in the View Options area.

Your First Slicing Job

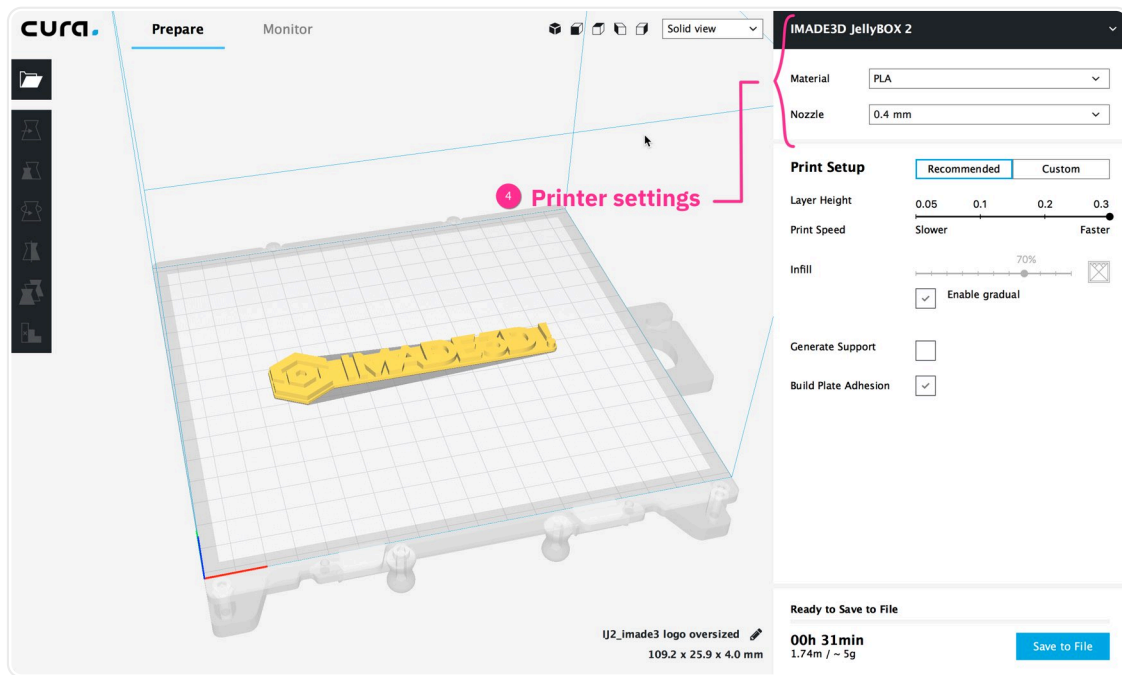
Start by loading a 3D model by clicking on the button or drag-and-dropping from file manager (or file launcher like Spotlight or Everything) into Cura.



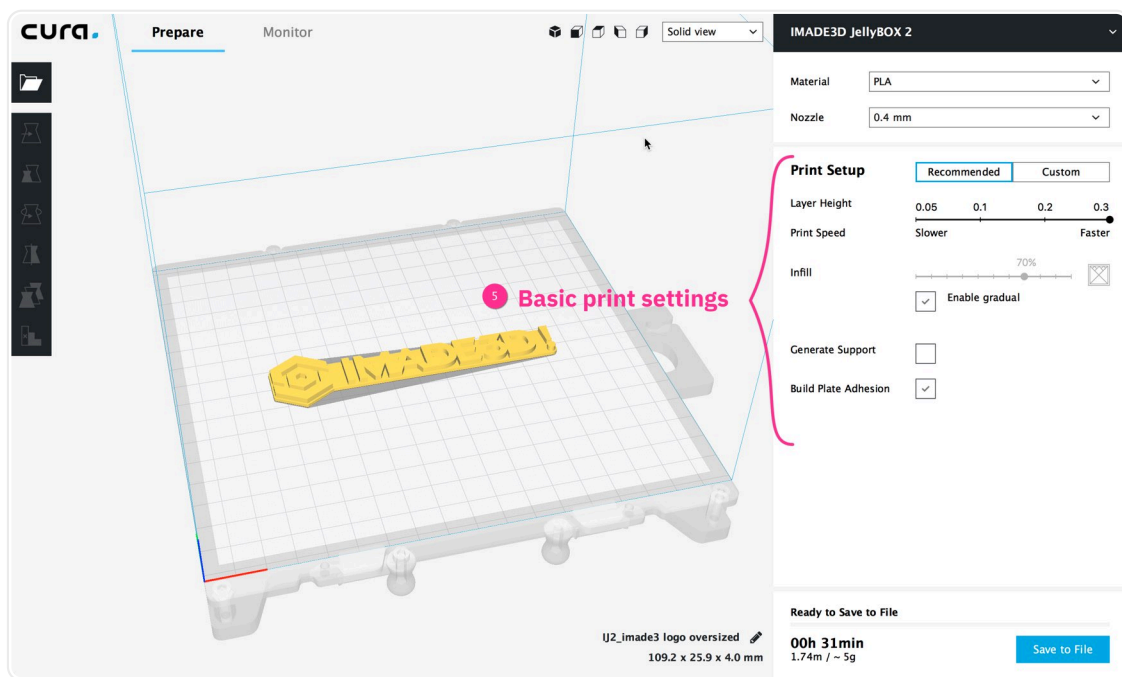
Adjust the model. You can move it around, resize it, rotate it, scale it up or down, flip it...



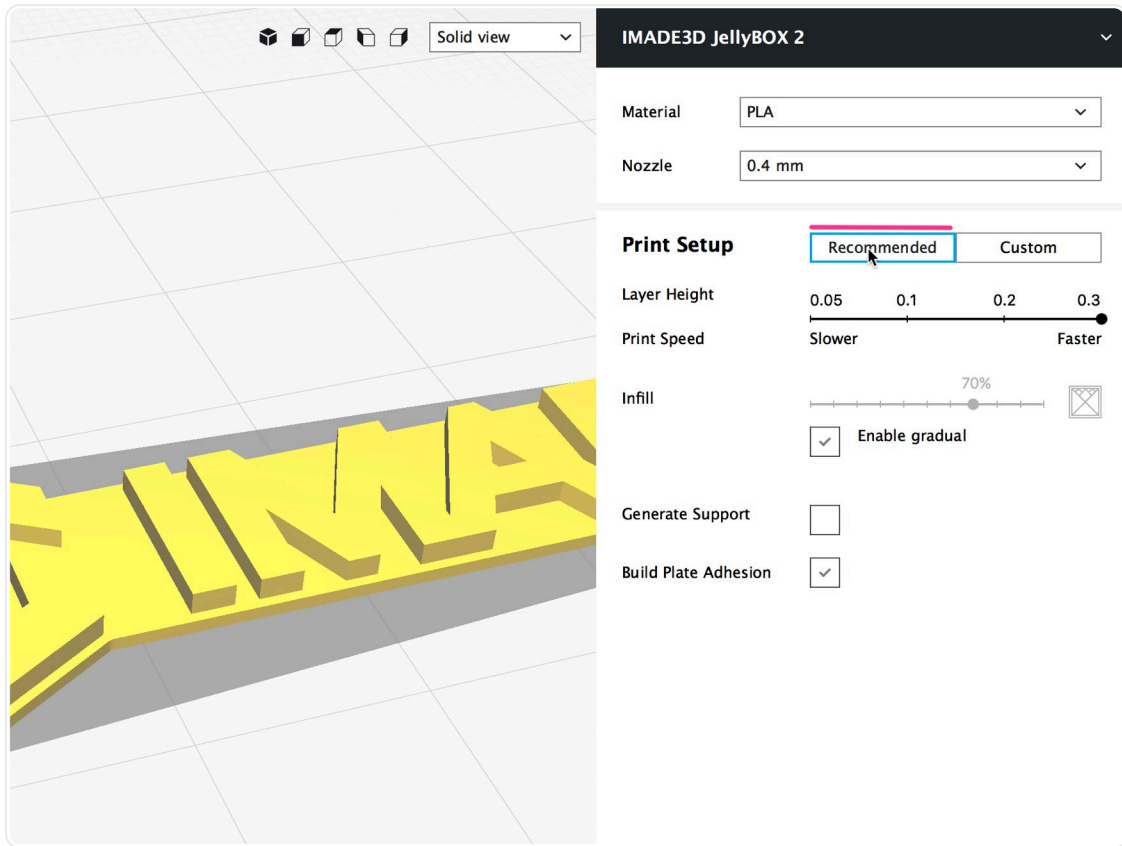
Choose a printer and nozzle (one fan, two fans, nozzle size...) The default is JellyBOX 2 and 0.4mm nozzle. Also choose a material profile. You probably start with PLA, which is widely available and prints marvelously.



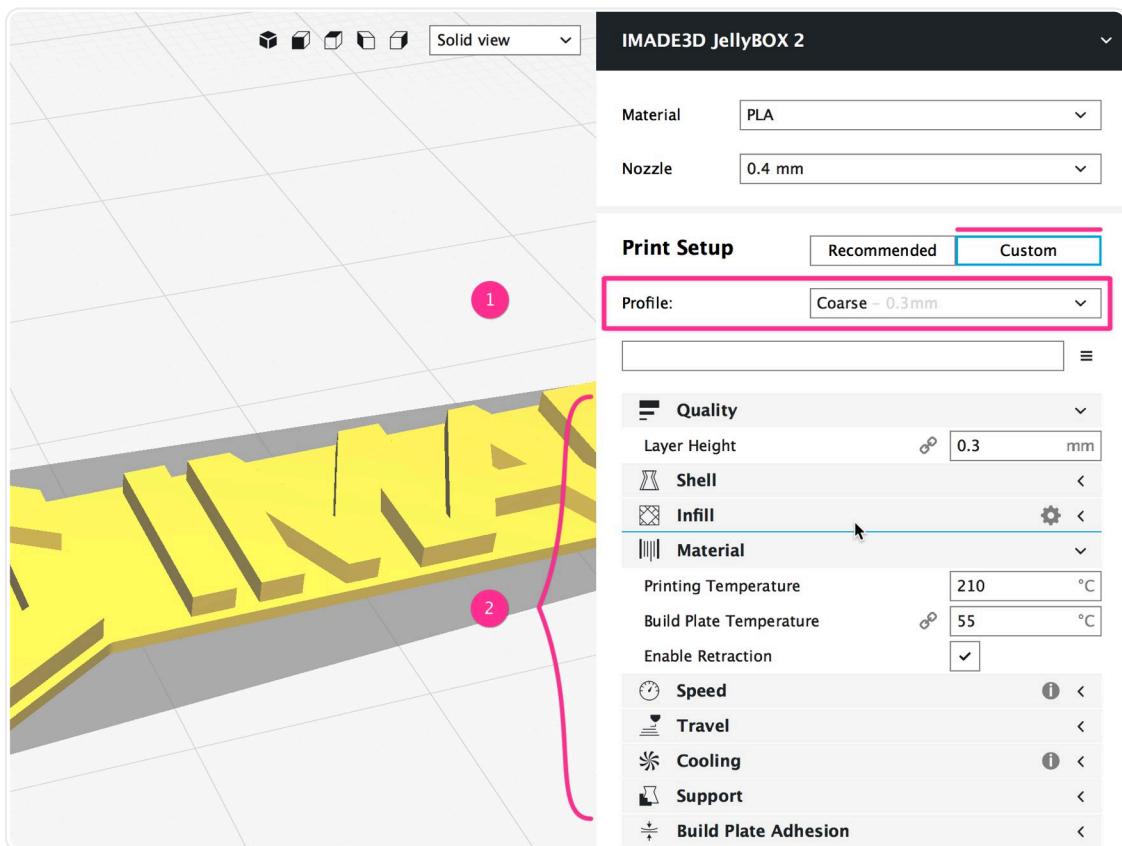
Adjust **slicing settings** in the Recommended or Custom mode.



In the Recommended mode, you can only adjust basic settings like infill density or the presence of supports. Honestly, we almost never use this 'simplified' mode.



We *highly recommend* the Custom mode! Here, you can select from our meticulously crafted *quality presets* (1) or modify any of the hundreds of settings (2).

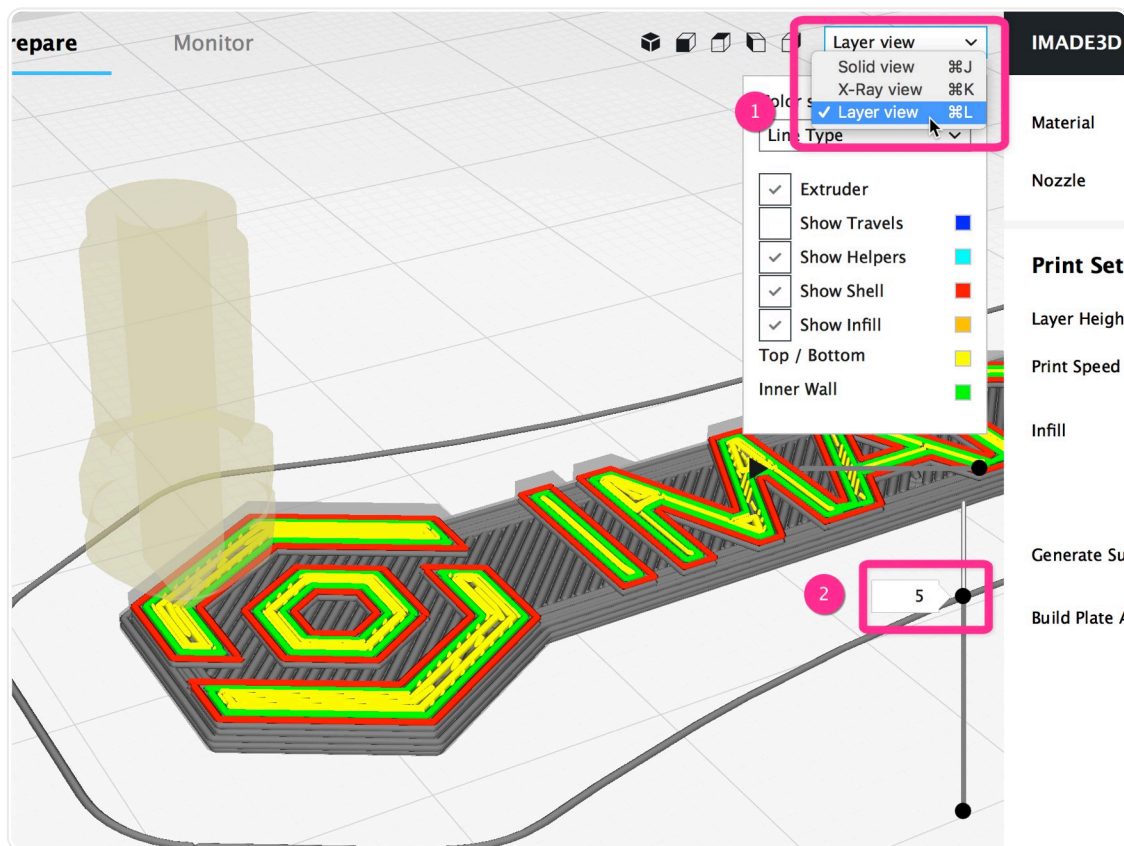


- **Quality Presets**

- **Coarse.** Recommended profile for general use. Prints the strongest parts in the least amount of time. Your printed part will have clearly visible 0.3 mm thick layers.
- **Medium.** Prints slower and slightly(!) more brittle parts than Coarse. Better for printing steep overhangs and small features.
- **Fine.** Prints slower and slightly more brittle parts than Medium. Great for printing steep overhangs and small features. Smooth-looking. Some people think it looks ‘better’ than coarser profiles.
- **UltraFine.** Takes a very long time to print (even days). Produces curiously smooth prints and amazing overhangs.

Whether you’re a beginner or a Pro, the Quality Presets are your best friend!

Preview with the indispensable Layer view. You always want to double-check your slice before export to see if you need to adjust any settings. Switch to the Layer view (1) and then use the slider (2) to scroll between layers.



Save and print. Get that gcode into your JellyBOX!

CURA Prepare Monitor Solid view

IMADE3D JellyBOX 2

Material: PLA
Nozzle: 0.4 mm

Print Setup Recommended Custom

Layer Height: 0.05 0.1 0.2 0.3
Print Speed: Slower Faster
Infill: 70% Enable gradual
Generate Support:
Build Plate Adhesion:

Ready to Save to File

00h 31min
1.74m / ~ 5g

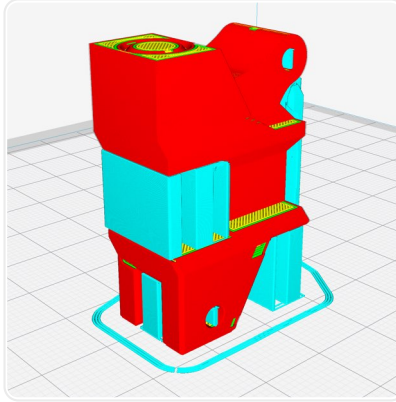
7 Save and print

Save to File

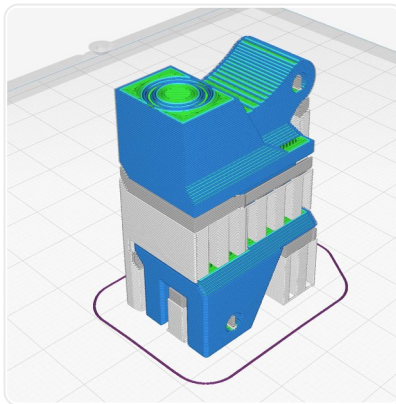
IJ2_imade3 logo oversized
109.2 x 25.9 x 4.0 mm

Pro Tip: Use our Simplify-Ed Theme to Get Pleasant Colors

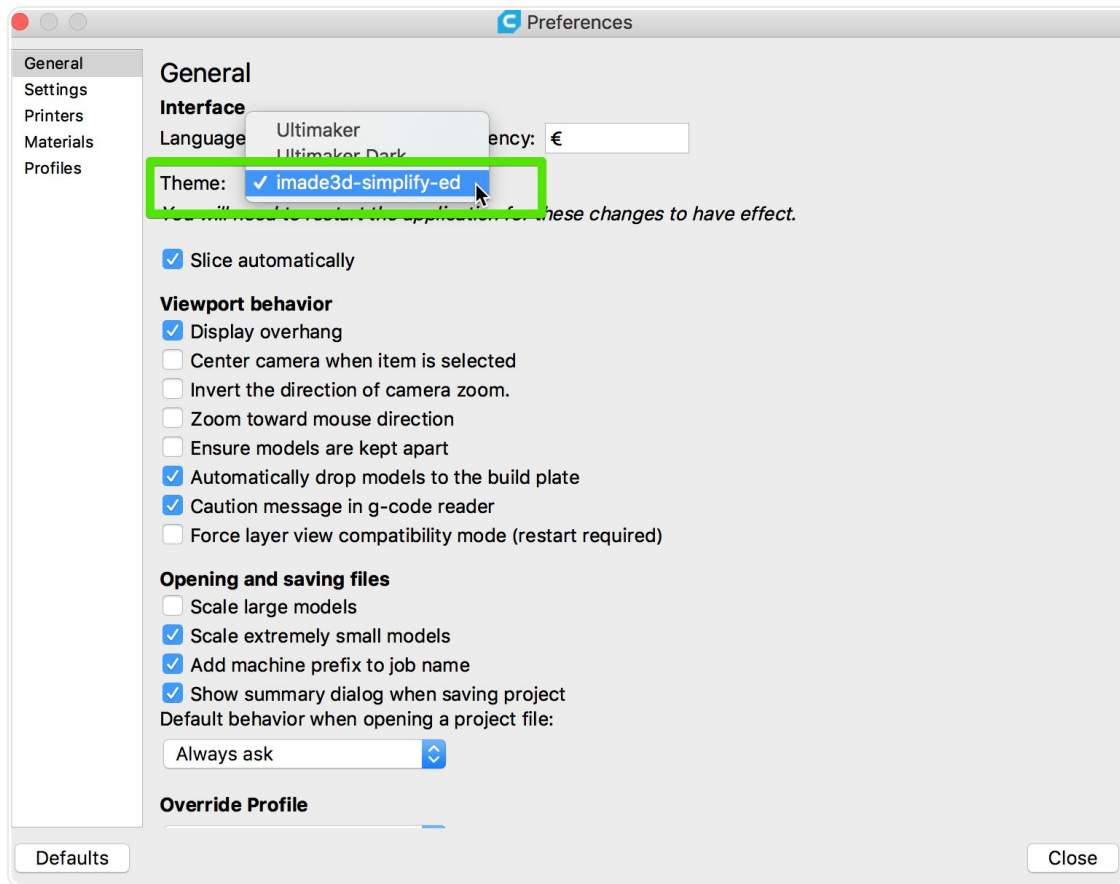
The default Cura theme has quite jarring colors that threaten to burn out your retinas.



Use our pleasant simplify-ed theme and feel like a boss.



Howto? Open the Preferences/ Settings, select our **Simplify-ed theme** and restart Cura. Voila! All this beauty! Honestly, I can't imagine using Cura without our Simplify-ed theme.

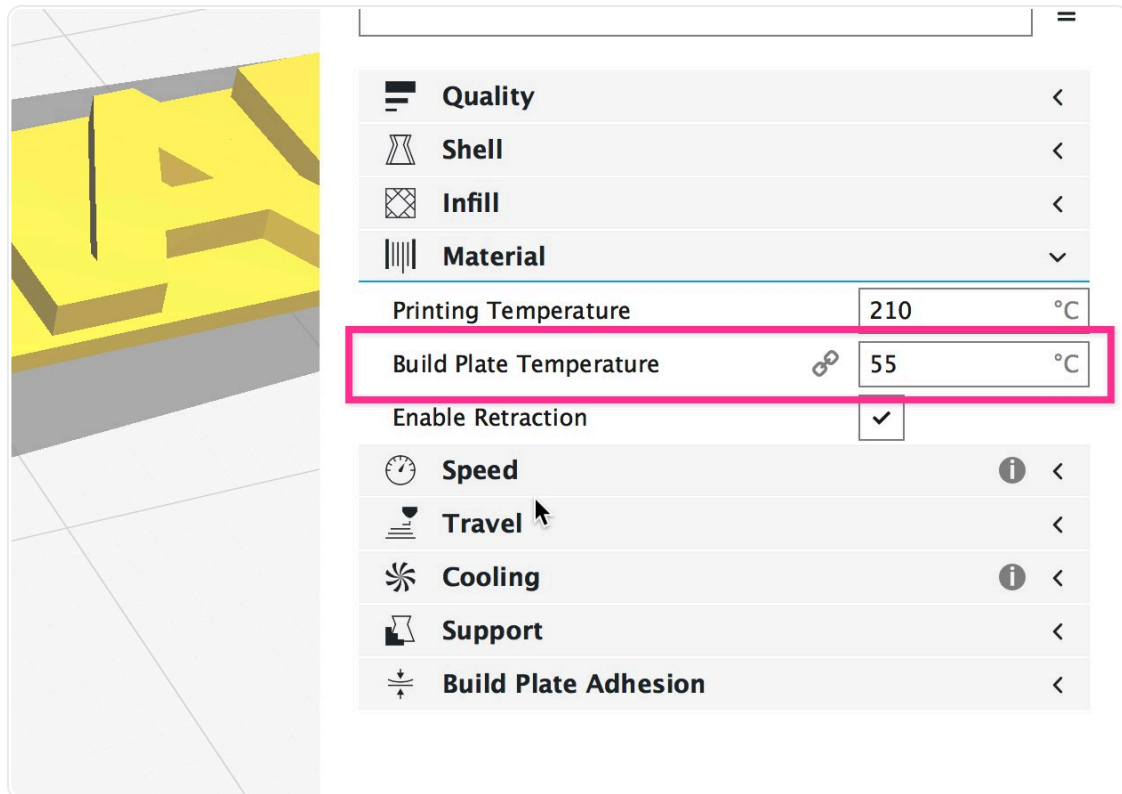


**The theme is called Simplify-Ed as a homage to Simplify3D slicer that (completely) inspired this color scheme. Say what you will about Simplify3D, but they do know how to pick a slicing preview color scheme.*

Cold vs. Heated Bed

All our print profiles **include heated bed turned on** by default.

Cold bed JellyBOXes ignore heated bed instructions. So, you **can** run gcodes with heated bed settings on a cold bed JellyBOX. No problem. And if in the future you upgrade to a heated bed (which we recommend if you will be printing other materials than PLA), you won't have to re-slice all your gcodes.



On the other side, if you have a heated bed and don't want to use it, set the Build Plate Temperature to 0°C.

The image shows a 3D printing software interface. On the left is a 3D model of a yellow object. On the right is a settings panel with the following sections:

- Quality** <
- Shell** <
- Infill** <
- Material** v

Printing Temperature: 210 °C

Build Plate Temperature 0 °C (highlighted with a pink border)

Enable Retraction:

- Speed** i <
- Travel** <
- Cooling** i <
- Support** <
- Build Plate Adhesion** <

What's Next?

Slicing is as important to the quality of your prints as the precision of your JellyBOX. To get the best possible results, you will need to move beyond the pre-made presets, and learn which settings to adjust depending on the model you want to print. Here's some resources to help you on the way:

All3DP Deep Inside the Cura 3D Slicer. This is a great and reasonably deep-dive into Cura's options. Text, images, and animated gifs.

<https://all3dp.com/1/cura-tutorial-software-slicer-cura-3d/>

Ultimaker Cura manuals are quite detailed, but can be a bit disjointed.

<https://ultimaker.com/en/resources/52340-quality>

And the [YouTube and search engines of the world](#) are full of tutorials on specific Cura features 😊