Using 'Cura'/ 'Cura IMADE3D Edition' / 'Ultimaker Cura' with IMADE3D JellyBOX and JellyBOX 2

Note: Cura on your computer may look a bit different depending on your system and Cura version. Yet, the procedure will be the same.

Naming Note: Cura is an well established open source slicer. Since version 3.04, "Cura" has been renamed to "Ultimaker Cura" to signify Ultimaker's huge contribution in leading and organizing the development of this open source gem. Cura IMADE3D Edition is a customized Cura with the latest and greatest slicing settings for your JellyBOX.

1. Add IMADE3D JellyBOX (2) machine

- Settings > Printer > Other > IMADE3D JellyBOX
- (When you start Cura for the first time, you will be presented with the 'add Printer' dialog automatically.)

ile	Edit	View	Settings	Extensi	ons Help		
			Printer Nozzle Material Profile		✓ IMADE3D J Add Printer Manage Pri	ellyBOX	3D JellyBOX
						Nozzle -	& Material:
	Ultimaker Custom Other IMADE3I IMADE3I	D JellyBOX D JellyBOX :	2	G	Add Printer		
Print	er Name:	IMADE3D	JellyBOX				Add Printer

- Choose the variant depending on your configuration of nozzle size/ number of filament fans.
- JellyBOX 2 has only one variant
- JellyBOX Original has one filament fan and two filament fan variants
 - Use 0.4 mm if you have JellyBOX with 0.4mm nozzle (default) and only one single filament fan on the left side.
 - Use 0.4 mm 2-fans if you have JellyBOX with 0.4mm nozzle (default) and the dual fan upgrade. In this case, you have a filament fan on both left and right side.

olid view	~	IMADE3D Je	IIybox				~
		Material	PLA				~
		Nozzle	0.4 mm				~
			0.4 mm	1 2-fans			
		Print Setup)	Recom	mended	Custor	n
		Layer Height		0.05	0.1	0.2	0.3
		Print Speed		Slower			Faster
		Infill		20%	able gradua		

3. Select a Material Profile

- PLA
 - a generic PLA profile for both cold and heated platforms.
- PETG
 - a generic PETG profile for both cold and heated platforms.
- IMADE3D Green/ Pink PETG
 - The settings are slightly tweaked to get the best results with PETG in IMADE3D colors that you get from us.

1 🗗	Solid view ~	IMADE3D Je	ellyBOX	~
		Material Nozzle	PLA PETG ✔ PLA	~
	Green PETG	PETG >	IMADE3D	
	Pink PETG	Print Setu	Manage Materials Recommended Custom	
		Layer Height	0.05 0.1 0.2	0.3

4. Select a Quality Profile

To switch between quality profiles, go to the Custom tab and use the drop down menu



- Coarse
 - 0.3mm layer height
 - Recommended profile for general use. Prints the strongest parts in the least amount of time. Your printed part with have clearly visible 0.3 mm thick layers.
- Medium
 - 0.2mm layer height
 - Prints slower and slightly(!) more brittle parts than Coarse. Better for printing steeper overhangs and small features.
- Fine
 - 0.1mm layer height

- Prints slower and slightly more brittle parts than Medium. Great for printing steep overhangs and small features. Smooth-looking.
- UltraFine
 - 0.05mm layer height
 - Takes a very long time to print (even days). Produces curiously smooth prints and amazing overhangs.

ne v	Material	PLA ~
ler	Nozzle	0.4 mm ~
Travels		
Helpers	Print Setup	Recommended Custom
Shell 📃		
Infill	Profile:	Coarse – 0.3mm 🗸
im 🔳		UltraFine - 0.05mm
		Fine - 0.1mm Medium - 0.2mm
	_	✓ Coarse - 0.3mm
	Q <mark>r</mark> ality	
	🖉 Shell	Update profile with current settings/overrides
	🔀 Infill	Discard current changes
	Materia	Manage Profiles

5. Pleasant IMADE3D Color Theme

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



• Use our pleasant simplify-ed theme and feel like a human again.



- Mac
 - Go to Ultimaker Cura > Preferences and select the simplify-ed theme

	C Preferences
General Settings Printers Materials Profiles	General Interface Language Uttimaker ancy: € Theme: ✓ Imade3d-simplify-ed Silice automatically Viewport behavior ✓ Display overhang Center camera when item is selected Invert the direction of camera zoom. Zoom toward mouse direction Ensure models are kept apart ✓ Automatically drop models to the build plate Caution message in g-code reader Force layer view compatibility mode (restart required) Opening and saving files Scale large models ✓ Scale extremely small models ✓ Add machine prefix to job name ✓ Show summary dialog when saving project Default behavior when opening a project file: Always ask ✓ Override Profile
Defaults	Close

- PC
 - Go to and select the simplify-ed theme



5. Difference Betwen Recommended and Custom tabs

- The **Recommended** tab in Cura is great for beginners. It's a simple mode, which only lets you tweak a few parameters.
- The Custom tab let's you select presets, but also see and change all the slicing settings.
- This can be very much overwhelming when you're getting started, but it's how you can eventually get the best results: by tweaking the settings to fit a specific 3D model.
- In The Custom tab, you can control which setting you actually want to see by clicking on the little hamburger menu.

Quality	Custom selection Basic ✓ Advanced	
Layer Height	Expert	Ţ
Initial Layer Height	Show All Settings	
Line Width	Manage Setting Vis	sibility
Wall Line Width	0.4	mm
Outer Wall Line Width	0.4	mm
Inner Wall(s) Line Width	0.4	mm
Top/Bottom Line Width	0.48	mm
Infill Line Width	0.6	mm
Initial Layer Line Width	100.0	%
Shell		<
🕅 Infill		<
Material		<

• You can also search for specific setting if you know what you want to tweak!

temperature			×	Ξ
Material			•	~
Default Printing Temperature		210		°C
Printing Temperature		210		°C
Printing Temperature Initial Layer		215		°C
Initial Printing Temperature		200		°C
Final Printing Temperature		195		°C
Default Build Plate Temperature	o	55		°C
Build Plate Temperature	o	55		°C
Build Plate Temperature Initial Layer	P	60		°C

6. Heated Bed ?

• All our print profiles include heated bed instructions 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.

Settings			Machine	e Settings			
Printers Materials	Machine Settings						
Plugins	Please enter the correct settings for your printer below:						
	Printer Settings			Printhead Settings			
	X (Width)	170	mm	X min	0	mm	
	Y (Depth)	160	mm	Y min	0	mm	
	Z (Height)	145	mm	X max	0	mm	
	Build Plate Shape Rectangular			Y max	0	mm	
4	GCode Flav	Bed For RepR	ap (Mar 🗘				
	Start Gcode			End Goode			
	; ; ; Jelly	/box Start S	cript Begin ; ; ; 	;;;; Jellybox End Script Begin ;;;			
	; Print Settings Summary ; (leave these alone: this is only a list of the M104 S0 :extruder heater off					ng Up	
						Close	

- We highly recommend the heated bed upgrade to print a wide variety of plastics.
- In general, you *do not* need a heated bed for most prints with *PLA*. PLA stick well to blue painter's tape, and you don't have to wait for the bed to heat up. Even if you have a heated bed, you may elect to set the bed temperature to only 25C-30C to combat unusually cold environments #printinginwinter

! Alert: Legacy Hotends with 10 mm Heat Block (Pre-2017) - If your hotend looks like this, with the heat block only 10mm long (current default is 20mm), then you have some old old JellyBOX, congrats on being a super-early adopter! - In general, you need to **set your material print temperature 10C higher** than the current JellyBOX profiles! - Alternatively, print up to 50% slower. - Else you're may have under extrusion problems.

