# Using 'Cura'/ 'Ultimaker Cura' with IMADE3D JellyBOX

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: that 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. They are doing a great job.

## 1. Add IMADE3D JellyBOX machine

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



## 2. Select Nozzle (= JellyBOX Variants)

- Choose the variant depending on your configuration of nozzle size/ number of filament fans.
- 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.

	IMADE3D JellyBOX		~	I	
	Nozzle & Material: Profile:	0.4 mm ✓ 0.4 mm 0.4 mm 2	► 2-fans	[ [4	~
	Print Setup Infill Enable Support Build Plate Adhesion	Recomm Hollow	Light	Custo Dense	om Solid
	Need help improving your Troubleshooting Guides	prints? Read	the <u>Ultim</u>	<u>aker</u>	
IJ_CuteOcto ♂ 82.6 x 82.7 x 56.5 mm      12.81 m / ~ 38 g	Ready to Save to File	ł		Save	to File

#### 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 PLA
  - These are only visual tweaks. The settings are identical to the generic PLA, but the model will be pleasingly green/pink.
- IMADE3D Green/ Pink PETG
  - These are only visual tweaks. The settings are identical to the generic PETG, but the model will be pleasingly green/pink.

IMADE3D Je	ellyBOX		~	X	
Nozzle & Nate	erial:	0.4 mm	~	PLA	~
Profile:		Coarse - 0	.3mm	PETG ✓ PLA	
Print Set u	Green PLA Pink PLA	PETG PLA Recomm		Manage	D Materials
Infill		Hollow	Light	Dense	Solid
Enable Suppo	rt				
Build Plate Ad	hesion				
Need help imp <u>Troubleshooti</u>	proving your p ng Guides	orints? Read	the <u>Ultim</u>	<u>naker</u>	
	IMADE3D Ja Nozzle & N ate Profile: Print Set u Infill Enable Suppo Build Plate Ad Need help imp Troubleshooti	IMADE3D JellyBOX Nozzle & Naterial: Profile:  Print Set Green PLA Pink PLA Infill  Enable Support Build Plate Adhesion Need help improving your p Troubleshooting Guides	IMADE3D JellyBOX         Nozzle & Naterial:       0.4 mm         Profile:       Coarse - 0         Print Set       Green PLA Pink PLA         Infill       Hollow         Enable Support       Hollow         Build Plate Adhesion       Need help improving your prints? Read         Troubleshooting Guides       Frank PLA	IMADE3D JellyBOX     Nozzle & Naterial:   0.4 mm   Profile:   Coarse - 0.3mm   PETG   Petg <td>IMADE3D JellyBOX     Nozzle &amp; N aterial:   0.4 mm   Perofile:   Coarse   Oarse   Perofile:   Perofil</td>	IMADE3D JellyBOX     Nozzle & N aterial:   0.4 mm   Perofile:   Coarse   Oarse   Perofile:   Perofil

## 4. Select a Quality Profile

- Coarse
  - 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
  - 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.

IMADE3D JellyB	ох	~	X		
Nozzle & Material:	0.4 mm	~	PLA	~	
Print Setup	Coarse - 0.3mm Medium - 0.2mm Fine - 0.1mm UltraFine - 0.05mm	3mm		~	
Infill Create profile from current settings/overr Update profile with current settings/overr Discard current changes					
Enable Support	Manage Profiles				
Build Plate Adhesio	n				
Need help improvir Troubleshooting G	ig your prints? Read t <u>uides</u>	he <u>Ultim</u>	aker		

## 5. Recommended/Custom

- The **Recommended** tab in Cura is great for beginners. It's a simple mode, which only lets you use our predefined presets and tweak a few parameters.
- The **Custom** tab let's you see and change all the slicing settings. This can be overwhelming when you're getting started, and there is no shame in using the Recommended mode.
- In The Custom tab, you can control which setting you actually want to see by going to Preferences > Settings (Settings Visibility).
  - Note that there are many settings that are actually hidden by default even if they are an important part of a preset
  - 0



#### 6. Heated Bed ?

- 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.
- The heated bed is **enabled by default** in Cura.

! **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 Settings							
Printers Materials Profiles Plugins	Machine Settings Please enter the correct settings for your printer below:							
	Printer Settings			Printhead Set	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 Bootongular		Pectangular A	Y max	0	mm		
ł	GCode Flavor RepRap (Mar			Ford Consider				
	Start Gcode		End Gcode					
	; ; ; Jellybox Start Script Begin ; ; ; 			; ;;; Jellybox E	nd Script Begin ;;	;		
	; Print Settings Summary : (leave these alone: this is only a list of the			M117 Finishing Up ;write Finishing Up M104 S0 :extruder heater off				
						Close		

• You have to make the heated be setting visible if you want to change the temperature. It's 'Build Plate Temperature' in the under 'Material'

Print Setup	Recommended	Custom		Q
Quality			i	<
🕂 Shell			i	<
🔀 Infill			i	<
Material			i	~
Printing Temperature		210		°C
Build Plate Temperature	2	55		°C
Diameter		1.75		nm
Flow		90		%
Enable Retraction		~		
( ) Speed			i	<

#### ! Alert: Legacy Hotends with 10 mm Heat Block - 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! - 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.

