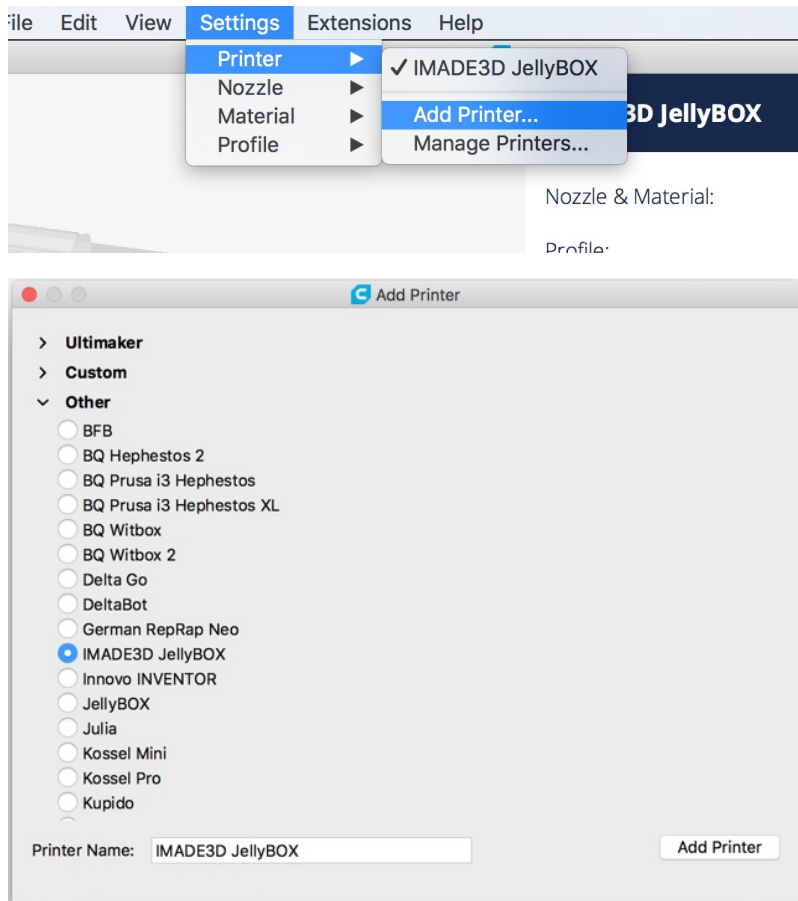


=== Using Cura IMADE3D Edition ===

Add IMADE3D JellyBOX machine

- Settings > Printer > Other > IMADE3D JellyBOX
- (Of course, only if you haven't already done that.)



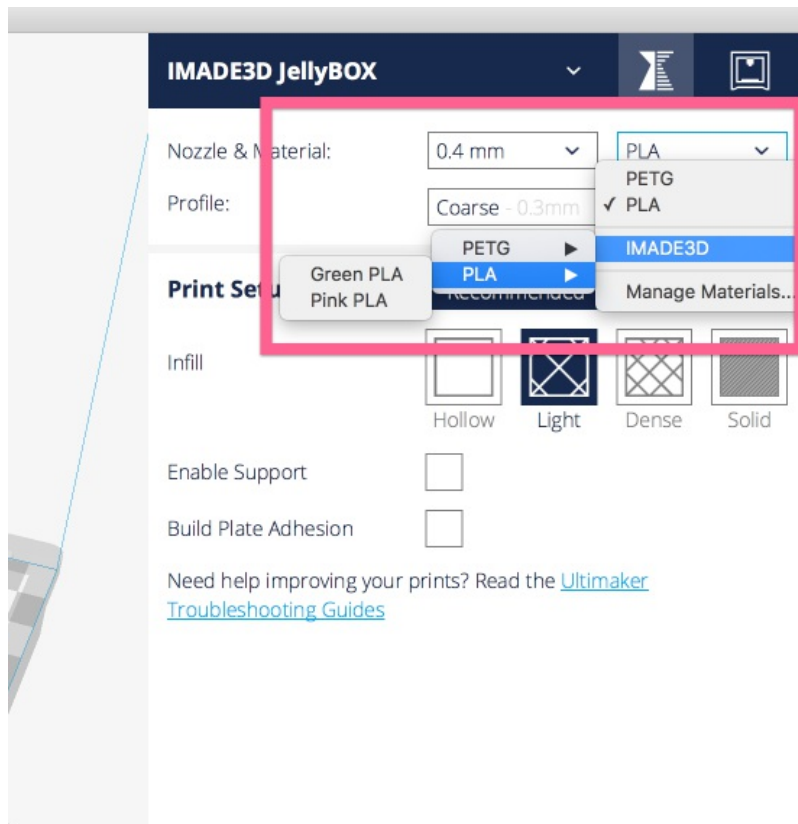
Select Nozzle = Variants

- 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.



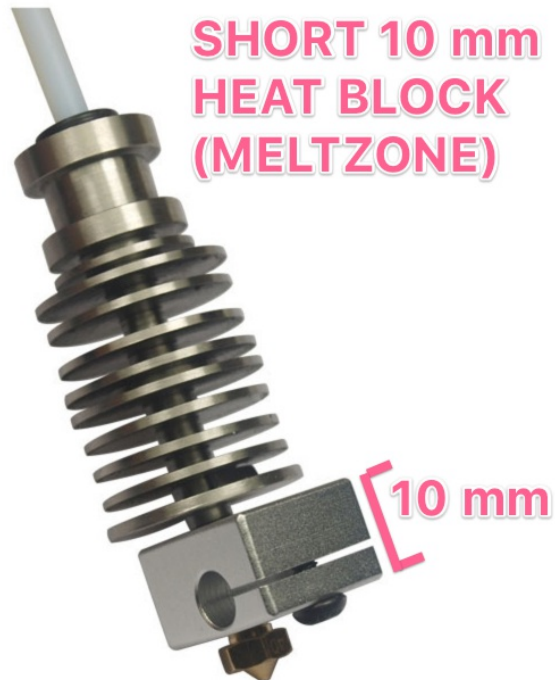
Select a Material Profile

- PLA
 - a generic PLA profile. Heats up nozzle to 210C and bed to 60C (If you have a heated bed.)
- PETG
 - a generic PETG profile. Heats up nozzle to 235C and bed to 50C (If you have a heated bed.)
- IMADE3D Green/ Pink PLA
 - These are only visual tweaks. The settings are identical to generic PLA, but the model will be pleasingly green/pink.
- IMADE3D Green/ Pink PETG
 - These are only visual tweaks. The settings are identical to generic PETG, but the model will be pleasingly green/pink.



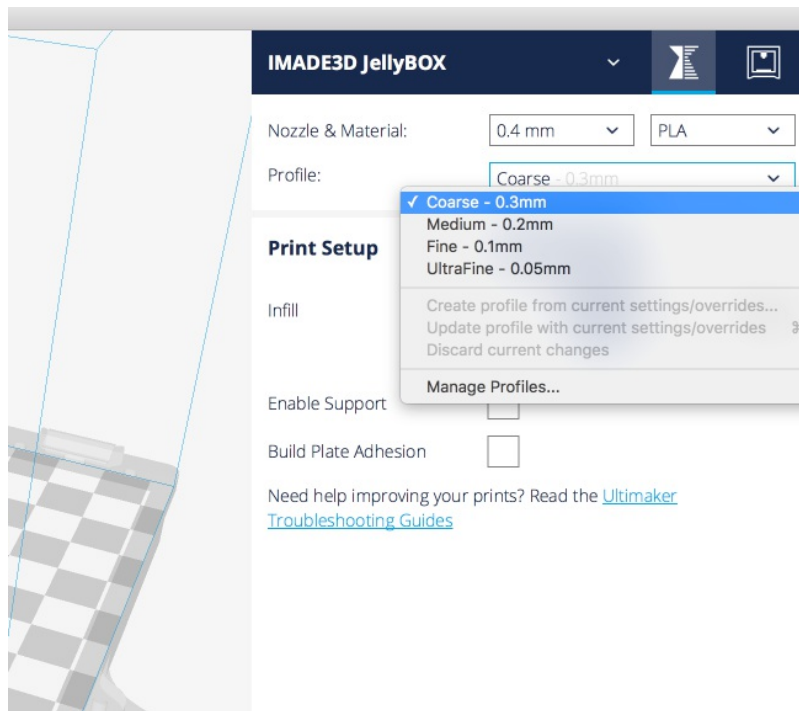
! Alert: Legacy Hotends with 10 mm Heat Block

- 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 probably gonna have under extrusion problems.



Select a Quality Profile

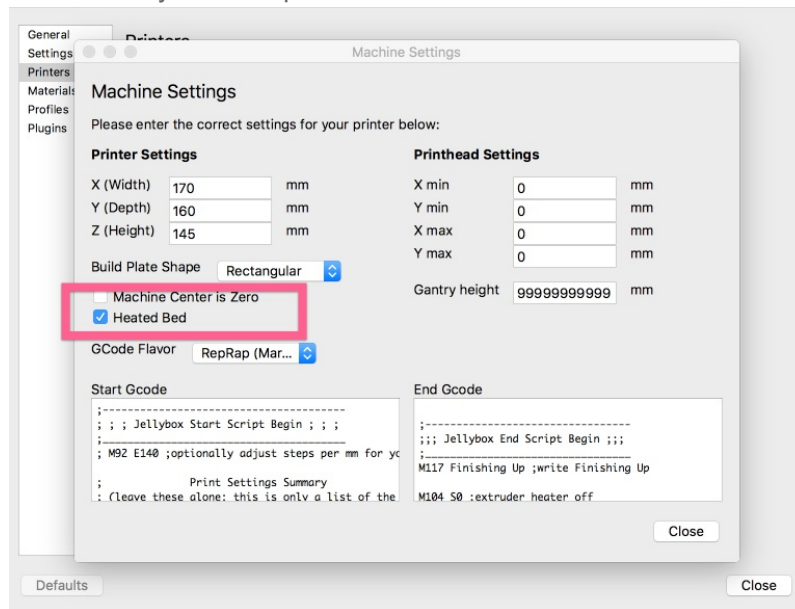
- Coarse
 - Recommended profile for general use. Prints the strongest parts in the least amount of time. Clearly visible 0.3 mm layers.
- Medium
 - Prints slower and slightly more brittle parts than Coarse. Better overhangs and small features.
- Fine
 - Prints slower and more brittle parts than Coarse. Great for overhangs and small features. Smooth looking. Some people think it looks better.
- UltraFine
 - Takes a crazy long time to print (even days). Curiously smooth prints and amazing overhangs.



Heated Bed

- For PETG, we highly recommend heated bed. It's possible to print PETG on cold bed, but warping can be an issue.
- In general, you don't need a heated bed for PLA. 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.
- You can **enable heated bed** in the printer configuration. (It is enabled by default.)

! Cold bed JellyBOX ignores heated bed instructions. So, you **can** run gcodes with heated bed settings on a cold bed JellyBOX. No problem.



- This will allow you to see the heated bed settings as 'Build Plate Temperature' in the 'Custom' tab under 'Material'



- Tip: See the "i" ? If you click on it, Cura will show you additional slicing settings that are explicitly defined as a part of the profile you have selected (1st layer build plate temperature for example.)

