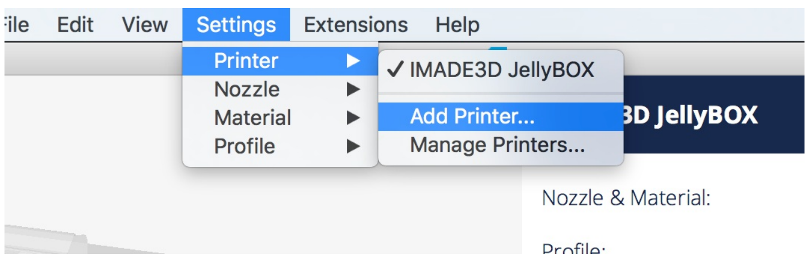
Using 'Cura'/ 'Cura IMADE3D Edition' / 'Ultimaker Cura' with IMADE3D JellyBOX and JellyBOX 2

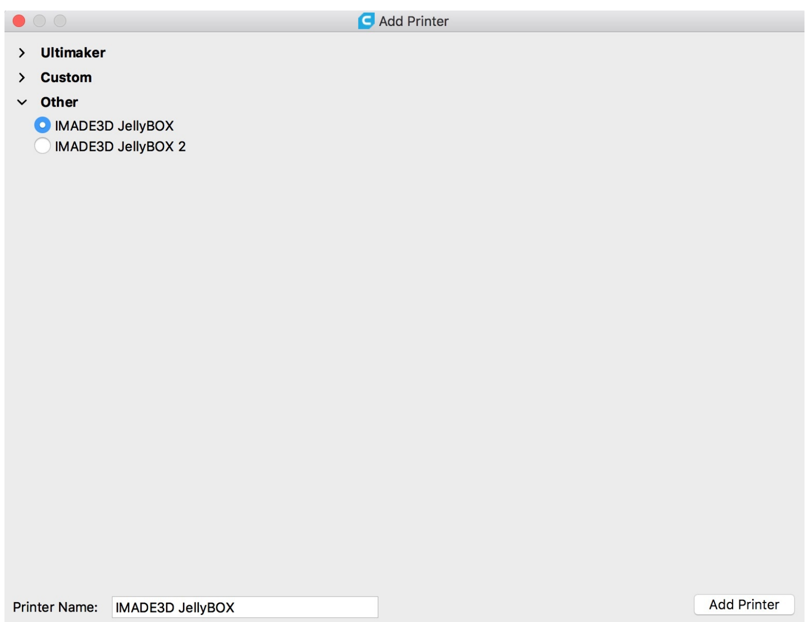
Note: Cura on your computer may look a bit different depending on your system and Cura version. Yet, the procedure will be the same.

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

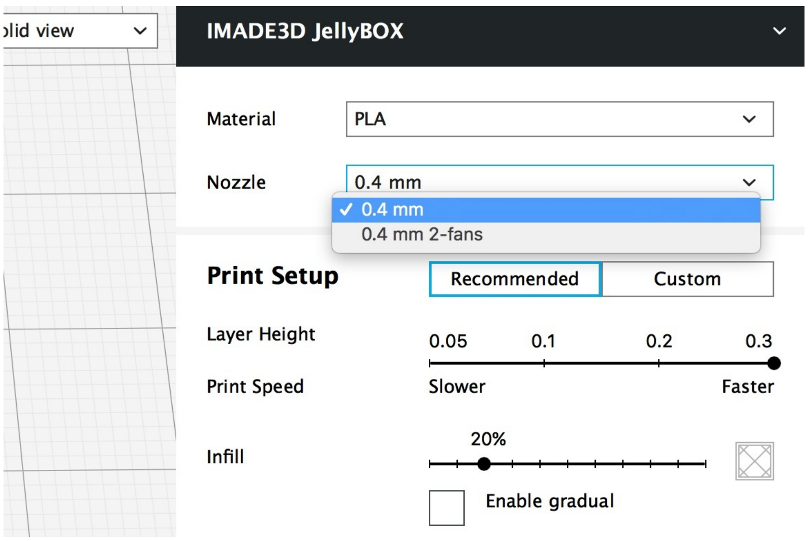
- Settings > Printer > Other > IMADE3D JellyBOX
- (When you start Cura for the first time, you will be presented with the 'add Printer' dialog automatically.)





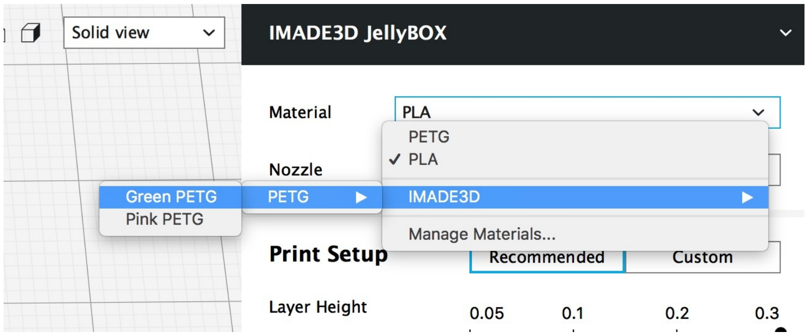
2. Select Nozzle (= JellyBOX Variants)

- 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.



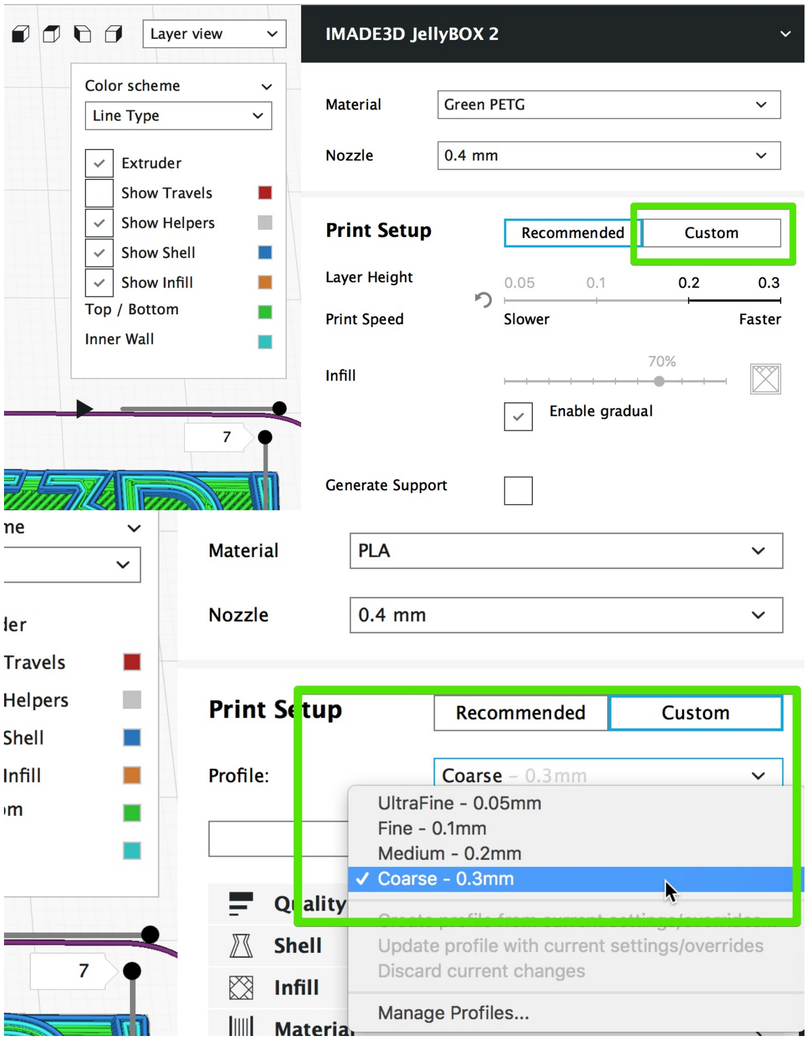
3. Select a Material Profile

- 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.



4. Select a Quality Profile

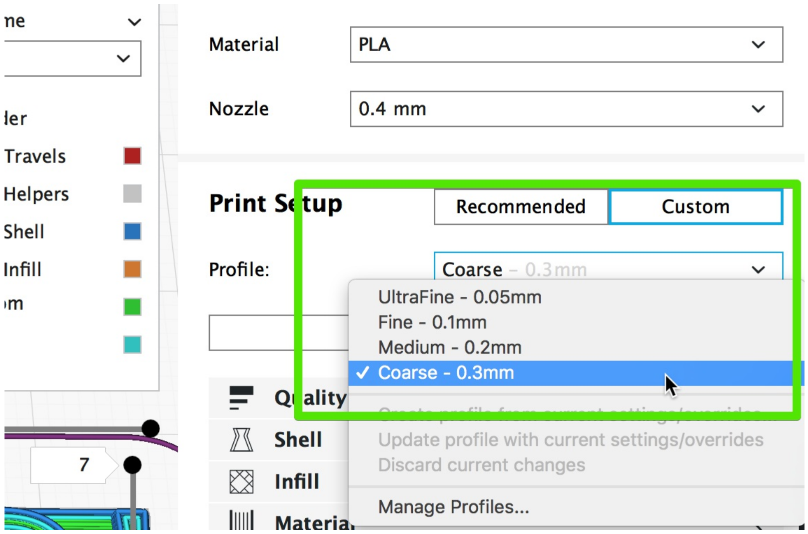
To switch between quality profiles, go to the Custom tab and use the drop down menu



- 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

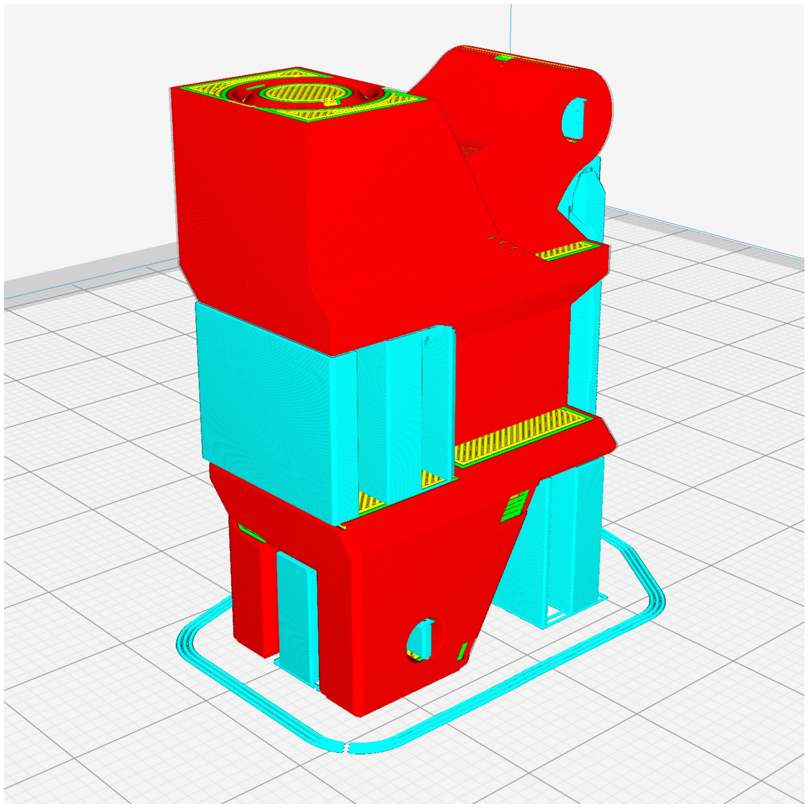
- Prints slower and slightly more brittle parts than Medium. Great for printing steep overhangs and small features. Smooth-looking.

- UltraFine
  - 0.05mm layer height
  - Takes a very long time to print (even days). Produces curiously smooth prints and amazing overhangs.

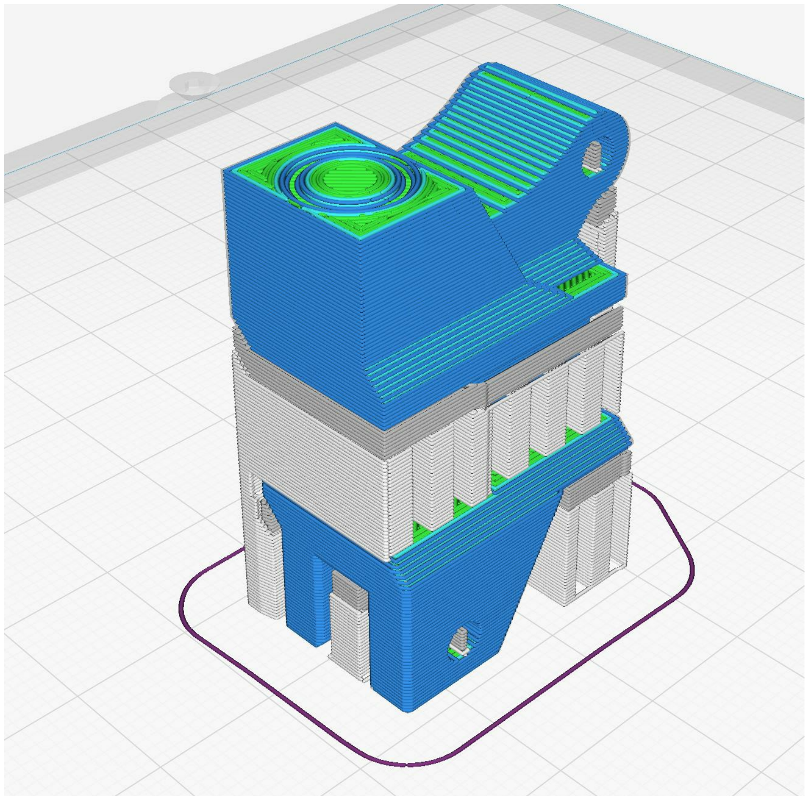


5. Pleasant IMADE3D Color Theme

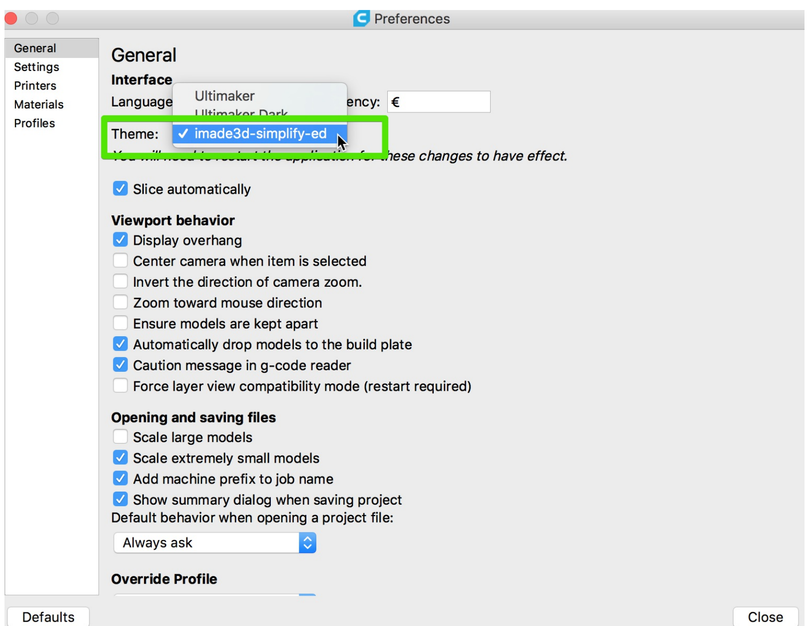
- The default Cura theme has quite jarring colors that threaten to burn your retinas.



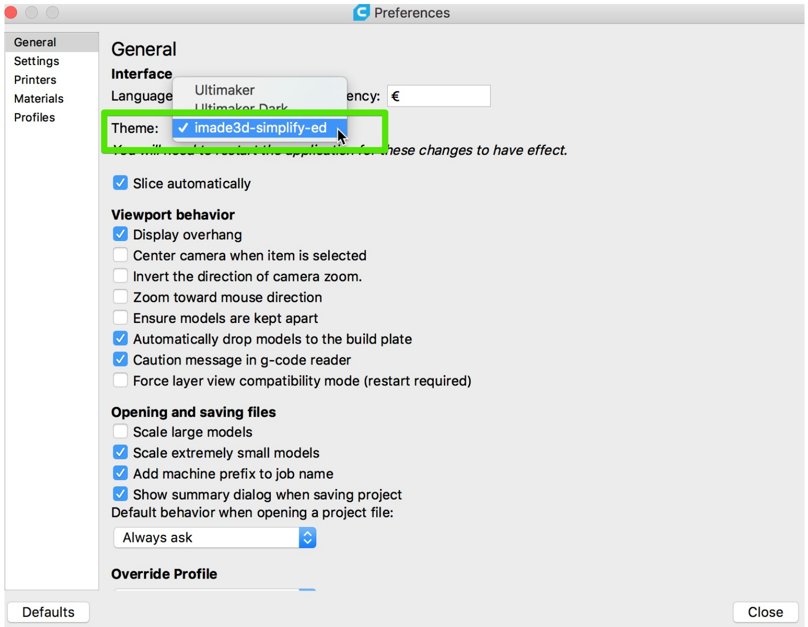
- Use our pleasant simplify-ed theme and feel like a human again.



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

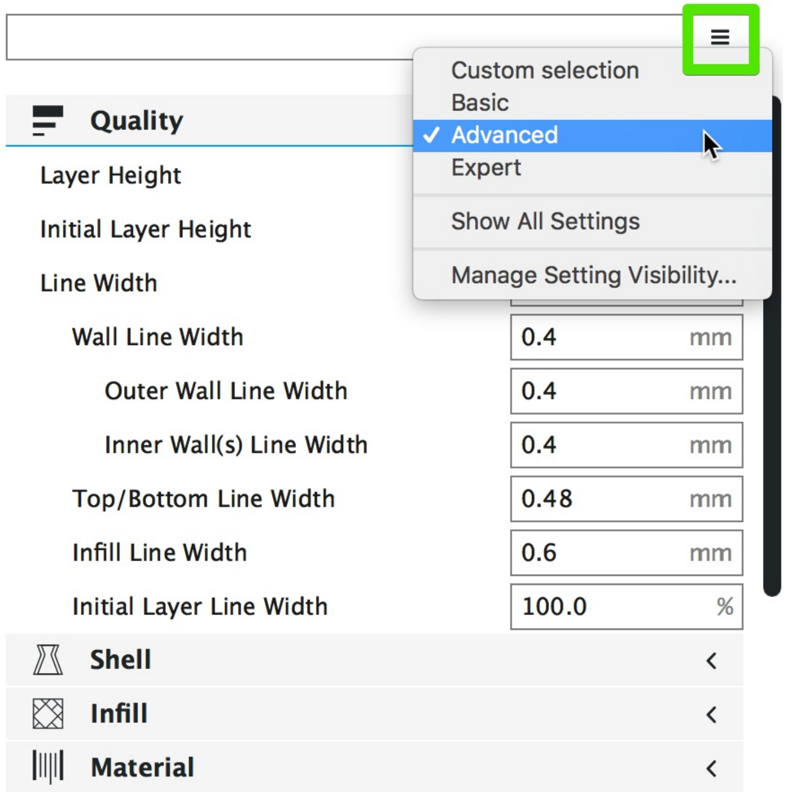


- PC
  - Go to and select the simplify-ed theme

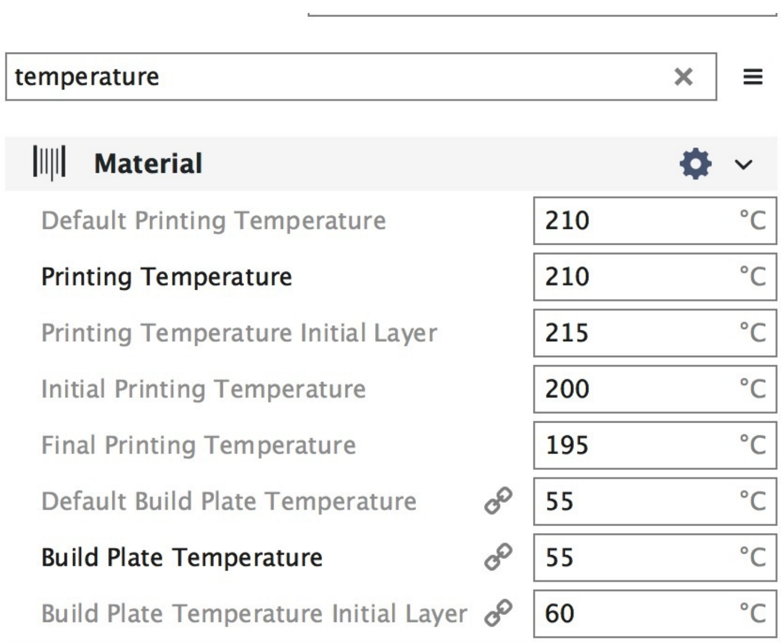


5. Difference Betwen Recommended and Custom tabs

- 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.

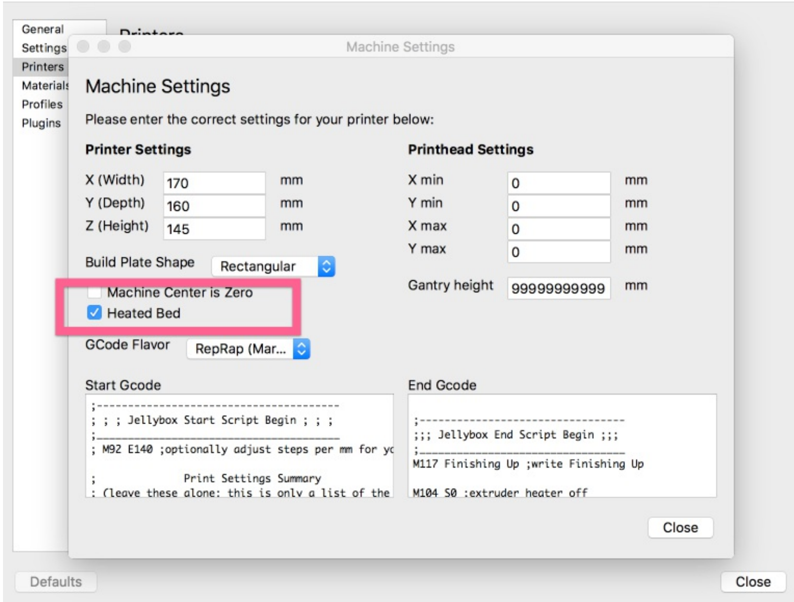


- You can also search for specific setting if you know what you want to tweak!



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.
>
> 
>

- 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.