

# Welcome Note

Dear Jellybox builders and owners. Getting your 3D printer up and running is just the first step in getting awesome prints. This document is a work in progress pointing you in the right direction to get satisfying 3D printing experience. As always, DO get in touch when you need a hand.

Permalink to this document; bookmarking recommended, share with whomever you wish.

<http://bit.ly/imade3d-jellybox-resources>

Note: Everyone has COMMENTING AND EDITING rights to this document. If you have questions, suggestions, or if there is a mistake or an omission, you can directly comment. Or, if you are confident, change as you wish. Changes are reversible if you make a mistake... Oh, and feel free to share with anyone.

~Filip

## This document changelog:

- 2017, 2018 update:
  - **This document has been abandoned for years!** Just keeping it here for posterity.
  - Please review the new resources (featuring for example [forum.imade3.com](http://forum.imade3.com)) at <https://www.imade3d.com/support/>
- Dec 5, 2016 update:
  - Updated Cura 2.3 start and end gcodes. They are awesome now!
- July 22nd, 2016 update:
  - I deleted a lot of things here because they are better organized now in the more refined **Jellybox Post Assembly Guide** available here: <http://bit.ly/jellybox-docs>
  - Added a major section "Jellybox Changelog" to help you see the development, and identify that some guides and notes are not for all versions...
  - Added this document changelog :-)
  - Updated PID values for newest hotend with 20mm melt zone (they were here, but unlabeled)
- July 19th, 2016 update
  - Added a major section - FAQ
- August 30, 2016
  - Minor cleanup, added some infographics under Good Resources
  - Rehailed and cleaned up start and end gcode scripts

## Table of Contents

(click to jump to a section)

[FAQ \(actual questions asked of Filip/imade3d by email\)](#)

[I also saw bubbles coming out of my nozzle... \(asks Matt\)](#)

## [Jellybox Changelog](#)

[Jellybox 1.0](#)

[Jellybox 1.1](#)

[Jellybox 1.2](#)

[Jellybox 1.3](#)

[Build Photos \(Add your own if you have any please. Just drag and drop.\)](#)

[2016](#)

## [Good Resources](#)

[Slic3r Manual](#)

[\(Slic3r Github\)](#)

[Astroprint Ultimate Cheat Sheet](#)

[3D Hubs: What is 3D Printing](#)

[Simplify3D Troubleshooting Guide](#)

[3DVerkstan Visual Ultimaker Troubleshooting Guide](#)

[Tom's Youtube Channel](#)

[Marlin's Gcode Reference](#)

[RepRap Gcode Reference and WIKI](#)

## [Jellybox DIY Build Instructions \(slides\)](#)

### [Slicing](#)

[Cura 2.x Slicing Profiles \(.curaprofile files\)](#)

[Slic3r Slicing Profiles \(Config; .ini file\)](#)

[Simplify3D Slicing Profiles \(.fff files\)](#)

[Making your own slicing profiles](#)

[\(Jellybox Start and End Gcode Scripts for slicers\) \(This is advanced stuff\)](#)

[Start for Cura 2.3+](#)

[End for Cura 2.3+](#)

[Start slic3r](#)

[End slic3r](#)

[Cura 15.x](#)

[Start Simplify3D](#)

[Start for cura 2+](#)

### [Slicers](#)

[Slic3r](#)

[Cura up to 15.x](#)

[Cura 2.1.2+ - the one we use in workshops now](#)

[Simplify3D](#)

[Craftware](#)

[Matter Control](#)

### [Gcode Visualizing](#)

[Gcode Analyzer](#)

[Upgrading Firmware \(Flashing Marlin\)](#)

[Getting the 1st layer height right](#)

[X Offset Troubleshooting - Proximity Sensor off the build plate](#)

[Extruder Calibration](#)

[Extrusion Calibration Guides online and videos online](#)

[Hotend \(Heater PID\) Calibration](#)

[Default PID for E3D Lite6 with 10mm meltzone](#)

[Default PID for E3D Lite6 with 20mm meltzone](#)

[Office Hours](#)

[Office Hours I - 05/17/16](#)

[Interesting times](#)

[Pre-written answers](#)

[Office Hours II - 05/19/16](#)

[Office Hours III - 05/24/16](#)

[Default PID](#)

---

## FAQ (actual questions asked of Filip/imade3d by email)

### 1. I also saw bubbles coming out of my nozzle... (asks Matt)

- a. It's moisture in the filament. One of the many things we have no time to cover in class. The trouble is that PLA, PETG, ABS, Nylon (a lot) and many other thermoplastics absorb moisture from the air. This water vaporizes in the melt zone, and shows up as a bubble of gas when leaving the nozzle. This leads to a variety of issues from bad quality surface finish to weaker prints.

Here is a good article on how to store filament in a dry environment to prevent this problem.

<http://www.3dprintinggeeks.com/3d-printer-filament-storage/>

Personally, I believe that a rather basic solution is enough for most people: simply store your filament in an airtight bag with some silica gel when you're not printing. All filament spools come with a small bag of silica gel, but you can get silica gel beads on Amazon pretty cheap these days. Get the ones that change color as they absorb moisture. That way you can monitor the state of the silica gel and revive it in a conventional oven once in awhile.

PS: this is a DIY industrial grade solution:

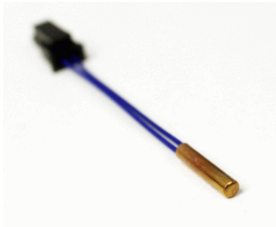
<https://3dprint.com/68083/airtight-filament-delivery/>

# Jellybox Changelog

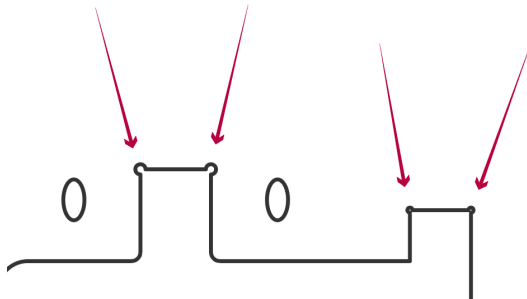
Doing my best to enumerate the major changes. Small tweaks are sometimes introduced unnoticed, and some feature roll out continuously rather than by major version numbers...

## Jellybox 1.0

- The basic public version.
  - Unless you met Filip and Ladi early on and wanted JB as soon as possible (in which case you have unmarked 0.9 version), you have JB 1.0. You can verify this by inspecting your acrylic parts. Most of them have little V 1.0 etched somewhere in the corner.
- Anything before JB 1.0 is considered continuous development, and there's no telling what and when was around.
- Hotend is all genuine E3D Lite6 with the newest thermistor cartridge system (before it was officially publicly available. Living on the edge here).



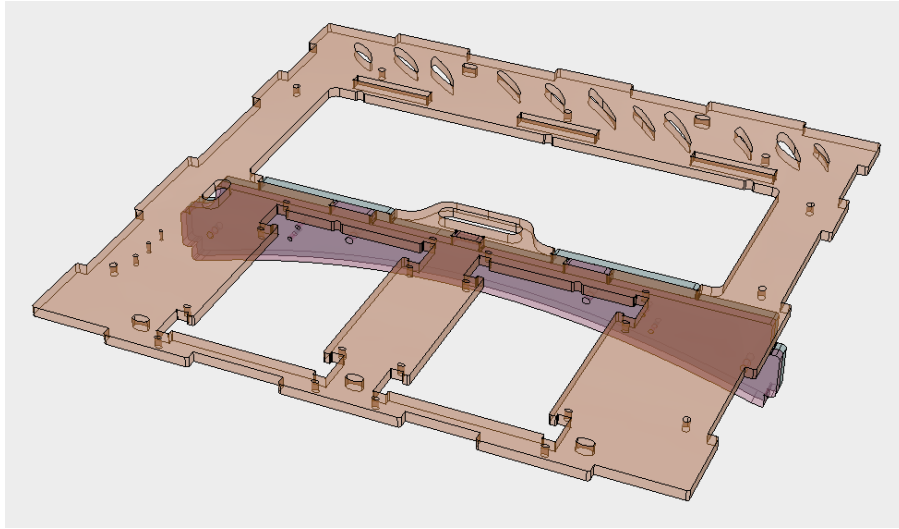
- 
- The bed assembly tensioning system has been re-done and greatly simplified. Inspired by a single crazy remark at a coffee with Conor. One of those moments you say something crazy and it turns out to be the solution...
- Relief radii in all corners of cuts in acrylic. Makes the parts less likely to crack by distributing the stresses.



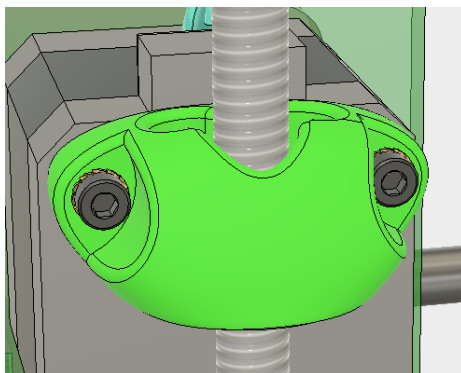
- 
- The power wires gain a new connector so that they don't have to be threaded through the power connector hole in the right acrylic painfully one by one. (A bit cryptic this one. Believe me. This change makes everyone happier.)

## Jellybox 1.1

- Proper part numbering: all acrylic parts now have "V 1.1" etched on them.
- The acrylic brace is doubled now and there is only one filament guide on top. This is a major improvement! It makes the printer more comfortable to carry on either side hashtag ambidextrous; everything is more rigid and less likely to break, and cooler looking.



- 
- We upgraded the hotend by adding our custom developed and custom manufactured heat block to genuine E3D Lite6 body with genuine E3D volcano style nozzle (long). This allows for:
  - Printing at lower temperature, which results in less stringing
  - Printing at higher speeds. E3D Lite6 is a great default hotend because it's reliable, easy to fix, robust, and nearly indestructible thanks to it's seamless stainless body. However, the steel has a worse heat transfer than aluminum (found in the original E3D V6) and the inner PTFE makes things even worse; long story short: E3D Lite6 officially is capable of printing at only 50% speed of the original E3D V6 - and even for that it needs to be some 10C hotter. Phew. We didn't like that. So, we essentially took the great E3D Volcano concept of longer meltzone, and combined it with Lite6 to cancel out its shortcomings. And it works great! So great!
  - Printing with thicker nozzle sizes (0.8, 1.0, 1.2) for reasons outlined above.
  - Easily wrapping the heatblock in silicone strip. This makes the hotend less susceptible to air flow induced fluctuations and leads to more even extrusion especially when running nozzle fans at 100% for PLA or bridges. Plus, you won't get burnt when you touch the heat block. Not that you should.
- We go back to 3D printed ZX coupler. The lasercut acrylic zip-tied ZX coupler used in JB 1.0 is cool both as a concept and visually, but it was difficult and time consuming to put together (and explain how to put it together). Trouble with novel solutions is that... they are novel, and therefore confusing. Making a custom 3D printed part makes all of this go away. Simply add two screws. Done. No explanation necessary.



- 
- The top has a neat 'carry here' and 'don't carry here' etching.
- The spool rollers have been widened by a few millimeters to accommodate Atomic Filament spools. Thanks to Joe for the suggestion! Atomic filaments are great.

- Switched to transparent zip ties for wiring. They're easier to use, stronger, and look arguably better. Also cheaper and easier to find if you need to source your own.
- Wires completely redone from scratch. Color coded ribbons look great and are joy to see.
- 3D printed tiny spacers for Arduino and for the Full Graphic Controller are replaced with lasercut ones that incorporate all the spacers into one. Believe me, it's much much easier and faster to put together.
- Bolts holding the LCD: switched from countersunk cap screws to countersunk flat heads. It's pretty obvious why. Flat heads are actually made to be countersunk; unlike cap screws.
- Trio of back panel holding discs transformed from circle to rounded hexagon. Functionally the same, but easier to handle, easier to tap the holes, and it's screaming IMADE3D.

## Jellybox 1.2

- The Y motor bracket is symmetrical. No more confusion during assembly which side is up and which side is down. Thanks to Bo from build group 4 (I think) for suggesting this.
- We removed a bunch of zip ties. The number of zip ties we used was definitely an overkill before. The JellyBox is faster to put together now, and there's less waste.
- Zip tie color coding. The big zip ties ('intermediate' designation) remain pink, small ones are now exclusively transparent ('miniature' designation), and the smallest ones remain black. No more confusion! Plus, the Jellybox actually looks more refined and less crowded.
- Wires completely redone from scratch, again. Ribbons were nice, but too costly and wasteful to manufacture, and expensive and a pain to fix. We go back to individual wires this time, but we developed a new and truly easy way to install the wires.

## Jellybox 1.3

- We got nozzle fans on both sides of the nozzle! This is made possible by switching to our own symmetrical heat block on the hot end and JellyBox version 1.1, and also by switching to a completely new and ridiculously cool kind of inductive proximity sensor. It's made by our favorite sense of manufacturer Aotoro, it's only 10 mm thick, and it's flat rather than tall.
  - We wanted to have nozzle fans on both sides of the hot end for the longest time. However, until we modified the commercially available hot end, and found and tested this new kind of proximity sensor, it simply wasn't viable. Now it's here.
  - This is a major improvement for print quality of various parts, especially if you printing from PLA. Before, sometimes you can clearly see that the left side of the printer was producing a better print quality and the right side. It's a common cooling issue that many 3-D printers have.
- We removed a bunch of zip ties going from 49 big pink 6" zip ties used in the frame to 37. The number of zip ties we used was definitely an overkill before. The JellyBox is faster to put together now, and there's less waste. We thought hard once more what zip ties to keep so then the function and the aesthetics are not compromised while the printer is faster to put together and waste less material.
- We also removed a bunch of smaller 4" zip ties. Same reasoning. Trimming fat.
- Y bracket lost its zip ties. The zip ties were useless - the bracket is held in place with the pressure from the y belt.
- Y bracket is symmetrical. Originally, I made the Y bracket asymmetrical, so that it could only be put in place the correct way. However, I think it was Bo who remarked that if the bracket was symmetrical, there would be no incorrect way. And he was right. This feature is probably not needed

anymore as we also added "y motor adjoins right here to this side" etching, but the logic is solid so why not make the design one layer more bullet proof.

- Fancy new "wire comb" to hold the wires above the RAMPS boards. Faster and easier wire installation here we come! Plus, less zip ties. (This is the first step in making the whole wiring faster and easier. More to come later.)
- Moved bed clip on front of the bed assembly into the middle. Originally, we had it on the side for easier manipulation, but reality showed it makes no difference and the middle is overall a better mounting point for stress distribution.
- X assembly front now has "front" etching and back has a "back" etching, just like front and back brace pieces. Because why not. It may help someone at some point.
- There's a new "start here" etching on the bottom piece drawing attention to the beginning of the popular squiggly line.
- The Arduino power source "+" and "-" etchings on the inner back were spatially correct, but invisible once the Arjuna was installed making them useless. We shifted the etching so that the plus and minus annotations regarding power source wires are easily visible even when Arduino is installed - which is really when you need it. It
- Packing tweak: we changed or rather codified the order in which we heat shrink the acrylic pieces together. they now come out in the order in which you need them for the build. Simple, I know, we just didn't think of it before. One step at a time.
- Added a 3rd zip tie to the `_strictly optional_` side covers. This prevents them from rotating and trapping tiny hands. (Thanks Joe!)
- Increased tolerance for the top rod in X assembly. Makes up for easier, more robust build experience.
- Added a few 'highlighting' etchings around bearing slots on X and Y. (Also for smooth rods on X). This should make the slots easier to see and lead to less people not aligning the bearings properly (which is rare, but hey. Perfection comes in small increments.)
- The 'top' acrylic piece has one if it's tabs 1mm too short. Hardly noticeable and made no difference for the function. Still, it bothered us so it's back to precision.
- The QR C clamp on top of X-carriage has slightly different shape again. This time to help hold the extruder assembly with more force. Just pushing millimeters around.
- Front LCD spacer didn't have the right cutout for the knob. In extreme cases, these could run against each other and make the knob turning into a true workout. (Simple fix was to pull the knob a bit out with pliers.) All good now.
- We are ditching the annoying hard to handle small black zip ties for X and Y end stops! Trouble with those is they are so small that they are hard to source in the US - which makes getting spare parts without us difficult. And we don't want you to have to rely on the kindness of one company even if it's us. Instead, we have M2 buttonhead bolts and nuts.
- The wire etchings around Z probe on inner back have been revised.
- Y Assembly (Bed assembly) Y belt attachment system has been tweaked and finalized. It looks very similar, but the procedure of attaching the belt is now faster and simpler than ever. Incremental, but UX wise major tweak.

**Build Photos (Add your own if you have any please. Just drag and drop.)**

## 2016

May 14th Build <https://goo.gl/photos/ai3CrFCTd2z9SFhZ9>

May 21st Build <https://goo.gl/photos/7jwDQEG991dydx5q9>

July 27th Build <https://goo.gl/photos/dzkuCvt8f8ewsdEGA>

July 9th Build GET THE LINK HERE

Sep 10th Build <https://goo.gl/photos/VTcibb8z84ZzoQez8>

Sep 10th Build 360 photo (!) thanks to Shawn <https://theta360.com/s/bxXap7wExHpn2qhP54ytXb7RI>

## Good Resources

### Slic3r Manual

<http://manual.slic3r.org/>

This manual is a truly a great introduction to all the intricacies of slicer settings. Even though it's Slic3r manual, the concepts and variables are universal across a wide range of slicing programs.

### (Slic3r Github)

<https://github.com/alexrj/Slic3r/wiki/FAQ>

Has good pointers for power users. Command line options etc. Not useful for most people.

### Astroprint Ultimate Cheat Sheet

<https://blog.astroprint.com/the-ultimate-3d-printing-cheat-sheet/>

Not much of an overstatement. It really is a great aggregation of links. We may not agree with some of the rankings or have different preferences, but it's a great all-in-one-place hub of links.

### 3D Hubs: What is 3D Printing

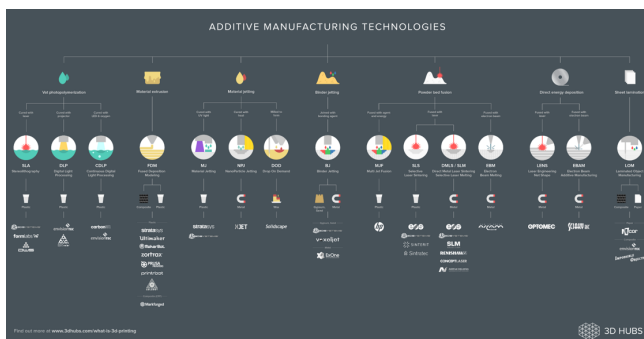
<https://www.3dhubs.com/what-is-3d-printing>

Very nice overview of various 3D printing techniques with applications, pros, cons, and photos. Good production value 3d hubs!

Update Aug 2016: Now includes this uber-cool infographic

<https://www.3dhubs.com/sites/all/themes/huographic--x-large.png>

<ring-infbs3d/images/landingpage/3d-101/technologies/additive-manufactu>



### Simplify3D Troubleshooting Guide

<https://www.simplify3d.com/support/print-quality-troubleshooting/>

A pictorial reference to fixing common quality issues with prints. Some solutions are S3D specific, but the diagnosis is useful.



## 3DVerkstan Visual Ultimaker Troubleshooting Guide

<http://support.3dverkstan.se/article/23-a-visual-ultimaker-troubleshooting-guide>

Many solutions are Ultimaker specific, but the diagnosis can be useful.

## Tom's Youtube Channel

<https://www.youtube.com/user/ThomasSanladerer>

Filip loves Tom. Down to earth, honest, succinct, and... basically always right. Tom has been an inspiration and a resource from our very beginnings.

## Marlin's Gcode Reference

<https://github.com/MarlinFirmware/MarlinDev/wiki/G-Code-in-Marlin>

If the explanation is there, it's a superb one. But most explanations are not there... this is one reference full of holes so far.

## RepRap Gcode Reference and WIKI

<http://reprap.org/wiki/G-code>

Very comprehensive gcode reference. Search it anytime you don't recall a gcode.

The rest of the wiki is a premier stop for most things ever related to open hardware 3d printers and printing. Marlin, hardware, software, slicing, gcode... everything. Search the wiki every time you don't know something. Beware! Some of the knowledge is arcane. While exhilarating to see years old posts of people developing calibration procedures for the first time, what you read is not always up to date or even correct.

## Jellybox DIY Build Instructions (slides)

Good if you need to refresh any part of the build. PowerPoint files with animations etc; pdf files with static images. <http://imade3d.com/support>

## Slicing

Slicers are programs that process 3D files (.stl and more) into 3D printing instructions (.gcode). To get the best quality it's worth tweaking slicing settings according to the model and according to what kind of effect you are looking for. Presets are a great starting point though!

## Cura 2.x Slicing Profiles (.curaprofile files)

To be use in conjunction with "clean slate" material selected in Cura.

- 0.1 mm / 0.2 mm / 0.3 mm [link to folder](#)
- Here's a video on how to import the profiles into Cura 2.x
- Note: at [www.bit.ly/jellybox-docs](http://www.bit.ly/jellybox-docs) you'll find a guide on how to get JellyBox printer into Cura (it's NOT self evident)

## Slic3r Slicing Profiles (Config; .ini file)

- 0.3 mm layer for PLA in Slic3r [JB 0.3 PLA Slic3r](#)

In Slic3r, you have to be in Expert mode to be able to use presets (which makes no sense to me, but that's what it is). This is how you do that:

1. File > Preferences > Mode > Expert
2. Restart Slic3r
3. Now you can load and save custom presets (slic3r calls these **configurations**)
4. Load configuration with File > Load Config...  
Or keyboard shortcut: Ctrl + L (PC) cmd + L (Mac)

### Simplify3D Slicing Profiles (.fff files)

- 0.3 mm layer for PLA in Simplify3D [JB 0.3 PLA S3D Lite6-10mm](#)
- 0.2 mm layer for PLA in Simplify3D [JB 0.2 PLA S3D Lite6-10mm](#)
- 0.1 mm layer for PLA in Simplify3D [JB 0.1 PLA S3D Lite6-10mm](#)

### Making your own slicing profiles

(Jellybox Start and End Gcode Scripts for slicers) (This is advanced stuff)

You can use any slicer you want and any profile you want, but you must include a "G28" and "G29" code in the starting gcode settings of the slicer! Without this, Jellybox cannot function properly!

Easiest way to go is to use one of our profiles as a starting point. If you use Imade3D slicing profiles, they all have the appropriate starting and ending gcodes included.

Here are the Start and End scripts that we use. Much of the code simply improves usability.

<u>Start for Cura 2.3+</u> To be used within the program's machine manager	<u>End for Cura 2.3+</u> To be used within the program's machine manager
<pre>----- - ; ; ; Jellybox Start Script Begin ; ; ; ----- - ; M92 E140 ;optionally adjust steps per mm for your filament  ;                               Print Settings Summary</pre>	<pre>----- ;;; Jellybox End Script Begin ;;; ----- M117 Finishing Up ;write Finishing Up  M104 S0 ;extruder heater off M140 S0 ;bed heater off (if you have it) G91 ;relative positioning G1 E-1 F300 ;retract the filament a</pre>

```

; (overwriting these values will NOT
change your printer's behavior)
;         sliced for:
{machine_name}
;         nozzle diameter:
{machine_nozzle_size}
;         filament diameter:
{material_diameter}
;         layer height:
{layer_height}
;         1st layer height:
{layer_height_0}
;         line width:
{line_width}
;         wall thickness:
{wall_thickness}
;         infill density:
{infill_sparse_density}
;         infill pattern:
{infill_pattern}
;         print temperature:
{material_print_temperature}
;         heated bed temperature:
{material_bed_temperature}
;         regular fan speed:
{cool_fan_speed_min}
;         max fan speed:
{cool_fan_speed_max}
;         support?
{support_enable}
;         spiralized?
{magic_spiralize}

M117 Preparing          ;write Preparing
M140 S{material_bed_temperature}
;set bed temperature and move on
M104 S{material_print_temperature}
;set extruder temperature and move on
M206 X10.0 Y0.0        ;set x homing
offset for default bed leveling
G21                    ;metric values
G90                    ;absolute
positioning
M107                   ;start with the
fan off
M82                    ;set extruder to
absolute mode
G28                    ;home all axes
M203 Z5                ;slow Z speed
down for greater accuracy when probing
G29                    ;auto bed
leveling procedure

```

```

bit before lifting the nozzle, to
release some of the pressure
G1 Z+0.5 E-5 X-20 Y-20 F9000 ;move Z
up a bit and retract filament even
more
G28 X0 ;home x, so the head is out
of the way
G1 Y160 ; bring y forward to have better
access
M84 ;steppers off
G90 ;absolute positioning

M117 Print finished ;write Print
finished

-----
---
;;; Jellybox Printer End Script End
;;;
-----

```

<pre> M203 Z7                ;pick up z speed again for printing M190 S{material_bed_temperature} ;wait for the bed to reach desired temperature M109 S{material_print_temperature} ;wait for the extruder to reach desired temperature G92 E0                 ;reset the extruder position G1 F200 E5            ;extrude 5mm of feed stock G92 E0                 ;reset the extruder position again M117 Print starting   ;write Print starting ----- ----- ; ; ; Jellybox Printer Start Script End ; ; ; ----- </pre>	
<p><u>Start slic3r</u></p>	<p><u>End slic3r</u></p>
<pre> ;Jellybox Start Script Begin ;write "Preparing" ; Jellybox Start Script Begin M117 Preparing  ; Start heating the bed, but don't wait around ; M140 S[first_layer_bed_temperature] ; Start heating the nozzle, but don't wait around M104 S[first_layer_temperature]  ; set x homing offset for default bed leveling M206 X11.0 Y0.0 ; optionally adjust steps per mm for your filament ; M92 E140  ; home all axes G28 ; slow Z speed down for greater accuracy M203 Z5 ; auto bed leveling procedure G29 ; pick up z speed again for print M203 Z7 ; reset extruder position G92 E0 </pre>	<pre> ;Jellybox Printer End Script Start ; extruder heater off M104 S0 ; bed heater off (if you have it) M140 S0 ; turn fan off M107  ; relative positioning G91 ; retract the filament a bit before lifting the nozzle to release some of the pressure G1 E-1.5 F1200 ; move Z up a bit and retract filament even more G1 Z+1 E-1 X-20 Y-20 ;move X/Y to so the head is out of the way G28 X0 G1 Y100 ; steppers off M84 ; absolute positioning G90  ; Jellybox Printer End Script End </pre>

<pre> ; Wait till bed temperature is reached ; M190 S[first_layer_bed_temperature] Wait till the nozzle temperature is reached M109 S[first_layer_temperature]  M117 Print starting ;Jellybox Printer Start Script End </pre>	
<p><u>Cura 15.x</u></p>	
<pre> ;Sliced at: {day} {date} {time} ;Basic settings: Layer height: {layer_height} Walls: {wall_thickness} Fill: {fill_density} ;Print time: {print_time} ;Filament used: {filament_amount}m {filament_weight}g ;Filament cost: {filament_cost} M190 S{print_bed_temperature} ;Uncomment to add your own bed temperature line M109 S{print_temperature} ;Uncomment to add your own temperature line ; M92 E140; optionally adjust steps per mm for your filament M117 Preparing ; write "Preparing" ; M206 X10.0 Y0.0; optionally, set x homing offset for default bed leveling G21 ;metric values G90 ;absolute positioning M82 ;set extruder to absolute mode M107 ;start with the fan off G28; home all axes M203 Z5; slow Z speed down for greater accuracy G29; auto bed leveling procedure M203 Z7; pick up z speed again for print G92 E0; reset extruder position ;Put printing message on LCD screen M117 Printing... </pre>	<pre> ;End GCode M104 S0 ;extruder heater off M140 S0 ;heated bed heater off (if you have it)  G91 ;relative positioning G1 E-1 F300 ;retract the filament a bit before lifting the nozzle, to release some of the pressure G1 Z0.5 E-5 X-20 Y-20 F{travel_speed} ;move Z up a bit and retract filament even more G28 X ;move X to min endstops, so the head is out of the way G0 Y140 ; move y forward for better access  M84 ;steppers off G90 ;absolute positioning ;{profile_string} </pre>
<p><u>Start Simplify3D</u></p>	
<pre> ;Jellybox Start Script Begin  M117 Preparing ; write "Preparing"  M104 S{material_print_temperature}; Start heating the nozzle, but don't wait around  M206 X11.0 Y0.0; set x homing offset for default bed leveling ; M92 E140; optionally adjust steps per mm for your filament  G28; home all axes M203 Z5; slow Z speed down for greater </pre>	<pre> ;Jellybox Printer End Script Start  ; extruder heater off M104 S0 ; bed heater off (if you have it) M140 S0 ; turn fan off M107  ; relative positioning G91 ; retract the filament a bit before lifting the nozzle to release some of the pressure </pre>

<pre> accuracy G29; auto bed leveling procedure M203 Z7; pick up z speed again for print G92 E0; reset extruder position  M109 S{material_print_temperature}; Wait till the nozzle temperature is reached  M117 Print starting; write "Preparing" ;Jellybox Printer Start Script End </pre>	<pre> G1 E-1.5 F1200 ; move Z up a bit and retract filament even more G1 Z+1 E-1 X-20 Y-20 ;move X/Y to so the head is out of the way G28 X0 G1 Y100 ; steppers off M84 ; absolute positioning G90  ; Jellybox Printer End Script End </pre>
<p><u>Start for cura 2+</u> (equivalent gcode, different formatting for backend .json file. Definitely only for advanced users)</p>	
<pre> "\n;Jellybox Start Script Begin\n;write "Preparing"\nM117 Preparing\n \n;optionally adjust steps per mm for your filament\n;M92 E140\n;set x homing offset for default bed leveling\nM206 X10.0 Y0.0 \n;metric values\nG21\n;absolute positioning\nG90\n;start with the fan off\nM107\n;set extruder to absolute mode\nM82\n\n;home all axes\nG28\n;slow Z speed down for greater accuracy\nM203 Z5\n;auto bed leveling procedure\nG29\n;pick up z speed again for print\nM203 Z7\n;reset the extruder position\nG92 E0\n;extrude 5mm of feed stock\nG1 F200 E5\n;reset the extruder position again\nG92 E0\n \n;write "Print starting"\nM117 Print starting\n;Jellybox Printer Start Script End\n" </pre>	

## Slicers

### Slic3r

Open Source. Free. <http://slic3r.org/>

Slic3r is Feature rich. Has a learning curve, but also a good profile management - both slicing profiles and material (filament) profiles.

### Cura up to 15.x

Open Source. Free. <https://ultimaker.com/en/products/cura-software>

Easier user interface than Slic3r. Arguably faster prints than Slic3r. However, less control and less features than Slic3r. One good trick is to get company specific Cura versions (Lulzbot, Type-A Machines), which

contain tons of material and quality presets to be inspired by. Note that you cannot just use these presets and expect good results though - presets are printer specific.

### **Cura 2.1.2+ - the one we use in workshops now**

Open Source. Free. <https://ultimaker.com/en/products/cura-software>

It's completely rebuilt from scratch. Has little to do with the old cura, interface and even features wise. Downside; does not work out of the box - needs a bit of tweaking during installation.

*The instructions for how to set up Cura 2+ and slice with it have moved to the Jellybox Post Assembly Guide Draft at <http://bit.ly/Jellybox-OneNote-Guides>*

### **Simplify3D**

Commercial. \$140 <http://simplify3d.com/>

Arguably the best slicer available today. Class leading supports that are easy to remove, and can be manually tweaked. Superior gcode visualization and print simulation. Fast and robust. Simple interface with time saving features. Custom format that saves 'project files' bundling the 3D objects with desired slicing settings. And much more. Hiccups: badly designed profile management; filament and slicing profiles are confusingly mixed. Also, expensive. Full disclosure: Imade3D does use Simplify3D for most of its slicing needs.

### **Craftware**

Commercial. Free.

Not ready for prime time, but keep an eye on this one. Getting better. Great visualization and preview of gcode. Good interface. However, not that robust, the quality of the prints not on par with competition it seems.

### **Matter Control**

I haven't used it enough to have an opinion.

## **Gcode Visualizing**

### **Gcode Analyzer**

<http://gcode.ws/>

This is one fine browser base gcode analyzer. The 3D view is pretty bad, but the 2D view is very good. Shows movement speeds color coded by line and has good estimates for time. I use it to compare code from different slicers as in how they treat bridges etc.

## **Upgrading Firmware (Flashing Marlin)**

*These instructions have moved to the Jellybox Post Assembly Guide Draft at <http://bit.ly/Jellybox-OneNote-Guides>*

## Getting the 1st layer height right

The instructions for how to set up Cura 2+ and slice with it have moved to the Jellybox Post Assembly Guide Draft at <http://bit.ly/Jellybox-OneNote-Guides>

## X Offset Troubleshooting - Proximity Sensor off the build plate

The magnetic proximity sensor moves off of the bed (metal build plate) and is unable to detect the bed below leading the extruder to crash into the build plate.

Troubleshooting:

1. Go to the root menu of the controller.
2. Scroll all the way down, select X Homing Offset
3. Set the X Homing Offset to 10 (or 11 as the default for the older version of the Jellybox. Makes no real difference other than aesthetic)
4. Set the Y Homing Offset to 0 (its default)

## Extruder Calibration

The instructions for how to set up Cura 2+ and slice with it have moved to the Jellybox Post Assembly Guide Draft at <http://bit.ly/Jellybox-OneNote-Guides>

## Extrusion Calibration Guides online and videos online

Our own video won't be ready until a bit later (working on it!) Until then, here's a lineup of guides using the same principle.

E Steps Calibration guide	Comments
Richrap's Guide (blog with photos) <a href="http://richrap.com/?p=54">http://richrap.com/?p=54</a>	Only extrudes 30mm, but the workflow is solid. Same principle.
E3D's guide (wiki with many photos) <a href="http://wiki.e3d-online.com/wiki/Your_First_Print">http://wiki.e3d-online.com/wiki/Your_First_Print</a>	There's some differences in the menu structure etc, but the principle is the same. See the "Calibrating The Filament" section.
Zennmaste (VIDEO and blog) <a href="http://zennmaster.com/makingstuff/reprap-101-calibrating-our-extruder-part-1-e-steps">http://zennmaster.com/makingstuff/reprap-101-calibrating-our-extruder-part-1-e-steps</a>	Fast and easy to follow, good visuals, good basic workflow. A few differences from how we do it, but principle remains.
Triffid Hunter's Calibration Guide (wiki no photos) <a href="http://reprap.org/wiki/Triffid_Hunter%27s_Calibration_Guide">http://reprap.org/wiki/Triffid_Hunter%27s_Calibration_Guide</a>	It's a classic often recommended in RepRap Community. Works fine. Guide is only a description, no images or videos.
Tom's Calibration Guide (VIDEO) <a href="https://youtu.be/YUPfBJz3I6Y?t=122">https://youtu.be/YUPfBJz3I6Y?t=122</a>	A bit long. A few differences from how we do it, but principle remains. Does quite a bit of explaining about the background of these things... which is great for beginners!



	Uses gcode - note that you can do all what he does over the Controller.
--	---

## Hotend (Heater PID) Calibration

The instructions for how to set up Cura 2+ and slice with it have moved to the Jellybox Post Assembly Guide Draft at <http://bit.ly/Jellybox-OneNote-Guides>

### Default PID for E3D Lite6 with 10mm meltzone

Input these using your controller under Settings > Temperature

OR, better, run the Marlin PID Autotune to measure your own values (need to use computer with USB connection and run "M303 S210 C5") and input those.

**Kp 25.80**

**Ki 2.85**

**Kd 58.34**

### Default PID for E3D Lite6 with 20mm meltzone

Newest measurement: July 2016

```
#define DEFAULT_Kp 20.49
```

```
#define DEFAULT_Ki 1.07
```

```
#define DEFAULT_Kd 97.67
```

```
echo: M301 P20.49 I1.07 D97.67 C100.00 L20
```

---

## Office Hours

### Office Hours I - 05/17/16

<https://plus.google.com/events/cqjt032m8irlf4unaplndnud7qko>

### Interesting times

**Flashing arduino** - [https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=268](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=268)

**Explaining the need for G29** -

[https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=551](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=551)

**Loading Slic3r preset and explaining the settings** -

[https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=971](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=971)

**Print over USB, how?** -

[https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=2175](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=2175)

**Simplify3d** - [https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=2540](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=2540)

**Troubleshooting Dan's thermistor** (unfinished?)-

[https://www.youtube.com/watch?feature=player\\_embedded&v=Ch-y9VmVkrS#t=2722](https://www.youtube.com/watch?feature=player_embedded&v=Ch-y9VmVkrS#t=2722)

## Pre-written answers

1. When the printer is idle and the nozzle temperature is still high (for example if I stop a print), melted filament oozes out the bottom until the nozzle cools down. Is this expected? I don't remember it from class, but we had only brief usage. (Daniel)
  - a. Yes. That's normal.
2. I'm noticing that when the printer draws the skirt, and the first layer, more filament appears on the surface on the left side than the right side. It seems as if the surface has a small slope and the height of the nozzle above the surface changes from left-to-right. This wasn't noticeable when we print the cube, but can be observed with larger prints. (Daniel)
  - a. Long story short: G29 is a must! More in the hangout.
3. Printing directly from Slic3r. I thought this should be possible, but can't find a "Print" menu. We connected the printer to the laptop with the USB cable, but I didn't see how to locate the printer from Slic3r. Of course here I should read the Slic3r manual. But let me know if another application should be used instead.
  - a. It's not possible. We HIGHLY recommend printing from SD CARD (or setting up Astroprint/Octoprint with raspberry pi).
  - b. You can use host software like Printron or Repetier Host to print directly from computer.
  - c. <http://www.pronterface.com/>
  - d. <https://www.repetier.com/>

## Office Hours II - 05/19/16

<https://plus.google.com/events/ciib7vnine4f7vk94vprol0tcug>

this time not much has happened: gave a troubleshooting tip about heater cartridge at the beginning... and that was it. rest of the video is just me troubleshooting some Fusion360 modeling issues. ~Filip

NOTE: the tip at the beginning about the heater did help. Daniel fixed his Jellybox.

## Office Hours III - 05/24/16

<https://plus.google.com/events/cfts3lead5tb48n5d1hfa7m3ddc>

I showed how to put custom start and end Gcode into Cura, Slic3r, and Simplify3d. Also how to edit gcode with a text editor.

### Default PID

Input these using your controller under Settings > Temperature  
OR, better, run the Marlin PID Autotune to measure your own values.

**Kp 25.80**

**Ki 2.85**

**Kd 58.34**