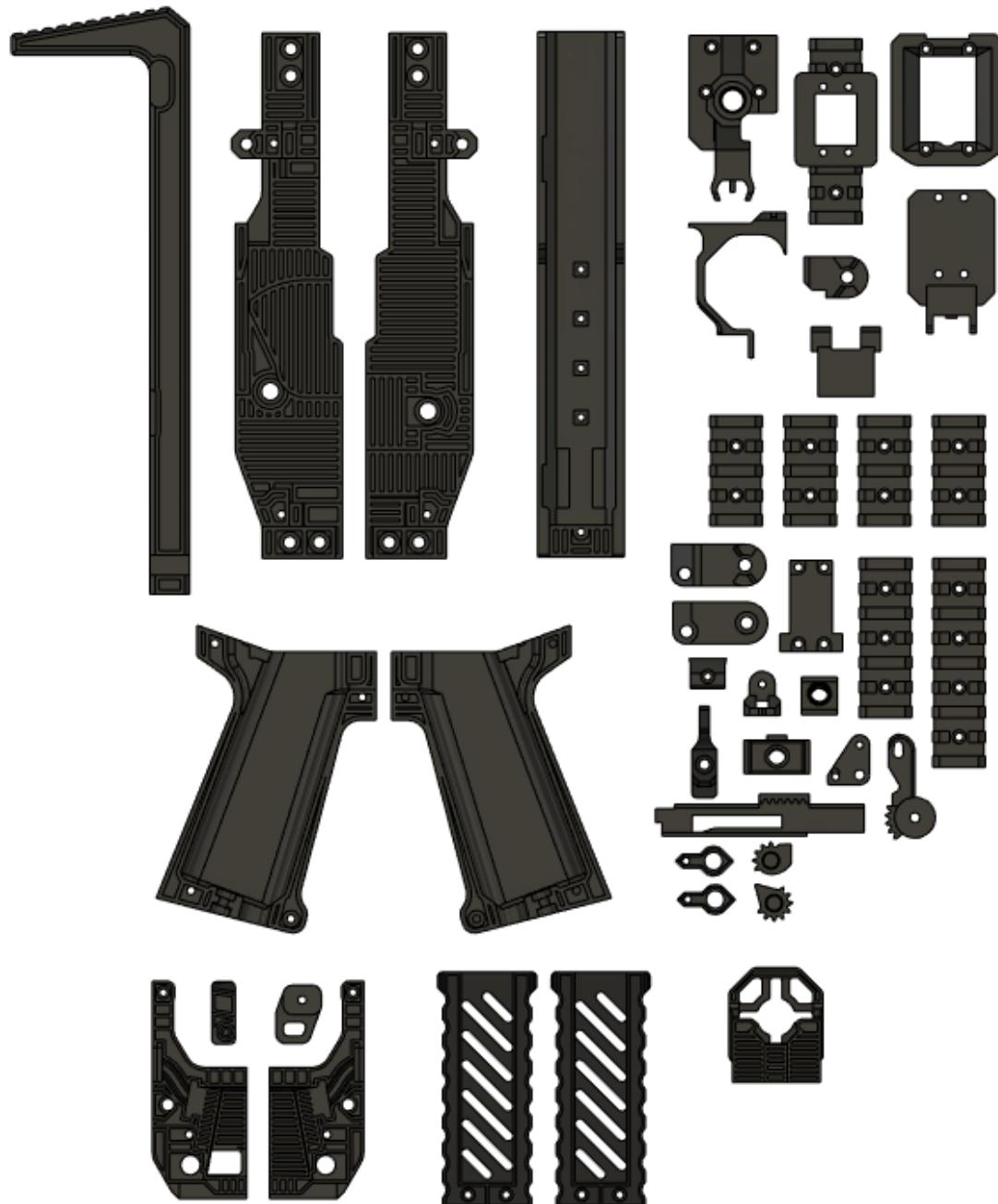


# MOSQUITO Open Beta 3D Printing Guide



By HYBRID AIRSOFT

# Contents

<b>1. Introduction</b>	<b>3</b>
1.1 Adapting PrusaSlicer To Your Printer	3
1.2 File Naming Convention	3
1.3 Recommended Materials	4
<b>2. Prerequisite Settings</b>	<b>5</b>
2.1 Recommended Nozzle Sizes	5
2.2 Recommended Layer Heights	5
2.3 Recommended Printing Speeds	5
2.4 Setting The Correct Perimeter Line Width	6
<b>3. Preparing The Build Surface</b>	<b>8</b>
3.1 How to Print on a Flat Build Surface	8
3.2 How to Print on a Textured Build Surface	8
<b>4. Printing Procedure</b>	<b>11</b>
4.1 Test Before You Commit!	11
4.2 Flat Build Surface Procedure With 0.6mm Nozzle	11
4.3 Textured Build Surface Procedure With 0.4mm Nozzle	12
4.4 Flat And Textured Build Surface Procedure With 0.6mm Nozzle	12
4.5 Parts That Require Support	13
4.6 Additional Tips And Tricks	18
4.6.2 Removing gap fill at the top	19
4.6.3 Seams	22
4.6.4 Variable layer height	24
<b>5. My Slicer Settings</b>	<b>26</b>
5.1 For 0.4mm Nozzle	27
5.2 For 0.6mm Nozzle	32

# 1. Introduction

The MOSQUITO has been designed and optimized with FDM 3D printing in mind where each layer is laid upon sequentially from the bottom up. It is not an easy model to print but great lengths have been made to make it as simple and supportless as possible to print. There are some parts that still require support but instructions will be there to aid the process.

These instructions and settings are based on using PrusaSlicer. Why? It's free, popular, and easy to use. If you are using other slicers such as Cura, Simplify3D, etc, you should be able to replicate these settings without too much trouble. Remember to do basic calibration on your own printer as most problems that people run into come from improper setup or omission of something.

There are two file types: STEP and STL. If you are 3D printing the kit, use the STL folder. STEP files are for 3D modelling and generally cannot be used in slicers for 3D printing.

If you need help or would like to suggest something about the kit, contact me via discord or email: hybridairsoftuk@gmail.com

---

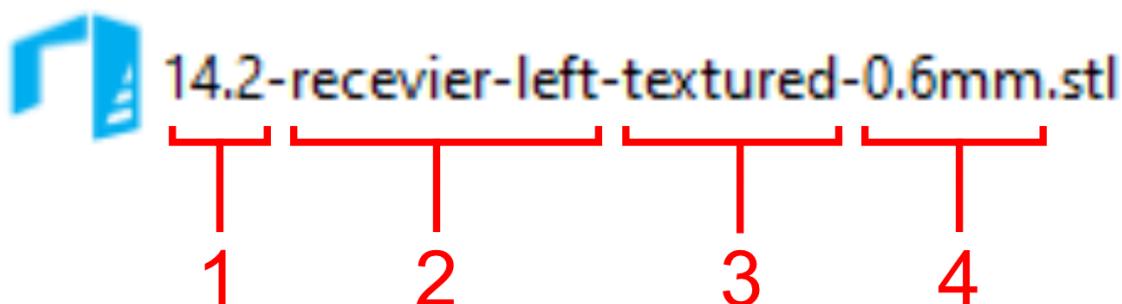
## 1.1 Adapting PrusaSlicer To Your Printer

PrusaSlicer, by default, has profiles for many different printers. Including popular machines from Creality, Lulzbot, Anycubic, etc. If your printer is not listed in the configuration wizard, you will need to create your own custom setup on it.

Instead of regurgitating basic information and taking up valuable space in this guide, this is a great [video](#) on setting up PrusaSlicer to work with any printer. After picking a machine preset, or making your own custom one, you can go down to the very bottom of the guide (section 5.2) and copy my print settings to get started.

---

## 1.2 File Naming Convention



1. Part number - decimal number denotes a variation of the part
  2. Part name - “left” and “right” parts are oriented as if you were holding the gun in a firing position
  3. The printed surface it is intended for - textured or flat (PEI, glass, etc)
  4. The nozzle size it is intended for - if not included in the file name, assume it is for a 0.4mm nozzle
- 

### 1.3 Recommended Materials

- PLA+ - eSUN PLA+, Polymaker Polymax PLA, etc
- ABS
- Nylon

Regular PLA isn't recommended due to its brittleness - you risk snapping it in the middle of a game. I have found PLA+ to be the best balance of strength and ease of printing (and cost).

Black filament is highly recommended if you are printing on a textured surface because, from my testing, most filaments are semi-transparent to an extent. Layers lines are quite visible with brighter materials even with the textured surface.

---

## 2. Prerequisite Settings

These are some settings and hardware that you should apply before doing anything else. The MOSQUITO has been designed around using certain parts, settings, and procedures.

Therefore, you could potentially run into some issues if you use default or premade profiles made by other companies and users.

If you really can't be bothered with the explanations (tl;dr as they say), you can just copy my slicer settings (section 5.2.) at the very bottom of the guide. Bear in mind that I am using a Prusa MK3S that has a direct drive extruder. You may need to alter some settings for your own printer setup.

---

### 2.1 Recommended Nozzle Sizes

- 0.4mm - can be used for all parts
- 0.6mm - optional for certain parts with the nozzle size in part name

For parts with options for both 0.4mm and 0.6mm, go with 0.4mm if you want it to just print, work, and look good. A 0.6mm nozzle is more difficult to work with and therefore requires a more finely tuned printer but will yield faster print times (printing time cut by around a half) and stronger parts. If there is no nozzle size in the part name, assume it is for a 0.4mm nozzle.

---

### 2.2 Recommended Layer Heights

- 0.3mm with a 0.4mm nozzle
- 0.45mm for a 0.6mm nozzle.

0.3mm is a good balance between detail and printing speed. However, you may decide to use a 0.2mm layer height for some small parts such as part 36 and 37 where a finer layer height will create a smoother transition for the stock locking mechanism.

---

### 2.3 Recommended Printing Speeds

- 20mm/s or less for external/small perimeters
- 40mm/s max for everything else

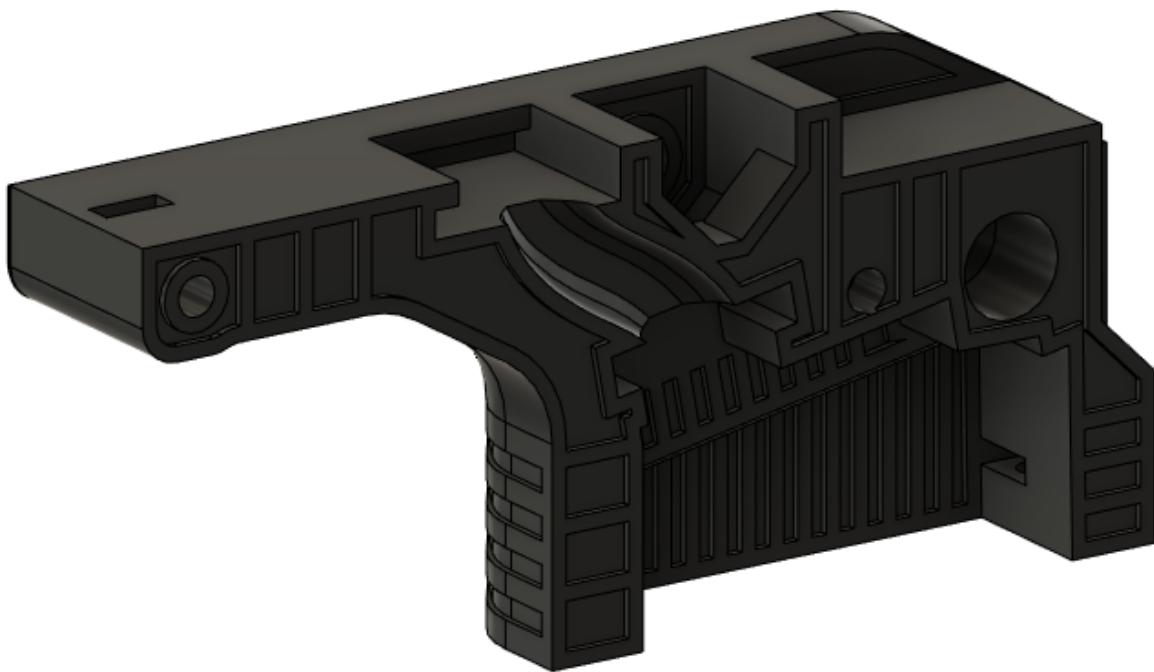
Over the course of the MOSQUITO project, these recommended printing speeds have been the result of vast experimentation to have the best balance between printing time, quality, and reliability.

---

## 2.4 Setting The Correct Perimeter Line Width

- 0.6mm for a 0.4mm nozzle
- 0.9mm for a 0.6mm nozzle.

It is important to set the correct perimeter line width so that the top surface of parts with two halves meets together as seamlessly as possible. This is due to a modelling technique that was ingeniously born over time from developing the MOSQUITO::

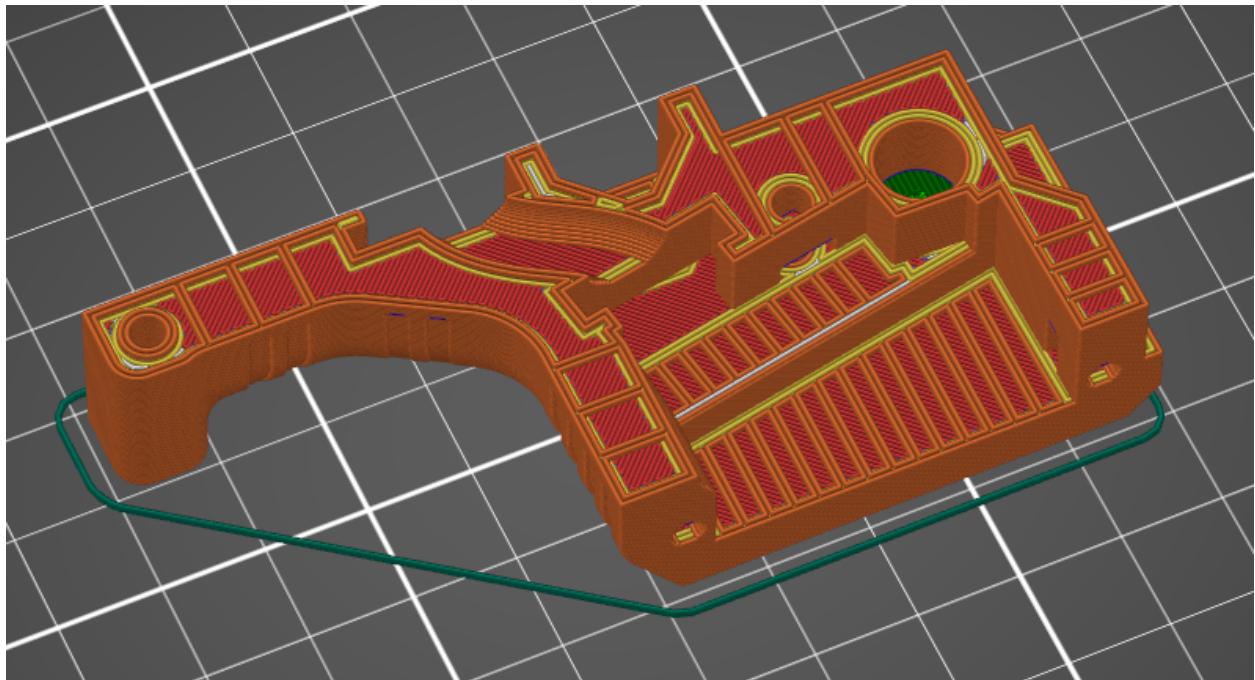


Normally with top surfaces, you'll have a zig-zag pattern that will bunch up a bit and cause it to... well, not be flat. This can be remedied using the ironing technique where the hot printer nozzle passes over the top surface to flatten it out. However, in my testing, even ironing can't reliably create a perfectly flat surface. In fact, it can make it worse if the bunching is especially bad because now it's just smushing over a turd.

What I noticed was that the perimeter walls at the edges are almost perfectly flat. So, with a stroke of genius, I decided to try and model perimeter wall thicknesses in the middle of models and lo-and-behold, it works! Therefore, to achieve a flat surface on top where the two halves

can mesh together well, you will need to set a correct multiple of the wall thickness so that the lines fit into it without having to do any additional infill or zig-zag pattern.

This technique results in the part looking like this in the slicer:



### 3. Preparing The Build Surface

Depending on what build surface you have, the procedure will be different. There are two main build surfaces the MOSQUITO is adapted for:

- Flat build surface - PEI, glass, etc
- Textured build surface - powder-coated, satin, etc

For ease of printing, a flat surface and the use of a 0.4mm nozzle is as fire-and-forget as possible. You will be able to put any combinations of parts onto the build surface and print them without trouble.

However, this kit was intended to have textured sides from printing on a textured bed but the process is more difficult and requires a more finely tuned printer (mainly a level bed) to succeed. In addition to your flat build surface, you will need a textured bed/sheet (some printers have removable sheets like the Prusa MK3S), and follow further instructions on how to set nozzle height and what combination of parts you can put on the build surface.

---

#### 3.1 How to Print on a Flat Build Surface

There really isn't much to it. Just make sure to use the "flat" version of the part if there are multiple options. In addition, you must print **internals first**. This is usually by default in slicers and 3D printing in general so don't worry too much about it. The reason is that the tolerances for the flat and textured versions are different; pinholes in the textured version are tighter than in the flat version.

And as always, make sure your printer bed is as level as possible so that you can avoid printing artifacts such as curling at corners.

---

#### 3.2 How to Print on a Textured Build Surface

To achieve the perfect textured surface, it is paramount that you make your printer bed as level as possible. Or rather, have your printer nozzle at a consistent z height from the printer bed at all points as much as possible. To create a uniform looking surface, you need the first layer to be squished as much and consistently as possible. This means having the printer nozzle at uncomfortably low z-axis heights. Otherwise, you will see the layer lines and those will break the continuity of the textured surface. With the flat model, you can get away with a bit of bed level variation. With the textured version, you almost cannot (unless you are fine with a discontinuous surface).

On some printers, you can adjust bed level with springs and knobs underneath. On other printers, like the Prusa MK3S, the bed is bolted down in place and requires community mods to adjust the levelling of it. Make sure to do research on your own printer and see what you can do to level the bed. If you cannot adjust it, try only printing in areas of the bed where you know it is level. If there is nothing available, go with the flat version instead (or perhaps ask someone to help).

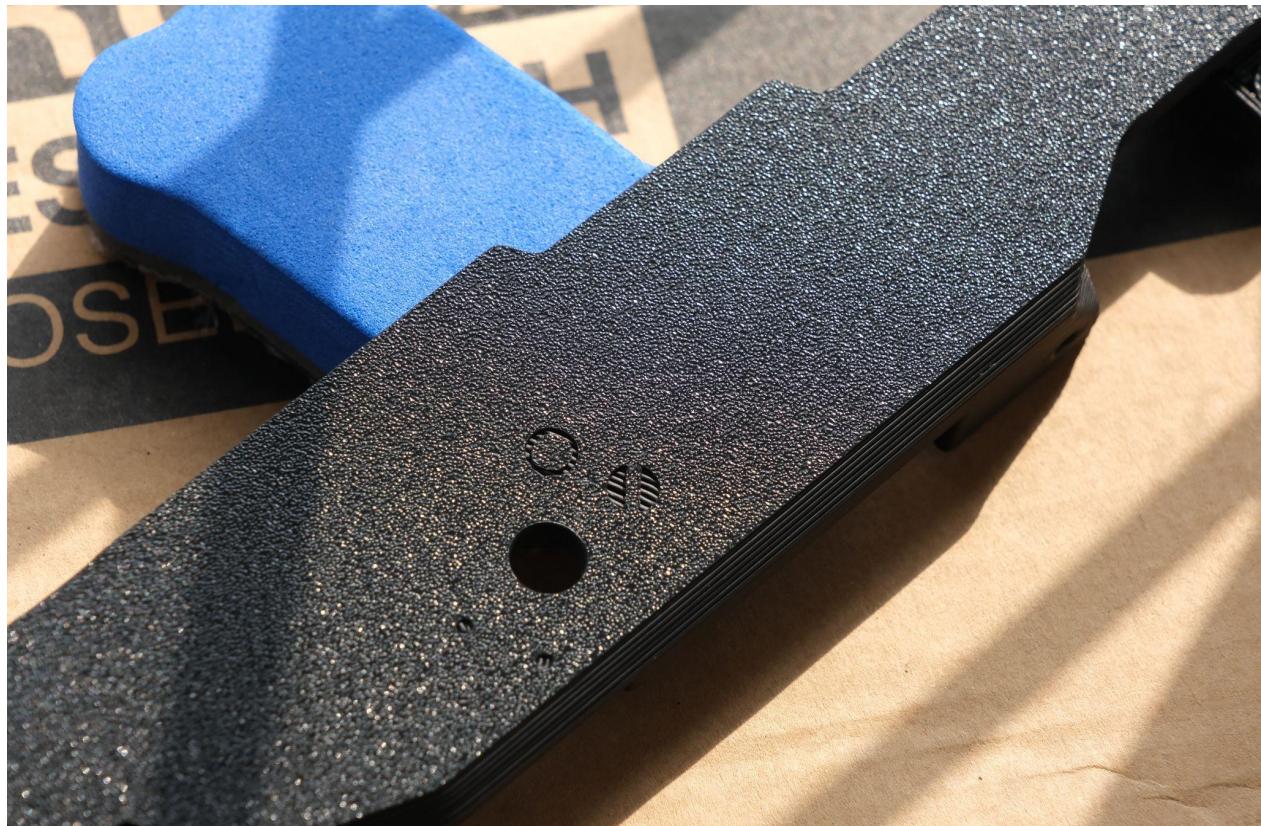
First layer settings:

- For a 0.4mm nozzle, set your first layer height to 0.1mm.
- For a 0.6mm, set it to 0.15mm.

This corresponds to 25% of the nozzle diameter (multiply nozzle diameter by 0.25).

Next, set your nozzle height (or live z) uncomfortably low but not touching the print bed. The first layer should be a bit mushy at the top but not scratchy/patchy. Do a test print using the group 7 test files and keep testing until the layer lines at the bottom have disappeared or are barely visible under strong lightning. If you find that the lines are not consistent and break up a bit, then it probably means the nozzle is too low.

The result should look like this (you may need to zoom in a bit):



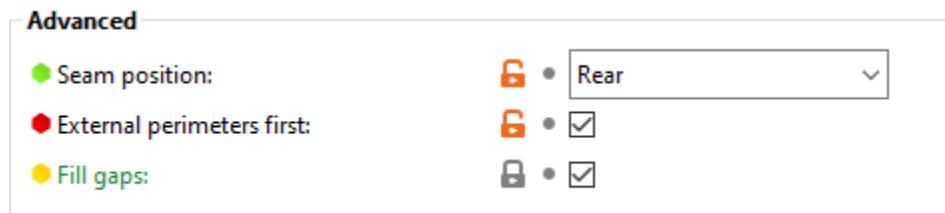
If you are using a Prusa MK3S and have both PEI and textured sheets, after setting your ideal live z on the PEI sheet, the corresponding live z for the textured sheet is 0.370mm lower than the PEI live z. For example, if your PEI live z is -1.000mm, then your textured live z is -1.370mm.

It is also very important that you print the textured parts **externals first** as they have been designed and tolerated with that in mind. By default in slicers, you print internals first then the external perimeters. However, this normally would expand the print in the x and y-axis by about 0.2-0.4mm compared to what it actually is in the CAD model due to the internal infill spreading out a bit and subsequently pushing out the external perimeters. This has an adverse effect on hole sizes as expansion can make them quite unpredictably uneven which is very undesirable for precise pins that are pushed through.

By contrast, when you print externals first, you lay down the foundations of the outer shell first, thus improving the overall external accuracy and consistency when comparing dimensions to the CAD model. This usually results in  $\pm 0.1\text{mm}$  and is much more preferable when it comes to precise holes for push-through pins.

To turn on printing external perimeters first, navigate to the top left: Print Settings > Layers and perimeters > Advanced

In the advanced section, you should see “External perimeters first”. Check that box:



To recap (accounting for attention span):

- Set first layer height to 25% of nozzle diameter
  - Lower nozzle height so that layer lines at the bottom disappear on test prints
  - **Print externals first**
-

## 4. Printing Procedure

If you only have one 3D printer, printing the MOSQUITO will take approximately two days assuming using 2-3 perimeters and 25% infill.

To begin composing parts on the build surface, simply drag and drop the 3D files into the slicer. Don't worry about finding the optimal orientation for printing as this has already been painstakingly done beforehand.

---

### 4.1 Test Before You Commit!

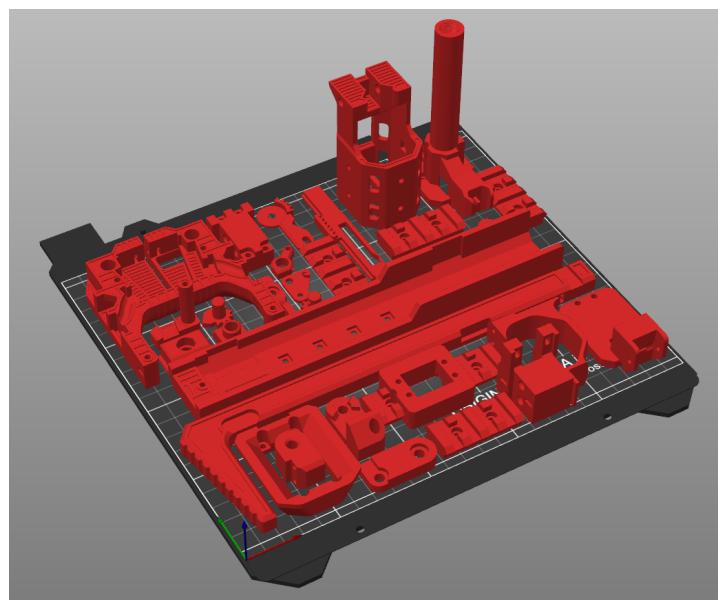
In the groups of files, you will see a folder named "6. Tests". In that folder are small test pieces that are chopped off from the main parts and they are there for you to get the supports, pinhole size, and textured surface correct before committing to printing entire parts.

Beginner or pro, I highly recommend testing first due to the highly unorthodox methods and procedures that govern this kit.

---

### 4.2 Flat Build Surface Procedure With 0.6mm Nozzle

If you are printing on a flat build surface, this is the easiest path as you can add any combination of parts to the slicer as long as they fit. But, before you get any ideas: yes, it is possible to do something like this (22 hours of non-stop printing with 0.4mm nozzle):



Do I recommend it? Not really. The reason is that the first layers for each part are going to be hanging around for so long that you risk it warping. Best to split it into 3-5 hour segments.

Remember to select the “flat” version of parts that have multiple variations.

---

#### **4.3 Textured Build Surface Procedure With 0.4mm Nozzle**

The process is mostly identical to the flat build surface procedure where you can slap on any combination of parts onto the build surface. Except, now there is a pool of textured parts that should be printed on a textured sheet. These parts are:

- Part 6
- Part 7
- Part 10
- Part 14
- Part 15
- Part 23
- Part 24
- Part 27
- Part 28
- Part 30
- Part 31
- Part 38

You can throw any combination of these parts onto the textured build surface as long as they fit.

---

#### **4.4 Flat And Textured Build Surface Procedure With 0.6mm Nozzle**

Some parts were designed for use of a 0.6mm nozzle which enables faster print times and yields stronger parts. They are very common and easy to find. Just make sure to find one that is compatible with your printer extruder, should you attempt it.

Due to the higher flow rate, it is more tricky to fine-tune and achieve high-quality results. Therefore, it is important that you test print first before committing to entire parts. For a quick start, copy my settings at the bottom (section 5.2) for a 0.6mm nozzle.

For a flat build surface, these parts are:

- Part 10.3
- Part 14.3

- Part 15.3
- Part 30.3
- Part 31.3
- Part 38.3

For a textured build surface, these parts are:

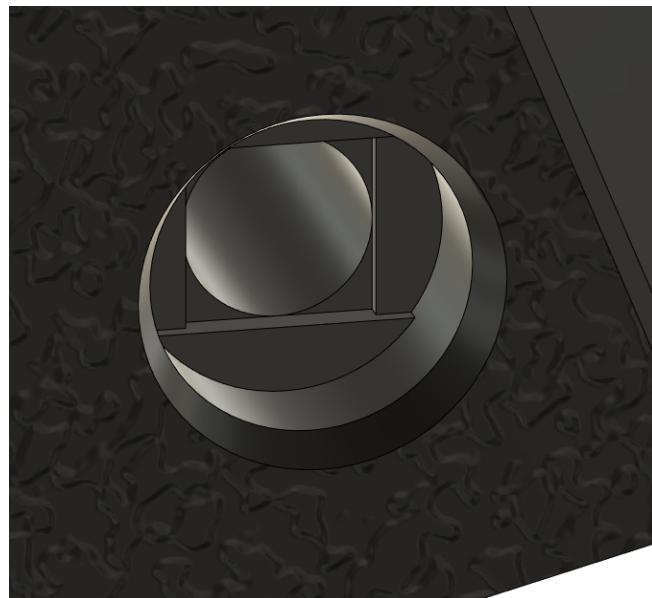
- Part 10.2
- Part 14.2
- Part 15.2
- Part 30.2
- Part 31.2
- Part 38.2

Remember not to mix parts for a 0.4mm nozzle on the same build surface while using a 0.6mm nozzle.

---

#### 4.5 Parts That Require Support

**DO NOT** add supports to parts except those properly defined in this section. The majority of parts have been designed in a way that allows the filament to climb without needing any additional supporting structure, even screw holes with some clever modelling:



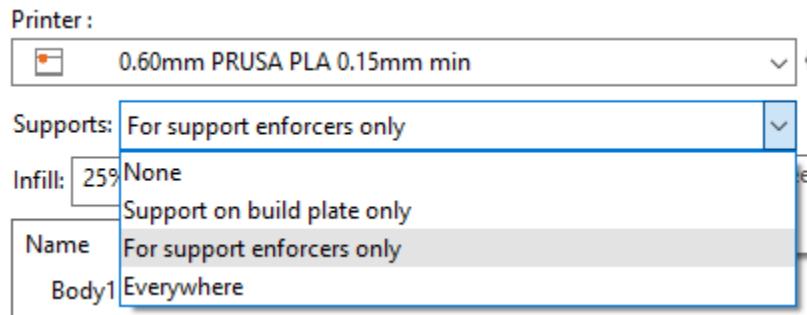
While the majority of parts do not require supports at all, some still require them in certain places as they have tricky geometry that hasn't been overcome just yet.

These parts are:

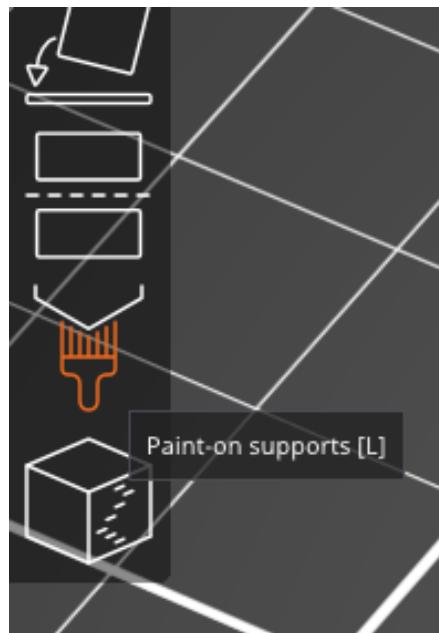
- Part 14 and 15, the pinholes for the magwell
- Part 23 and 24, the areas that house the magazine catch

If you are unsure of what support settings to use, scroll down to the bottom (section 5.2) and copy my support settings for whatever nozzle size you are using.

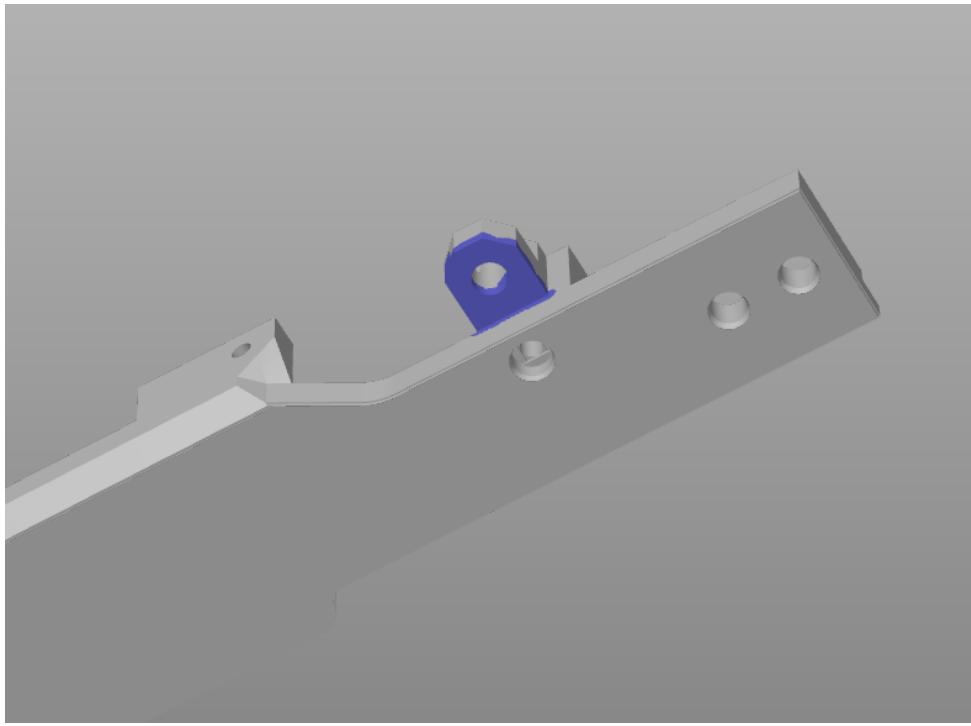
To start, select “support enforcers only”. To do this, make sure you are in expert mode (top right) and on the right-hand side, there is a drop-down menu that shows you the options on what kind of support you want:



From there, pick “For support enforcers only”. This means that it will only support the places you want it to. And how to do that? Well, there is a handy tool in PrusaSlicer that allows you to paint the areas that you want to support. It is second from the bottom on the left-hand side menu:

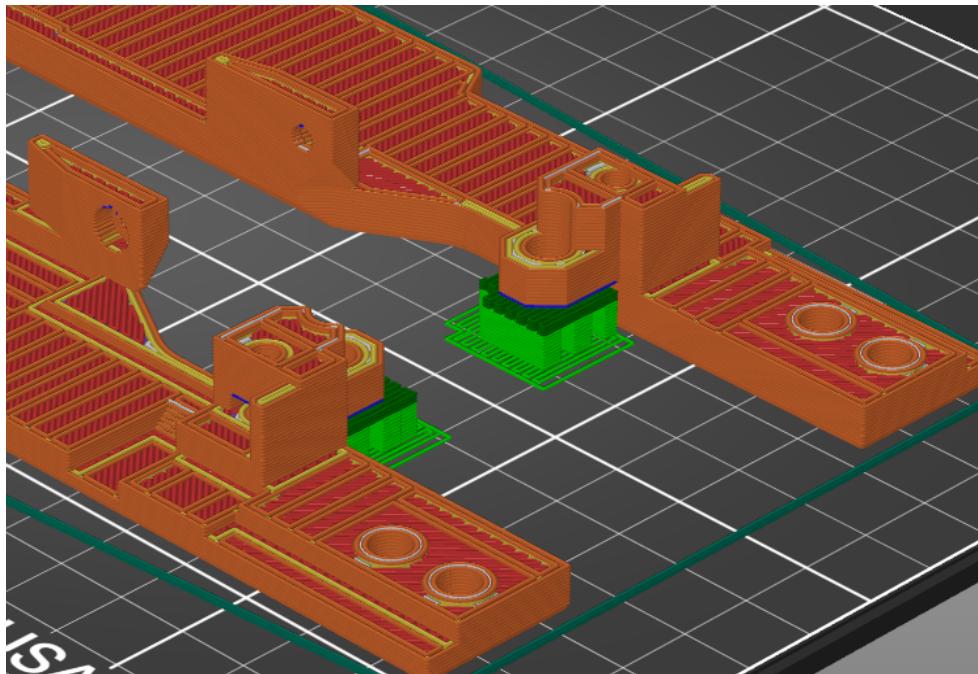


There will be instructions that pop out. Follow those to apply supports on areas that you want. For parts 14 and 15, you need to paint the area for the magwell pin:

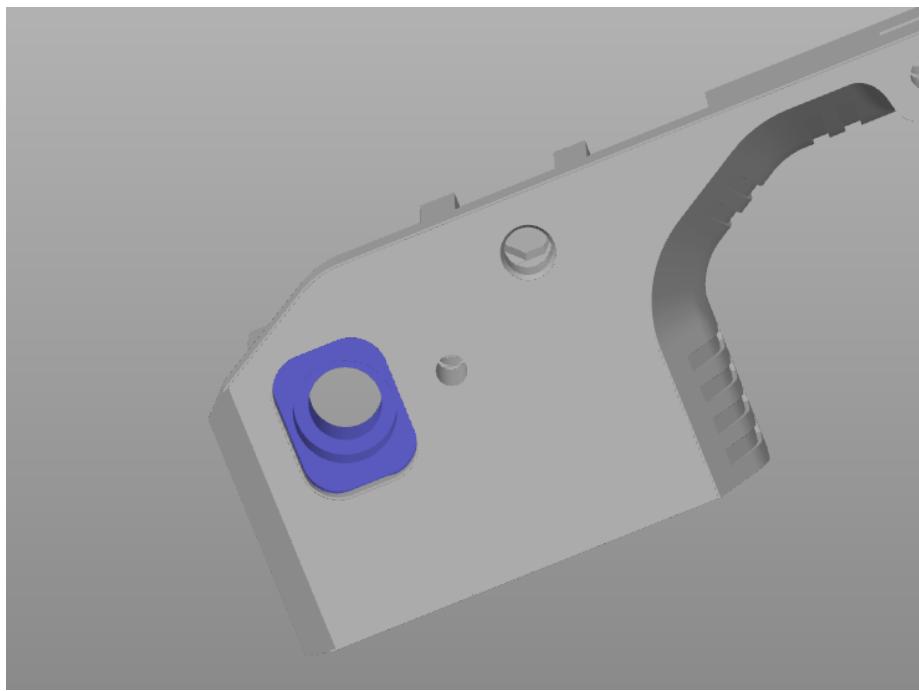
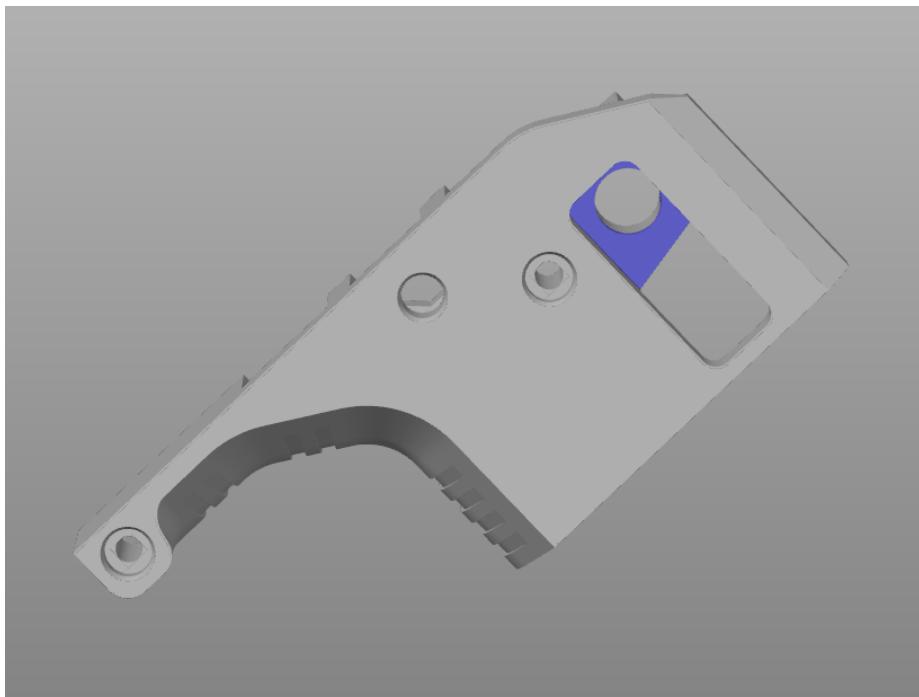


By pressing 2 in PrusaSlicer, you can change your view to the underside of the part which will make it easier to paint.

Once those areas are supported, it should look like this when sliced:

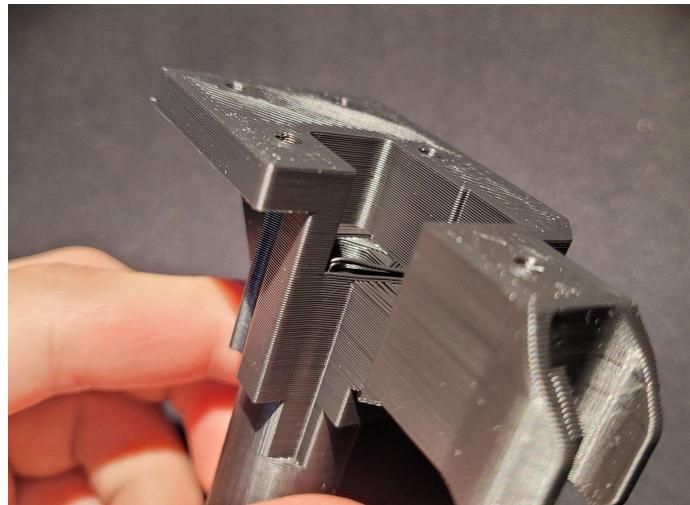


For parts 23 and 24, it is areas where the magazine catch goes through as highlighted:

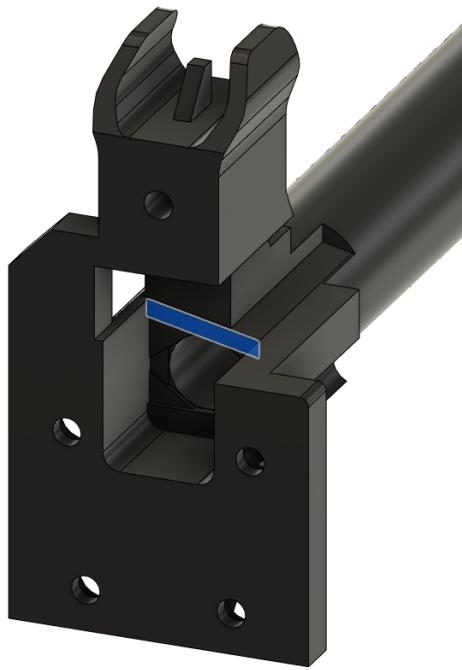


For part 24 (image above), the support may be difficult to remove due to it being on two levels. The easiest way is to poke through the other end with a screwdriver or an allen key. Be sure to not make the contact distance between the part and support interface too close or you may face a lot of difficulties removing it. 0.2mm difference in contact should be enough.

For part 1, the barrel, while it does not need support, you may come across some strings that droop down:



This corresponds to the highlighted blue strip on the 3D model:

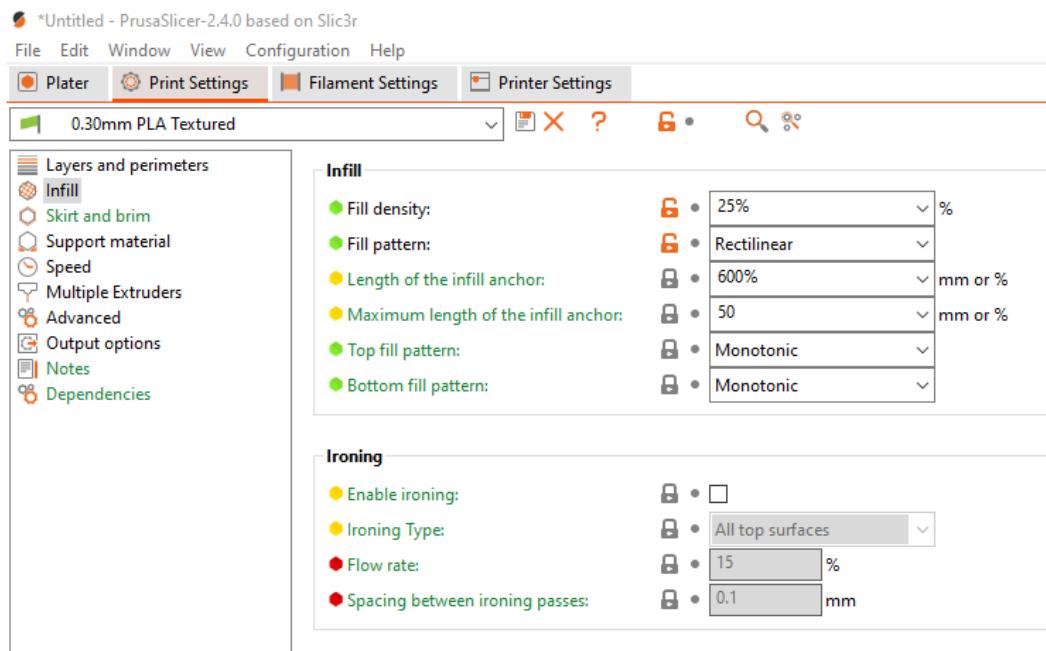


While the best efforts thus far have been made to make it supportless, it is not perfect yet. The blue strip is a necessary long bridge that may not complete perfectly and droop down. It acts as the foundation for the rest of the supportless structure to build upon. You may need to clip the strings out if they interfere with the hop-up unit.

## 4.6 Additional Tips And Tricks

### 4.6.1 Ironing

If you wish to improve the top surface appearance on some parts, you can iron the top surfaces by turning on the ironing settings. To do this, navigate to: Print settings > Infill > Ironing



Check “Enable Ironing” and you will be presented with a few options. The default settings in the above image should be perfectly fine. In the “Speed” section, set ironing speed around 20mm/s or slower for best results.

**Do not** iron surfaces with the perimeter wall modelling technique on the top surface as described in section 2.4. These parts have been designed to be as flat as possible with their top surfaces from normal printing and ironing can add on unnecessary bulk which defeats the purpose. The parts to **avoid ironing** are:

- Part 6
- Part 7
- Part 10
- Part 14.2
- Part 15.2
- Part 23
- Part 24
- Part 30
- Part 31
- Part 38

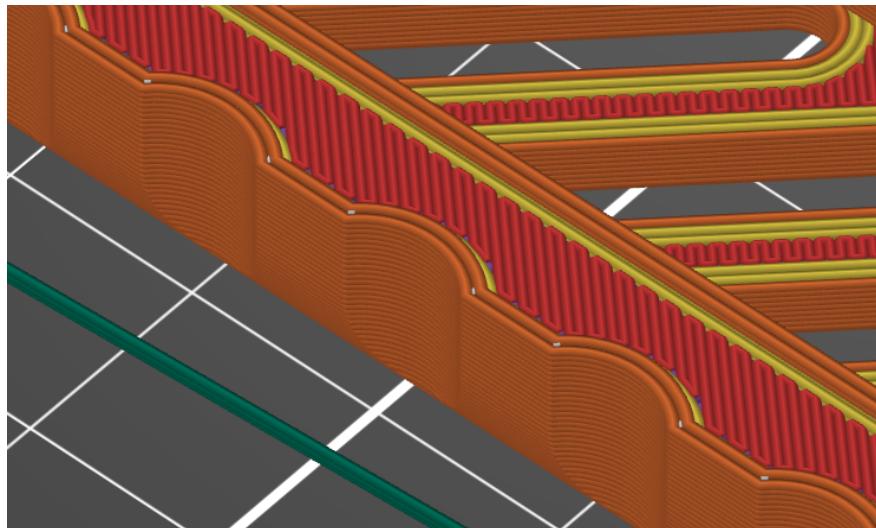
The rest are fine to iron and should turn out something like this:



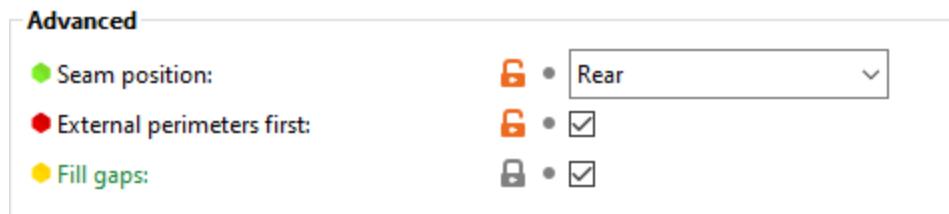
Some filaments do iron better than others, particularly those that are less viscous.

#### 4.6.2 Removing gap fill at the top

When slicing parts in PrusaSlicer, you may notice that some parts have unwanted gap fill at the top, as denoted by white streaks or dots. For example, notice these white dots in between the external perimeters:

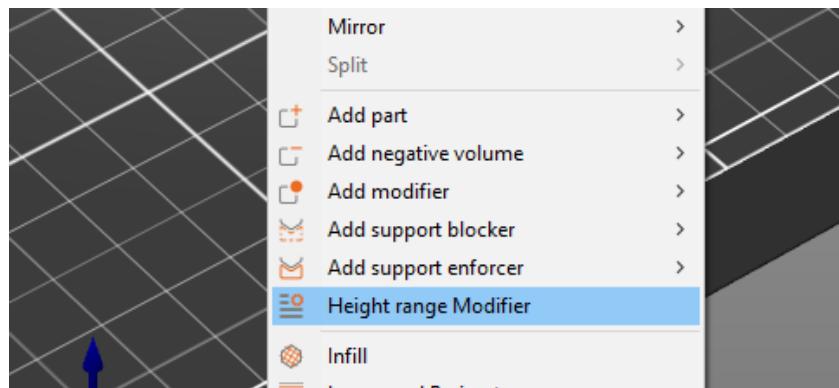


These aren't a huge problem but for parts like the battery grip where two halves mesh together, this may cause gaps between them. One solution to remove them is to turn off gap fill in the slicer. This can be done by navigating to Print settings > Layers and perimeters > Advanced

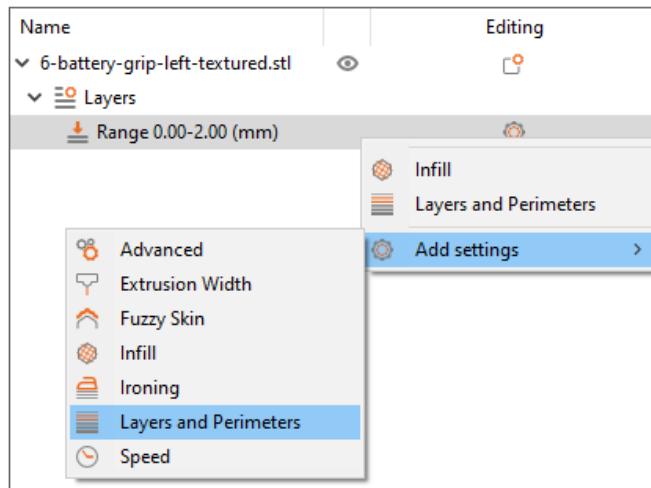


Uncheck gap fill and it will be gone when slicing. However, this removes all gap fill from the model including internally, so it may not be practical to do this.

There is a way to only remove the gap fill at specific layers. To do this, right-click on the object and select "Height range modifier" on the drop-down menu:



Then, on the right side, click on the cogwheel on the object and navigate to "Layers and Perimeters" as shown below:



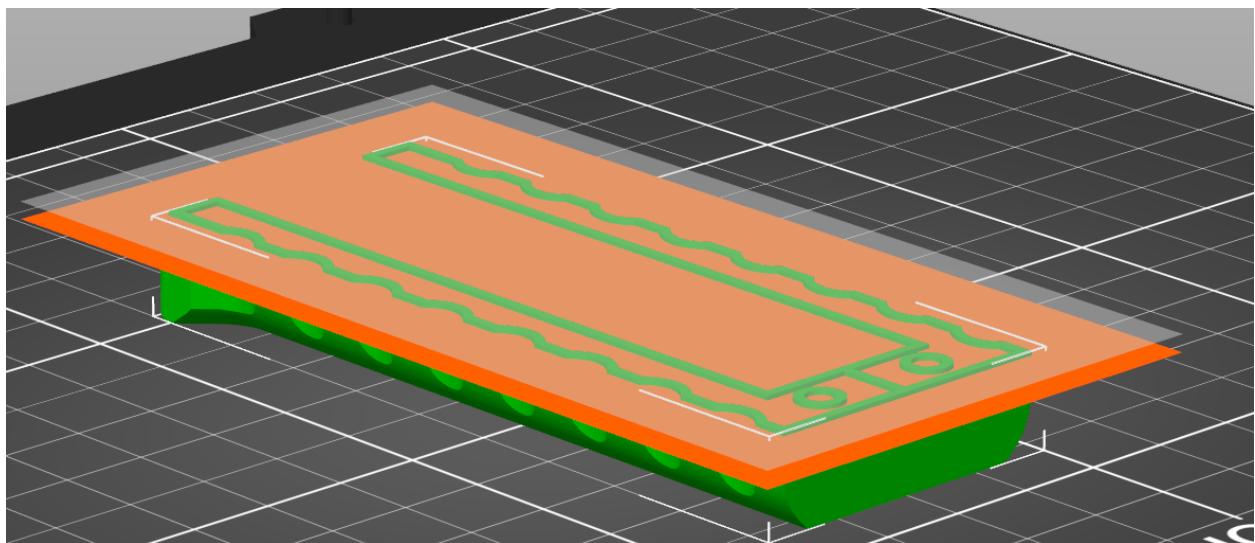
When another menu pops up, tick “Fill gaps”. After that, click on “Layers and Perimeters” on the right at the bottom of the object hierarchy and uncheck “Fill gaps” at the bottom:



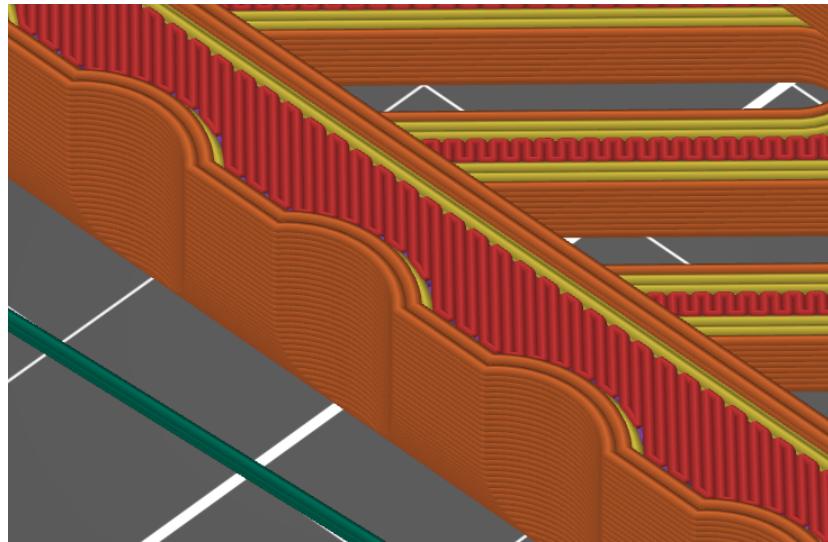
The way the height range modifier works is very simple: you select two heights and anything between those two heights, you can choose what happens to them out of the options available. To select those two heights, click on “Range” on the right side, and at the bottom, you will be able to input the start and stop heights:



In this case, for the battery grip, the heights of 11mm and 13mm encase the topmost layers as shown:



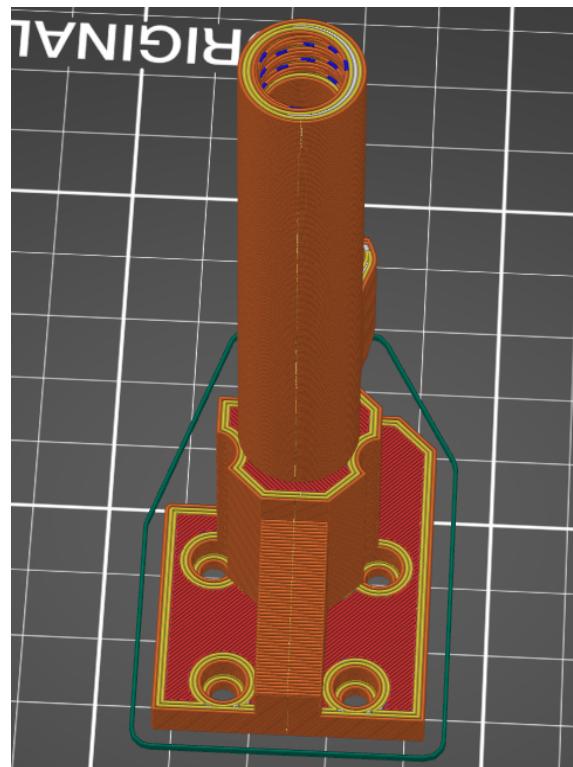
Since gap fill in this height range has now been turned off from above, the sliced top surface should look like this:



The gap fill has now been removed. You can use this method to do other things such as turning off ironing for specific objects, changing infill, or even how many layers or perimeters you want. There are many options you can play and experiment with.

#### 4.6.3 Seams

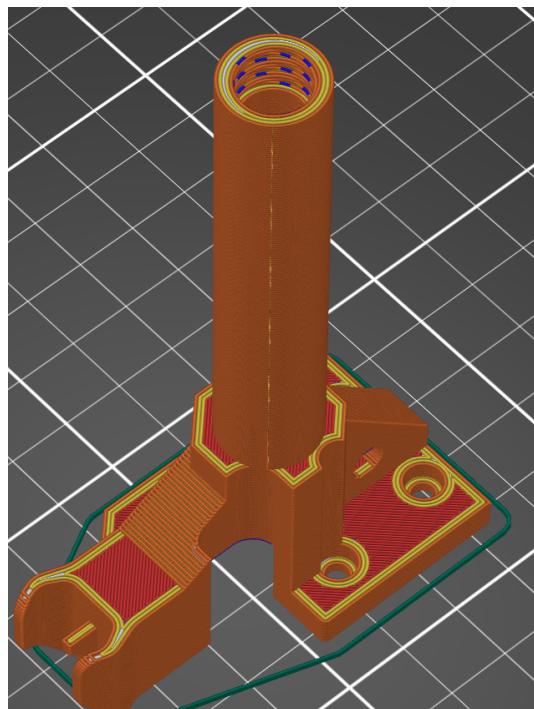
Seamlines are visible vertical layer transitions that can be seen on the exterior of a 3D print. For example on part 1, the barrel:



Unfortunately, they are unavoidable in 3D printing but there are ways of managing them. In PrusaSlicer this can be adjusted by navigating to Print Settings > Layers and Perimeters > Advanced. In the Advanced section, you will see “seam position”. The best choices for aesthetic quality are “rear” and “aligned”. “Nearest” is a bit of a 50/50 on whether it works or not, and random you should avoid completely as it will create ugly zits all over the model.

“Aligned” is a good default setting to have. This mostly keeps the vertical seamlines together as much as possible, usually preferring corners first. For most of the parts, “aligned” will work just fine. However, sometimes the placement of the seams can be unpredictable and undesirable.

“Rear” will align the seams towards the back of the build plate. It will try to align it at one spot in the middle of the model as much as possible like in the image of the barrel above. By default, PrusaSlicer tends towards putting seams in corners of models to hide them as much as possible. However, with cylindrical shapes, this is not as easy and can result in a mess. Therefore, having it all lined up neatly using the rear position is hugely beneficial. The main benefit of using “rear” is the predictability of the seam positions. For example, using “aligned” on the barrel results in this:

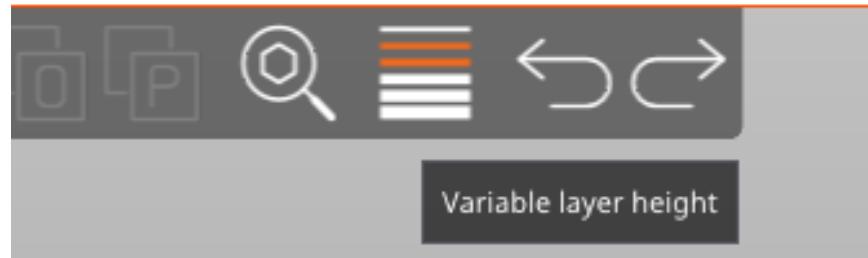


While the seams are straight, they are in an undesirable position. It is much more preferable to use “rear” to position the seams at the bottom of the barrel, hidden away.

With all that said, all the parts have been made and positioned in such a way that takes advantage of the “rear” setting. So there isn’t any rotational guesswork to do on your part. If “rear” doesn’t look appealing, then you can always change it to “aligned” without repositioning anything.

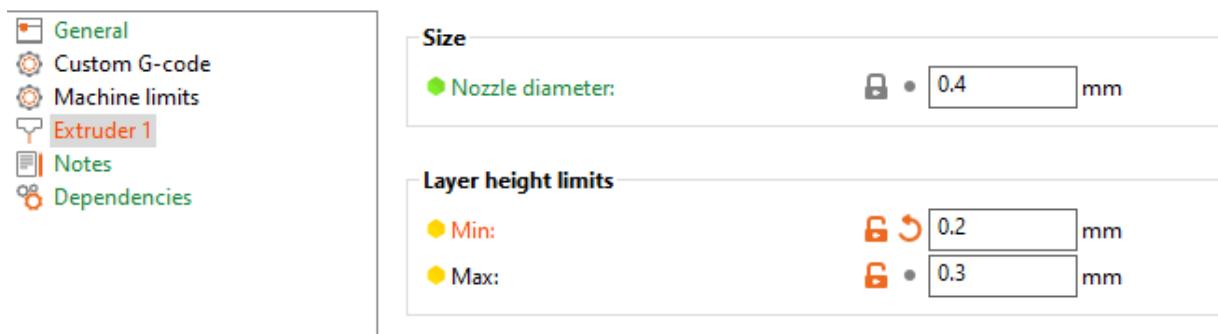
#### 4.6.4 Variable layer height

For parts that require a little more precision, you can print them in 0.2mm layer height or less. Or, you can vary the layer height for specific areas during the actual print. To do this, navigate to the top menu bar, and on the right, you will see an icon that denotes variable layer height as shown:

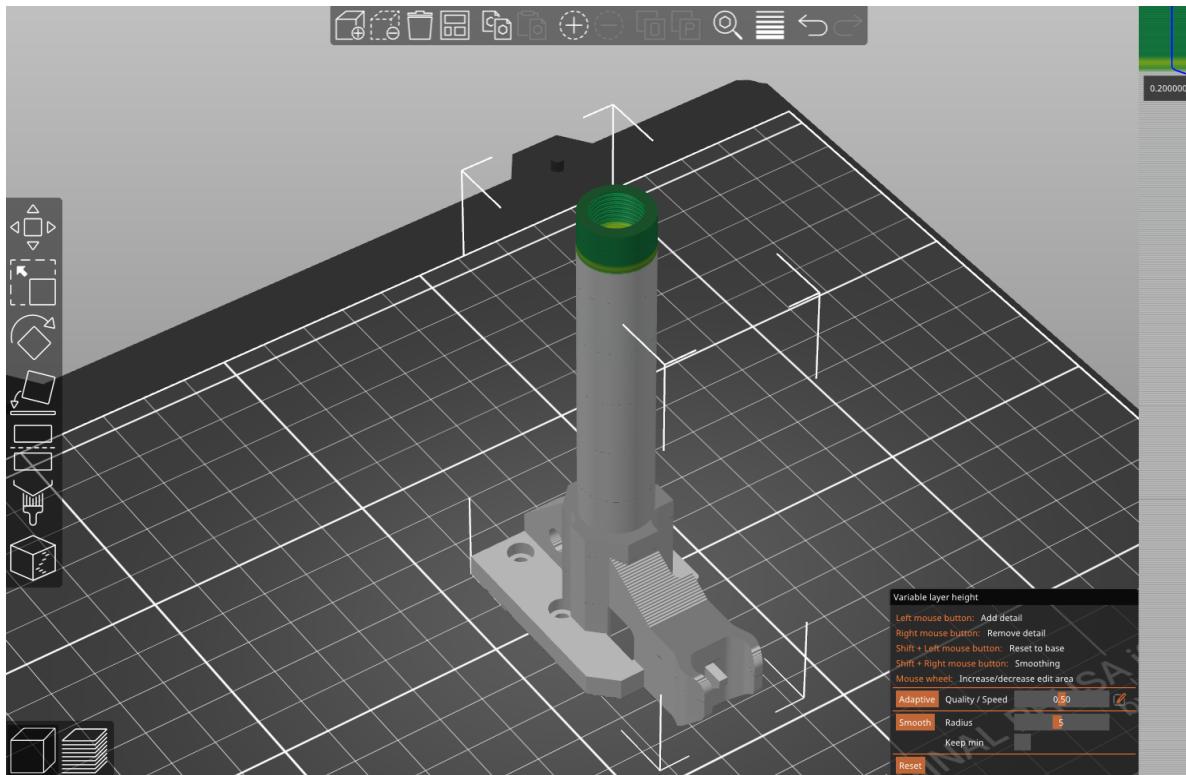


Click it and select the desired part you want fine-tuning on the right. The selected part will turn grey. On the right side of the editor, you will see a vertical bar that shows the layer height of the part. If you hover over it and observe the part at the same time, you will see that where you hover over corresponds to the layer height of the part. There are instructions on the bottom right on to vary the layer height.

You may notice that by left-clicking, it will shrink the layer heights of that area but not to the layer height that you want. To control the minimum layer height that it shrinks to, navigate to Printer Settings > Extruder > Layer height limits and set your desired minimum layer height, for example:



In this case, I set it to 0.2mm. Going back to the editor, this will now keep the minimum layer height to 0.2mm. Using the barrel again as an example, while 0.3mm for the internal thread will work just fine, 0.2mm will produce a better result. So by only adjusting the threaded part of the barrel, the new edited result will look like this:



This can be done for other parts such as the stock locking mechanism on parts 36 and 37 for a smoother transition on the locking mechanism.

---

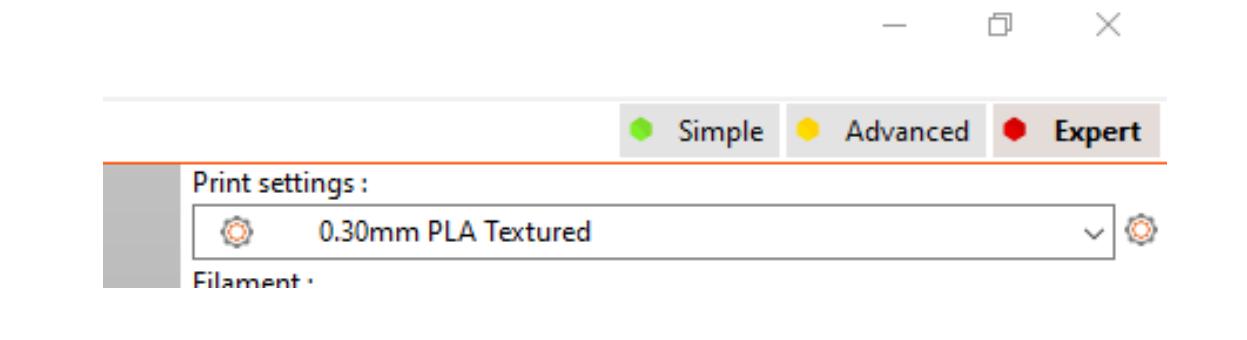
## 5. My Slicer Settings

Can't be bothered reading explanations and just want to quickly copy, paste, and get going? You've come to the right place. Make sure to find the right combination of settings for your printer (this means read the titles properly for once). These are general settings I use for most 3D prints.

**IMPORTANT:** The profiles I use for flat and textured build surfaces are almost identical. The only difference is the first layer height - the same as layer height for flat build surface, or 25% of nozzle size for textured build surface. Be sure to adjust it when switching between profiles, or make a separate profile for each build surface.

Please note that although it says PLA in the profile name, they can be applied to all the recommended materials.

Also, be sure to enable expert mode by going to the top right of the slicer and clicking on it:



## 5.1 For 0.4mm Nozzle

The screenshot shows the Cura slicing software interface for a 0.4mm nozzle. The left sidebar lists various slicing options: Layers and perimeters, Infill, Skirt and brim, Support material, Speed, Multiple Extruders, Advanced, Output options, Notes, and Dependencies. The main panel is divided into several sections:

- Layer height**:
  - Layer height: 0.3 mm
  - First layer height: 0.1 mm
- Vertical shells**:
  - Perimeters: 3 (minimum)
  - Spiral vase: Off

Recommended object thin wall thickness for layer height 0.30 and 2 lines: 1.14 mm , 4 lines: 2.21 mm , 6 lines: 3.28 mm
- Horizontal shells**:
  - Solid layers: Top: 4, Bottom: 4
  - Minimum shell thickness: Top: 0 mm, Bottom: 0 mm

Top shell is 1.2 mm thick for layer height 0.3 mm. Minimum top shell thickness is 0.4 mm.  
Bottom shell is 1.2 mm thick for layer height 0.3 mm. Minimum bottom shell thickness is 0.4 mm.
- Quality (slower slicing)**:
  - Extra perimeters if needed: Off
  - Ensure vertical shell thickness: On
  - Avoid crossing perimeters: Off
  - Avoid crossing perimeters - Max detour length: 0 mm or % (zero to disable)
  - Detect thin walls: Off
  - Thick bridges: Off
  - Detect bridging perimeters: On
- Advanced**:
  - Seam position: Rear
  - External perimeters first: On
  - Fill gaps: On
- Fuzzy skin (experimental)**:
  - Fuzzy Skin: None
  - Fuzzy skin thickness: 0.3 mm
  - Fuzzy skin point distance: 0.8 mm

0.30mm PLA Textured

Layers and perimeters

- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

Infill

- Fill density: 25% %
- Fill pattern: Rectilinear
- Length of the infill anchor: 600% mm or %
- Maximum length of the infill anchor: 50 mm or %
- Top fill pattern: Monotonic
- Bottom fill pattern: Monotonic

Ironing

- Enable ironing:
- Ironing Type: All top surfaces
- Flow rate: 15 %
- Spacing between ironing passes: 0.1 mm

Reducing printing time

- Combine infill every: 1 layers
- Only infill where needed:

Advanced

- Solid infill every: 0 layers
- Fill angle: 45 °
- Solid infill threshold area: 0 mm<sup>2</sup>
- Bridging angle: 0 °
- Only retract when crossing perimeters:
- Infill before perimeters:

0.30mm PLA Textured

Layers and perimeters

Infill

Skirt and brim

Support material

Speed

Multiple Extruders

Advanced

Output options

Notes

Dependencies

### Support material

- Generate support material:
- Auto generated supports:
- Overhang threshold:
- Enforce support for the first:
- First layer density:  %
- First layer expansion:  mm

### Raft

- Raft layers:  layers
- Raft contact Z distance:  mm
- Raft expansion:  mm

### Options for support material and raft

- Style:
- Top contact Z distance:  mm
- Bottom contact Z distance:  mm
- Pattern:
- With sheath around the support:
- Pattern spacing:  mm
- Pattern angle:  °
- Closing radius:  mm
- Top interface layers:  layers
- Bottom interface layers:  layers
- Interface pattern:
- Interface pattern spacing:  mm
- Interface loops:
- Support on build plate only:
- XY separation between an object and its support:  mm or %
- Don't support bridges:
- Synchronize with object layers:

0.30mm PLA Textured

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

---

### Speed for print moves

Perimeters:	<input type="text" value="40"/> mm/s
Small perimeters:	<input type="text" value="20"/> mm/s or %
External perimeters:	<input type="text" value="20"/> mm/s or %
Infill:	<input type="text" value="40"/> mm/s
Solid infill:	<input type="text" value="40"/> mm/s or %
Top solid infill:	<input type="text" value="20"/> mm/s or %
Support material:	<input type="text" value="20"/> mm/s
Support material interface:	<input type="text" value="20"/> mm/s or %
Bridges:	<input type="text" value="40"/> mm/s
Gap fill:	<input type="text" value="20"/> mm/s
Ironing:	<input type="text" value="20"/> mm/s

---

### Speed for non-print moves

Travel:	<input type="text" value="80"/> mm/s
Z travel:	<input type="text" value="0"/> mm/s

---

### Modifiers

First layer speed:	<input type="text" value="10"/> mm/s or %
Speed of object first layer over raft interface:	<input type="text" value="30"/> mm/s or %

---

### Acceleration control (advanced)

Perimeters:	<input type="text" value="500"/> mm/s <sup>2</sup>
Infill:	<input type="text" value="1000"/> mm/s <sup>2</sup>
Bridge:	<input type="text" value="1000"/> mm/s <sup>2</sup>
First layer:	<input type="text" value="1000"/> mm/s <sup>2</sup>
First object layer over raft interface:	<input type="text" value="0"/> mm/s <sup>2</sup>
Default:	<input type="text" value="1000"/> mm/s <sup>2</sup>

---

### Auto Speed (advanced)

Max print speed:	<input type="text" value="200"/> mm/s
Max volumetric speed:	<input type="text" value="0"/> mm <sup>3</sup> /s

The screenshot shows the Cura slicing software interface with the following settings:

- Layers and perimeters**:
  - Infill
  - Skirt and brim
  - Support material
  - Speed
  - Multiple Extruders
  - Advanced** (selected)
  - Output options
  - Notes
  - Dependencies
- Extrusion width**:
  - Default extrusion width: 0.6 mm or %
  - First layer: 0.6 mm or %
  - Perimeters: 0.6 mm or %
  - External perimeters: 0.6 mm or %
  - Infill: 0.6 mm or %
  - Solid infill: 0.6 mm or %
  - Top solid infill: 0.5 mm or %
  - Support material: 0.45 mm or %
- Overlap**:
  - Infill/perimeters overlap: 20% mm or %
- Flow**:
  - Bridge flow ratio: 0.9
- Slicing**:
  - Slice gap closing radius: 0.049 mm
  - Slicing Mode: Regular
  - Slice resolution: 0 mm
  - G-code resolution: 0.0125 mm
  - XY Size Compensation: 0 mm
  - Elephant foot compensation: 0.2 mm
- Other**:
  - Clip multi-part objects: checked

## 5.2 For 0.6mm Nozzle

The screenshot shows the Cura slicing software interface with the following settings:

- Layers and perimeters**:
  - Infill
  - Skirt and brim
  - Support material
  - Speed
  - Multiple Extruders
  - Advanced
  - Output options
  - Notes
  - Dependencies
- Layer height**:
  - Layer height: 0.45 mm
  - First layer height: 0.15 mm
- Vertical shells**:
  - Perimeters: 2 (minimum)
  - Spiral vase:
- Horizontal shells**:
  - Solid layers: Top: 3, Bottom: 3
  - Minimum shell thickness: Top: 1, Bottom: 1

Top shell is 1.35 mm thick for layer height 0.45 mm. Minimum top shell thickness is 1 mm.  
Bottom shell is 1.35 mm thick for layer height 0.45 mm. Minimum bottom shell thickness is 1 mm.
- Quality (slower slicing)**:
  - Extra perimeters if needed:
  - Ensure vertical shell thickness:
  - Avoid crossing perimeters:
  - Avoid crossing perimeters - Max detour length: 0 mm or % (zero to disable)
  - Detect thin walls:
  - Thick bridges:
  - Detect bridging perimeters:
- Advanced**:
  - Seam position: Aligned
  - External perimeters first:
  - Fill gaps:
- Fuzzy skin (experimental)**:
  - Fuzzy Skin: None
  - Fuzzy skin thickness: 0.3 mm
  - Fuzzy skin point distance: 0.8 mm

0.45mm PLA 0.6mm Textured

Layers and perimeters

- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

### Infill

- Fill density:  25% %
- Fill pattern:  Rectilinear
- Length of the infill anchor:  600% mm or %
- Maximum length of the infill anchor:  50 mm or %
- Top fill pattern:  Monotonic
- Bottom fill pattern:  Monotonic

### Ironing

- Enable ironing:
- Ironing Type:  Topmost surface only
- Flow rate:  15 %
- Spacing between ironing passes:  0.1 mm

### Reducing printing time

- Combine infill every:  1 layers
- Only infill where needed:

### Advanced

- Solid infill every:  0 layers
- Fill angle:  45 °
- Solid infill threshold area:  0 mm<sup>2</sup>
- Bridging angle:  0 °
- Only retract when crossing perimeters:
- Infill before perimeters:

0.45mm PLA 0.6mm Textured

**Support material**

- Generate support material:
- Auto generated supports:
- Overhang threshold:  °
- Enforce support for the first:
- First layer density:  %
- First layer expansion:  mm

---

**Raft**

- Raft layers:  layers
- Raft contact Z distance:  mm
- Raft expansion:  mm

---

**Options for support material and raft**

- Style:
- Top contact Z distance:  mm
- Bottom contact Z distance:  mm
- Pattern:
- With sheath around the support:
- Pattern spacing:  mm
- Pattern angle:  °
- Closing radius:  mm
- Top interface layers:  layers
- Bottom interface layers:  layers
- Interface pattern:
- Interface pattern spacing:  mm
- Interface loops:
- Support on build plate only:
- XY separation between an object and its support:  mm or %
- Don't support bridges:
- Synchronize with object layers:

0.45mm PLA 0.6mm Textured

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes
- Dependencies

---

### Speed for print moves

Perimeters:	• <input type="text" value="40"/> mm/s
Small perimeters:	• <input type="text" value="20"/> mm/s or %
External perimeters:	• <input type="text" value="15"/> mm/s or %
Infill:	• <input type="text" value="40"/> mm/s
Solid infill:	• <input type="text" value="40"/> mm/s or %
Top solid infill:	• <input type="text" value="20"/> mm/s or %
Support material:	• <input type="text" value="20"/> mm/s
Support material interface:	• <input type="text" value="20"/> mm/s or %
Bridges:	• <input type="text" value="40"/> mm/s
Gap fill:	• <input type="text" value="20"/> mm/s
Ironing:	• <input type="text" value="40"/> mm/s

---

### Speed for non-print moves

Travel:	• <input type="text" value="100"/> mm/s
Z travel:	• <input type="text" value="0"/> mm/s

---

### Modifiers

First layer speed:	• <input type="text" value="10"/> mm/s or %
Speed of object first layer over raft interface:	• <input type="text" value="30"/> mm/s or %

---

### Acceleration control (advanced)

Perimeters:	• <input type="text" value="500"/> mm/s <sup>2</sup>
Infill:	• <input type="text" value="1000"/> mm/s <sup>2</sup>
Bridge:	• <input type="text" value="1000"/> mm/s <sup>2</sup>
First layer:	• <input type="text" value="1000"/> mm/s <sup>2</sup>
First object layer over raft interface:	• <input type="text" value="0"/> mm/s <sup>2</sup>
Default:	• <input type="text" value="1000"/> mm/s <sup>2</sup>

---

### Auto Speed (advanced)

Max print speed:	• <input type="text" value="200"/> mm/s
Max volumetric speed:	• <input type="text" value="0"/> mm <sup>3</sup> /s

0.45mm PLA 0.6mm Textured

Layers and perimeters

Infill

Skirt and brim

Support material

Speed

Multiple Extruders

Advanced

Output options

Notes

Dependencies

### Extrusion width

- Default extrusion width:  mm or %
- First layer:  mm or %
- Perimeters:  mm or %
- External perimeters:  mm or %
- Infill:  mm or %
- Solid infill:  mm or %
- Top solid infill:  mm or %
- Support material:  mm or %

### Overlap

- Infill/perimeters overlap:  mm or %

### Flow

- Bridge flow ratio:

### Slicing

- Slice gap closing radius:  mm
- Slicing Mode:
- Slice resolution:  mm
- G-code resolution:  mm
- XY Size Compensation:  mm
- Elephant foot compensation:  mm

### Other

- Clip multi-part objects: