My Balsa & Glass Workshop

3D Modeling & Printing Information

Websites

Creality Website - https://www.creality.com/

Creality Print - https://www.crealitycloud.com/downloads/software/creality-print

Creality Ender-3 V3 – Best Cura Settings & Profile - https://the3dprinterbee.com/creality-ender-3-v3-cura-settings-profile/

Cults 3D Models - https://cults3d.com/en

Thingiverse 3D Models - https://www.thingiverse.com/

Printables 3D Models - 3D models database | Printables.com

MakerWorld 3D Models - https://makerworld.com/en

Thangs 3D Models - https://thangs.com/

Yeggi 3D Models - https://www.yeggi.com/

Autodesk Fusion 360 - https://www.autodesk.com/products/fusion-360/personal?

Good Video on using Variable Layer Heights in Creality Print - https://www.youtube.com/watch?v=jtbcYLDfg14&t=53s

These are the most important slicer settings for the Ender-3 V3:

Quality:
Layer Height: 0.2 mm
Line Width: 0.4 mm
The layer height should be between 25% and 75% of the nozzle diameter. For a 0.4 mm nozzle, this means between 0.1 and 0.3 mm.
The line width should be between 100% and 120% of the nozzle diameter. For a 0.4 mm nozzle, this means between 0.4 and 0.48 mm.

Walls:
Wall Line Count: 2-4
Wall Ordering: Inner/Outer/Inner
The wall thickness mainly depends on how stable the object has to be. More walls = more stability.

Top/Bottom:
Top Layers: 4-8
Bottom Layers: 4-8
Monotonic Top/Bottom Order: Yes
The same applies here: more top/bottom layers = more stability.

Infill:
Decorative objects: 0-15%
Standard objects: 15-50%
Stable objects: 50-100%
If you print thin walls, it can sometimes happen that the infill shows through from the inside. In this case, you should decrease the “Infill Overlap Percentage” settings.

Material:
Printing Temperature:
PLA: 215 °C
ABS: 260 °C
PETG: 250 °C
TPU: 225 °C
Build Plate Temperature:
PLA: 60 °C
ABS: 100 °C
PETG: 70 °C
TPU: 50 °C
The print temperature is one of the settings that should definitely be calibrated.
You can’t go too far wrong with the temperature of the build plate, as the print bed of the Ender-3 V3 offers excellent adhesion for most filaments.

Speed:
Print Speed: 250 mm/s
Initial Layer Speed: 20 mm/s
The considerably reduced speed of the first layer helps to increase the print bed adhesion.

Travel:
Retraction Distance:
PLA: 0.7 mm
ABS: 0.7 mm
PETG: 0.8 mm
TPU: 0.8 mm
Retraction Speed:
PLA: 40 mm/s
ABS: 40 mm/s
PETG: 45 mm/s
TPU: 45 mm/s
Z-Hop When Retracted: Yes
Z-Hop Only Over Printed Parts: Yes
Z-Hop Height: 0.6 mm
Retraction is also one of the settings that must be calibrated. Otherwise, stringing will quickly occur.

Cooling:
Fan Speed:
PLA: 100%
ABS: 50%
PETG: 100%
TPU: 100%
Minimum Layer Time: 8 s
Cooling only needs to be reduced for ABS. ABS has a strong tendency to warp as it shrinks during cooling. As the Ender-3 V3 does not have a closed print volume, it can only process ABS to a limited extent. However, smaller objects are certainly possible.
If you cannot avoid drafts, a draft shield could be an option.

Build Plate Adhesion:
Build Plate Adhesion Type: Skirt
As already mentioned, the print bed adhesion of the Ender-3 V3 is excellent. It is therefore only necessary to print a Brim or Raft in very few cases.

Here is a table of the 4 most common Ender 3 filaments to see all settings at a glance:

	PLA	ABS	PETG	TPU
Extruder Temperature	180 – 230 °C	220 – 240 °C	230 – 250 °C	220 – 240 °C
Printing Bed Temperature	40 – 60 °C	80 – 100 °C	70 – 90 °C	40 – 60 °C
Print Speed	max. 60 mm/s	max. 60 mm/s	max. 60 mm/s	max. 30 mm/s
Retraction Distance	5 mm	5 mm	5 – 7 mm	as small as possible
Retraction Speed	45 mm/s	45 mm/s	min. 40 mm/s	max. 30 mm/s
Cooling	Yes	No	Yes	Yes

Creality Ender-3 V3
Best Slicer Settings for Ultra-Fine Detail Printing using 0.2mm Nozzle

These parameters form the core of your ultra-fine detail configuration.

Layer Height Best Setting: 0.05 - 0.1 mm
Why It Matters:
A 0.2mm nozzle excels at ultra-thin layers. Printing at 25 – 50% of the nozzle diameter ensures strong layer bonding and high resolution. Avoid using a layer height higher than 0.12 mm.
Typical Use:
0.05 mm for extremely fine surface detail (miniatures, figurines)
0.08 – 0.1 mm for detailed mechanical models

Line Width Best Setting: 0.2 - 0.24 mm
Why It Matters:
Setting line width too narrow causes under-extrusion; too wide causes flow inconsistencies. Most slicers automatically set line width equal to the nozzle diameter, but you can adjust it slightly for better surface control.
Recommendation:
Start with a line width of 0.22 mm for perimeter and wall printing.

Print Speed Best Setting: 10 – 30 mm/s
Why It Matters:
Fine nozzles cannot push filament as fast as standard nozzles. Printing too quickly causes gaps, missed steps, or uneven extrusion. Slower speeds provide better detail and prevent vibrations from affecting accuracy.
Suggested Speeds:
Walls/perimeters: 15 – 25 mm/s
Infill: 25 – 30 mm/s
Small details and features: 10 – 15 mm/s

Retraction Settings
Recommended Settings: Distance: 1 – 2 mm for direct drive; Speed: 20 – 40 mm/s
Why It Matters:
Because 0.2mm nozzles are narrow, they are more prone to stringing and clogs. Precise retraction helps reduce oozing, but overly aggressive retraction can cause jams or delay re-priming.
Test retraction carefully with small models before larger prints.

Cooling Fan Speed Recommended Setting: PLA: 80 – 100%; PETG: 50 – 60%
Why It Matters:
Fine nozzles lay down less filament per second, so cooling becomes more important to solidify small features. Too much cooling can cause warping on high-temp filaments, while too little cooling causes stringing and sagging.

Extrusion Multiplier / Flow Rate Best Setting: 95 – 100% (start with 100%)
Why It Matters:

Under- or over-extrusion affects layer adhesion and print quality more noticeably with fine nozzles. Perform flow calibration tests using a single-wall cube to fine-tune this value.

First Layer Settings

Recommended Settings: First layer height: 0.15 – 0.2 mm; First layer speed: 10 – 15 mm/s

Z-offset calibration: Crucial due to narrow gap

Why It Matters:

The first layer must be accurate and uniform. With a 0.2mm nozzle, improper bed leveling causes nozzle drag, uneven extrusion, or poor adhesion.

Calibrate the Z-offset carefully and use a clean, level bed surface.

Advanced Tips for Ultra-Fine Detail Printing

Use Adaptive Layer Height: Reduces print time by using finer layers in detailed areas and thicker layers elsewhere.

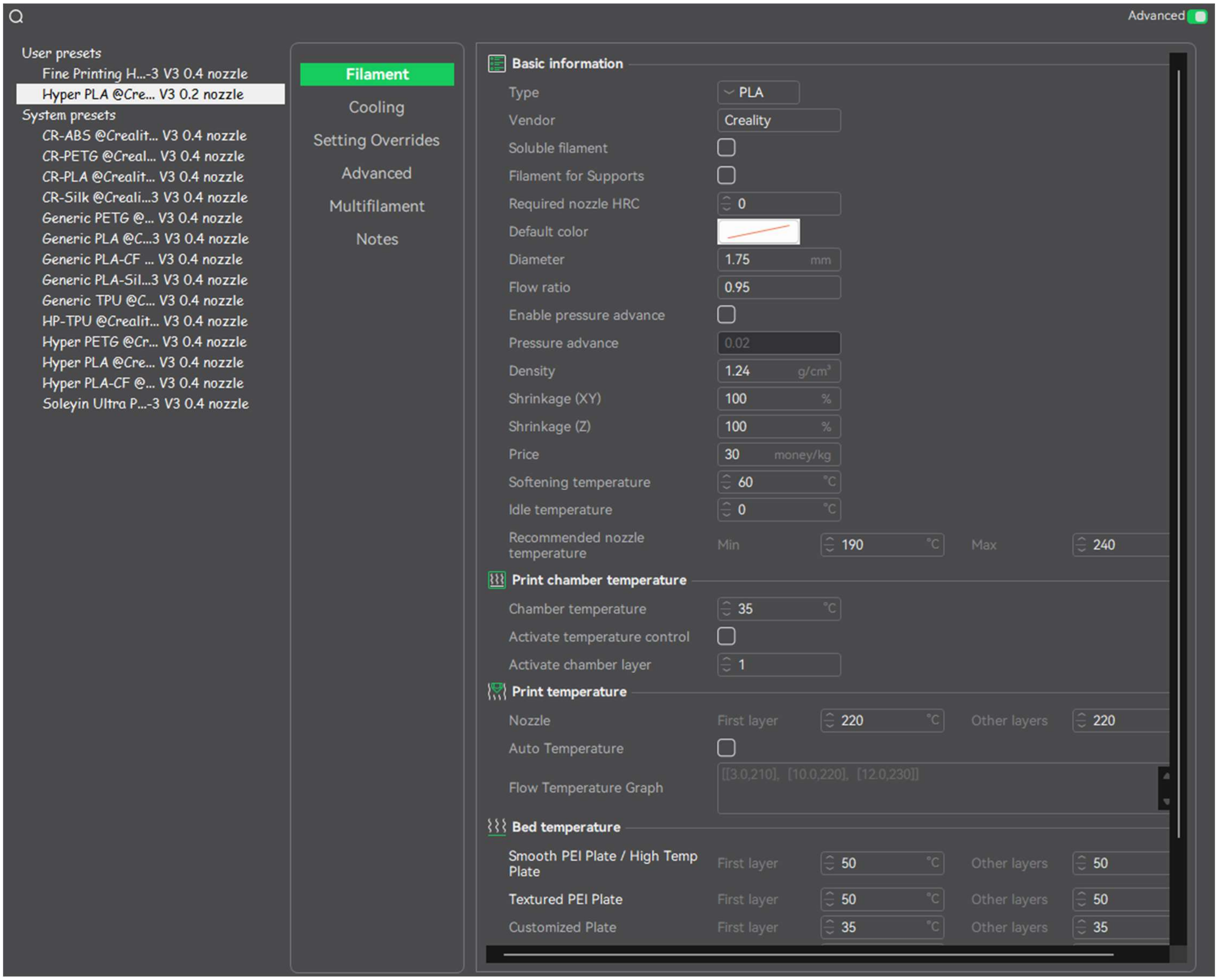
Print at Cooler Temperatures for PLA: Helps reduce stringing and drooping on small features.

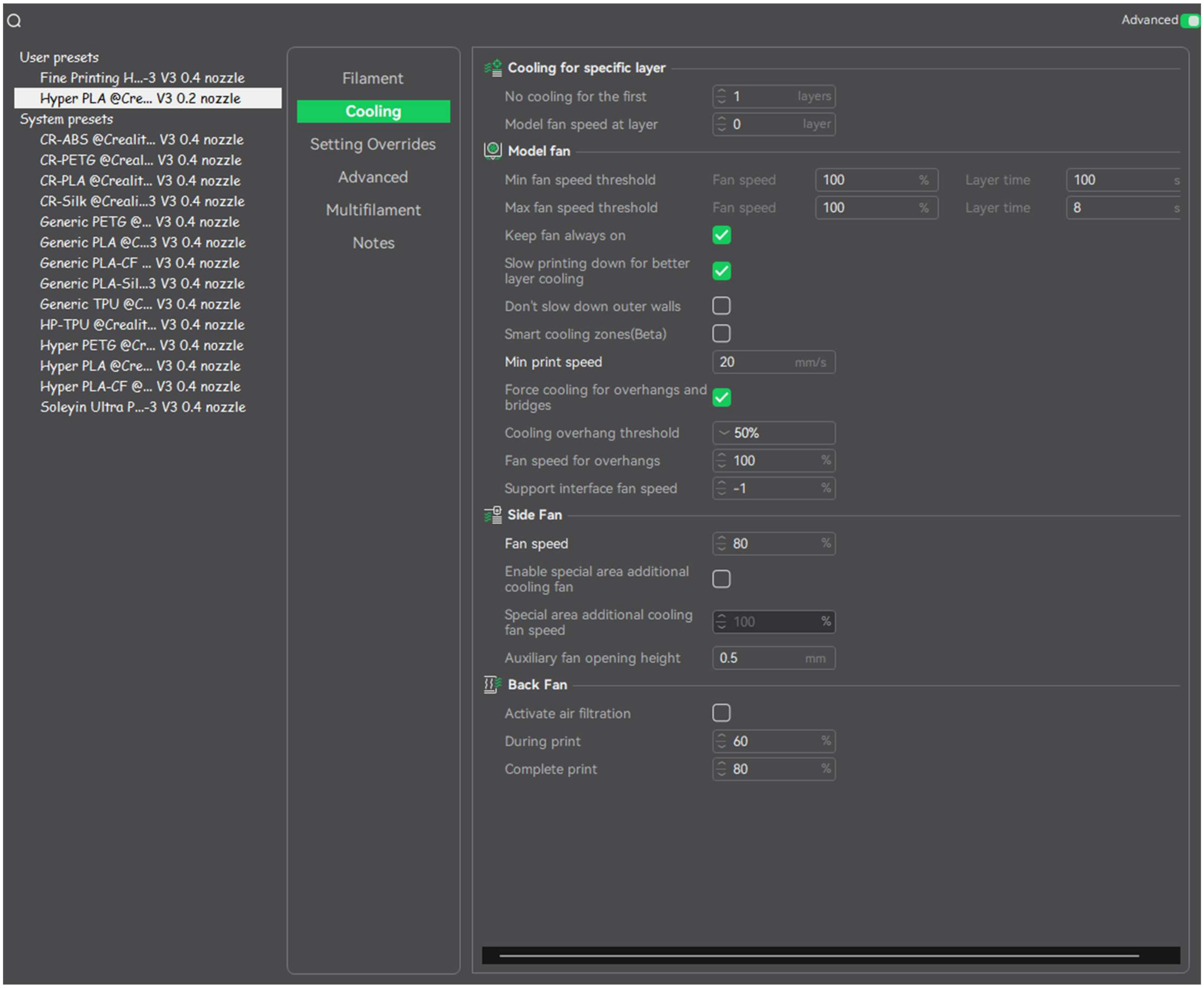
Enable "Outer Walls First" in Slicer: Improves surface accuracy by printing visible areas before internal ones.

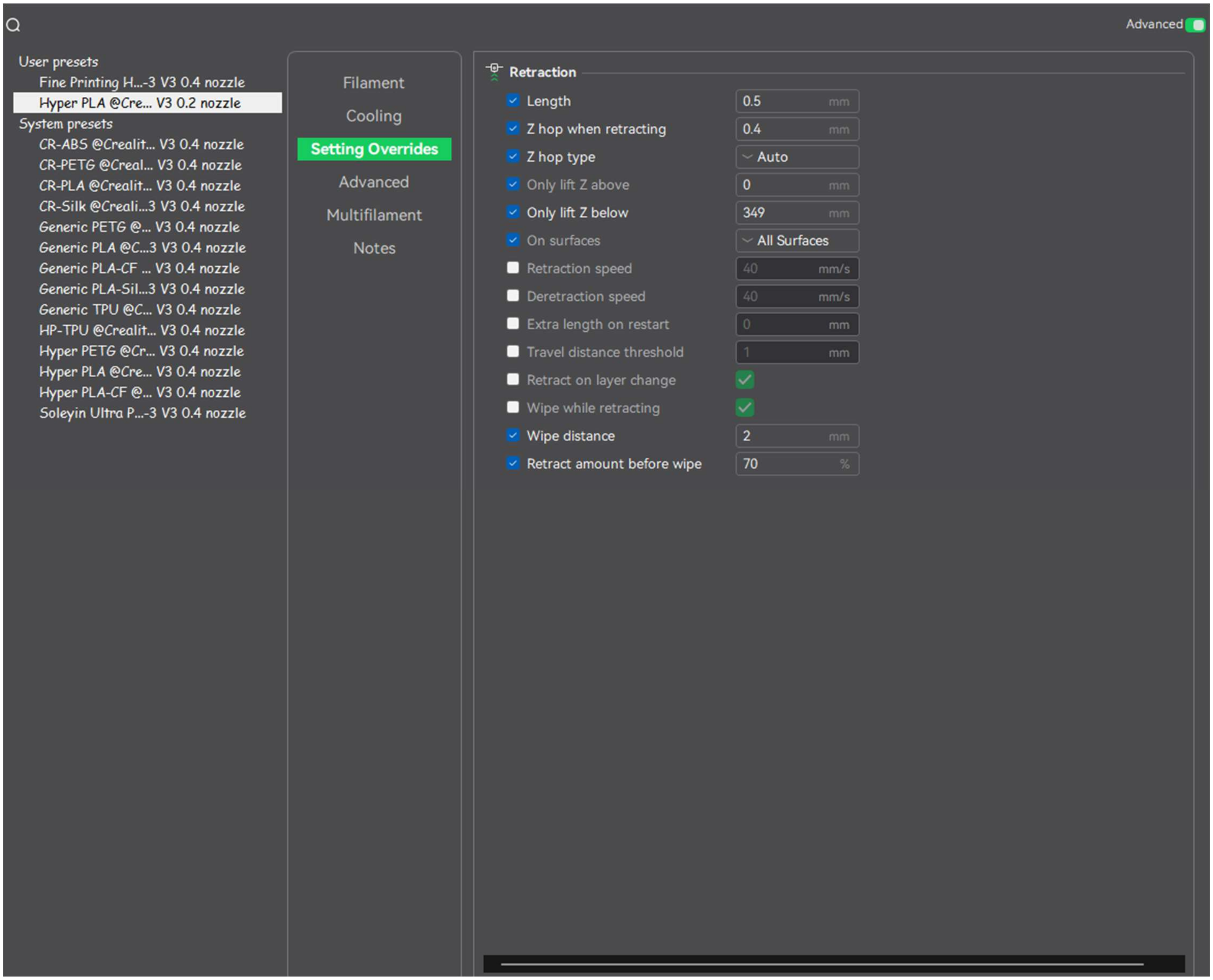
Reduce Acceleration and Jerk Settings: Lowers vibrations and enhances fine feature consistency.

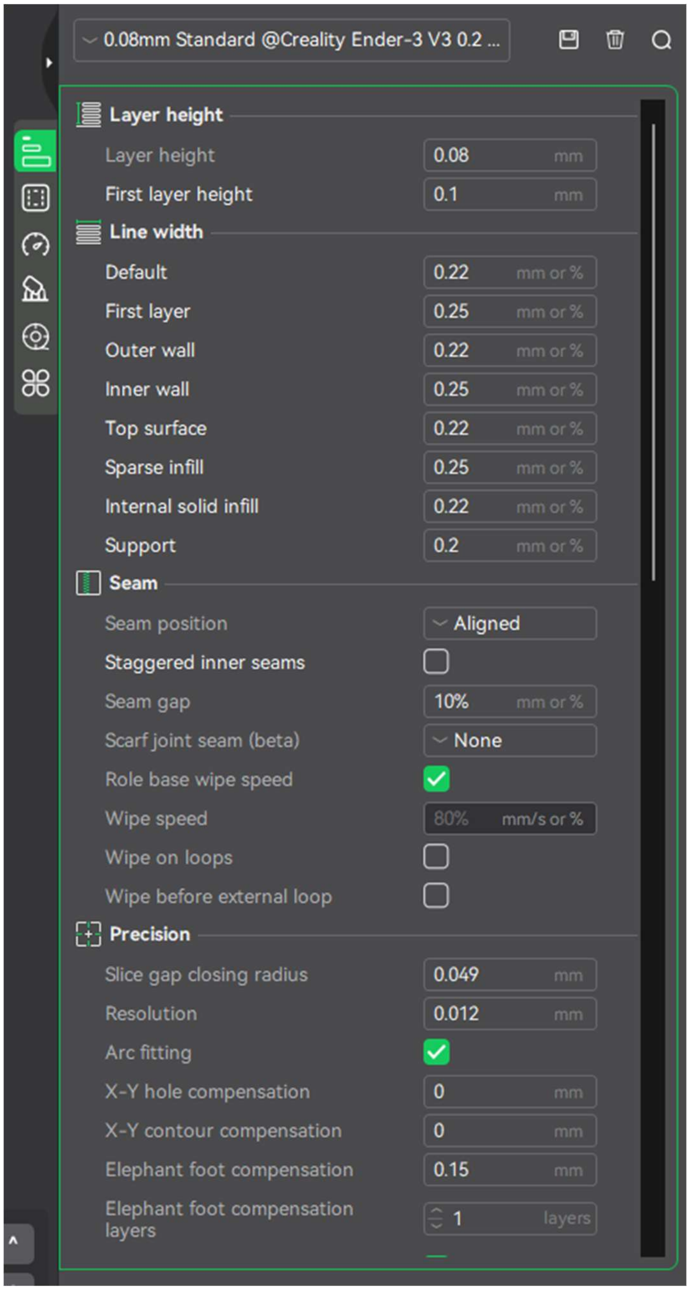
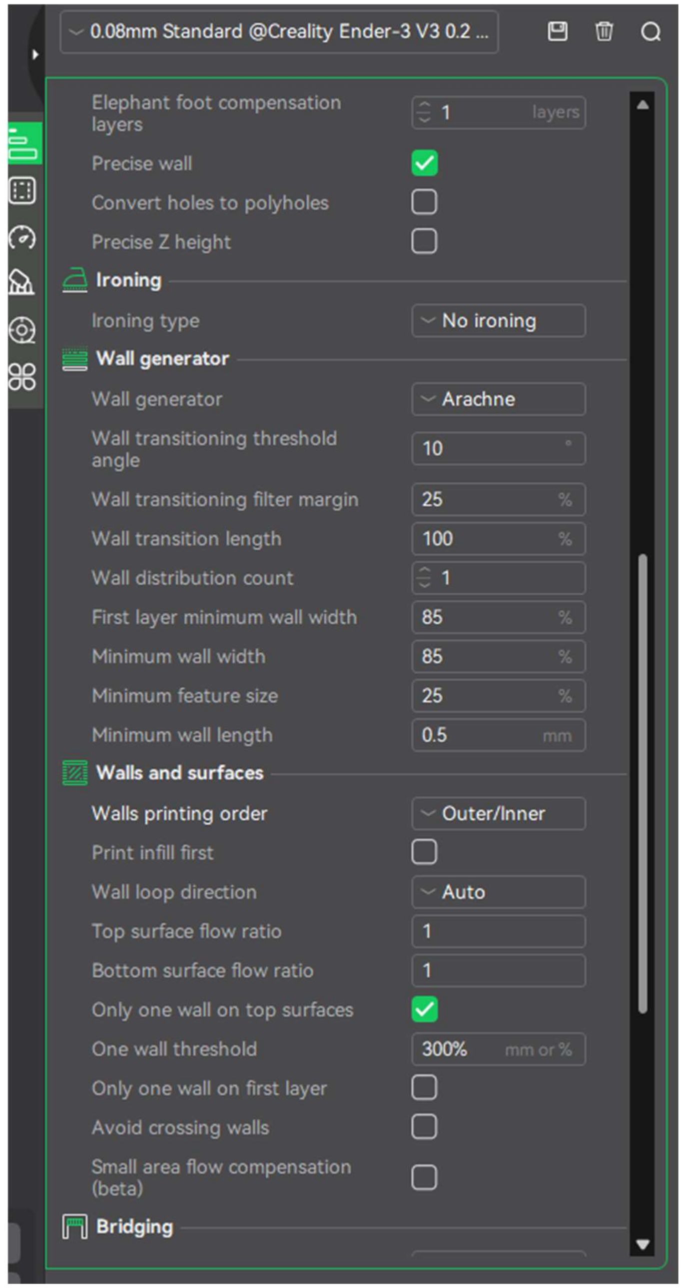
Use Smaller Text Sizes Effectively: A 0.2mm nozzle can cleanly render text as small as 3–4 mm in height when properly tuned.

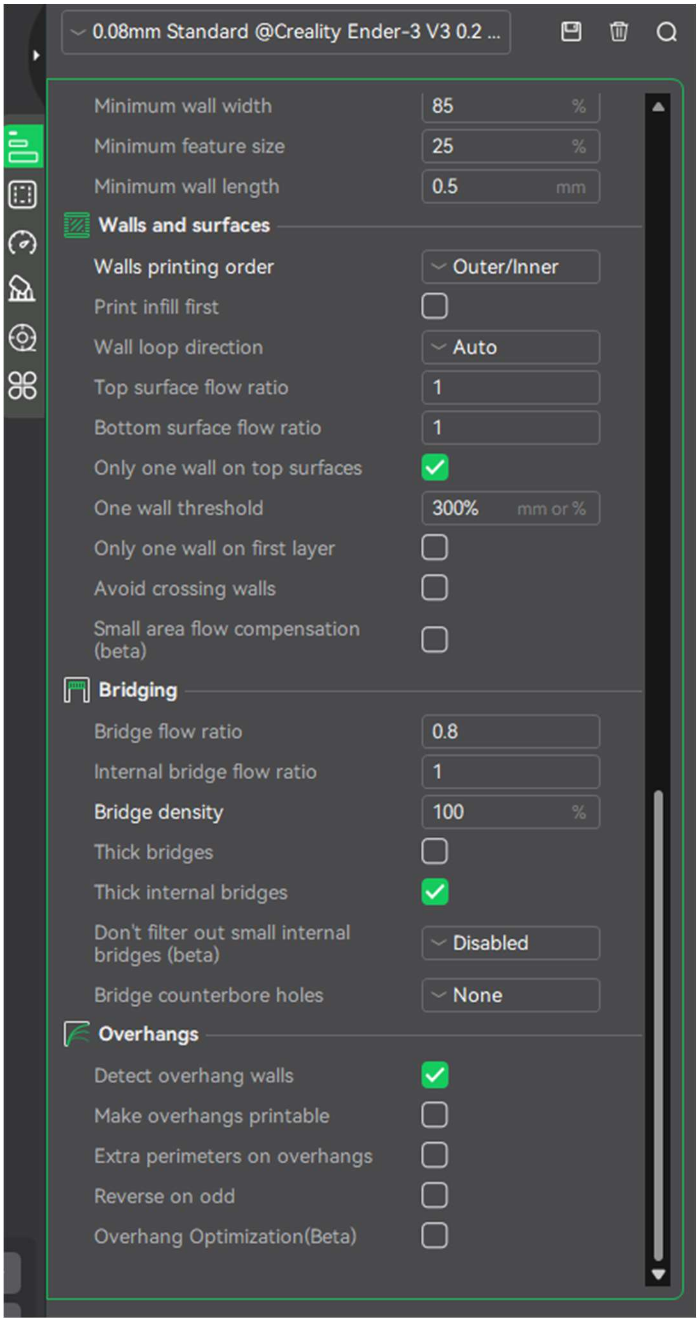
Creality Print 6.2 Settings for Ender-3 V3 with Hyper-PLA using 0.2mm Nozzle

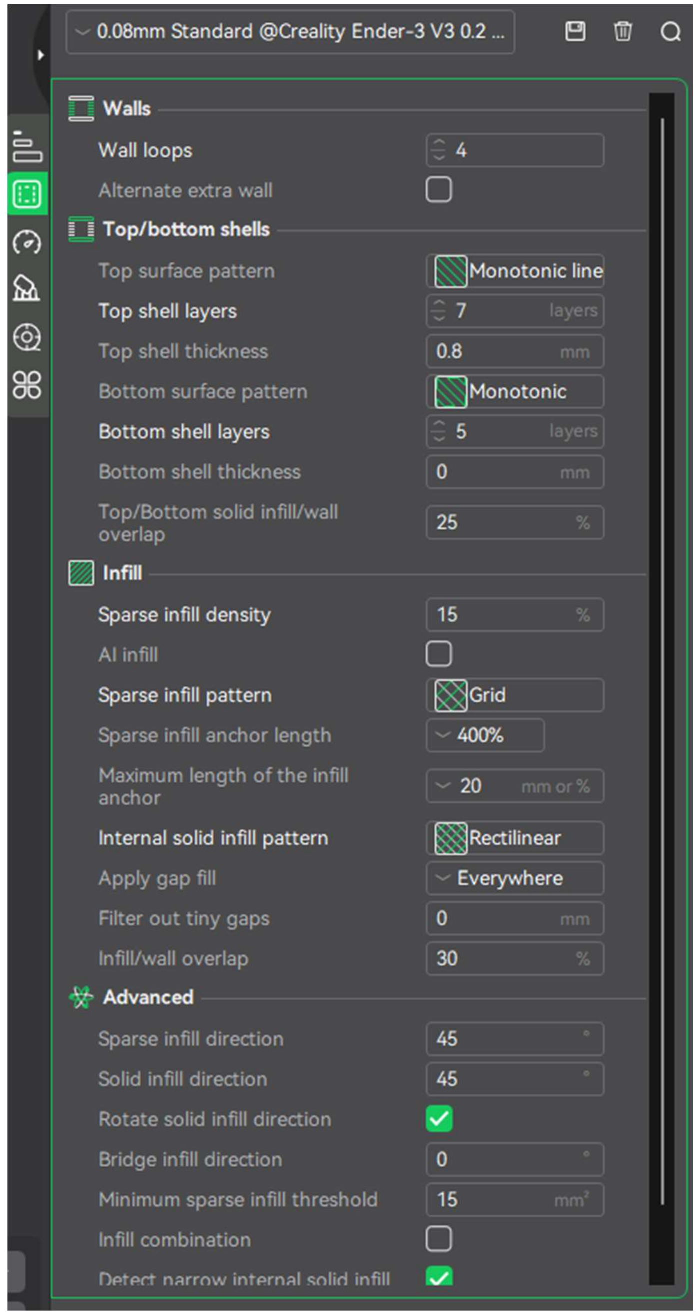


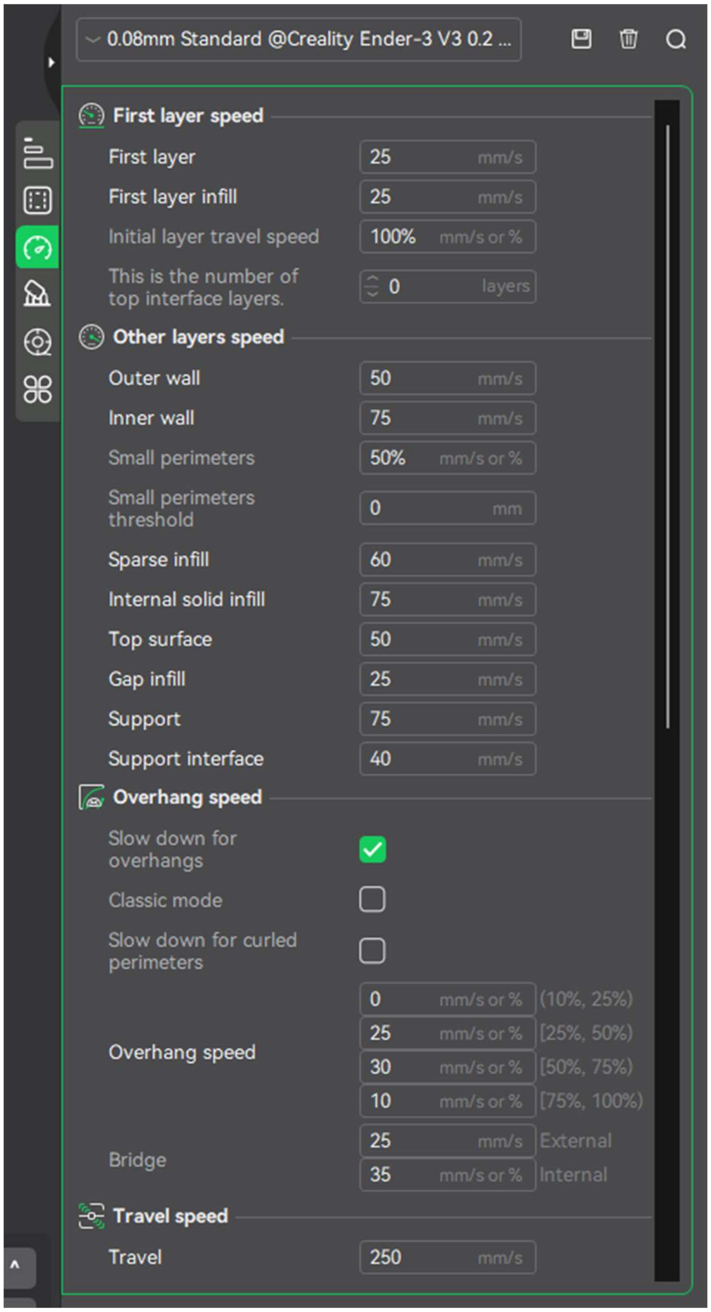

Advanced
User presets
Fine Printing H...-3 V3 0.4 nozzle
Hyper PLA @Cre... V3 0.2 nozzle
System presets
CR-ABS @Crealit... V3 0.4 nozzle
CR-PETG @Creal... V3 0.4 nozzle
CR-PLA @Crealit... V3 0.4 nozzle
CR-Silk @Creali...3 V3 0.4 nozzle
Generic PETG @... V3 0.4 nozzle
Generic PLA @C...3 V3 0.4 nozzle
Generic PLA-CF ... V3 0.4 nozzle
Generic PLA-Sil...3 V3 0.4 nozzle
Generic TPU @C... V3 0.4 nozzle
HP-TPU @Crealit... V3 0.4 nozzle
Hyper PETG @Cr... V3 0.4 nozzle
Hyper PLA @Cre... V3 0.4 nozzle
Hyper PLA-CF @... V3 0.4 nozzle
Soleyin Ultra P...-3 V3 0.4 nozzle
Filament
Cooling
Setting Overrides
Advanced
Multifilament
Notes
Cooling for specific layer
No cooling for the first
1
layers
Model fan speed at layer
0
layer
Model fan
Min fan speed threshold
Fan speed
100
%
Layer time
100
s
Max fan speed threshold
Fan speed
100
%
Layer time
8
s
Keep fan always on
Slow printing down for better layer cooling
Don't slow down outer walls
Smart cooling zones(Beta)
Min print speed
20
mm/s
Force cooling for overhangs and bridges
Cooling overhang threshold
50%
Fan speed for overhangs
100
%
Support interface fan speed
-1
%
Side Fan
Fan speed
80
%
Enable special area additional cooling fan
Special area additional cooling fan speed
100
%
Auxiliary fan opening height
0.5
mm
Back Fan
Activate air filtration
During print
60
%
Complete print
80
%

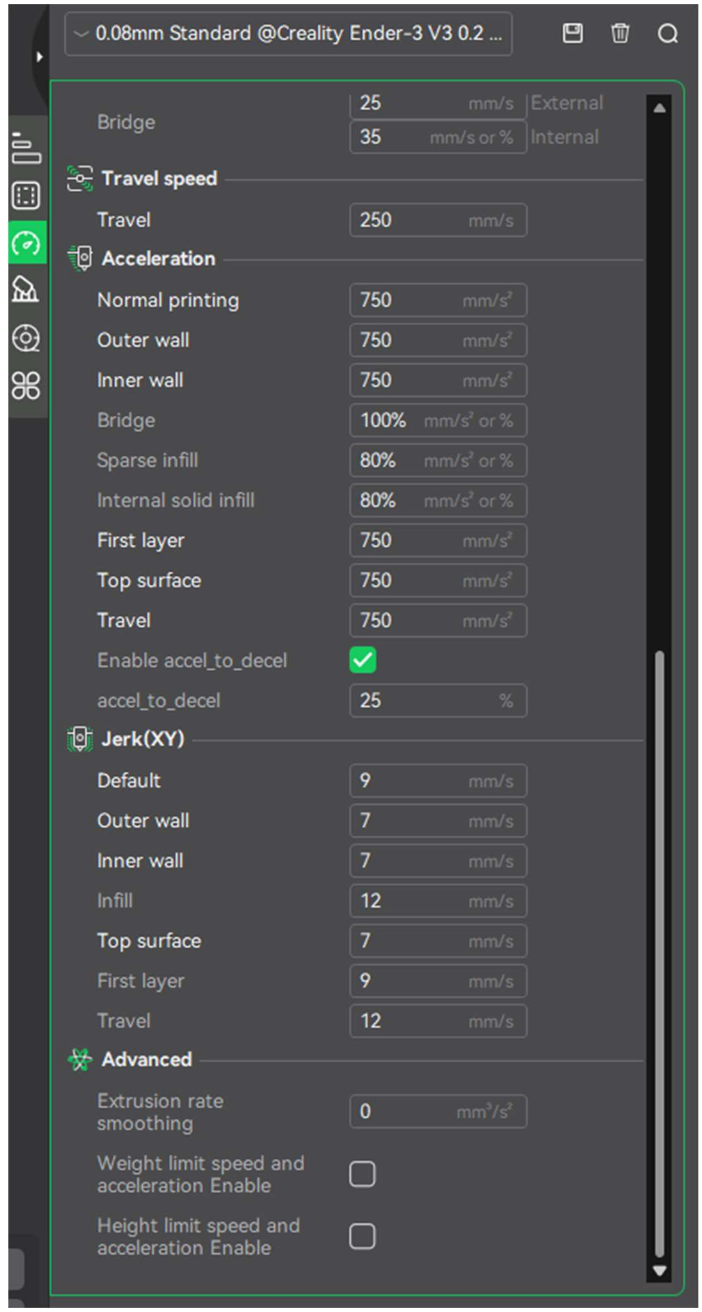

Advanced
User presets
Fine Printing H...-3 V3 0.4 nozzle
Hyper PLA @Cre... V3 0.2 nozzle
System presets
CR-ABS @Crealit... V3 0.4 nozzle
CR-PETG @Creal... V3 0.4 nozzle
CR-PLA @Crealit... V3 0.4 nozzle
CR-Silk @Creali...3 V3 0.4 nozzle
Generic PETG @... V3 0.4 nozzle
Generic PLA @C...3 V3 0.4 nozzle
Generic PLA-CF ... V3 0.4 nozzle
Generic PLA-Sil...3 V3 0.4 nozzle
Generic TPU @C... V3 0.4 nozzle
HP-TPU @Crealit... V3 0.4 nozzle
Hyper PETG @Cr... V3 0.4 nozzle
Hyper PLA @Cre... V3 0.4 nozzle
Hyper PLA-CF @... V3 0.4 nozzle
Soleyin Ultra P...-3 V3 0.4 nozzle
Filament
Cooling
Setting Overrides
Advanced
Multifilament
Notes
Retraction
Length 0.5 mm
Z hop when retracting 0.4 mm
Z hop type Auto
Only lift Z above 0 mm
Only lift Z below 349 mm
On surfaces All Surfaces
Retraction speed 40 mm/s
Deretraction speed 40 mm/s
Extra length on restart 0 mm
Travel distance threshold 1 mm
Retract on layer change
Wipe while retracting
Wipe distance 2 mm
Retract amount before wipe 70 %

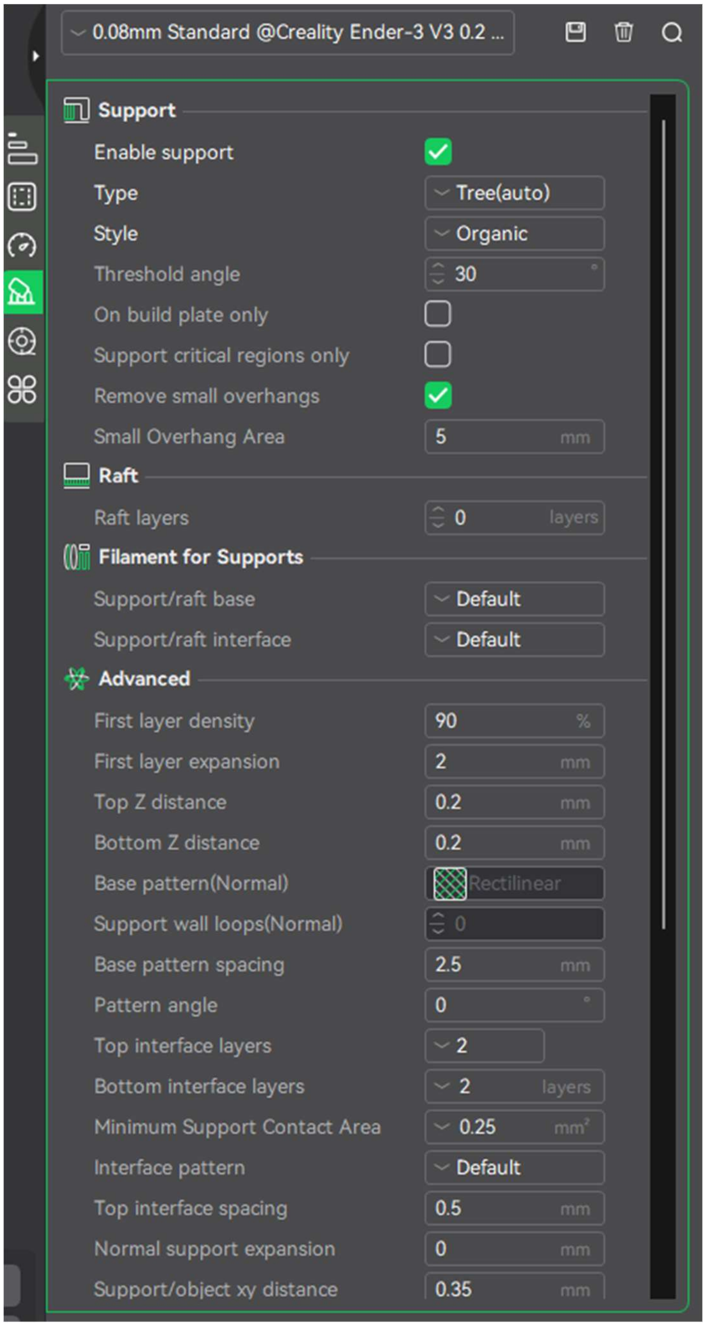
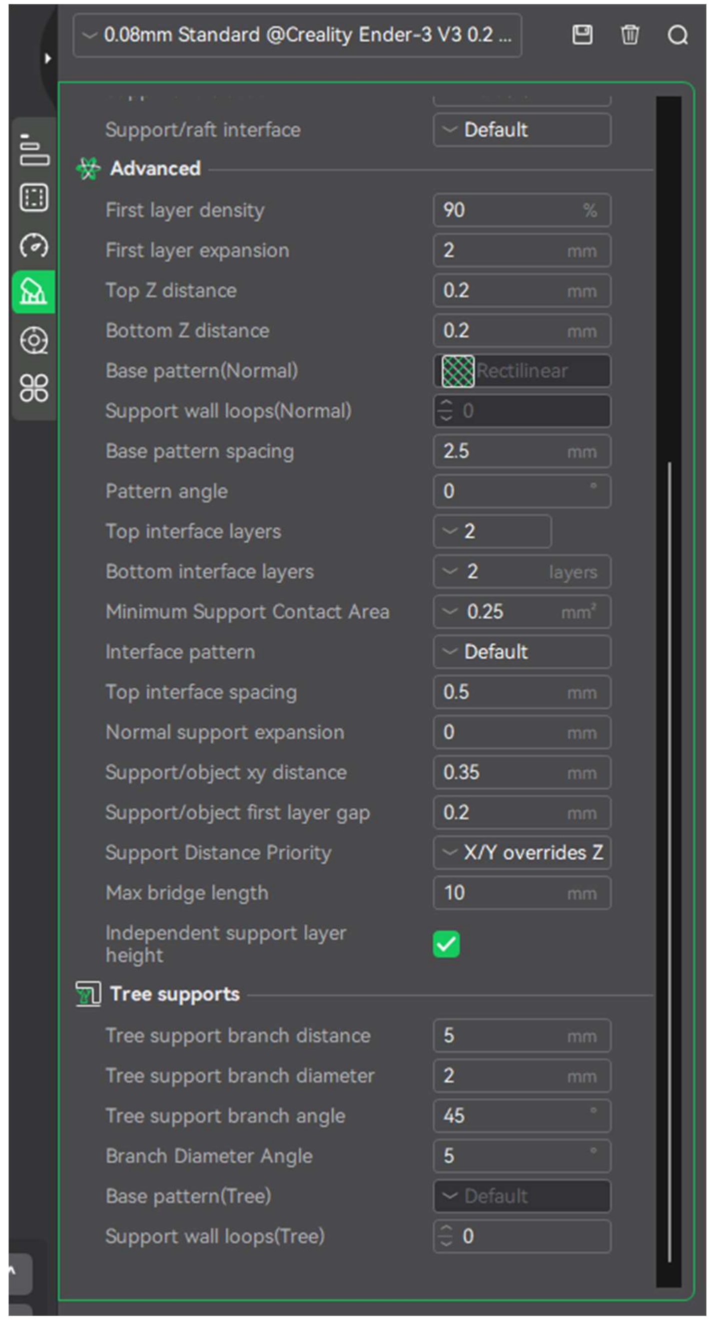

0.08mm Standard @Creality Ender-3 V3 0.2 ...
Layer height
Layer height 0.08 mm
First layer height 0.1 mm
Line width
Default 0.22 mm or %
First layer 0.25 mm or %
Outer wall 0.22 mm or %
Inner wall 0.25 mm or %
Top surface 0.22 mm or %
Sparse infill 0.25 mm or %
Internal solid infill 0.22 mm or %
Support 0.2 mm or %
Seam
Seam position Aligned
Staggered inner seams
Seam gap 10% mm or %
Scarf joint seam (beta) None
Role base wipe speed
Wipe speed 80% mm/s or %
Wipe on loops
Wipe before external loop
Precision
Slice gap closing radius 0.049 mm
Resolution 0.012 mm
Arc fitting
X-Y hole compensation 0 mm
X-Y contour compensation 0 mm
Elephant foot compensation 0.15 mm
Elephant foot compensation layers 1 layers

0.08mm Standard @Creality Ender-3 V3 0.2 ...
Elephant foot compensation layers 1 layers
Precise wall
Convert holes to polyholes
Precise Z height
Ironing
Ironing type No ironing
Wall generator
Wall generator Arachne
Wall transitioning threshold angle 10 °
Wall transitioning filter margin 25 %
Wall transition length 100 %
Wall distribution count 1
First layer minimum wall width 85 %
Minimum wall width 85 %
Minimum feature size 25 %
Minimum wall length 0.5 mm
Walls and surfaces
Walls printing order Outer/Inner
Print infill first
Wall loop direction Auto
Top surface flow ratio 1
Bottom surface flow ratio 1
Only one wall on top surfaces
One wall threshold 300% mm or %
Only one wall on first layer
Avoid crossing walls
Small area flow compensation (beta)
Bridging

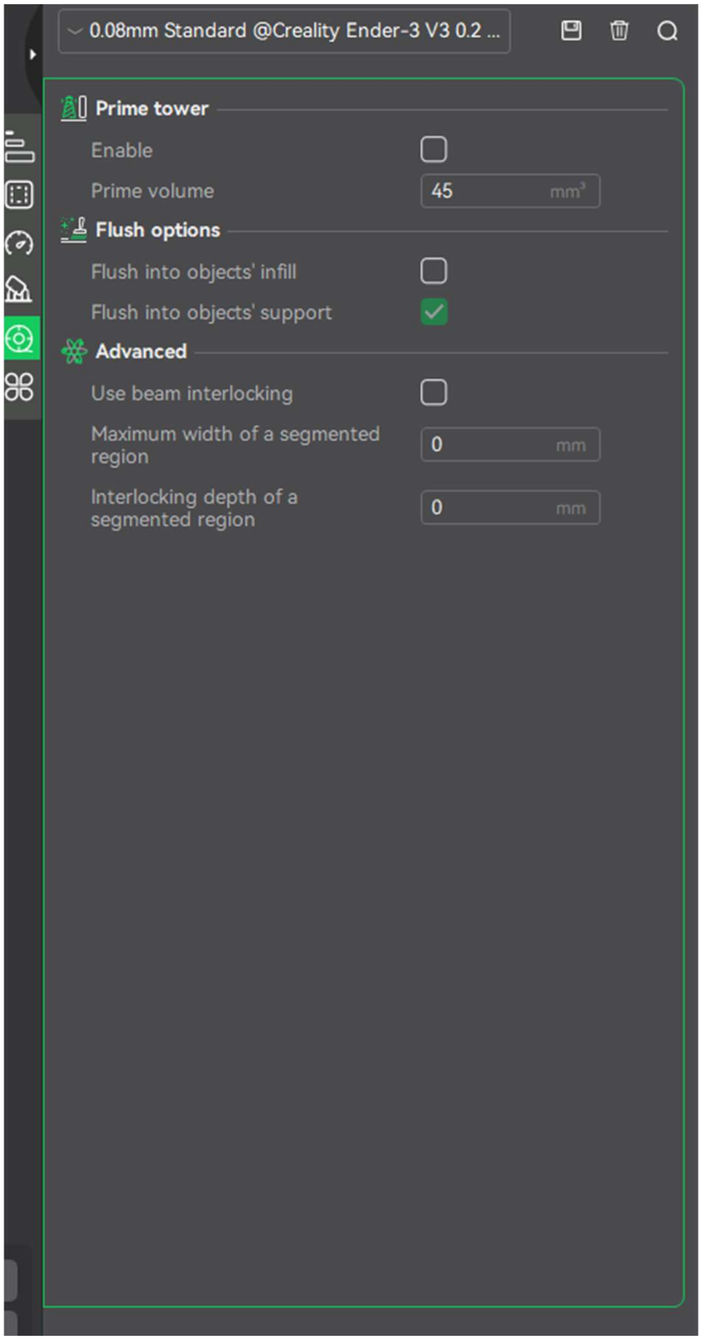

0.08mm Standard @Creality Ender-3 V3 0.2 ...
Minimum wall width
85 %
Minimum feature size
25 %
Minimum wall length
0.5 mm
Walls and surfaces
Walls printing order
Outer/Inner
Print infill first
Wall loop direction
Auto
Top surface flow ratio
1
Bottom surface flow ratio
1
Only one wall on top surfaces
One wall threshold
300% mm or %
Only one wall on first layer
Avoid crossing walls
Small area flow compensation (beta)
Bridging
Bridge flow ratio
0.8
Internal bridge flow ratio
1
Bridge density
100 %
Thick bridges
Thick internal bridges
Don't filter out small internal bridges (beta)
Disabled
Bridge counterbore holes
None
Overhangs
Detect overhang walls
Make overhangs printable
Extra perimeters on overhangs
Reverse on odd
Overhang Optimization(Beta)

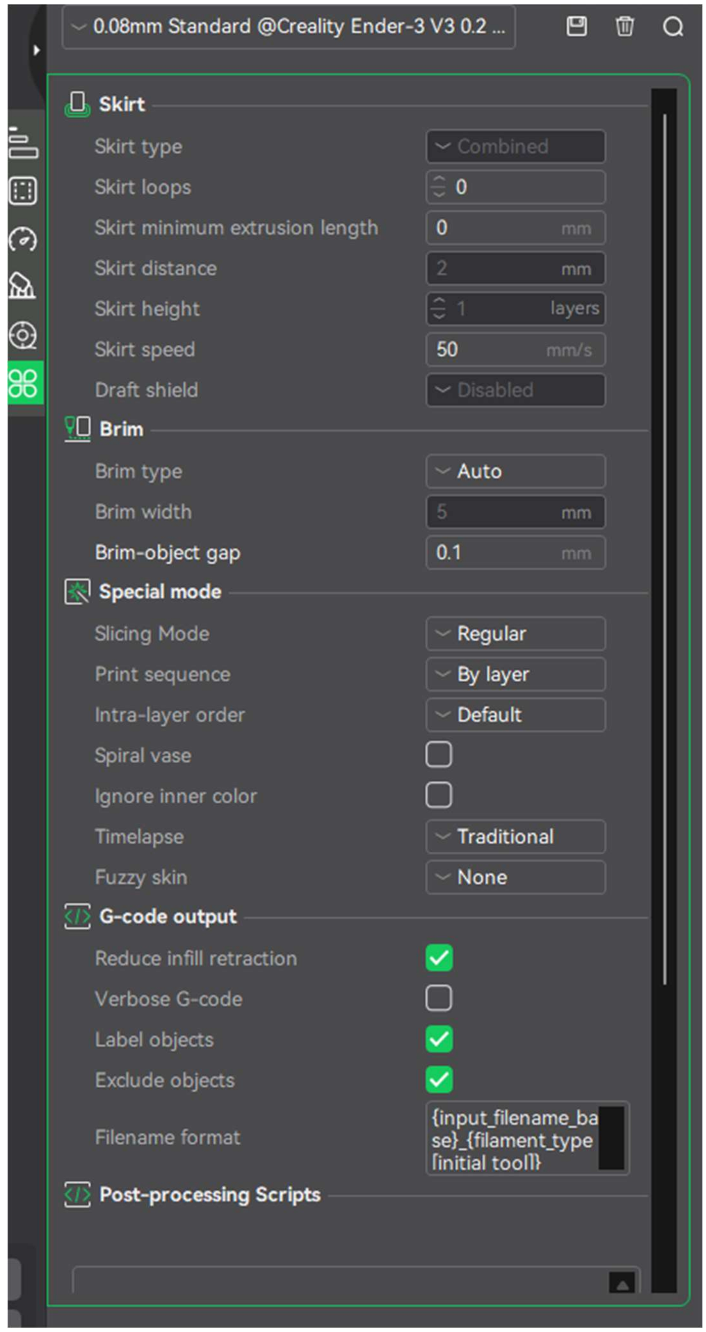

0.08mm Standard @Creality Ender-3 V3 0.2 ...
Walls
Wall loops
4
Alternate extra wall
Top/bottom shells
Top surface pattern
Monotonic line
Top shell layers
7 layers
Top shell thickness
0.8 mm
Bottom surface pattern
Monotonic
Bottom shell layers
5 layers
Bottom shell thickness
0 mm
Top/Bottom solid infill/wall overlap
25 %
Infill
Sparse infill density
15 %
AI infill
Sparse infill pattern
Grid
Sparse infill anchor length
400%
Maximum length of the infill anchor
20 mm or %
Internal solid infill pattern
Rectilinear
Apply gap fill
Everywhere
Filter out tiny gaps
0 mm
Infill/wall overlap
30 %
Advanced
Sparse infill direction
45 °
Solid infill direction
45 °
Rotate solid infill direction
Bridge infill direction
0 °
Minimum sparse infill threshold
15 mm²
Infill combination
Detect narrow internal solid infill


0.08mm Standard @Creality Ender-3 V3 0.2 ...
First layer speed
First layer 25 mm/s
First layer infill 25 mm/s
Initial layer travel speed 100% mm/s or %
This is the number of top interface layers. 0 layers
Other layers speed
Outer wall 50 mm/s
Inner wall 75 mm/s
Small perimeters 50% mm/s or %
Small perimeters threshold 0 mm
Sparse infill 60 mm/s
Internal solid infill 75 mm/s
Top surface 50 mm/s
Gap infill 25 mm/s
Support 75 mm/s
Support interface 40 mm/s
Overhang speed
Slow down for overhangs
Classic mode
Slow down for curled perimeters
Overhang speed
0 mm/s or % (10%, 25%)
25 mm/s or % [25%, 50%)
30 mm/s or % [50%, 75%)
10 mm/s or % [75%, 100%)
Bridge
25 mm/s External
35 mm/s or % Internal
Travel speed
Travel 250 mm/s


0.08mm Standard @Creality Ender-3 V3 0.2 ...
Bridge
25 mm/s External
35 mm/s or % Internal
Travel speed
Travel 250 mm/s
Acceleration
Normal printing 750 mm/s²
Outer wall 750 mm/s²
Inner wall 750 mm/s²
Bridge 100% mm/s² or %
Sparse infill 80% mm/s² or %
Internal solid infill 80% mm/s² or %
First layer 750 mm/s²
Top surface 750 mm/s²
Travel 750 mm/s²
Enable accel_to_decel
accel_to_decel 25 %
Jerk(XY)
Default 9 mm/s
Outer wall 7 mm/s
Inner wall 7 mm/s
Infill 12 mm/s
Top surface 7 mm/s
First layer 9 mm/s
Travel 12 mm/s
Advanced
Extrusion rate smoothing 0 mm³/s²
Weight limit speed and acceleration Enable
Height limit speed and acceleration Enable


0.08mm Standard @Creality Ender-3 V3 0.2 ...
Support
Enable support
Type
Tree(auto)
Style
Organic
Threshold angle
30
On build plate only
Support critical regions only
Remove small overhangs
Small Overhang Area
5 mm
Raft
Raft layers
0 layers
Filament for Supports
Support/raft base
Default
Support/raft interface
Default
Advanced
First layer density
90 %
First layer expansion
2 mm
Top Z distance
0.2 mm
Bottom Z distance
0.2 mm
Base pattern(Normal)
Rectilinear
Support wall loops(Normal)
0
Base pattern spacing
2.5 mm
Pattern angle
0 °
Top interface layers
2
Bottom interface layers
2 layers
Minimum Support Contact Area
0.25 mm²
Interface pattern
Default
Top interface spacing
0.5 mm
Normal support expansion
0 mm
Support/object xy distance
0.35 mm

0.08mm Standard @Creality Ender-3 V3 0.2 ...
Support/raft interface
Default
Advanced
First layer density
90 %
First layer expansion
2 mm
Top Z distance
0.2 mm
Bottom Z distance
0.2 mm
Base pattern(Normal)
Rectilinear
Support wall loops(Normal)
0
Base pattern spacing
2.5 mm
Pattern angle
0 °
Top interface layers
2
Bottom interface layers
2 layers
Minimum Support Contact Area
0.25 mm²
Interface pattern
Default
Top interface spacing
0.5 mm
Normal support expansion
0 mm
Support/object xy distance
0.35 mm
Support/object first layer gap
0.2 mm
Support Distance Priority
X/Y overrides Z
Max bridge length
10 mm
Independent support layer height
Tree supports
Tree support branch distance
5 mm
Tree support branch diameter
2 mm
Tree support branch angle
45 °
Branch Diameter Angle
5 °
Base pattern(Tree)
Default
Support wall loops(Tree)
0


0.08mm Standard @Creality Ender-3 V3 0.2 ...
Prime tower
Enable
Prime volume
45
mm³
Flush options
Flush into objects' infill
Flush into objects' support
Advanced
Use beam interlocking
Maximum width of a segmented region
0
mm
Interlocking depth of a segmented region
0
mm


0.08mm Standard @Creality Ender-3 V3 0.2 ...
Skirt
Skirt type
Combined
Skirt loops
0
Skirt minimum extrusion length
0
mm
Skirt distance
2
mm
Skirt height
1
layers
Skirt speed
50
mm/s
Draft shield
Disabled
Brim
Brim type
Auto
Brim width
5
mm
Brim-object gap
0.1
mm
Special mode
Slicing Mode
Regular
Print sequence
By layer
Intra-layer order
Default
Spiral vase
Ignore inner color
Timelapse
Traditional
Fuzzy skin
None
G-code output
Reduce infill retraction
Verbose G-code
Label objects
Exclude objects
Filename format
{input_filename_base}_{filament_type[initial tool]}
Post-processing Scripts

Ender-3 V3 Nozzle Replacement Step-by-Step

Select "Movement/Temp" - "Z-axis origin", wait for the machine homing; Select "30mm" - "Z-axis up", and raise the Z-axis to middle position (120mm).

Select "Extrude/Retract" – "Retract", wait for the machine to heat up to 220 °C, complete the filament unloading; "Open" the filament clamp and remove the Teflon tube and filament.

Select "Movement/Temp" – "Temp", set temperature to "0"; Select "Cooling" – turn on the "Auxiliary Cooling" - "Model Cooling", and adjust both to "100%", wait for the temperature to drop to room temperature.

Remove the two 5/64" hex head screws from the fan cover (one on each side), disassemble the fan cover, and unplug fan motor from printer head circuit board; now remove the silicone sleeve from around the nozzle at the bottom of extruder.

On control panel "Home" screen set nozzle temperature to 220 °C and press "OK", wait for the temperature to rise to the set value.

Loosen the nozzle throat 5/64" hex head set screw on right side of extruder head; using the nozzle socket wrench turn counterclockwise to remove the old nozzle (**caution, the nozzle will be very hot**).

Apply thermal grease to nozzle smooth center copper sleeve, spread the thermal grease evenly; insert the new nozzle into the bottom of the extruder and using the nozzle socket wrench turn clockwise to install the new nozzle (do not overtighten) now tighten the nozzle throat hex head set screw.

On control panel "Home" screen set nozzle temperature to 0 °C and press "OK"; Select "Cooling" – turn on the "Auxiliary Cooling", and adjust to "100%", wait for the temperature to drop to room temperature.

Install silicone sleeve back over printer nozzle on bottom of extruder head; plug fan motor back into printer head circuit board and reinstall the fan cover using the two hex head screws removed earlier.

Reload the filament and "Close" the filament clamp; Select "Extrude/Retract" – "Extrude", wait for the machine to heat up to 220 °C, observe whether the nozzle discharges smoothly.

Perform a self-test of the printer; Select "System Icon" – "Self-Check" – "Select Input Shaping and Auto Leveling" – "Start Detecting"; and wait for machine self-test to complete.

Printing with PLA

PLA, the "polylactic acid", is the most widely used printing filament of all. That is why it's processing on the Ender 3 is particularly problem-free. In fact, the 3D printer is optimized to use PLA.

When printing PLA it is important that the printer is not enclosed. Even if you like it "neat" and have built a nice housing for your printer, this can cause problems when printing PLA.

The print data for PLA are as follows:
Extruder temperature: 190 – 220 °C
Print speed: 60 mm/s (recommended value)
Retraction distance: 5 mm
Retraction Speed: 45 mm/s
Cooling: required
These basic technical characteristics are fulfilled by the Ender 3. Its temperature at the tip of the nozzle is exactly in the range required for the production of extruded PLA.

PLA has the pleasant property of not necessarily requiring a heated printing bed. This is available in an Ender 3. But the printer is equipped with a flexible printing plate as standard. By bending it slightly, practically any product can be easily removed.

A disadvantage of the printing plate is that it produces a rough underside. When printing with PLA, a glass plate is therefore recommended as a base. This is always absolutely flat. It also ensures that the underside of the print is mirror smooth. In order to create a sufficiently stable adhesive base, use a 3D printer glue stick to coat the glass plate prior to printing.

Of course, it is always advisable to find out exactly which PLA filament you want to order. This applies especially to the temperature range. It is quite possible that one or the other manufacturer will state a deviation here that is outside the maximum temperature of Ender 3.

The low melting temperature of the filament reacts sensitively to trapped heat. Especially the gears in the feed can heat up so much in a closed housing that they reach the extrusion temperature of the material. Then your filament thread wraps around these gear wheels and does not reach the hot end tip anymore. This not only ruins the print but always results in costly repair and cleaning work.

PolyLite™ Metallic PLA Pro

Printing Temperature: 190°C - 220°C
Bed Temperature: 30°C - 60°C
Printing Speed: 30mm/s - 70mm/s
Fan: ON

Direct Drive:
Retraction Distance: 1mm
Retraction Speed: 30mm/s
Indirect Drive:
Retraction Distance: 3mm
Retraction Speed: 60mm/s

Printing with Silk PLA

Silk PLA filament is a type of 3D printing material made by adding special additives (such as metallic particles) to PLA base material. Typically, Silk PLA has a shiny and silky appearance and also has a soft and smooth texture similar to silk fabric, so it is called: silk PLA. Compared to traditional PLA, Silk PLA has stronger inter-layer adhesion, resulting in a smoother and more delicate surface of the printed model and a more obvious silk-like feel. These characteristics make Silk PLA filament very popular; it can be used in a wide range of applications and is an excellent choice for producing high quality and more attractive looking prints.

Below, you can find the Silk PLA settings that we recommend using with the Ender 3, which are the values that we have arrived at through our testing:
Nozzle (Print) Temperature: 200 - 215 °C
Bed Temperature: 60 °C
Print Speed: 30 - 50 mm/s (Slow down the printing speed to as low as 20 mm/s if you want the best surface finish.)
Layer Height: 0.1 - 0.2 mm

Some Other Recommended Silk PLA Slicer Settings

Filament Drying

Recommendation Temp:	55 °C
Time:	2 – 4 hr

Material

Print temperature (+/- 5 °C based on hardware):	215 °C
Printing temperature initial layer:	215 °C
Initial printing temperature:	215 °C
Final printing temperature:	60 °C
Build plate temperature:	60 °C

Speed

Print speed:	50 mm/s
Initial layer speed:	20 mm/s

Optimize Wall Printing Order:	Open
Flow:	95 - 100%

Retraction

Enable retraction:	Yes
Retraction distance (direct drive):	0.8 mm
Retraction distance (bowden):	5.5 mm
Retraction speed:	40 - 45 mm
Print Jerk:	8 mm/s

Travel

Combing mode:	Within Infill
Retract before outer wall:	Yes

Z hop when retracted:	No

Cooling

Fan speed:	100%
Initial fan speed:	0%
Regular fan speed at layer:	3
Minimum layer time:	10s

Build plate adhesion

Build plate adhesion type:	Skirt
Skirt line count:	3

Printing with PETG

PETG is a variant of polyethylene terephthalate. Basically, it is normal PET, as it is used for beverage bottles, with the addition of glycol. It combines the advantages of PLA and ABS. Printing is easy and the results are detailed and precise.

PETG filament requires the following settings:
Print temperature: 230 – 250 °C
Bed temperature: For the first layers 110 °C, from 5 mm height it can be reduced to 70 °C.
Plate adhesion: required, but in combination with a release agent
Printing speed: 60 mm/s max.
Retraction height: 5 – 7 mm
Retraction speed: 40 mm/s minimum
Cooling: required
First layer height on the base plate: 0.3 mm

PETG requires a very hot print. The printing temperature can be around 250 °C. However, most filament manufacturers have now set a processing temperature of 240 °C for this type. This is just within the temperature range with which the Ender 3 can work optimally.

PETG also tends to warp. However, this is not as pronounced as with ABS filament. It is therefore sufficient to let the first layers adhere securely and straight through a hot adjusted printing plate. Afterward, the temperature can be reduced without any problems. This not only saves expensive electricity but also makes handling safer.

The ideal combination for the adhesive/release layer is blue painter's tape and hair spray. During printing, the solid then adheres sufficiently firmly. However, it can be easily removed after completion.

PETG can be printed at up to 60 mm/s. However, you should only select this printing speed if you do not want to place high demands on the outer contour and strength. With a 50% reduction in speed, you will get much better results. This doubles the time needed for printing but is better than wasting filament on an unsatisfactory result.

PETG is a little bit tricky when it comes to the topic "retraction". Due to its high processing temperature, it tends to drip and pull threads. At the same time, it cools down quickly. Too high a pullback quickly forms a drop on the hot end, which blocks the print. It is recommended that you determine the retraction individually for each project. It is always between 5 and 7 mm. Proceed in the same way step by step when selecting the retraction speed. Starting at 40 mm/s you can work your way to the optimum setting in 5 mm/s increments.

The quality of your print is significantly influenced by a powerful cooling system. With the fan turned on, the details can be seen more precisely. On the other hand, a switched-off fan ensures that the individual layers bond more firmly together. So, if you want to produce a detailed, aesthetic component, a switched-on fan is recommended. But if you want to make a robust component for static applications, leave the fan off. The difference can be big.

Always work with a base layer when printing with PETG. This should be applied as loosely as possible. So let your extruded thread glide from a height of about 0.3 mm onto the prepared heating plate without pressing it down. Only after you have applied a base layer, can you reduce the printing distance again.

Printing with Foaming LW-PLA

Creality quote "Yes, the Ender-3 V3 is compatible with 1.75 mm Light Weight PLA (LW-PLA) filaments. However, please note that **Light Weight PLA foams and expands when printed**, so it requires special print settings."

LW-PLA is a unique filament especially suited for RC flight and it's the first filament of its kind. It uses an active foaming technology to achieve lightweight, low-density printed parts. At around 230 °C LW-PLA starts foaming and it increases its volume by nearly 3 times. Decreasing the material flow to 35% or 40% in your preferred slicer allows you to achieve very lightweight parts.

How Does LW-PLA Work?
LW-PLA is a PLA material mixed with a foaming additive that activates at elevated temperatures. It will print much like standard PLA at lower temperatures (210 – 220 °C) but once you print above 230 °C the foaming technology activates. This foaming effect becomes more aggressive as you continue to elevate the temperature. The key to success when printing with LW-PLA is balancing between print temperature, speed, and flow rate. Once your printer and slicer settings are dialed in, you should achieve parts that match the wall thickness of the designed part but are 35 - 40% the weight of the same part printed in traditional PLA.

In order to print parts that are dimensionally accurate and as light as possible, you need to determine how much the material will expand with your printer at different print temperatures. We do this by printing single perimeter test cubes at various temperatures and flow rates. (https://www.3daeroventures.com/s/LW-PLA-Test-Cube.STL) Then follow the following steps to dial in your printer's settings for LW-PLA.

Step 1
Print various cubes in 10 degree increments between 200 – 280 °C. Use the following settings to print the supplied test cube:

Bed Temp: 60 °C
Speed: 30 - 40 mm/s
Bottom Layers: 4
Top Layers: 0
Perimeter: 1
Infill: 0%
Cooling Fan: 0% for First Layer, up to 25% for subsequent layers

Step 2
Measure the wall thickness of each cube and note the temperature that gave you the cube with the thickest wall/highest expansion.

Now, print various cubes, all at the same print temperature you determined in Step 2, but each cube at a different flow percentage. Print the first cube at 90% flow, then step down by 10% for each cube until your last cube is printed at 30% flow. Note that Cura calls this setting "Flow" and it is entered as a percentage and Simplify3D calls this setting "Extrusion Multiplier" and it is entered as a decimal value (e.g. 0.4 instead of 40%)

Measure the wall thickness of each cube and find the one that matches the nozzle diameter you are printing with. If you are printing with a 0.4mm nozzle then the wall thickness of your perfectly optimized cube should measure 0.4 mm.

Foaming LW-PLA slicer settings:

Nozzle Temperature Suggested Range: 210 – 260 °C
Effect of Temperature: Lower temperatures (around 210 – 220 °C) yield minimal foaming, resulting in denser prints. Higher temperatures (250 °C+) significantly increase the foaming effect, which is ideal for lightweight parts but can sacrifice some surface detail.

Bed Temperature Suggested Range: 40 – 60 °C
Bed Adhesion: LW-PLA generally adheres well to a heated bed set around 50 °C. If your prints have trouble sticking, consider adding a thin layer of glue stick or using a textured build surface.

Print Speed Suggested Range: 30 - 60 mm/s
Optimal Speed: Printing at slower speeds, like 30 - 40 mm/s, helps manage the foaming process and ensures better print quality. Faster speeds may work but can lead to inconsistencies in extrusion due to rapid foaming.

Layer Height Suggested Range: 0.1 - 0.3 mm
Considerations: A smaller layer height will yield finer details but may increase print time. For larger, less detailed models, a 0.2 - 0.3 mm layer height is suitable.

Retraction Settings Retraction Distance: 3 - 6 mm for Bowden setups, 1 - 2 mm for direct drive
Retraction Speed: 20 - 40 mm/s
Tips: Since LW-PLA tends to be more susceptible to stringing due to foaming, fine-tuning retraction settings are crucial to reduce unwanted stringing.

Cooling Fan Speed: 0 - 50%
Recommendations: To control foaming, it's often best to print with reduced fan speeds or even turn off cooling entirely. Too much cooling can lead to brittle foamed parts, while minimal cooling ensures smoother extrusion.

As a reference, we have had great success printing LW-PLA with the following settings:
Nozzle Temp: 250 °C
Flow Rate (Extrusion Multiplier): 40% (0.4)
Speed: 40 mm/s

Tips for Optimal LW-PLA Printing

Printing with LW-PLA requires a bit of experimentation to get the ideal balance between weight and quality. Here are some advanced tips to help you master printing with this filament:

1. Experiment with Temperature for Desired Foaming
Different models and applications call for varying levels of foaming. If you want a more solid part, use a lower temperature (210-220°C). For maximum weight reduction, increase the temperature gradually to 240-250°C but monitor your print for any signs of over-foaming.

2. Adjust the Flow Rate
Because of LW-PLA's expansion properties, you can reduce the flow rate to save material. For instance, if you're printing at a high foaming temperature, reducing the flow rate by 20-50% can still yield a full-size part without excess material.

3. Use Lower Infill for Lightweight Parts
In most cases, you can use an infill percentage as low as 10-20%, especially for parts where structural integrity is less critical. However, keep in mind that foamed parts can be more fragile than dense prints, so choose infill settings based on your part's functional requirements.

4. Post-Processing Possibilities
LW-PLA prints can be post-processed similarly to standard PLA. You can sand, paint, or even add finishes to achieve a smooth and professional look. Due to its foamed nature, LW-PLA is often easier to sand, which can be advantageous for detailed projects.

Troubleshooting Common Issues with LW-PLA

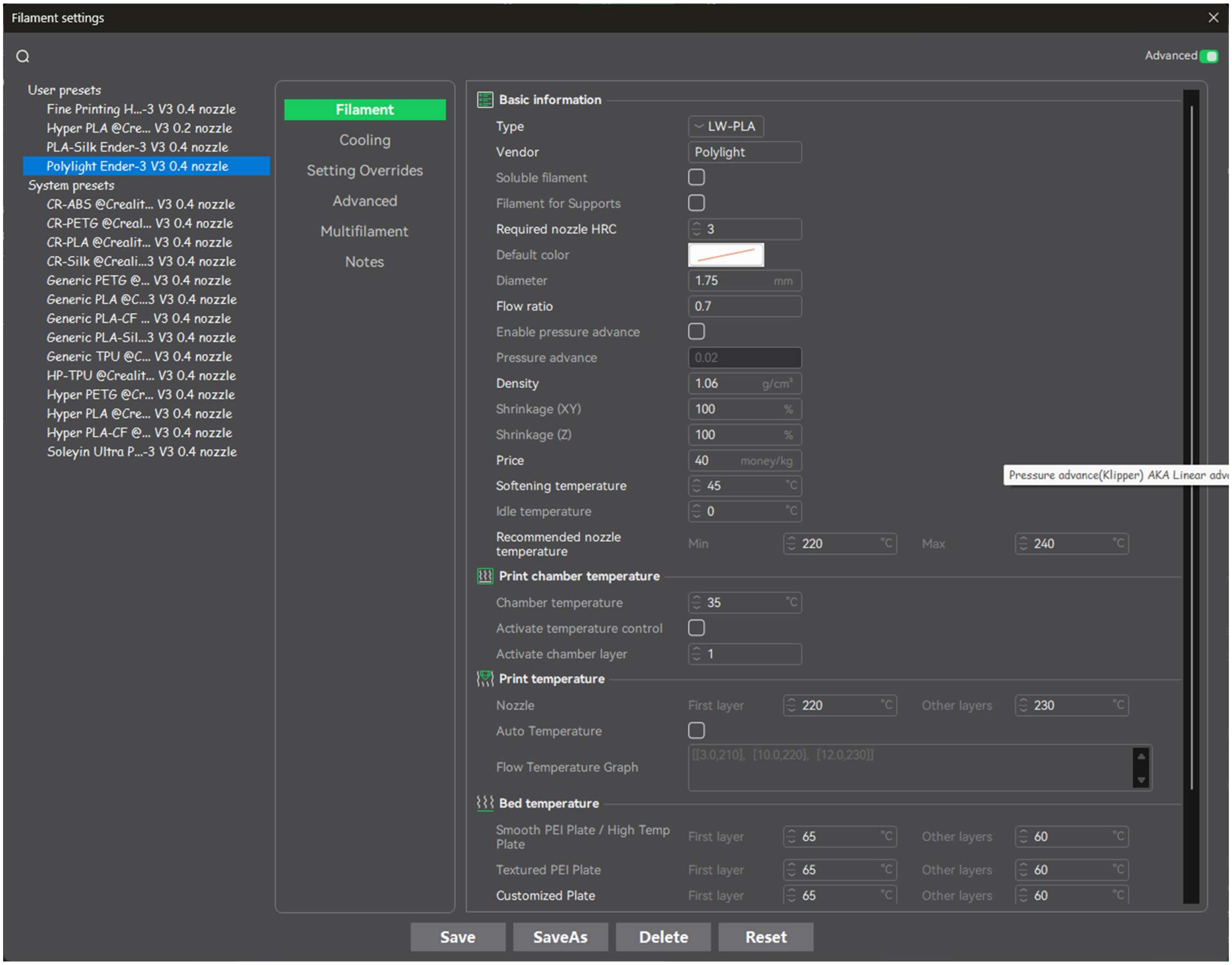
Over-Foaming
If you notice that your print is excessively foamed and lacks detail, try lowering the print temperature. Start with small temperature adjustments, like reducing by 5°C at a time, until you achieve the desired texture.

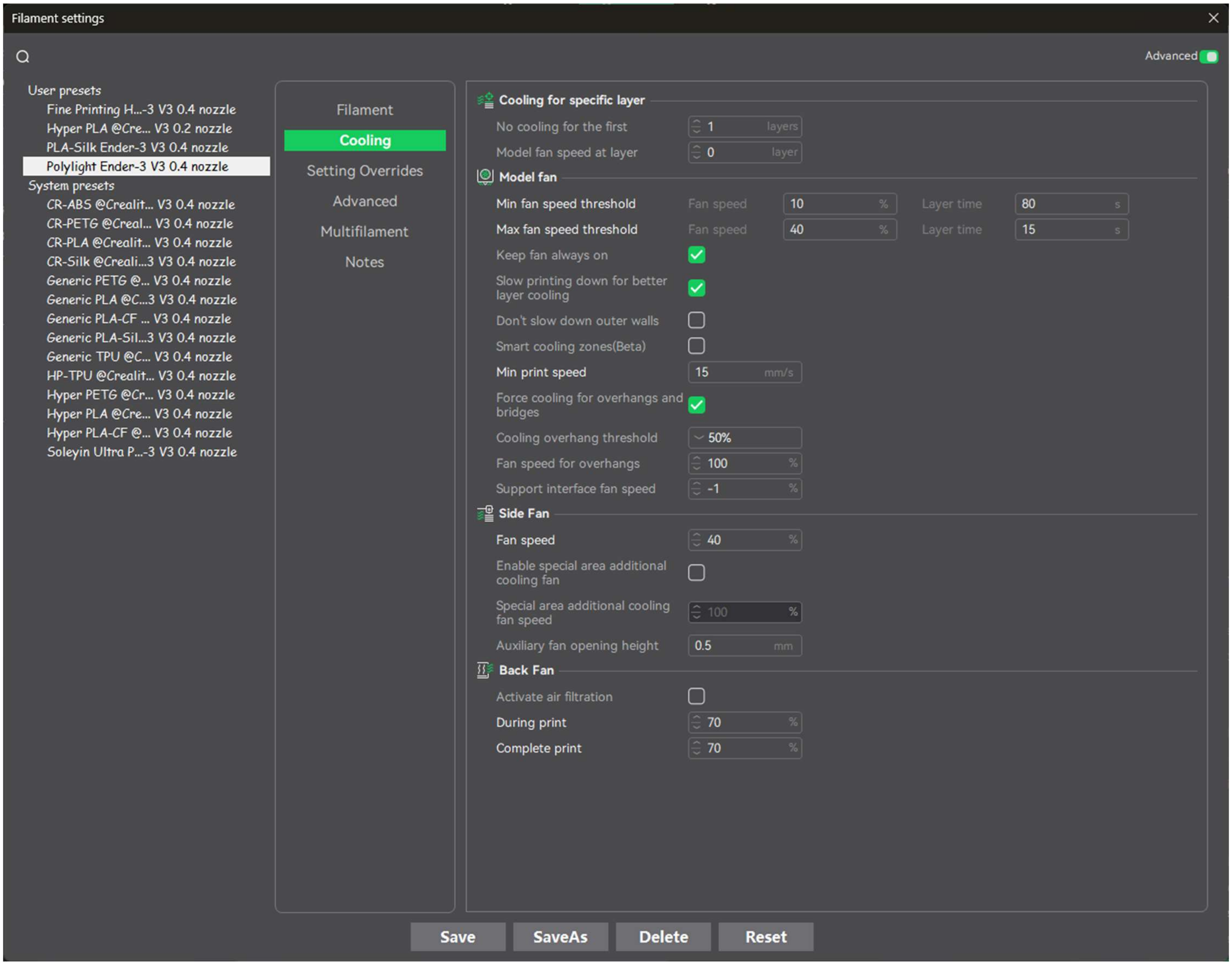
Stringing and Blobs
LW-PLA can be prone to stringing due to its foaming characteristics. Adjusting retraction distance and speed can help, as can lowering the print temperature slightly. Running a series of stringing tests may be helpful to fine-tune these settings.

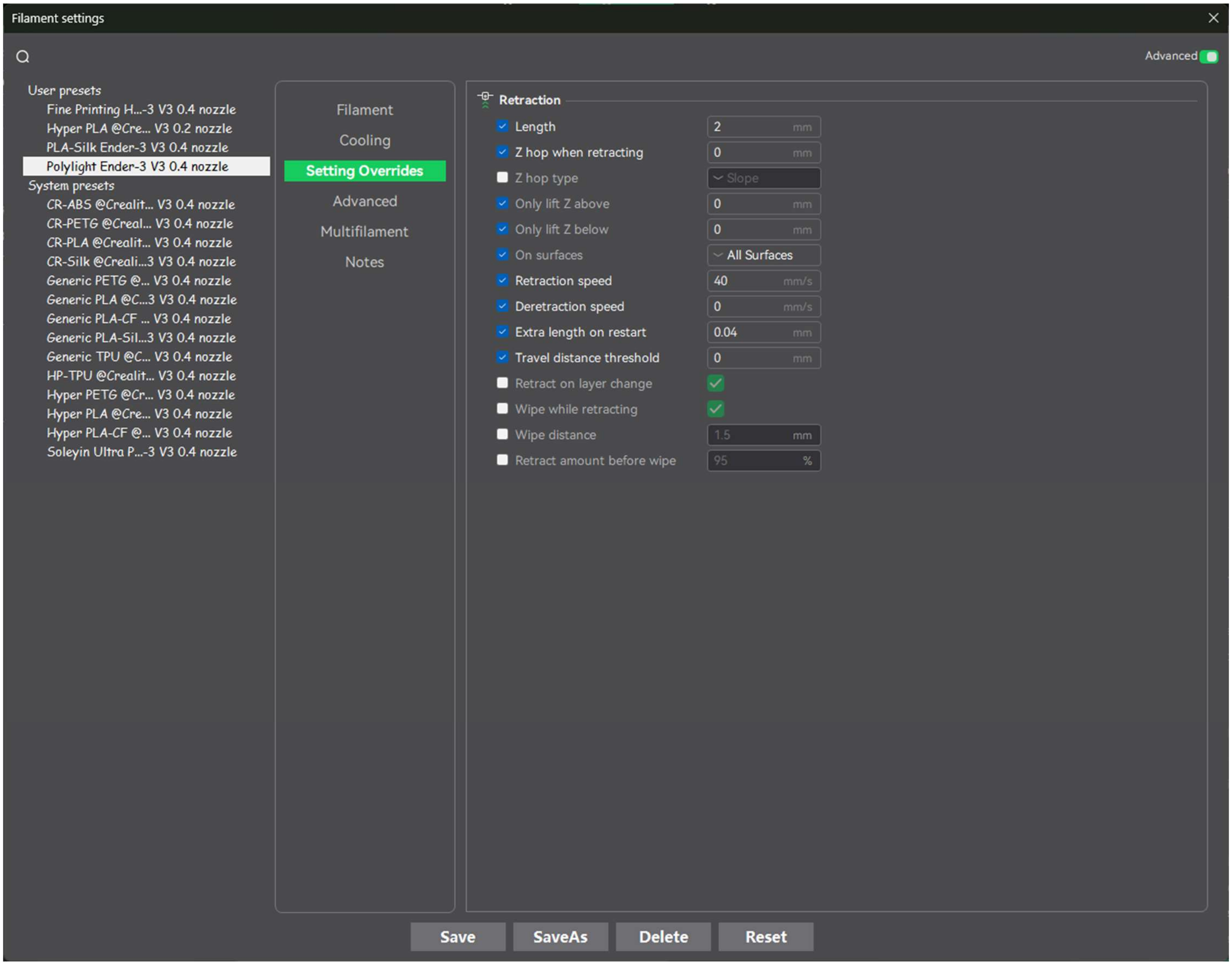
Inconsistent Extrusion

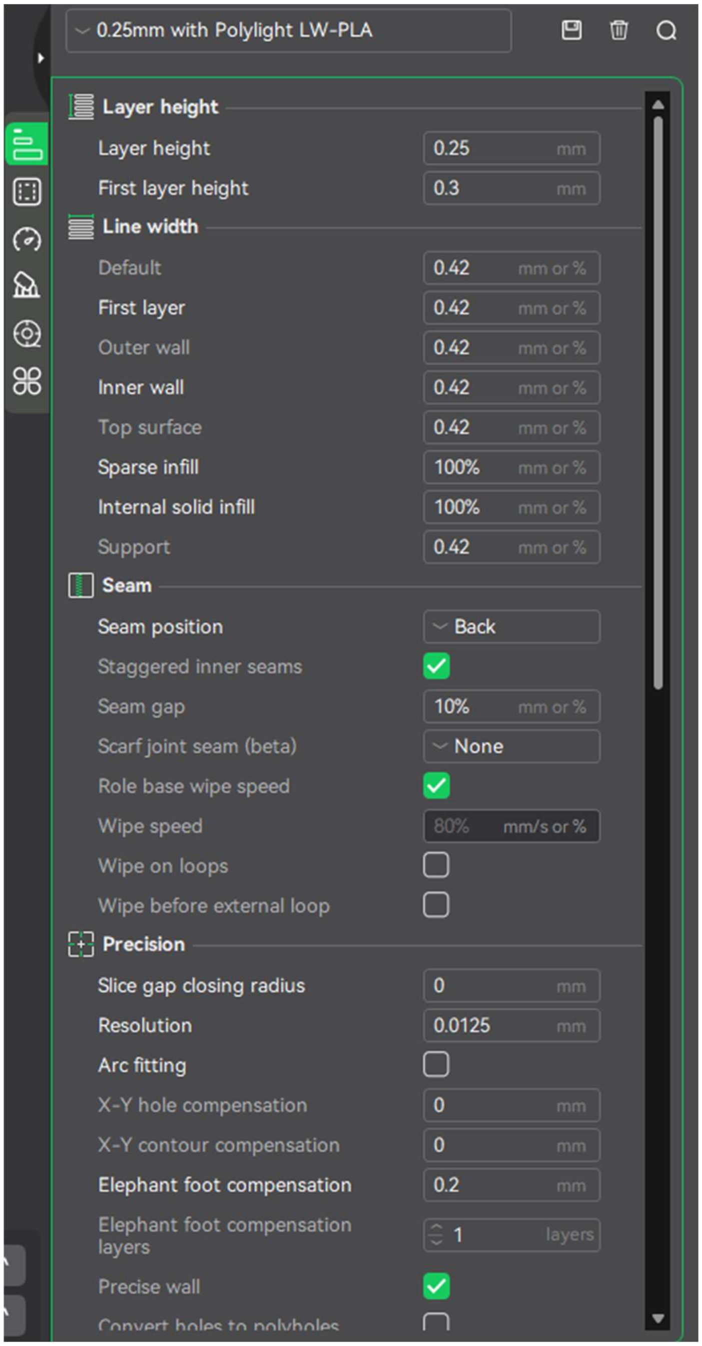
If you experience inconsistent extrusion, it may be due to rapid expansion in the filament. Slowing down your print speed and reducing the flow rate can provide more consistent results. Also, ensure that the filament path in your extruder is free from obstructions.

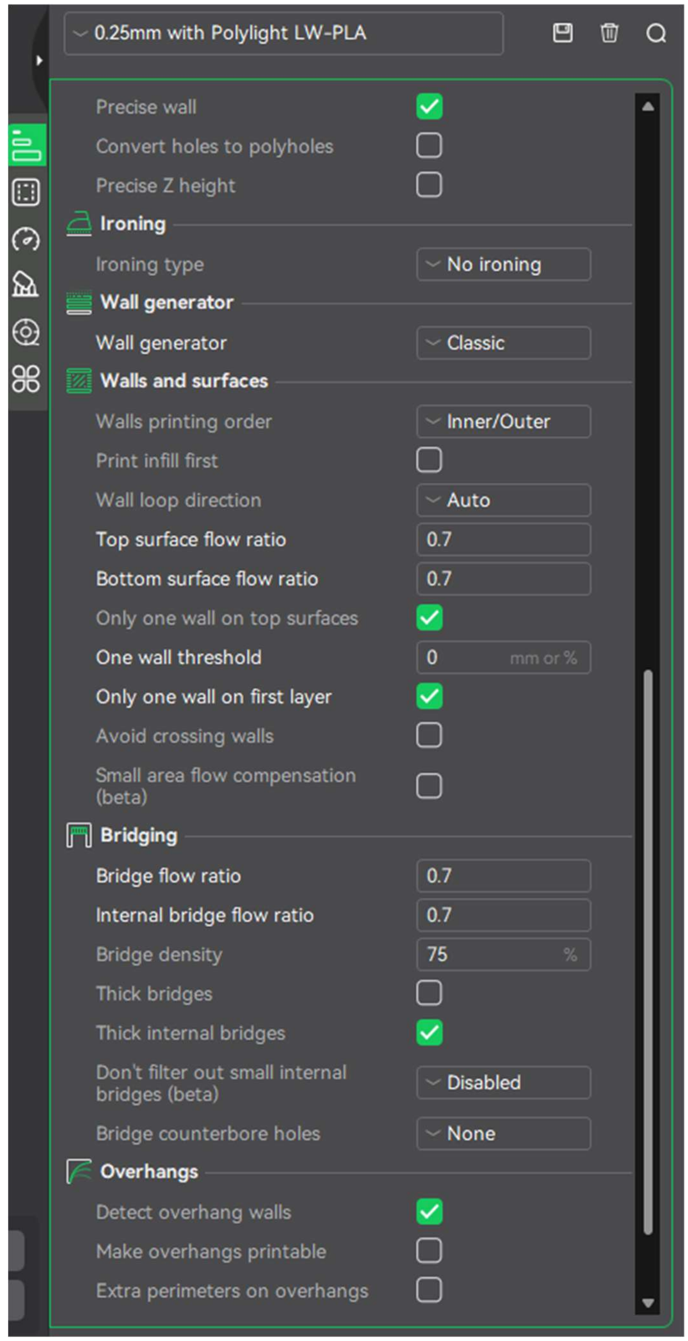
Creality Print 6.2 Settings for Ender-3 V3 with Polylight 1.0 LW-PLA using 0.4mm Nozzle

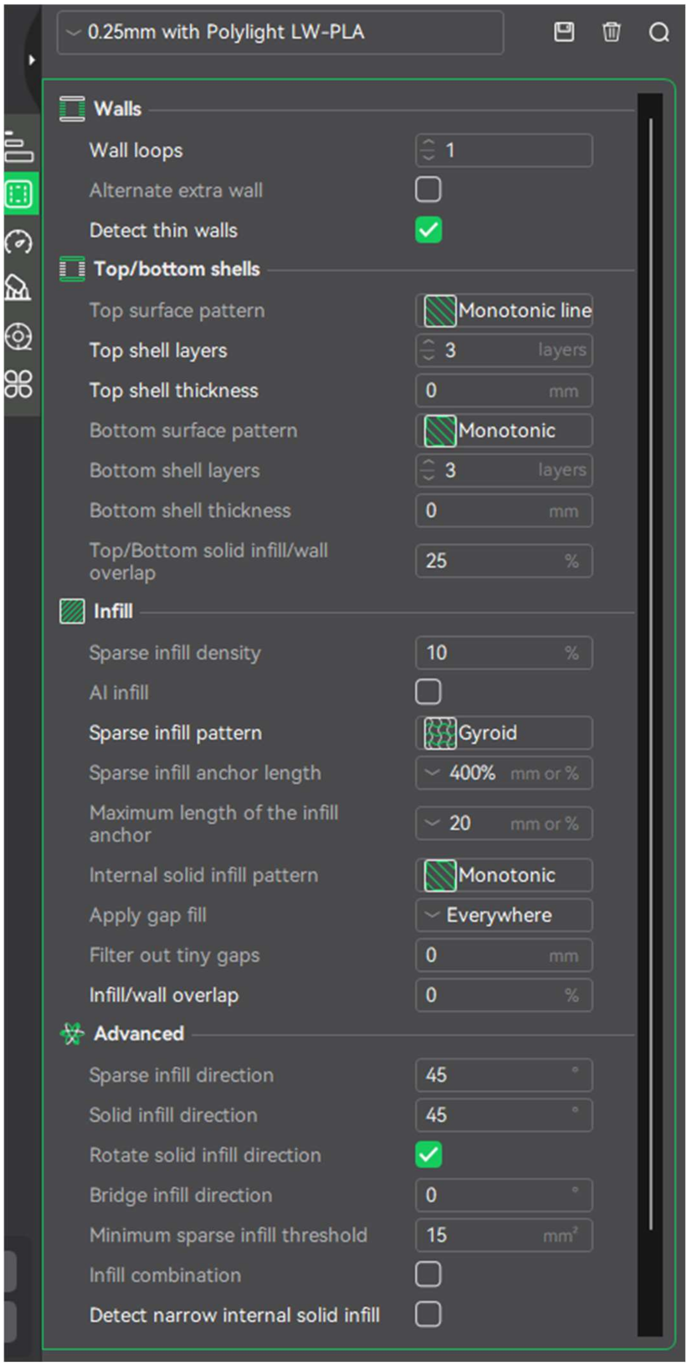


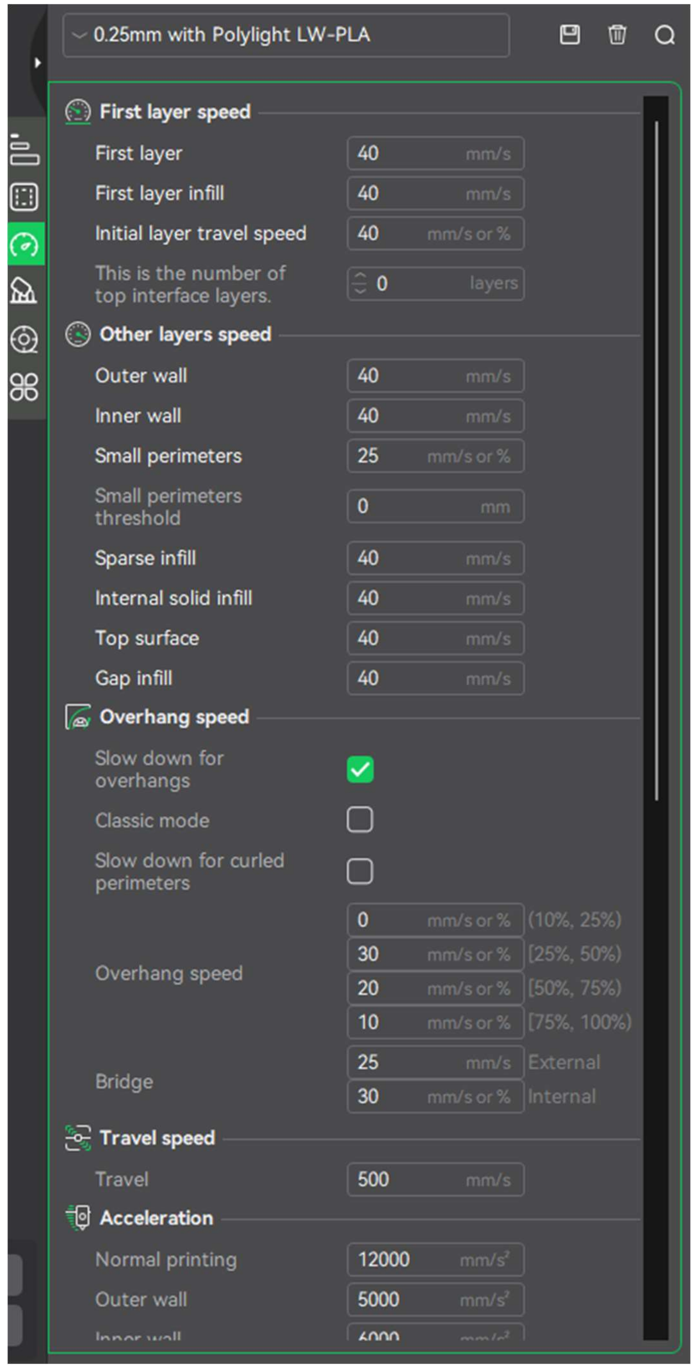

Filament settings
Advanced
User presets
Fine Printing H...-3 V3 0.4 nozzle
Hyper PLA @Cre... V3 0.2 nozzle
PLA-Silk Ender-3 V3 0.4 nozzle
Polylight Ender-3 V3 0.4 nozzle
System presets
CR-ABS @Crealit... V3 0.4 nozzle
CR-PETG @Creal... V3 0.4 nozzle
CR-PLA @Crealit... V3 0.4 nozzle
CR-Silk @Creali...3 V3 0.4 nozzle
Generic PETG @... V3 0.4 nozzle
Generic PLA @C...3 V3 0.4 nozzle
Generic PLA-CF ... V3 0.4 nozzle
Generic PLA-Sil...3 V3 0.4 nozzle
Generic TPU @C... V3 0.4 nozzle
HP-TPU @Crealit... V3 0.4 nozzle
Hyper PETG @Cr... V3 0.4 nozzle
Hyper PLA @Cre... V3 0.4 nozzle
Hyper PLA-CF @... V3 0.4 nozzle
Soleyin Ultra P...-3 V3 0.4 nozzle
Filament
Cooling
Setting Overrides
Advanced
Multifilament
Notes
Cooling for specific layer
No cooling for the first 1 layers
Model fan speed at layer 0 layer
Model fan
Min fan speed threshold Fan speed 10 % Layer time 80 s
Max fan speed threshold Fan speed 40 % Layer time 15 s
Keep fan always on
Slow printing down for better layer cooling
Don't slow down outer walls
Smart cooling zones(Beta)
Min print speed 15 mm/s
Force cooling for overhangs and bridges
Cooling overhang threshold 50%
Fan speed for overhangs 100 %
Support interface fan speed -1 %
Side Fan
Fan speed 40 %
Enable special area additional cooling fan
Special area additional cooling fan speed 100 %
Auxiliary fan opening height 0.5 mm
Back Fan
Activate air filtration
During print 70 %
Complete print 70 %
Save
SaveAs
Delete
Reset

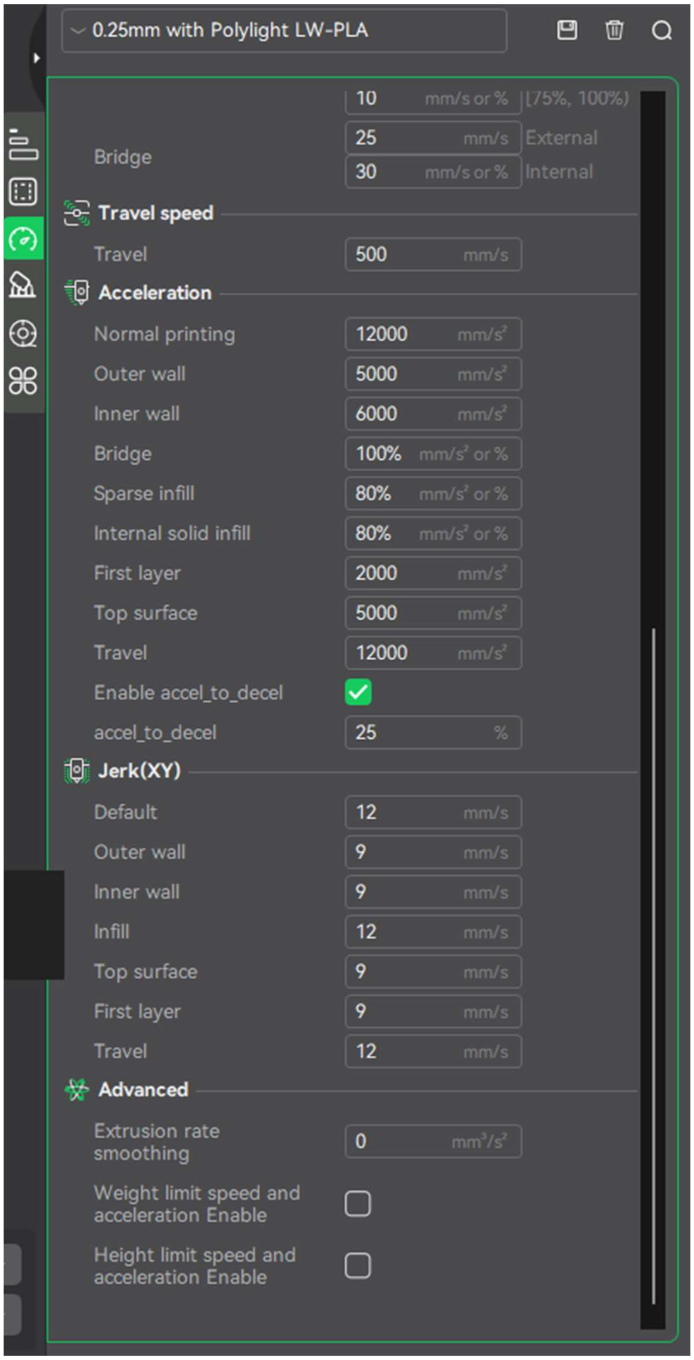

Filament settings
Advanced
User presets
Fine Printing H...-3 V3 0.4 nozzle
Hyper PLA @Cre... V3 0.2 nozzle
PLA-Silk Ender-3 V3 0.4 nozzle
Polylight Ender-3 V3 0.4 nozzle
System presets
CR-ABS @Crealit... V3 0.4 nozzle
CR-PETG @Creal... V3 0.4 nozzle
CR-PLA @Crealit... V3 0.4 nozzle
CR-Silk @Creali...3 V3 0.4 nozzle
Generic PETG @... V3 0.4 nozzle
Generic PLA @C...3 V3 0.4 nozzle
Generic PLA-CF ... V3 0.4 nozzle
Generic PLA-Sil...3 V3 0.4 nozzle
Generic TPU @C... V3 0.4 nozzle
HP-TPU @Crealit... V3 0.4 nozzle
Hyper PETG @Cr... V3 0.4 nozzle
Hyper PLA @Cre... V3 0.4 nozzle
Hyper PLA-CF @... V3 0.4 nozzle
Soleyin Ultra P...-3 V3 0.4 nozzle
Filament
Cooling
Setting Overrides
Advanced
Multifilament
Notes
Retraction
Length 2 mm
Z hop when retracting 0 mm
Z hop type Slope
Only lift Z above 0 mm
Only lift Z below 0 mm
On surfaces All Surfaces
Retraction speed 40 mm/s
Deretraction speed 0 mm/s
Extra length on restart 0.04 mm
Travel distance threshold 0 mm
Retract on layer change
Wipe while retracting
Wipe distance 1.5 mm
Retract amount before wipe 95 %
Save
SaveAs
Delete
Reset


0.25mm with Polylight LW-PLA
Layer height
Layer height 0.25 mm
First layer height 0.3 mm
Line width
Default 0.42 mm or %
First layer 0.42 mm or %
Outer wall 0.42 mm or %
Inner wall 0.42 mm or %
Top surface 0.42 mm or %
Sparse infill 100% mm or %
Internal solid infill 100% mm or %
Support 0.42 mm or %
Seam
Seam position Back
Staggered inner seams
Seam gap 10% mm or %
Scarf joint seam (beta) None
Role base wipe speed
Wipe speed 80% mm/s or %
Wipe on loops
Wipe before external loop
Precision
Slice gap closing radius 0 mm
Resolution 0.0125 mm
Arc fitting
X-Y hole compensation 0 mm
X-Y contour compensation 0 mm
Elephant foot compensation 0.2 mm
Elephant foot compensation layers 1 layers
Precise wall

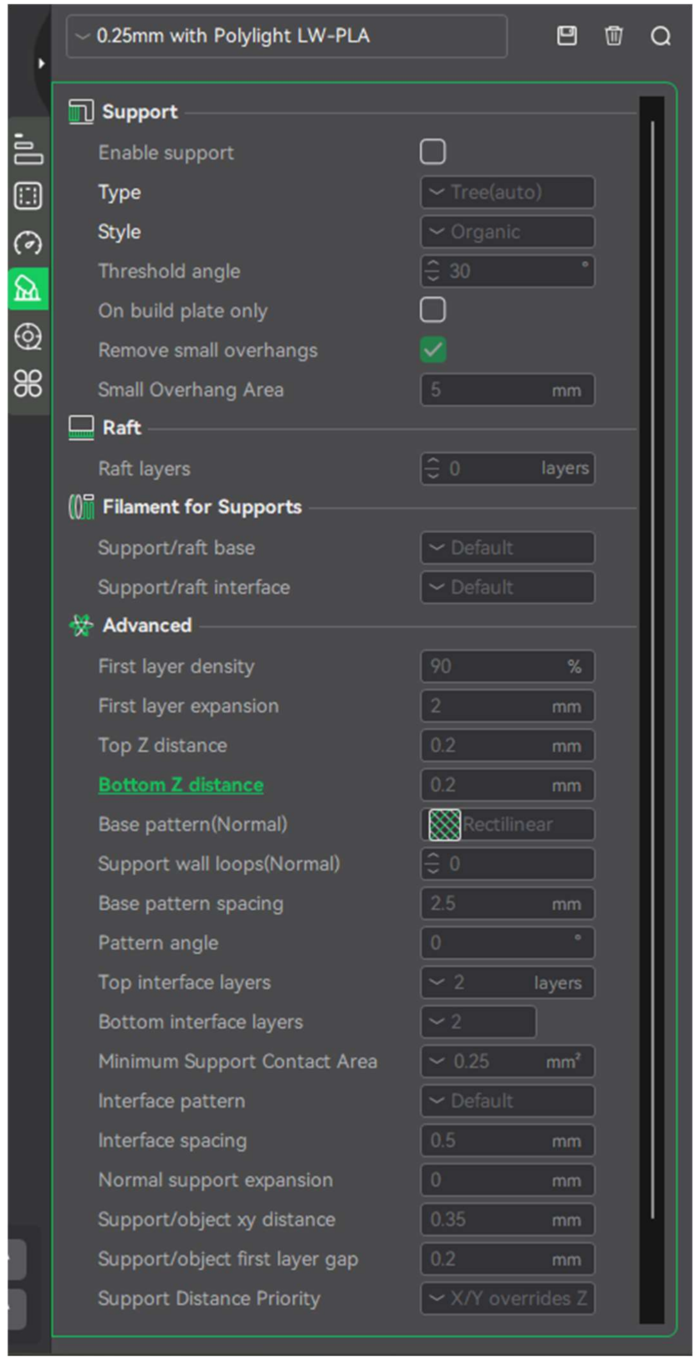

0.25mm with Polylight LW-PLA
Precise wall
Convert holes to polyholes
Precise Z height
Ironing
Ironing type No ironing
Wall generator
Wall generator Classic
Walls and surfaces
Walls printing order Inner/Outer
Print infill first
Wall loop direction Auto
Top surface flow ratio 0.7
Bottom surface flow ratio 0.7
Only one wall on top surfaces
One wall threshold 0 mm or %
Only one wall on first layer
Avoid crossing walls
Small area flow compensation (beta)
Bridging
Bridge flow ratio 0.7
Internal bridge flow ratio 0.7
Bridge density 75 %
Thick bridges
Thick internal bridges
Don't filter out small internal bridges (beta) Disabled
Bridge counterbore holes None
Overhangs
Detect overhang walls
Make overhangs printable
Extra perimeters on overhangs


0.25mm with Polylight LW-PLA
Walls
Wall loops 1
Alternate extra wall
Detect thin walls
Top/bottom shells
Top surface pattern Monotonic line
Top shell layers 3 layers
Top shell thickness 0 mm
Bottom surface pattern Monotonic
Bottom shell layers 3 layers
Bottom shell thickness 0 mm
Top/Bottom solid infill/wall overlap 25 %
Infill
Sparse infill density 10 %
AI infill
Sparse infill pattern Gyroid
Sparse infill anchor length 400% mm or %
Maximum length of the infill anchor 20 mm or %
Internal solid infill pattern Monotonic
Apply gap fill Everywhere
Filter out tiny gaps 0 mm
Infill/wall overlap 0 %
Advanced
Sparse infill direction 45 °
Solid infill direction 45 °
Rotate solid infill direction
Bridge infill direction 0 °
Minimum sparse infill threshold 15 mm²
Infill combination
Detect narrow internal solid infill

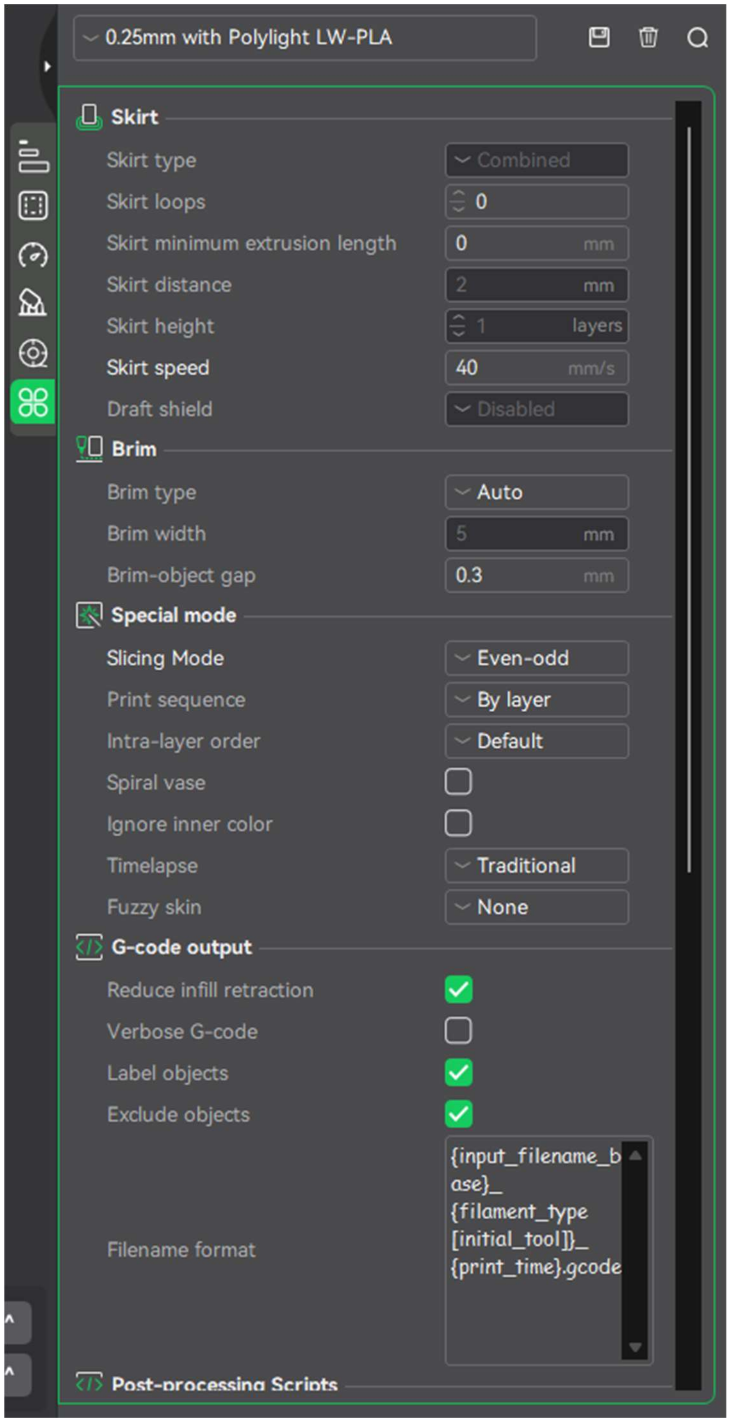

0.25mm with Polylight LW-PLA
First layer speed
First layer 40 mm/s
First layer infill 40 mm/s
Initial layer travel speed 40 mm/s or %
This is the number of top interface layers. 0 layers
Other layers speed
Outer wall 40 mm/s
Inner wall 40 mm/s
Small perimeters 25 mm/s or %
Small perimeters threshold 0 mm
Sparse infill 40 mm/s
Internal solid infill 40 mm/s
Top surface 40 mm/s
Gap infill 40 mm/s
Overhang speed
Slow down for overhangs
Classic mode
Slow down for curled perimeters
Overhang speed
0 mm/s or % (10%, 25%)
30 mm/s or % [25%, 50%)
20 mm/s or % [50%, 75%)
10 mm/s or % [75%, 100%)
Bridge
25 mm/s External
30 mm/s or % Internal
Travel speed
Travel 500 mm/s
Acceleration
Normal printing 12000 mm/s²
Outer wall 5000 mm/s²



0.25mm with Polylight LW-PLA
10 mm/s or % [75%, 100%)
Bridge
25 mm/s External
30 mm/s or % Internal
Travel speed
Travel 500 mm/s
Acceleration
Normal printing 12000 mm/s²
Outer wall 5000 mm/s²
Inner wall 6000 mm/s²
Bridge 100% mm/s² or %
Sparse infill 80% mm/s² or %
Internal solid infill 80% mm/s² or %
First layer 2000 mm/s²
Top surface 5000 mm/s²
Travel 12000 mm/s²
Enable accel_to_decel
accel_to_decel 25 %
Jerk(XY)
Default 12 mm/s
Outer wall 9 mm/s
Inner wall 9 mm/s
Infill 12 mm/s
Top surface 9 mm/s
First layer 9 mm/s
Travel 12 mm/s
Advanced
Extrusion rate smoothing 0 mm³/s²
Weight limit speed and acceleration Enable
Height limit speed and acceleration Enable



0.25mm with Polylight LW-PLA
Support
Enable support
Type Tree(auto)
Style Organic
Threshold angle 30 °
On build plate only
Remove small overhangs
Small Overhang Area 5 mm
Raft
Raft layers 0 layers
Filament for Supports
Support/raft base Default
Support/raft interface Default
Advanced
First layer density 90 %
First layer expansion 2 mm
Top Z distance 0.2 mm
Bottom Z distance 0.2 mm
Base pattern(Normal) Rectilinear
Support wall loops(Normal) 0
Base pattern spacing 2.5 mm
Pattern angle 0 °
Top interface layers 2 layers
Bottom interface layers 2
Minimum Support Contact Area 0.25 mm²
Interface pattern Default
Interface spacing 0.5 mm
Normal support expansion 0 mm
Support/object xy distance 0.35 mm
Support/object first layer gap 0.2 mm
Support Distance Priority X/Y overrides Z



Recommended Settings for eSun eLW-PLA

Temperature Settings:

Printing Temperature: 235°C

Build Plate Temperature: 60°C

Flow Rates:

Set the flow rate to 100% for standard prints. Adjust as necessary based on your printer's calibration and the specific model being printed.

Print Speed:
Print Speed: 100 mm/s
Outer Wall Speed: 40 mm/s
Inner Wall Speed: 40 mm/s
Travel Speed: 80 mm/s

Layer Height:
Layer Height: 0.25 mm
Line Width: 0.4 mm

Infill and Shell Settings:
Infill Density: 0% (for lightweight prints, adjust as needed for strength)
Wall Line Count: 1

Retraction Settings:
Enable Retraction: Checked
Retraction Distance: 2 mm
Retraction Speed: 30 mm/s

Cooling Settings:
Enable Print Cooling: Checked
Fan Speed: 20%

Additional Tips

Calibration: Ensure your printer is well-calibrated, especially for flow rates and retraction settings, as LW-PLA can be sensitive to these parameters.

Single Part Printing: It is recommended to print one part at a time to minimize stringing and improve print quality.

Foaming Behavior: Be aware that eSun PLA-LW foams during printing, which can affect the final dimensions of your print. Adjust settings accordingly to achieve the desired results.

By following these settings and tips, you should be able to achieve high-quality prints with eSun PLA-LW filament, suitable for lightweight applications such as model aircraft.

Active Foaming LW-PLA Settings (Cura)

Quality

Layer Height	0.25 mm
Initial Layer Height	0.25 mm
Line Width	0.4 mm
Wall Line Width	0.4 mm
Outer Wall Line Width	0.4 mm
Inner Wall(s) Line Width	0.4 mm
Top/Bottom Line Width	0.4 mm

Infill Line Width	0.4 mm
Skirt/Brim Line Width	0.4 mm
Initial Layer Line Width	100%

<u>Walls</u>

Wall Thickness	0.4 mm
Wall Line Count	1
Outer Wall Wipe Distance	0.0 mm
Outer Wall Inset	0.0 mm
Optimize Wall Printing Order	☑
Wall Ordering	Inside to Outside
Alternate Extra Wall	☐
Print Thin Walls	☑
Horizontal Expansion	0.0 mm
Initial Layer Horizontal Expansion	0.0 mm
Hole Horizontal Expansion	0.0 mm
Z Seam Alignment	Sharpest Corner
Seam Corner Preference	Smart Hiding

<u>Top/Bottom</u>

Top Surface Skin Layers	0
Top/Bottom Thickness	0.75 mm
Top Thickness	0.75 mm
Top Layers	3
Bottom Thickness	0.75 mm
Bottom Layers	3

<u>Top/Bottom Pattern Lines</u>

Bottom Pattern Initial Layer	Lines
Monotonic Top/Bottom Order	☐
Top/Bottom Line Directions	☐
No Skin in Z Gaps	☐
Extra Skin Wall Count	1
Enable Ironing	☐
Skin Overlap Percentage	10%
Skin Overlap	0.04 mm
Skin Removal Width	0.4 mm
Top Skin Removal Width	0.4 mm
Bottom Skin Removal Width	0.4 mm
Skin Expand Distance	0.4 mm
Top Skin Expand Distance	0.4 mm
Bottom Skin Expand Distance	0.4 mm
Maximum Skin Angle for Expansion	90°
Minimum Skin Width for Expansion	0.0 mm

<u>Infill (Gyroid)</u>

Infill Density	3%
Infill Line Distance	13.333 mm
Infill Pattern	Gyroid
Connect Infill Lines	☐
Randomize Infill Start	☐
Infill Line Multiplier	1
Extra Infill Wall Count	0
Infill Overlap Percentage	10%
Infill Overlap	0.04 mm
Infill Wipe Distance	0.1 mm
Infill Layer Thickness	0.25 mm
Gradual Infill Steps	0
Infill Before Walls	☐
Infill Minimum Area	0.0 mm
Infill Support	☐
Skin Edge Support Thickness	0.0 mm
Skin Edge Support Layers	0

<u>Infill (Cubic Subdivision)</u>

Infill Density	3%
Infill Line Distance	40.0 mm
Infill Pattern	Cubic Subdivision
Infill Line Directions	☐
Randomize Infill Start	☐
Infill Line Multiplier	1
Cubic Subdivision Shell	0.4 mm
Infill Overlap Percentage	10%
Infill Overlap	0.04 mm
Infill Wipe Distance	0.1 mm
Infill Layer Thickness	0.25 mm
Gradual Infill Steps	0
Infill Before Walls	☐
Infill Minimum Area	0.0 mm
Infill Support	☐
Skin Edge Support Thickness	0.0 mm
Skin Edge Support Layers	0

<u>Material</u>

Printing Temperature	235 °C
Printing Temperature Initial Layer	235 °C
Initial Printing Temperature	235 °C
Final Printing Temperature	235 °C
Build Plate Temperature	60 °C
Build Plate Temperature Initial Layer	60 °C
Scaling Factor Shrinkage Compensation	100%
Horizontal Scaling Factor Shrinkage Compensation	100%

Vertical Scaling Factor Shrinkage Compensation	100%
Flow	60%
Wall Flow	60%
Outer Wall Flow	60%
Inner Wall(s) Flow	60%
Top/Bottom Flow	60%
Infill Flow	60%
Skirt/Brim Flow	60%
Prime Tower Flow	60%
Initial Layer Flow	80%

<u>Speed</u>

Print Speed	60 mm/s
Infill Speed	60 mm/s
Wall Speed	30 mm/s
Outer Wall Speed	30 mm/s
Inner Wall Speed	30 mm/s
Top/Bottom Speed	30 mm/s
Travel Speed	120 mm/s
Initial Layer Speed	30 mm/s
Initial Layer Print Speed	30 mm/s
Initial Layer Travel Speed	120 mm/s
Skirt/Brim Speed	30 mm/s
Number of Slower Layers	2
Flow Equalization Ratio	100%
Enable Acceleration Control	☐
Enable Jerk Control	☐

<u>Travel</u>

Enable Retraction	☑
Retract at Layer Change	☐
Retraction Distance	0.0 mm
Retraction Speed	35 mm/s
Retraction Retract Speed	35 mm/s
Retraction Prime Speed	35 mm/s
Retraction Extra Prime Amount	0.3 mm
Retraction Minimum Travel	1.5 mm
Maximum Retraction Count	90
Minimum Extrusion Distance Window	6.5 mm
Combing Mode	All
Avoid Supports When Traveling	☑
Travel Avoid Distance	0.625 mm
Layer Start X	0.0 mm
Layer Start Y	0.0 mm
Z Hop When Retracted	☐

Cooling	
Enable Print Cooling	☐
Regular/Maximum Fan Speed Threshold	10 s
Regular Fan Speed at Height	0.27 mm
Regular Fan Speed at Layer	2
Minimum Layer Time	2.0 s
Maximum Speed	10 mm/s
Lift Head	☐
Support	
Generate Support	☐
Build Plate Adhesion	
Build Plate Adhesion Type	Brim
Skirt/Brim Minimum Length	250 mm
Brim Width	8.0 mm
Brim Line Count	20
Brim Distance	0.0 mm
Brim Only on Outside	☑

Printing with Pre-Foamed LW-PLA

I purchased a roll of Bright Yellow Polymaker Light Weight PLA Filament from Amazon, where you can find this on the web @: https://www.amazon.com/dp/B0B1DGJSTQ/?tag=lstir-20&th=1. The features of this **Pre-Foamed** filament are captured in the next few paragraphs below.

Lightweight PLA: Polymaker LW-PLA filament has a low density of 0.9g/cm^3, compared to 1.17g/cm^3 for standard PLA. **It uses the same printing settings as regular PLA**, ensuring easy compatibility with most 3D printers. *Unlike the traditional foaming LW-PLA, Polymaker LW-PLA does not foam during printing, preventing common foaming issues. Compared with the filament of Foaming-When-Printing, it has higher printing density stability and is better at suppressing stringing.*

High Rigidity & Good Layer Adhesion: Polymaker LW-PLA filament provides high rigidity and excellent layer adhesion, making it ideal for creating stiff parts for radio-controlled planes. This reliable LW-PLA features great bed adhesion, consistent color, and dimensional accuracy, while eliminating issues like warping, jamming, blobs, and layer delamination. Note: **Turn off the fan for the first layer to improve bed adhesion.**

7 Colors Available & Matte Finish: Polymaker LW-PLA is available in white, black, grey, wood, bright yellow, bright orange, and bright green. It produces a matte surface finish that conceals layers and is easy to paint.

Tangle-Free & Moisture-Free: Polymaker LW-PLA filament is meticulously wound to prevent tangling and is dried and vacuum-sealed in a resealable Ziplock bag with a desiccant. Always hold the filament tip to avoid nodes and utilize the spool holes to help prevent tangling.

Pre-Foamed LW-PLA Printing Settings: Printing Temperature: 190 – 210°C; Bed Temperature: 30 – 50°C; Printing Speed: 25 – 60 mm/s; Fan: ON; For Direct Drive: Retraction Distance: 3 mm; Retraction Speed: 40 mm/s; For Indirect Drive: Retraction Distance: 6 mm; Retraction Speed: 60 mm/s

Here's a helpful video on Pre-Foamed LW-PLA: https://youtu.be/eDBtZeIY5aE. It seems like a happy medium between foaming LW-PLA and standard PLA in both strength and weight, as in it is stronger than foaming LW-PLA it seems, and lighter than standard PLA, but not quite as light as the ColorFab LW-PLA. However, it is allegedly very easy to print, and at a lower price.

Another source of pre-foamed LW-PLA is **OVERTURE Air PLA Filament**, Pre-Foamed PLA Low-Density (0.82g/cm^3), which you can also purchase on Amazon @: https://www.amazon.com/OVERTURE-Filament-Cardboard-Dimensional-Probability/dp/B0BQRG6FBW/?th=1.

Overture Air Filament settings for the Ender-3 V3 with a 0.4 nozzle:

All Flow Ratios: 120%
Recommended Nozzle Temperature: 190 – 210°C
Nozzle Print Temperatures:
- First Layer - 210°C
- Other Layers - 190°C

Smooth PEI Plate / High Temp Plate (bed) Temperatures:
- First Layer: 65°C
- Other Layers: 60°C

Fan Cooling:
- No cooling for first layer
- Model fan speed @ 2nd layer w/ 10% min fan speed & 25% max fan speed

For Direct Drive:
- Retraction Distance: 3.5 mm
- Retraction Speed: 35 mm/s
- Extra length on restart: 0.35mm

Layer Height: 0.20 mm
- First layer height: 0.25 mm

Line Width: 0.30 mm

First Layer & Infill Speed: 15 mm/s
Other Layers Speed: 30 – 50 mm/s

Top Shell Layers: 4 @ 1mm
Bottom Shell Layer: 3 @ 1mm
Sparse Infill Density: 5%
Sparse Infill Pattern: Gyroid
Enable Support: use Tree(auto) Type and Organic Style
Brim: Outer Brim Only
Travel speed: 250 mm/s
Acceleration: 750 mm/s^2

Prefoamed LW-PLA Settings (Cura)

Quality

Layer Height	0.25 mm

Initial Layer Height	0.25 mm
Line Width	0.4 mm
Wall Line Width	0.4 mm
Outer Wall Line Width	0.4 mm
Inner Wall(s) Line Width	0.4 mm
Top/Bottom Line Width	0.4 mm
Infill Line Width	0.4 mm
Skirt/Brim Line Width	0.4 mm
Initial Layer Line Width	100%

Walls

Wall Thickness	0.4 mm
Wall Line Count	1
Outer Wall Wipe Distance	0.0 mm
Outer Wall Inset	0.0 mm
Optimize Wall Printing Order	☑
Wall Ordering	Inside to Outside
Alternate Extra Wall	☐
Print Thin Walls	☑
Horizontal Expansion	0.0 mm
Initial Layer Horizontal Expansion	0.0 mm
Hole Horizontal Expansion	0.0 mm
Z Seam Alignment	Sharpest Corner
Seam Corner Preference	Smart Hiding

Top/Bottom

Top Surface Skin Layers	0
Top/Bottom Thickness	0.75 mm
Top Thickness	0.75 mm
Top Layers	3
Bottom Thickness	0.75 mm
Bottom Layers	3

Top/Bottom Pattern Lines

Bottom Pattern Initial Layer	Lines
Monotonic Top/Bottom Order	☐
Top/Bottom Line Directions	☐
No Skin in Z Gaps	☐
Extra Skin Wall Count	1
Enable Ironing	☐
Skin Overlap Percentage	10%
Skin Overlap	0.04 mm
Skin Removal Width	0.4 mm
Top Skin Removal Width	0.4 mm
Bottom Skin Removal Width	0.4 mm
Skin Expand Distance	0.4 mm

Top Skin Expand Distance	0.4 mm
Bottom Skin Expand Distance	0.4 mm
Maximum Skin Angle for Expansion	90°
Minimum Skin Width for Expansion	0.0 mm

<u>Infill (Gyroid)</u>

Infill Density	3%
Infill Line Distance	13.333 mm
Infill Pattern	Gyroid
Connect Infill Lines	☐
Randomize Infill Start	☐
Infill Line Multiplier	1
Extra Infill Wall Count	0
Infill Overlap Percentage	10%
Infill Overlap	0.04 mm
Infill Wipe Distance	0.1 mm
Infill Layer Thickness	0.25 mm
Gradual Infill Steps	0
Infill Before Walls	☐
Infill Minimum Area	0.0 mm
Infill Support	☐
Skin Edge Support Thickness	0.0 mm
Skin Edge Support Layers	0

<u>Infill (Cubic Subdivision)</u>

Infill Density	3%
Infill Line Distance	40.0 mm
Infill Pattern	Cubic Subdivision
Infill Line Directions	☐
Randomize Infill Start	☐
Infill Line Multiplier	1
Cubic Subdivision Shell	0.4 mm
Infill Overlap Percentage	10%
Infill Overlap	0.04 mm
Infill Wipe Distance	0.1 mm
Infill Layer Thickness	0.25 mm
Gradual Infill Steps	0
Infill Before Walls	☐
Infill Minimum Area	0.0 mm
Infill Support	☐
Skin Edge Support Thickness	0.0 mm
Skin Edge Support Layers	0

<u>Material</u>

Printing Temperature	210 °C
Printing Temperature Initial Layer	210 °C

Initial Printing Temperature	210 °C
Final Printing Temperature	210 °C
Build Plate Temperature	60 °C
Build Plate Temperature Initial Layer	60 °C
Scaling Factor Shrinkage Compensation	100%
Horizontal Scaling Factor Shrinkage Compensation	100%
Vertical Scaling Factor Shrinkage Compensation	100%
Flow	100%
Wall Flow	100%
Outer Wall Flow	100%
Inner Wall(s) Flow	100%
Top/Bottom Flow	100%
Infill Flow	100%
Skirt/Brim Flow	100%
Prime Tower Flow	100%
Initial Layer Flow	100%

<u>Speed</u>

Print Speed	60 mm/s
Infill Speed	60 mm/s
Wall Speed	30 mm/s
Outer Wall Speed	30 mm/s
Inner Wall Speed	30 mm/s
Top/Bottom Speed	30 mm/s
Travel Speed	120 mm/s
Initial Layer Speed	30 mm/s
Initial Layer Print Speed	30 mm/s
Initial Layer Travel Speed	120 mm/s
Skirt/Brim Speed	30 mm/s
Number of Slower Layers	2
Flow Equalization Ratio	100%
Enable Acceleration Control	☐
Enable Jerk Control	☐

<u>Travel</u>

Enable Retraction	☑
Retract at Layer Change	☐
Retraction Distance	6.0 mm
Retraction Speed	50 mm/s
Retraction Retract Speed	50 mm/s
Retraction Prime Speed	50 mm/s
Retraction Extra Prime Amount	2.0 mm
Retraction Minimum Travel	1.5 mm
Maximum Retraction Count	100
Minimum Extrusion Distance Window	5.0 mm
Combing Mode	All

Avoid Supports When Traveling	☑
Travel Avoid Distance	0.625 mm
Layer Start X	0.0 mm
Layer Start Y	0.0 mm
Z Hop When Retracted	☐

<u>Cooling</u>

Enable Print Cooling	☑
Fan Speed	100%
Regular Fan Speed	100%
Maximum Fan Speed	100%
Regular/Maximum Fan Speed Threshold	10 s
Initial Fan Speed	0.0%
Regular Fan Speed at Height	0.9 mm
Regular Fan Speed at Layer	10
Minimum Layer Time	5.0 s
Maximum Speed	15 mm/s
Lift Head	☐

<u>Support</u>

Generate Support	☐
Build Plate Adhesion	
Build Plate Adhesion Type	Brim
Skirt/Brim Minimum Length	250 mm
Brim Width	8.0 mm
Brim Line Count	20
Brim Distance	0.0 mm
Brim Only on Outside	☑

Printing with TPU

https://the3dprinterbee.com/3d-printing-with-tpu-properties-tipps-best-settings/

TPU is short for *Thermoplastic Polyurethane* and is a **flexible filament**. Flexible materials are suitable for numerous printing projects (like main landing gear and tailwheel tires) due to their elasticity.

Here are the most important TPU settings (compatible with Cura, PrusaSlicer, Creality Slicer and many other slicers):

Setting	Value
Nozzle Size:	0.4 - 0.6 mm (0.6mm nozzle can help reduce clogging & ensure smoother extrusion)
Print temperature:	210 - 230 °C
Print bed temperature:	50 - 60 °C
Print speed:	20 - 30 mm/s
Flow Rate:	95 - 105%
Z-Hop:	Enable the Z-hop option and set it to 0.2 - 0.4 mm lift
Retraction distance:	0.5 - 2 mm
Retraction speed:	20 mm/s
Retraction Min. Travel:	1.5 mm
Layer height:	25 - 50% of the nozzle diameter (i.e. 0.1 to 0.2 mm with a 0.4 mm nozzle)
Line width:	80 - 100% of the nozzle diameter (i.e. 0.32 to 0.4 mm for a 0.4 mm nozzle)
Infill density:	20 - 30%. The higher the stiffer.
Infill pattern:	Gyroid, concentric
Extruder:	Direct drive extruder
Cooling	0% (10 - 50% for overhangs)
Storage:	Airtight (vacuum bag, plastic box)
Print bed:	PEI* and isopropyl alcohol for loosening, blue tape, no raft.
Enclosure necessary?	No
Nozzle:	Bigger is better with Bowden extruders

Through the infill pattern, you can influence the stiffness of the object in different directions. For example, the gyroid pattern is equally stable in all spatial directions, which also makes the finished TPU object evenly flexible. If you want to make one axis of the object more stable than the other two, you can use the concentric infill pattern. It consists of circles formed around one axis. This gives a relatively low stability along the main axis and almost none along the other two.

Due to the flexible nature of TPU, it is also important where the coil is placed relative to the extruder and what the resistance is when the coil is unwound. If the extruder has to pull too hard on the filament, either because the spool is placed under the extruder or the spool does not have good bearings, the filament will be drawn stretched in the extruders. This can lead to under-extrusion. Many 3D printers have a rigid spool holder. Try replacing it with a holder that allows the spool to rotate easily.

TPU has extremely good print bed adhesion to PEI. PEI print bed surfaces are available in various designs. They are smooth or rough, flexible or inflexible. Whether the print bed is flexible is not all that important with TPU, since the flexibility would be compensated for by TPU anyway and you wouldn't be able to release the object any better by flexing the print plate. TPU often adheres too well to PEI, so it can be difficult to remove the object from the print bed after printing. However, if you apply isopropyl alcohol generously to the base of the object, the TPU object will come off easily. Even large objects can be easily removed without leaving any residue. Another option and to increase (or in the case of PEI slightly decrease) the print bed adhesion for TPU is to use Blue Tape. This tape has a roughened surface and provides good adhesion. The downside is that you'll have to replace it after a few prints.

Multicolor 3D Printing with a Single Extruder

Pause at Height and Swap Filament

One of the most common ways to print 3D with multiple colors is to pause the print and to manually swap out the filament with another filament of a different color. Often this is done when the printer increases its Z position to move to a new layer. It is often referred to as the pause at height or pause at layer method. You are not limited to swapping filament colors only once; it is possible to swap filaments many times during a single print to create cool visuals. You can either pause the 3D printer by hand or edit the G-code so that the 3D printer pauses automatically at specific points during the print.

Edit G-code to Pause the Print

If you need more precise control, for example, if you want to swap filament on a layer change, it is best to edit the G-code of the print. This causes the printer to automatically interrupt the print and enter a pause state. One advantage of this method is that there is no need to monitor the 3D printer. Once the printer reaches a specific layer that you want to swap the filament at, it does the pause without any human input. Editing the G-code involves inserting pause commands. You can do this by hand, or you can have the slicer software do it for you.

How to Manually Edit G-code to Pause a Print

To manually edit G-code to set up a 3D print for automatic pausing, you can do the following:

1. First, find in your slicing software the exact layer(s) you want to swap colors at.
2. Generate the regular G-code for the print and open it in a text editor like Notepad.
3. Edit the G-code by adding pause commands. For example, the G-code I add to pause the print is as follows:

```
G91            ; set relative positioning
G1 Z10 F4000; move Z up 10mm
G90            ; set absolute positioning
M600           ; suspend/pause
```

This moves the nozzle up 10mm and then pauses the print. I let the hotend move upwards first so that any filament that oozes out of the nozzle will be in mid-air instead of oozing on the print.

In the example from before, where I want to pause the print at Z2.20, the resulting section of G-code would look something like this:

```
G92 E0.0000
G1 E-1.7000 F3000
; layer 11, Z = 2.200
G91            ; set relative positioning
G1 Z10 F4000; move Z up 10mm
G90            ; set absolute positioning
```

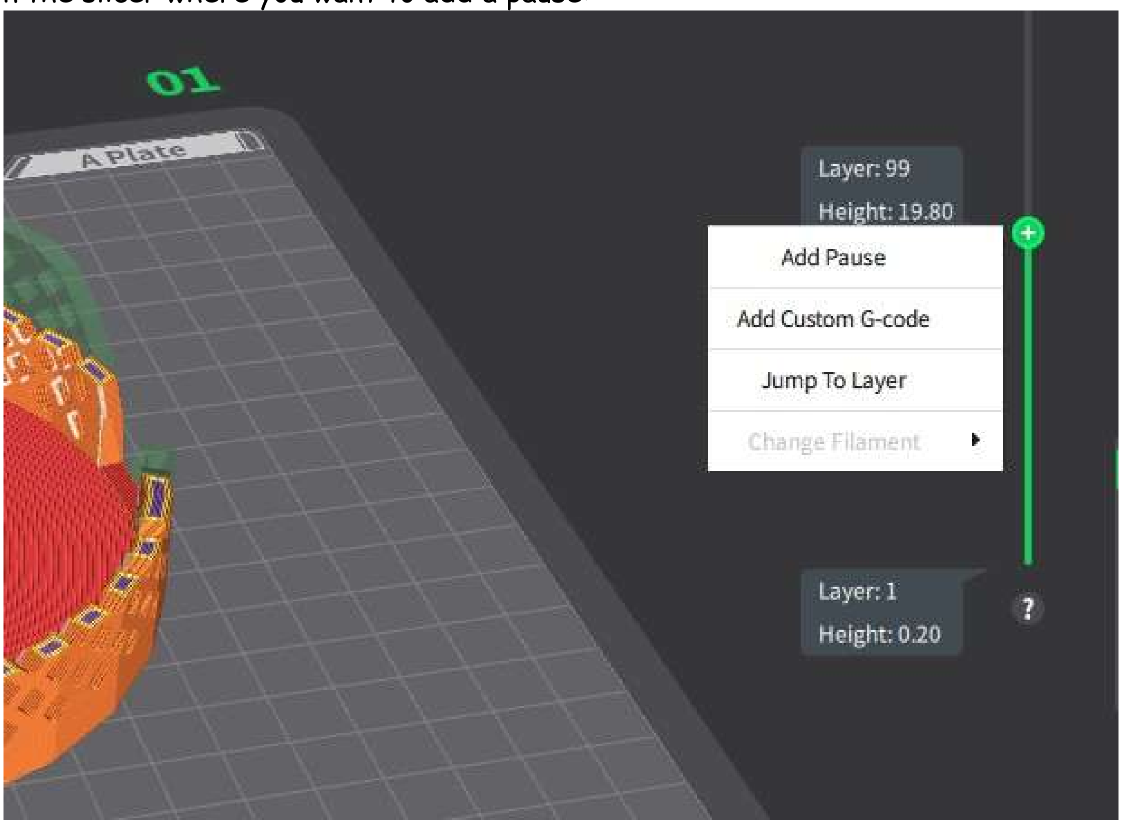
```
M600          ; suspend/pause
; feature outer perimeter
G1 Z2.200 F300
G1 X-2.589 Y3.860 F12000
```

So, what I did there was insert the pause G-code right before the command that moves the 3D printer to Z2.20 (G1 Z2.200 F300).

For the Ender-3 V3 using Creality Print: Simply insert the word PAUSE following the line of code for the ;LAYER: you wish to complete with the OLD color. The printer will pause AFTER that layer finishes and allow you to manually change the filament using "Retract" followed by "Extrude" commands on the touch screen. When the NEW filament is ready to go and you have manually removed the thin strands of filament from the nozzle during the Retract/Extrude routine, hit HOME and then RESUME. The printer will move to its last location and begin printing with the NEW color.

I use this on my Ender-3 V3's:

Right click in the slicer where you want to add a pause:



This is the code that's added. (After you add the pause reslice to make sure it's written in the code.)

Pause-Test

Pause-Test

294×121 5.83 KB

While it's paused you can change filaments using Extrude/Retract as usual. Bed temp will remain hot, but the nozzle will cool.

How to Automatically Insert Pause Commands in G-Code

Aside from manually editing G-code, it is also possible to have the slicer software automatically insert pause commands when the G-code for the print is generated. This saves quite a bit of time and effort when you want to pause a print several times. Slicers usually do this by running a post-processing script after generating the G-code. This script inserts the pause commands at the relevant locations.

Post-Processing in Other Slicers

Other slicers have their own implementation of post-processing scripts. For example, Cura has a built-in post-processing script for pausing at specific heights, whereas Slic3r supports external post-processing scripts in a variety of languages such as Perl, Python, Ruby and Bash.

In short, you need to insert the right post-processing script in your slicer, and make sure that the G-code in the script is correct for your specific 3D printer. In many cases you will be able to find an example pause at layer height script online for your 3D printer that you can use. After setting the right layer heights in the script, you will be able to pause 3D prints at the exact height you want.

How to Change Filament Colors After Pausing the Print

After pausing the print, whichever method you use, the next step is to change to a different filament color. Let's see how to do that.

Before swapping the filament, make sure that the hotend is heated to the extrusion temperature. It is important for the filament in the nozzle is melted, otherwise you might not be able to pull it out.

1. To unload the old filament, press the extruder lever to disengage the drive gear, and use your other hand to pull the filament out. Alternatively, use the printer control panel or host software to reverse the filament out mechanically.
2. To load the new filament, press the extruder lever once again, and insert the new filament as far as possible. Make sure that the filament passes the extruder drive gear, before releasing the lever.
3. Purge the old filament from the hotend. I find the best way to do this is to extrude using the printer's control panel or host software.

Once the color of the extruded filament is the same as the filament going into the hotend, you can resume the print. Make sure to remove any strings of extruded filament from the nozzle with a pair of tweezers first.

To resume the print, once again use the printer's control panel or the host software.

Additional Considerations

Make sure to keep the hotend hot during pausing. This allows you to pull all of the old filament out.

The same goes for the heated bed. If you are using one, make sure that it does not cool down while the printer is paused. This can cause the part(s) to be released, ruining your print.

It is also important to keep the stepper motors of your 3D printer on/engaged while the print is paused, so that the hotend can resume printing from the exact X/Y coordinates it paused at. Turning the stepper motors off makes it possible for the extruder carriage to move. If it moves, the print will shift on the next layer after resuming.

You can often set the pause behavior for the three above items in the printer's firmware. That way you don't need to set them manually every time you pause a print.

Editing STL Files using Fusion 360

Step 1: Import STL File
Click on the + button on the top bar to pick a new design.
Click on the Create button from the menu bar and a drop-down menu will be displayed.
By clicking on the Create Base Feature from the drop-down menu, it will turn off all the extra features and design history will not be recorded.
Click on the Insert > Insert Mesh, browse your STL file, and open to import it.

Step 2: Edit & Modify STL File
Once the file is imported, an Insert Design box will appear on the right side to change your model's position using a mouse or inserting numerical inputs.
Right-click on the model and click on Mesh to BRep > OK to convert it into a new body.
Click on Model > Patch from the top left corner to remove unnecessary facets.
Click Modify > Merge, select the facets you want to remove and click
Click on Finish Base Feature to go back into regular mode.
Click Modify > Change Parameters, click the + button, and modify parameters as you want.
Click on Sketch and put a center using angles.
Go to Create > Pattern > Pattern on Path, modify the settings and parameters according to your need.

Step 3: Export STL File
Go to the save icon on the top bar, give a name to your file and click
Go to the left side window, Right Click > Save as STL > OK > Save.

Check out the video below for a tutorial for modifying STL files.
https://youtu.be/DWv2uK-XpA4

Converting a Mesh to a Solid in Fusion 360

1. Open **Fusion 360** and go to the **File** menu.
2. Select **Open** and import the desired **STL file**.
3. Switch to the **Mesh** workspace by selecting it from the top toolbar.
4. Use the **Prepare** tools in the **Mesh** toolbar to clean up or repair the mesh body.
5. If needed, convert the mesh to a solid body by selecting **Modify** > **Convert Mesh**.
6. Choose the **BRep** option to convert the mesh into a solid for further editing.
7. Use the **Modify** tools (e.g., **Extrude**, **Cut**, or **Fillet**) to make changes to the solid body.
8. Save your edited file by selecting **File** > **Save As** and choosing the desired format.

Converting a Mesh to a Solid in Fusion 360

1. Open the **Design** workspace in Fusion 360.
2. Click on the **Mesh** tab in the toolbar.
3. Select **Modify** > **Convert Mesh**.
4. In the canvas, select the mesh body you want to convert.
5. In the **Convert Mesh** dialog, choose an **Operation**: Parametric: Maintains upstream parametric relationships. Base Feature: Does not maintain upstream parametric relationships.

6. Choose a **Method**: Faceted: Converts each face of the mesh to a corresponding face on the solid body. Prismatic: Merges groups of faces into singular faces for prismatic features. Organic: Converts the mesh into an organically shaped solid (requires Fusion Design Extension).
7. Adjust additional settings if needed: For Organic, select a Resolution (e.g., Low, Medium, High, Precise) or specify the Number of Faces. Optionally, check Preprocess Holes to refine open hole boundaries.
8. Click **OK** to complete the conversion.

Tips for Best Results:

- Ensure the mesh has fewer than 10,000 facets to avoid conversion issues.
- Use tools like **Repair**, **Generate Face Groups**, and **Direct Edit** to refine the mesh before conversion.

If the mesh is watertight, it will convert to a solid body. Otherwise, it will convert to a surface body, which can be stitched into a solid using surface modeling tools.

3D Model a Canopy Step-by-Step Workflow in Fusion 360

Gather References:
Import side/top view drawings or blueprints of your RC airplane into Fusion 360.
Scale them to match your fuselage dimensions.

Create Base Sketches:
Draw the canopy outline on the fuselage using the Sketch tool.
Define cross-sections (front, middle, rear) to capture the canopy's curvature.

Use Loft or Sweep:
Apply the Loft tool between multiple profiles to generate the canopy's curved surface.
Alternatively, use Sweep if you have a guiding path for the canopy's arc.

Refine the Shape:
Use Fillet to smooth sharp edges.
Apply Surface Patch or Surface Trim to clean up transitions between canopy and fuselage.

Hollow the Canopy:
Use the Shell tool to create a thin-walled canopy suitable for 3D printing.
Adjust thickness depending on material strength (usually 1–2 mm for lightweight RC parts).

Add Details:
Model framing lines or cutouts for cockpit visibility.
Use Split Body if you want detachable canopy sections.

Prepare for Printing:
Export as STL.
Orient the canopy in your slicer for minimal supports and best surface finish.

Pro Tips:
Symmetry: Use the Mirror tool to ensure both sides of the canopy are identical.
Fit Check: Model the canopy as a separate body so you can test-fit it digitally with the fuselage.
Printing Strategy: Consider printing the canopy in clear PETG or resin for transparency.

Step-by-Step Bubble Canopy Example

Set Up Reference Geometry:
Open Fusion 360 and create a new component called Canopy.
Import or sketch the fuselage outline (side view) so you know where the canopy will sit.

Sketch the Canopy Base:
On the Top Plane, sketch an oval or rounded rectangle that matches the footprint of the canopy on the fuselage.
This will be the bottom edge of your canopy.

Create Cross-Section Profiles:
On the Front Plane, sketch a curved profile (like a dome shape).

On the Mid Plane (halfway along the canopy length), sketch another profile, slightly flatter.
On the Rear Plane, sketch a smaller curve tapering down to meet the fuselage.

Loft the Canopy:
Use the Loft tool: select the front, mid, and rear profiles.
This generates a smooth bubble canopy surface.

Refine the Shape:
Apply Fillet to soften edges where the canopy meets the fuselage.
If needed, use Surface Patch to close gaps and then Stitch to make it a solid body.

Hollow the Canopy:
Use the Shell tool to hollow the canopy to ~1.5 mm thickness.
This makes it lightweight and printable.

Add Details:
Sketch canopy framing lines (thin ribs) and extrude them slightly.
Use Split Body if you want the canopy detachable.

Prepare for Printing:
Export as STL.
Orient canopy upside down in slicer for best surface finish.

Advanced Detailing Steps

Add Canopy Framing:
Create a new sketch on the canopy surface.
Draw thin lines where the frame would be (like cockpit window divisions).
Use Extrude with a small thickness (0.5–1 mm) to make raised frame ribs.
Apply Fillet to soften edges so they look realistic.

Design Hinges:
Sketch hinge outlines on the fuselage and canopy.
Use Extrude or Revolve to create cylindrical hinge pins.
Position them so the canopy can pivot open.
Use Assemble → Joint to test hinge movement digitally.

Latch Mechanism:
Create a small rectangular cutout in the fuselage for a latch.
Model a simple tab on the canopy that fits into this cutout.
This ensures the canopy stays secure during flight.

Transparency for Printing:
If you plan to print in clear PETG or resin:
Keep canopy walls thin (1–1.5 mm).
Avoid sharp corners — smoother curves print clearer.
Orient canopy upside down in slicer to minimize support marks.

Split Body for Detachability:
Use Split Body to separate canopy from fuselage.
This lets you print them separately and assemble later.
Add alignment tabs so the canopy snaps into place.

Tips for RC Builders:
Weight balance: Keep canopy light to avoid shifting CG (center of gravity).
Material choice: PETG for durability, resin for clarity, PLA for easy prototyping.
Test fit: Print a small section first to check hinge/latch tolerances.

OA-1K Skyraider II Canopy Modeling Workflow in Fusion 360

The OA-1K Skyraider II has a distinctive canopy with a long, slightly angular profile and raised framing. Here's how to replicate it:

1. Import Reference Images
Use side, top, and front view drawings of the OA-1K.
Insert them as canvases in Fusion 360 and scale them to match your canopy dimensions.

2. Sketch the Canopy Footprint
On the top plane, sketch the canopy base outline — typically a rounded rectangle that tapers toward the rear.
Use the Project tool to align it with the fuselage.

3. Create Profile Sketches
On vertical planes (front, mid, rear), sketch cross-sections of the canopy:
Front: Slightly curved dome.
Mid: Flattened arch.
Rear: Tapered down to fuselage.

4. Use Loft to Shape the Canopy
Select the profiles and apply the Loft tool to generate a smooth canopy surface.
Use Rails if needed to control curvature along the sides.

5. Shell the Canopy
Convert the surface to a solid body using Thicken.
Apply Shell to hollow it out to ~1.5 mm wall thickness — ideal for lightweight RC printing.

6. Add Framing Ribs
Sketch framing lines on the canopy surface.
Use Split Face and Extrude to create raised ribs.
Apply Fillet for smooth transitions.

7. Integrate with Fuselage
Use Split Body to separate the canopy from the fuselage.

Add alignment tabs or hinge pins for detachable mounting.
Test fit using Assemble → Joint to simulate movement.

Materials & Printing Tips
Material: PETG clear filament for transparency; LW-PLA for lightweight builds.
Orientation: Print canopy upside down to reduce support marks.
Finish: Sand and polish for clarity if using transparent filament.

References:
How to Model an Aircraft in Fusion 360 | Tutorial 2 - Fuselage | Step-by-Step (4K).
https://www.youtube.com/watch?v=eBTAxZGYUug

1000mm Air-Tractor AT-802 (+ sky warden) - RC Groups.
https://www.rcgroups.com/forums/showthread.php?4413841-1000mm-Air-Tractor-AT-802-(-sky-warden)

Free Fusion 360 Tutorials - 3D Modeling Airplane Course | RC CAD.
https://www.rccad2vr.com/fusion-360-aviation-design-tutorials

Creating an airplane canopy - Autodesk Community - Fusion.
https://forums.autodesk.com/t5/fusion-design-validate-document/creating-an-airplane-canopy/td-p/10916325

Master Your RC Plane Building Skills with Fusion 360 - YouTube.
https://www.youtube.com/watch?v=ib73dIZiNZ4

Editing STL Files using MeshMixer

Step 1: Import STL File
Click on Import, browse your computer, and open the STL file.

Step 2: Edit & Modify STL File
Click Select and mark different parts of your model.
Press Del from the menu to delete or remove the unnecessary marked tiles.
To open different forms for the model, go to the Meshmix
You can select various options from the sidebar, such as letters.
Click on Stamp, choose the patterns, and draw them on the model using your mouse.
To smooth or extrude different parts of the model, go to the Sculpt

Step 3: Export STL File
Go to File > Export > File Format (.stl).

Check out the video below for a tutorial for modifying STL files.
https://youtu.be/WMyxVyQ9nFM

Step-by-Step Guide to Editing STL Files in Tinkercad

1. Create an Account and Start a New Project:
   If you haven't already, create an account on Tinkercad's website. Once logged in, open a new project from your dashboard.
2. Import the STL File:
   Click on the "Import" button located at the top right corner of the workspace.
   Select the STL file from your device that you wish to edit. You can adjust the scaling options if necessary before clicking "Import" again. The model will appear in your workspace.

3. Familiarize Yourself with the Interface:
   Ensure you understand the layout of Tinkercad, including the shapes panel on the right, which contains various geometric forms you can use to modify your STL file.

4. Editing the STL File:
   Positioning: Click and drag the imported object to the desired location on the grid. Use the ruler tool for precise placement.
   Resizing: Select the model to see white handles around it. Click and drag these handles to resize the object while maintaining proportions.
   Adding Shapes: To add or cut portions of the model, drag shapes from the shapes panel into the workspace. Position them where you want to make alterations.
   Creating Holes: Use the “Hole” feature to hollow out sections of your model. Select a shape, convert it to a hole using the inspector panel, and position it on the STL model. Group the two shapes together to execute the cut.
   Grouping Objects: After making all adjustments, select all components by holding down the Shift key and clicking on each object, then click the “Group” button to merge them.

5. Exporting the Modified File:

Once satisfied with your modifications, click the “Export” button located at the upper right. Choose the “.STL” option from the available formats to save the modified design to your device.

By following these steps, you can effectively modify STL files in Tinkercad, allowing for custom designs and adjustments to your 3D models. Happy designing!

Dark Techno Odonatopter Gunship

https://www.cgtrader.com/3d-print-models/miniatures/sci-fi/dark-techno-odonatopter-gunship

This multi-part kit contains all you need to build a set of flyer vehicles for the Dark Techno army.

The kit comes with various options :
multiple cockpit versions, with/without interior detail, and with flat/curved glass. A PDF with cutout for clear sheets is also included
2 hull versions with/without interior
2 different wing styles (some parts are shown in clear on images, but can of course be printed in solid colours
3 back turret weapons (lasers, machineguns, bombs)
2 front weapons options (heat, machinegun)
doors on side of hull with/without gunner crew.
3 font legs poses (retracted, extended, lowered)
It fits on standard cross shaped stems, or 8mm acrylic rod.

Models are scaled for 28/30mm gaming. Parts require support. Recommended resin printing, or **FDM with a 0.2mm nozzle.**

Scarab Beetle Box (with secret lock) Remix with magnets and customized "Catch_drawer" and "catch_lid"

https://www.thingiverse.com/thing:3718159

I'd like to extend my congratulations to LOUBIE (Louise Driggers) for the outstanding design of the scarab beetle box. It truly looks exceptional. However, I encountered a problem with the closing mechanism - it didn't work for me. To rectify this issue, I made some adjustments to a few parts. In the "Catch_drawer" and in the "middle_part", holes were added for mounting 4 neodymium magnets (4 x 2mm). These neodymium magnets are available on Ebay, for example, at https://www.ebay.com/itm/303166219032 - 50 pieces for $1.85. Additionally, I made adjustments to the closure parts "Catch_drawer" and "catch_lid". It's recommended that these two parts be printed with support structures, as the 10° angle of the nose required for proper function may not be printed accurately enough without them. With these modifications, the closing mechanism now works perfectly. All parts were printed at a layer height of 0.1. The additional components can be found on https://www.thingiverse.com/thing:521463

The scarab beetle is a small model with a secret compartment which is accessed by pulling back a "drawer" and opening up the beetle's wing case. The box is locked by closing the wing case and pushing the drawer back into place. Another Thingiverse user (sherpa_chris) came up with the idea of fixing the "head jewel" differently, so it can be used as a button to open the wing case (providing the drawer is pulled back) which enhances the model in my opinion. You can see his make here:
http://www.thingiverse.com/make:101797

This model is styled after the "Egyptian Revival" of the 19th and early 20th centuries rather than Ancient Egypt. For more information, visit this excellent page:

http://www.metmuseum.org/toah/hd/erev/hd_erev.htm

It is worth researching this style as it is beautiful and striking.
To the ancient Egyptians, the scarab beetle was symbolic of the sun (Ra) moving across the sky every day (the beetle rolling a ball of dung was seen to represent this). My model has a stylized beetle with a winged Ra between his two front feet.
The model consists of 12 parts which must be fitted together. All piece's print without support.
It also has my design logo on the bottom of it:-)

UPDATE: I have uploaded a third version of the drawer file after it was noted that it had a tendency to fall open but the second version was too big. The file drawer_v3.stl file has a height of 12.45mm whereas the fitting for it int he middle_part.stl file is 12.5mm. This may require some sanding Thanks to MacGyver for pointing out the original issue to me and Botmaster for alerting me to the second problem.

Instructions
The assembly instructions are here:
http://www.instructables.com/id/The-Scarab-Beetle-Box-Assembly-Instructions/
Please read these instructions as they will contain any updates I have had to make and instructions for modified files.

Double Native American Flute in the key of F#

https://www.printables.com/@FluteCraft_427746

https://www.printables.com/model/452234-double-flute

Printing recommendations:
The whole model is designed to be printed without support. The screenshots show the correct positioning on the table. The height of each model does not exceed 180mm.

Speed: 60mm/s
Print in 3 perimeters!
Infill: 15%
Supports: No
Material: PLA

The model is designed with a 0.1mm offset. On calibrated printers the fit will be perfect. With a larger offset, the sound is not good. So, if your printer is not calibrated, you will have to work with sandpaper on the joints. I used some silicone grease to make the connection easier. The model consists of 4 parts and a sound channel cover. I suggest printing one model at a time. There is also a decorative element of the head of an eagle. It consists of 2 halves, which then have to be glued. You can play without it; it is just a decoration.

After printing, the holes for the fingers should be calibrated with a drill of the appropriate diameter and chamfered. Each time after drilling, the chamfer should be removed with a chamfer knife. If this is not done, the note will sound lower.

For the best results, you can download a guitar tuner and check each note while tuning. This is an important step; you need to tune responsibly for good tuning and sound. The tuning should be done from the bottom up. The fundamental note is played with all the holes closed! You can tune the next note only after the previous one is finished.