# English: How to print with Hueforge

## Information

### Introduction to Hueforge

Hueforge uses information about the color and transparency (TD: Transmission Distance) of filaments to help us reconstruct an image from an FDM 3D printer, a technique called filament painting.
It works on the same principle as overlapping transparent sheets of different colors.
The 3D model is therefore a succession of layers of filaments.

### Layer thickness 0.08 vs. 0.04

Layer thickness 0.08:

- Prints more easily and quickly
- Perfect for images with well-separated colors (anime, manga, etc.)
- Less effective on images with shadows and gradients (photos, etc.)

Layer height 0.04:

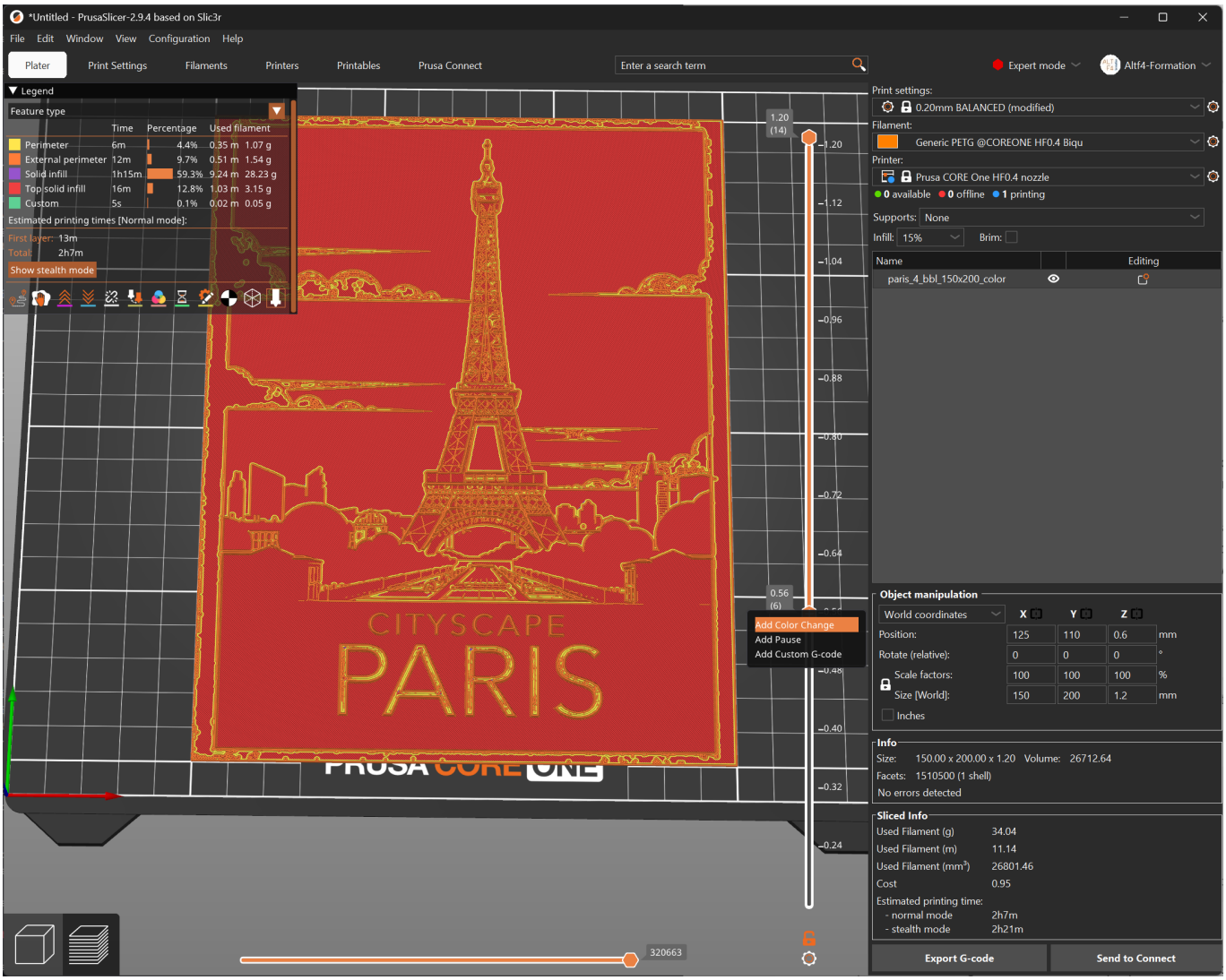
- Takes longer to print
- Better rendering on images with shadows and gradients
- More difficult to print; it is often necessary to calibrate the filaments to achieve the best result

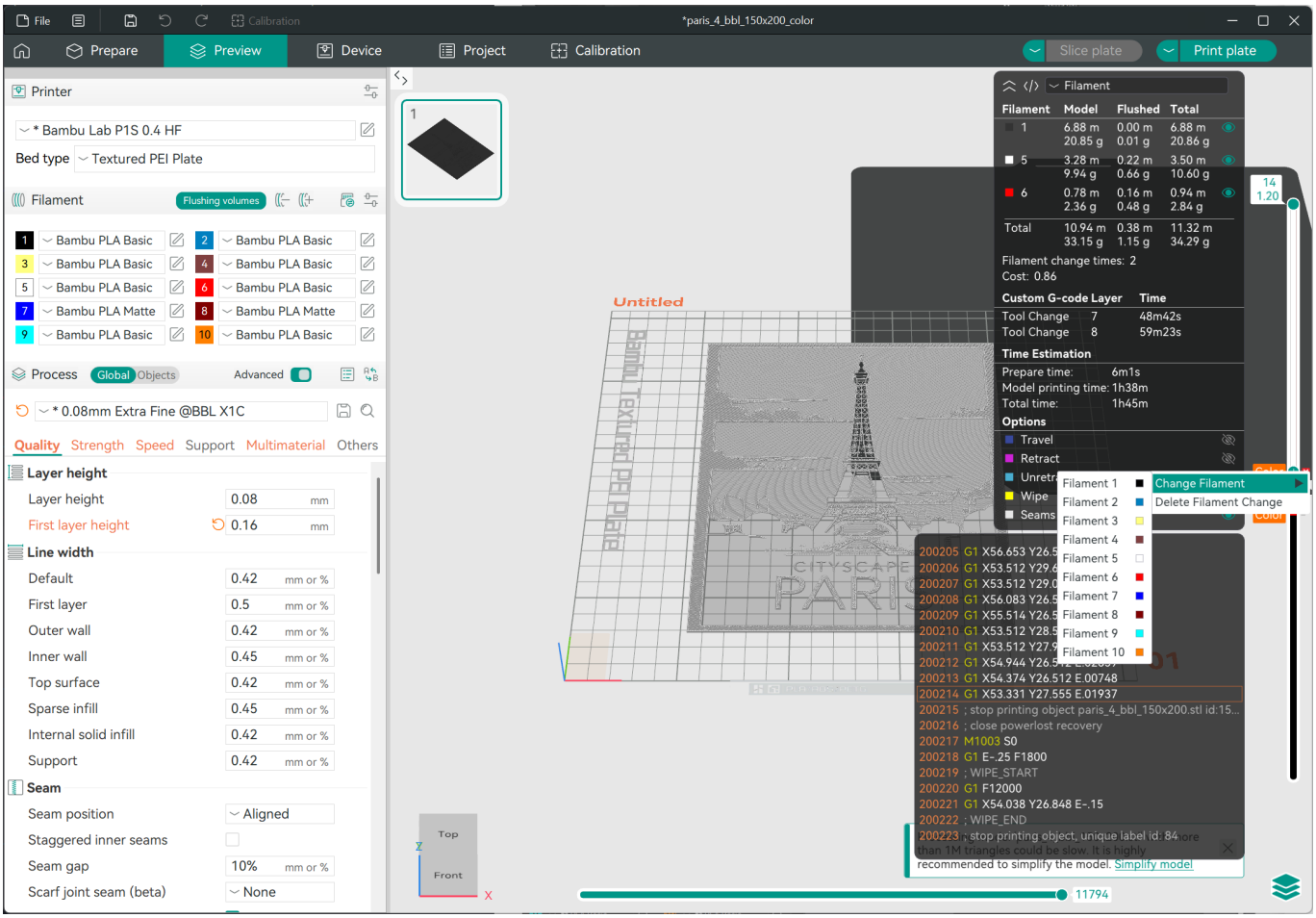
## From a print profile

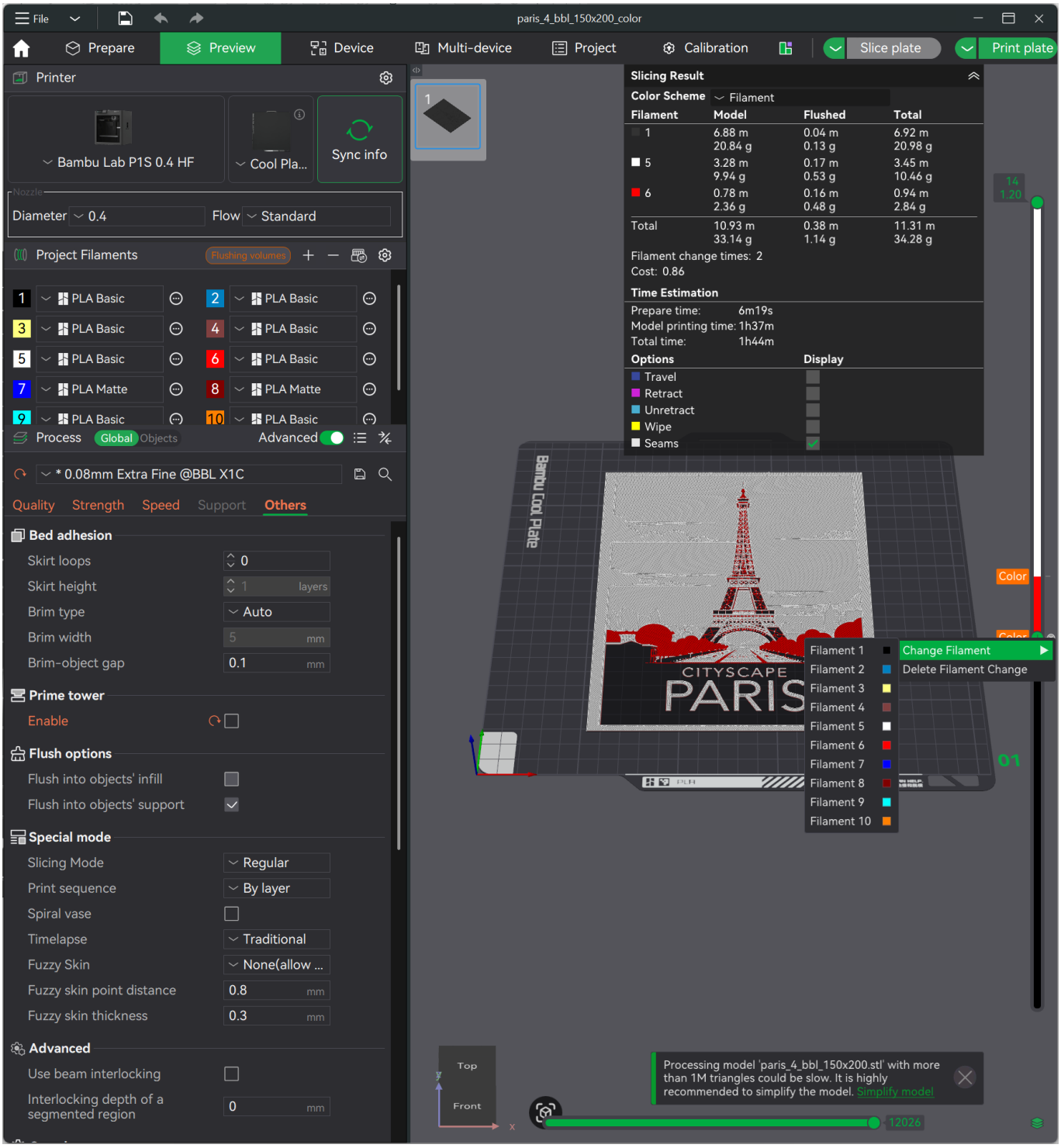
### With MMU, AMS, CFS, etc

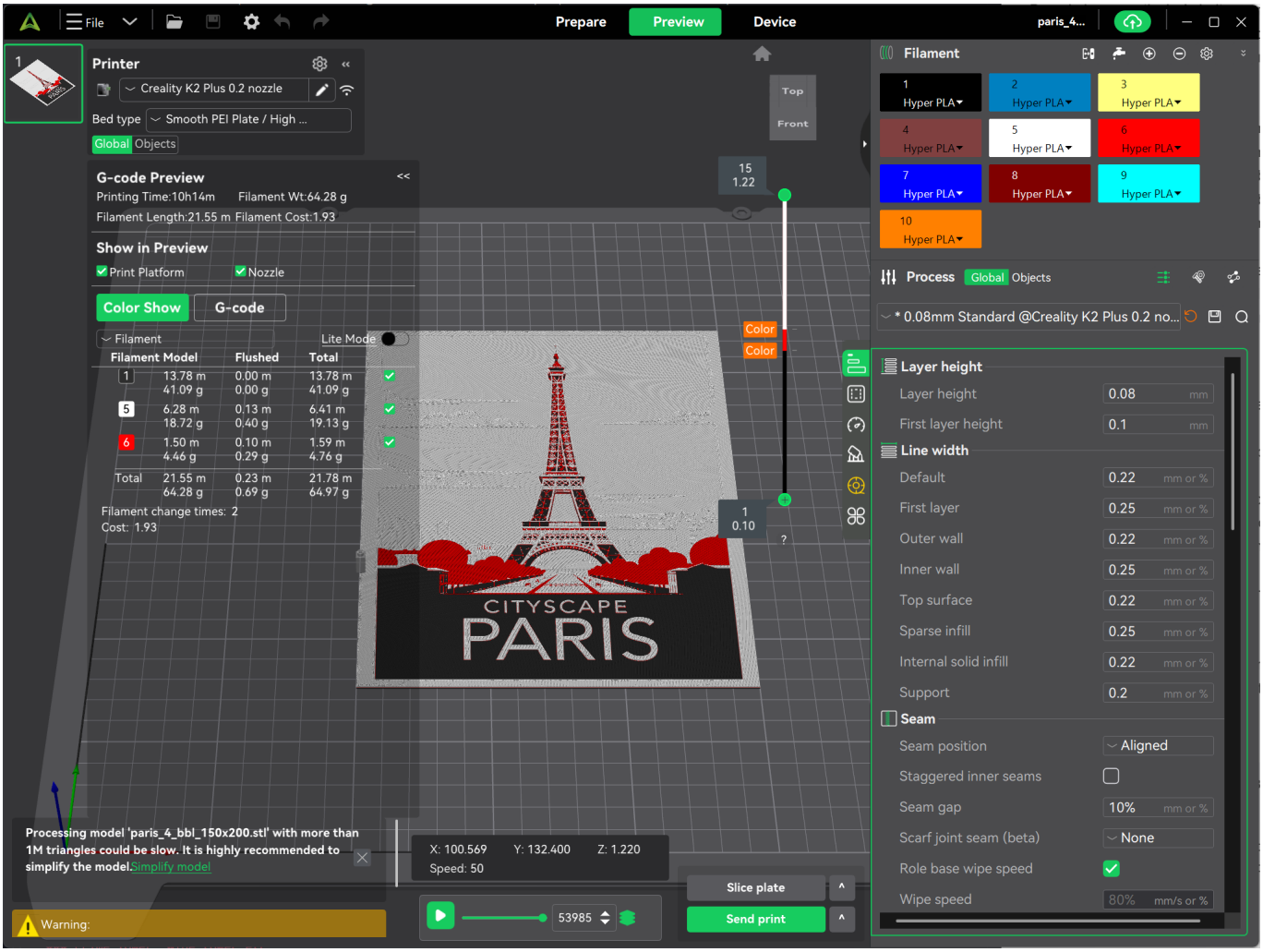
Open the print profile in your slicer and “slice”; you should see the color changes already configured in the vertical bar on the right.
Depending on the slicer, you may need to reconfigure the colors correctly directly in the slicer by right-clicking (usually on the plus sign) and changing the color in the bar on the right. Feel free to compare the documentation indicating at what height to change colors to verify that there are no errors in the print profile.
Change the settings in the slicer if necessary (speed, etc.).
All that's left to do is start printing and, if necessary, associate the colors with the filaments in your AMS/CFS...

Images example


*Untitled - PrusaSlicer-2.9.4 based on Slic3r
CITYSCAPE
PARIS
Add Color Change
Add Pause
Add Custom G-code
Export G-code
Send to Connect


*paris_4_bbl_150x200_color
Prepare
Preview
Device
Project
Calibration
Slice plate
Print plate
Untitled
Change Filament
Delete Filament Change
Simplify model


File
paris_4_bbl_150x200_color
Prepare
Preview
Device
Multi-device
Project
Calibration
Slice plate
Print plate
Printer
Bambu Lab P1S 0.4 HF
Cool Pla...
Sync info
Nozzle
Diameter 0.4
Flow Standard
Project Filaments
Flushing volumes
1 PLA Basic
2 PLA Basic
3 PLA Basic
4 PLA Basic
5 PLA Basic
6 PLA Basic
7 PLA Matte
8 PLA Matte
9 PLA Basic
10 PLA Basic
Process Global Objects
Advanced
* 0.08mm Extra Fine @BBL X1C
Quality Strength Speed Support Others
Bed adhesion
Skirt loops 0
Skirt height 1 layers
Brim type Auto
Brim width 5 mm
Brim-object gap 0.1 mm
Prime tower
Enable
Flush options
Flush into objects' infill
Flush into objects' support
Special mode
Slicing Mode Regular
Print sequence By layer
Spiral vase
Timelapse Traditional
Fuzzy Skin None(allow ...
Fuzzy skin point distance 0.8 mm
Fuzzy skin thickness 0.3 mm
Advanced
Use beam interlocking
Interlocking depth of a segmented region 0 mm
Slicing Result
Color Scheme Filament
Filament Model Flushed Total
1 6.88 m 20.84 g 0.04 m 0.13 g 6.92 m 20.98 g
5 3.28 m 9.94 g 0.17 m 0.53 g 3.45 m 10.46 g
6 0.78 m 2.36 g 0.16 m 0.48 g 0.94 m 2.84 g
Total 10.93 m 33.14 g 0.38 m 1.14 g 11.31 m 34.28 g
Filament change times: 2
Cost: 0.86
Time Estimation
Prepare time: 6m19s
Model printing time: 1h37m
Total time: 1h44m
Options Display
Travel
Retract
Unretract
Wipe
Seams
Bambu Cool Plate
CITYSCAPE
PARIS
Filament 1
Filament 2
Filament 3
Filament 4
Filament 5
Filament 6
Filament 7
Filament 8
Filament 9
Filament 10
Change Filament
Delete Filament Change
Color
Top
Front
Processing model 'paris_4_bbl_150x200.stl' with more than 1M triangles could be slow. It is highly recommended to simplify the model. Simplify model




## Without MMU, AMS, CFS, etc

Open the print profile in your slicer and “slice.”
Use the documentation indicating at what height to change filaments
In the vertical bar on the right, select the + and move it up to the height of the first color change (0.88 in the following example), add a pause, and continue until you have configured all the pauses for color changes
**Example** : Swap Instructions:
Start with BambuLab Basic Black
At layer #18 (0.88mm) swap to BambuLab Basic Blue Gray
At layer #25 (1.16mm) swap to BambuLab Basic Black
At layer #26 (1.2mm) swap to BambuLab Basic Red
At layer #36 (1.6mm) swap to BambuLab Basic Blue Gray
At layer #44 (1.92mm) swap to BambuLab Basic Jade White for the rest.
Change the settings in the slicer if necessary (speed, etc.).
Slice.
All that's left to do is start printing and change the filaments at each pause in the order indicated.

# From an STL

## With MMU, AMS, CFS, etc

Open the STL file in your slicer.
Set the layer height and initial layer height, and the infill percentage as indicated in the hueforge documentation (Example: Print at 100% infill with a layer height of 0.04mm and a base layer of 0.2mm).
Optional but recommended: based on the Hueforge information, feel free to configure a Height Range Modifier (right-click on the Height Range Modifier model) with a layer height as indicated (base layer of 0.2mm) from 0 to what is indicated ("You may print at higher layer heights below the Base Thickness of **0.8mm**").
Both values must be multiples (0.16 and 0.48, 0.16 and 0.64, 0.16 and 0.80, 0.20 and 0.60, etc.).
"Slice"
In the vertical bar on the right, select the + and move it up to the height of the first color change (0.88 in the following example), right-click to change filament and select the filament (it must exist in your filament list) and continue until you have configured all the color changes
**Example**: Swap Instructions:
Start with BambuLab Basic Black
At layer #18 (0.88mm) swap to BambuLab Basic Blue Gray
At layer #25 (1.16mm) swap to BambuLab Basic Black
At layer #26 (1.2mm) swap to BambuLab Basic Red
At layer #36 (1.6mm) swap to BambuLab Basic Blue Gray
At layer #44 (1.92mm) swap to BambuLab Basic Jade White for the rest.
Slice.
If you are in filament display mode in the preview, you should have a rough preview of the hueforge (the slicer does not know how to mix colors).
If things look strange, check the heights of the filament changes.

## Without MMU, AMS, CFS, etc

Open the STL file in your slicer.
Set the layer height and initial layer height, and the infill percentage as indicated in the Hueforge documentation (Example: Print at 100% infill with a layer height of 0.04mm and a base layer of 0.2mm).
Optional but recommended: based on the Hueforge information, feel free to configure a Height Range Modifier (right-click on the Height Range Modifier model) with a layer height as indicated (base layer of 0.2mm) from 0 to what is indicated ("You may print at higher layer heights below the Base Thickness of **0.8mm**").
Both values must be multiples (0.16 and 0.48, 0.16 and 0.64, 0.16 and 0.80, 0.20 and 0.60, etc.). Slice.
Use the documentation indicating at what height to change filament
In the vertical bar on the right, take the + and move it up to the height of the first color change (0.88 in the following example), add a pause, and continue until you have configured all the pauses for color changes
**Example**: Swap Instructions:
Start with BambuLab Basic Black
At layer #18 (0.88mm) swap to BambuLab Basic Blue Gray
At layer #25 (1.16mm) swap to BambuLab Basic Black
At layer #26 (1.2mm) swap to BambuLab Basic Red
At layer #36 (1.6mm) swap to BambuLab Basic Blue Gray
At layer #44 (1.92mm) swap to BambuLab Basic Jade White for the rest.
Change the settings in the slicer if necessary (speed, etc.).
Slice.
All that's left to do is start printing and change the filaments at each pause in the order indicated.

## Prime Tower, Wipe Tower, etc

If your printer generates poops between each color change, it is not necessary to use Prime Tower, Wipe Tower, etc.

# Printing issues

## Things to check

- Check the initial layer height
- Check the layer height
- Check that the infill is set to 100%
- Try a slower print speed

## "Waves" on my print

Filament compression can be caused by:

- A nozzle that is too close to the plate (on the Creality K2+, I had to change the offset by 0.1 mm)
- A printing speed that is too fast (on my two P1S printers, one prints perfectly at 250 mm/s (internal solid infill) and the other I have to reduce to 220 mm/s or even 200 mm/s, on the Creality K2+ 125mm/s. I now use 200 mm/s in my printing profiles
- Filament flow rate too high
  You can use a height range modifier over the entire height of the Base Thickness (for example, layer height of 0.2 from 0 to 0.8 with 0.8 base thickness), this eliminates the wave problem on the first layers and speeds up printing

## Incorrect layer number

Depending on the print profile settings, the layer number may not be correct.
For optimization reasons, as indicated:
"You may print at higher layer heights below the Base Thickness of 0.8mm."
if you set the first layer to 0.2 and a layer height of 0.04 for the following layers, you need 0.20 + (0.04 * 15) layers to reach 0.80 (16 layers)
Or you can use a Height Range Modifier (HRM) that allows you to define, for example, layers higher from 0.20 to 0.8mm and you need
0.20 + 0.20 + 0.20 + 0.20 to reach 0.80mm (4 layers), which is faster and for some people reduce the risk to get some waves (filament compression) at the start of printing
So the number of layers is lower (12 less) than in the first case.
It is preferable to use the indicated height in parentheses rather than the number of layers, as the height is always correct even if there is a height range modifier.

## Thin or missing filament in small areas

If the filament is under-extruded, it is necessary to increase the flow rate (or equivalent) and/or the pressure advance (or equivalent).
Try to calibrate the filament for the layer height used.
Hueforge 0.04 layers require more attention to certain filament settings, and in some cases a lower print speed than 0.08 layers.

## Warping

- Make sure the platen is clean
- Deactivate auxiliary fans

## Stringing

- Set shrink length to 0.4mm
- Reduce filament printing temperature