FELIX BIOprinters

Quick Start Guide

BIOprinting with Simplify3D

SIMPLIFY3D®

www.FELIXprinters.com/support
support@felixprinters.com

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands

1 Introduction

The goal of this guide is to explain how to use Simplify3D (software that prepares CAD files) together with the FELIX BIOprinter and ensure you get successful prints as quickly as possible for your specific case.

The following topics are covered.

- Installation of software
- General 3D printing Process
- Preparing the printer and software for 2 example cases
    - Single head (simple)
    - Dual head print with UV curing (complex)
- Tips and tricks

For more in depth information about simplify3D or 3D printing in general, go to the following resources:
www.FELIXprinters.com
www.Simplify3D.com

If you are unable to continue or have any questions, you can check at the support section of our website or you can contact us directly:

Website: www.felixprinters.com/support
Email: support@felixprinters.com
Telephone: +31 (0)30 30 31 387
Address: Zeemanlaan 15, 3401MV IJsselstein, The Netherlands

2 Contents

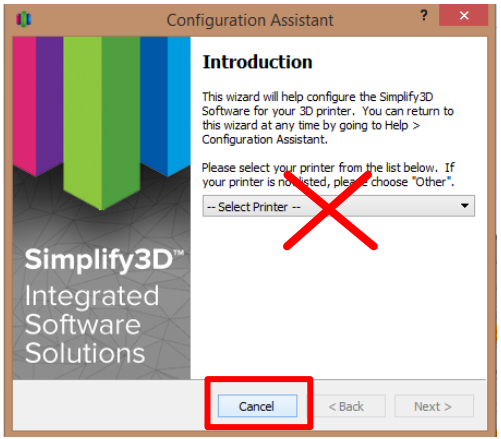
3 Install Simplify3D

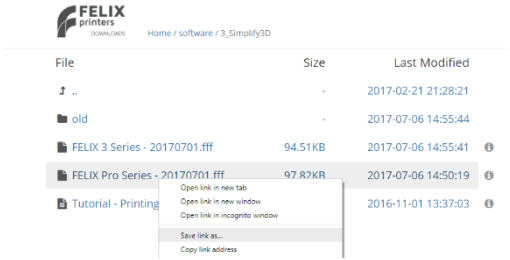
- ✓ **Download and Install simplify3D**, follow instructions on the supplied voucher card to activate your product.

- ✓ **Run simplify3D**
- ✓ Press **Cancel** when the configuration assistant pops-up after startup of the program. **Use the printer settings (slice profiles) provided by FELIXprinters**. They are most up to date and give best print results.

- ✓ Get slice profiles from:
  - **SD card** provided with the printer.
    - Go to folder Software\Simplify3D
    - Locate the *.fff file corresponding to your printer.
  - **Download section FELIXprinters website,** www.felixprinters.com/downloads
    - Go to folder Software -> 3_Simplify3D. Press right mouse click and choose **save as...** on the **.fff** file for your printer.

IMPORTANT:
Please note that using correct slice profiles for your printer is essential for successful 3D printing and stable operation.



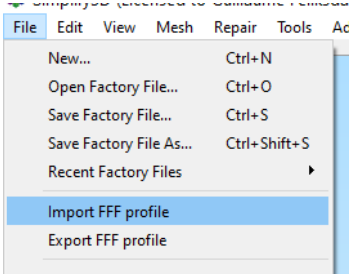


4 Import slice profiles

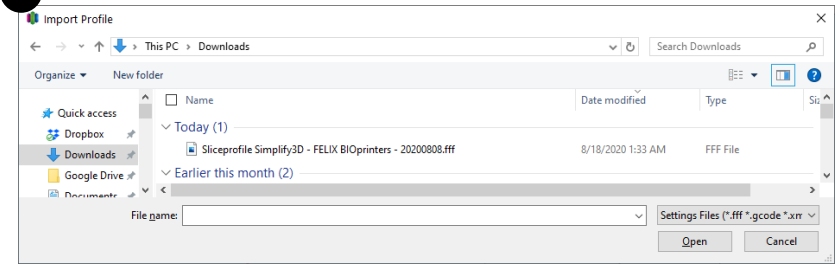
1. After opening Simplify3D choose **File** and then *import FFF profile.*
2. Navigate to the *.fff file for your printer and open it.
3. A new window will pop-up with printing process settings, for now press OK.

> **WARNING:**
> Make sure to use correct slice profiles. It can make or break your 3D print/printer.

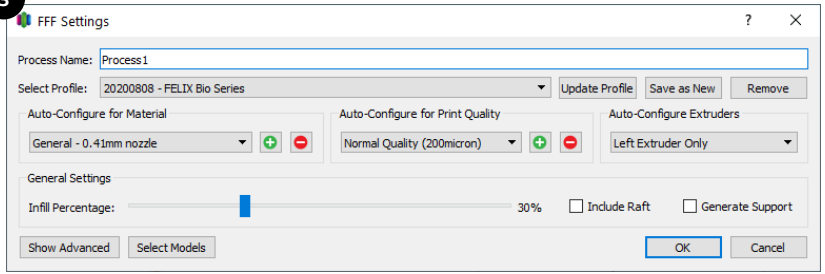
1



2



3



5 3D printing process

In general the 3D printing process from design to printed object is followed for each 3D print.

1. Import CAD file

Simplify3D software generally accepts STL files.

2. Prepare printer and software parameters

Prepare the printer and software parameters for the application.

3. Slice and inspect

The object is cut in thin layers and paths are calculated per layer for printing.

4. Print

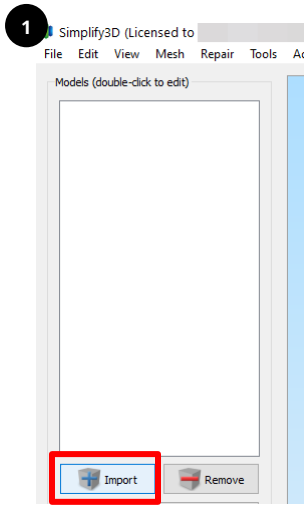
The printer will extrudes material on the calculated path, layer by layer. Till you have a finished product

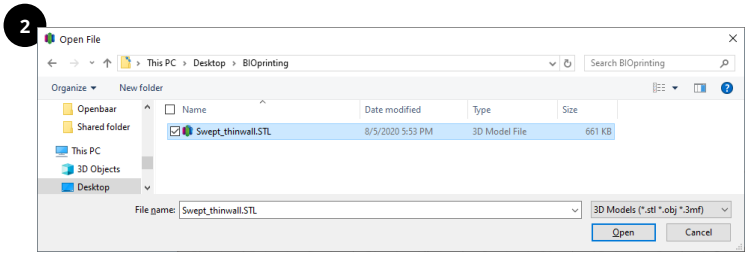
Each process is described in detail in following chapters. To make it easy to understand two example cases (simple and complex) are described, with intention to give insight in how to address all possible features of the FELIX BIOprinter.

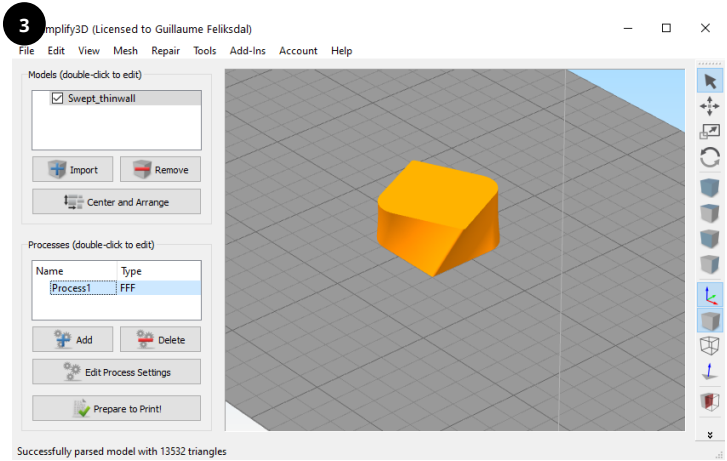
6 Single head print

6.1 Import STL file

- ✓ Press the import button to import an STL file.
- ✓ In this example, we take the *swept_thinwall.STL, locate this file on the supplied SD card or get it online in the download section of the FELIX BIOprinter*







6.2 Prepare printer and software parameters

Because there are a lot of specific cases for bioprinting, it is strongly recommended to follow below preparation steps for an optimal printing setup.

1. What material type is printed?
2. Extruder setup
    1. Cooling or heating?
    2. Single head/dual head
3. Print bed setup
    1. Standard heated build plate with flex plate.
    2. Bed add-on, place petri dish and printer.
    3. Calibrate first layer offset with bed coating parameter (Printer is standard calibrated for supplied petri dish)
4. UV module
   Determine exposure strategy

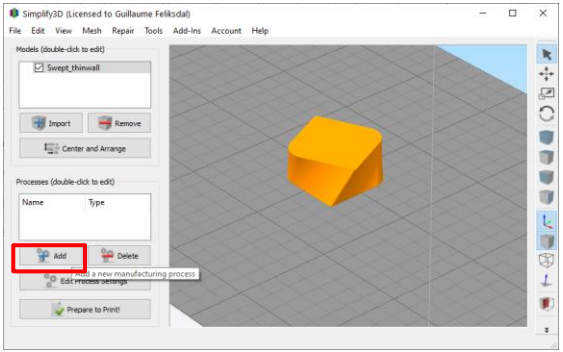
6.2.1 What material is printed?

To properly prepare the printer and the software parameters we need to know at least the following information.

	Left extruder
Material name	MatA
Nozzle diameter (mm)	0.51
Printing temperature (˚C)	4
Idle temperature (˚C)	10
Bed temperature (˚C)	8
Viscosity (Pa s)	64
Printed on carrier?	Petri dish
Cross linking required?	No

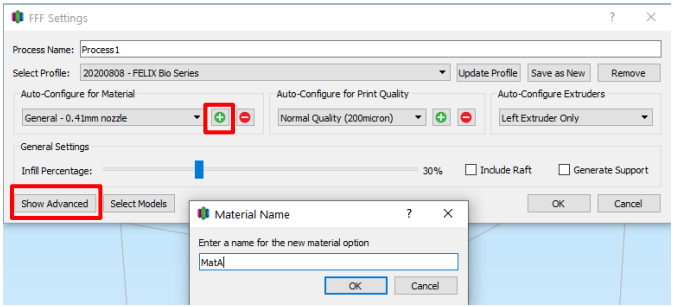
To setup any of the above parameters first a process needs to be created in simplify3D and a new material has to be defined.

Add a print process:

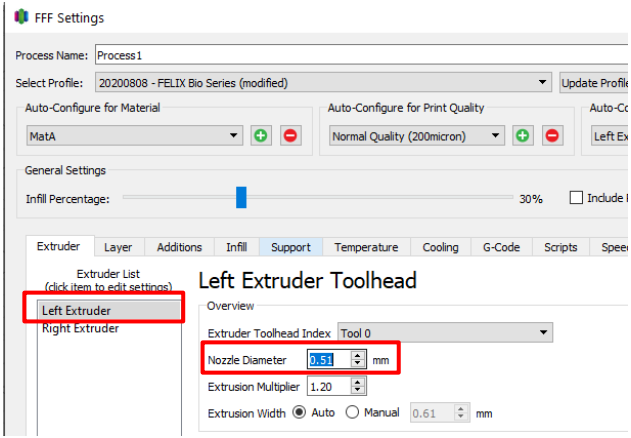


6 Single head print

Add a material and press show advanced

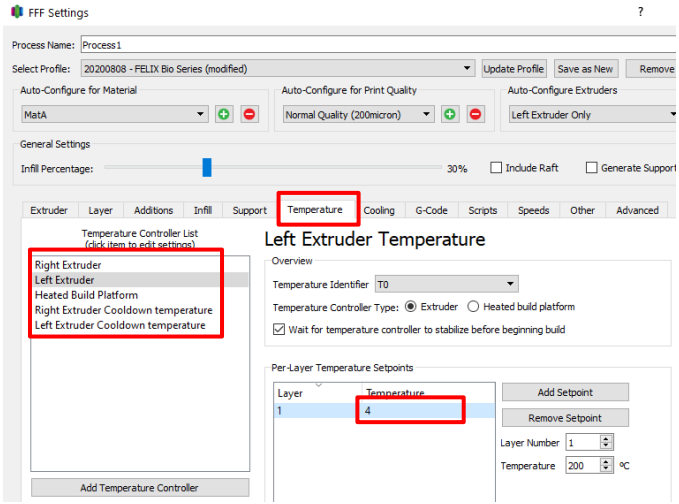


Goto the Extruder tab and select the Left Extruder and then fill on the right side the correct nozzle diameter (0.51)



Goto the temperature Tab and fill in the correct temperatures.

- Select Left Extruder and fill in for temperature 4 degC
- Select Heated Build Platform and fill in temperature 8 degC
- Select Left Extruder Cooldown temperature 10 degC



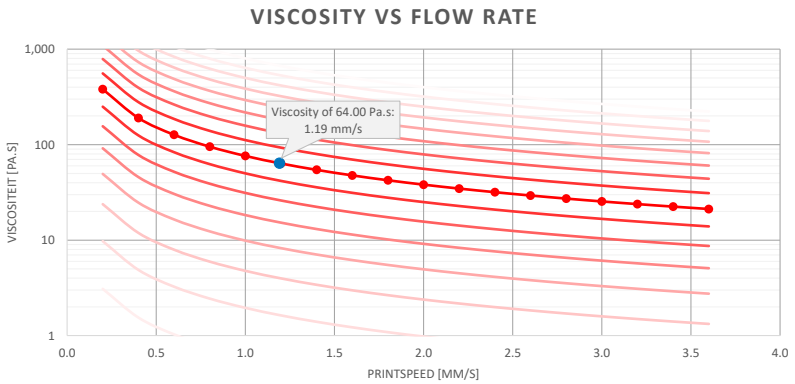
6 Single head print

When the viscosity is known, a maximum print speed can be determined which the extruder can handle. How a viscous material behaves inside the syringe with needle is modeled and put into an excel sheet. This conveniently gives insight in the max. recommended print-speeds versus viscosity of the material.

In this case:

Syringe		
Sectie 1 (needle)		
Diameter	0.51	mm
Length	12.7	mm
Sectie 2 (filled syringe)		
Diameter	12	mm
Lengte	35	mm
Setup		
Printspeed	2	mm/s
Layer height	0.2	mm
Line width	0.35	mm
Max extrusion force	80	N
Viscositeit	64	Pa.s

Recommended speeds		
Flow rate	**1.445**	mm³/s
Print Speed	**20.64**	mm/s
Print Speed	**1238.58**	mm/min



So according to the model, for MatA with its parameters the max recommended print speed is 20.64 mm/s

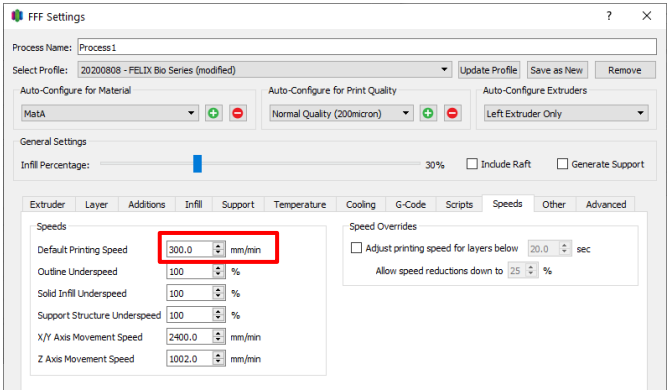
Find the excel sheet in the download section of our website.

Go to the speed tab and fill in the max speed.

According to the model the max recommended print-speed is 20mm/s, convert that to mm/min it comes to 20 x 60 = 1200mm/min

It is recommended to start slow and increase the speed when feeling comfortable so instead of the max speed of 20mm/s we choose a conservative print speed of 5mm/s (300mm/min)

6 Single head print



6.2.2 Extruder setup

To properly setup the extruders we need to take the follow steps.

1. **Does the extruder need cooling or heating?**
   The left extruder (MatA) needs to be set to cooling mode. Factory standard it is set to cooling mode.
2. **Single/Dual head printing?**
   Single head. Load the syringe according to the quick start manual

6.2.3 Print bed setup

For the print bed setup we have two options. Both will be explained below.

1. Standard heated bed with flex plate.
    1. Perform bed leveling (this only needs to be done occasionally)
    2. Place carrier on the print surface and fix it with for instance tape.
2. **Bed add-on module (since we need cooling this is the option required)**
    1. **Perform bed leveling** (this only needs to be done occasionally)
    2. **Ensure switch is set for cooling** on the side the of bed module. (Factory standard setting is cooling mode)
    3. **Place carrier** (petri dish in this case) onto bed addon module according quickstart manual.

Now calibrate the first layer offset with bed coating parameter (Printer is standard calibrated for supplied petri dish). See quick start manual

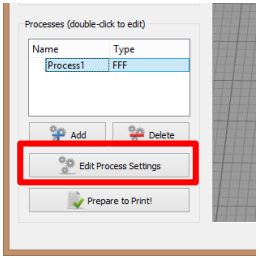
This calibration value determines the offset between nozzle and printed surface when printing the first layer. If too close, the material will be squashed onto the surface or worse the needle gets damaged. When too far away the material will be not attach properly to the surface.

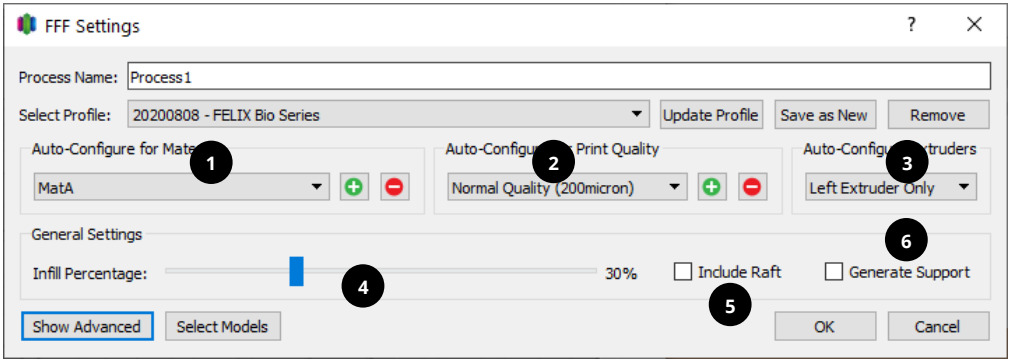
The printer is capable of probing (finding a reference z-position) directly onto the print surface where calibrating this value is not necessary. There is chosen not to do this but to choose an outside reference. The reason for this is, when a delicate carrier (plastic petri dish) is used, it can get damaged during probing.

6.3 Slice model

We have prepared the printer and the printing parameters properly. So now:

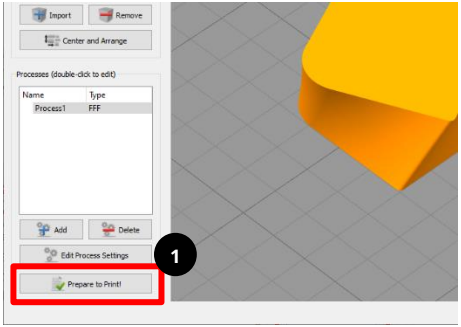
Press *edit process settings* in main window



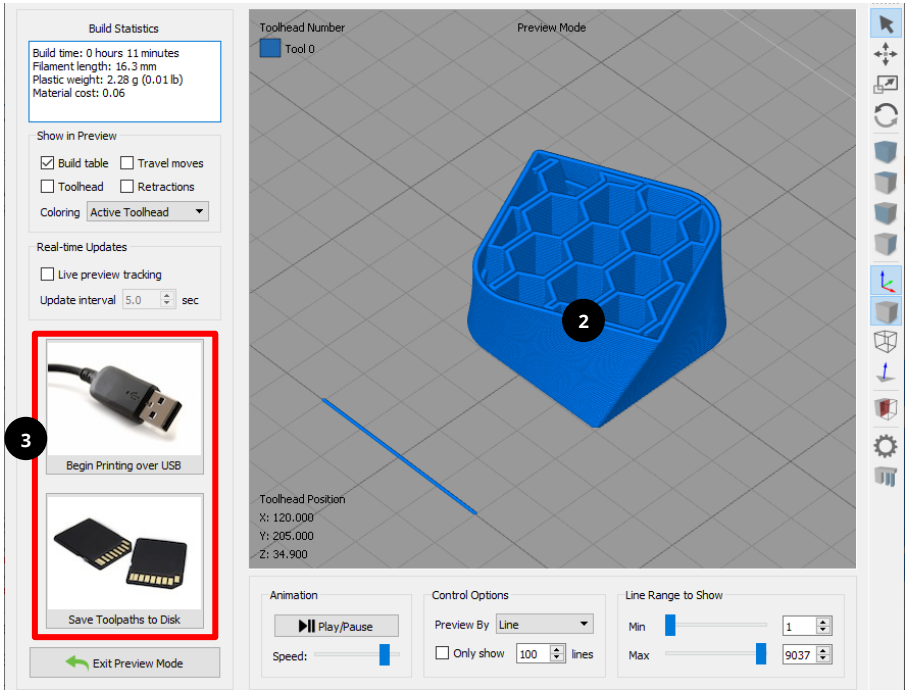


1. Select **material type**
2. Select **print quality**
3. Select **which extruder** to print with.
4. Select the amount of **infill**.
5. **Raft** is an additional thick layer between part and build surface, only recommended in cases where there is not a lot of contact surface or for special materials.
6. **Generate Support** is required to print objects successfully with large overhang angles or where otherwise printing in mid-air. Usually not suitable for viscous materials.
7. Press **OK** to continue

6.4 Print!



1. Press *Prepare to print!*
2. Inspect the sliced object in slice preview mode and verify if it is what you expect.
3. Print over USB cable if printer is connected or
   Save print file (*.gcode) to a microSD card or locally on your disc for later printing.



CONGRATULATIONS!

You've finished the quick start manual for single head printing!

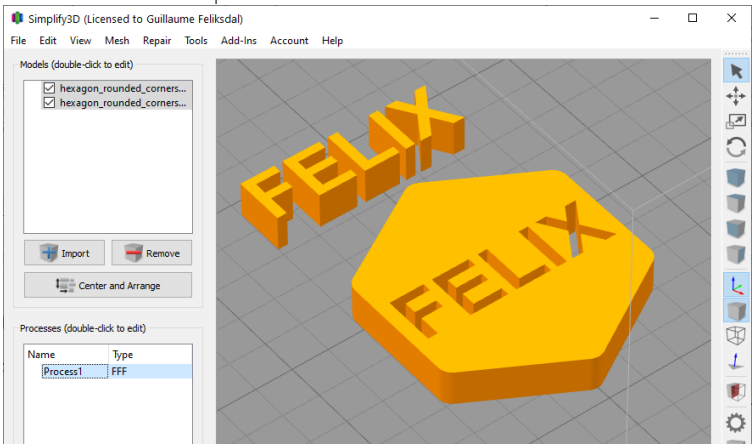
Now you can enjoy watching your idea to become reality.

7 Dual head print

7.1 Import STL files

This chapter describes how to print one object with two print-heads colors. It comes down to importing two objects and align them properly. Then assign each object with the correct extruder to print.

1. Import two files, we used the *hexagon_rounded_corners_logo_letters.STL* and *hexagon_rounded_corners_logo_body.STL*. Located on the supplied SD card with the printer or found online.



7.2 Prepare printer and software parameters

Because there are a lot of specific cases for bioprinting, you need to ensure the also follow the below preparation steps are done for an optimal printing setup.

1. What material type is printed?
2. Extruder setup
    1. Cooling or heating?
    2. Single head/dual head
3. Print bed setup
    1. Standard heated build plate with flexplate.
    2. Bed add-on, place petri dish and printer.
    3. Calibrate first layer offset with bed coating parameter (Printer is standard calibrated for supplied petri dish)
4. UV module
    Determine exposure strategy

7.2.1 What material is printed?

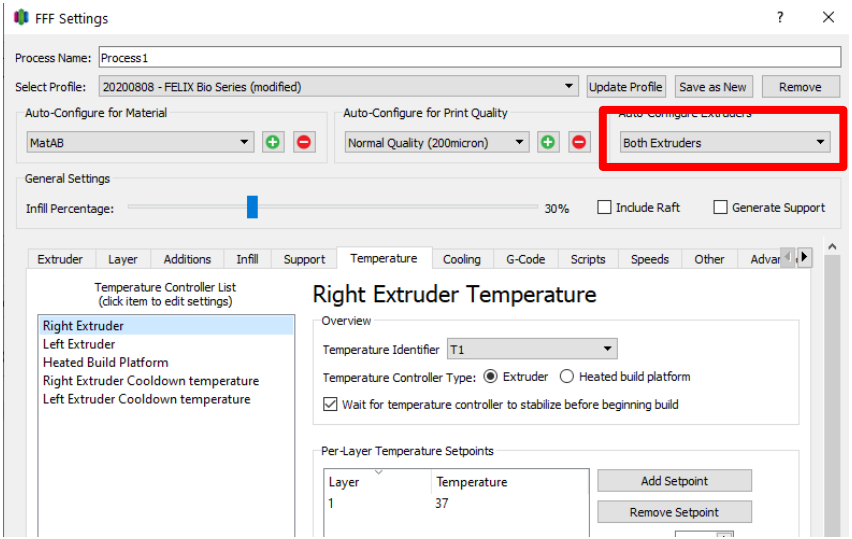
To properly prepare the printer and the software parameters we need to know the following information. We use the example materials MatA and MatB:

	Left extruder	Right extruder
Material name	MatA	MatB
Nozzle diameter (mm)	0.25	0.6
Printing temperature (˚C)	4	37
Idle temperature (˚C)	10	45
Bed temperature (˚C)	8	8
Viscosity (Pa s)	64	40
Printed on carrier?	Petri dish	
Cross linking required?	No	Yes

7 Dual head print

To setup any of the above parameters first a process needs to be created and a new material has to be defined.

- Add a print process to create a new material.
- Like in chapter 6.2.1 create a new material and call it **MatAB**
- Goto the extruder tab and Fill in for the **left** and **right** extruder the Nozzle diameters.
- Goto the temperature tab and ensure to select BOTH EXTRUDERS
    - Fill in all the temperatures.

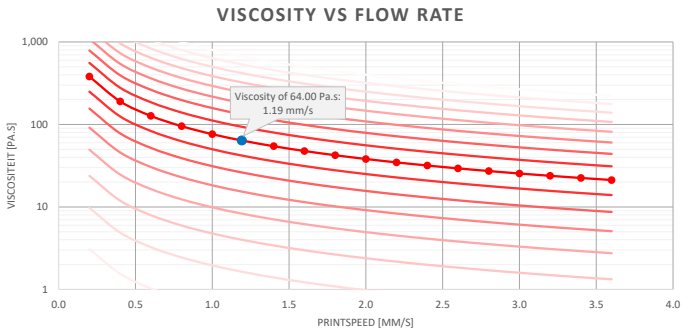


When the viscosity is known, a maximum print speed can be determined which the extruder can handle. We've worked out a model which can easily give insight in max recommended print-speeds versus viscosity of the material also taking into account important parameters.

For MatA the recommended print speed is 1.2mm/s

Syringe		
Sectie 1 (needle)		
Diameter	0.25	mm
Length	12.7	mm
Sectie 2 (filled syringe)		
Diameter	12	mm
Lengte	35	mm
Setup		
Printspeed	2	mm/s
Layer height	0.2	mm
Line width	0.35	mm
Max extrusion force	80	N
Viscositeit	64	Pa.s

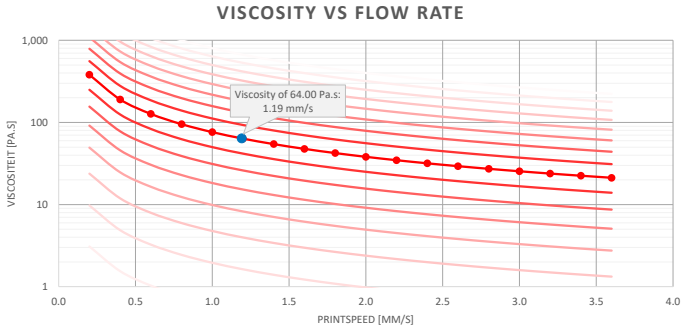
Recommended speeds		
Flow rate	**0.083**	mm³/s
Print Speed	**1.19**	mm/s
Print Speed	**71.52**	mm/min



For MatB, the max recommended print-speed is 63mm/s

Syringe		
Sectie 1 (needle)		
Diameter	0.6	mm
Length	12.7	mm
Sectie 2 (filled syringe)		
Diameter	12	mm
Lengte	35	mm
Setup		
Printspeed	2	mm/s
Layer height	0.2	mm
Line width	0.35	mm
Max extrusion force	80	N
Viscositeit	40	Pa.s

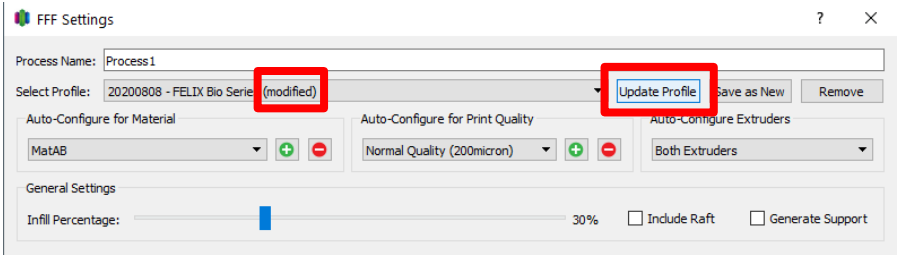
Recommended speeds		
Flow rate	**4.429**	mm³/s
Print Speed	**63.27**	mm/s
Print Speed	**3796.34**	mm/min



So according to the model for MatA the printspeed is max to 1.2mm/s and for MatB it is maximized to 63mm/s. It is recommended to use the minimal value of the two to create the print job.

Fill this into the speed tab of the print process.

You will see at the *Select Profile*: option that after the name the word ***modified*** is displayed. To store the changes please press **Update Profile**



7.2.2 Extruder setup

To properly setup the extruders we need to take the follow steps.

1. **Does the extruder need cooling or heating?**
   The left extruder (MatA) needs to be set to cooling mode. The right extruder (MatB) needs to be set to heating mode.
   Set the switch on the side of each extruder accordingly. Factory standard it is set to cooling mode.

2. Single/Dual head printing?

Dual head

1. Load both syringes according to quick start manual
2. Perform **Auto XYZ calibration**, will calibrate relative difference between both syringe tips.
3. In case above fails, a manual method can be performed. **(Calibrate Z then do manual XY calibration)**
4. Prepare slicing software for dual printing. We have two options.
    a) Use one of the print-heads for auto generated soluble support. (use a single head print process, but select both extruders and select the option support material)
    b) **Use both extruders in each layer. Where each head prints a different geometry. This option is covered in this chapter.**

7.2.3 Print bed setup

See chapter 6.2.3 for reference.

7.2.4 UV module

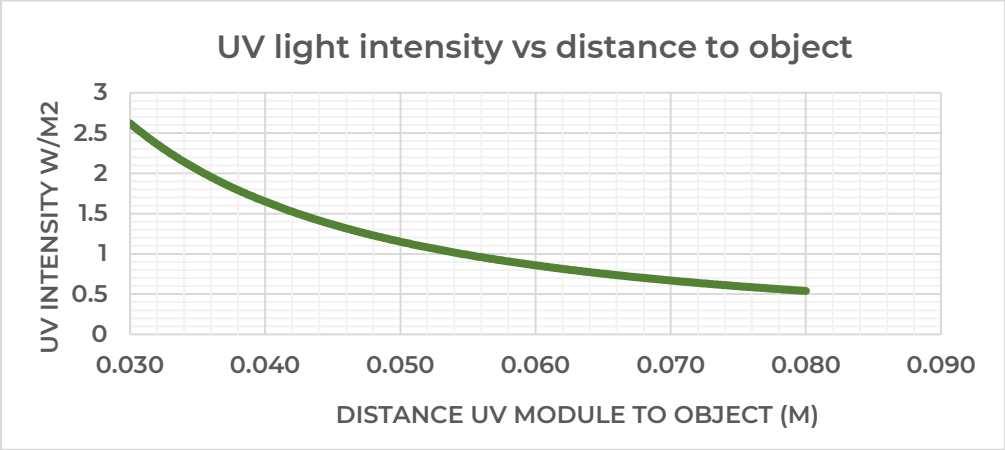
The UV module is normally used for curing printed material.

There are 2 strategies:

1. **Scanning exposure.** This means that while the light source is active, the object moves underneath it at a determined speed.
2. **Stationary exposure.** This means that when the light source is activated, the printer remains stationary, until the desired exposure is reached.

For above strategies the following parameters need to be set.

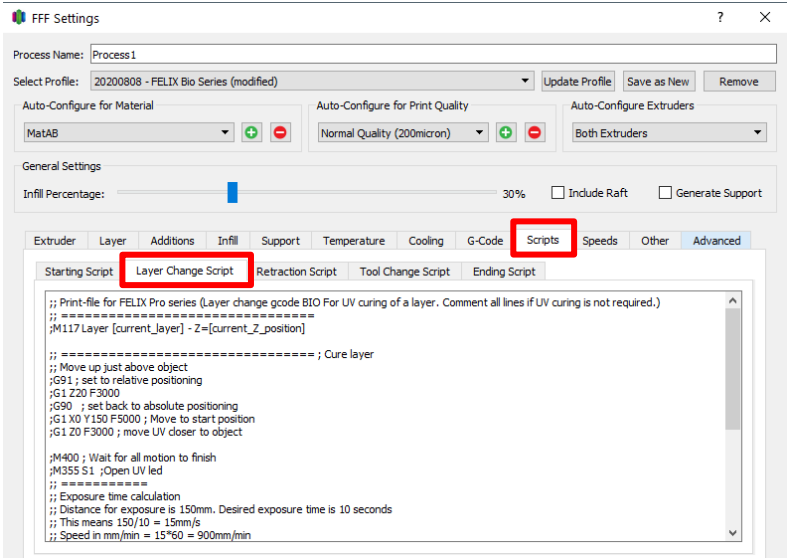
Distance between object and UV module. This determines the intensity.


UV light intensity vs distance to object
UV INTENSITY W/M2
3
2.5
2
1.5
1
0.5
0
0.030
0.040
0.050
0.060
0.070
0.080
0.090
DISTANCE UV MODULE TO OBJECT (M)

7 Dual head print

To enable the UV module, go to the process window and go to the *scripts* tab. Then go to the sub-tab *Layer Change Script*

Then remove the only the first ";" at the beginning of each line.



It should look like this:

7 Dual head print

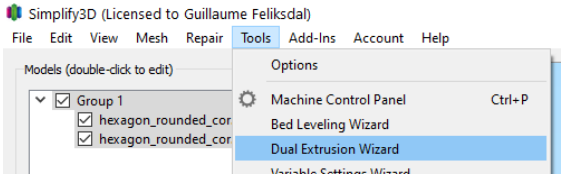
```
; Print-file for FELIX Pro series (Layer change gcode BIO For UV curing of a layer. Comment all lines if UV curing is not required.)
; ================================
M117 Layer [current_layer] - Z=[current_Z_position]

; ================================ ; Cure layer
; Move up just above object
G91 ; set to relative positioning
G1 Z20 F3000
G90 ; set back to absolute positioning
G1 X0 Y150 F5000 ; Move to start position
G1 Z0 F3000 ; move UV closer to object

M400 ; Wait for all motion to finish
M355 S1 ;Open UV led
; ===========
; Exposure time calculation
; Distance for exposure is 150mm. Desired exposure time is 10 second
; This means 150/10 = 15mm/s
; Speed in mm/min = 15*60 = 900mm/min
; ===========

G1 X0 Y0 F4500; Move to end curing posiiton
M400 ; Wait for all motion to finish
M355 S0 ;Close UV led

; ================================ ; Wipe nozzle
G1 Y40 F3000
G1 X5 F10000 ; fast wipe to the right
G1 Z20 F1000 ; Move back up again.
G1 X40 Y100 F5000 ; Go back to to position of printed object

; ======= end layer change code
```

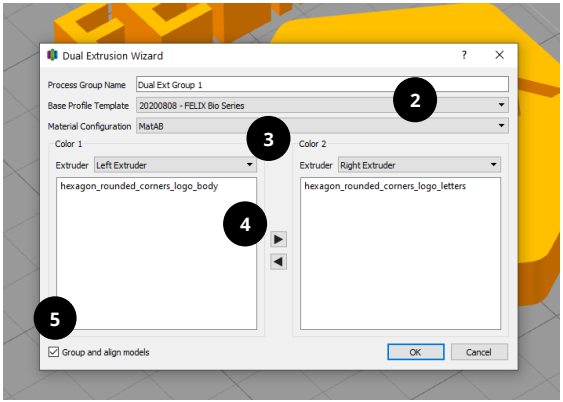
Only change this line to bring UV module closer or further away

Change this value to determine exposure time

7.3 Slice Model

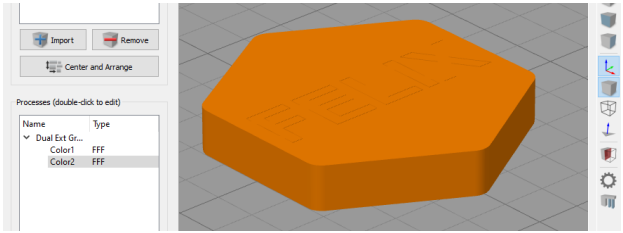
The model is prepared and the the printer and printing parameters as well now





1. Click **Tools**, then ***Dual Extrusion Wizard.***
2. Select a **base profile Template** corresponding to your printer.
3. Select **material**.
4. Configure which extruder has to print what object. Select the object name and then press the left or right arrow button to move it to the different extruder.
5. **IMPORTANT:** ***Select group and align Models checkbox***. This is only useful if you want two objects axis to be aligned with respect to each other in the way they are created. (if you forget this, you can undo the grouping of parts by Edit -> ungroup Selection -> Click on Center and Arrange)
6. Press **OK**

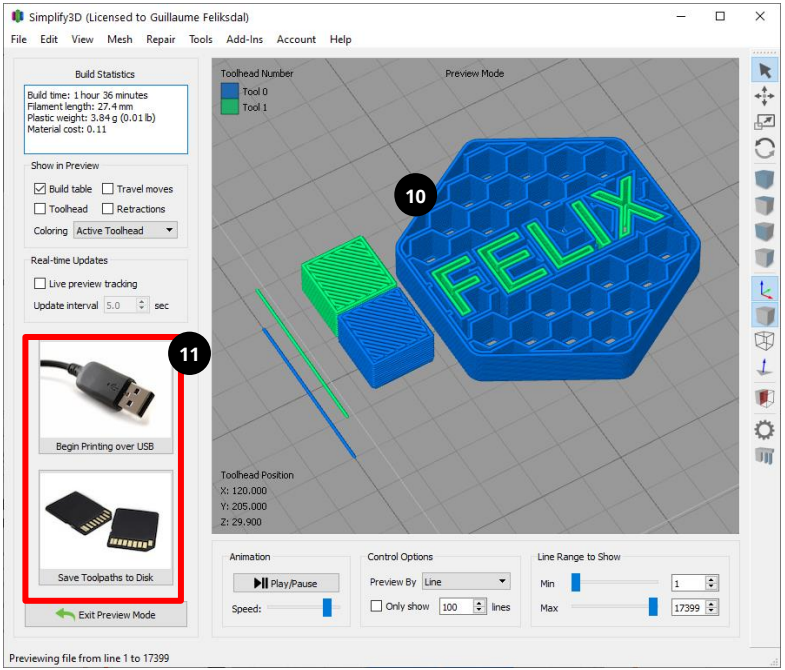
7. **A grouped process shows up** in the processes overview. And the objects will be grouped and aligned with respect to each other.
8. In some cases the object is standing upright. this is not ideal for 3D printing.
    a. Press **CTRL+L** to re-orientate the grouped object (or press *Edit -> place surface on bed*)
    Point and click on a surface which we lay flat on the build surface, you will see a highlighted area.



7.4 Print!

9. A pop-up comes up which processes need to be sliced.
   Choose *Select All* and *OK*
10. The parts get sliced and the preview mode is shown.
    Please note the different colors and the wipe tower. In some occasions two wipe towers are created. NOTE: The wipe towers should not overlap the two thin lines. If so, please move the object a few mm's away.

11. After inspection, print over USB cable if printer is connected or Save print file (*.gcode) to a microSD card or locally on your disc for later printing.



CONGRATULATIONS!

You've finished the quick start manual for dual head printing!

Now you can enjoy watching your idea to become reality.

While printing your first print, please take a moment to continue to read. We'll explain the following in a quick overview.

- ✓ 3D printing tips and tricks
- ✓ 3D design best practice.
- ✓ Dual head printing with soluble support
- ✓ Dual head multi-color printing

8 3D printing tips and tricks

5 Steps to a successful print

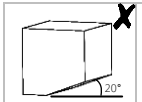
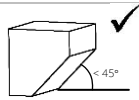
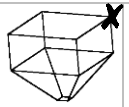
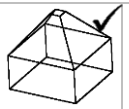
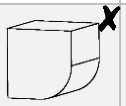
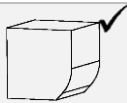
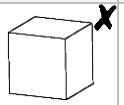
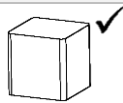
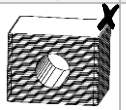
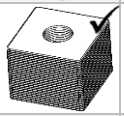
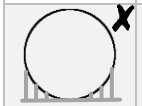
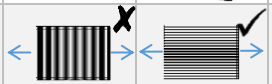
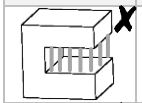
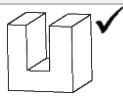
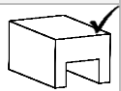
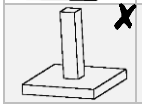
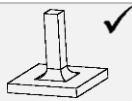
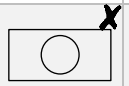
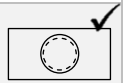
- ✓ **Clean print surfaces.**
  Ensure print-surface is clean. Use recommended detergent suitable for the print-process.
- ✓ **Clean exterior of nozzle.**
  Remove remainders of printmaterial with tweezer or cloth before starting each print.
- ✓ **Use supplied accessories.**
  The use of supplied accessories ensures optimal machine performance.
- ✓ **Wait for first layer to finish.**
  The success of a print is mostly depending on the first layer. Make sure first layer is finished properly Before leaving your printer unattended.
- ✓ **Let print heads settle down at correct temperature.**
  Before starting a print it might be better to let the printhead and bed temperature settle on their correct temperatures before printing for a few minutes.

General 3D printing tips

- ✓ **Make sure build plate is leveled.**
  Especially for larger print object a properly leveled bed is essential for good first layer adhesion to the build plate.
- ✓ **Use latest slicing profiles and firmware**
  Make sure to check for updates regularly. Using the latest profiles and settings improves performance.
- ✓ **Use a wipe tower when printing dual head prints.**
  Using a wipe tower results in 'clean' prints. The wiper tower is standard enabled but be sure to always double check before printing
- ✓ **Use low cost Vaseline grease as print material.**
  After searching for easy to use off the shelve material for general purpose testing, we found that vaseline grease is an excellent material to mechanically simulate a wide range of biomaterials. It is low cost and easy to print. Before you use the expensive bio-inks, please practice with this material first. You can buy it at most pharmacy/grocery stores at low cost.

9 3D design best practice

Besides choosing proper slice settings, life can be made easier if you print 3D objects suitable for 3D printing. The next table shows an overview of best practice when creating your own design.

		Create overhangs greater than 45° with respect to the print surface.				Reorient model, for largest adhesive surface to print surface. Reduce chance to tipping over objects.
		Avoid rounded corners touching the build plate, create a chamfer to have at least an angle of 45°.				Avoid small surface to volume ratio. Use surface area larger than 5x5mm Ratio (length or width)/height < 1:5, to prevent tipping over during printing.
		Make small fillets on sharp corners to improve print results.				Orient model for maximum strength. Holes are stronger when printed in plane.
		Divide objects in parts, to prevent support material and reduce print time.				Tension in plane of layer is much stronger than tension in direction of layer.
		Re-orient part to prevent support material to save material, print and post-processing time.				Bridges can be unsupported when bridge is larger than 10 mm, support material is recommended
		Apply fillets to reduce stress and increase strength on small pillars and features.				For holes with a diameter up to 10 mm correct size in design by an increase of about 2 to 4 %.

Review our products

Help us to improve our product and services

www.felixprinters.com/survey

Manuals & tutorials

www.felixprinters.com/support
www.simplify3D.com

FELIX community

www.felixprinters.com/forum

Quick Start Guide **BIOprinting with Simplify3D**

V1-2020

FELIX printers

www.felixprinters.com/support
support@felixprinters.com

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands