



Quick Start Guide

Simplify3D



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

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands

1 Introduction

1 Introduction

Simplify3D is software to control your printer and prepare your 3D files for printing. Please read this quick start manual and benefit from:

- Best prints in the least amount of time.
- Essential tips and tricks for 3D printing.

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

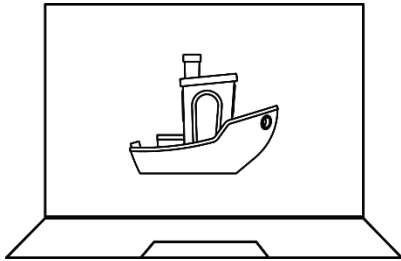
Address: Zeemanlaan 15, 3401MV IJsselstein, The Netherlands

2 Contents

1	Introduction	1
3	3D printing process overview	2
4	Install Simplify3D	3
5	Import slice profiles	4
6	Import STL file	5
7	Slice model.....	6
8	Print	7
9	3D printing tips and tricks	9
10	3D design best practice	10
11	Dual head printing with soluble support	11
12	Dual head multi-color printing	13

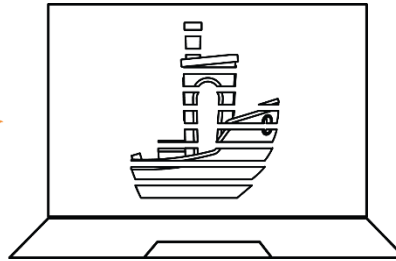
3 3D printing process overview

The process from design to printed object at a glance



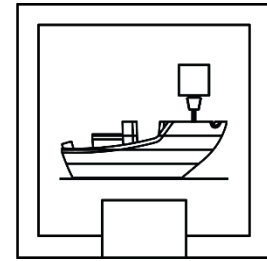
Import STL file

Simplify3D software generally accepts STL files.



Slice it in thin layers

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



Print

The printer will extrude filament on the calculated path, layer by layer.

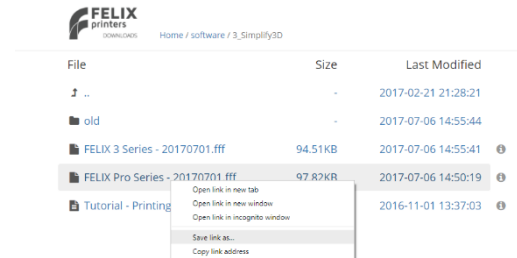
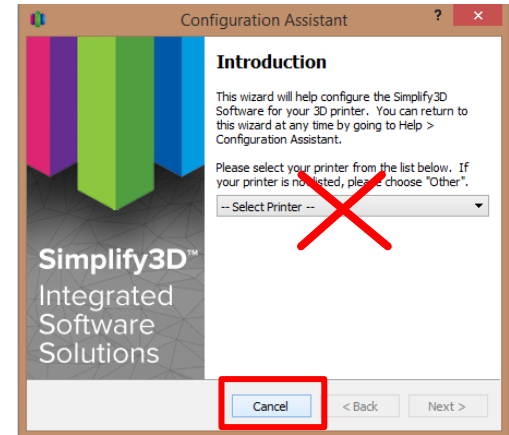
4 Install Simplify3D

4 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.



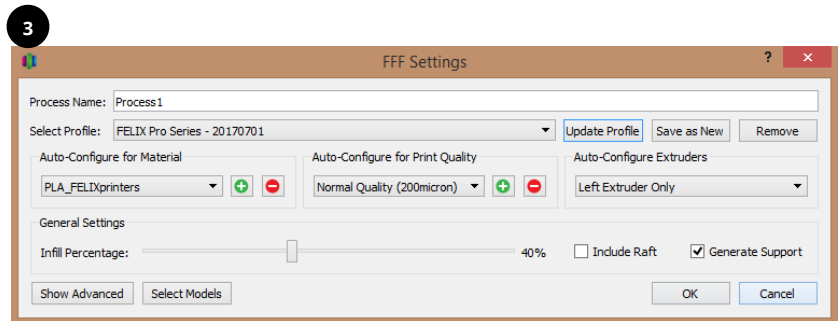
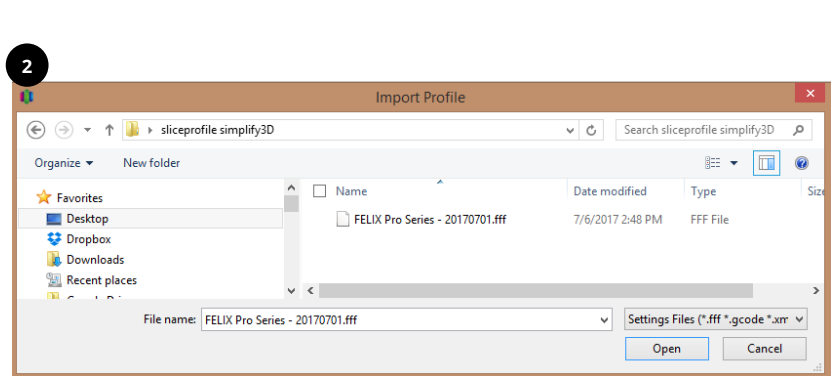
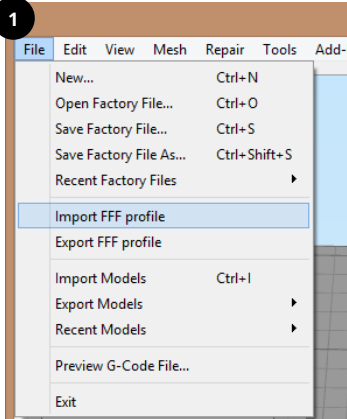
5 Import slice profiles

5 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:

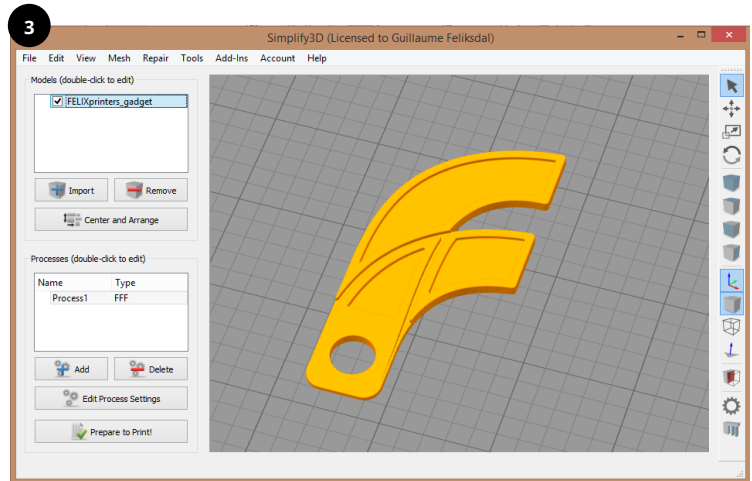
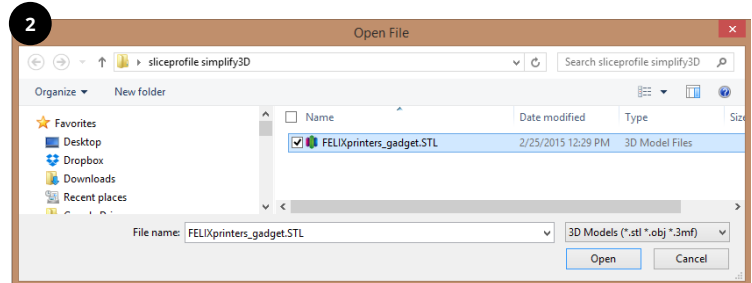
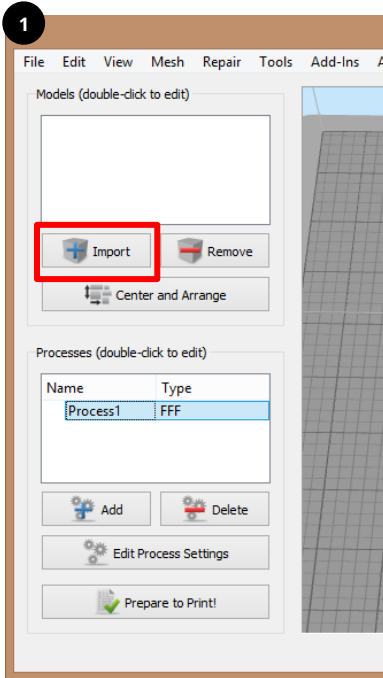
Make sure to use correct slice profiles. It can make or break your 3D print/printer.



6 Import STL file

6 Import STL file

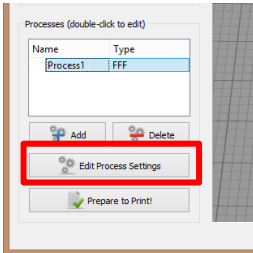
- ✓ Press the import button to import an STL file.
- ✓ In this example, we take the *FELIXprinters_gadget.STL*, locate this file on the supplied SD card or get it online.



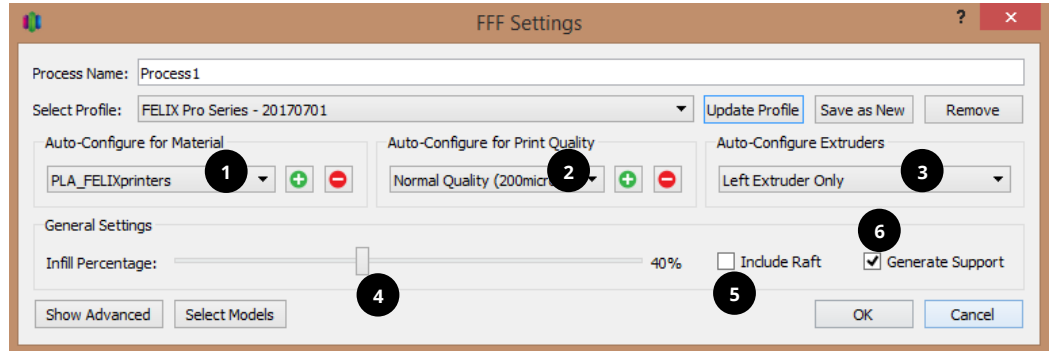
7 Slice model

7 Slice model

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.
7. Press **OK** to continue



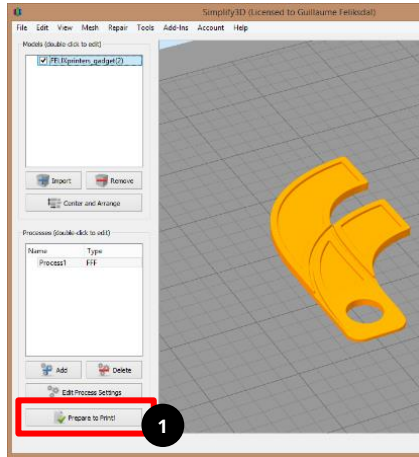
TIP:

Choose infill less than 70% To prevent extruder clogs. Going higher requires very good filament flow calibration. Strength difference between 70% and 100% is negligible

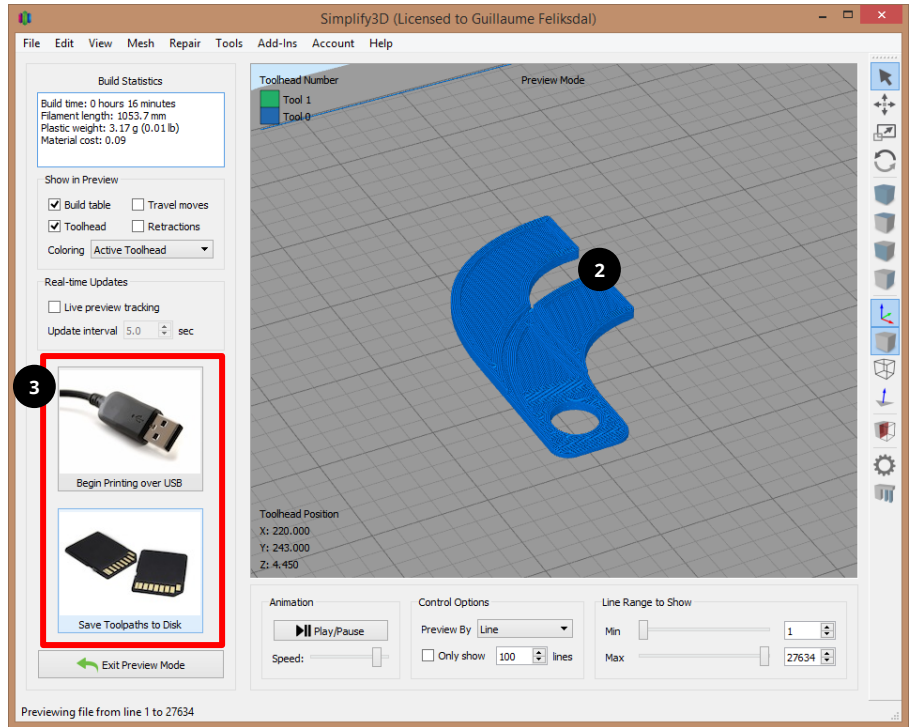
Start with lower quality and single head printing to get familiar with the printer and printing process.

8 Print

8 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 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.

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

9 3D printing tips and tricks

5 Steps to a successful print

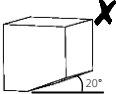
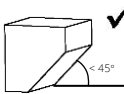
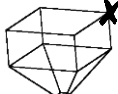
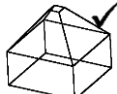











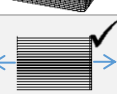
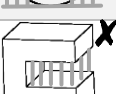
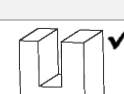
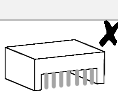
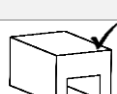
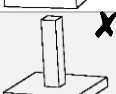
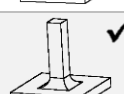
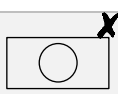
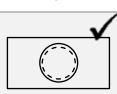
- ✓ **Clean print surface.**
Use recommended detergent like methylated spirit, acetone or alcohol.
- ✓ **Clean exterior of nozzle.**
Remove remainders of filament with tweezer or cloth before starting each print.
- ✓ **Use filament accessories.**
The use of supplied accessories ensures proper guiding, cleaning of filament and 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 cool down.**
Below 100 °C before turning off the printer, to prevent clogging of print heads

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.
- ✓ **Never leave print heads at elevated temperatures.**
Leaving the heaters on for elongated periods of time without extruding could cause clogged print heads.
- ✓ **Flush out old material first.**
Extrude approx. 200 mm of new material to ensure old material is fully removed to prevent clogs when switching to different filament.
- ✓ **Use SD card for +5hr long prints.**
Standalone printing (instead of via USB cable) improves reliability. Unexpected computer updates can ruin your print.
- ✓ **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

10 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.

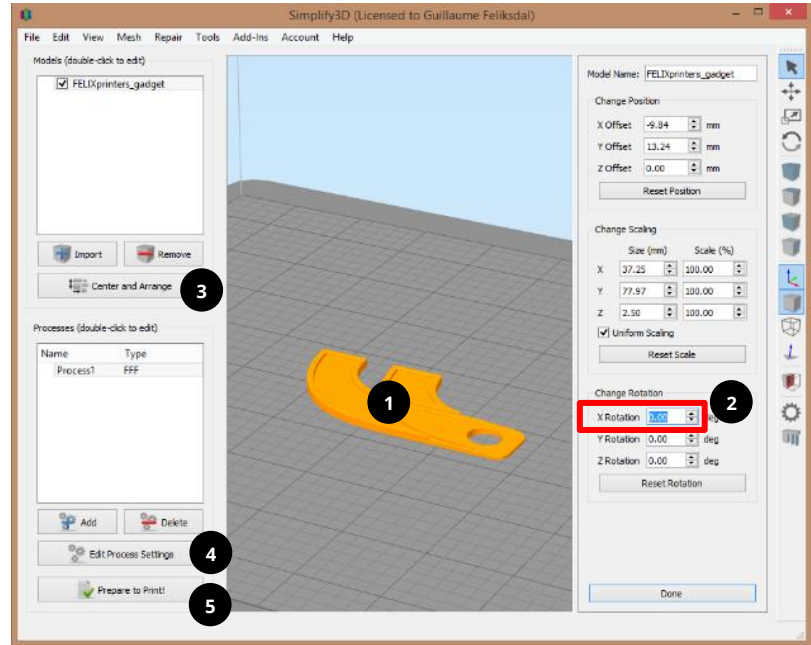
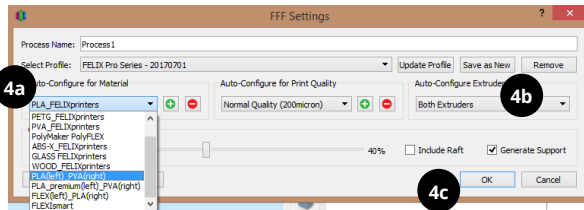
		Create overhangs greater than 45° with respect to the print surface.			Reorient model, for largest adhesive surface to print surface. Reduce chance to tipping over objects.
		Avoid rounded corners touching the build plate, create a chamfer to have at least an angle of 45°.			Avoid small surface to volume 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 %.

11 Dual head printing with soluble support

11 Dual head printing with soluble support

For this example, we use the same file as in previous chapter.

1. Double click on the imported STL object,
2. Rotate the file around it's x-axis by rotating it 20degC
3. Press *Center and arrange* button
4. Click on *edit process settings*.
 - a. Select *PLA(left) PVA(right)*
 - b. Both Extruders
 - c. Press OK
5. Press *Prepare to Print!* and inspect results.



11 Dual head printing with soluble support

11.1 Inspect the results.

To be sure the print will be successful, there must be the following visible in the print preview area:

- ✓ **Visible wipe tower.** This wipes strings from the non-used extruder ensuring clean prints
- ✓ **Support material should have a different color from the part.** This indicates that both extruders are used for the print. Make sure the coloring of the preview is set to *Active Tool head*.
- ✓ 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.

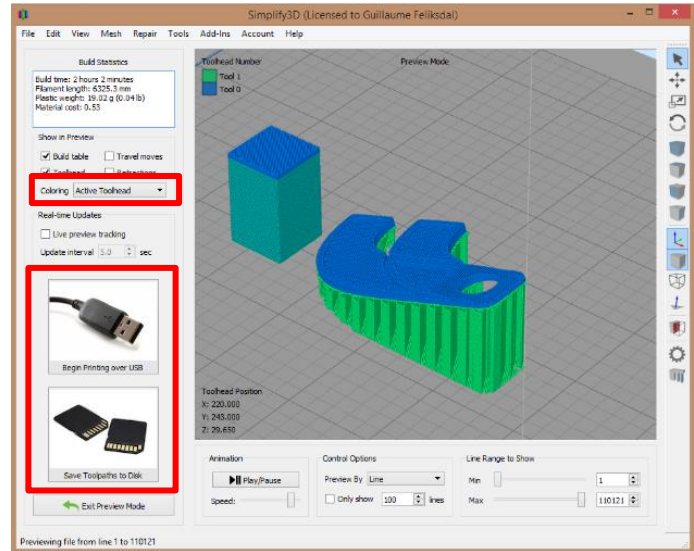
TIP:

Make sure to use fresh PVA. PVA has a short shelf life because besides dissolving in water it also absorbs water from the air.

It is best to cut off a small piece of PVA and put the spool back into its reseal able bag with desiccants for best performance.

Never leave PVA at elevated temperatures without printing. It will crystallize and clog your nozzle.

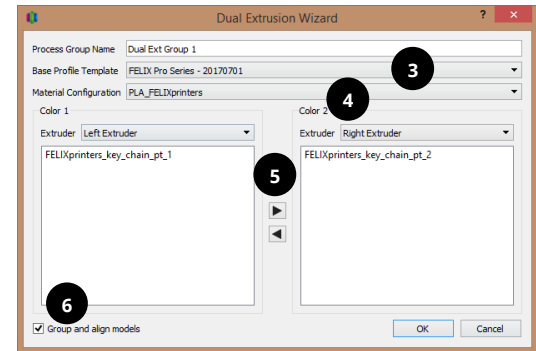
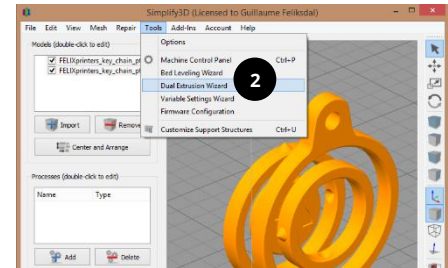
It is recommended to consider a redesign of your part to enable single head printing. This will save you printing time and costs.



12 Dual head multi-color printing

This chapter describes how to print one object with two 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 *FELIXprinters_key_chain_pt1.STL* and *pt2*. Located on the supplied SD card with the printer.
2. Click **Tools**, then *Dual Extrusion Wizard*.
3. Select a **base profile Template** corresponding to your printer.
4. Select **material**.
5. Configure which extruder to print what object. Select the object name and then press the left or right arrow button to move it to the different extruder.
6. **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)
7. Press **OK**
8. **A grouped process shows up** in the processes overview. And the objects will be grouped and aligned with respect to each other.
9. In this case 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.
10. Press *Prepare to Print!*



12 Dual head multi-color printing

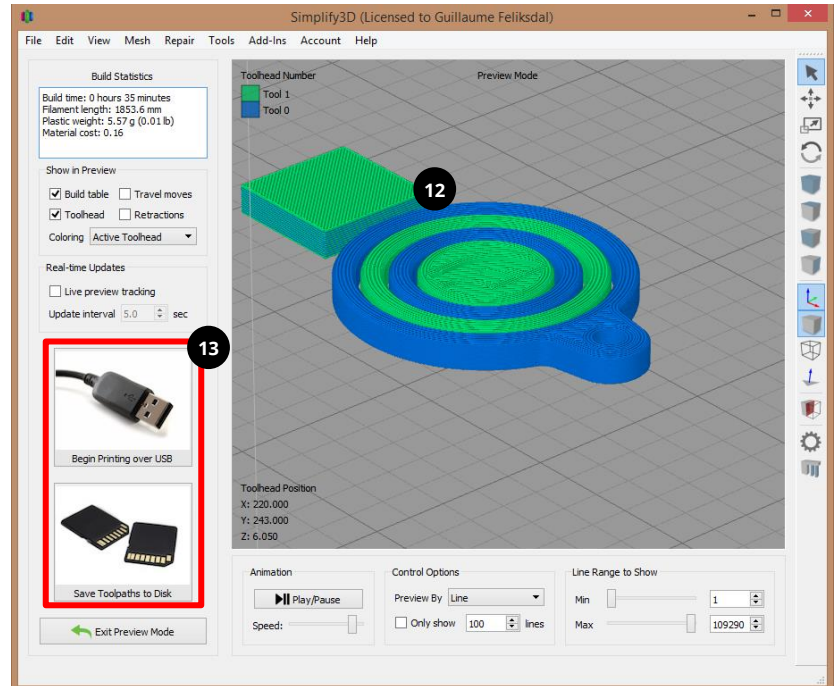
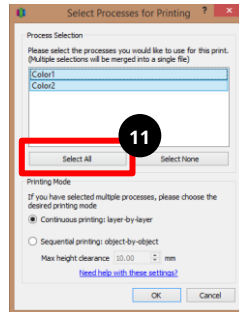
11. A pop-up comes up which processes need to be sliced.

Choose *Select All* and *OK*

12. 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.

13. 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.





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 Simplify3D

V1-2017



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

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands

