



Quick Start Guide

Repetier-host for FELIXprinters

(PC only)



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

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands

1 Introduction

1 Introduction

Repetier-host for FELIXprinters 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 Repetier-host or 3D printing in general, go to the following resources:

www.FELIXprinters.com

www.repetier.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

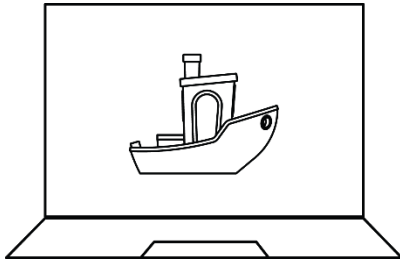
Address: Zeemanlaan 15, 3401MV IJsselstein, The Netherlands

2 Contents

1	Introduction	1
3	3D printing process overview	2
4	Install Repetier-host for FELIXprinters	3
5	Run Repetier-host for FELIXprinters	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

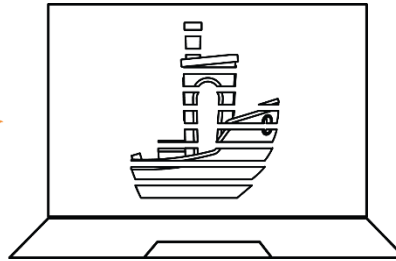
3 3D printing process overview

The process from design to printed object at a glance



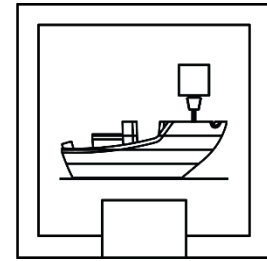
Import STL file

Repetier 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 Repetier-host for FELIXprinters

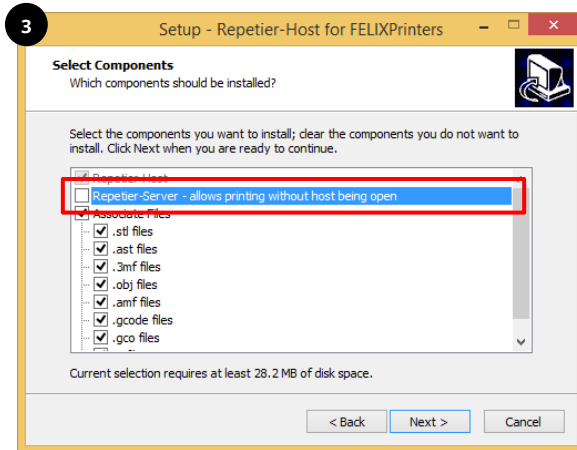
4 Install Repetier-host for FELIXprinters

1. Get the installation files from:
 - a. The supplied SD card. Location on SD card (Software -> Repetier-Host)
 - b. Online. <http://www.felixprinters.com/downloads>. Then software -> Repetier-Host -> Installation
2. Follow instructions during installation.
3. At some point during the installation it asks to install repetier-Server. This is not recommended to start with, so cancel this at this point. If you still want to install it, please make sure there is an internet connection it will try to download the installation files.

What is repetier-server?

Repetier-Server enables you to control your printer via a web browser and enables access to your printer from within the local network.

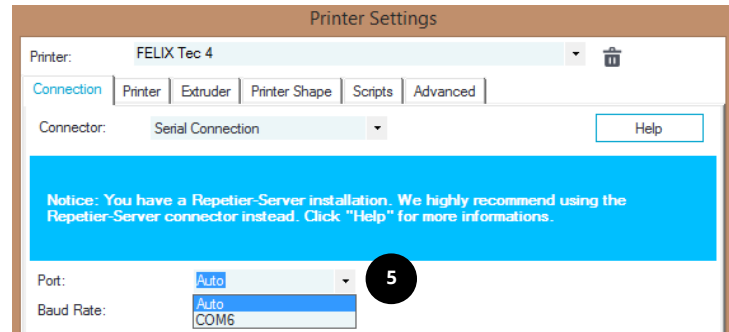
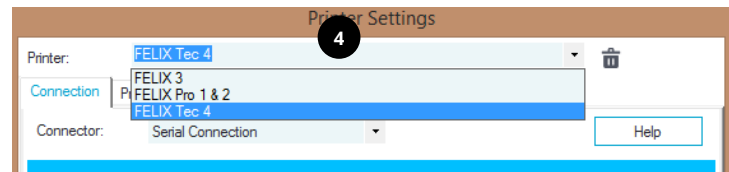
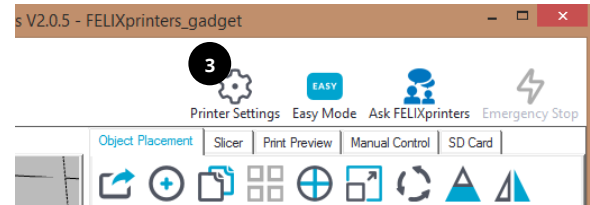
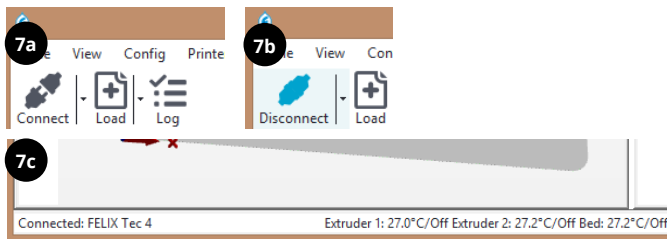
4. Finally, it will ask to install drivers. Select the drivers for your printer type.



5 Run Repetier-host for FELIXprinters

5 Run Repetier-host for FELIXprinters

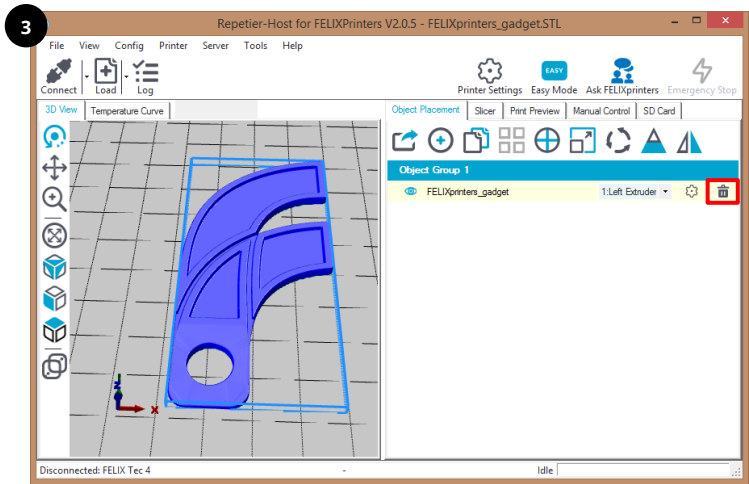
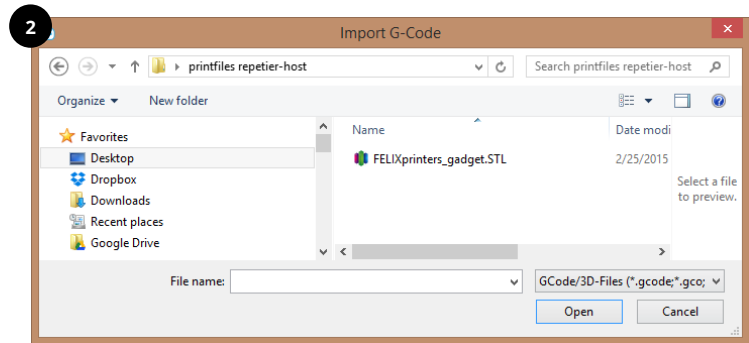
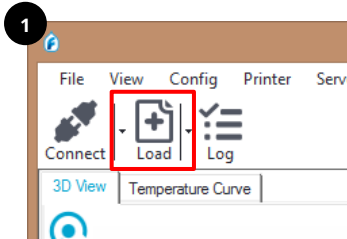
1. Start Repetier-host program.
2. Plug in the USB cable in the printer and Computer and turn on the printer.
3. Go to **Printer Settings**
4. Select your printer type
5. Check in the Port settings, if you see anything other than Auto. If yes, that is most likely the port for your FELIXprinter. If no, the printer is either not plugged in or not recognized by your computer. If this is the case, please make sure the printer is turned on and make sure the driver installation is done properly during installation of repetier-host.
6. Press OK to close the window
7. In the main window,
 - a. press the **connect button**.
 - b. The connection icon should change
 - c. Another indication to see if the printer is properly connected is the bottom of the window. It should display that the printer is connected and also you should see the temperatures variate a little bit.



6 Import STL file

6 Import STL file

1. Press the Load button to import an STL file.
2. Search the file to print In this example we take the *FELIXprinters_gadget.STL*, locate this file on the supplied SD card or get it online in our download section. www.felixprinters.com/downloads.
3. The 3D object will be shown in the 3D view window.
Tip: to remove it press the recycle bin button



7 Slice model

7 Slice model

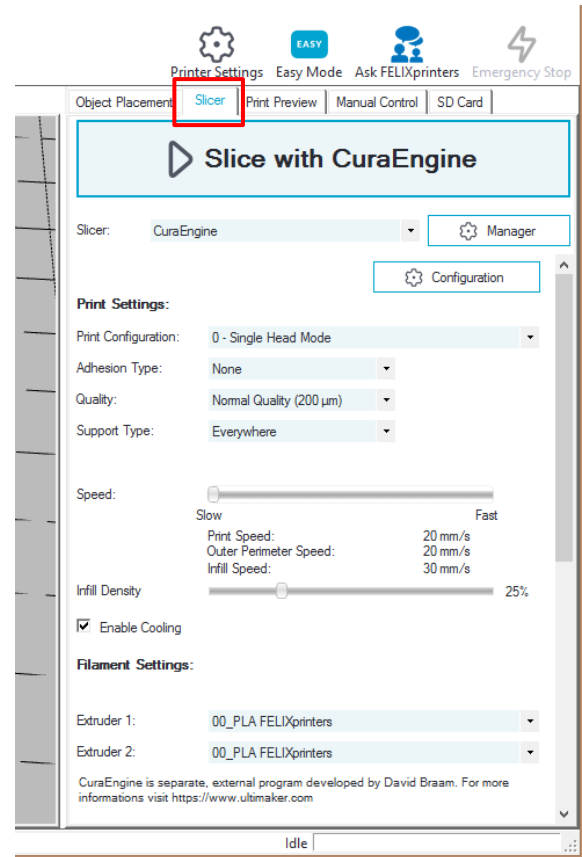
Go to the *slicer tab settings* in main window

1. Select for Print configuration: *0 – Single Head Mode*
Other print modes
1 – Soluble support mode. This is applicable to dual head printers where you use soluble support material in the 2nd head
2 – Dual Color mode. Where two of the same materials are used in a single print.
3 – Vase mode. Where printing an object in a corkscrew way.
2. Adhesion Type: *None*. This is useful when printing with an object which has a large volume, but small contact surface. Use this to prevent tipping over of the object during printing.
3. Select Quality: *Normal Quality (200µm)*. This is thickness of each layer.
4. Select Support Type: *Everywhere*.
Support material is required to print objects successfully with large overhang angles or where otherwise printing in mid-air. Touching bed, means that only the exterior of the object is supported.
5. Slide speed to *Slow*. It is recommended to start printing at slow speeds.
6. Slide infill Density to *25%*. It is not recommended to choose infill rates above 80%. This increases chances of extruder clogging if filament is not properly calibrated in software.
7. Leave *enable cooling ON*.
8. Select the **material to print**. As beginner, we recommend to use PLA. If you have a single extruder, ignore settings set in Extruder 2.
9. Press *Slice with CuraEngine*

TIP:

Choose infill less than 80% To prevent extruder clogs. Going higher requires very good filament flow calibration. Strength difference between 80% and 100% is negligible.

Start with lower quality and single head printing to get familiar with the printer and printing process before diving into dual head printing.

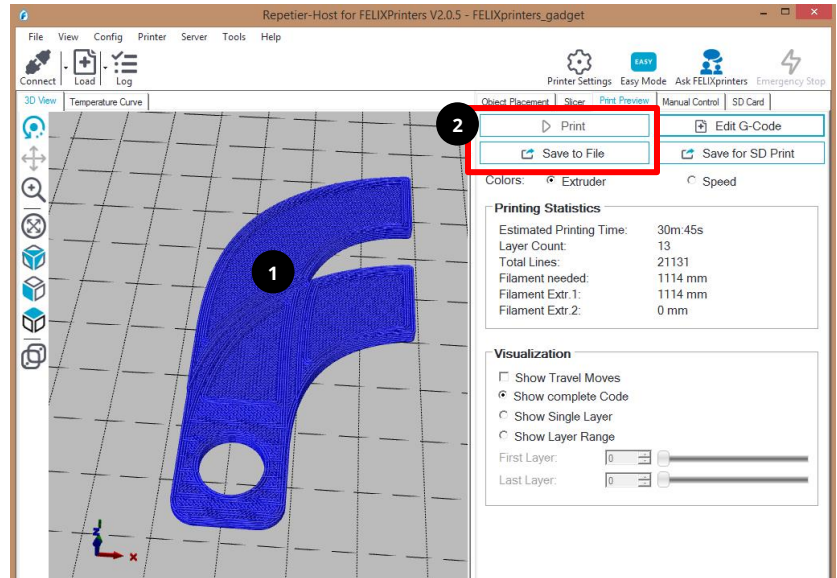
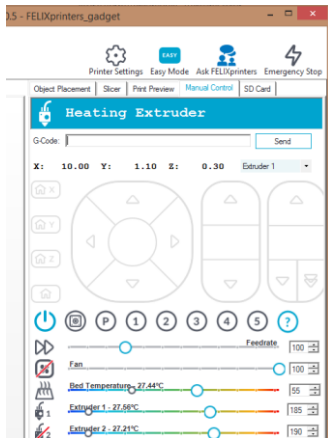


8 Print

8 Print

After slicing you will see a preview of how the object will be printed.

1. Inspect the sliced object in slice preview mode and verify if it is what you expect.
2. Print the object via
 - a) USB connection if printer is connected or
 - b) Save print file (*.gcode) to a microSD card or locally on your disc for later printing.
3. The interface will jump to the Manual Control Tab, where online parameters can optionally be adjusted during printing.



TIP:

For prints longer than 6 hours we recommend to print via SD card. This reduces chances of failure. Computers are able to restart themselves or turn off or go into hibernation, start an unwanted auto update, reset USB connection if plugging in new devices. This causes the print to stop resulting in a failed print and wasted filament.

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

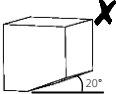
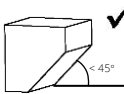
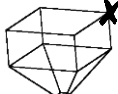
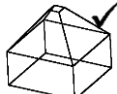

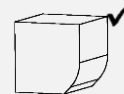


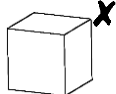
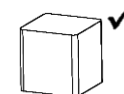
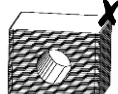
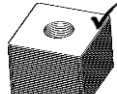




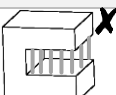
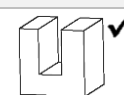
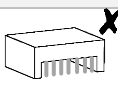
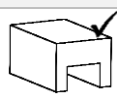
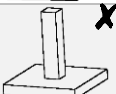
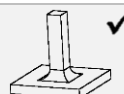
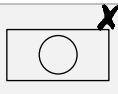
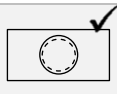
- ✓ **Clean print surface.**
Use recommended detergent like methylated spirit, acetone or alcohol.
- ✓ **Clean exterior of nozzle.**
Remove remainders of filament with tweezer or cloth before starting each print.
- ✓ **Use filament accessories.**
The use of supplied accessories ensures proper guiding, 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.

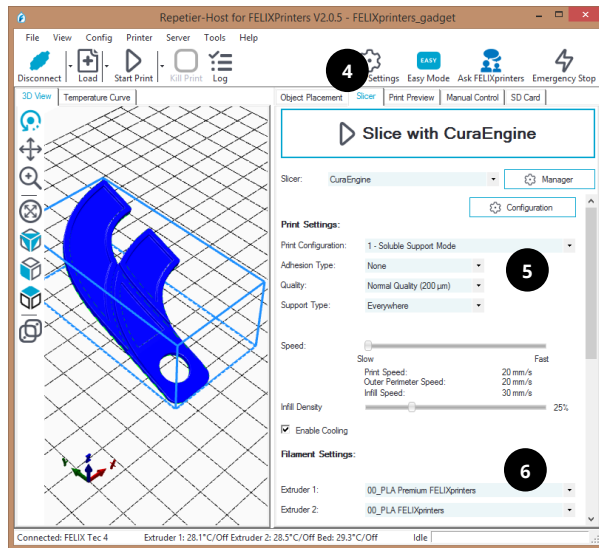
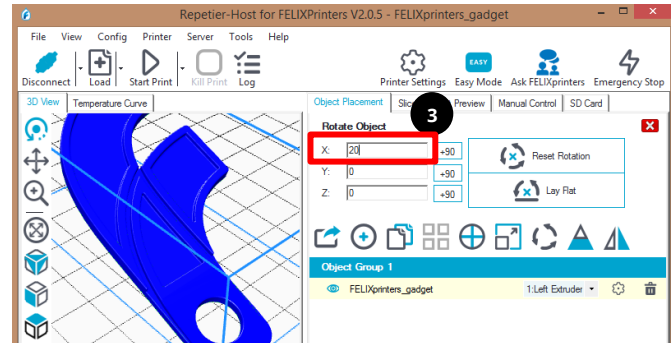
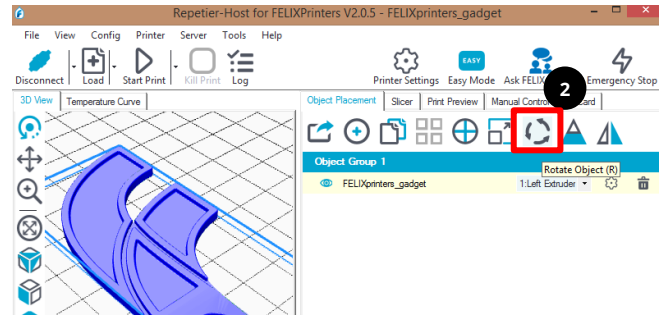
		Create overhangs greater than 45° with respect to the print surface.			Reorient model, for largest adhesive surface to print surface. Reduce chance to tipping over objects.
		Avoid rounded corners touching the build plate, create a chamfer to have at least an angle of 45°.			Avoid small surface to volume 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. Click on the imported STL object,
2. Press the rotate icon.
3. Fill in 20 for X to rotate the part.
4. Go to the slicer tab.
5. For print configuration Select: 1 – Soluble Support Mode.
6. Select correct material.
7. Press Slice with CuraEngine



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:

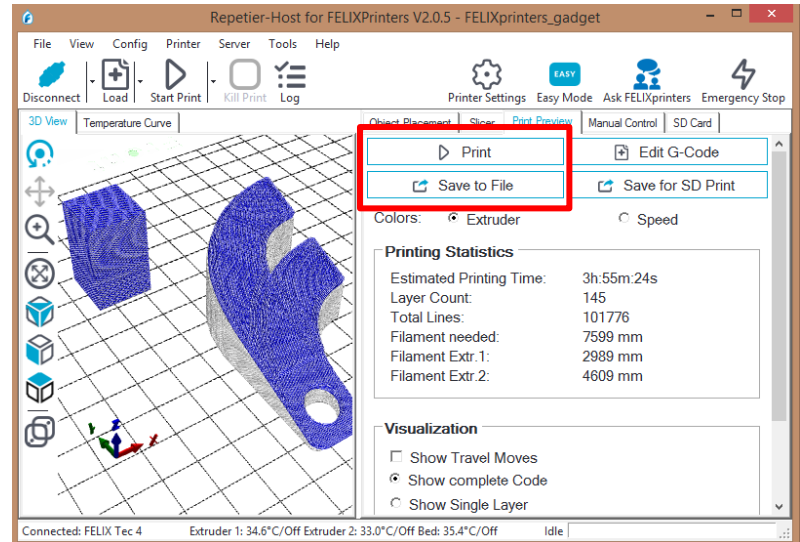
- ✓ **Visible wipe tower.** This wipes strings from the non-used extruder ensuring clean prints
- ✓ **Support material should have a different color from the part.** This indicates that both extruders are used for the print.
- ✓ After inspection, print over USB cable if printer is connected or
Save print 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.

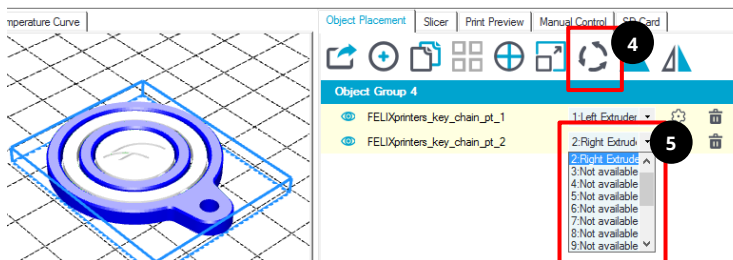
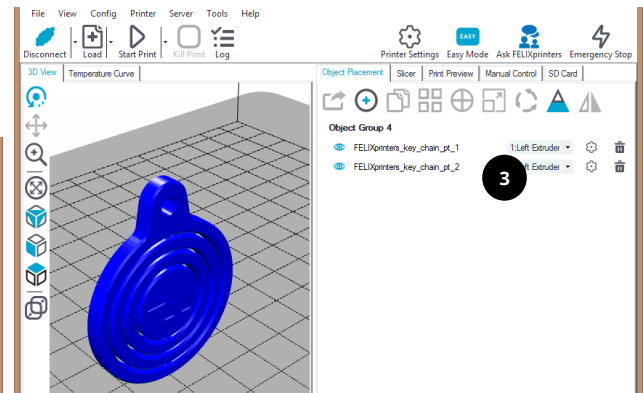
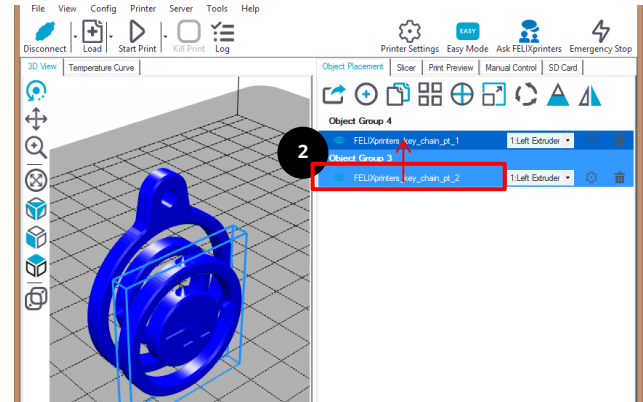


12 Dual head multi-color printing

12 Dual head multi-color printing

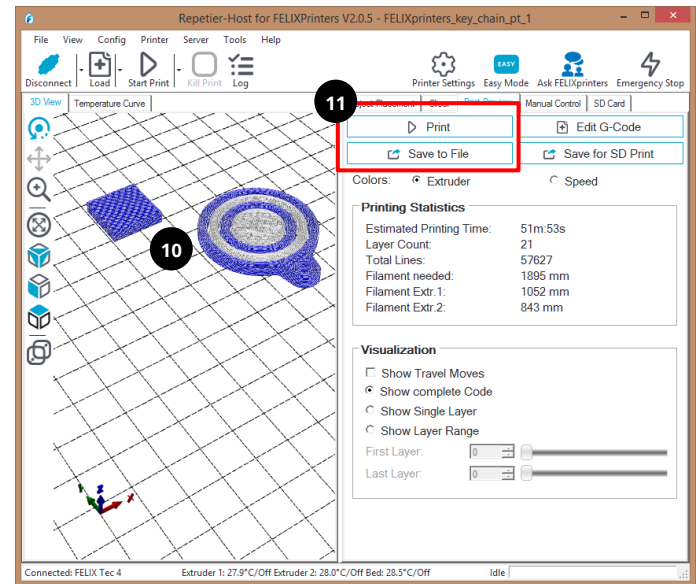
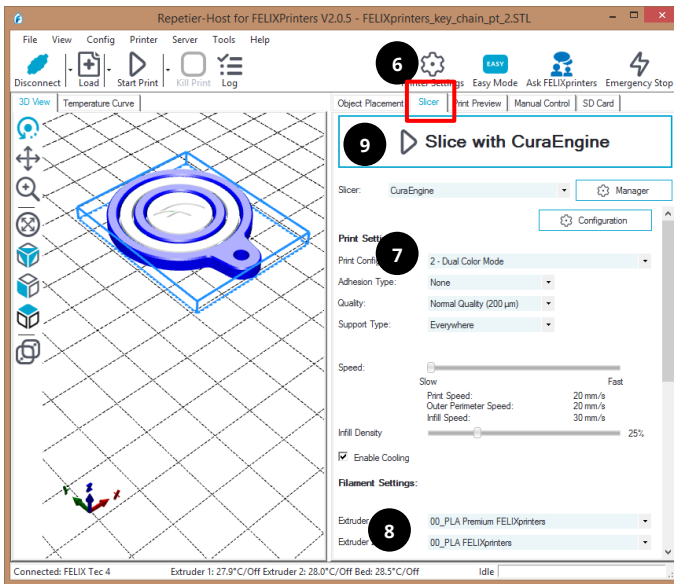
This chapter describes how to print one object with two colors. Basically, what we do is import two objects and align them properly. Then assign each object with the correct extruder to print.

1. **Import two files**, we used the *FELIXprinters_key_chain_pt1.STL* and *pt2*. Find in SD card.
2. Drag object pt2 onto pt1 as indicated in the picture
3. Objects will be matched according to their respective coordinate systems and you will see that they are displayed in a group.
4. Press the rotate button and set the x rotation to 90 deg. The object will lay flat on the surface.
5. Then Select Extruder 2 for pt_2. The 2nd part will show white.



12 Dual head multi-color printing

6. Go to slicer tab
7. Choose Dual Color Mode for print configuration
8. Select Correct Material for each extruder.
9. Now press the *slice with CuraEngine* button
10. Inspect the preview window. Make sure a wipe tower is present and that you see two colors in the object.
11. Press Print to print via USB or Save to File for printing it later





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 Repetier-Host for FELIXprinters

V1-2017



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

Zeemanlaan 15
3401 MV IJsselstein
The Netherlands

