

Tutorial

Printing with Repetier-Host

27 October

2016

Revision 0



1 Introduction

This tutorial describes how to get up and running to use Repetier-Host to print with your FELIX printer.

It assumes you have access to the Repetier-Host website and that you are familiar with the basic user interface elements. The tutorial will not describe in detail how to configure all the parameters of the application.

For more detailed information about the myriad of options, please check the Repetier-Host website:

<https://www.repetier.com/>

The tutorial assumes you are going to use the latest Windows version (Repetier-Host 1.6.7) and that you have a FELIX Pro series printer connected to the computer (this is a dual head printer).

The tutorial can also be used for Felix 3 series printers, but some details will look a little different.

2 What can Repetier-Host do?

The Repetier-Host application has four important functions:

- It allows you to import a 3D file and view and manipulate it so it can be printed by the printer
- It allows you to import a prepared print job file and preview it
- It can communicate with the printer to send it the right commands via a USB cable.
- It provides a user interface to manually control the printer from a computer

3 Installing Repetier-Host

3.1 Preparation

Please make sure the computer, on which you are going to install Repetier-Host, has access to the internet.

Personal firewalls, virus scanners and corporate firewalls are known sources for interference with the installation process of the software.

We recommend to use the customized version from the FELIXprinters website. It includes the right drivers for the FELIXprinters and printing profiles for the integrated slicer software. You can find the installation files here:

http://www.felixprinters.com/downloads/?dir=software/1_Repetier_Host

If you decide to use the generic version from the Repetier website, please make sure the right drivers for the printer are installed as described in the printer user manual. You need to import the profiles manually.

If you experience connection problems later on with the FELIX 3 series printers, it is recommended to install the latest FTDI VCP serial port drivers. You can find them here: <http://www.ftdichip.com/Drivers/VCP.htm>

3.2 Installation

For Windows, run the installer file and follow the directions.

For Linux, check the Repetier website:

<https://www.repetier.com/documentation/repetier-host/rh-installation-and-configuration/>

For Mac: run the installer file and follow the directions.

Note: The OSX version has not been kept up to par with the Windows and Linux versions. You will need more manual actions to get things up and running, and you will not be able to use some features that have been incorporated in the other versions.

4 User interface

This chapter will show a short overview of the Repetier-Host interface.

First, open the application. The interface will present itself.

The main components of the interface are organized as follows:

The screenshot shows the Repetier-Host V1.6.2 interface. Callout 1 points to the menu bar (File, View, Config, Printer, Server, Tools, Help). Callout 2 points to the general menu (Connect, Load, Toggle Log, Show Filament, Show Travel). Callout 3 points to the 3D view area showing a grid. Callout 4 points to the control area with various buttons and sliders. Callout 5 points to the log output window at the bottom. Callout 6 points to the status line at the very bottom.

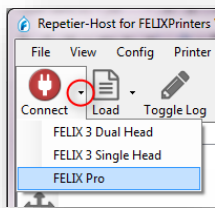
1. Menu bar
2. General menu
3. View area
4. Control area
5. Log output window
6. Status line

The control area is roughly organized according to the work flow of 3D printing. You start with the tab "Object placement", move on to the tab "Slicer", continue with "Print preview", and finish with "Manual control" and "SD card".

5 Connecting to the printer

First, select what model printer you have.

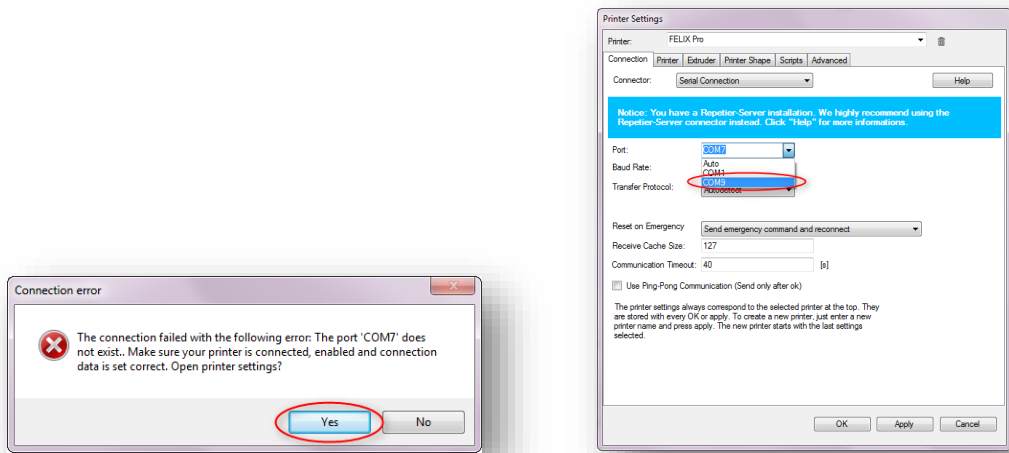
You can do this either via the Config menu bar or via the pull-down menu next to the "Connect" button.



Then, click the connect button:

If the connection is successfully made, the button will turn green:

If the program cannot find a printer at the port that is specified in the printer settings, it will complain and present you with a dialog to select the right port if you click “Yes”:



You can select the right COM port with the drop-down menu and click “Ok”. Try to connect again.

If this fails, exit the program, turn off the printer, turn it on again and restart Repetier-Host. Then try the previous steps again.

6 Preparing a model for single head printing

6.1 Create a 3D file

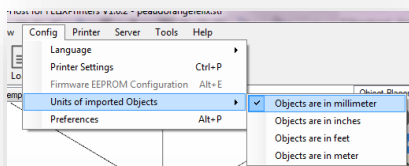
Repetier-Host accepts 3D files in the [STL file format](#), AMF file format, OBJ file format and 3DS file format. If you designed a 3D object yourself, you need to export it from the design software into one of these formats. The STL format is the most commonly used format in the 3D printing community. Note which dimensional unit you use for the export (for example, millimeter). In this tutorial we assume you are going to print an STL file that has the mm as unit.

6.2 Import the 3D file

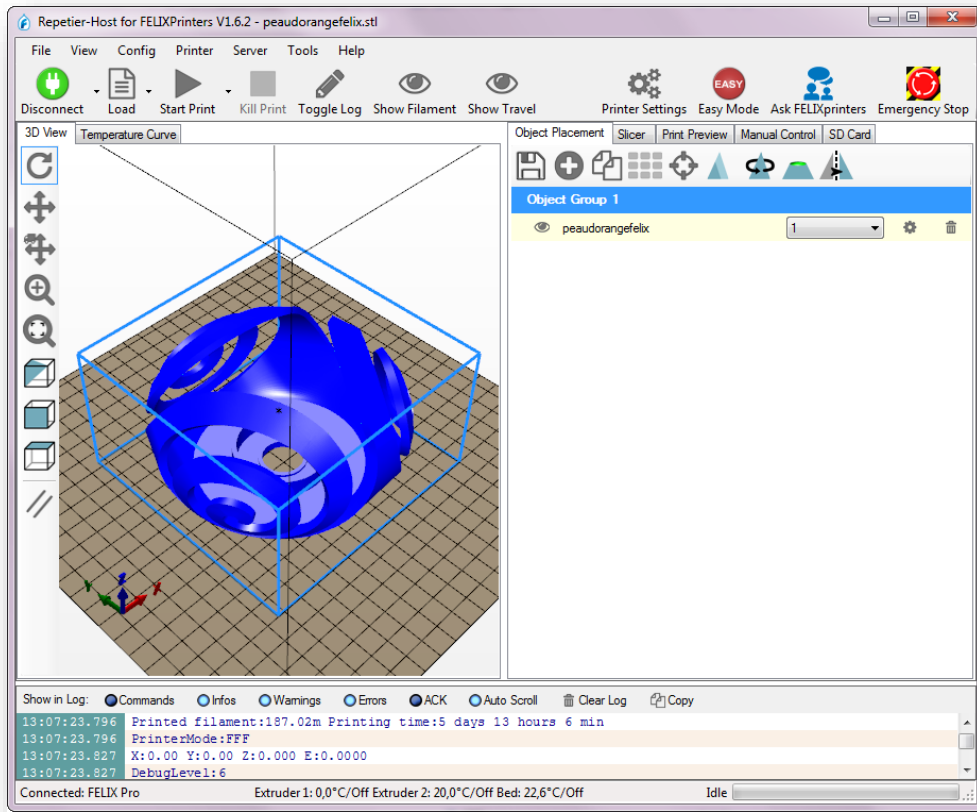


Click the “Load” button:

A file selection dialog will open that allows you to select your STL file. By default, the program assumes your STL file is in mm. You can change this in the “Config / Units of imported Objects” menu:



The file will be imported and it will be placed centered on the build platform.

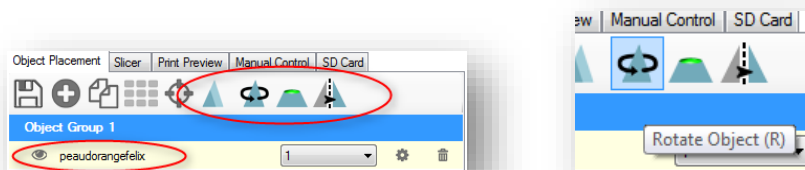


6.3 Orient the part

Now, take a look at the control area of the interface.

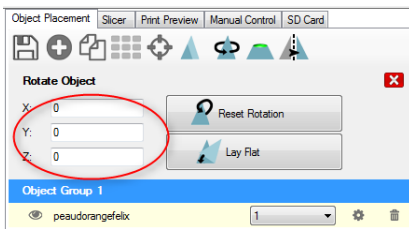
The "Object Placement" tab is active, and it shows the imported object. If the object is selected, there will be a blue bar above it and a blue box will appear around the object in the view area.

The tab also shows some icons that allow manipulation of the selected object. If you hover the mouse arrow above an icon, a short description will appear:

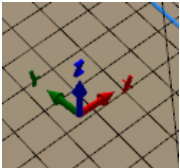


If for some reason you are not satisfied with the position of the object, you can change it here.

For example, if we want to rotate the object, we click the “Rotate Object (R)” button. The rotate tool appears:



The tool allows you to enter the angle in degrees that the object will rotate around the workspace axis. You can see the alignment of the axes in the bottom left corner of the view area:



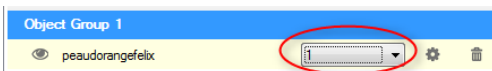
(Please note that the tool has a fixed rotation sequence: It will first rotate around X, then around Y, then around Z. The result can sometimes be surprising, but if you keep this in mind it will become clearer)

It is left to the reader to explore the rest of the manipulation tools. Look here for more information:

<https://www.repetier.com/documentation/repetier-host/object-placement/>

6.4 Choose the extruder

Next, we need to tell the program what print head we want to use for printing this object. This can be done in the object list.



The drop-down list indicates which print head will be used to print the part. It depends on your printer how many choices you have here.

6.5 Choose a slicer

To prepare a 3D object for printing, the object data must be processed into a format that the 3D printer understands. This processing is called “slicing” since the software (virtually) cuts the 3D objects into slices.

The software will generate an optimal path for the print head to recreate each slice in plastic. This is a rather complex trick, so it will take some time (sometimes a lot of time for big and complex objects).

The printer will stack each next slice (also called layer) on top of the previously printed slice.

Repetier-Host has bundled two slicing programs that you can choose from: CuraEngine and Slic3r.

FELIX has bundled printing profiles for the slicing programs with the FELIX version of Repetier-Host.

Both slicing programs have their strong points and weaknesses.

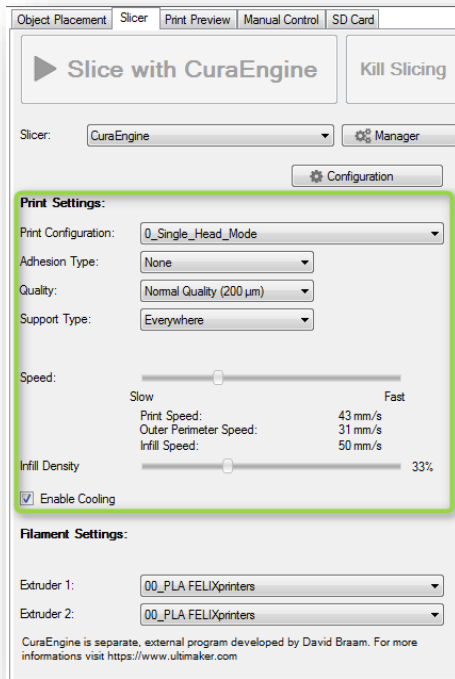
It is recommended to use CuraEngine to begin. It has good speed and it is in general pretty reliable.

Slic3r has a multitude of options to tweak and adjust, which makes it both flexible and complicated. Its speed is lower.

In this tutorial we will be using CuraEngine. Look at chapter 6.11 for a short description of the Slic3r program.

6.6 Preparing the slicer

In the slicer-tab, you'll find the CuraEngine slicer with its settings. The screenshot below shows what it looks like and what the available options are:



First, we are going to set the print settings.

6.6.1 Print Configuration

Depending on the printer you have chosen, you can choose between two print configurations.

- 0_Single_Head_Mode
This enables single head printing. It takes the activated extruder to print with. Everything gets printed with one extruder.
- 1_Dual_Head_Mode
This enables dual head printing. It takes extruder 1 to print the object and extruder 2 to print the possibly required support structures or parts of the object in another material.

6.6.2 Adhesion Type:

This is disabled by default. For special cases, where the object is very small or has a limited contact area with the build plate, it is recommended to choose brim, or, in very special cases, the raft to get extra adhesion.

6.6.3 Quality:

Quality here means basically layer height of each printed layer. The smaller the layer height, the more detailed the printed object will be.

For reference: the 100-micron layer height is the same as the thickness of a human hair.

We recommend to use 150 micron; it is a nice tradeoff between quality and print-time.

The extreme quality (50 micron) is experimental and might require some fine tuning for complex objects.

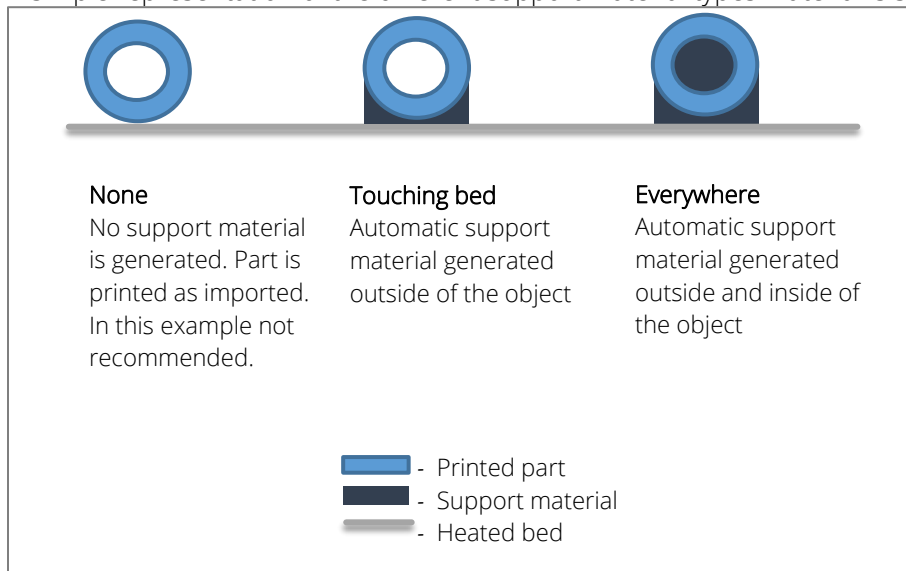
6.6.4 Support type:

Support structures are generated to support parts of the object when necessary. When printing layer after layer, each layer needs some supporting material to lean on, to prevent the layer from sagging. Sometimes extra support structures are necessary to provide enough support.

There are two options:

- *Touching bed:*
Support material is only generated which is physically touching the build platform.
- *Everywhere:*
Here the support material is also generated inside a model

A simple representation of the different support material types material is shown here.



6.6.5 Speed slider:

This will set the print speed. Increasing the print speed comes at a cost: you will get a less nice print-quality.

6.6.6 Infill Density:

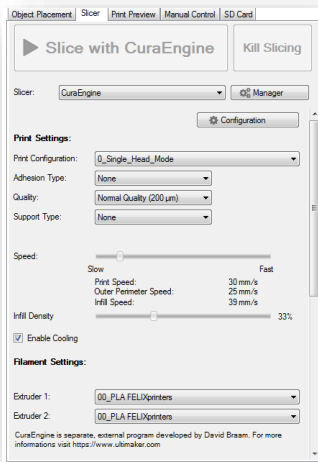
Solid objects are created by printing their contours and fill up the rest of the object with a structure of a certain density; the structure is called infill. The density can range from empty to fully filled. More than 70% infill is not recommended; it can cause deformations on the object.

6.6.7 Enable cooling:

When enabled, the extruded filament will be actively cooled with a workpiece cooling fan. This is to ensure conservation of details. It is especially important for detailed objects.

6.7 Example

So, let's assume you want to print the imported object in PLA with print head 1.



Choose the Print configuration "0_Single_Head_Mode".

Select Adhesion type: None, Quality: Normal, Support type: None.

Set the Print Speed to approximately 30 mm/s. Enable the cooling.

For each extruder, you need to choose what filament type currently is in the extruder.

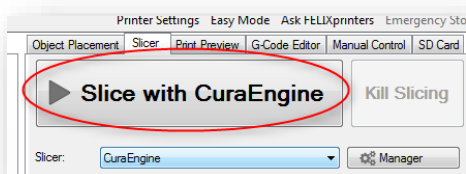
Set the material for extruder 1 to PLA. The other material is not important for now; you can leave it as it is.

6.8 Generating a preview

Now, we are curious of course what the slicing program will tell the printer to do.

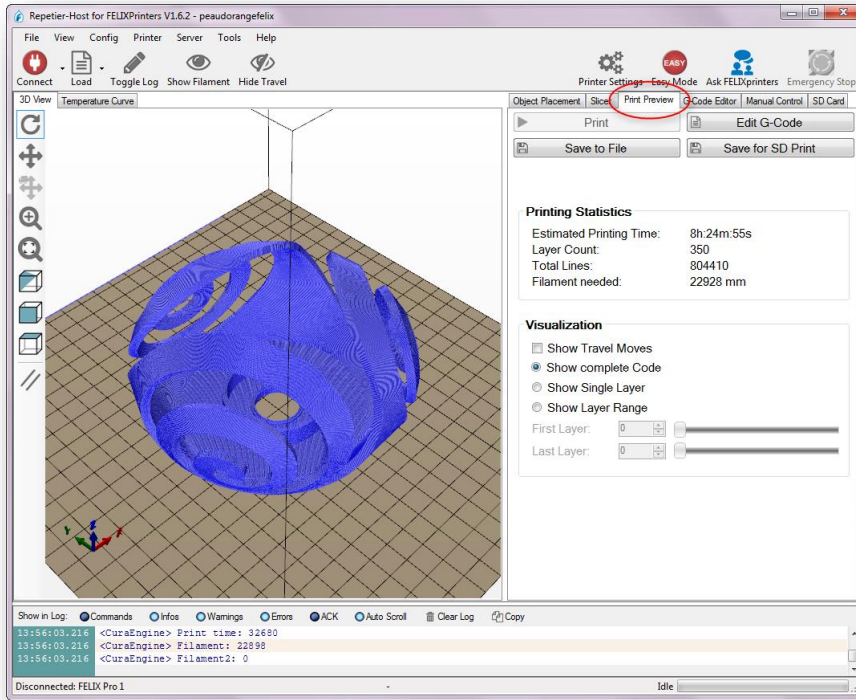
So instead of starting to print straight away, we will tell the program to generate a preview for us.

To do that, first click the "Slice with CuraEngine" button:

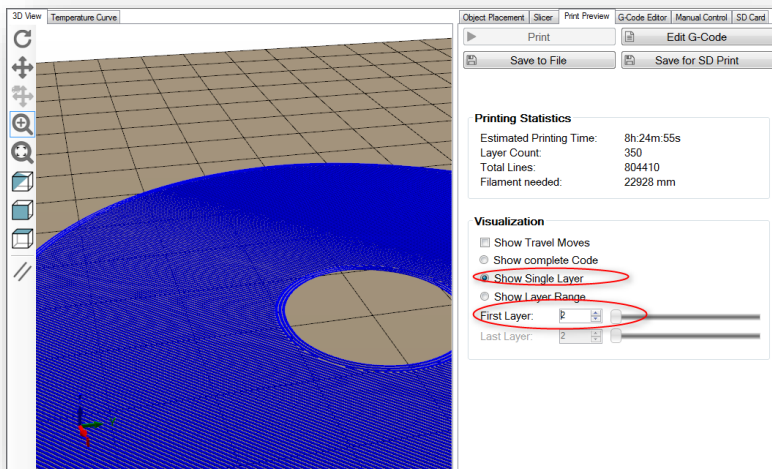


The program will start slicing the imported model (See also 6.5).

When the slicing is done, click the "Print preview" tab to show the sliced model:



The sliced layers will be displayed. You can also see one layer at a time by clicking "Show Single Layer".

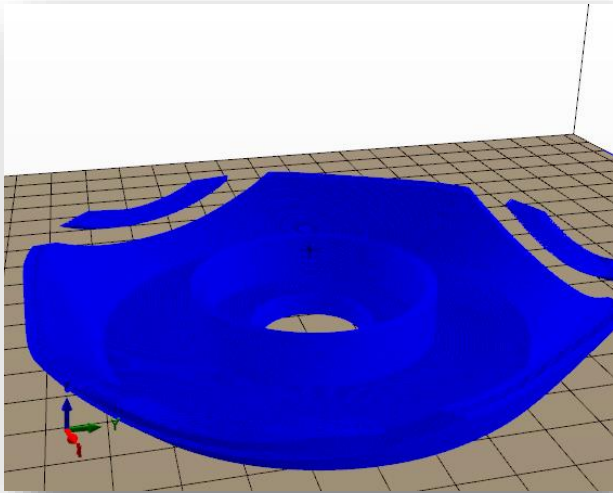


If you zoom in more, you can see the lines the printer will "draw" in plastic.

The slicer also generates an estimate of the printing time and the filament used for the print job. You can see these numbers in the "Printing Statistics" section of the "Print Preview" tab.

(The software can use different colors for each extruder. This is useful later if you want to print with multiple extruders).

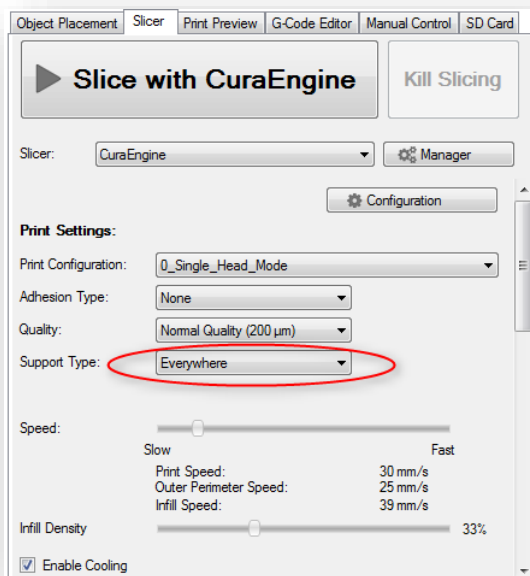
If we inspect some more layers, it becomes obvious the generated print job will encounter issues:



Some of the layers contain isolated parts that start in mid air! That is impossible of course, the material would just fall down.

The printer however has no clue about this. It will just follow orders and extrude the material as instructed. This is why previewing your print job is a good habit, it allows you to spot issues beforehand instead of after many hours of printing. The more experience you gain, the better you will become at spotting issues beforehand.

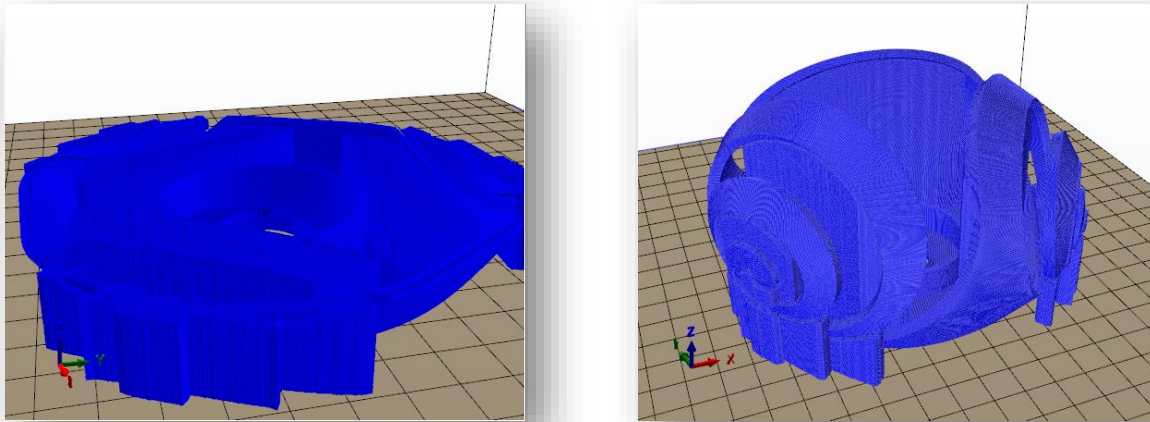
So, to correct this we need to change some settings. We will need a support structure for this object. You can enable the generation of that structure in the Slicer tab:



Choose "Everywhere" (see 6.6.4 for more information).

This will tell the program to generate support structures. It will use the same print head as used for the part itself.

If you slice again, you can see in the review tab that the program has added extra material:



This material provides the support for the parts that would otherwise fall down or deform too much. The material must be removed later, after the print job is finished. The slicer program uses some tricks that help easing the separating of the support structure from the printed part.

6.9 Print the file

Note: It is assumed the printer has been fully calibrated before. If not, check the FELIX Pro series / Felix 3 user manual to see how the printer should be calibrated.

Make sure the Repetier-Host program is connected to the printer (see chapter 5)

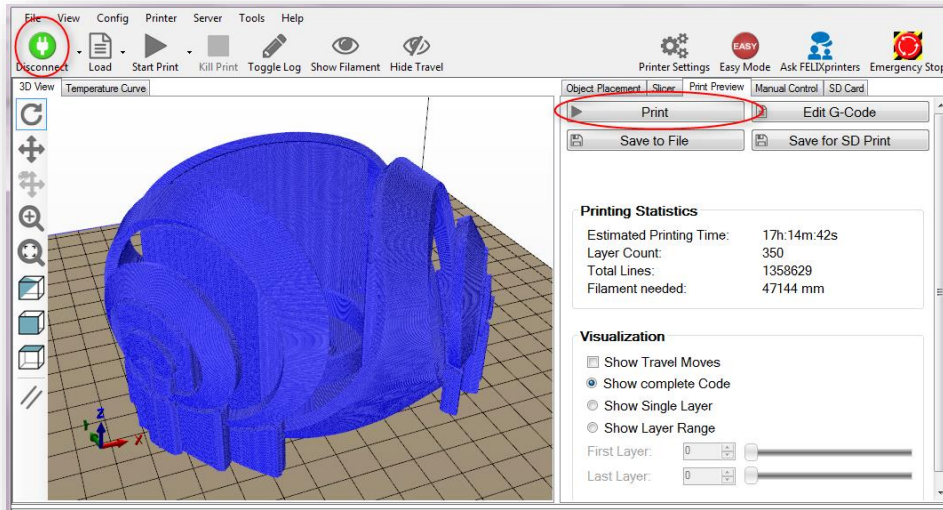
If it is connected (the button is green), check / make sure that the following is true:

- ✓ The printer axes can move without obstructions
- ✓ All plastic residues on the heated bed are removed, preferably with the supplied tweezers
- ✓ The heated bed surface is clean and degreased (blue spirit is the preferred cleaning agent).
- ✓ When all axes are homed, the hot-end is not touching the table.

Make sure the table is moved at least 10mm away from the extruder tip and insert the right filament in the extruder. Run the extruder until a steady flow of plastic comes out.

See also the printer manual (“loading filament into printer”)

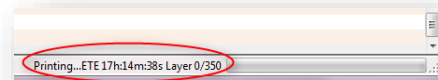
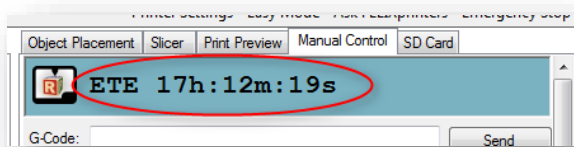
After that, press the "Print" button":



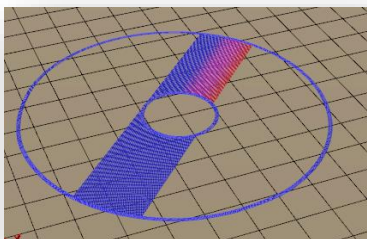
The printer will start the print job.

The program will show the manual controls so you can check what is going on with the temperatures of the extruders and the heated bed and with the workpiece cooling fan.

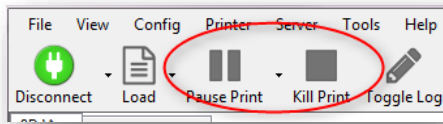
It will also display the progress of the print job in the top of the Manual control tab (and in the lower corner of the application window).



The program will show the print progress in the graphics view as well:



You can pause or cancel the current job with these buttons.



6.10 Save the print job

It is also possible to save a prepared print job.

The job will be saved as a so-called "g-code" file.

This file can be sent to another location with the same FELIX printer, or it can be stored for repeated print jobs or batch printing.

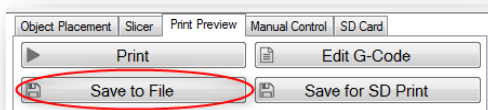
Another very practical use of a g-code file is that you can make the printer print without having to be connected to a computer. In case of a lengthy print job this minimizes the risk of interrupted communication because the computer decides to run an update, go to sleep mode, crash, ...

It also saves energy.

To print the saved print job, you need to copy the g-code file on a micro SD card, insert the card into the printer and then start the print job from the printer interface. See the printer user manual for more details.

To save the prepared print job, you first must have sliced a 3D object with the right settings.

Then, press the "Save to File" button:

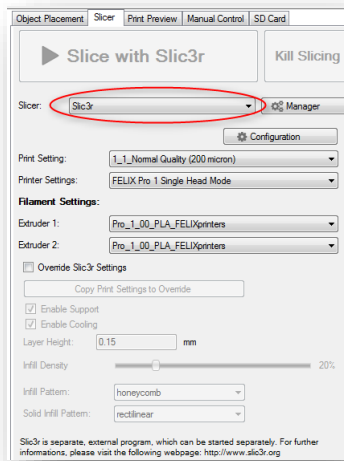


(You can also use the Save for SD print button, that option allows you to save the job in a binary format that is more compact but not human readable.)

A file dialog will appear that allows you to enter a filename and location for the g-code print file. Once you click "Ok", the program will save the prepared job.

6.11 Extra information: Slicing with Slic3r

To slice with the other slicer that is packed with Repetier-Host, select “Slicer: Slic3r”:



The setting options will change to that belonging to the Slic3r program.

The general principle is the same: You use profiles to select default behavior of the slicing program for print quality and the number of heads used.

Print and printer settings:

With these settings you can print in two modes.

1. Single head printing
2. Dual head printing.

1. Single head printing:

To enable this, configure it like this:

Print settings: choose one of the qualities with where the second number is a **1**.

So for instance 1_1_Normal Quality (200 micron). This enables single head mode.

Printer settings: Choose FELIX Pro Single Head Mode

2. Dual head printing:

Print settings: choose one of the qualities with where the second number in the *print settings* is a **2**.

So for instance 1_2_Normal Quality (200 micron) enables dual head mode.

Printer settings: Choose FELIX Pro Dual Head Mode

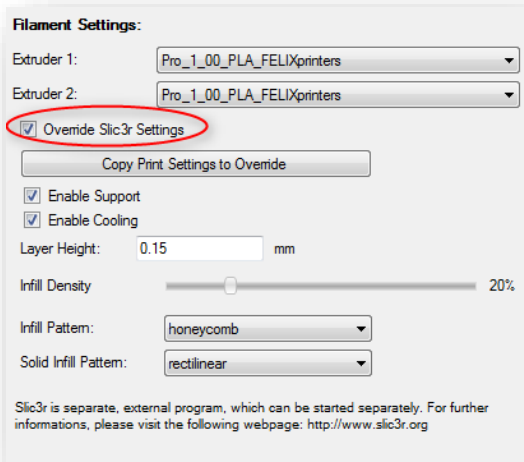
This will print support material with the second extruder.

Filament Settings:

Extruder 1: choose your filament type

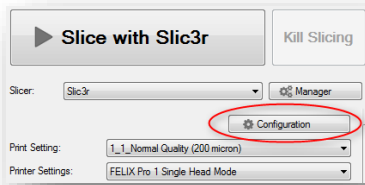
Extruder 2: choose your filament type, this one will be ignored if you have the print and printer settings configured for single head mode.

The profiles can be managed in a separate Slic3r window. Repetier-Host allows you to override some of the settings defined in the profiles. You can do it by ticking the “Override Slic3r Settings” box:

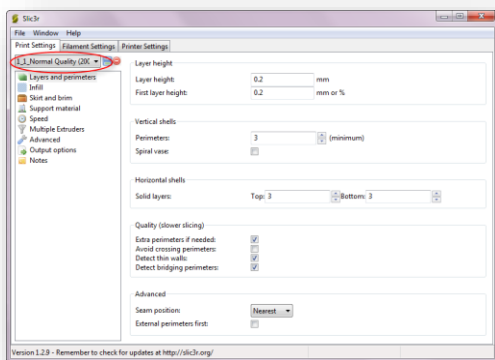


You can copy the settings in the profile to the Repetier-Host window by clicking the “Copy Print Settings to Override” button. After that you can alter the values.

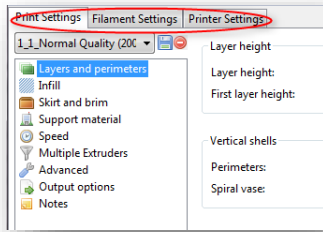
If you want to get more involved, you can also edit the Slic3r profiles. To do this, click the “Configuration” button:



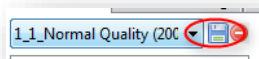
The Slic3r interface will appear.



It will show the “Print setting” parameters for the current selected Quality profile. The same window allows you to show and edit the profiles for the filaments and the printer itself:



Be careful, if you alter a setting it will not be saved by default (the program will ask you what to do if you made changes). On the other hand, if you save your changes the current profile will be overwritten with the new settings. It is wise to always create a copy of the standard profile before editing it. You can do that by entering a new name for the profile after hitting the “Save” button:



A brief description of the options in the “Print settings” window:

Copy Print Settings to Override. This will copy the settings in the selected print setting to the fields of this current window.

Enable support, this enables the generation of support material where needed.

Enable cooling, enables cooling of the object during printing. This normally improves print quality especially for detailed objects.

Layer height, the height of each printed layer.

Infill density, the percentage of infill of the object. 0 means a hollow object. 100% means fully filled.

Infill pattern, pattern of the infill.

Solid infill pattern, pattern of the infill for the bottom and top of the object.

For a (very) detailed overview of the parameters and their functions in the Slic3r software, please take a look at the Slic3r website: <http://manual.slic3r.org/>

7 Preparing a part for dual head printing

7.1 Starting points

Dual head prints expand the possibilities of what kind of objects you can print. You can print objects with more colors. You can create objects with different properties (for example a wheel with a PLA (tough) rim and a Flex (soft) tire. You can also print the supporting structure in a material that is soluble in water or limonene. That allows you to print virtually any shape and eliminates the tedious process of manually chipping off the support structure.

For dual head printing, you (obviously) need a dual head printer.

You also need a 3D model that consists of multiple parts. You can determine per part what print head will be used for printing that part.

It is important that the 3D models for the different print heads share the same origin, otherwise the object will not be imported at the position where they fit together perfectly.

You will also need some experience with printing single head prints successfully. Printing dual head files is more difficult because more things can go wrong. If you are unable to successfully print single head jobs, please keep practicing to master it. Only then move on to dual head printing.

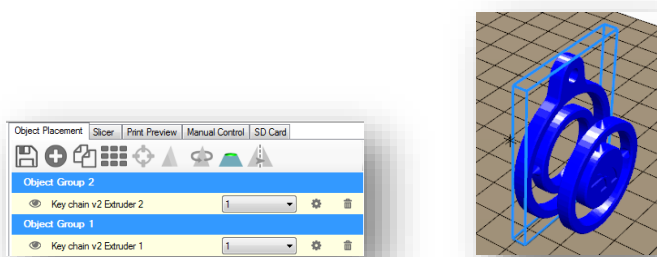
7.2 Importing the parts

This is an example of a simple 2 color CAD model, a key chain with rotating parts:

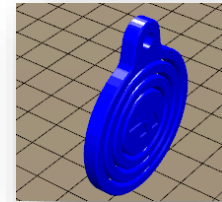
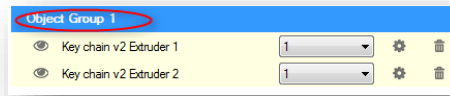
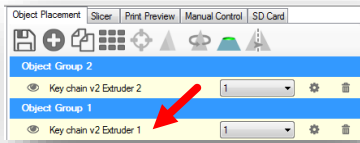


The objects have been exported as two STL files each containing 2 parts.

The exported STL files have been imported in Repetier-Host, just the same as in 6.2. Each imported object will be placed in a separate Object Group by default:



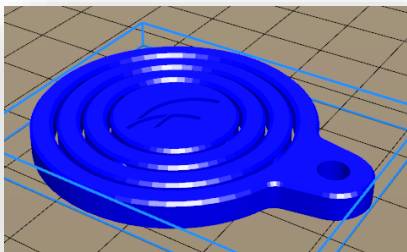
You can drag one object with the mouse into another group:



The program will automatically put the objects at the same origin.

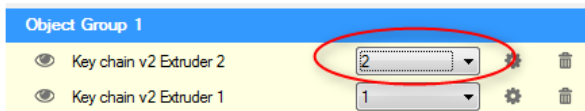
7.3 Orient the parts

Because you grouped the parts together, they will move as one object. You can use the same tools as with a single object to put the object into the desired position.



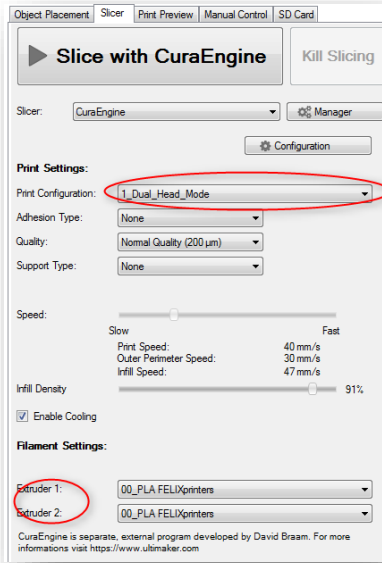
7.4 Select extruders

In the object list, set the nozzle for the second object to print head 2.



7.5 Choose materials

Check if the materials for both print heads are set as you want it.
Select the "1_Dual_Head_Mode" print profile.

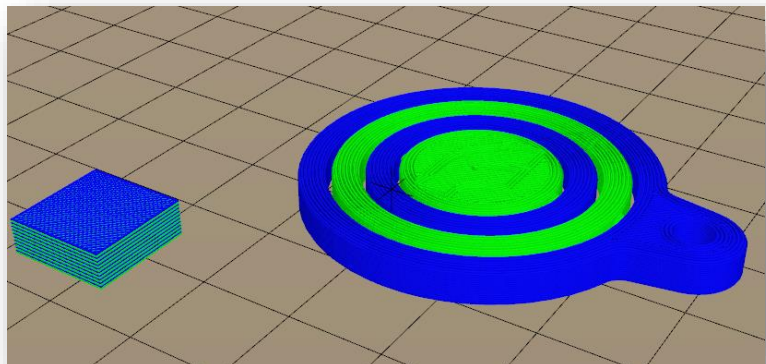
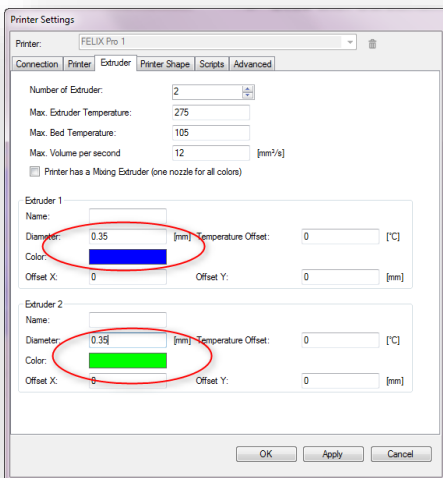


We don't need any support structure for these parts.
The rest of the settings work the same as explained in 6.6

7.6 Previewing

You can preview the print job just as with a single head print (see 6.8).
You can see the left and right extruder lines in different colors.

To choose these colors, check the printer settings, tab Extruder:



You can also see that an extra structure is added (the wipe and prime tower). This structure helps the extruder to get going after it has been idle for a while, and it will increase the chance of wiping off any excess filament that could be hanging from the idle extruder.

7.7 Print/Save the file

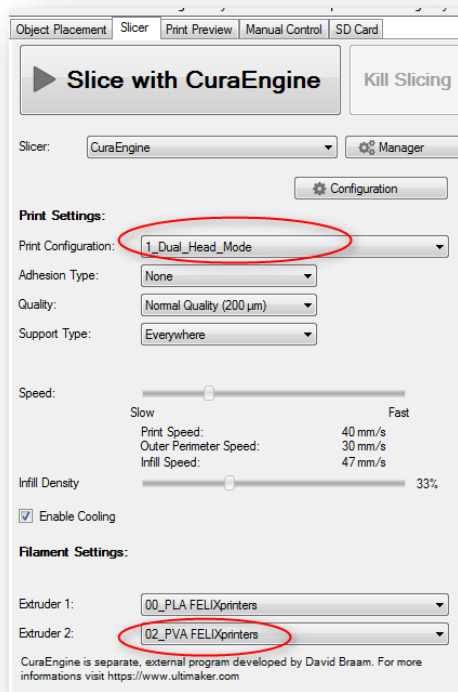
Printing and saving the job is pretty much the same as a single head print, see 6.9 and 6.10.

When printing the job, you will see that the printer will alternate which head it is using. It will wait a while in between to allow the active extruder to reach the right temperature. At the same time the printer will lower the temperature of the idle extruder to prevent it to drip molten material on the object.

8 Printing a part with soluble support

Printing a part that has a support structure that is made of another material is very similar to dual head printing. The difference is you don't need to have a 3D object in two separate parts, since the support structure is generated by Repetier-Host itself.

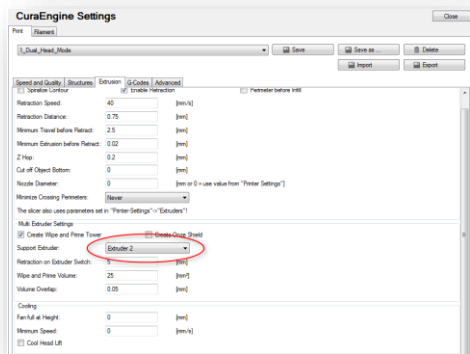
To instruct the program to prepare a job like this:



So:

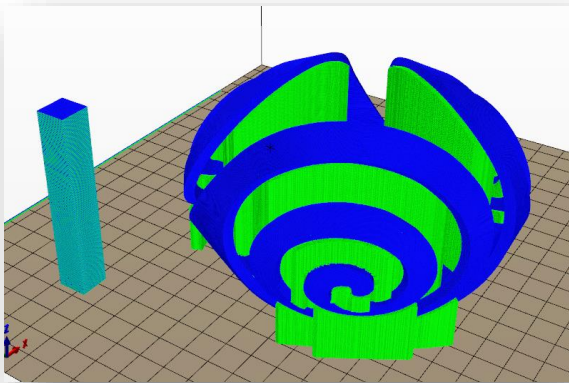
- We will use PLA in print head 1. We tell the program that the parts will be printed with that nozzle.

- We will use a water-soluble material (PVA) in print head 2. This head will be used for the support.
(You can check this in the CuraEngine settings:)



- We instruct the program to generate a support structure.

This is a preview of the output of the slicing program:



You can see the program has generated the support structure and that it will be printed with the other extruder.

The rest of the printing process is identical to that of single or dual head printing (see 6 and 7).

9 Felix Support

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

Kind regards,

FELIXprinters