

Tutorial

# Printing with FELIXbuilder

27 October

2016

Revision 0



# 1 Introduction

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

It assumes you have a copy of the FELIXbuilder manual 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 FELIXBuilder user manual.

The tutorial assumes you are going to use the latest version (FELIXbuilder 2.1) 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 FELIXbuilder do?

The FELIXbuilder application has three important functions:

- It allows you to import a 3D file and view and manipulate it so it can be printed by the printer
- 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 FELIXbuilder

### 3.1 Preparation

Please make sure the computer, on which you are going to install FELIXbuilder, has unrestricted access to the internet. Personal firewalls, virus scanners and corporate firewalls are known sources for interference with the activation process of the software.

Please make sure the right drivers for the printer are installed as described in the printer user manual.

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 installation details, please see the "FELIXbuilder User Manual", chapter 2.

## 4 User interface

For an overview of the features of the User interface, check the FELIXbuilder user manual chapter 2.2 and further.

## 5 Preparing a model for single head printing

### 5.1 Create an STL file

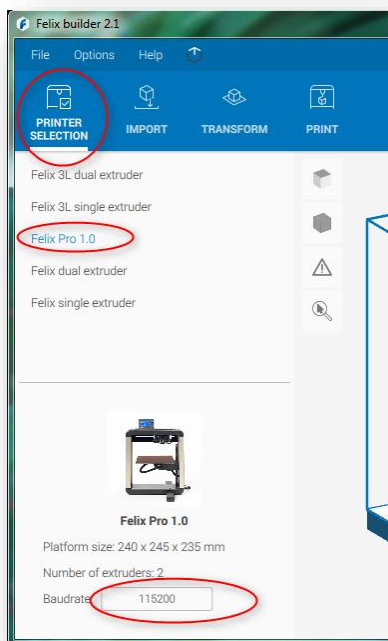
FELIXBuilder only accepts 3D files in the [STL file format](#).

If you designed a 3D object yourself, you need to export it from the design software into this format. Make sure you use the millimeter as dimensional unit for the STL export.

### 5.2 Prepare FELIXbuilder

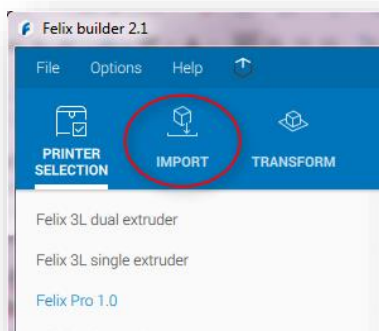
Open the FELIXBuilder application.

Make sure you select the right printer in the Printer Selection menu. Also, check the communication speed (baud rate).



### 5.3 Import the 3D file

Click the Import button:

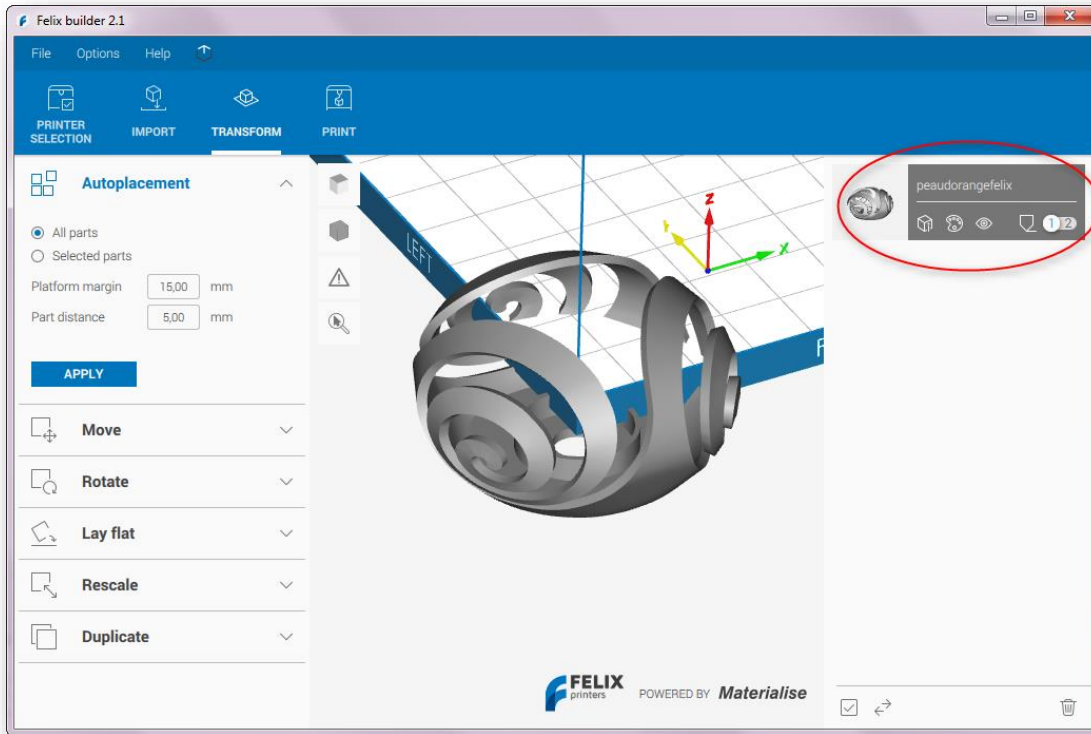


A file selection dialog will appear that allows you to select your 3D model. Select an STL file and click "Open".

The file will be imported.

The origin of the 3D object will be placed on the origin of the printer heated bed.

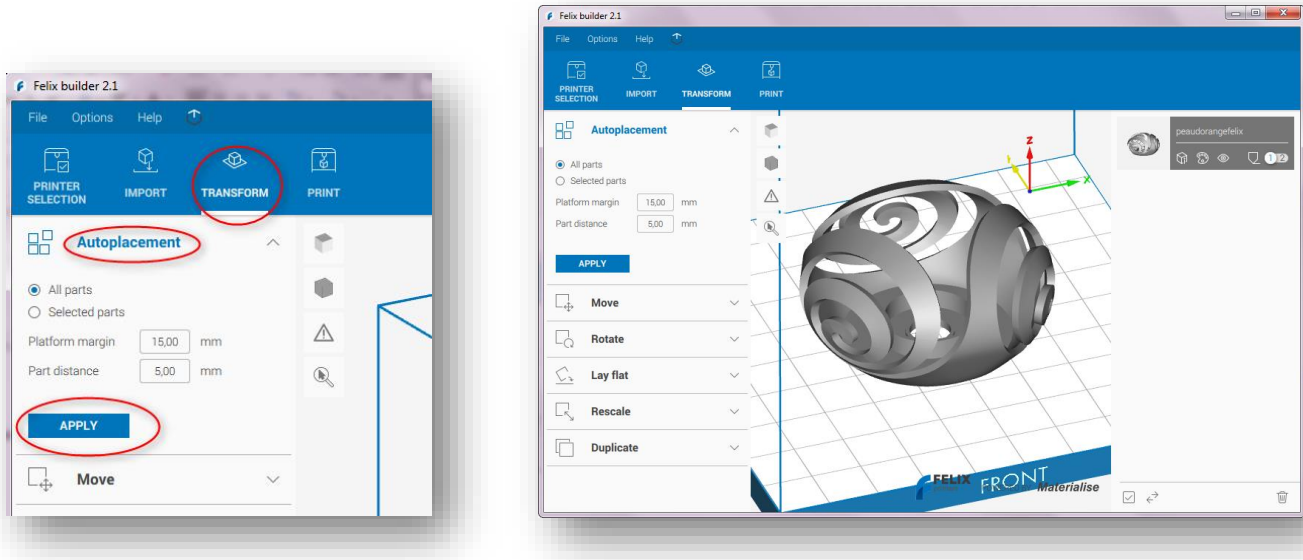
You can see the name and more properties of the imported 3D object in the **part list** on the right side of the interface. (See chapter 5.2 of the user manual for more information on the part list).



## 5.4 Orient the part

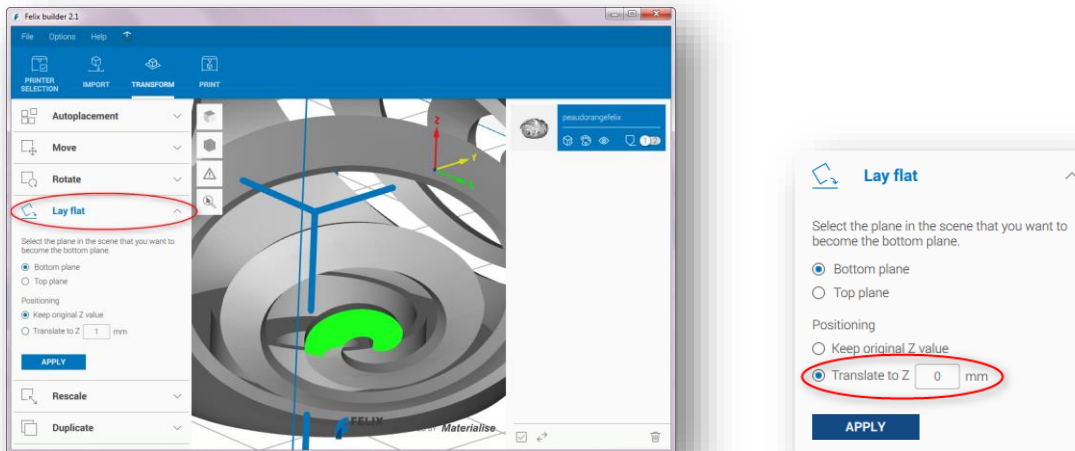
The origin is not a very convenient place for this part. It cannot be printed like this.

You can use the “Autoplacement” function in the “Transform” panel to let FELIXbuilder calculate a better spot. Click “Apply” to use the default settings. This will place all imported parts in a convenient position:

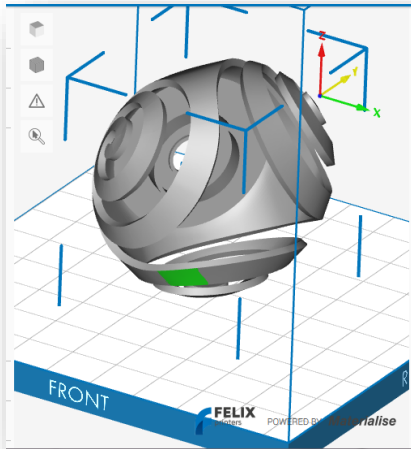


You can also select a different face of the part to put on the build platform of the printer. Use the “Lay Flat” transform for this.

Select the face that you want put on the platform. Set the “Translate to Z” to 0 and click “Apply”: This ensures the model touches the platform with the selected face.



The part will be positioned with the selected face on the platform.



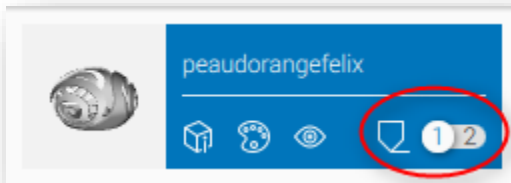
However, the first orientation was better for printing, so we will continue with that one.

The other transforms mostly speak for themselves. Check the manual (chapter 6) for more details.

(The transforms you apply will be applied to the objects you selected in the **part list**. A selected part will be highlighted blue in the list, and a wireframe cube will be drawn around them in the workspace).

## 5.5 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 part list.

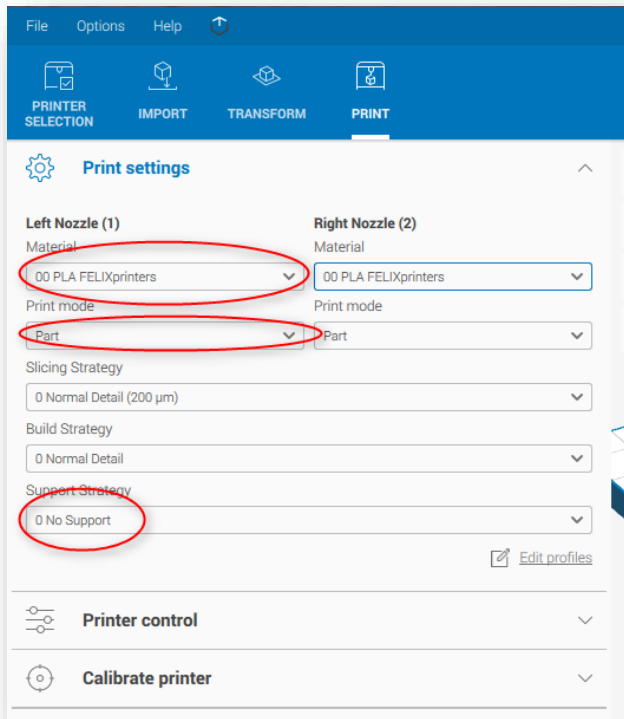


The slider indicates which print head will be used to print the part.

As an advanced user, you can also choose later if you want the inside (and possibly the support structure) of the part to be printed with a different print head. But that is beyond the scope of this tutorial.

## 5.6 Choose a material

We also need to tell the program what material is loaded in the print heads. This can be done in the “Print” tab.



Let’s assume you want to print this object in PLA with print head 1.

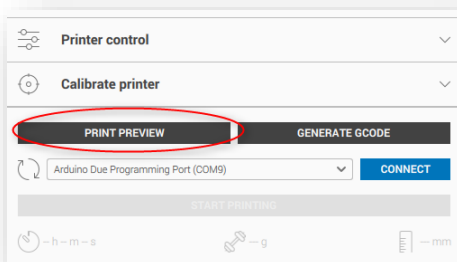
Set the material to PLA, set the print mode to Part.

We want the print to have a normal level of detail for the Z direction (slicing strategy) and a normal level of detail for the XY direction (build strategy\*). For educational reasons, we will not be using a support structure.

\*: See chapter 5.10 for more information on the Build Strategy settings

## 5.7 Generating a preview

Now, we are curious of course what the 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, click the “Print Preview” button:



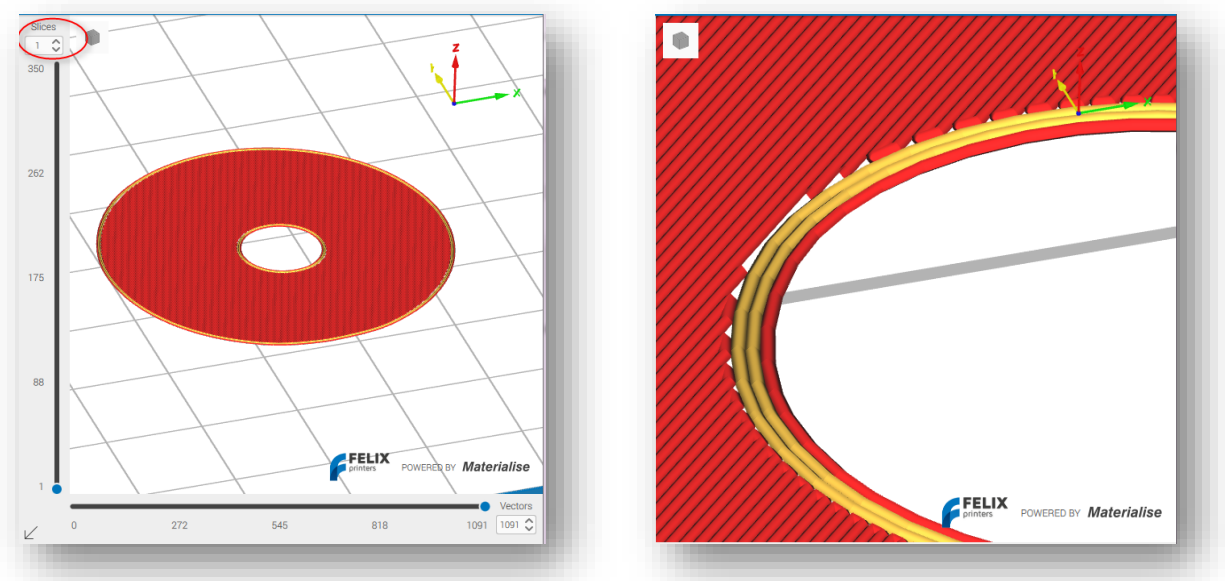
The software will start processing the parts.

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.

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

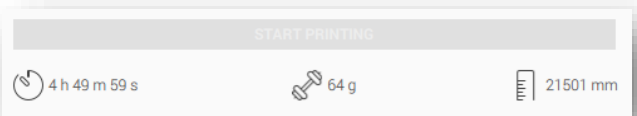
When the slicing is done, the first slice (or layer) will be displayed. If you zoom in more, you can see the lines the printer will “draw” in plastic:



In the top left of the object display area you can see which slice is being displayed.

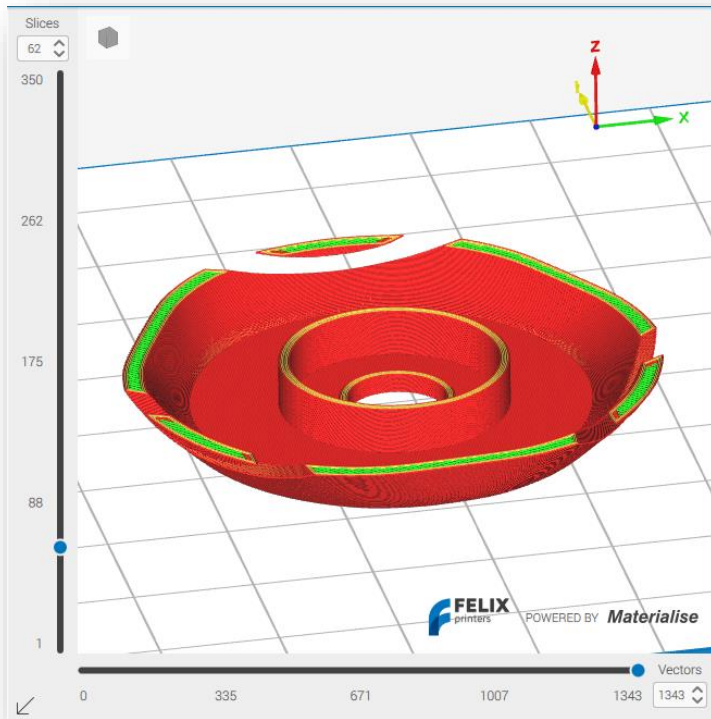
(The software uses different colors for different types of line in the slice. This can tell you something about how the printer will print that line. This is beyond the scope of this tutorial).

The preview also generates an estimate of the printing time, the part weight and the filament used for the print job. You can see these at the bottom of the print tab:





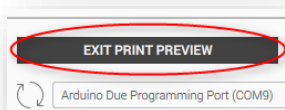
If we inspect some more layers, it becomes obvious the generated print job will encounter issues:



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

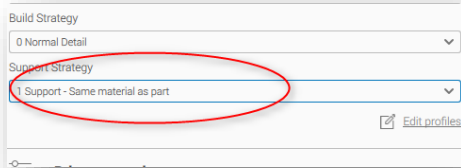
The printer however has no clue about this. It will just follow orders and extrude the material as instructed. This is why previewing your print job is a good habit, it allows you to spot issues beforehand instead of after many hours of printing. The more experience you gain, the better you will become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

So, to correct this we need to change some settings. First, click Exit Preview to activate the Print menu again:



For this object, we will need a support structure. You can enable the generation of that structure in the Support strategy selector.

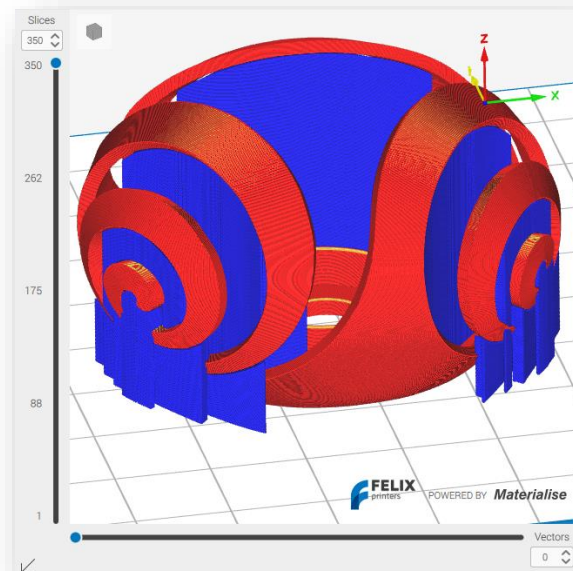
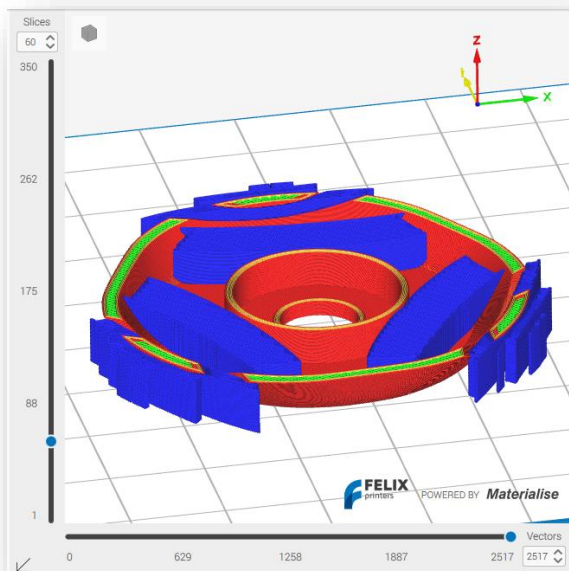
Choose "1 - Support: Same material as part":



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

(Other options are to add an extra brim of material around the part and/or the support structure (to increase the adhesion of the printed objects to the heated bed), and using the other print head to print a support structure from a soluble material.)

If you generate a preview again, you can see the program has added extra material (colored blue in the pictures):



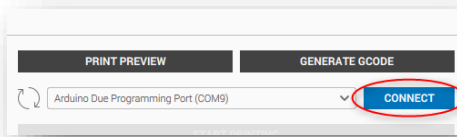
This material provides the support for the parts that would otherwise fall down or deform too much. It must be removed later, after the print is finished. The program uses some tricks that help easing the separating of the support structure from the printed part.

## 5.8 Print the file

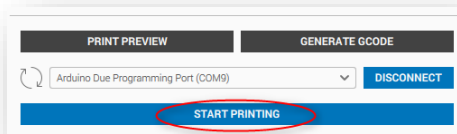
*Note: It is assumed the printer has been calibrated before. If not, check the FELIX Pro series / Felix 3 user manual to see how the printer should be calibrated*

Make sure the FELIXbuilder program is connected to the printer.

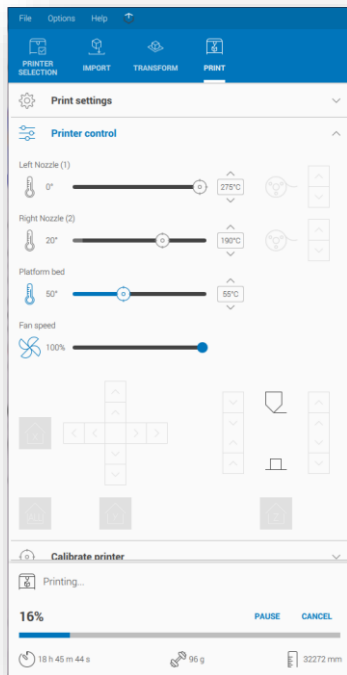
If it is not connected, select the right communication port and press the “Connect” button:



If it is connected, press the “Start printing” button”:



The program will start the slicing process once more. It 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. You can pause or cancel the current job as well.

## 5.9 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.

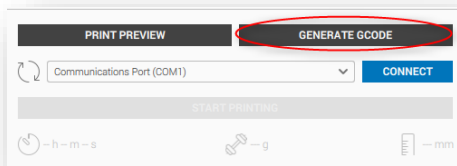
This file can be sent to another location with the same FELIX printer, or it can be stored for repeated print jobs or batch printing.

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, press the “Generate Gcode” button:



A file dialog will appear that allows you to enter a filename and location for the print job. Once you click “Ok”, the program will slice the parts once more with the latest settings and save the prepared job.

## 5.10 Extra information: Build strategy options

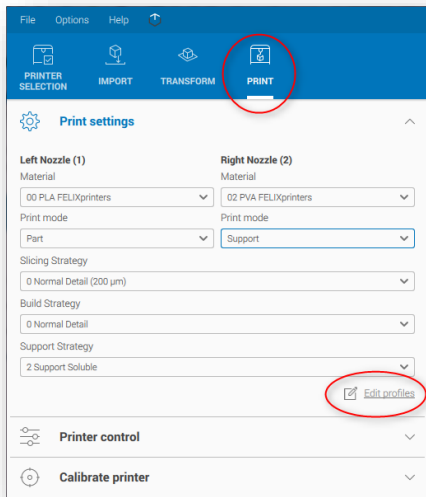
The build strategy does not only contain information about the level of detail in the X/Y direction.

It also comprises a lot of options on how the inside of the part will be generated, how thick the outer shell of the part will be, etc. We advise to get familiar with these settings, sooner or later you will want to change certain default settings.

Be warned though that there are many options available, and changing some options can cause the print to fail horribly.

To experiment, it is advised to first create a copy of a suitable standard profile, and then change the copied profile one parameter at a time.

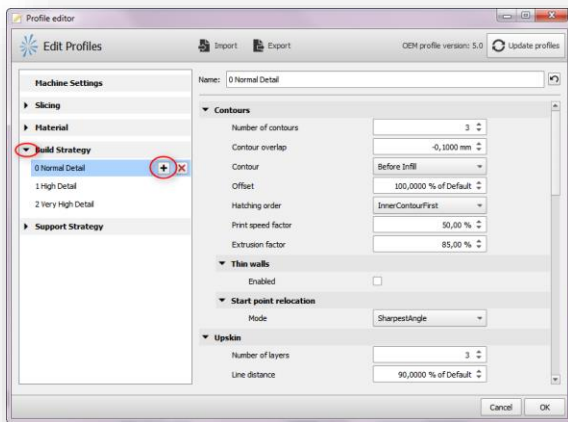
To do this, open the profile editor by clicking the “Edit profiles” button in the “Print” tab:



The profile editor will open.

Unfold the Build profiles section by clicking the triangle in front of it if it is folded.

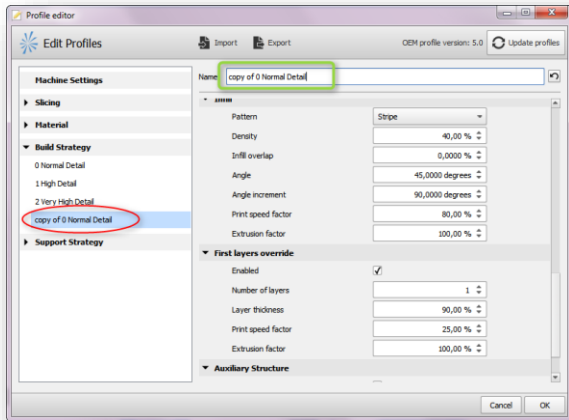
Copy a profile by clicking the “+” button that will appear if you hover the mouse above the profile:



A copy will appear in the Build Strategy list.

You can change the name and all the parameters of this profile.

Look in the FELIXbuilder manual for a description of each parameter (chapter 9.5).



The same applies to Material profiles and Slicing strategy profiles. See also the FELIXbuilder manual.

## 6 Preparing a part for dual head printing

### 6.1 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.

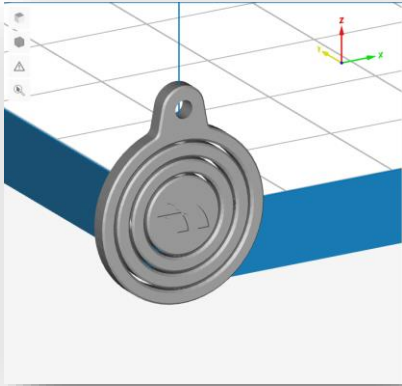
### 6.2 Importing the parts

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 FELIXbuilder, just the same as in 5.3. You can import multiple files at the same time by CTRL-selecting them just as you would in the file explorer.



### 6.3 Orient the parts

Manipulating multiple parts at once is kind of cumbersome in FELIXbuilder.

You don't want them to move separately, but as a whole.

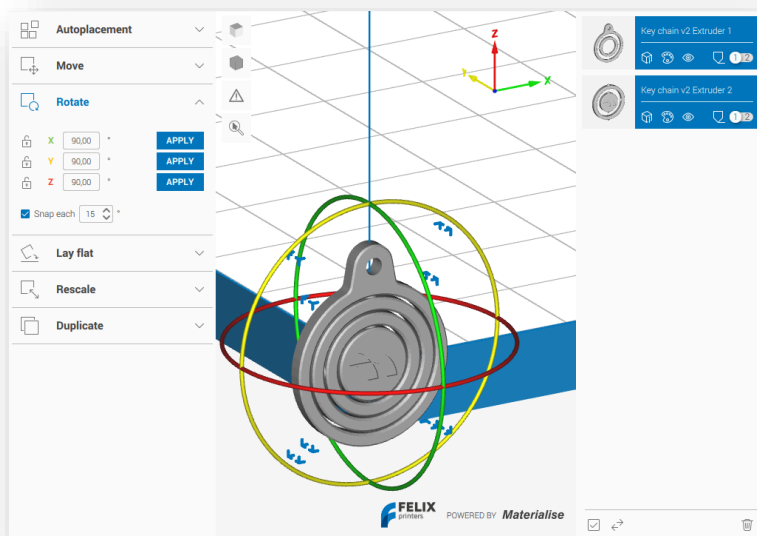
This is not possible with the Autoplace and Lay flat transformations at the moment of writing this tutorial

So you need some careful manipulation with the Rotate and Move transforms. Let's look at the example.

First, Rotate:

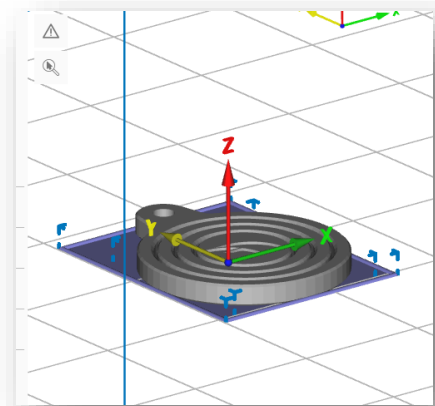
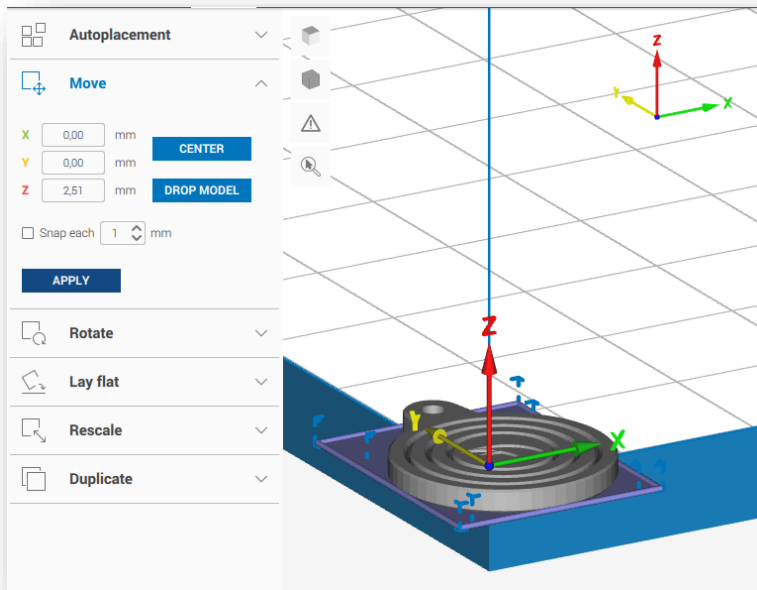
Make sure you select both parts in the part list by CTRL-clicking them. Use the colored manipulators to rotate the object in the right orientation. There is no undo option, so plan ahead a little.

If you can't get it right, it is often quicker to just delete the parts (select them and hit the "Delete key") and start over.



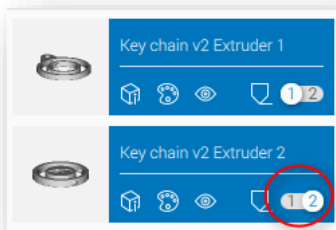
Then, Move

You can click the “Center” button to put both objects in the middle of the platform. Then, click “Drop model” to make both objects touch the platform



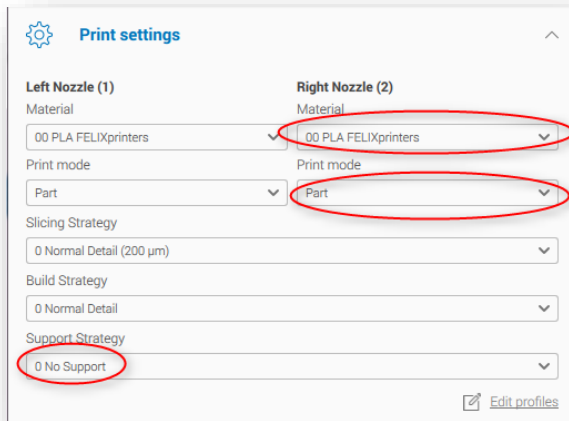
## 6.4 Select extruders

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





## 6.5 Choose materials



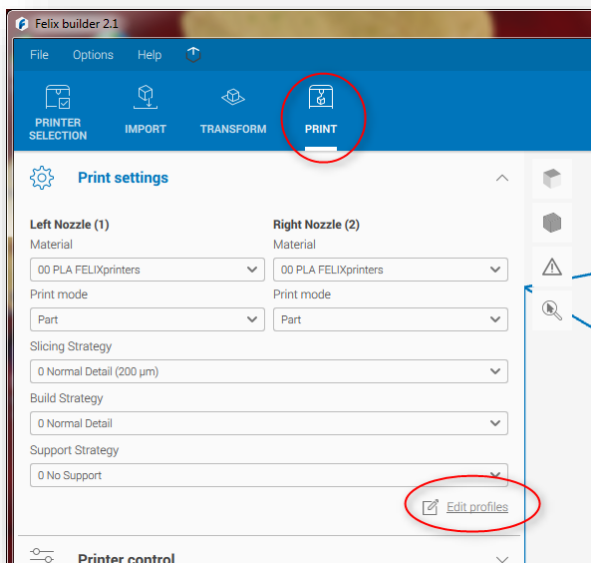
Check if the materials for both print heads are set as you want it. The print mode for the second head needs to be "Part" if you have a two-color print. We don't need support for these parts.

## 6.6 Bug workaround

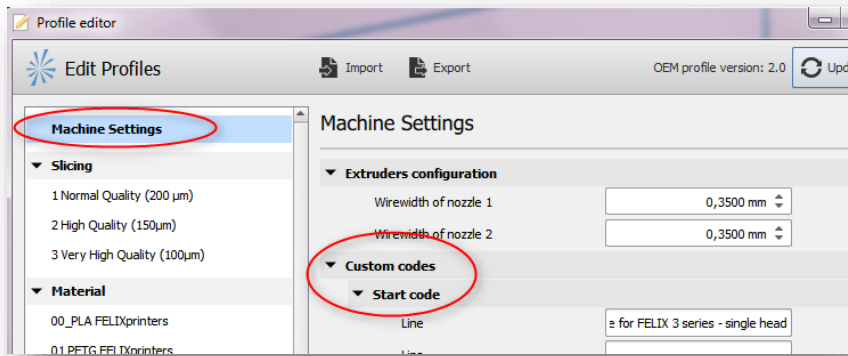
The most recent version of FELIXbuilder contains an inconvenient bug:

In the first layer, it will not heat the second extruder in case of a two-material print (Part-Part) like this example.

To work around this bug, open the profile editor:

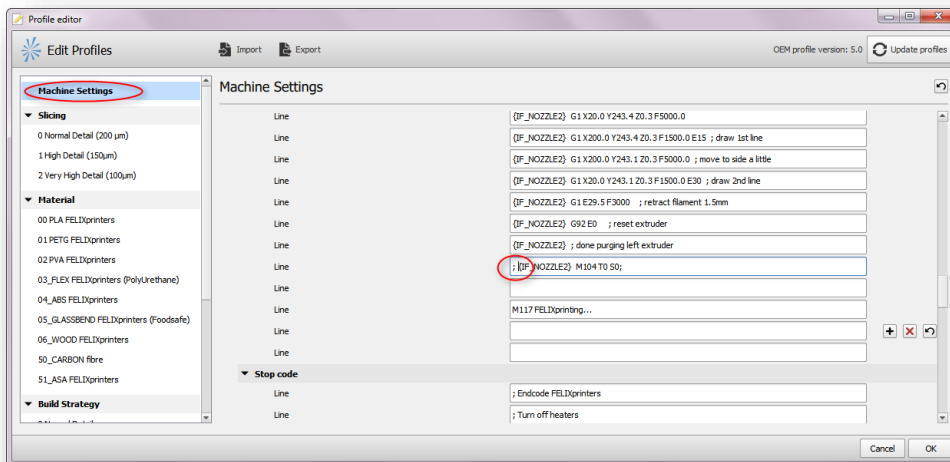


Go to the machine settings and look in the "Custom codes" section:



Scroll down to the end of the “Start code” section.

Put a semicolon in front of the line “`{IF_NOZZLE} M104 TO S0;`”:



Now the program should generate correct code for prints that use both heads for part printing.

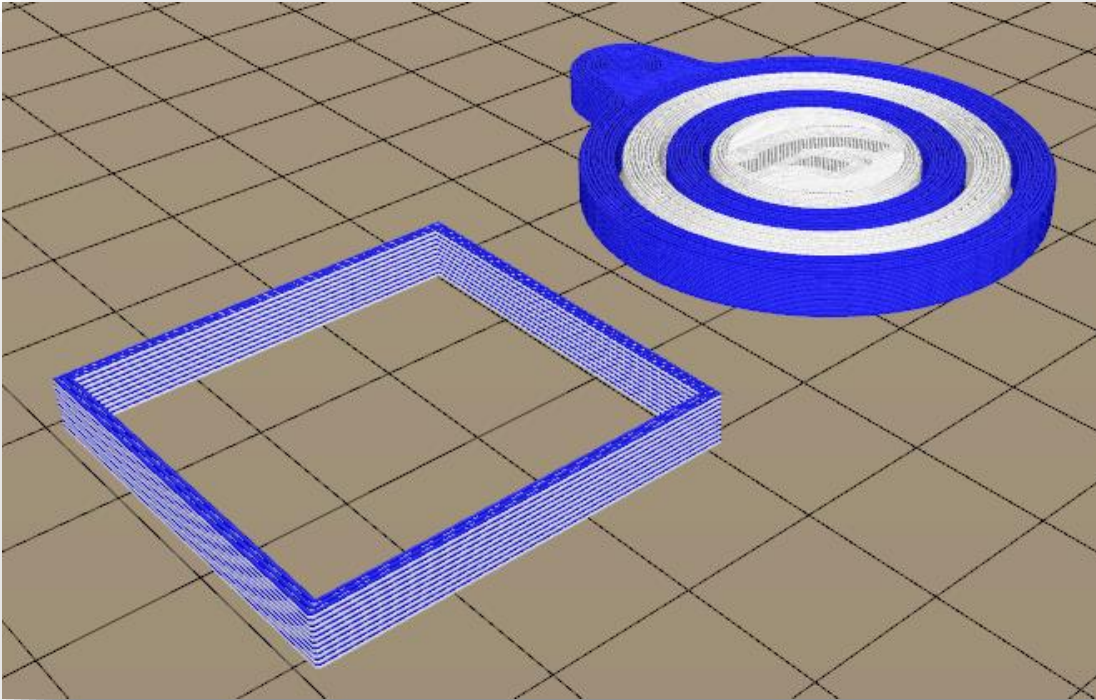
For prints that use:

- a single head (see 5)
- one head for part material and the other head for support material (See later, 7)

remove the semicolon again.

## 6.7 Previewing

You can preview the print job just as with a single head print (see 5.6). It is not possible to view what line will be printed with what extruder. To give you some insight, we visualized the generated print job in another external program (Repetier-Host).



Here you can see the left extruder in blue and the right in white. An extra structure is added (the auxiliary structure, also known as wipe / prime tower). This structure helps the extruder to get going after it has been idle for a while.

## 6.8 Print/Save the file

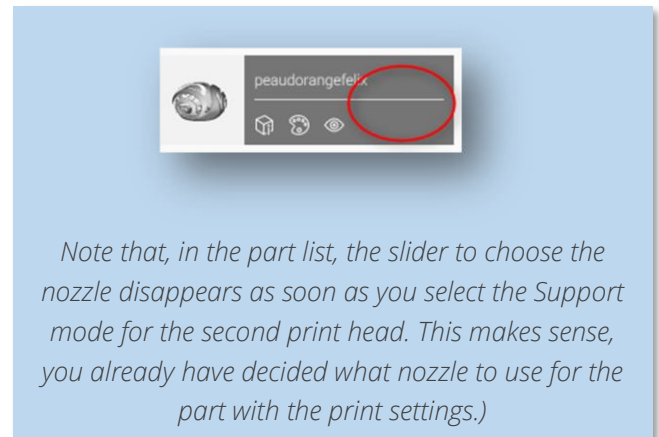
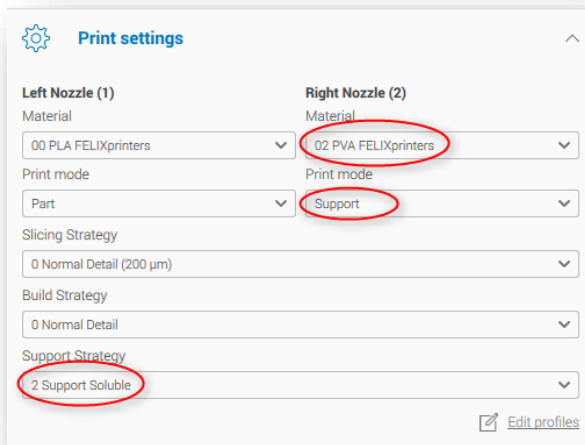
Printing and saving the job is pretty much the same as a single head print, see 5.8 and 5.9.

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.

## 7 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 FELIXbuilder 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.
  - We instruct the program to generate a support structure that is suitable for soluble material. (The support structure can be placed very tightly against the part, since it does not need to be manually peeled off later).

The rest of the printing process is identical to that of single or dual head printing (see 5 and 6).

## 8 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](http://www.felixprinters.com/support)

Email: [support@felixprinters.com](mailto:support@felixprinters.com)

Telephone: +31 (0)30 30 31 387

Address: Zeemanlaan 15, 3401MV IJsselstein, The Netherlands

Kind regards,

FELIXprinters