Tutorial

Printing with KISSlicer

26 October 2016

Revision 0

FELIX printers

1 Introduction

This tutorial describes how to get up and running to use KISSlicer to prepare print jobs for your FELIX printer.

It assumes you have access to the KISSlicer 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 single extrusion this program is free of charge, to get the full functionality and enable dual extrusion a license needs to be obtained from www.kisslicer.com.

The tutorial assumes you are going to use the latest Windows version (KISSlicer 1.5)

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

2 What can KISSlicer do?

The KISSlicer application allows you to import a 3D file and view and manipulate it so it can be prepared to be printed by the FELIX printers. It can generate print jobs for dual head prints.

3 Installing KISSlicer

3.1 Preparation

You can find and download the installation file here:

http://www.felixprinters.com/downloads/?dir=software/2_KISSlicer

The Zip file contains 32-bit and 64-bit versions for Windows and Mac.

We also bundled the profiles for the FELIXprinters inside.

3.2 Installation

Copy the entire directory that contains the version that is suitable for your operating system to a convenient location on your computer.

Start the executable main file (KISSlicer.exe) from inside the directory.
(You probably need to have administrative rights.)

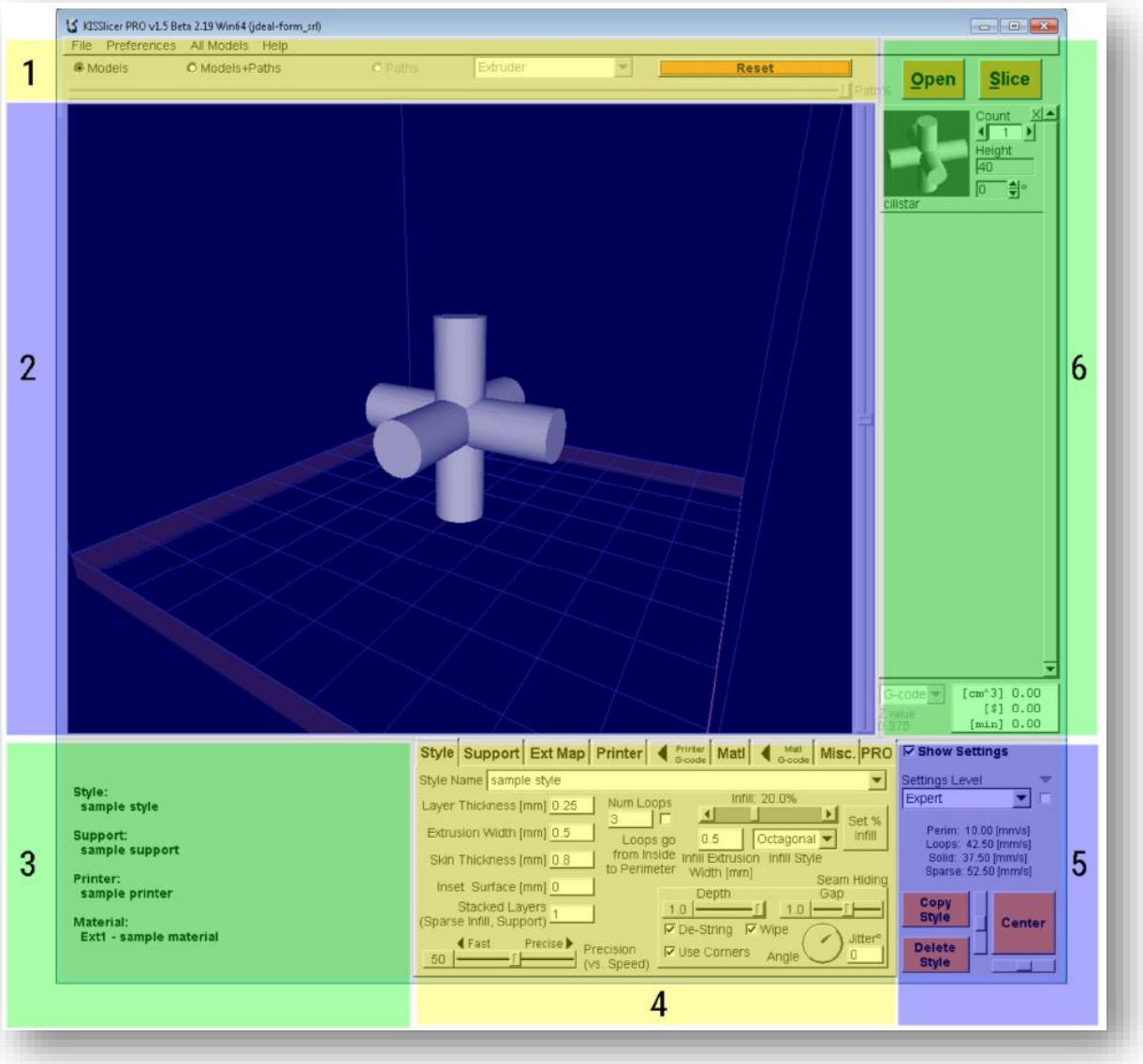
The program will now start up.

4 User interface

The main software interface will present itself.

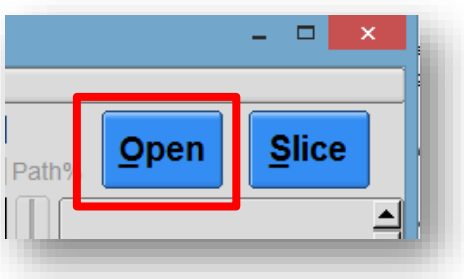
The KISSlicer interface can be subdivided into six main areas:

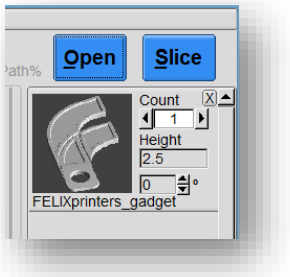
1. Upper menu, comprised of drop down menus and visualization settings
2. Visualization, where you can visualize you model and/or the paths of the resulting g-code
3. Setting recap, as a reminder of current slicing setups
4. Parameters, the place where most of the action is: choosing all the settings of the printing process
5. Setting handling: copy or delete profiles, adjust the placement of part(s) on printing bed
6. STL area, to insert and modify geometrical properties of chosen STL files



5 Preparing a model for single head printing

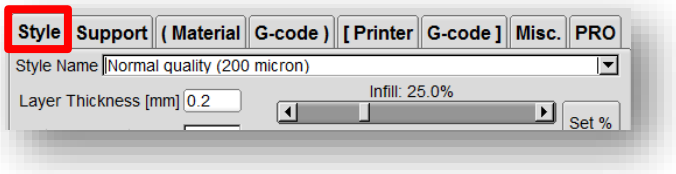
1. Open an STL file by clicking the Open button:



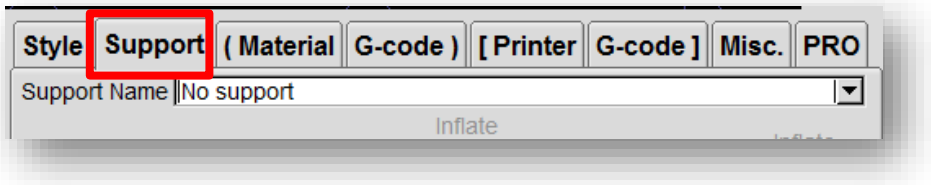


2. Configure your slicing settings.

Choose quality:

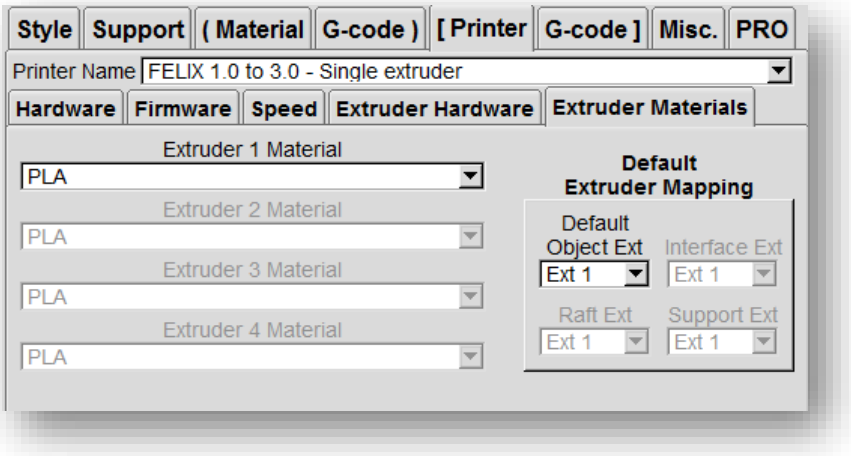


Choose support type:

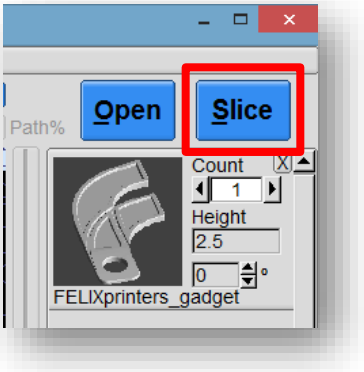


Choose print-mode (single or dual extrusion)

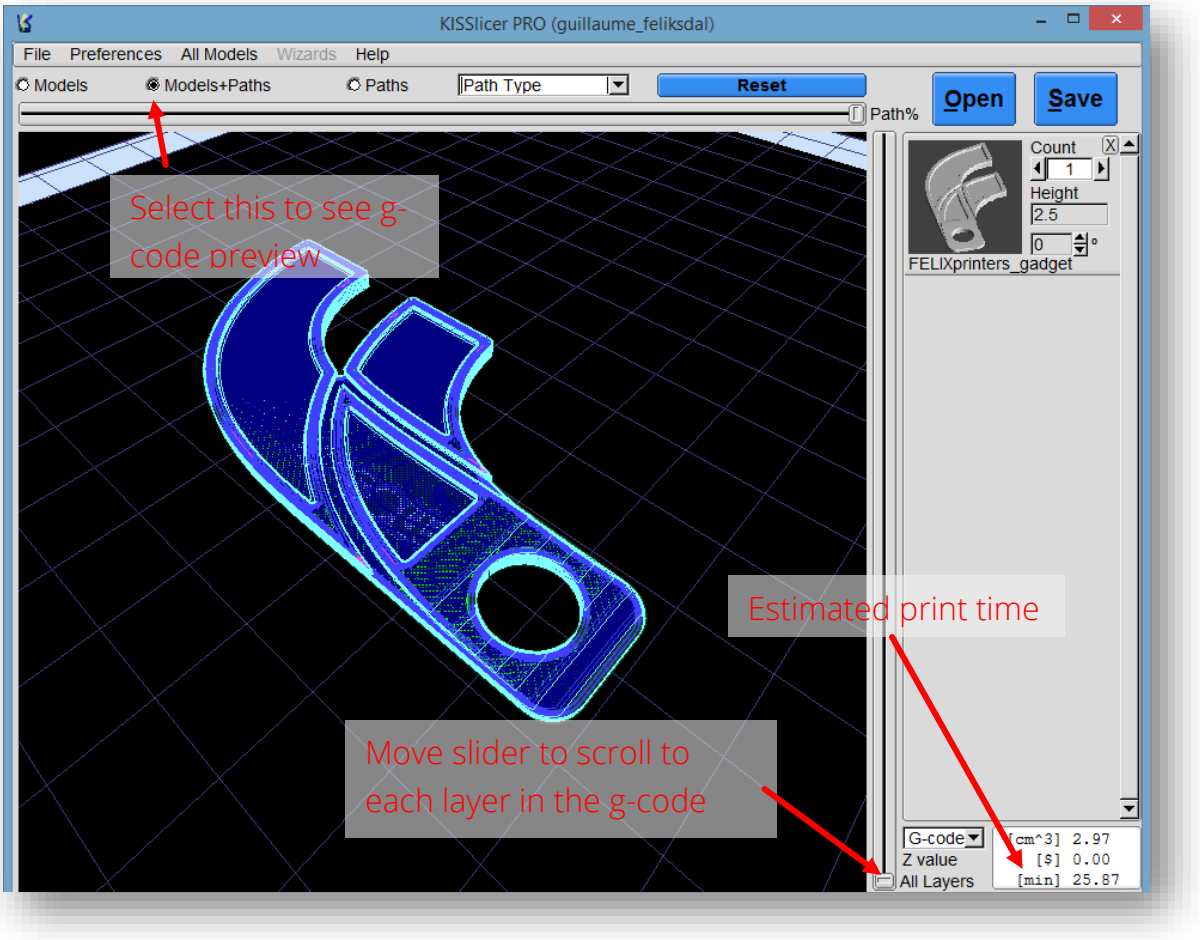
Choose what material the extruders are printing:



Press the “slice” button:



Check results:



5.1 Save the print job

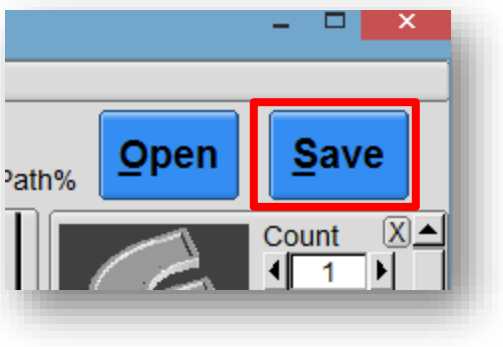
It is also possible to save a prepared print job.
The job will be saved as a so-called “g-code” file.
This file can be sent to another location with the same FELIX printer, or it can be stored for repeated print jobs or batch printing.

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.

If you are satisfied with the result, press the save button:



This saves your print file to a user-defined location, such as a folder on your computer or an SD card. The software will save a .gcode file.

Load the just saved .gcode file in a host program (like Repetier-Host) to print it on your FELIXprinter.

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.

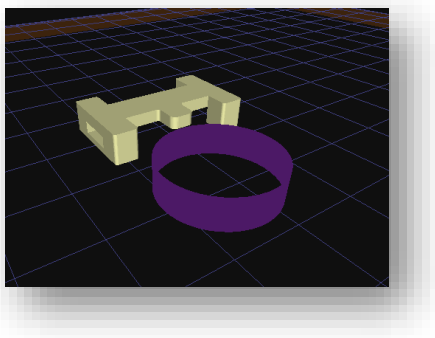
6.2 Make a dual extrusion print

These steps will demonstrate how to make a dual extrusion print with Kisslicer. Two examples will be shown.

- How to use the second extruder to print support material
- How to use the second extruder to print a selected object.

6.2.1 Support material with second extruder

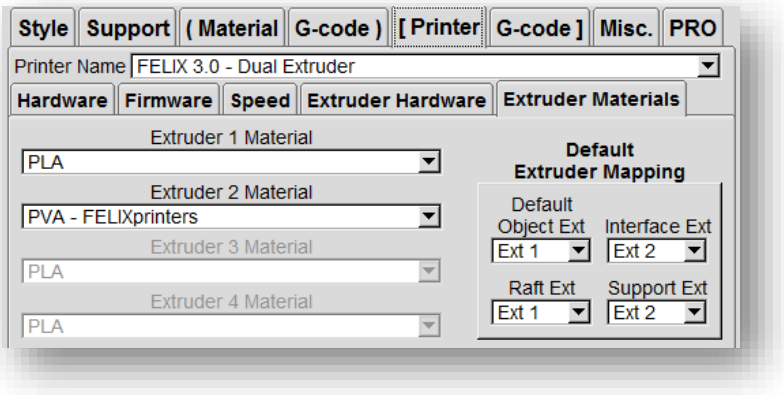
First, import an STL file which requires support material to print properly.



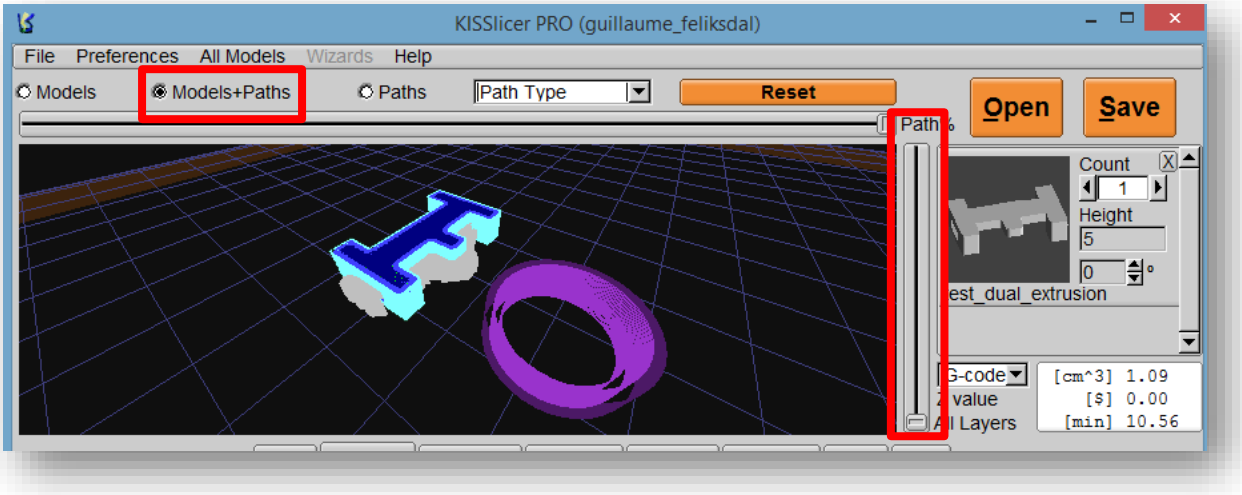
Choose a **quality**, it is recommended to start with normal quality.
Support type: Normal – Dual extrusion
Printer: Select your printer, use the Dual extruder profile.

Inside the printer tab, go to extruder material and choose the materials currently loaded in the extruders.

Also, note the default extruder mapping area. Here you can also select which structure types each extruder should print. In this example just go with the default settings.



Press slice and, after slicing, activate the "Models+Path" option and move the Z value slider to 0. This is the result:



The white lines are the support material and the other colors are the object itself. Next to the object there is a prime pillar. This functions as a wipe tower and also primes the extruder when switching to the new extruder.

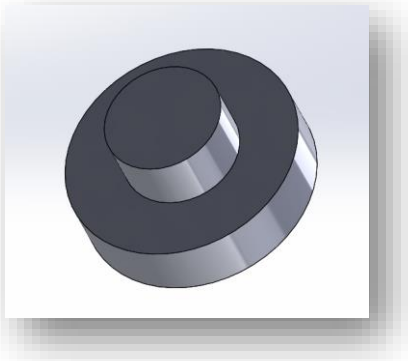
To print the generated g-code, press "Save". Then, open it in a host program (like Repetier-Host) or place it on a microSD card for directly printing it on your printer.

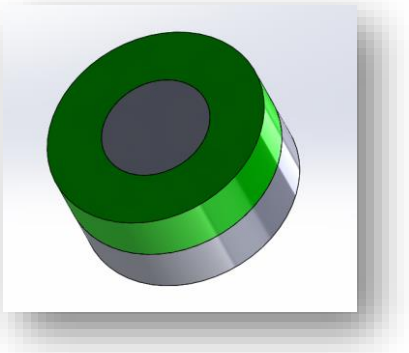
6.2.2 Use second extruder for different object or part of an object.

This short tutorial will show you how to print two objects in a single print with different extruders.

You basically need two STL files which are created in the same coordinate system and with exact needed distance with respect to each other.

For this example, two simple parts were created. See the pictures below.

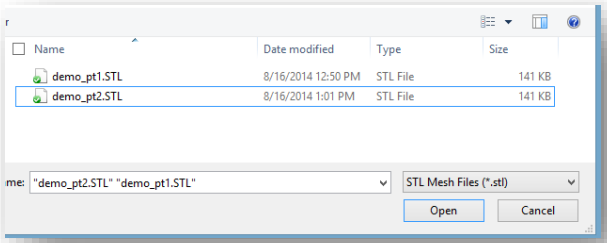


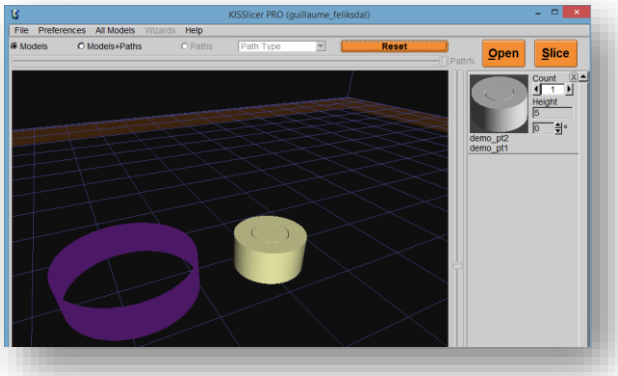


Press the “Open” button.

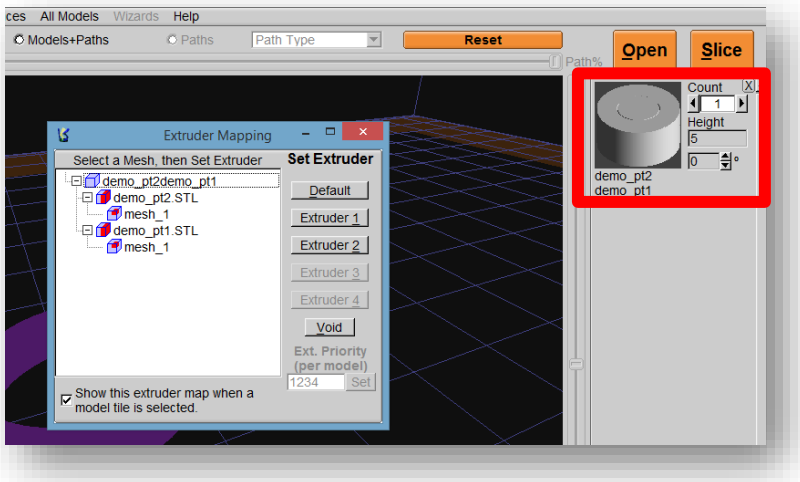
Assumed is that you configured the slicing settings as in previous section for dual extrusion.

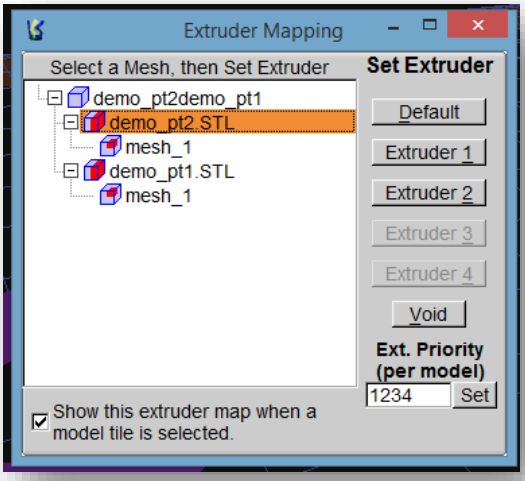
Select two STL files at once. In this case the demo parts are used.

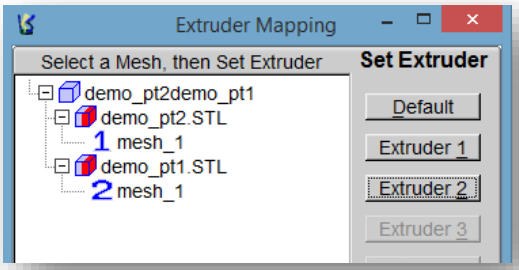




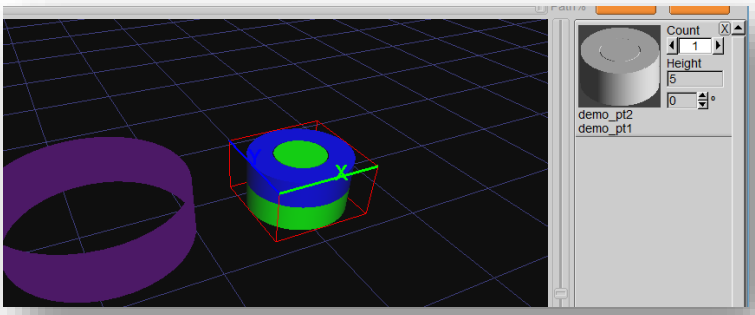
Click on the model in the right pane and a small window pops up. Here you can assign an extruder to every part.





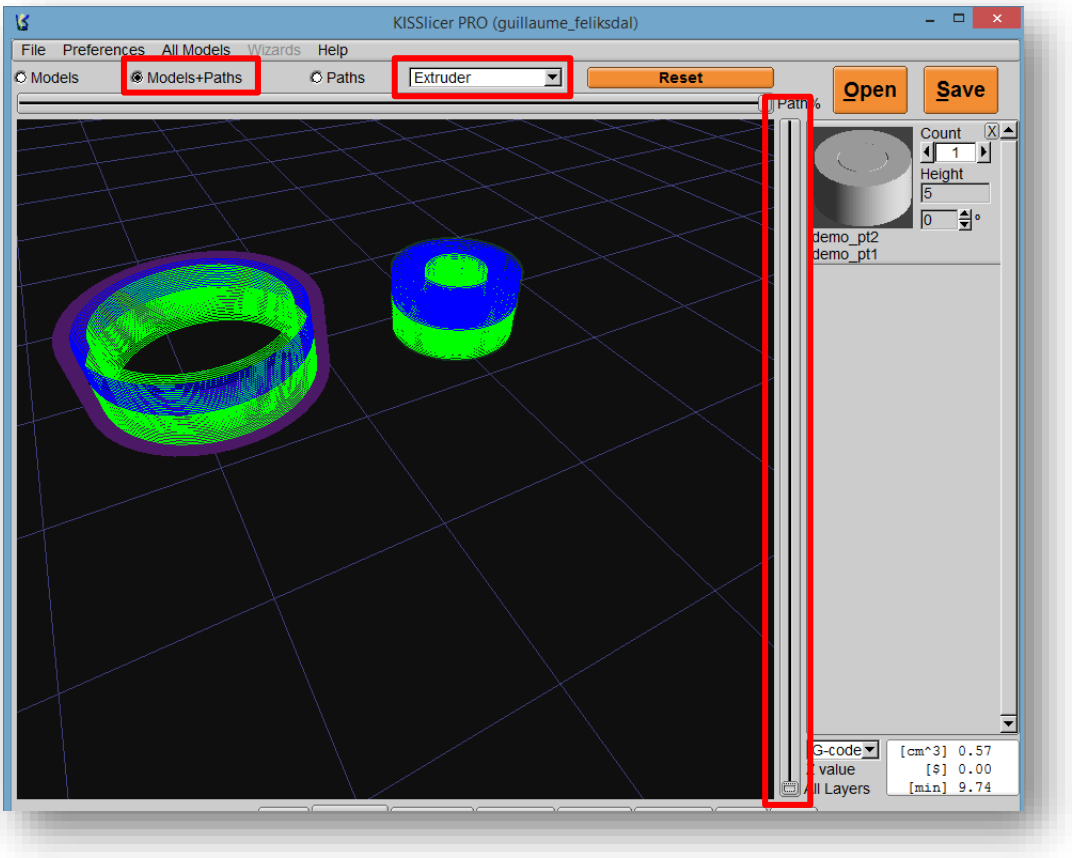


Close the window and it will look like this:



Press the “Slice” button.

Play with the red accentuated parts to see what each extruder will print.



You can now save the file and open it in repetier-host or put it straight on a microSD card to print it directly on your printer.

7 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