

KISSlicer Quick-Start Guide

Welcome to KISSlicer! This short document will guide you through the creation of your first print file. The second part of the document contains a concise reference guide to the parameters and some general tips to better exploit the printer capabilities. For an in-depth description please refer to the pdf manual enclosed with the download.

Download and install KISSlicer

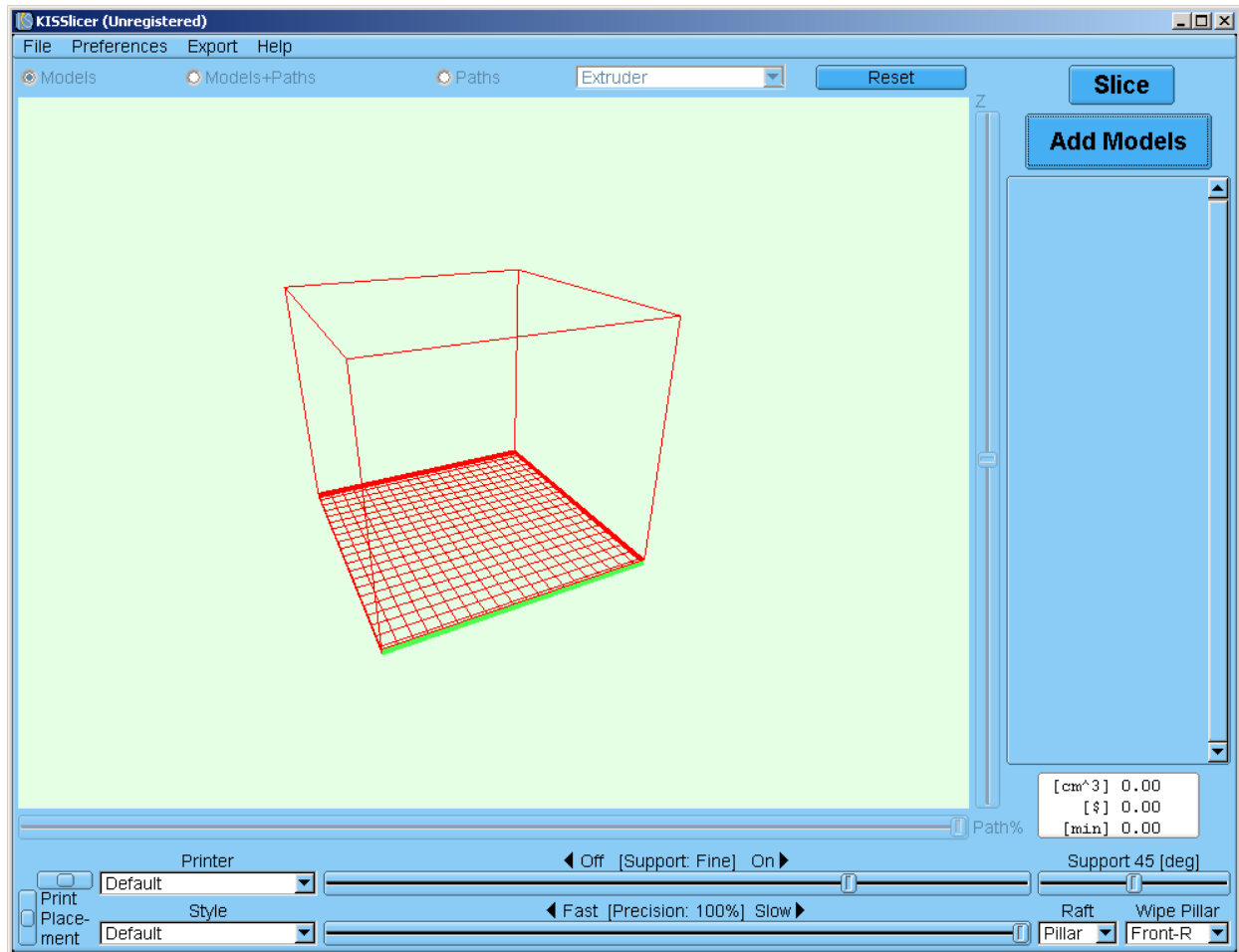
Go to the download page: <http://www.kisslicer.com/download.html>

Select the version matching your Operating System (at the moment Windows and Linux versions are available) and save it locally. You can unzip the executable directly into the desired folder. For example, under Windows you can use “C:\KISSlicer\” or “C:\Program Files\KISSlicer”. No installation is required. At the first run the executable will create support files in the same folder.

If you downloaded the KISSlicerQuickStart.pdf file and/or Glasswalker’s sample settings, simply place them all into the same directory as the executable. Under Windows you can place a shortcut to ‘KISSlicer.exe’ in your ‘Send To’ folder, allowing you to right-click on any model(s) in the Windows File Explorer, and [Send To]->KISSlicer.

Your first print

Run the executable (KISSlicer.exe) and the following window will open.



First of all go to preferences->advanced settings to set the printer, material and print style. This first run only some setting will be (in case) modified, for a parameters summary check the end of the document, and for an in depth description refer to the manual.

Settings

Style

KISSlicer Settings

Style | Material | Printer | ...Speed | ...Extruders | ...G-Code | KISSlicer

Name: Default Delete Style

Skin Thickness [mm]: 0.8 Extrusion Width [mm]: 0.5

Number of Loops: 3 Infill Extrusion Width [mm]: 0.5

Stacked Layers: 1 Layer Thickness [mm]: 0.25

Infill: 20.0% De-String

Save Settings Close Without Saving Settings

Under style you can save as many profiles as you like. For the first print you can leave the parameters as they are.

Material

KISSlicer Settings

Style | Material | Printer | ...Speed | ...Extruders | ...G-Code | KISSlicer

Name: ABS Delete Material

Calculate Fan: ON for Loops Color:

0.00549771 Extruder RPM: Min 3 Max 75

Warm-Up Factor Destring [mm]: Suck 1 Prime 1

Calculate Min Layer Time [s]: 10 Wipe [mm]: 10

2 Fiber Dia [mm]: 3 Flow Tweak: 1

\$ / cm³ Temp [C]: 250 Keep-Warm: 160 First Layer: 255

Save Settings Close Without Saving Settings

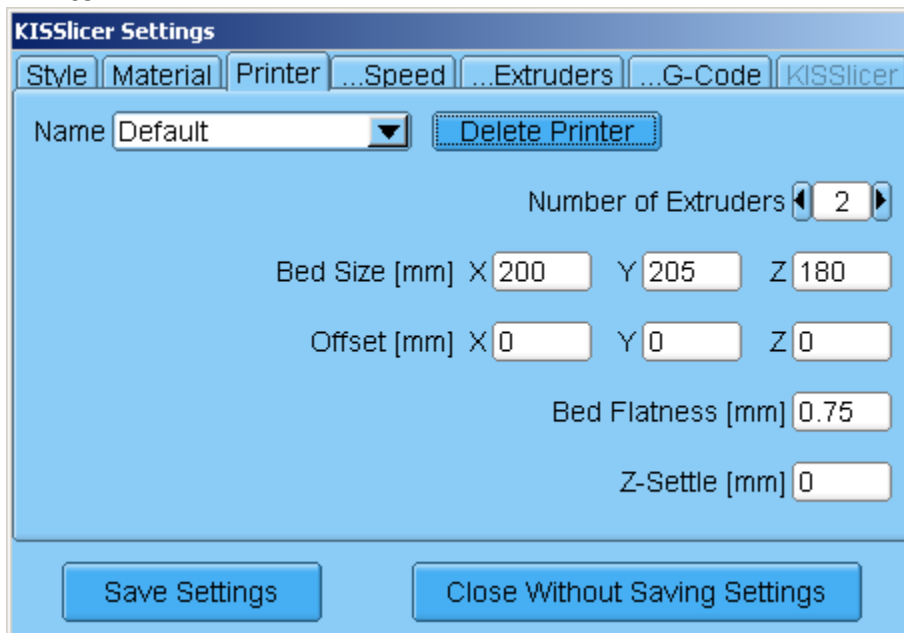
You can select the material in the extruder.

ABS and PLA default profiles are provided. You can select if the Fan should be on all the time, only when printing the perimeter or never. You can set printing temperature, min/max printing speeds, destring settings and modify the flowrate to adapt to your material. You can also set a print cost and a warm up parameter to estimate print time and cost.

Select your material from one of the default options, and for the moment keep the default settings. In case you already have a working temperature profile for your material update it

accordingly.

Printer



The image shows the 'Printer' tab in the KISSlicer Settings window. It features a dropdown menu for 'Name' set to 'Default', a 'Delete Printer' button, and a 'Number of Extruders' spinner set to 2. Below these are input fields for 'Bed Size [mm]' (X: 200, Y: 205, Z: 180), 'Offset [mm]' (X: 0, Y: 0, Z: 0), 'Bed Flatness [mm]' (0.75), and 'Z-Settle [mm]' (0). At the bottom are 'Save Settings' and 'Close Without Saving Settings' buttons.

Parameter	Value
Name	Default
Number of Extruders	2
Bed Size [mm] X	200
Bed Size [mm] Y	205
Bed Size [mm] Z	180
Offset [mm] X	0
Offset [mm] Y	0
Offset [mm] Z	0
Bed Flatness [mm]	0.75
Z-Settle [mm]	0

Here you can rename the printer profile, create a new one and set some of the parameters.

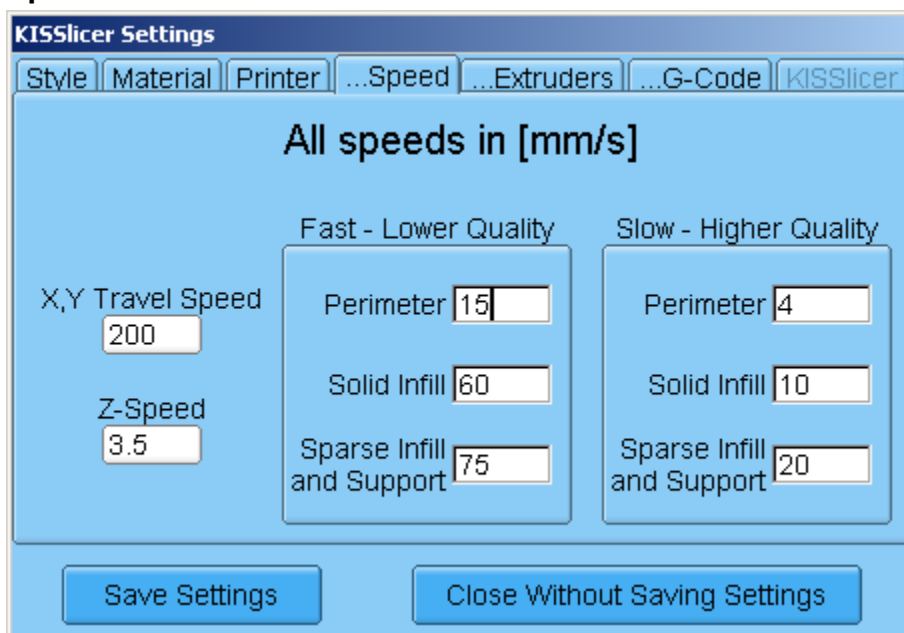
For rapman dual head set Bed Size to 120-130mm

For rapman firmware 4.2.x set Offset Y = 20mm

Bed flatness will drive the pillar height, while Z-settle will move the bed a little lower and then will raise it back, useful for some anti wobble designs.

For now leave the other parameters as they are. You can refer to the manual to modify the others parameters.

Speed



The image shows the 'Speed' tab in the KISSlicer Settings window. It is titled 'All speeds in [mm/s]'. It contains two main sections: 'Fast - Lower Quality' and 'Slow - Higher Quality'. The 'Fast' section has input fields for 'Perimeter' (15), 'Solid Infill' (60), and 'Sparse Infill and Support' (75). The 'Slow' section has input fields for 'Perimeter' (4), 'Solid Infill' (10), and 'Sparse Infill and Support' (20). To the left of these sections are input fields for 'X,Y Travel Speed' (200) and 'Z-Speed' (3.5). At the bottom are 'Save Settings' and 'Close Without Saving Settings' buttons.

Parameter	Fast - Lower Quality	Slow - Higher Quality
Perimeter	15	4
Solid Infill	60	10
Sparse Infill and Support	75	20

X,Y Travel Speed: 200
Z-Speed: 3.5

This will define a range of speed parameters, from the main window you can set the printer

speed between the two extremes here shown. Leave them as they are for the moment.

Extruder

The screenshot shows the 'KISSlicer Settings' dialog box with the 'Extruders' tab selected. The dialog has a blue header with tabs for 'Style', 'Material', 'Printer', '...Speed', '...Extruders', '...G-Code', and 'KISSlicer'. The main area is light blue and contains settings for three extruders. Each extruder has a '[mm/rev]' speed input (all set to 1.25) and a 'Material' dropdown menu (Extruder 1 is ABS, Extruder 2 is PLA, Extruder 3 is ABS). Below these are four dropdown menus for assigning extruders to different print parts: 'Object Extruder', 'Raft Extruder', 'Support-Obj Interface Ext', and 'Support Main Ext', all currently set to 'Ext 1'. At the bottom are two buttons: 'Save Settings' and 'Close Without Saving Settings'.

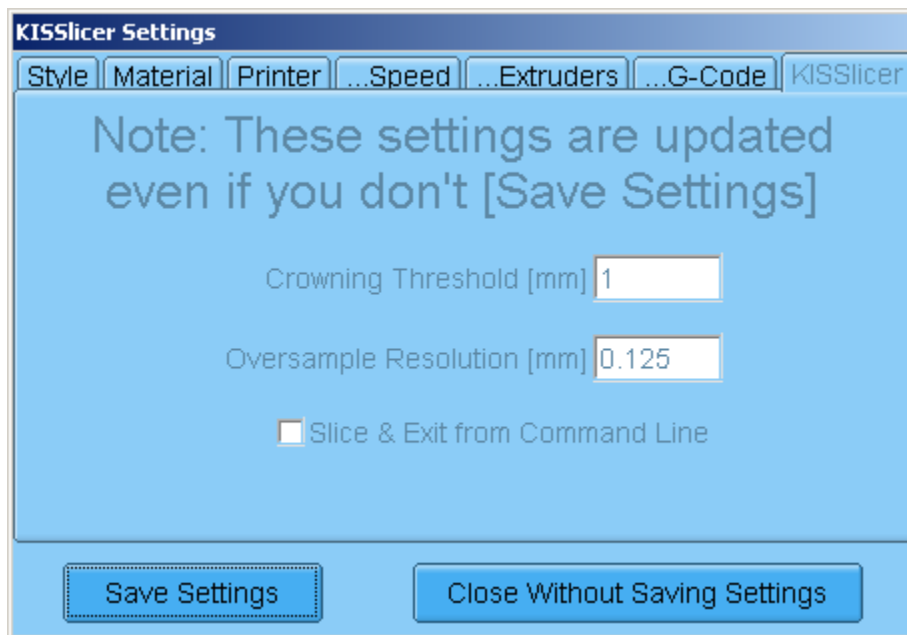
You can assign the extruder material to your available heads.

With free version you can set one extruder for all the print, while the the pro version you can assign the head to part, raft, support and part/support interface. Check at the end of the guide for an example

G-Code

The screenshot shows the 'KISSlicer Settings' dialog box with the 'G-Code' tab selected. The dialog has a blue header with tabs for 'Style', 'Material', 'Printer', '...Speed', '...Extruders', '...G-Code', and 'KISSlicer'. The main area is light blue and contains settings for G-code generation. It includes a 'G-Code Prefix' text area, a 'G-Code Postfix' text area, and a 'G-Code Extruder Change' text area. To the right, there is a 'Firmware Type' dropdown menu set to 'Bits From Bytes' and a 'File Extension' input field containing 'bfb'. At the bottom are two buttons: 'Save Settings' and 'Close Without Saving Settings'.

Kisslicer settings



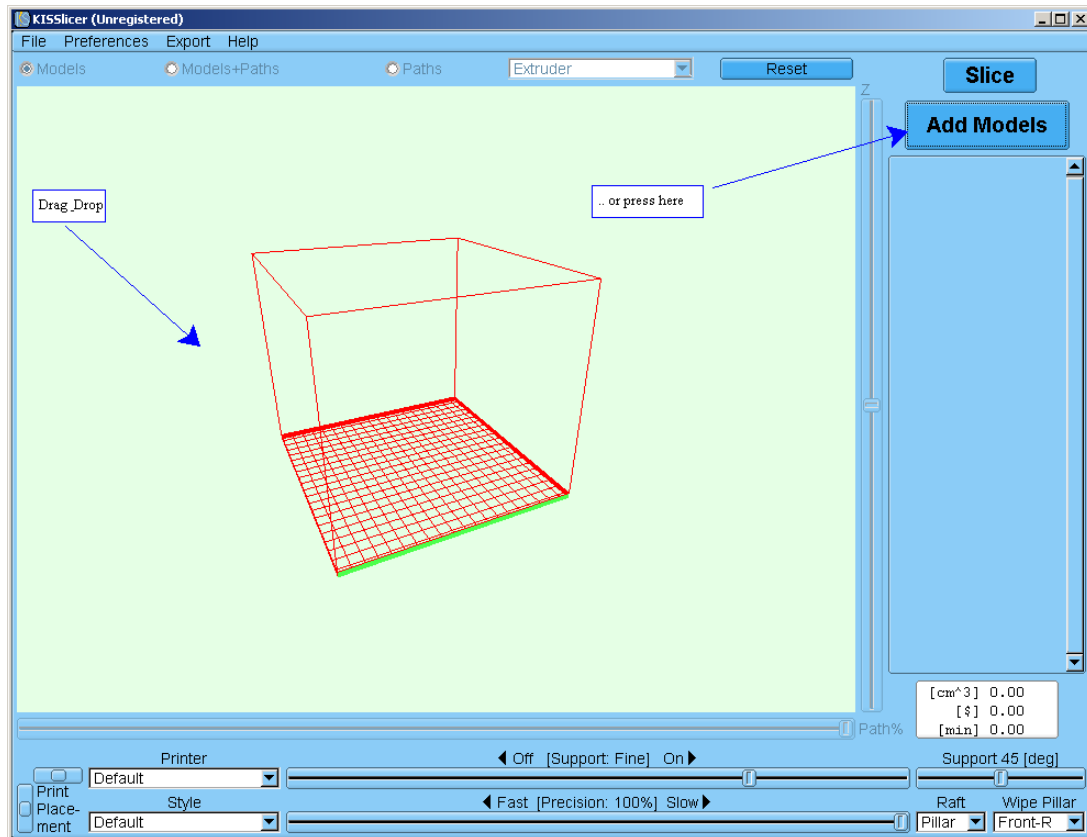
The settings allow so fine tune the crowning threshold. This is a pro feature, and for the free version the values are not modifiable. For sure it is a parameter to be modified after getting acquainted with the printer, so for the first print leave it as it is.

Loading a STL file

A new part can be loaded either by pressing the Add model button, or by drag and drop over the 3D view area.

With pro version you can upload multiple stl models.

The stl file must be correctly oriented before loading it to kisslicer. Free Netfabb is a nice option to correctly orient the part.



View the part

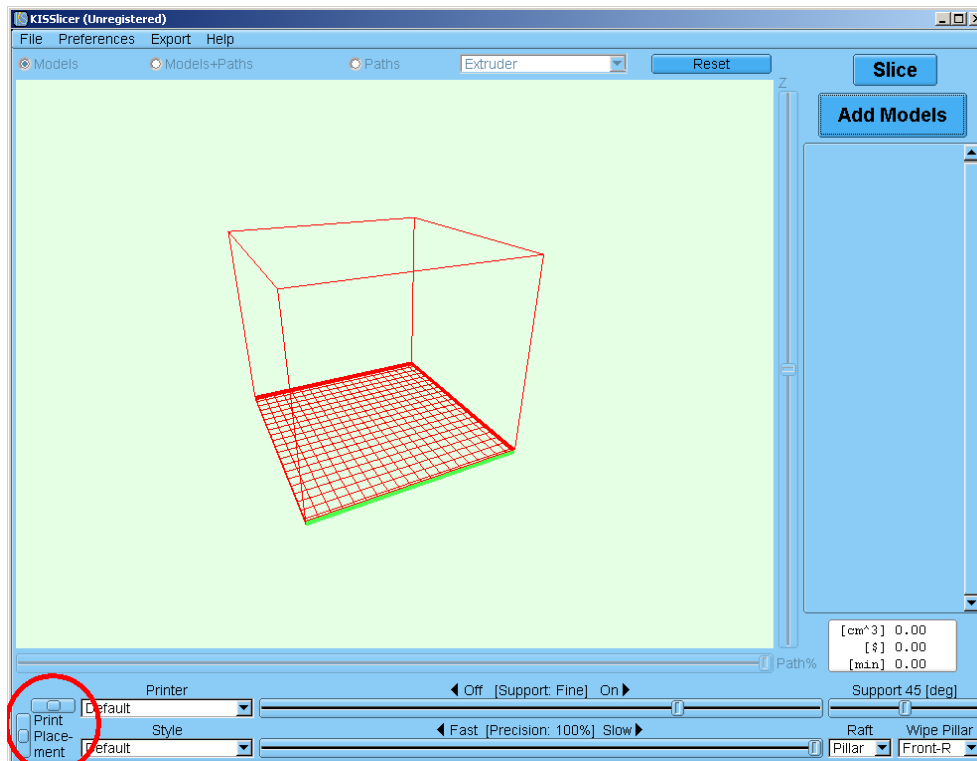
The 3D view can be modified in the following way:

Left mouse	pan view (translate view)
Mouse wheel, or right+left mouse button	zoom in/out
Right or center mouse button	rotate view

The reset button will restore the view.

Positioning the part

A part can be positioned in the print area with the use of the two sliders on the bottom left. This is a coarse positioning, to avoid damaged print area, for instance.



Setting the printer parameters

From the main view the printer parameters can be adjusted and tailored to suit the printing needs. Some of the options use the profiles or value ranges already set in the preferences menu.

Printer profile

From the printer select menu a different printer profile can be selected.

The printer settings drive the number of extruders, the bed size and offset, bed flatness (to be used in the raft calculation) and z-settle.

Style

The style selector allow to choose between preset parameters groups, mainly infill%, use of destring, loops and stacked layers, extrusion width and layer thickness. Choose a suitable profile.

Support

On the main window it is possible to choose the support characteristics. Mainly if the support is to be switched on or off. In case of support on it can be chosen a varying finesse of the structure. Moreover the support angle decides the overhang limit before using the support structure. 45 degrees is a general value, but it can be tailored depending on the material and model design.

Speed

The precision slider will allow to set the speed between the speed extreme values available in settings->speed. A faster print is generally less precise, and could cause overheating. A slower print will take longer.

Raft

3 raft stiles are available:

- **OFF** creates a raftless print
- **Grid** creates a “traditional” stile raft
- **Pillar** creates a first layer of small pillars, and then creates the traditional grid. The height of the pillars is calculated with the bed flatness parameters. The higher the value the bigger the pillars. This raft style is more forgiving when the bed is not flat anymore

Wipe pillar

The wipe pillar is a cylinder printed before every layer to clean the extruder tip. It reduces hairing and it is extremely important in dual head prints. Can be switched off, or the position with respect to the main part can be chosen.

Creating (slicing) a file

After having reviewed the model, and selected the printer parameters it is time to create the print file. Press the SLICE button and let the magic begin.

Saving the generated file

Immediately after the file is generated you will be prompted to save it. In case you press cancel you can export again the file pressing once more the slice button. The path is already calculated so only the file export will take place.

Reviewing the print path

The radio buttons on the top part of the main window will now be active and it is possible to review the print path.

Registering

/* better at the beginning of the document..

Parameters quick reference

This section will shortly describe the meaning of the single parameters.

Menu Settings->Style:

Skin Thickness [mm]: *****

Number of loops: number of loops of filament for the edge of the model

Extrusion width [mm]: cross section of the extruded material for external model portion

Infill extrusion width [mm]: cross section of the filament to be used for the infill

Layer thickness [mm]: Thickness (along Z) of the single print layer

Infill [%]: a solid model can be printed hollow inside, or with a partial fill, with an honeycomb like **Structure:** It reduces print time and material, at the price of a less sturdy model.

Destring: activates the destring function: it will “suck” the filament when moving without printing to avoid that the material in the hot head goes out without control creating filaments around (oozing). The material will be pushed forth before restarting the print. The destring amount will

be set in the material menu for each material.

Stacked layers (Pro only): Sets the number of external layers to be printed before printing a cumulative infill layer. For instance stacked layers = 2 means the sparse infill will be printed every second layer, with the required material do fill the double thickness gap. It allows to print the exterior with finer layers, while speeding things up while printing the infill.

Menu Settings->Material:

Fan: will switch on/off the fan during the print, or only when printing the external layers (loops)

Color: sets the color of the material for print review

Extruder RPM, Min/Max: sets a range of print speed. When using the speed slider the program will use the two values as extremes

Destring, Suck/Prime: sets the mm of filament to be sucked and pushed forward (prime) if destring option is on.

Min Layer Time [s]: if a layer prints in less than this min value the printer will waste some time to reach this minimum value. Useful for reducing overheating in slender/thin parts

Wipe [mm]: after a warm up the printer will push (prime) this amount of filament to ensure the printer is ready.

Fiber Dia [mm]: Sets the diameter of the filament. Use a caliper to measure it.

Flow tweak: can adjust the amount of material being pushed, to tailor the print. This is material and supplier dependent.

Temperature: sets the temperature of the material for the main part

Keep Warm: when printing with multiple materials sets the idle temperature

First layer: sets the temperature of the first layer, usually a little bit higher to better stick to the bed

Warm up and \$/CM3: You can set a warm up time parameter. It will allow a more accurate print time estimate. When using multiple head printing the warm up time could be a big player. The cost per volume will allow to estimate the material cost during printing.

Menu Settings->Printer:

Number of extruders: will allow setting the number of extruders available. The material, and the proprieties of the extruders are to be set in the "extruders" tab.

Bed size (x,y,z): sets your printable area

Offset (x,y,z): you can tailor the offsets of the printer, in case the firmware zero is shifted

Bed flatness: This parameter will be used with pillar raft to tailor the pillar dimensions. A higher value will bring bigger and fatter pillars.

Z-settle: In case of anti wobble solutions it could be useful to have the z stepper moving over the expected position and coming back to the desired z.

\$/hour: Cost per hour of the printer, to estimate a total printing cost.

Menu Settings->Speed:

X,Y travel speed: speed along x and y while not printing

Z-speed: speed along z

Perimeter speed: minimum/maximum speed range for the external iso line (perimeter of the print)

Solid infill and support speed: minimum/maximum speed range for the support and for the inner iso lines (with the exception of the perimeter)

Sparse infill speed: minimum/maximum speed range for the rest of the infill

Menu Settings->Extruders:

Extruder [n] mm/rev: This value come from the geometry of the extruder (an M8 is typically 1.25 mm/revolution)

Extruder [n] Material: Select a material for the extruder from the ones you have created in the Settings->Material menu

Object Extruder: Will select which of the available extruders will be used for the object printing. The free version will print everything with this material/extruder. Pro version allows the following options

Raft Extruder: extruder to be used for the raft

Support-Object interface Extruder: The last layers of the support can be printed with a different extruder with respect to the main support. You can then print almost all the support with the main extruder, and only the interface layers with an alternative extruder. In some cases this will allow to speed up the print reducing extruder switching and extruder heating wai time

Support main Extruder: This is the most of the support material, see previous description.

Menu Settings->G-code:

G-code prefix: *****

G-code postfix: *****

G-code extruder change: *****

Firmware type: Select one of the possible firmwares

5D Temp G-code: *****

File extension: selects the file extension to be appended to the generated file

Menu Settings->Kisslicer:

Pro only, be very careful with the following parameters!!

Crowning Threshold [mm]: *****

Oversample Resolution [mm]: *****

Slice & Exit from Command Line: *****

Print suggestions - tips and tricks

/* design tips

/* model orientation

/* destring tweak

/* ...

Print suggestions - Pro version

/* use of dual head - model tweaking

/* use of stacked layer