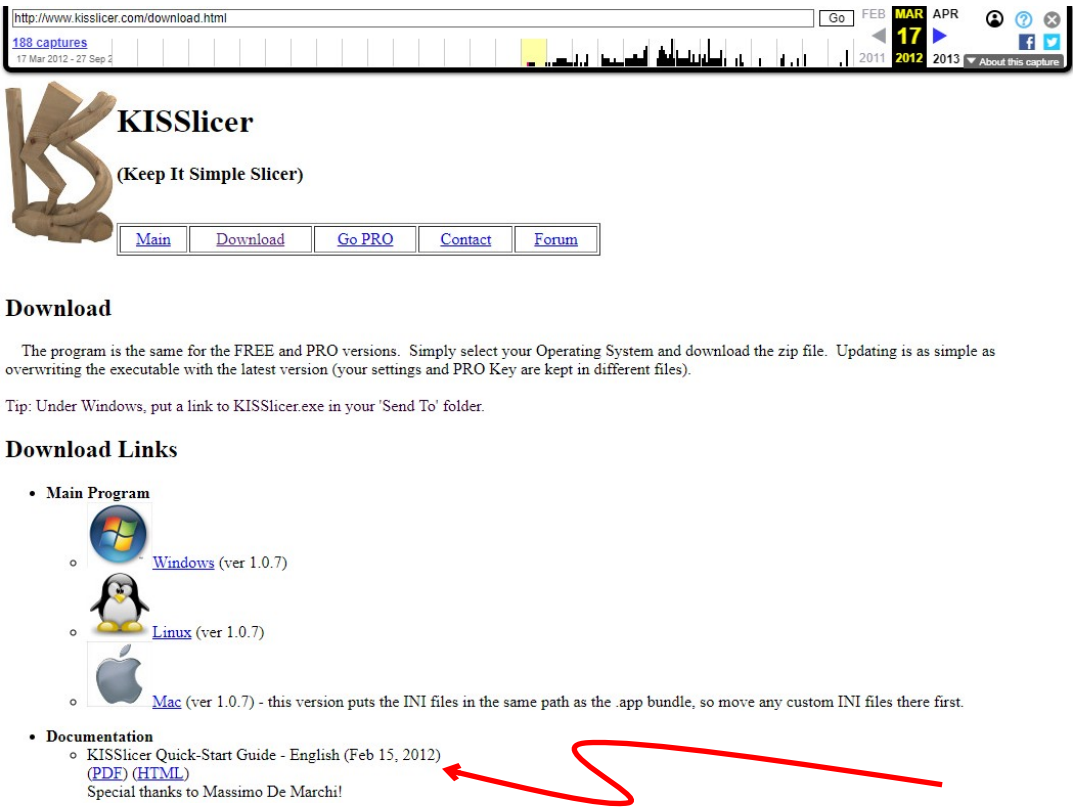
DECLARATION OF JONATHAN DUMMER

I, Jonathan Dummer of Redding, California, being of sound mind, declare the following to be true to the best of my knowledge, information and recollection:

1. I am the creator of KISSlicer, a powerful software tool that facilitates the conversion of 3D object models to specific path instructions for 3D printing by slicing STL files into 3D printer-ready G-code files.

2. Since January 2012, the KISSlicer software has been available in both free and pro versions at the www.kisslicer.com website, which became popular to many in the 3D printing field by early 2012. *See* Appendix A, p. 1. The free version was and still is freely available and does not require any login credentials or payment to access. Additionally, the website also provided free access for users to also download the "KISSlicer Quick-Start Guide" (Appendix B), which was publicly accessible by interested users no later than February 2012—a fact that is corroborated by the archived version of the www.kisslicer.com website:



See Appendix A, p. 1 ("February 15, 2012"). I have personal knowledge that, throughout February 2012, members of the public interested in 3D printing were able to find, access, and download the KISSlicer software and Quick-Start Guide from www.kisslicer.com using publicly

available search engines, blog posts, and through word of mouth in the 3D printing community. I recall that, during February 2012, the kisslicer.com forum included many active members discussing topics relevant to the KISSlicer software, and the KISSlicer Quick-Start Guide was a helpful tool for many of those new members who also downloaded the KISSlicer software.

3. Appendix A is a true and correct copy of an archive.org capture of the Download page of www.kisslicer.com (http://www.kisslicer.com/download.html) as it appeared in February and March of 2012 (and for some time thereafter). This page included links for downloading the KISSlicer Quick-Start Guide (Appendix B). Appendix B is a true and correct copy of the KISSlicer Quick-Start Guide as published on www.kisslicer.com on February 15, 2012. This Quick-Start Guide (Appendix B) was publicly accessed by members of the 3D printing community in February 2012 and was available in both PDF and HTML format. No password or access credentials were required to download this Quick-Start Guide.

4. I declare under penalty of perjury that the foregoing is true and correct to the best of my knowledge. I hereby declare that all statements made herein of my own knowledge are true and that all statements made on information and belief are believed to be true; and further that these statements were made the knowledge that willful false statements and the like so made are punishable by fine or imprisonment, or both, under Section 1001 of Title 18 of the United States Code.

By: 

Jonathan Dummer

Date: December 4, 2024

Appendix A

KISSlicer

(Keep It Simple Slicer)

Download

The program is the same for the FREE and PRO versions. Simply select your Operating System and download the zip file. Updating is as simple as overwriting the executable with the latest version (your settings and PRO Key are kept in different files).

Tip: Under Windows, put a link to KISSlicer.exe in your 'Send To' folder.

Download Links

- **Main Program**
  -  Windows (ver 1.0.7)
  -  Linux (ver 1.0.7)
  -  Mac (ver 1.0.7) - this version puts the INI files in the same path as the .app bundle, so move any custom INI files there first.
- **Documentation**
  - KISSlicer Quick-Start Guide - English (Feb 15, 2012)
    (PDF) (HTML)
    Special thanks to Massimo De Marchi!
    Place in the same directory as the main program for [Help]->[Quickstart Guide]
    [Work In Progress]
  - KISSlicer Anleitung V1.0a - German Manual
    (PDF)
    Special thanks to Jörg Schnyder!
- **Sample Setting Files**
  - Glasswalker's Settings (Feb 15, 2012)
    Unzip into the same directory as the main program.
    [Work In Progress]
- **KISSlicer 3D Logo**

- Logo

KISSlicer FREE License

- Has NO time limit
- Has NO model size limit
- MAY be used for commercial applications
- MAY be shared freely
- MUST NOT be reverse engineered

Changelog

- 2012 March 9 (v1.0.7)
  - Support
    - if using a raft, use dense bottom support layer in cross direction
    - disable stacked support on base layers (better adhesion)
    - More unified support (no random reversals) for easier removal
    - Sheathed Support (perimeter around the support base, strengthens and adds material for upper support to bond)
  - Shows filename when extension check fails
  - Removed clamp on flow tweak
  - Improved timing estimate (still doesn't handle acceleration)
  - Loops always go CCW now
  - Solid / Loop overlap is now a setting
  - All adaptive layers (support and infill) use the fine setting
  - Use a "sleep(1)" instead of yield to hopefully allow other apps to run simultaneously
  - No exit while slicing or exporting G-code
  - Allow transparent view before paths are generated for model error checking
  - Added "Raft Inflation" to expand the raft area
  - KISSlicer advanced settings tab is now available in FREE mode
  - on Mac
    - Put the configuration INI files in the same directory as the .app bundle
    - Handle "-psn_*" on command line
    - Added Mac icon (Thanks, setar!!)

- 2012 February 25 (v1.0.6)
  - Breakdown of estimated extrusion time by extruder and path type (in G-code)
  - Can optionally extrude loops from inside to outside
  - 1st layer temperature only happens after the raft if the material is different from the raft
  - Changed the name "Wipe Pillar" to "Prime Pillar"
  - Changed the name "Bed Flatness" to "Bed Roughness".
  - Changed the math for solid infill
  -
    - to remove overlap with loops
    - to account for material in short connecting pieces
    - NOTE: you will probably need to change your "Flow Tweak" in material settings (close to 1.0)
  - Allows a 0mm wipe (will still trigger destring on 5D firmware)
  - Added a "Scale by X" right-click menu option to each object tab
  - Added an "All Models Menu" (to scale, revert, or delete)
  - Added an option to *NOT* rotate the model on load or packing
  - Fixed grid raft thickness bug
  - On Drag-n-Drop to 3D pane, can now handle filenames with spaces in them under Linux & Mac (was changing space to %20)

- 2012 February 15 (v1.0.5)
  - Can handle unicode characters in filenames and paths
  - Opening STL files can now sort by date
  - Fixed Z-speed not saving
  - Keep original filename when changing save directories
  - Sped up the support pass
  - Added more descriptive message if STL fails to load
  - Updated 'Close Settings Window' button text
  - Report extra settings and build info in output file
  - Clamp some user settings to resonable values
  - Remove very short paths (< 2 * extruder width)
  - 5D firmware:
    - Use M104 for cool or keep-warm temp
    - No default purge code after extruder change
    - <EXT> replacement uses numbering 0,1,2
    - Clamp wipe to >= 0.01mm (for destring...temporary fix)

- 2012 February 4 (v1.0.4)
  - Default prime after extruder change now works under 5D firmware
  - Used new time and cost estimates in the output G-code file
  - first layer temp now occurs on the first layer after the raft as well
  - "T0" only happens on 5D firmware if > 1 extruder
  - Wipe Pillar now works under FREE version as well as PRO
  - G-code includes (prefix, postfix, extruder change) now have an 8KB limit (instead of 512 bytes)

- 2012 January 31 (v1.0.3)
  - 5D firmware can select M104 or M109 for setting temperature
  - Improved settings handling (especially deletion and creation)
  - Fixed flow-rate errors on 1st layer (especially raftless)
  - Support extruded at the same speed as solid paths (not sparse-infill)
  - Remember window size & location
  - Added tooltips to main GUI
  - Infill density slider buttons now advance by 1
  - Added machine cost per hour (for cost estimates)
  - Fatten the base layers of the wipe pillar
  - Added Mac support

- 2012 January 28 (v1.0.2) - unreleased

- 2012 January 23 (v1.0.1)
  - Added website to [About] box.
  - Fixed Drag-N-Drop files under Linux.

- 2012 January 20 (v1.0.0)
  - Initial Release

Appendix B

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

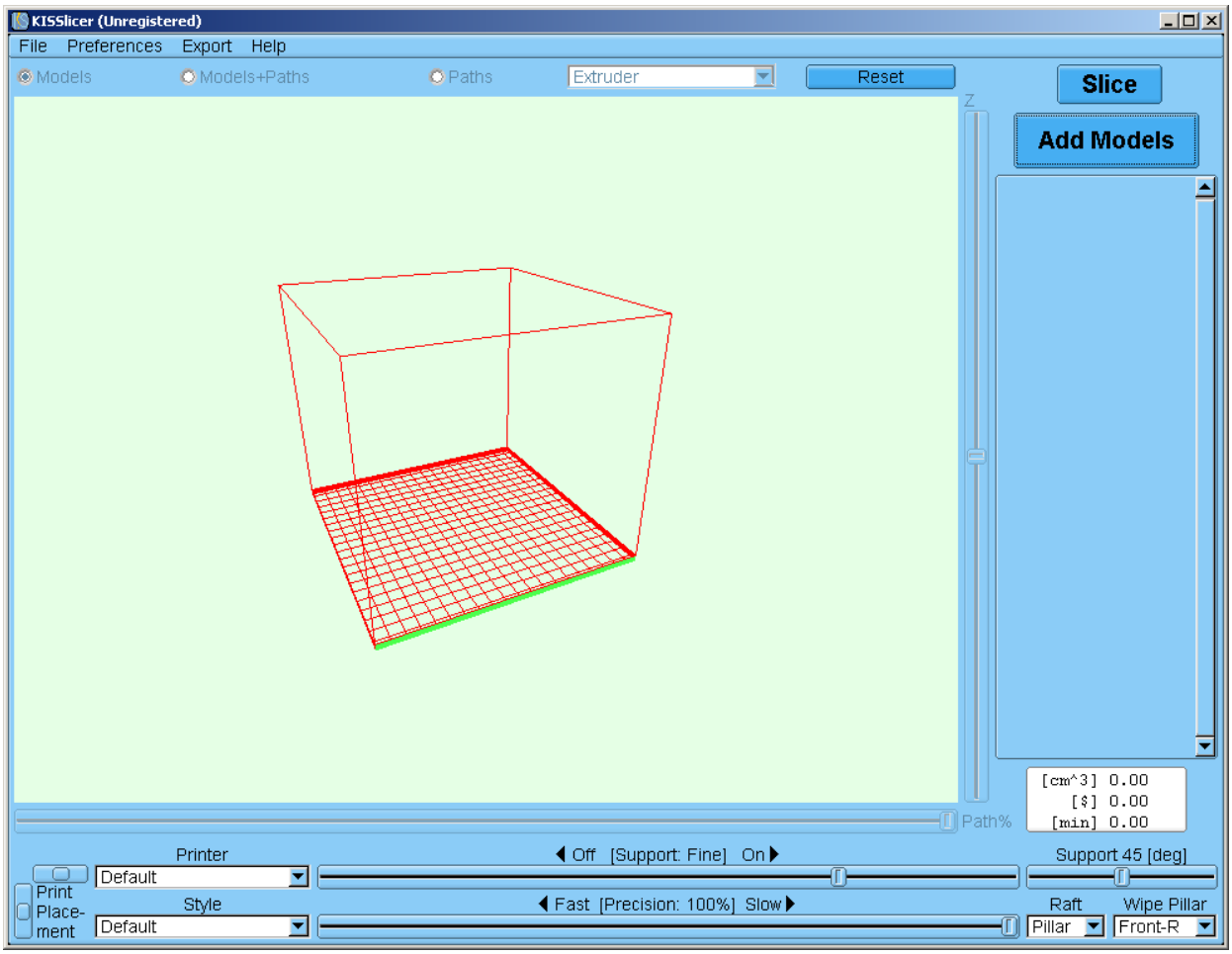
Go to the download page: http://www.kisslicer.com/download.html

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

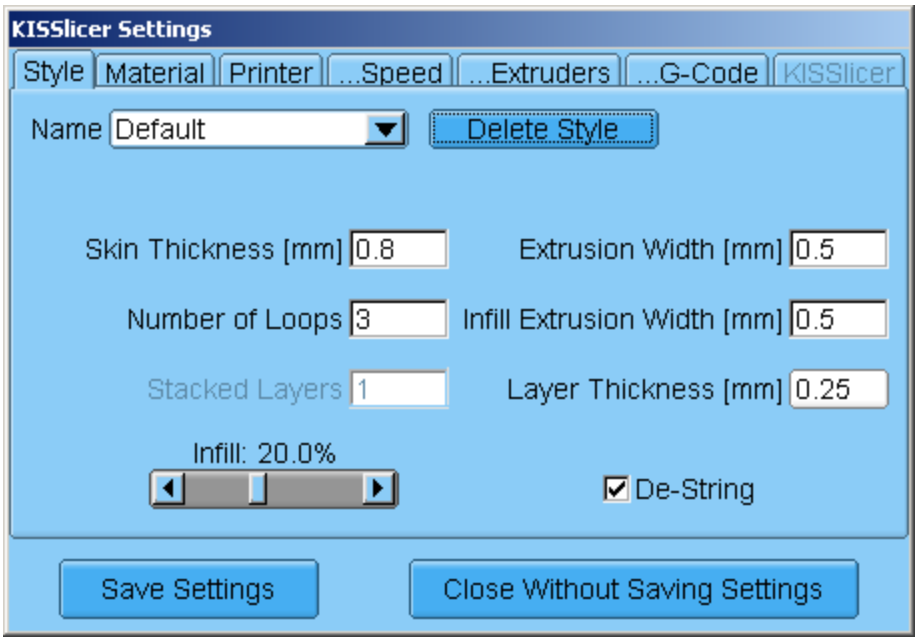
Run the executable (KISSlicer.exe) and the following window will open.



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.

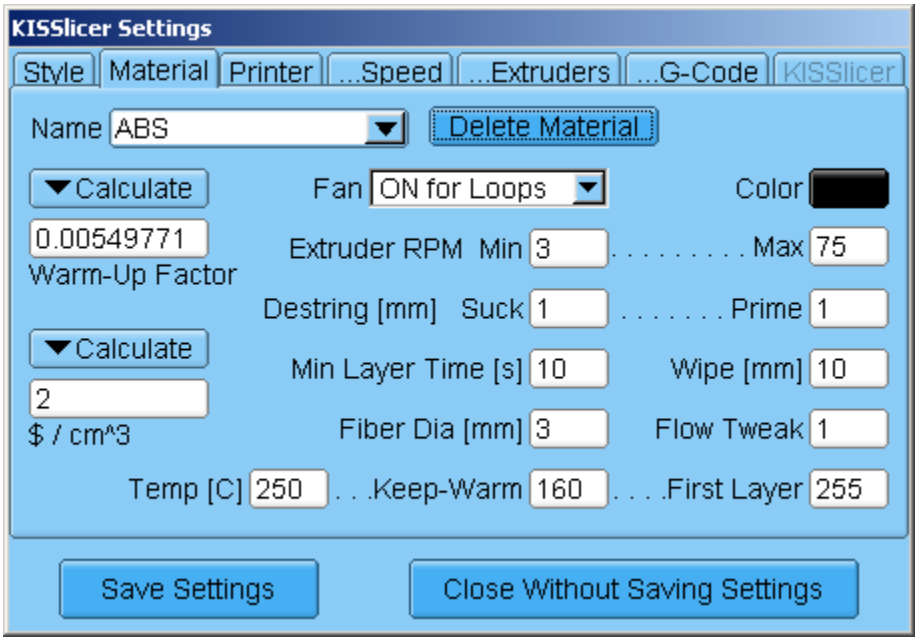
Settings

Style



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

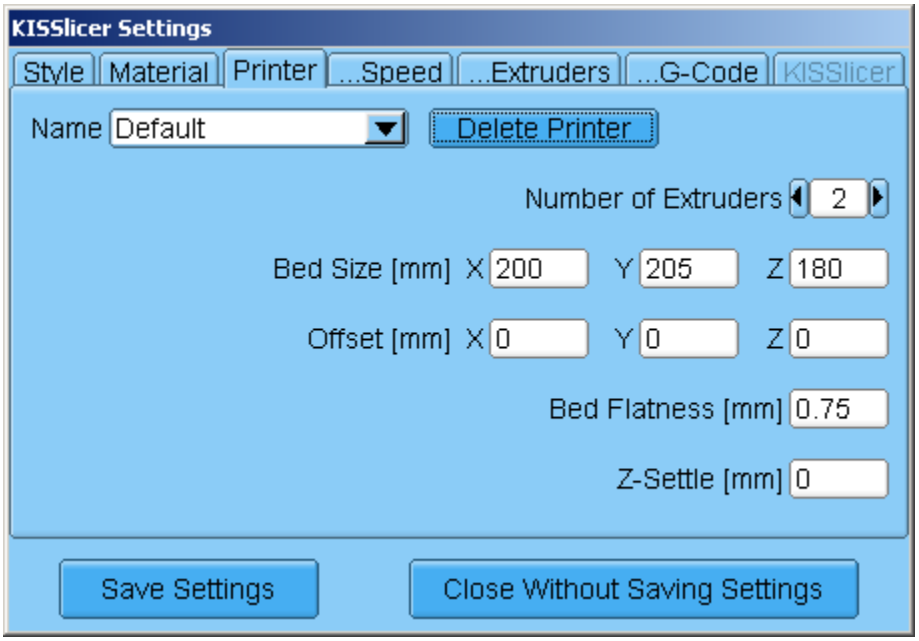
Material



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

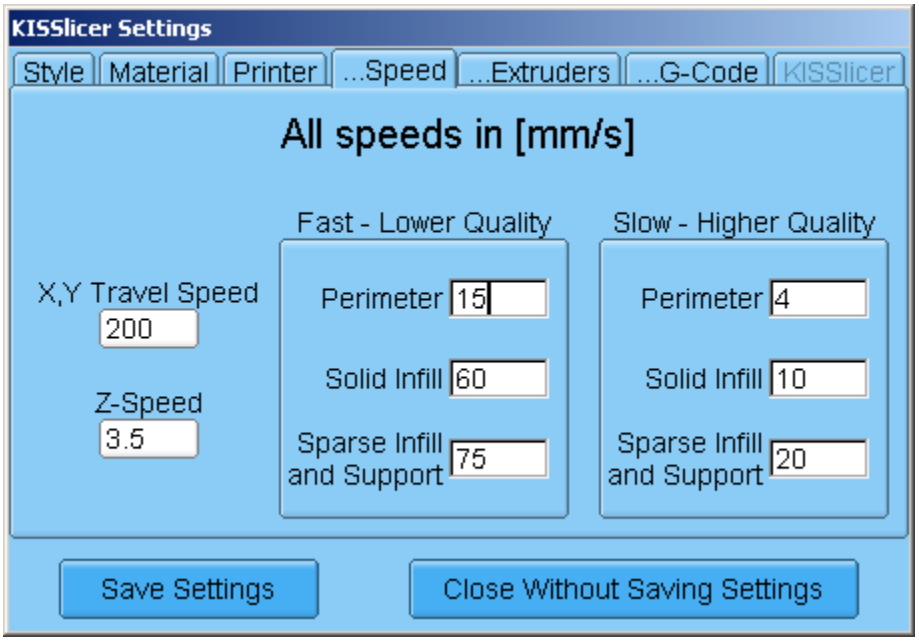
accordingly.

Printer



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.

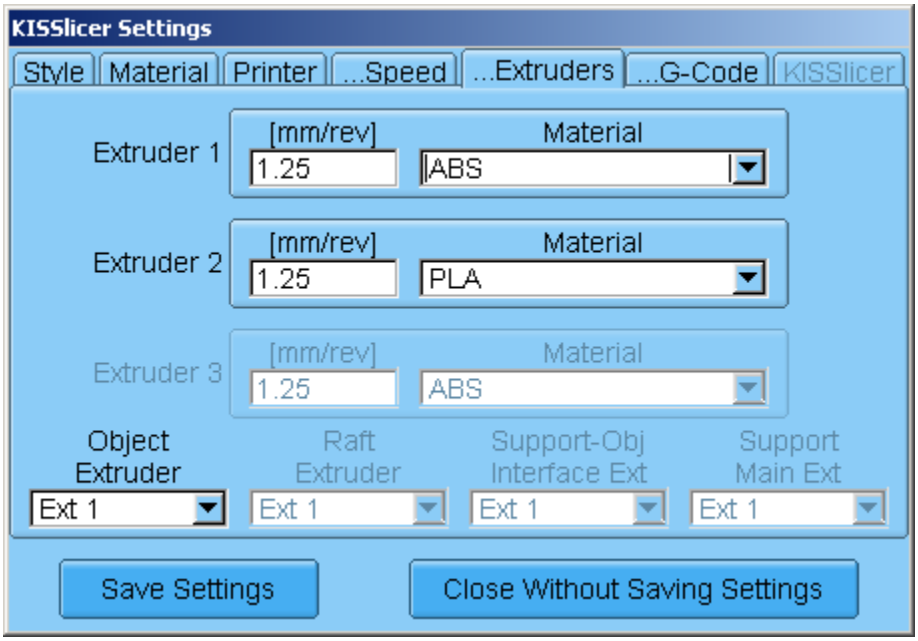
Speed



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

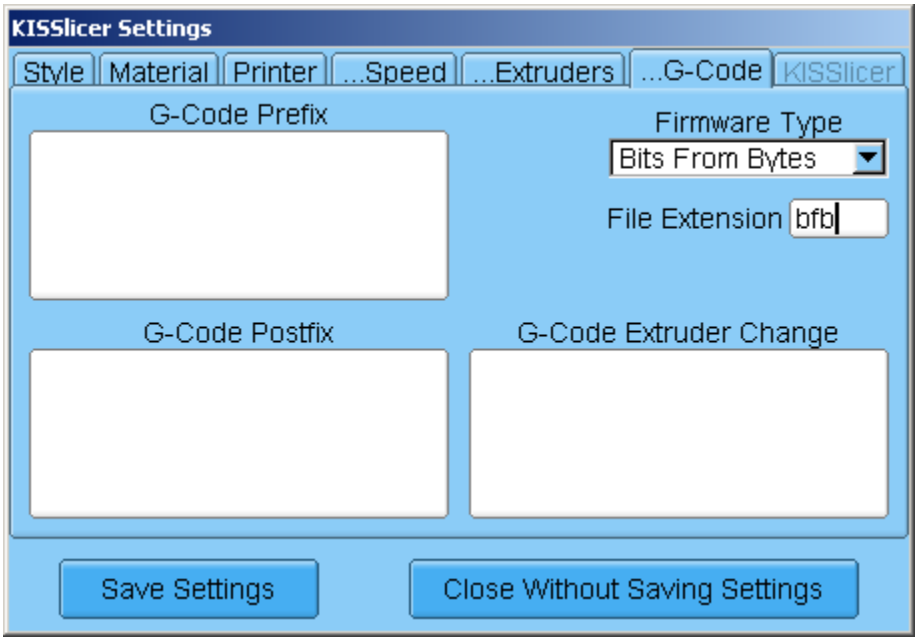
speed between the two extremes here shown. Leave them as they are for the moment.

Extruder



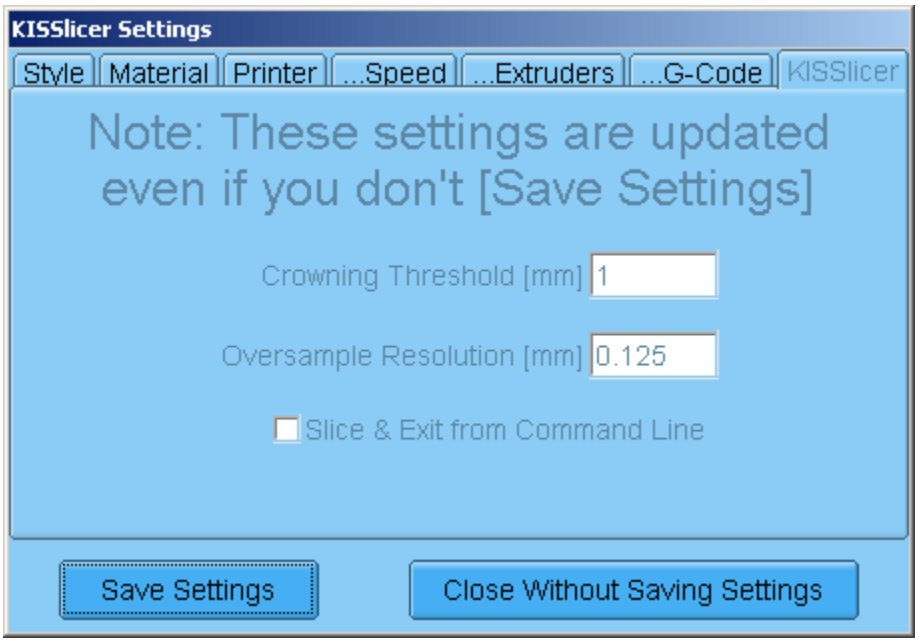
You can assign the extruder material to your available heads.
With free version you can set one extruder for all the print, while the the pro version you can assign the head to part, raft, support and part/support interface. Check at the end of the guide for an example

G-Code



****

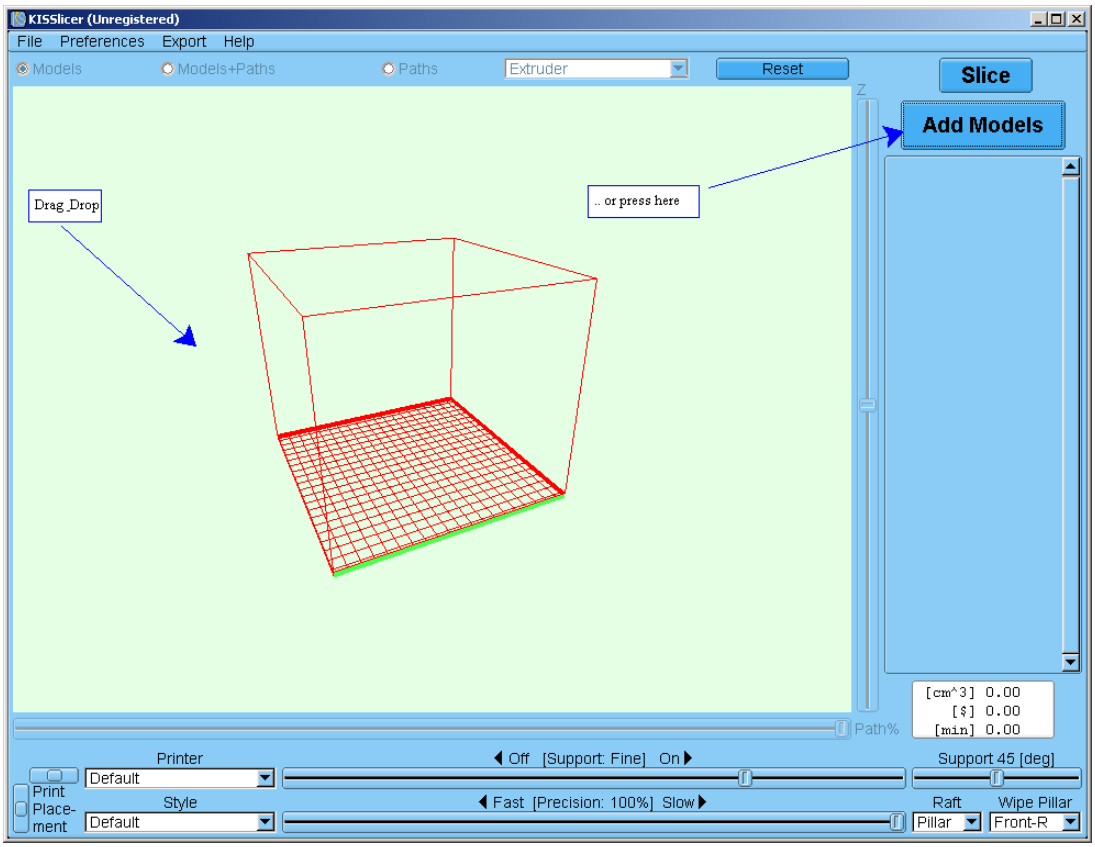
Kisslicer settings



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

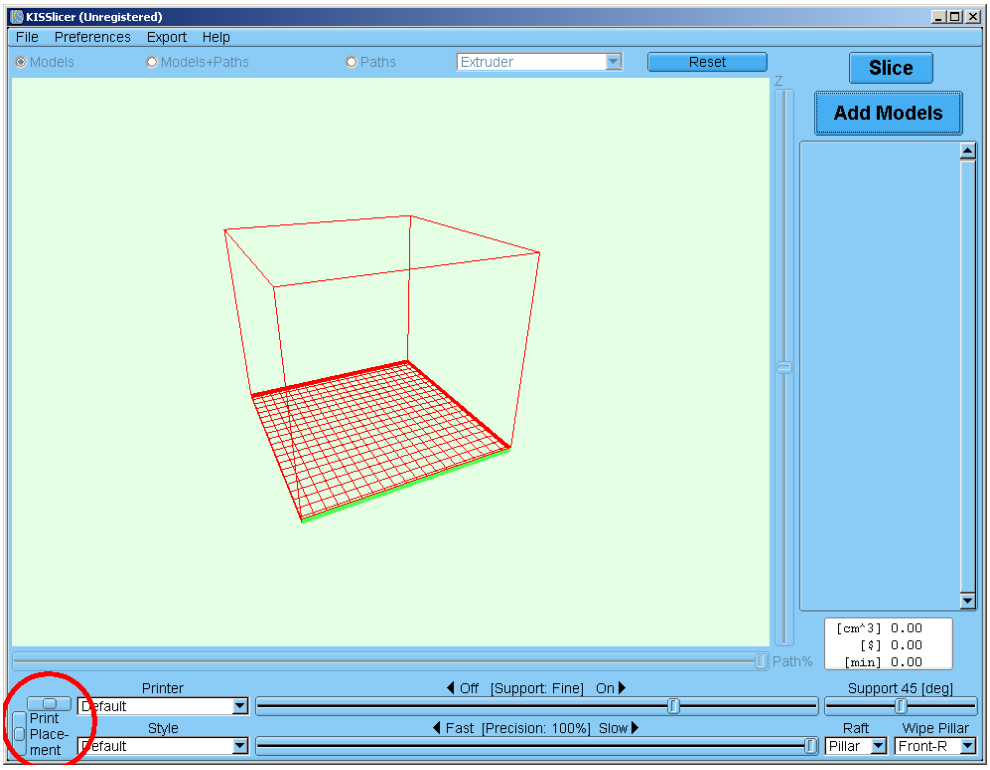
The 3D view can be modified in the following way:

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** greats 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 axtremes
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 tipically 1.25 mm/revolution)
Extruder [n] Material: Select a material for the extruer 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