KISSlicer©

User manual

Revision 1-2-15

Quick start

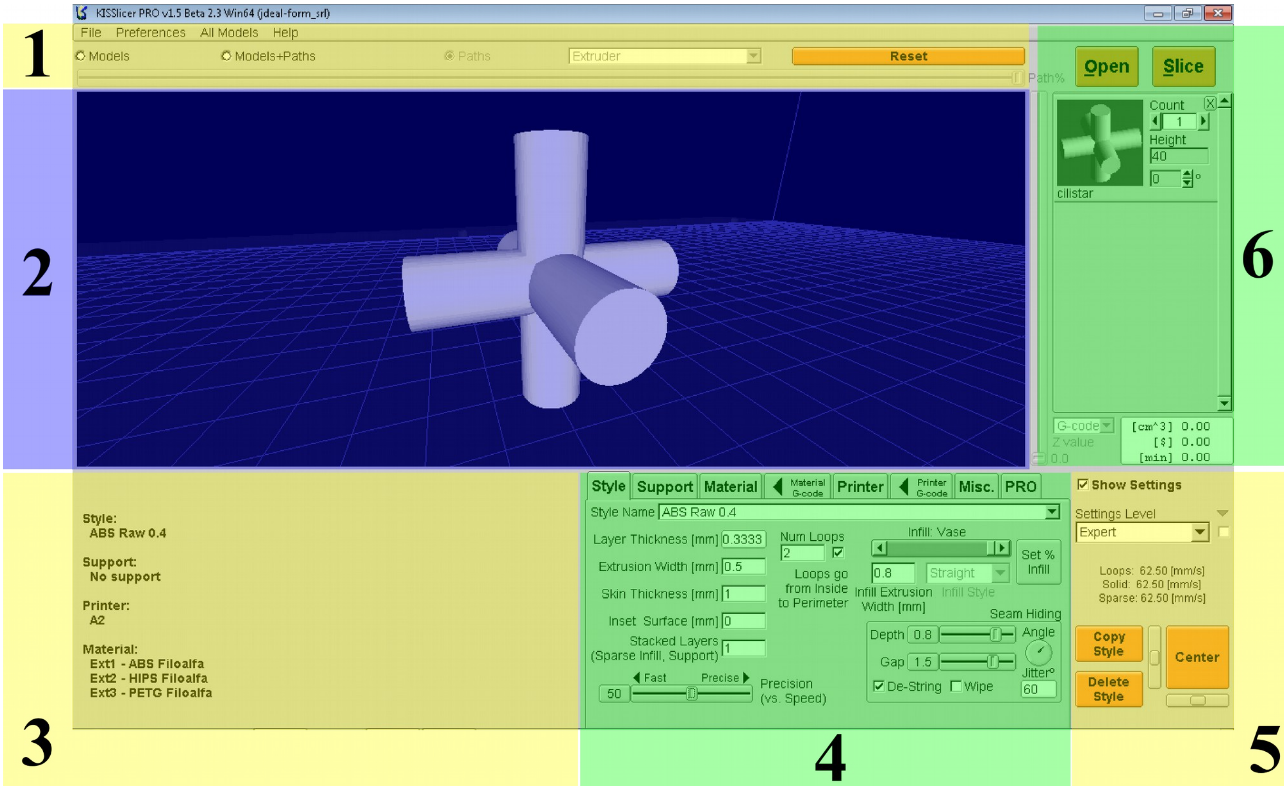

Fig. 1: KISSlicer screen areas

The KS interface can be subdivided in five 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 gcode
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

First thing to do is checking that KISSlicer setup is matching your printer configuration: in the “parameters” area, select the “printer” tab.

All the relevant printer parameters are easily obtained from your printer data sheet: be sure to seek expert advice if you aren't too confident with this step.

The “default printer” is already configured with standard values that should make no harm to most of the printers, yet, as there are countless printer models around, for sure you will need to modify some parameters.

Be sure to select the correct firmware type: most of printers will work with “5D absolute E” setting.

Before going on, you must verify the “sample material” parameters:

- diameter: check the filament you are using, measure its diameter(s) as precisely as you can and insert the found value in the “Diameter” box (Material Tab)
- Temperature: if you are using PLA, 205°C for the Main/First layer/Keep warm. If you are using ABS the temperature should be set at higher value (240°C). Bed temperature should be set (if your printer has this option) at 65°C for PLA and 110°C for ABS.

KISSlicer default style is provided to let you quickly prepare your first print.

At his point you can load the desired STL geometry, clicking on the large “LOAD” button top right: it is wise to start with some easy part, that can let you quickly verify the calibration of your parameters.

Set the printing speed with the slider found in the “style” tab: don't go too fast at first, try lowest speed to avoid damaging the feeding parts of your printer.

Clicking on the “Slice/Save” button will start the slicing of the part, saving the machine instruction (gcode) at the desired location. The resulting file can be sent to your printer: if the parameters have been appropriately set you should get your first print with KISSlicer !

Preface

KISSlicer is a slicing software available from

http://www.KISSlicer.com

This booklet refers to the soon-to-be-released beta (1.5): still, it can be used to better understand the use of almost any KISSlicer version.

Join KISSlicer users here:

http://www.KISSlicertalk.com/index.php

.

The screen images are taken from a Windows machine: your interface may look slightly different, depending on the operating system you are using.

It is released as it is: no liability for anything, use at your own advantage and responsibility.

Released as a work-in-progress documentation for 3ntr (3d printers.) customers.

Please report any typo, conceptual or really incredible mistakes at

ardizzoiadavide@gmail.com

If you think that this manual has been useful to you, and would like to see more development, you can motivate me using Paypal and sending your donation at my email address (ardizzoiadavide@gmail.com).

...Any amount will do!

Terms used in this booklet

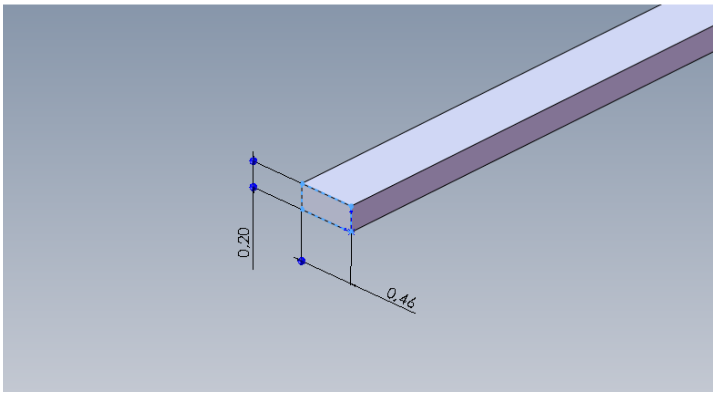

Fig. 2: Extrusion and layer height

- Extrusion width: the width of the molten plastic laid down by the extruder during the printing process. In the above picture it's 0.46mm
- Gcode: the set of instructions accepted by 3d printers.
- Infill: the inner part of an object, that can be filled with user selectable pattern and density.
  Can be
  - solid: when it is filling an exposed surface
  - sparse: when it is used to fill the inside of the part
- KS = KISSlicer
- Interface: between support and the part there is a cushion zone, affected by Z band, solid support interface switch and flow gain. If your support is actually an expensive polymer, you may want to keep using the part-polymer for support, too, and using it (ie.: PVA) just for interface. This can be done using the extruder mapping functions (Printer->Extruder material)
- Layer: horizontal slice of your model. Your model can be considered like building a house: you must lay down a layer of bricks before laying down another one. The layer height is like the brick height: if you want a rationalistic house (vertical wall and flat tops) you can get along with 20cm high bricks. If you want a “Gaudì style” one, you should use thinner ones. Same applies to your models: the thinner the layer, the higher the print quality BUT also way longer print times.
  The picture above depicts an extrusion with a 0.2mm layer height.
- Loops: the part skin can be made of several concentric lines, the outer one being the perimeter while the others being the loops.
- Nozzle diameter: the molten polymer is extruded thru a small orifice on the tip of a heated nozzle. Most widely used are 0.4 and 0.5mm diameter, but smaller (0.3) or bigger (0.8) nozzles are available.
- Ooze: most common problem on reprap based 3d printers. It is used to describe the tendency to leak molten polymer over printed part. To minimize it there are several methods: retraction, lowering nozzle temperature when not in use, using lowest possible extrusion temperature, dehydration of filament, fast movement between printing areas.
- Perimeter: outer part of an object
- Skin: KS considers as skin ANY outer surface, either vertical or horizontal. Therefore, “skin thickness” is meant for ANY outer part surface.
- Seam: the small discontinuity on the vertical surfaces given by the start and stop extrusion points.

Installing the PRO version

Just download from

http://www.KISSlicer.com/download.html

Follow the instruction for your operating system (don't worry, it's an extraordinary small package).

This document refers to the PRO version: given the power of this tool, and the really low price asked, I see no reason not to to reward Jonathan (the author of this wonderful tool) for all his efforts.

Therefore do yourself a favor and buy a PRO license:

http://www.KISSlicer.com/buy.html

If you use a cracked copy, no matter what is your excuse,
please repent and do something right in your life: BUY it !

Once installation is terminated, to get customized setup for your 3ntr printer, please download customized .ini files from

http://3ntr.eu/?page_id=330

Menu area – upper part

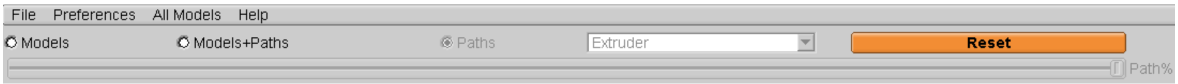

Fig. 3: Menu area

This part of the screen is the home of the drop down menus and the visualization mode selectors.

File

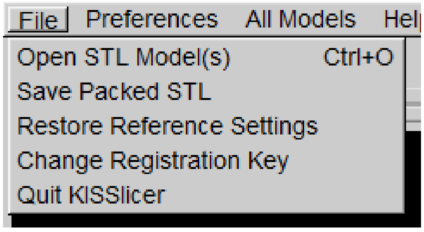

Fig. 4: File menu

- OPEN STL FILES (can be recalled pressing Ctrl key and “o”), replicated by second rightmost orange button, lets you choose the STL file to open. You can open multiple file at once (press CTRL while selecting files): this way you will be able to print multiple parts with common origin, useful when dealing with assemblies or multicolored prints.
- SAVE PACKED STL, lets you save the current 3d print scene (useful when you get load several parts and you want to save them as a single print job)

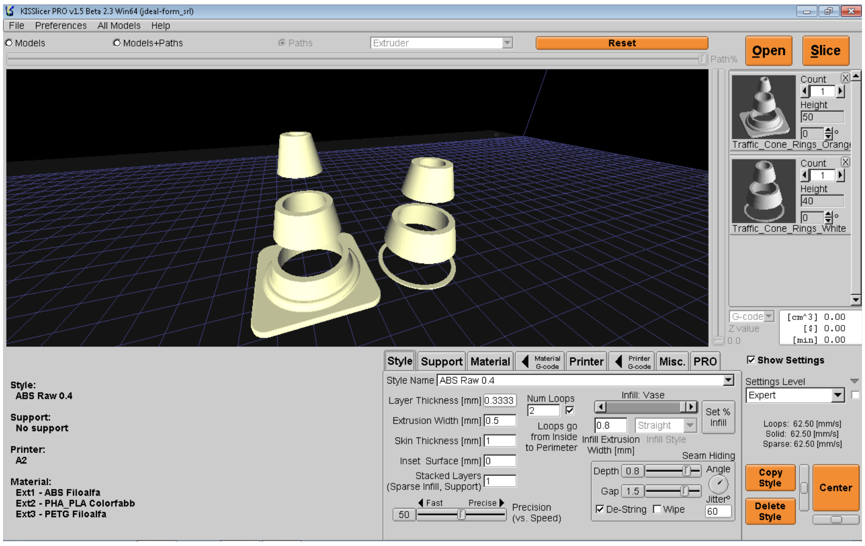

Fig. 5: Multiple files

- RESTORE REFERENCE SETTINGS: you can restore your KS settings to a copy previously saved. Best would be saving into a directory on a trusted media (cloud backup, removable USB drive, ...anything ! Keep in mind that KS settings files are really tiny, therefore you could even keep a “safety copy” into your 3d printer SD card!!)
- ENTER/CHANGE REGISTRATION KEY, to upgrade to PRO license (DO IT NOW!)
- QUIT KISSlicer, to finish current session.

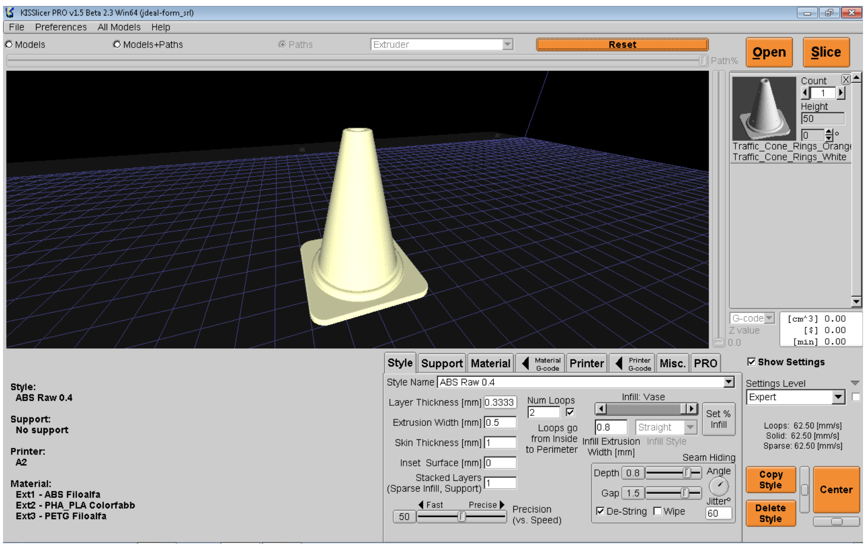

Fig. 6: Multiple files loaded at once

Preferences

Fig. 7: Preferences menu

File handling, lets you enable several options

- Preview open STL, handy but needs powerful PC to be useful
- Load multiple STL as a single model
- Use unique file names when saving, meaning that it won't ever overwrite anything: you can keep a history of your print, at the expense of increase disk space
- Slice&save: enabling this feature you will be asked for file name to save each time you slice a part

Languages

- Revert to English: reset to the original language setup
- Load a language file: load a translation for the user interface
- Translate tooltips only: will keep all the original user interface, but moving the mouse pointer over any part of interface will bring on a translated tooltip (small help window)
- Save reference language file: generate a file you can later edit to get a localized copy of KS with your own language

Display options

- Disable 3d view on slice: to speed up the slicing process. Enabling this option will get you a faster slicing, with less feedback during the process.
- Color scheme: to alter the look of your KS
- Foreground
- Background
- 3d background
- Color scheme presets: change user interface colors
- Widget scheme: change user interface graphic

Warnings

- Material change warning: useful at first, since some users mistakenly assume that browsing materials in the “Materials” tab automatically sets the current extruder to the selected material (the correct procedure is selecting material in the “Printer->Extruder materials” tab)

All models

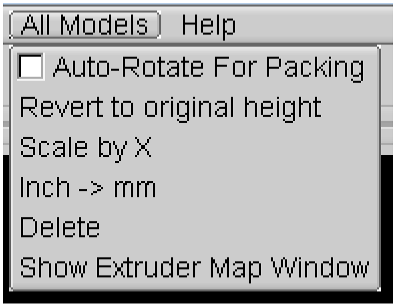

Fig. 8: All models menu

- Auto rotate for packing: if enabled, when you will place multiple copies of one part, KS will try the best orientation to improve space utilization
- Revert to original height: restore the original size of the selected part
- Scale by x: the selected part will be scaled by the selected factor
- Inch->mm: scale existing part by 25.4
- Delete: remove all currently loaded STL models
- Show extruder map window: lets you choose an extruder for every selected mesh. Useful for multicolored objects

Help

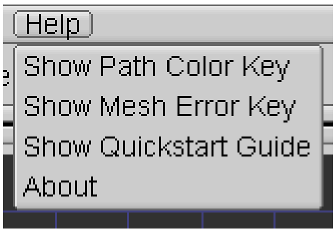

Fig. 9: Help menu

This menu is used to display some popup guides

- Show path color key – when MATERIAL display option is chosen, you may want to enable a popup color guide
- Show mesh error key – It pop up automatically when loading new parts and error are detected (default behaviour, but you can disable it clicking the option box at the bottom of the popup)
- Show quickstart guide
- About (version, etc..)

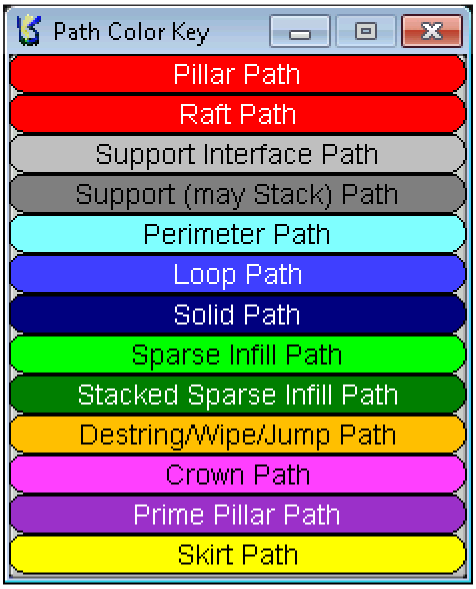

Fig. 11: Path color key - Path type

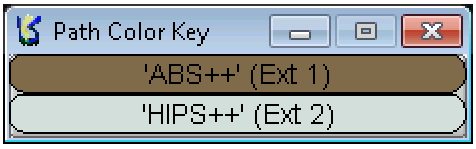

Fig. 12: Path color key - Material

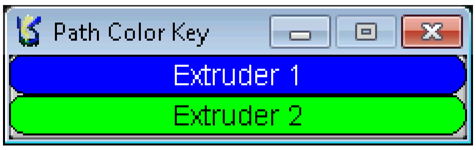

Fig. 13: Path color key- Extruder

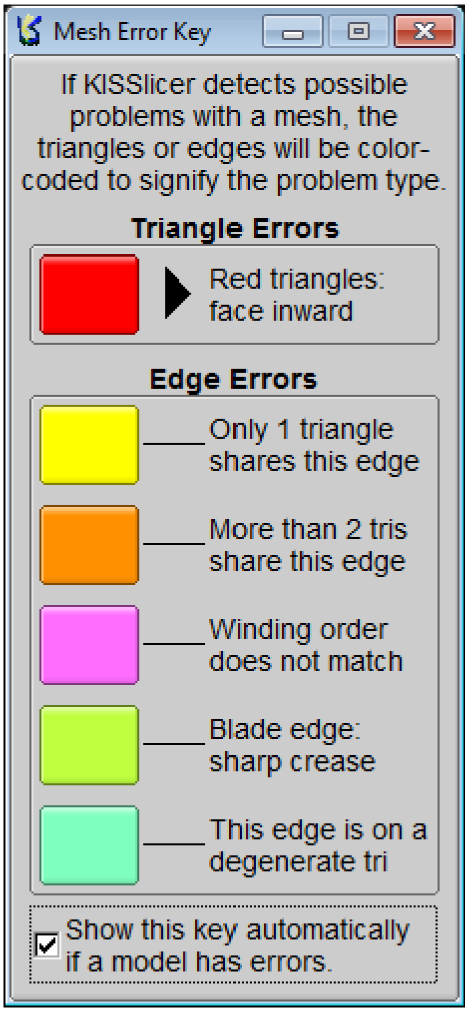

Fig. 10: Mesh error key

Menu area – lower part

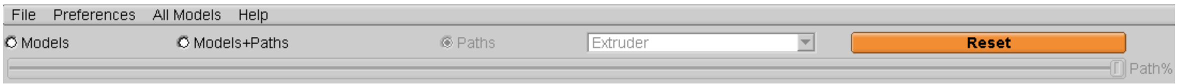

Fig. 14: Menu area

In the lower part of the menu area, there are several key controls:

Display modes

- Models: shows just the STL geometry
- Models+path: after slicing, the original geometry is superimposed with generated paths.
- Paths: after slicing, will show a top view of generated path

Path color selector (pull down selector)

- Path type: each path type will be highlighted in a different color (it is useful to enable the help->Show path color option to better understand this function)
- Extruder: will show each extruder path with different color
- Material: will color each path with the material color (you set material color in the Material Tab)

Reset button: will re-center the 3d view

Path% slider: when PATHS option is selected, you can see the printing progression during current layer, just move the slider.

Visualization area

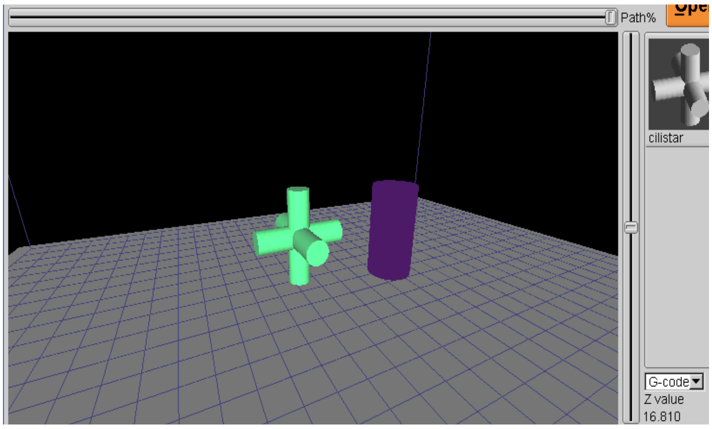

Fig. 15: 3d visualization area (Models)

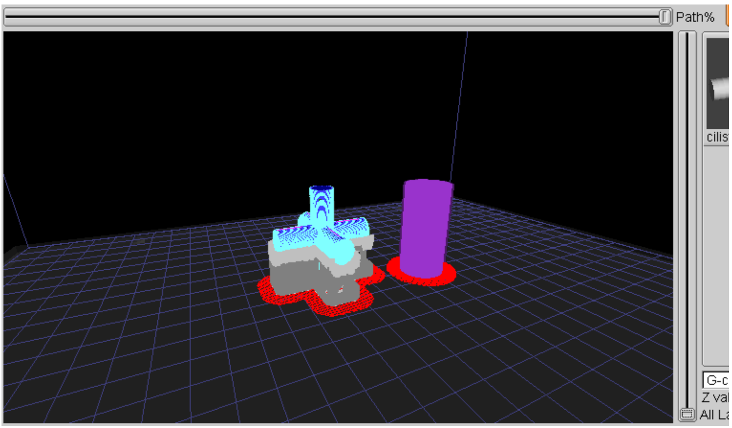

Fig. 16: 3d visualization area (model+paths)

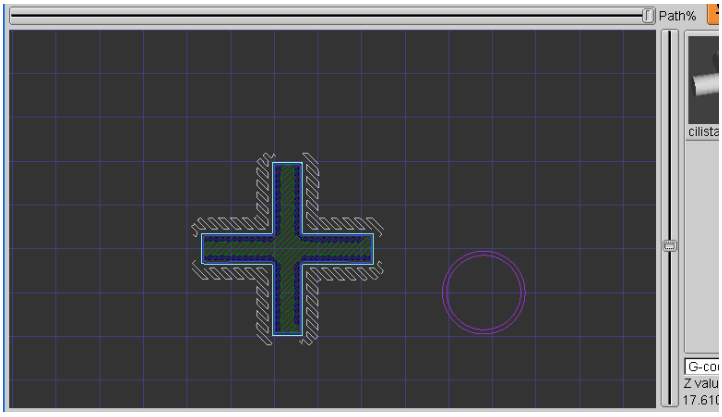

Fig. 17: 3d visualization area (paths)

It's the largest part of the screen, where KS renders the graphical display of part, path or both of them.

Use the left mouse button to move and the right button to rotate. The mouse wheel is used to pan and zoom.

The size of the print area depicted depends from the size information entered into the Printer->Hardware tab.

If you have set a print bed .STL file name in PRINTER tab, then it will be shown to help you better understand part placement.

When paths are displayed there are two scroll bars available:

the upper (horizontal) scroll bar will show the movements on the current layer as they will happen during print process.

The right (vertical) scroll bar will navigate across the layers. Moving slider all down when in model+paths mode will display an all-layer view

Settings recap

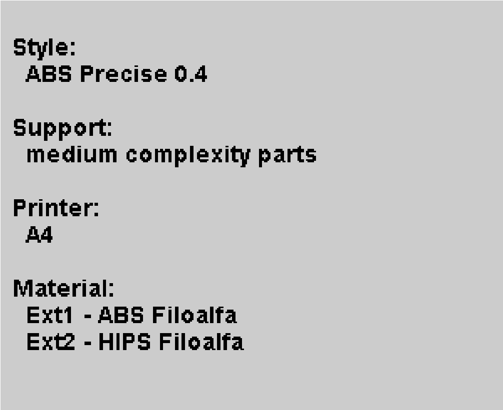

Fig. 18: Settings recap

This screen area is just a reminder of the current slicing setup: you can do nothing here, but it is always useful to look here before clicking on the Save button

Style

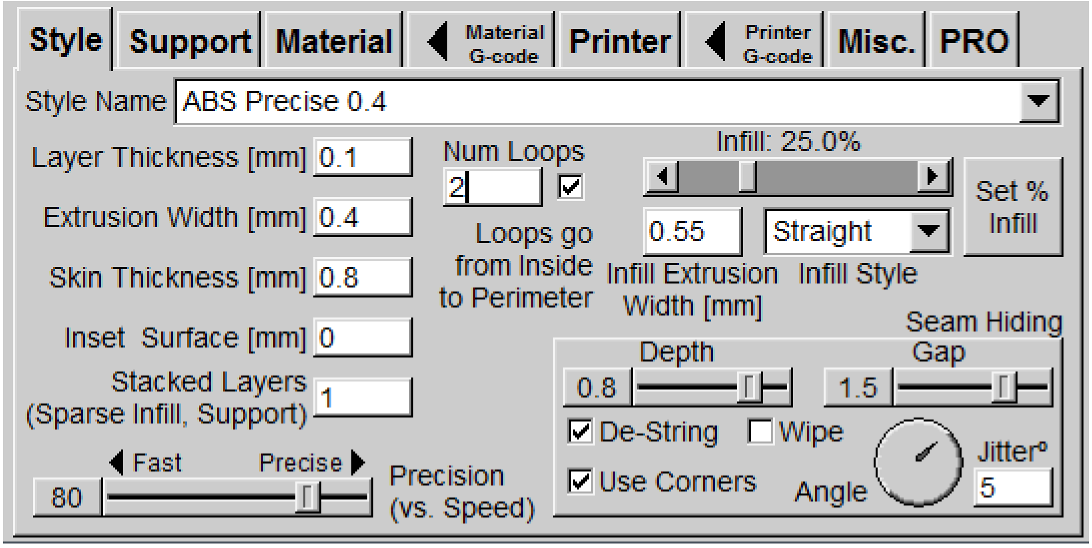

Fig. 19: Style tab

This tab is the most important one: getting a good understanding of the parameters will reward you with good quality prints.

You can build a library of styles, using the COPY STYLE and DELETE STYLE buttons at the bottom right.

Navigating and choosing the current style is easy, just use the drop-down selector at the top of the tab area ("ABS Medium 0.4" in above screen capture).

The parameters (CAUTION: a lot of "it should be" ahead):

- Layer thickness:
  - For quick prints, with low quality requirements, 75% of nozzle diameter will suffice, therefore it should be 0.3mm in case of a 0.4mm nozzle
  - For good quality prints, 50% of nozzle diameter is usually a good choice
  - For very good detail, but long print times, it is possible to use 20% or even 10% of nozzle diameter: do note that it may be necessary to greatly slow down and fine tune the temperatures to get really good results.
- Extrusion width: it should be always greater than nozzle orifice diameter. Wider extrusions will speed up prints, at the expense of lack of detail. On the other hand, thin extrusion will give you greater detail but may prove to be too brittle to be practical, and could require lower speeds.
- Skin thickness: you want it to be a multiple both of extrusion width and layer thickness.
- Inset surface: usually left to zero, this parameter will do shrink the solid at the selected amount. Can be used to remedy to lack of clearance for holes or shafts.
- Stacked layers (PRO VERSION): used to make support and infill each N layer. If left at 1 (default) it will print support and infill at each layer. If you set it to 2, it will print one infill and support every two layers, making for a faster print. Part strength may suffer a bit.
- Speed slider: the speed used to print this style.
- Number of loops: should be used to get skin thickness (= number of loops x extrusion width). Best would be having skin thickness as a perfect multiple of layer thickness.
- Loops go from inside to perimeter: enabling this option, will get better looking perimeters but a bit less precise outer dimensions. If you want precise (mechanical) parts, disable this option.
- Infill slider: choose the amount of infill, moving the slider from left (100% infill) to the right (vase infill = empty infill and no top infill). You can also use the "Set % infill" button to precisely enter the desired amount of fill percentage
- Infill extrusion width: three possibilities
  - Thinner than extrusion width: less warping, at the expense of a weaker part
  - Same as extrusion width: the molten polymer pressure in the nozzle will be constant, leading to a good looking part

- Bigger than extrusion width: part will be stronger, but appearance may suffer from worse perimeter definition and/or increased risk of warping

□ Infill style

- Rectilinear: fast but not too strong.
- Octagonal: stronger than rectilinear, but slower to print

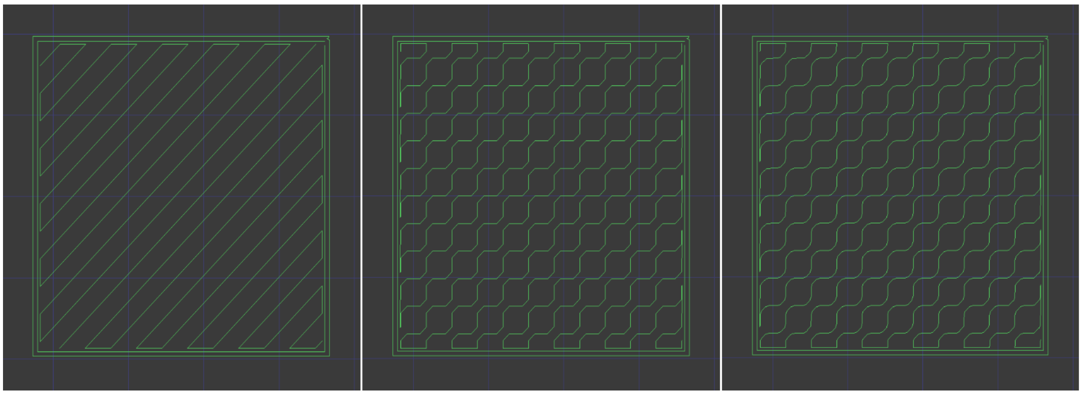

Fig. 20: Straight, Octagonal and Rounded infill

- Rounded: a bit weaker than octagonal, but faster to print. At first sight, it looks exactly like octagonal, but looking closer will reveal rounded corners instead of sharp corners.

□ Depth: moving the slider from 0.00 to 1.00 will extend toward the inside of the part the start and stop of each perimeter line. Carefully setting this parameter can hide the start and stop “blobs” on the vertical surfaces.

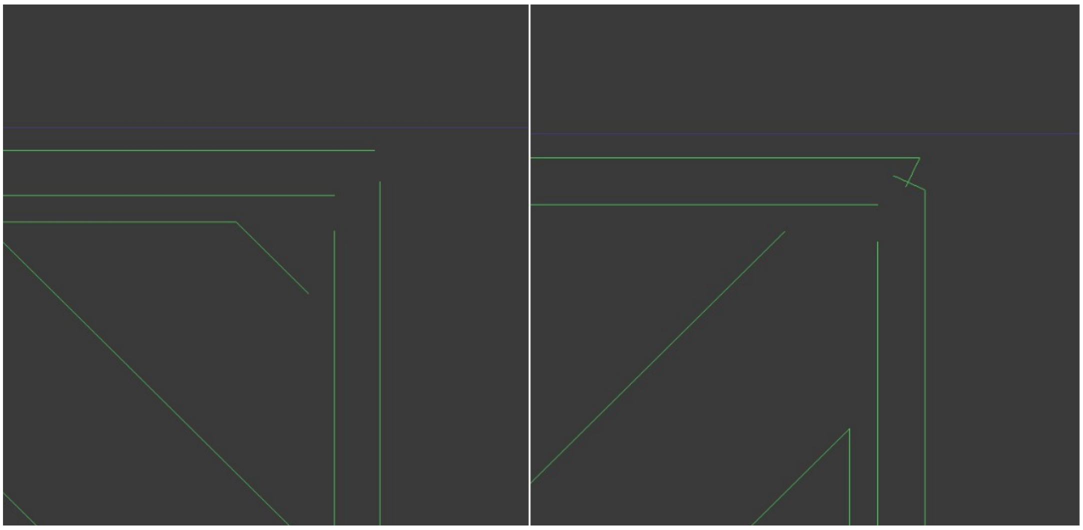

Fig. 21: Minimum (left) and maximum (right) depth seam hiding

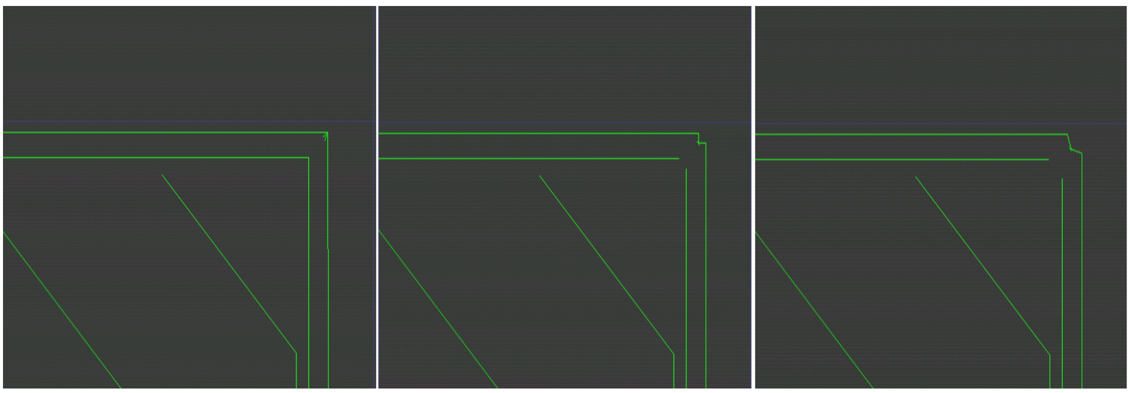

Fig. 22: Gap settings: 0, 1 (default), 2

□ Gap: the distance between start and stop of each extrusion loop.

- Jitter: the effect of this parameter is more useful for rounded surfaces. It will spread the seams over the selected angle range. Usually, 10° is enough to minimize the start-stop esthetic impact on the vertical surfaces.
- Angle: visually choose the placement of the seams on the object(s)
- Wipe on: enabling this feature, once extrusion is finished, the nozzle will backtrack after filament retraction. Useful with very fluid polymers. Backtrack distance is set in the corresponding MATERIAL tab.
- De-String: perform a Prime (positive filament feed) at the extrusion start and a Suck (negative filament feed) at the end, as set in the corresponding fields of MATERIAL tab.
- Use corners: will try to place start and stops on the corners

Support

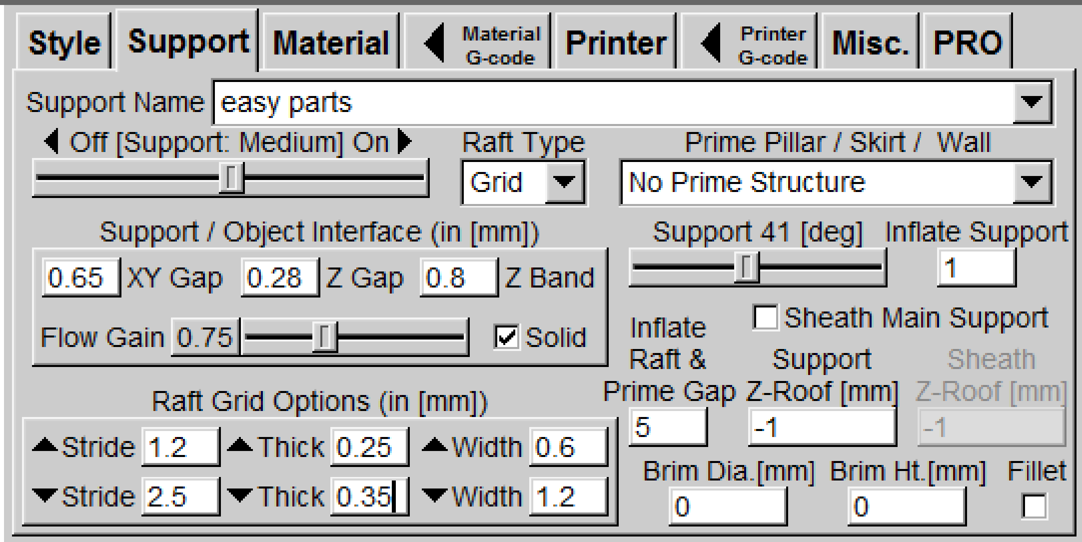

Fig. 23: Support tab

Sooner or later you will step into a part with overhangs, forcing you to choose a supporting strategy.

KISSlicer will give you several tools to face those situations.

The SUPPORT tab lets you set up a library of support strategies. Like the STYLE tab, you can browse your settings with the drop-down selector and use copy and delete buttons to organize your archive.

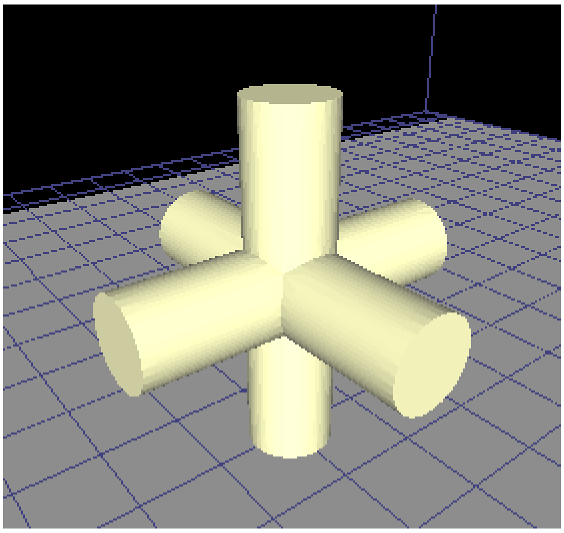

Fig. 24: Part with overhangs

The parameters:

- Support slider: it has several steps (from left to right):
  - Off: no support geometry will be generated
  - Coarse: very basic parts, with low quality requirements
  - Rough: medium-low supporting density
  - Medium: as the name implies...medium density supporting
  - Dense: increased support density, useful for sphere-like objects
  - Fine: very high support density, print time will increase noticeably
  - Ultra: very high density: for challenging geometries

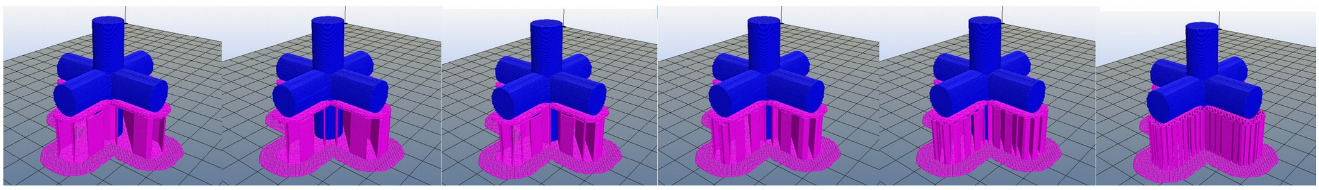

Fig. 25: Support (left to right):coarse,rough,medium,dense,fine,ultra

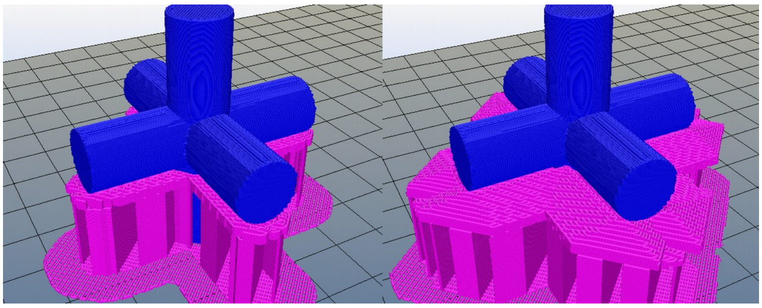

Fig. 26: Inflate support: 1 (left) 5 (right)

- Inflate support: default is 1, you can decide to increase it to strengthen the supporting structure for large parts
- Solid support-object interface: if you want to get perfectly flat bottom surfaces, enable this option. Keep in mind that with this option double extruder use is mandatory, possibly using a soluble support.
- Flow gain: this slider will control the adhesion between support and part (also known as interface area). Higher numbers will make better adhesion (thicker extrusion)

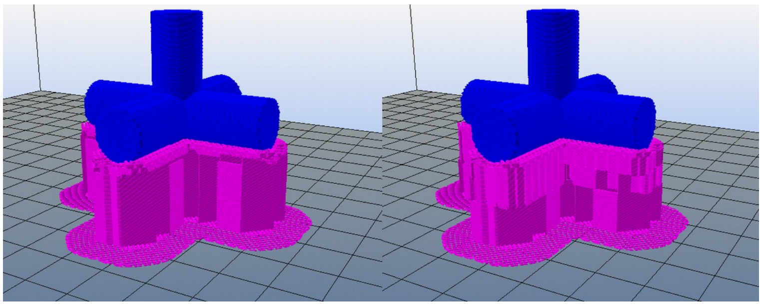

Fig. 27: Z Band: 0.8 (left) 4 (Right)

- z-band: controls the thickness of last stage of supporting. Thicker sections will help with "difficult" parts
- Support degree slider: decide what is the maximum slope that can print unsupported

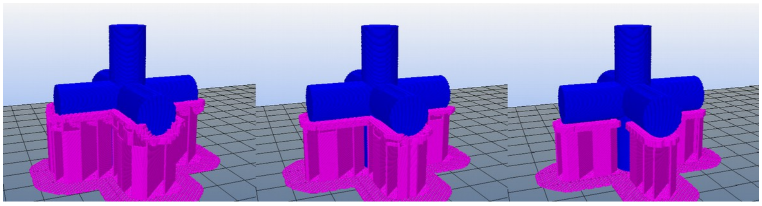

Fig. 28: Support degree (left to right): 5, 40 and 70 degrees

- Gap (XY): the horizontal distance between support and object. You want it to be low (0.15) for round/spherical surfaces and high (0.7) for vertical/flat surfaces. Just keep in mind that low values will stick with the part making hard to remove supporting structure.
- Gap (Z): the vertical clearance between support and part. Usually this value is a fraction of layer thickness.
- Support z-roof: if left at default value (-1) the program will create supports where needed. Entering any value here will stop the support creation for any part of the object whose Z is bigger than the value entered.
- Sheath main support: enabling this option will put a perimeter around the supporting structure (sparse support). The resulting supporting assembly will be stronger but tougher to remove.
- Sheath z: maximum height to be reached with sheath. Default value (-1) will make sheath as high as your support.

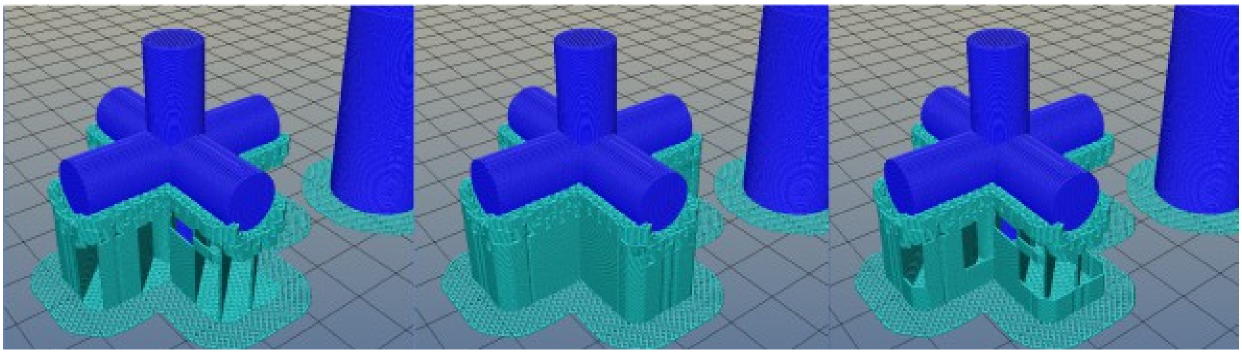

Fig. 29: Left to right: support, sheath, 5mm sheath z-roof (all with prime pillar)

- Raft type:
  - GRID – a mesh will be printed between part and print bed. The mesh is printed with the extruder selected in PRINTER->Extruder materials tab. This way, if you have a multi extruder setup, you can use soluble polymer to speed up print cleanup process
  - PILLAR – same as GRID, but a layer of micro pillars will be added between mesh and print bed. Useful if your soluble polymer is too sticky

- Raft grid stride / thickness / width: the grid raft is made of two layers, and you can choose to alter the default behavior. You may want to have wide, thick and quite spaced lower layer to improve adhesion and compensate for printing surface defects. You may also want to have a thinner and more dense upper layer, to get better support interface.
- Inflate raft: will let you "inflate" the skirt or grid (if you choose to use them, of course)
- Prime pillar / skirt / wall: KISSlicer gives you several extruder priming strategies. You can choose among several options:
  - Single pillar: one column (pillar) will be created for every nozzle involved. The column will be automatically sized and placed near the part to be printed.
  - Skirt (first layer) : useful for priming at the very beginning of the print. No other priming will be performed during printing process.
  - Short wall: a bit like single pillar, but it will enclose the current print and will last until the last extruder excange.
  - Wall (all layers) like short wall, but will last until the end of the print

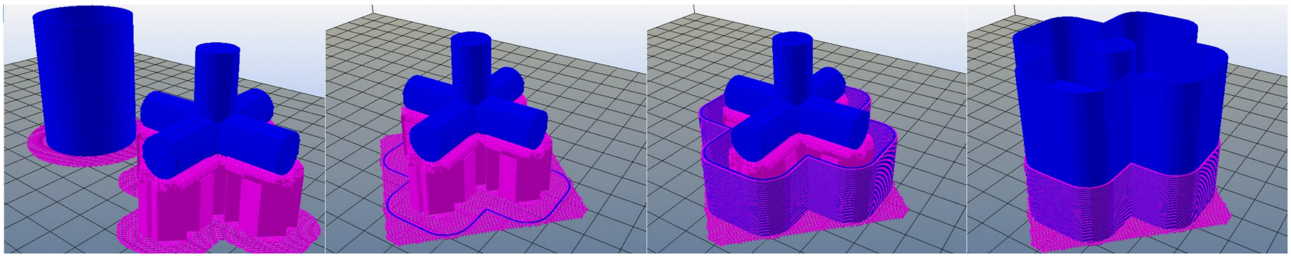

Fig. 30: Prime pillar / Skirt / Short Wall / Wall

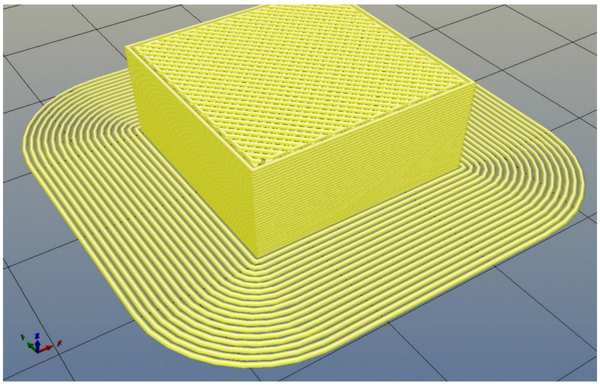

Fig. 31: 10mm brim

- Brim diameter – when adhesion is difficult, brim will make for a larger "foot" to increase adhesion (at the expense of increased time spent to manual finish of the part once printed). The diameter is how far the "adhesion improvement surface" should extend from part "foot" area.
- Brim ht: usually left at zero, if this value is increased the "adhesion improvement surface" will be as thick as you want
- Brim fillet: if BRIM HT is quite big, a filleting will be applied to the generated surfaces to improve adhesion and minimize warping

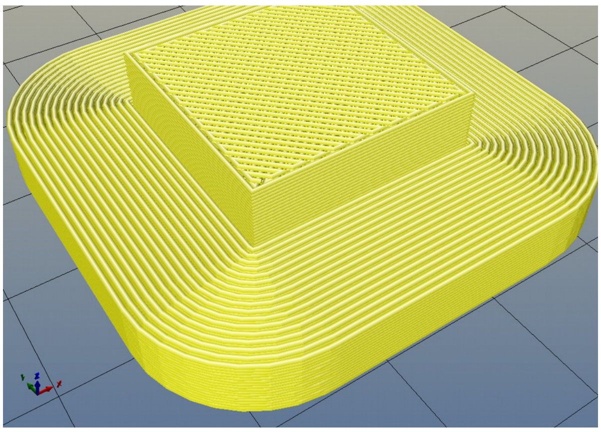

Fig. 32: 10mm Brim + 5 mm Ht

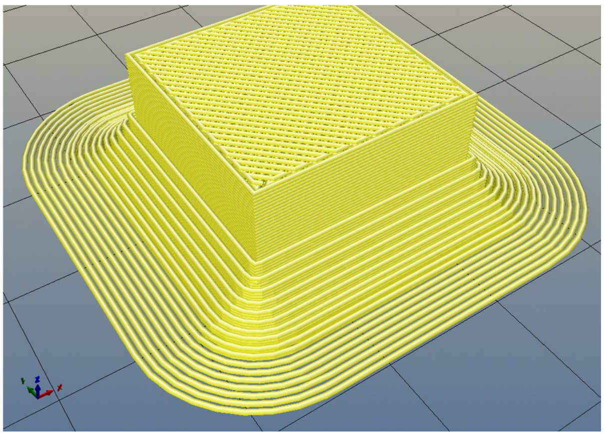

Fig. 33: 10mm Brim + 5mm Ht + fillet

Material

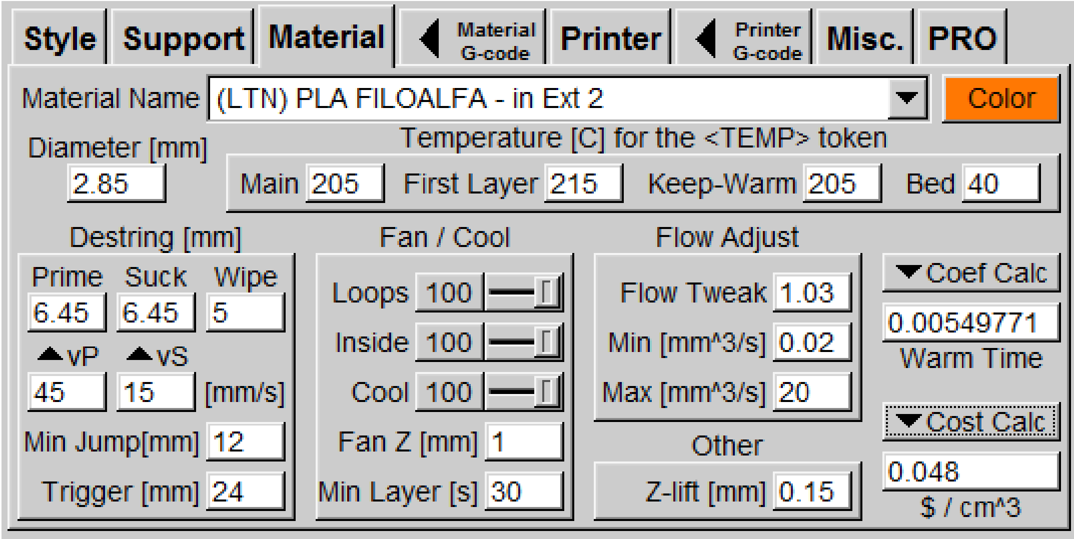

Fig. 34: Material tab

The parameters of the Material tab:

- Diameter(mm) - Being able to correctly measure the filament diameter is one of the fundamental aspect of the printing process: all the slicing procedure relies on this information, and if we supply approximate information we can't expect precise output.
  Therefore use a caliper (nowadays a digital caliper is cheap enough) to precisely measure the filament diameter: don't settle with just one measurement, take several measurement and find the average diameter.
- Temperature for the <TEMP> token: KS uses the “tokens” to allow the customization of the printing process. The default behaviour is to warm the extruder when needed, then keep it warm whem not in use and turn off all of them when part is finished.
  - Main: the temperature for the print of the entire part
  - First layer: set this temperature at the higher temperature than Main, to improve the adhesion to the bed (printing surface)
  - Keep-warm: when using multiple extruders, you can choose to lower the temperature of the inactive ones to minimize oozing. Set the temperature wisely (i.e.: not too low) otherwise your extruder swap will be painfully slow, waiting for the activated extruder to warm up to the selected working (main) temperatures
  - Bed: if a heated bed is available, you can set its temperature to improve adhesion
- Color button: will let you set the color that will be used to visualize the path extruded with this material
- Destring: 3d printer push filament at the start of printing process and retract it when it's done. You can decide several parameter of this behavior:
  - Prime (mm): the amount of filament to feed before starting the current layer printing process. You want this value to be greater or equal to the next parameter.
  - Suck (mm): the amount of filament to retract when the printing is done for any given layer.
    If you set this value too high you risk extruder jam.
  - Wipe(mm): the length of the backtracking to minimize the oozing.
  - vP / vS : the speed at which filament is fed during PRIME and SUCK
  - Min jump (mm): the maximum movement that can be performed without oozing. Longer jumps will start the wipe process.
  - Trigger (mm): the maximum movement that can be performed after wipe (if enabled). Longer jumps will start the wipe AND destring (suck) process.
- Fan/cool – Almost all 3d printers have fans to cool down the nozzle area. KS gives you tools to handle the part cooling process.
  - Loops (0..100%): the fan speed to use when printing everything except infill areas.

    - Inside (0..100%) : the fan speed to use when printing infill areas (inside the part or the top and bottom planar faces)
    - Cool (0..100%): when slowing down the printing process to comply with Min Layer setting, keep at least this fan speed
    - Fan Z(mm): fan cooling will be enabled after printing process has printed the selected thickness
    - Min layer (s): the minimum time granted to a layer to cool down. If the layer printing time will be smaller, then the speed will be reduced to comply with this setting
- Flow/adjust
    - Flow/Tweak: it may happen that despite you precise filament measurements, the print looks “skinny” or “fatty”. You can use this parameter to compensate. Default is 1, increase it if you seen gaps between fill (i.e. 1.05) or decrease it if you nozzle is surfing over molten polymer waves.
    - Min (mm3/s): the minimum volume that can be reliably extruded. The better control you have on you extruder feeder, the smaller the limit can be. If you reach this minimum, KS will speed up the printing speed.
    - Max (mm3/s): the maximum volume you can successfully melt and extrude in a second. Don't be too optimistic when setting this parameter, otherwise you risk filament grinding (feeding wheel “eats” the filament)
- Other
    - Z-lift (mm): when extrusion is finished, before moving to the next print area the extruder(s) will be raised from the printed part by the required amount
- Coef calc – When KS computes the time required to print, needs to know several parameters such as acceleration and warm up time for extruders. Clicking on “Coef calc” a warm time calculator will pop up, letting you easily set up this parameter: Extruder temp. is an average between working and keep-warm temp. Minutes to reach temp. is meant the time it takes to reach above said average value from room temp. The warm time value will be automatically inserted in the appropriate text box: you can still edit the value if you want.

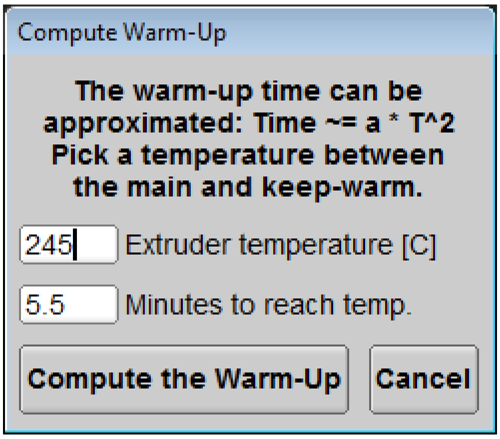

Fig. 35: Warm time calculator

- Cost calc – KS will give you also the cost of object(s) being sliced. You need to calibrate this value just once: slice any part, print it and weight it as precisely as possible then click on “Cost calc” to bring out the material cost calculator. Insert the required values (just be sure to use same unit of measure for spool and part weight!)

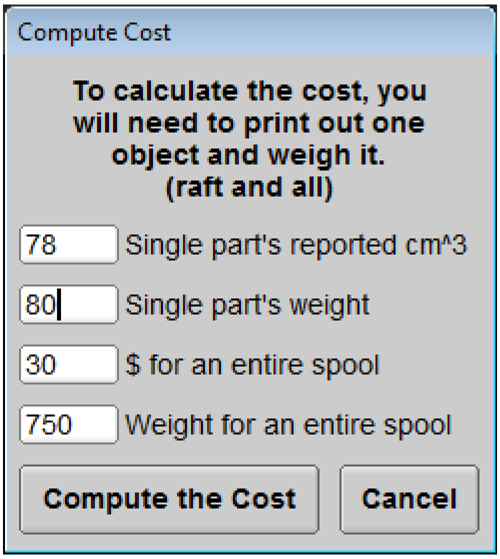

Fig. 36: Material cost calculator

Material GCODE

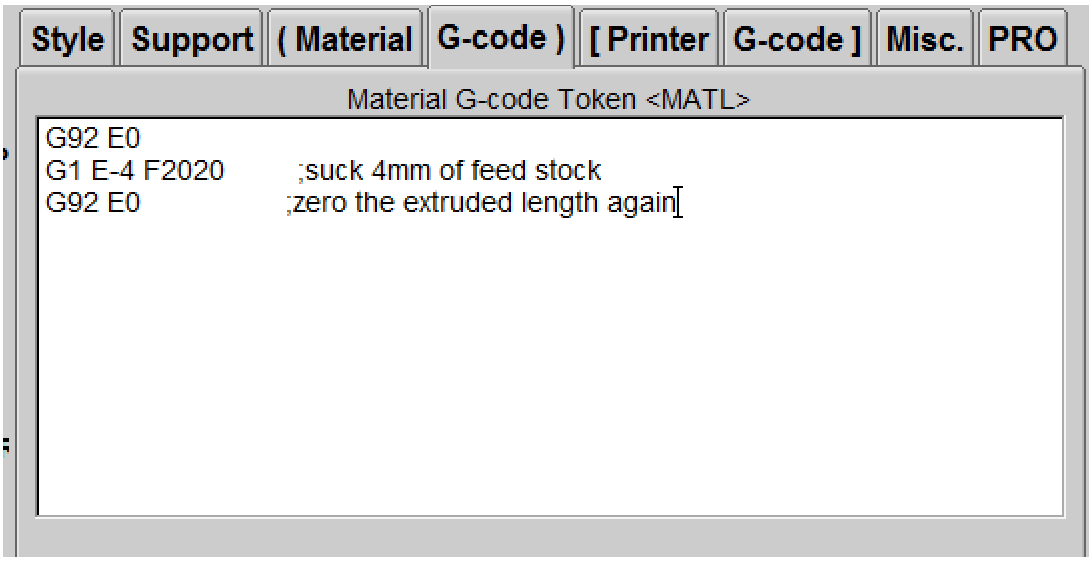

Fig. 37: Material Gcode

You may want to perform material-specific tasks: this is the reason for the <MATL> placeholder.

In the printer Gcode area you can insert the <MATL> keyword anywhere you like, and during gcode generation it will be replaced with the text you are going to insert into the above shown tab.

You could:

- Do some extra nozzle purging/cleaning in a specific print area
- Feed or retract more filament at nozzle change
- Set a custom pre- or post- heating procedure

Printer - Hardware

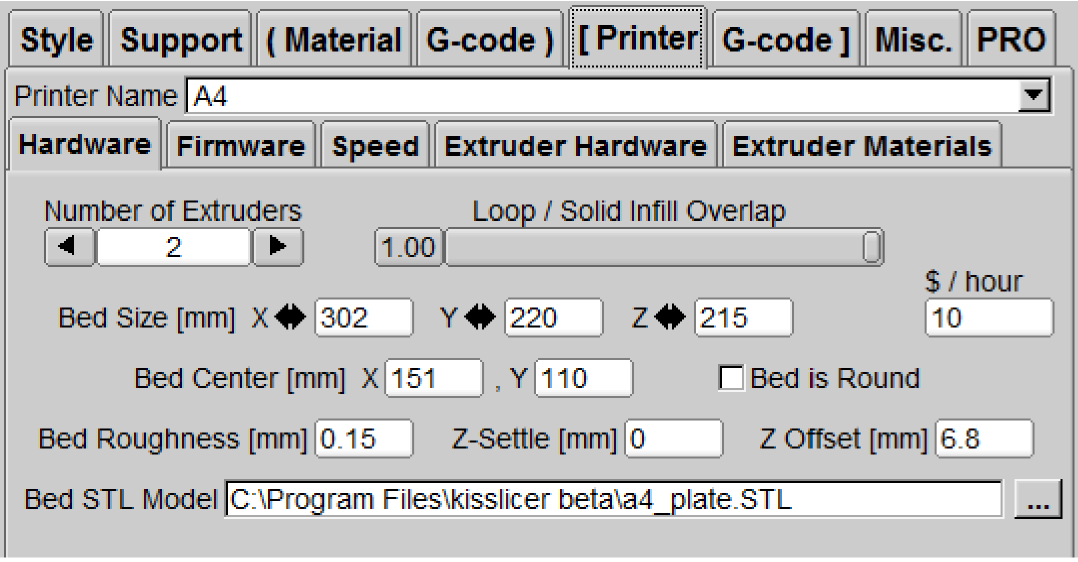

Fig. 38: Printer tab - hardware

The hardware setup of your printer takes place here: you have as usual the opportunity to set up a library of printer profile you can copy and delete at will.

Almost all of the parameters here are very important, therefore don't set them without really knowing well what you are doing – on some printer architectures you may even damage the printer itself!

On this sub-tab you will find the following parameters:

- Number of extruders: how many physical extruders you have on your printer
- Loop/solid infill: this slider sets the overlap between infill and perimeter. Minimum value will make practically no contact between them, resulting into a cleaner print but way weaker. Maximum value will give you strongly bonded inner parts at expense of a little less sharp outer surface.

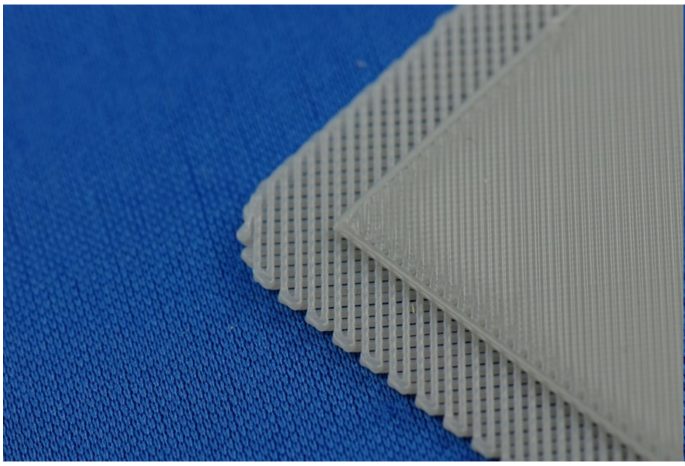

Fig. 39: Infill overlap = 0

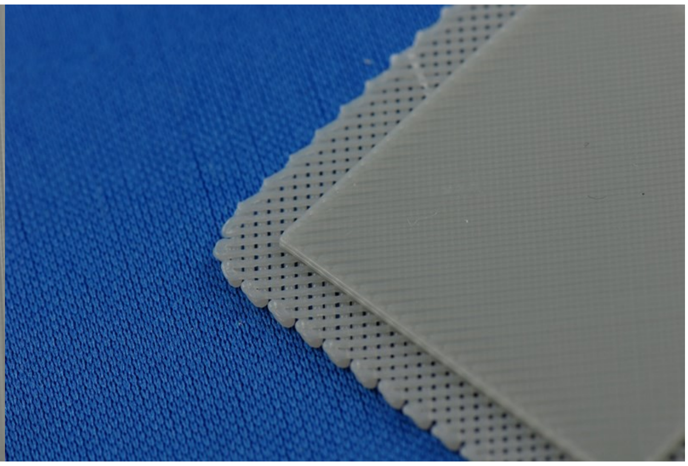

Fig. 40: Infill overlap = 1

- Bed size: your printer working envelope
- $/Hour: this is your cost/hour. If you charge anybody for your work, you should calculate your business cost per hour + printer running cost (electric power cost per hour). KS will use this to tell you the final cost of each print
- Bed center: your bed center coordinates
- Bed is round: enable this if you are using a delta printer
- Bed roughness: you can use this parameter to inflate the first layer to match irregular plates. Wise use of this parameter will help you to improve first layer adhesion.
- Z-Settle: when the Z movement is not precise, KS will compensate for given slackness.

- Z-offset: this is the extra Z distance that will be considered at the first layer. You may want to enter (small?) negative values to make for thinner first layers (BUT some firmware setups will not allow this). In any case, this is one of the key parameter for a good quality print.
- Bed STL model: if you have one for your printer, it will greatly improve your visual experience with KS

Printer - Firmware

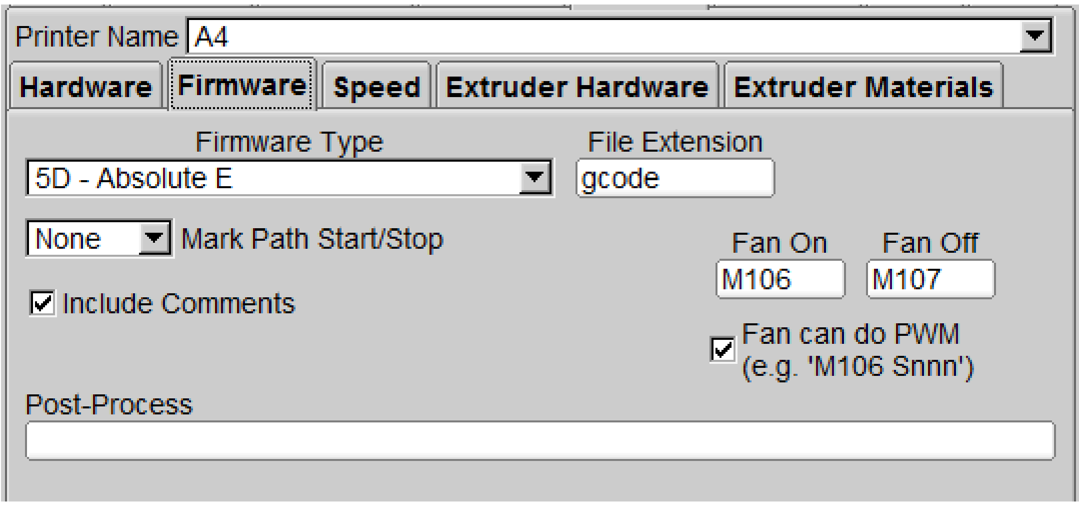

Fig. 41: Printer tab - Firmware

This is probably one of the least used tabs, unless you change your firmware day in, day out. Before attempting any parameter change here, be sure to know what you are doing. Seek GOOD advice from somebody with you same printer or from your printer supplier!

You have few parameters to set:

- Firmware type: if you are unsure, stick with “5d – Absolute E” or better yet, seek good advice.
- File extension: default is “gcode”. You may alter it, but keep in mind that your choice could make your files invisible to other softwares.
- Mark Path start/stop: let it at NONE if you don't know. Other options are absolutely firmware specific.
- Include comments: will place all parameter used to slice your part atop of the GCODE.
  Handy feature, if you don't have memory space problems.
- Fan ON/OFF commands: the vast majority of Firmwares use M106/M107 for this. If your machine uses more exotic values you can enter them here
- Fan can do PWM: if your hardware cannot decide the speed at which the extruder cooling fan runs (meaning that you can get only turn ON or OFF you fan, no speed control) you should un-tick this.
- Post-Process: if you need to know what this is for, you shouldn't care about this feature :-). Jokes apart, this option is used to tell KS which post processor is needed to handle special processing on the gcode saved.

Printer – Speed

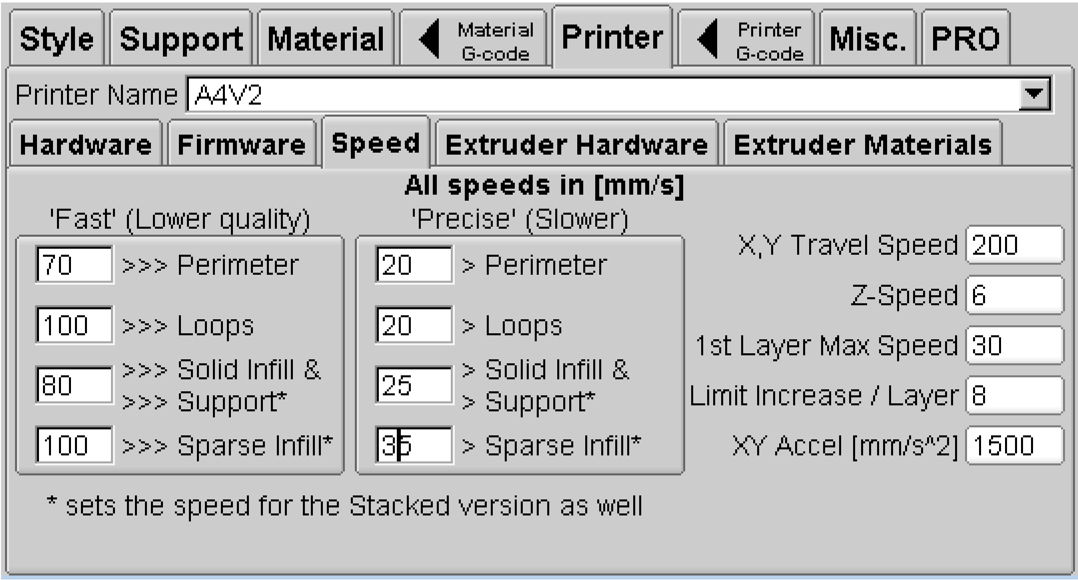

Fig. 42: Printer tab - Speed

Here you set the speed limits for all your prints: set the value your machine can really print at!

- Fast (lower quality): set the fastest values your printer can handle. Usually perimeter should be a little slower than infills. Solid infill should be the slowest of the infills.
- Slow (higher quality): usually nobody wants to go slower than 10-15 mm/sec, unless you are using a very thin nozzle orifice
- X,Y travel speed: the maximum speed your printer can go (KS uses this speed for non-printing moves)
- Z-Speed: maximum Z speed your machine can handle. This will be the speed used for z-lift movements and layer change
- 1st layer max speed: usually you want the first layer to be printed quite slow to insure good adhesion to the printing surface
- Limit increase: maximum allowable speed increase on second layer onward
- XY accel: KS will use this value just to compute total printing time. It won't be used into Gcode generation.

Printer - Extruder Hardware

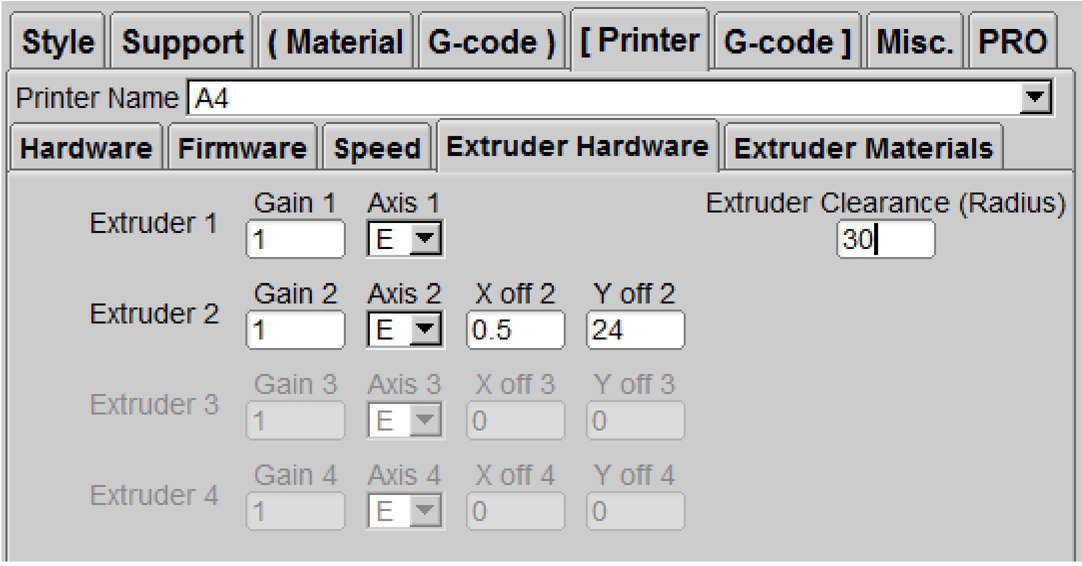

Fig. 43: Printer tab - Extruder Hardware

This is another seldom visited tab: usually you set it once then forget about it, unless you have altered the extruder relative positions.

For each extruder you can set the gain: some firmwares (i.e.: Marlin) won't let you adjust individual feed parameters, therefore here you can put up with this limitation and tweak as finely as possible the feed parameter.
The procedure is basically detaching the feed unit from the extruder, feeding a known amount, measure the effectively fed material and putting into GAIN field the needed multiplication factor.
For example: if you asked for 100 mm and you got 105 then your GAIN field for given extruder should be 0,952 (100 / 105 = 0,952).

Also, you can reconfigure the logic addresses of your extruder(s): just use the pull down selector and you will be able to choose other physical addresses other than default (E).

If your printer has more than one extruder, you can adjust the offset of secondary extruder(s) to the primary extruder. Basically you print a test pattern with all involved extruder, then adjust offsets here.

When doing sequential printing (i.e.: printing multiple object one at a time, instead of printing them all at once) you absolutely need to avoid hitting printed parts with your print head: the EXTRUDER CLEARANCE will keep a round clearance centered on your extruder(s), therefore you need to know how wide and deep is your printhead “footstep” and insert the bigger size divided by 1.5 here (if your print head is 60mm wide and 30mm deep then you will insert 60 / 1.5 = 40)

Printer - Extruder materials

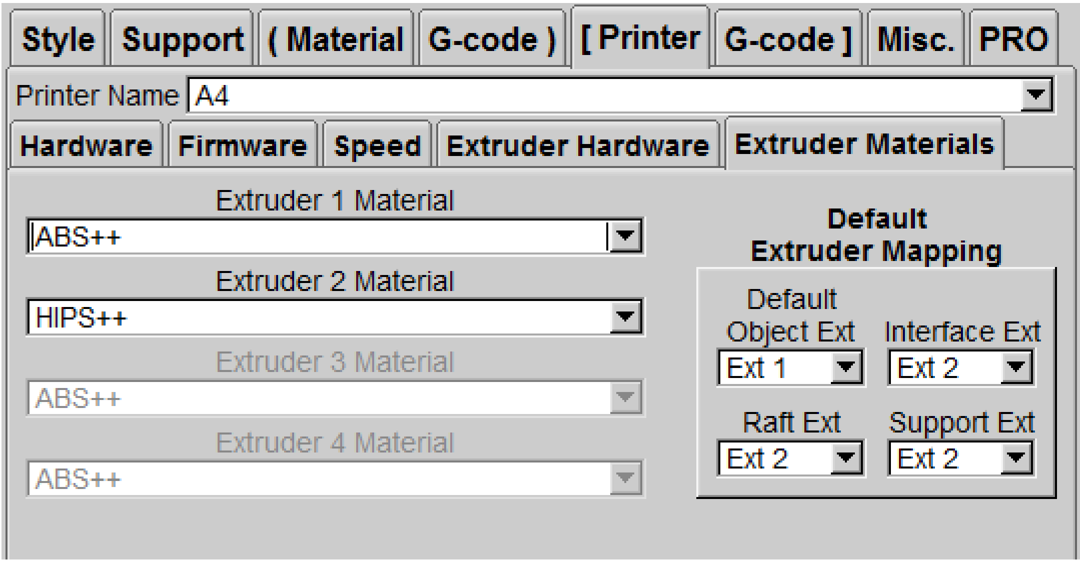

Fig. 44: Extruder materials

You can control the filament being loaded on your printer extruder(s): KS will then generate the Gcode accordingly.

For each extruder you can choose from the Materials you have defined so far, and if you are a paid PRO license user, you can also decide the extruder mapping, meaning that you can choose what duties perform with each extruder:

- Default object (Perimeter, Infill, upper and lower faces)
- Support (the structure to help printing overhanging parts of the object, but not in contact with it)
- Interface (the part of the support in contact with the object)
- Raft (the part used to improve adhesion to the printing bed, excluding BRIM)

Printer Gcode

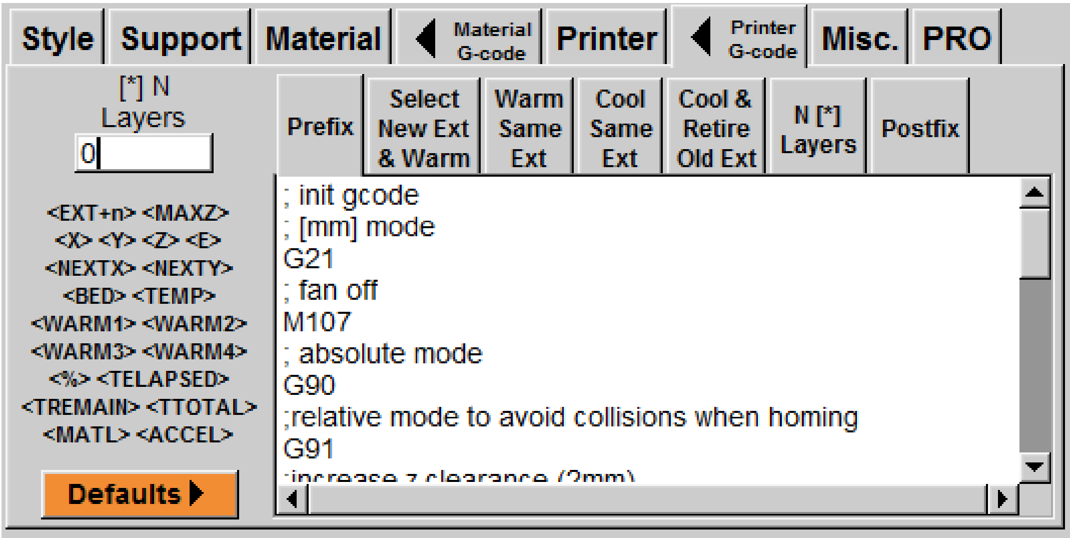

Fig. 45: Printer Gcode

Most of the Gcode generated by KS can be customized without too much effort (if you know a bit of Gcode programming): the rest of this chapter assumes that you are comfortable with Gcode and your firmware M and G values.

There are several phases where some degree of customization is allowed:

- Prefix – The code that is run once at the beginning of the print process.
- Select new extruder & warm – each time an extruder is selected immediately before printing
- Warm same extruder – each time an extruder is selected while it was at pre-heating temperature
- Cool same extruder – each time an extruder is de-selected (in a single extruder setup it will be run just before Postfix code)
- Cool & Retire old extruder – if extruder is no longer needed (in a single extruder setup it will be run just before Postfix code)
- N[*] layers – code to be inserted every * layers. The * value is set on the text field at top left of Gcode Tab
- Postfix – the code executed at the end of the print job

The button “Defaults” will delete all the Gcode fields with default scheme (there is no way back if you confirm this option).

KISSlicer uses several “tokens” (keywords that are going to be replaced with values, coordinates, or even many lines of code).

Tokens may be parsed (interpreted) as integer and some kind of math can be done with them.

<TOKEN> is the same as <TOKEN0> is the same as <TOKEN+0> is the same as <TOKEN-0> E.g. If <TOKEN> is valued at 100, then <TOKEN15> = <TOKEN+15> = 115

Here is the list of the tokens currently accepted by KISSlicer (some description of parameters are cut and pasted from old post on the KISSlicer site)

- <ACCEL> can be used to set the machine acceleration set into PRINTER SPEED, passed as an argument to M201 (print move acceleration) or M202 (travel mode acceleration)
- <BED> : the bed temperature as set in MATERIAL settings
- <EXT> : is the extruder, this is base 0, as in the 1st extruder is 0. If you need it to be a 1, you can use <EXT+1>
  'Prefix' and 'Select Extruder' will have <EXT> evaluate to the next extruder to be used.
  All other inserts will have <EXT> evaluate to the just used extruder
- <MATL> : replace the token with the MATERIAL GCODE text (any gcode token inside MATERIAL GCODE will be replaced, too)

- <MAXZ> same as <Z>, but if used during a sequential printing is the maximum Z reached so far. Useful to safely move above all the printing area for nozzle cleaning/priming
- <NEXTX><NEXTY> X or Y coordinates of the next point where extruder(s) will move.
- <TEMP>is the temperature of the extruder -
  'Select Extruder' KISSlicer will substitute the correct temp (first layer or regular) as needed 'Deselect Extruder' will select the Keep-Warm temp if the extruder will be used again, or 0 if it just finished
- <X><Y><Z><E> are the placeholder of current axis/extruder positions
- TELAPSED is the time elapsed from print start
- TREMAIN is the time left to complete the print process
- TTOTAL is the time needed to perform the complete printing process
- <WARM1><WARM2><WARM3><WARM4> are the keep-warm temperatures for each extruder
- <%> will resolve to the percent done estimate at that point in the print (using the timing estimates, so it should be decently accurate). It will have one decimal place (e.g. 17.3), and will be 0 in 'Prefix', 100 in 'Postfix', and guaranteed to be between those for all other inserts. Offsets are allowed.

Misc tab

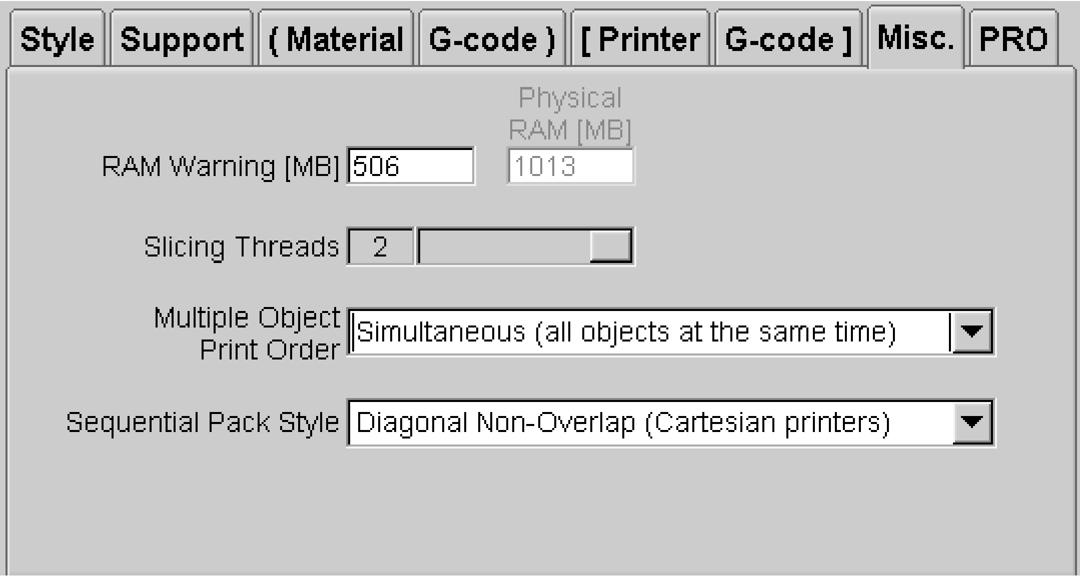

Fig. 46: Misc tab

This tab lets you set the following values:

- Ram warning: if the slicing process needs more memory than the set limit, a warning will be issued. The physical RAM memory is displayed for your convenience.
- Slicing threads: more is better, but too many may strangle your PC, giving you USB problems (read: aborted prints) if you run out of system resources
- Multiple object print order: you have two choices
  - Simultaneous (all objects at once)
  - Sequential (one object at a time)
- Sequential pack style: if you choose sequential print style, you have to select also how object are packed for printing
  - Diagonal (needed for cartesian printers)
  - Extra space (needed for delta printers)

PRO tab

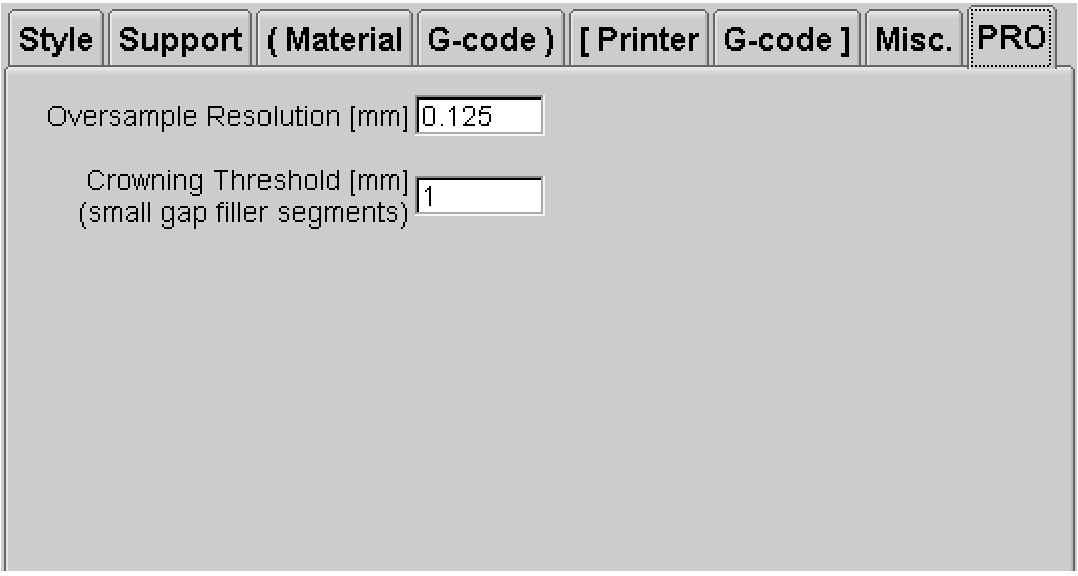

Fig. 47: PRO tab

If you buy a PRO license, you will be able to use the parameters on this page:

- Oversample resolution: changes the minimum detail level. On large models you may want to increase the resolution to conserve memory. On the other hand, smaller numbers (higher resolution) will let you print higher level of detail and get past troubled objects with sharp areas. This parameter is related with slicing speed (lower numbers = higher resolution = longer slicing time): if resolution is set high (ie.: 0.01) slicing time will increase considerably as well as memory requirements – may cause problems with slow PCs or with scarce RAM.
- Crowning threshold: when trying to fill small gaps, segment shorter than set value will be deleted. Inserting negative value will disable crowning.

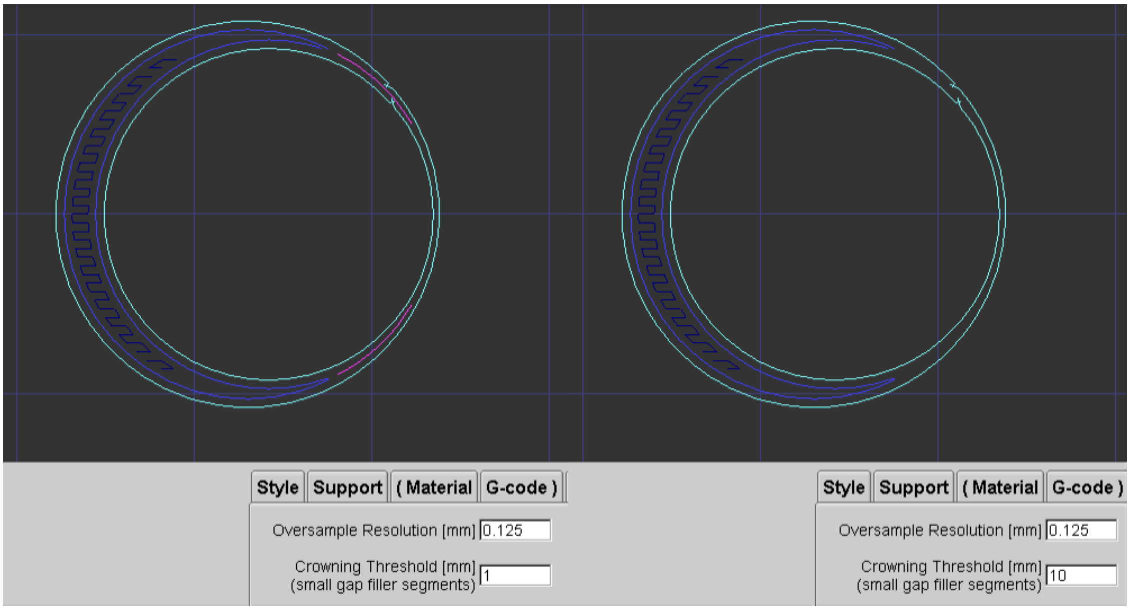

Fig. 48: Different crowning setup

Setting area

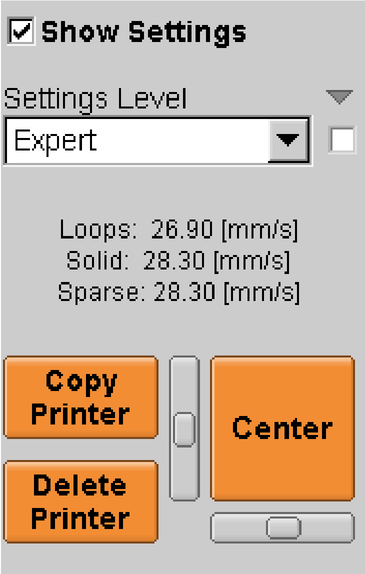

Fig. 49: Right side parameter area commands

At the right side of the parameter area there is a small area with several key commands:

- Show setting check box: tick this one to show the parameter tabs
- Settings level: (beginner/medium/expert) depending on the chosen level, the user will be granted access to different level of parameters (all this guide refers to expert mode)
- A small un-labeled check box: enabling this one will force all setting levels above the current one to use the defaults values.
- Current speed values: the speed is set with the slider found at the “style” tab
- Copy and Delete buttons: will let you organize STYLE / SUPPORT / MATERIAL / PRINTER setups.
- Center Button: pressing this will automatically center the part(s) on printing plate
- horizontal and vertical slider are provided to move around the part(s) on printing bed

STL area

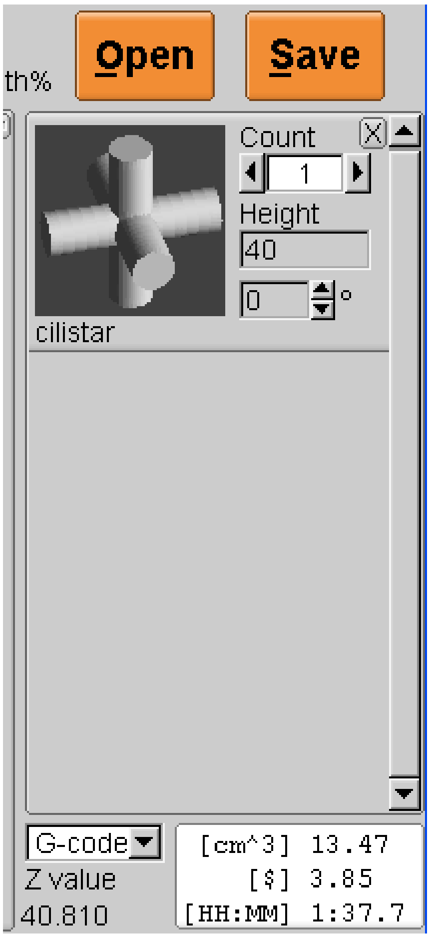

Fig. 50: STL area

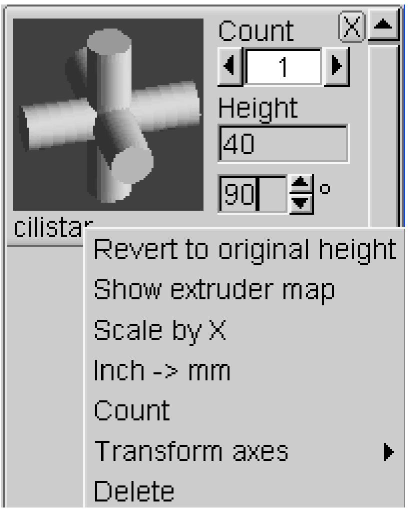

Fig. 51: Popup menu

This screen area deals with part management.

At the top you find two buttons:

- OPEN: to load the STL files of desired parts
- SAVE (or CALC) : to start the slicing process and save resulting Gcode

At the bottom you will find (left to right):

- Gcode / Model drop down selector: will change the coordinate system used by the vertical slider. Very handy to verify vertical slicing precision
- Z value: display of the current vertical slider positions
- Part data:
  - volume (expressed as Cm3)
  - cost (expressed in US$)
  - print duration

For each part loaded, a small rendering will be displayed along with several quick to use commands:

- Count: to insert copies of the selected part
- Height: part height expressed in STL units. Entering any value will scale the part to reach desired height
- Part rotation: you can enter directly the amount or interactively use up and down arrows to set it

Right-clicking on the part information will bring a popup menu:

- Revert to original height – to undo any scaling
- Show extruder map: when an object is a multimesh one, it will show a pop up screen, letting you associate different extruder to each mesh. Click on a mesh then click on an extruder you want to use to print that mesh. Use the VOID button when you don't want to print the selected mesh. In case of overlapping paths, you can decide extruder priority. This is the tool that will let you print multi color parts.
- Scale by X: scale the object by the user supplied value
- Inch-mm: same as entering 25.4 to Scale by X
- Count: to insert copies of the selected part

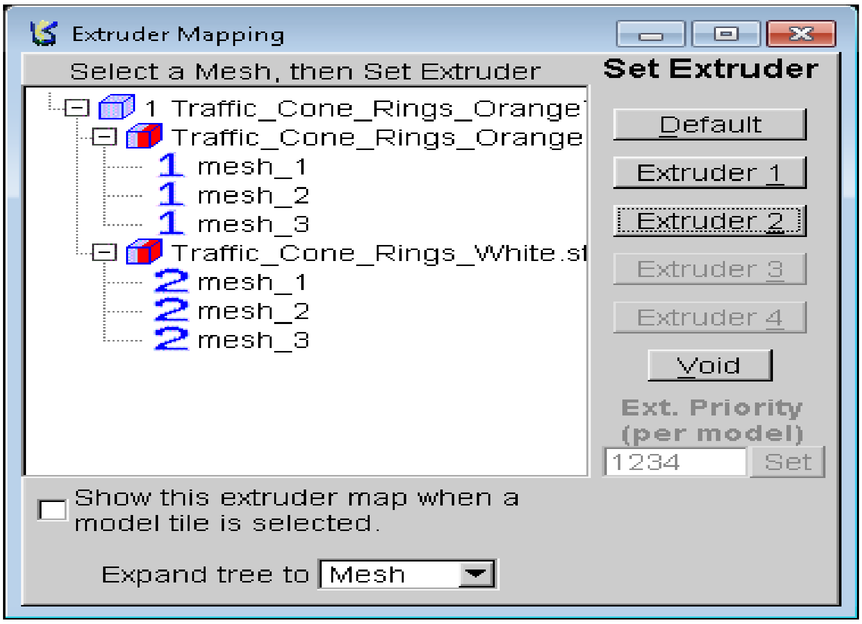

Fig. 52: Show extruder map

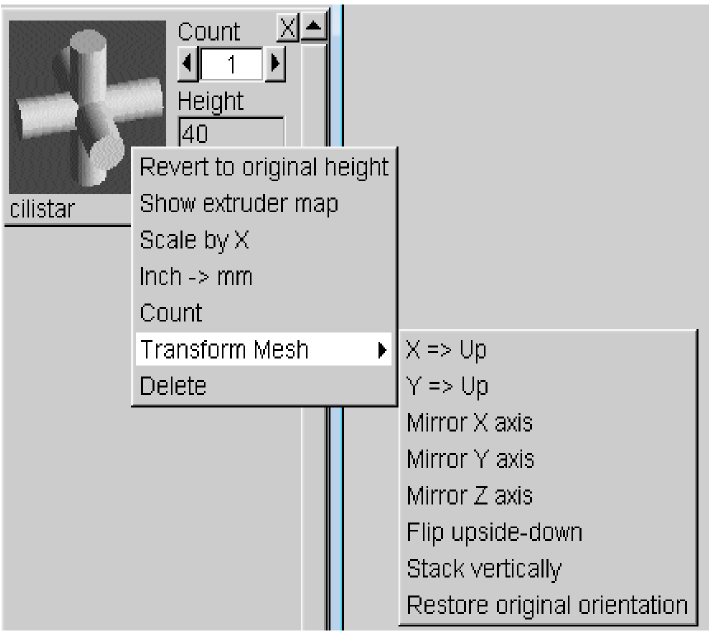

Fig. 53: Transform mesh pop-up

- **Transform mesh: will bring up a popup window allowing some basic part orientation:**
  - **X=>UP / Y=>UP will orient the part in a way that desired part axis (X or Y) will be aligned to the Z print area axis**
  - **Mirror X/Y/Z : will reflect the part on selected axis. Think of it as a geometric reflection tool. Be careful with this tool: if your part has some letters on it..those will be reflected too, making them not readable anymore.**
  - **Flip upside down: if you part is sitting on the printing plate the wrong way, this is the tool to use to remedy it.**
  - **Stack vertically: you can make multiple copies, stacked. Useful when you are printing large, flat parts and want to print several copies in a single batch.**
  - **Restore original orientation: if you lost track of the situation, this is your panic button.**
- **Delete: to delete selected part**

Calibration

To get good consistent good quality prints you need to periodically check your printer calibration, especially the filament-related ones: keeping this habit you will spare yourself waste of time and material.

You need a caliper, better if digital, and being able to use it correctly. May sound funny, but it isn't: there are plenty of tutorials on the Net about correct measure procedures.

IMPORTANT: the calibration sequence must be performed following the below sequence. Failing to do so, you may not be able to get to desired result.

Firmware parameter correctness is the key to printer behavior: inserting the wrong values may lead you to unwanted results, maybe you could even damage the printer itself!
Therefore PLEASE be sure to dutifully check the entered values and be sure to follow procedure with the following sequence!

Before going on with the calibration, be sure to have

1. A digital caliper: nowadays you can buy a good quality one for 20-25 $.
2. Information about the filament you are going to use: printing temperatures, preferred printing surface, recommended layer thickness and print speed
3. Your printer user manual
4. Printing plate leveled with the nozzle

The procedures are targeted to a medium-experience user, able to perform basic duties: if you are a complete beginner, you should seek expert advice.

1 – Filament feed check

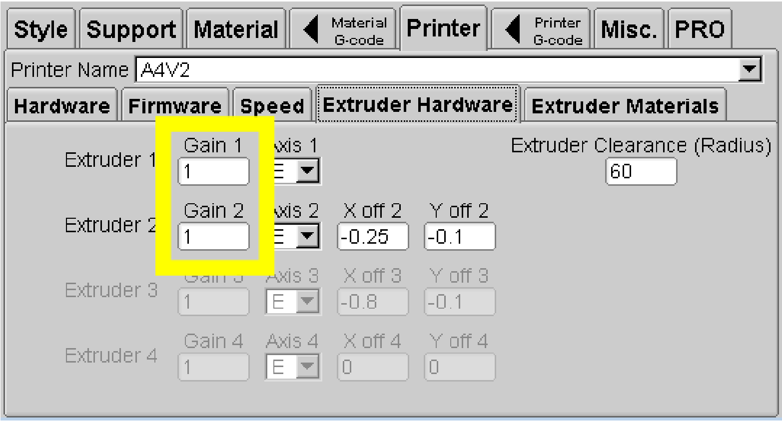

Fig. 54: Extruder gain

Print precision is first and foremost depending on precision of the filament feeding mechanism: the check basically consists of feeding some known quantity (i.e.:100mm) then checking the actually fed quantity.
Two possible approaches: changing (if possible) the firmware parameters, or changing the extruder gain parameters (Printer->Extruder hardware tab) inside KISSlicer.

Procedure detail:

- ◻ Take the provisions to let you precisely measure the fed filament. Procedure is different for each printer, basically consists into detaching the feeding unit from extruder (it may be trivial with bowden-like machines, or really cumbersome with direct-drive extruders).Therefore dutifully follow your printer documentation, to avoid costly mistakes.
- ◻ Feed 100mm
- ◻ Precisely measure the effective measure of the filament fed. If there is a measurable difference, you need to change existing setup with following formula:

  NG = OG * (VT / NM)

  NG = new gain
  OG = old gain
  VT = theoretical measure
  NM = effective measure

  Should you want to change the firmware parameters, change above “gain” with “step/mm”
- ◻ Update the gain value of selected extruder, in KS text box (“Printer->Extruder hardware” tab)
  Should you decide to change firmware parameter, follow your printer instructions.

2 – Filament diameter

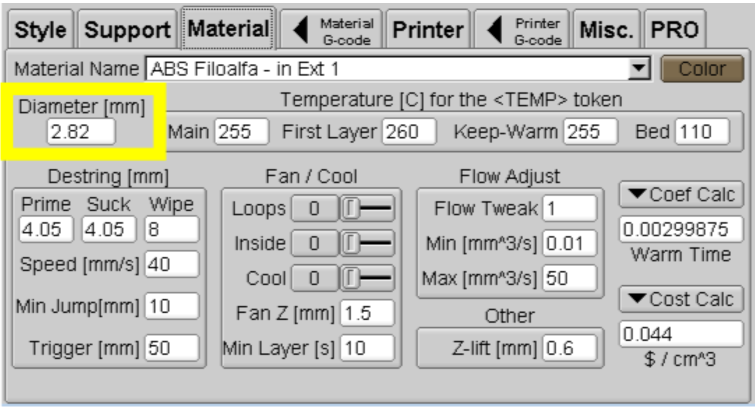

Fig. 55: filament diameter

This procedure should be done as often as possible, especially with cheap consumables, since diameter may change consistently into a single spool.
Use a caliper to check X and Y diameter, every 20-30 cm, along at least two meter length of the filament, find the average diameter.
Set the DIAMETER (Material tab) to the average found.

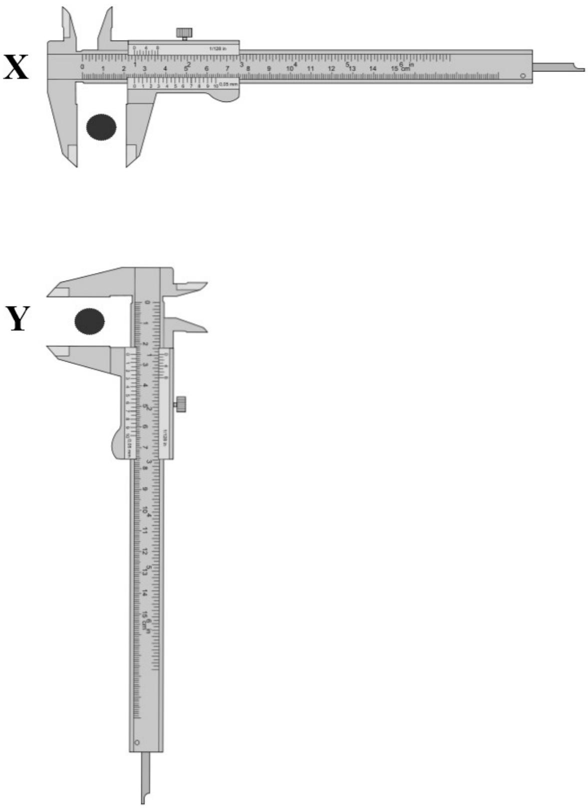

Fig. 56: X and Y diameter measurement

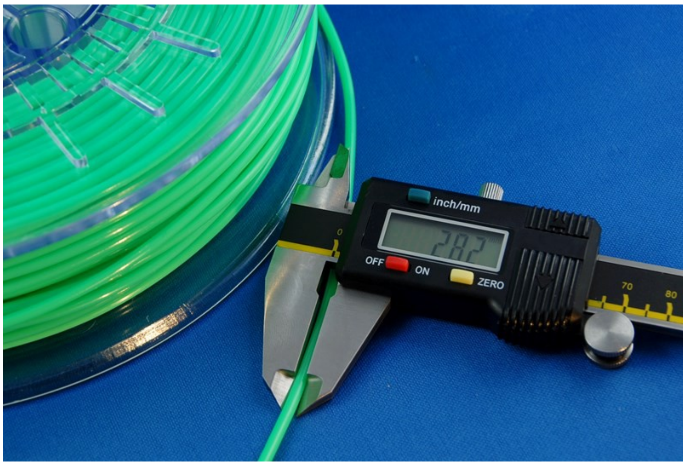

Fig. 57: A digital caliper will improve your life !

3 – Z offset

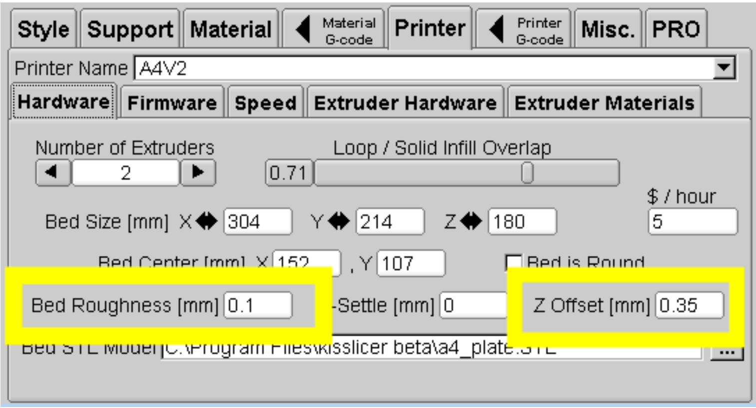

Fig. 58: Z offset and bed roughness

First layer thickness is fundamental and must be tuned as precisely as possible to get optimal print results:

- If nozzle is too far away from plate, the print will be brittle, infill will show gaps and adhesion to print bed will be poor: print will warp or even fully detach from the plate.
- If nozzle is too close to plate the extrusion will be "trapped": printed parts will lack definition and flat surfaces will look rough and lack definition. When the clearance is really low the risk of nozzle jam is quite high.

Fig. 59: Z offset too high, nozzle too far away from plate

To find the correct z-offset parameter is wise to start with very high values (0.5 – 1 mm) to avoid hitting the print bed with extruder nozzle(s), slowly decreasing it until prints are looking OK.

When plate flatness is not perfect, the "bed roughness" parameter can improve the situation: increasing its value will make for a thicker first layer, that will fill the possible voids.

Do note that if z offset value is correctly set, it will be easier to get desired precise part thickness.

Fig. 60: Perfect Z offset

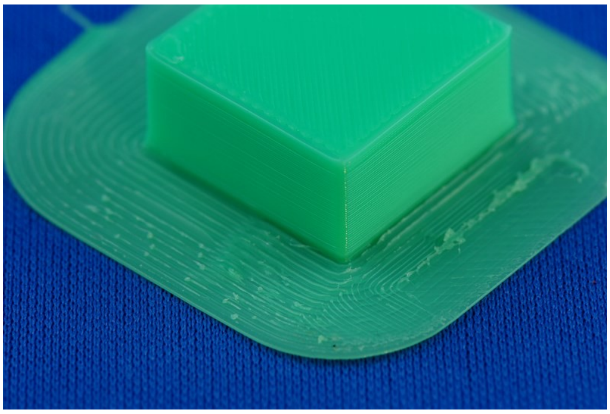

Fig. 61: Z offset too low: nozzle dragging on print

4 – Extrusion width

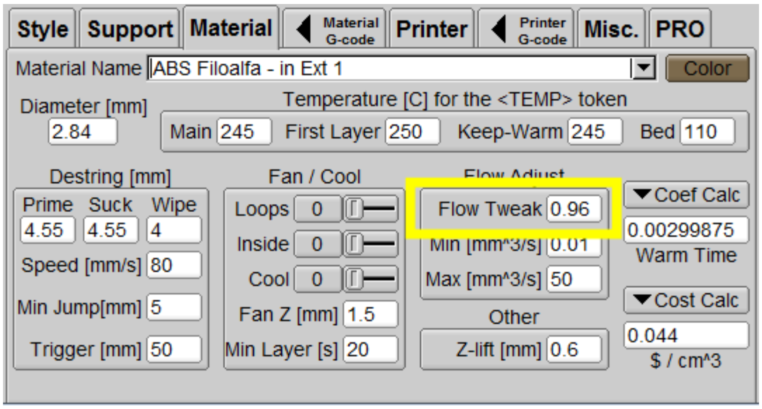

Fig. 62: Flow tweak

If you print a simple object using “vase” infill style (i.e.: [20x20x10 box]) with a single loop, it will be easy to check the extrusion width.

Use the recommended following settings to get a good test part:

- Any layer thickness (max 60-70% nozzle diameter)
- Extrusion width = nozzle diameter
- Num loops = 1
- Skin thickness = extrusion width
- Inset surface = 0
- Vase infill

Check the cube wall thickness, using a caliper, measuring the value at the middle of the edges.

If the thickness measure isn't equal to the theoretical value (and you are sure bout the filament diameter and feed calibration) you need to change the “Flow tweak” parameter (found in the Material tab): the default value (1) must be changed according the logic (i.e.: if measure is 0.5 instead of 0.55 expected, you must increase 10% the flow value, therefore new flow tweak value will be 1.1).

Print again and verify the results.

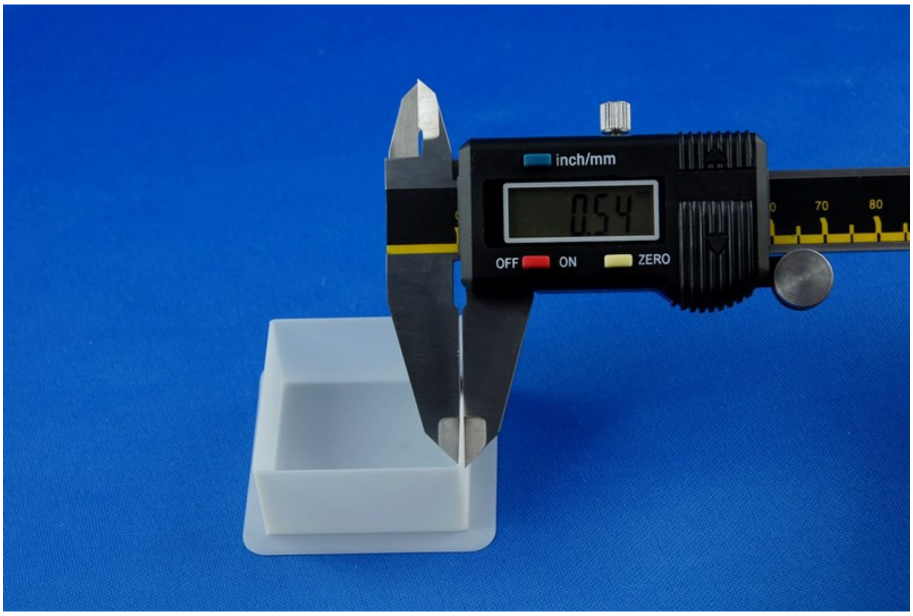

Fig. 63: Extrusion width measurement

5 – Dimensional accuracy

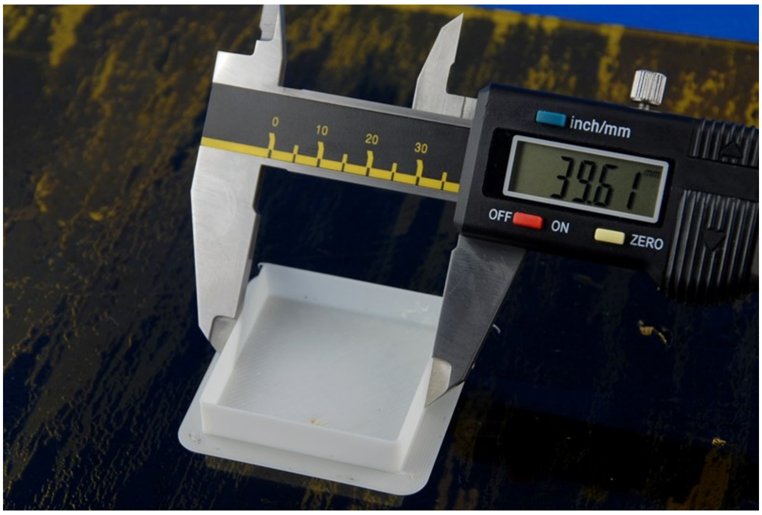

Fig. 64: Measure the printed part size along each axis

Dimensional accuracy is important, and achieving good results isn't hard, if you follow few easy steps.

Distinction should be made between accuracy and shrinkage: the first is due to the mechanical characteristics of the printer, and the latter happens due to the physical / chemical features of the polymer in use.

Dimensional accuracy can be checked precisely with just a few layers (2-3mm) while shrinking can happen immediately or some hours/day after printing.

IMPORTANT: as dimensional accuracy is HIGHLY dependent on the polymer used, you should do this test ANYTIME you change polymer (or even same polymer, different color) if you are really serious about precision. When dealing with large parts, the shrinkage may be changing abruptly as you modify any printing parameter

Procedure:

1. Select an easy part such as the 20x20x10 box
2. Calc&save the gcode, using an appropriate Style setup, insuring that
   - "inset surface" is set to zero
   - "loops go from inside to perimeter" switch is deselected (style tab)
3. If polymer in use is prone to warping, be sure to select an infill value >30% (it will make for sturdy prints, without deformation that will disrupt the precision)
4. Print the part, 2-3mm thickness may be enough. Don't remove the part from printing bed.
5. Measure along the chosen axis, avoiding the corners (that may be a bit thicker due to sharp turns and extrusion irregularity)
6. If measure doesn't match theoretical value, you can follow two approaches, each one with advantages and disadvantages that must be considered by the end user (it means YOU):
   1. Scale the part you must print according to the shrink factor found. Take care about the scaling method, as usually Z is quite precise while the real shocker is about X and Y shrinking.
   2. Change** the STEP/mm value according to the following formula

      NG = OG * (VT / NM)
      NG = new step/mm value
      OG = old step/mm value
      VT = theoretical measure value
      NM = measure of the printed part
7. Whatever is your chosen approach, print a new part to verify the set up.

**Altering firmware parameter is a job that must be done (if user is allowed to do so) strictly following the instructions from the user's printer model.

6 – Nozzle alignment

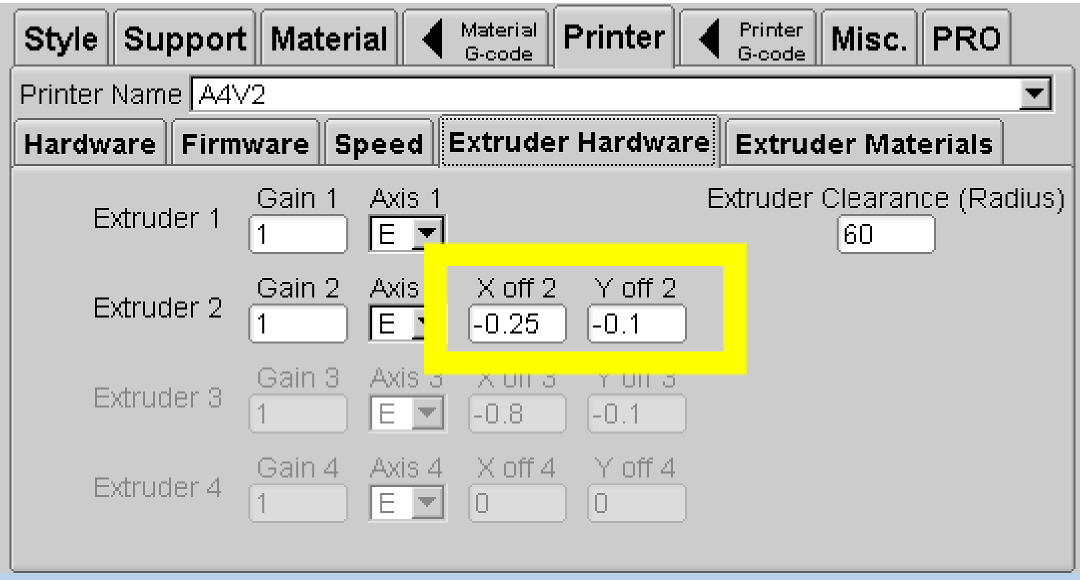

Fig. 65: Extruder offset

If your printer has more than one nozzle, you want KS to know as precisely as possible the x-y distance between each one.

Failing to do so, you could get unwanted results, resulting from overlapping extrusions or unwanted gaps in your parts.

To speed up the alignment, you can [download from 3ntr website a multi mesh object]() that has been designed to give immediate feedback about numerical displacement along x and y axis.

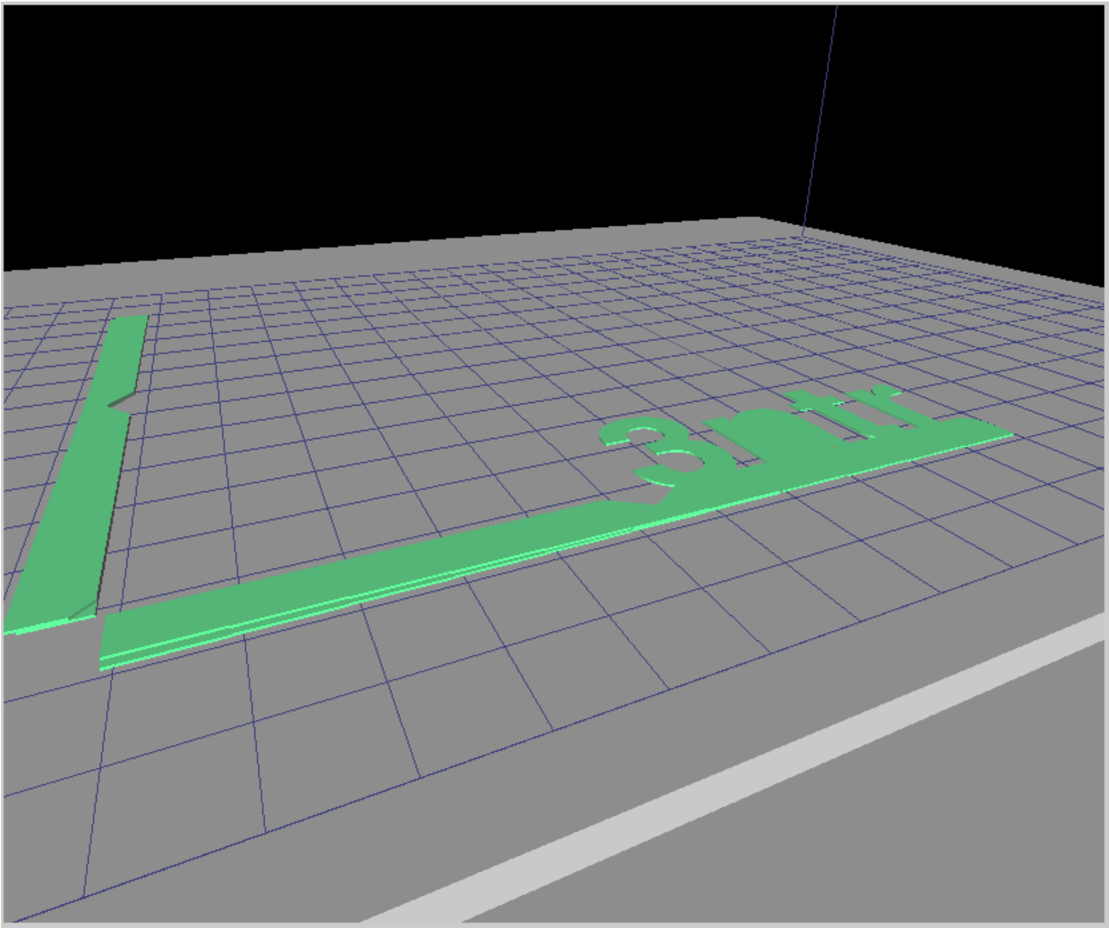

Fig. 66: Nozzle alignment STL

Load the files as a single part (hold SHIFT while selecting filenames) then select the EXTRUDER MAPPING option (right-click on the STL part picture). You will be shown a situation like the one in Fig.67:

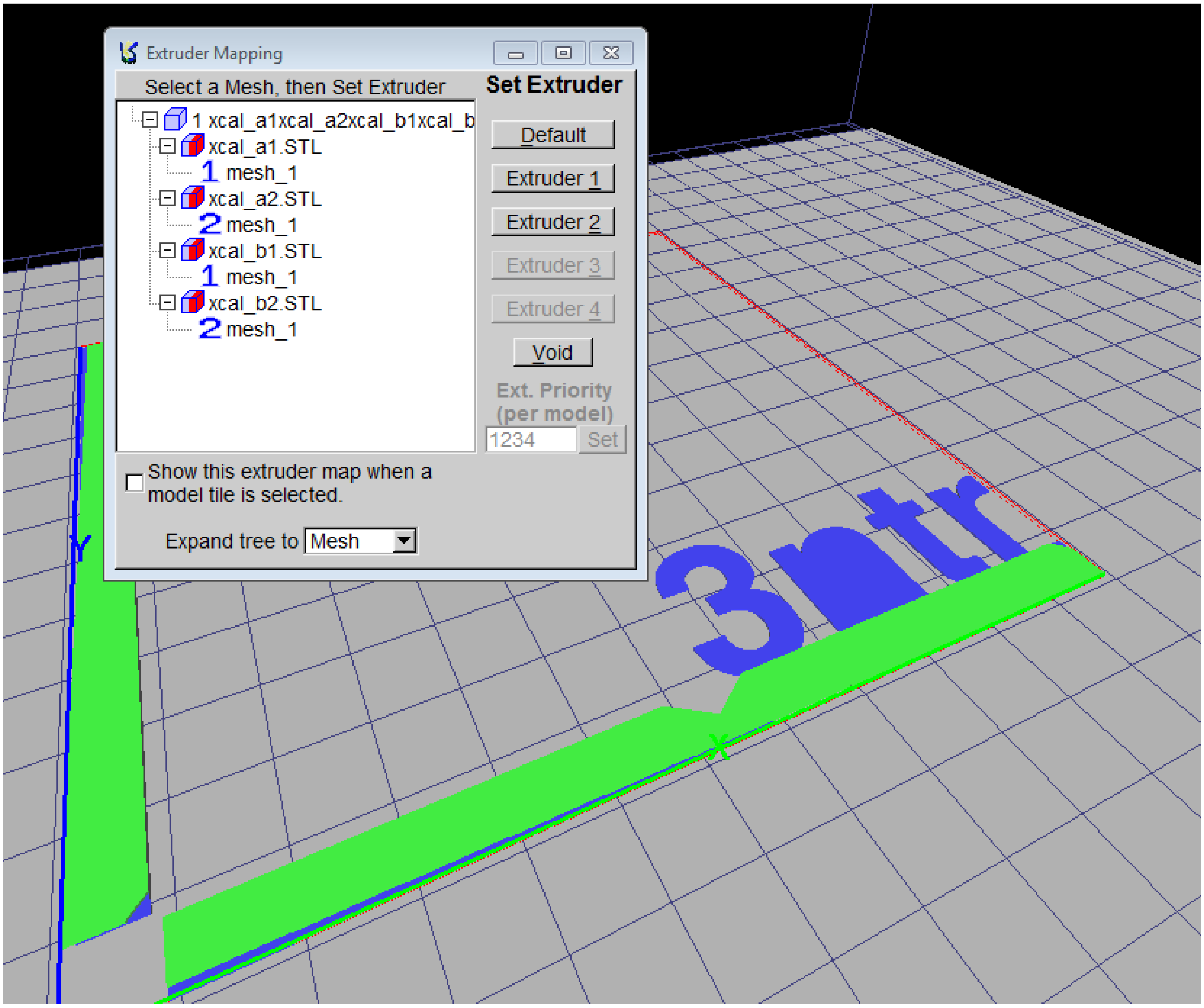

Fig. 67: Extruder mapping

If you look carefully, you will see that parts show some kind of “steps”, each one 0.1mm high and 10mm long.

Those parts are reversed and superimposed to give you a quick indication of what is going on: if the parts stairs are aligned at the center (where the “V” shaped cut is) will mean perfection. If the parts are aligned elsewhere, you just need to count the steps from the center to understand how many tenths of mm the displacement is.

To correct the situation you can either change the KS parameters or the printer firmware (it it is possible).

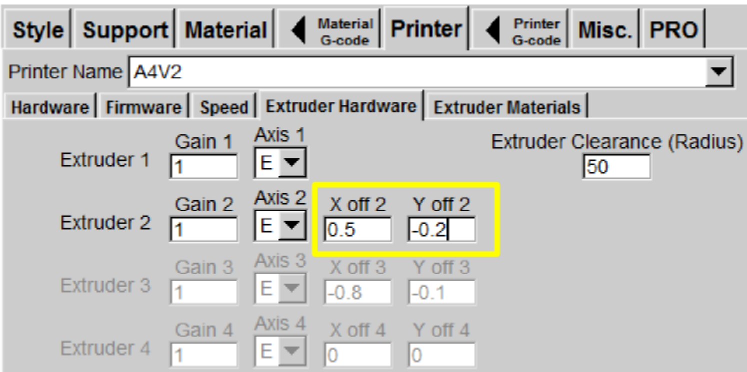

Fig. 68: Offset correction

Fig. 69: Test print

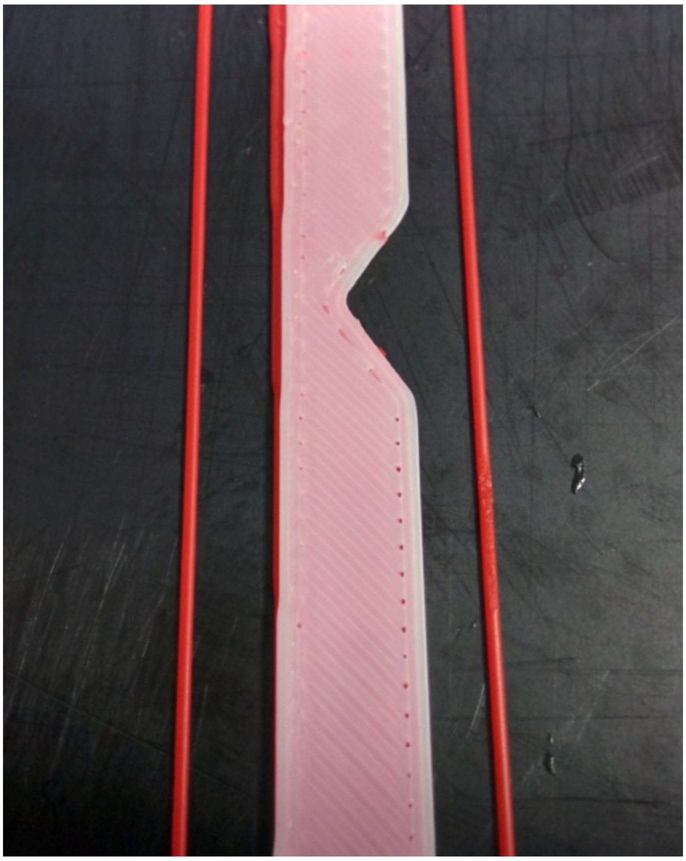

Fig. 70: detail of X calibration part

You can see below an enlarged portion (Fig70) of the test print (Fig69): you can easily understand that the extruders alignment along X axis is not perfect.

The “steps” are aligned on the lower part of the picture, three steps away from the center (where the “V” shape is): therefore you must correct the displacement of the extruder no.2 (printing white material) and add 0.3mm to the X offset (to correct the left-right alignment).

Same approach will be needed into evaluating the horizontal part (the one with “3ntr”) that will help you align along the Y (up-down) axis.

For best results, it's advisable to use a printing profile with 0.3mm thick layers (giving you just one layer per color) and assuring to have a good calibration of the materials used.

Index

Picture index