KISSlicer
Keep It Simple

1.1.1.1

KISSlicer v2

User Manual

KISSlicer
Keep It Simple

Contents

1.0 3D Printing Basics 7

1.1 The Fused Filament Fabrication Process 7

1.2 Definitions 8

2.0 Installation 8

2.1 Profiles 8

2.2 Wizards 9

2.2.1 Printer Wizard 9
2.2.2 Material Wizard 10
2.2.3 Style Wizard 10
2.2.4 Support Wizard 10

3.0 KISSlicer Interface Overview 10

3.1 Menu and Open 10

3.2 Print Settings 11

3.3 Slice and Save G-Code 11

3.4 Model List 11

3.5 Viewport 12

3.6 Viewport Settings 12

4.0 Menu 12

4.1 File 12

4.2 Project 14

4.3 Preferences 14

4.3.1 File Handling 15
4.3.2 Languages 15
4.3.3 Display Options 16
4.3.4 Warnings 18
4.3.5 Restrict Advanced Settings 18

4.4 All Models 18

4.4.1 Extruder Mapping 19

4.5 Wizards **20**
4.5.1 Create Profile Wizards 21

5.0 Printer **21**

5.1 Printer Name **22**

5.2 Extruder Mapping **23**
5.2.1 Default Extruder for… 23
5.2.2 Advanced Extruder Options 24

5.3 Speed Settings **24**
5.3.1 Path 'Fast' Speeds (Lower quality) 24
5.3.2 Path 'Precise' Speeds (Slower) 25
5.3.3 Printer Speeds 25

5.4 Associated Profiles **26**

5.5 Extruder Hardware **26**
5.5.1 All Other Extruders 28
5.5.2 N:1 28

5.6 Firmware **29**
5.6.1 Firmware Type 29
5.6.2 File Extension 30
5.6.3 Mark Path Start/Stop 30
5.6.4 Include Comments 30
5.6.5 Fan On and Fan Off 30
5.6.6 Fan can do PWM 30
5.6.7 Max Nodes/s 30
5.6.8 Post-Process 30
5.6.9 G4 Dwell P is in Seconds 31
5.6.10 Arc Support (G2/G3) 31
5.6.11 Speed Units 31

5.7 Hardware **31**
5.7.1 Number of Extruders 32
5.7.2 Loop/Solid Infill Overlap 32
5.7.3 $/Hour 33
5.7.4 Z Step [mm] 33
5.7.5 Min Segment Length [mm] 33
5.7.6 Bed Size X [mm] 33
5.7.7 Bed Size Y [mm] 33
5.7.8 Bed Size Z [mm] 33
5.7.9 Bed Center X [mm] 33
5.7.10 Bed Center Y [mm] 33
5.7.11 Bed Is Round 33
5.7.12 Bed Roughness [mm] 33

5.7.13 Z-Settle [mm] 34
5.7.14 Z Offset [mm] 34
5.7.15 Bed STL Model 34
5.7.16 Printer Color Scheme Hint 34
5.7.17 Blackbelt-Style Printer 34
5.7.18 Final Z 35

5.8 Printer G-Code Editor 35
5.8.1 Prefix 35
5.8.2 Pre-Heat 35
5.8.3 Select New Ext & Warm 35
5.8.4 Warm Same Ext 35
5.8.5 Cool Same Ext 35
5.8.6 Cool & Retire Old Ext 35
5.8.7 N [*] Layers 36
5.8.8 Postfix 36
5.8.9 Pause & Res 36

6.0 Print Style 36

6.1 Style Name 36

6.2 Extrusion Control 37
6.2.1 Extrusion Width [mm] 37
6.2.2 # Loops 37
6.2.3 Skin Thickness [mm] 37
6.2.4 Sparse Infill 38
6.2.5 Advanced Style Options 41

6.3 Layer Control 42
6.3.1 Layer Thickness 42
6.3.2 Max Layer Thickness 42
6.3.3 Adaptive Options 42

6.4 Seam Hiding 43
6.4.1 Seam Angle 43
6.4.2 Jitter 43
6.4.3 Corners 43
6.4.4 Join-Loop 43
6.4.5 Seam Depth 44
6.4.6 Seam Gap 44

6.5 Improve Quality 44
6.5.1 Speed Slider 44
6.5.2 Monotonic Sweep 44
6.5.3 Iron 44
6.5.4 Travel 45
6.5.5 Inset Surface 45

6.6 Link Style 45

7.0 Support Style 45

7.1 Support Name 46

7.2 Prime & Purge 46

7.2.1 Prime/Purge Structure 47
7.2.2 Pillar Location 47

7.3 Support Density 47

7.4 Main XY Gap 47

7.5 Support/Object Interface 48

7.5.1 Also on Top 48
7.5.2 Make Solid 48
7.5.3 Z Band 48
7.5.4 Interface XY Gap 48
7.5.5 Z Gap 48
7.5.6 Bond 49
7.5.7 Angle 49
7.5.8 Stride 49
7.5.9 Width 49
7.5.10 1st Width 49
7.5.11 1st Z Gap 49
7.5.12 Raft Interface Layers 49

7.6 Inflate Support 49

7.7 Inflate Raft & Prime Gap 49

7.8 Raft Options 49

7.8.1 Upper Stride 50
7.8.2 Upper Thick 50
7.8.3 Upper Width 50
7.8.4 Lower Stride 50
7.8.5 Lower Thick 50
7.8.6 Lower Width 50
7.8.7 Raft Type 50

7.9 Sheathe 51

7.9.1 Mode 51
7.9.2 Z- Roof 52

7.10 Brim/Skirt 52

8.0 Material 52

9.0 Miscellaneous Settings **52**

10.0 Models List **52**

11.0 Viewport **52**

ts

2.0 3D Printing Basics

2.1 The Fused Filament Fabrication Process

Sometimes called Fused Deposition Modelling (FDM) this is the process of pushing a plastic filament (a) using an extruder (b) through a heated nozzle to create a 3D object (c). The slicing software (KISSlicer) generates the toolpath that the nozzle will follow to create the object. The build plate € moves down by the desired layer height as the printer finishes each layer. Some objects will require support material (d) to print successfully.

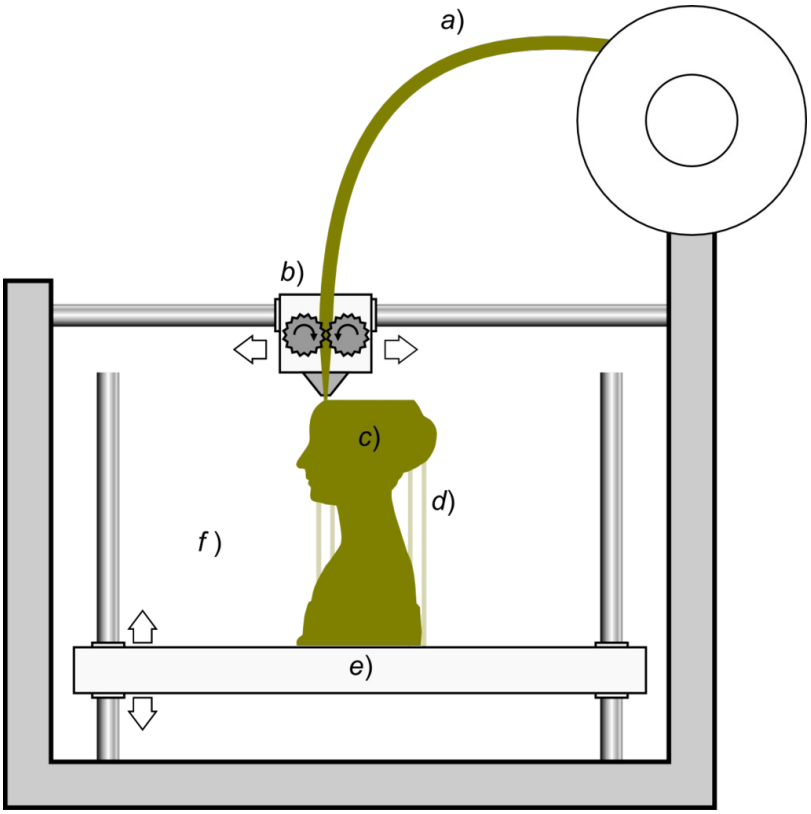


Figure 1 - The FFF printing process.

Scopigno R., Cignoni P., Pietroni N., Callieri M., Dellepiane M. (2017“. "Digital Fabrication Techniques for Cultural Heritage: A Surv”y". Computer Graphics Forum 36 (1): 6–21. DOI:10.1111/cgf.12781.

2.2 Definitions

- Slicing
  - The process of cutting a model into thin slices that are extruded one on top of the other to create the 3D model.
- Layer Height
  - The distance between the print nozzle and the build plate (for the first layer) or the previous layer.
- Extrusion Width
  - The width of the plastic extruded by the nozzle in a single pass.
- Perimeter
  - The outer surface of the printed part on each layer.
- Skin
  - The outer surface of the part on all sides (perimeter and top and bottom surfaces).
- Loops
  - The number of toolpaths around the perimeter on each layer. The perimeter is a single loop.
- Infill
  - Any material inside the part that is not considered perimeter.
- Mesh
  - An STL model
  - Multiple meshes can be opened together as one item in the model list
- G Code
  - Code used by the machine firmware to orchestrate the coordinated motion of the printer.
- M Code
  - Contained within the G Code file.
  - Typically used for setting variables in the code
- Raft
  - Layers of material (typically support) that are printed on the print bed. The part is then printed on top.
- Brim
  - The first layer of model material is expanded to provide better adhesion to the print bed.
- Skirt
  - An outline of model material that surrounds the part but does not touch it. Typically used for priming extruders.

3.0 Installation

Installing KISSlicer is a very simple process. Place the KISSLICER64.exe (or the KISSLICER.exe for older 32-bit systems) into a folder of your choosing and run it.

3.1 Profiles

Every aspect of the printing process can be saved as a different type of print profile. If you have material, printer, styles, or support profiles from another source you can paste them into the appropriate folders shown below. If you do not have any profiles then a series of setup wizards will guide you through the creation of your first profiles.

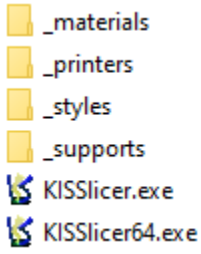
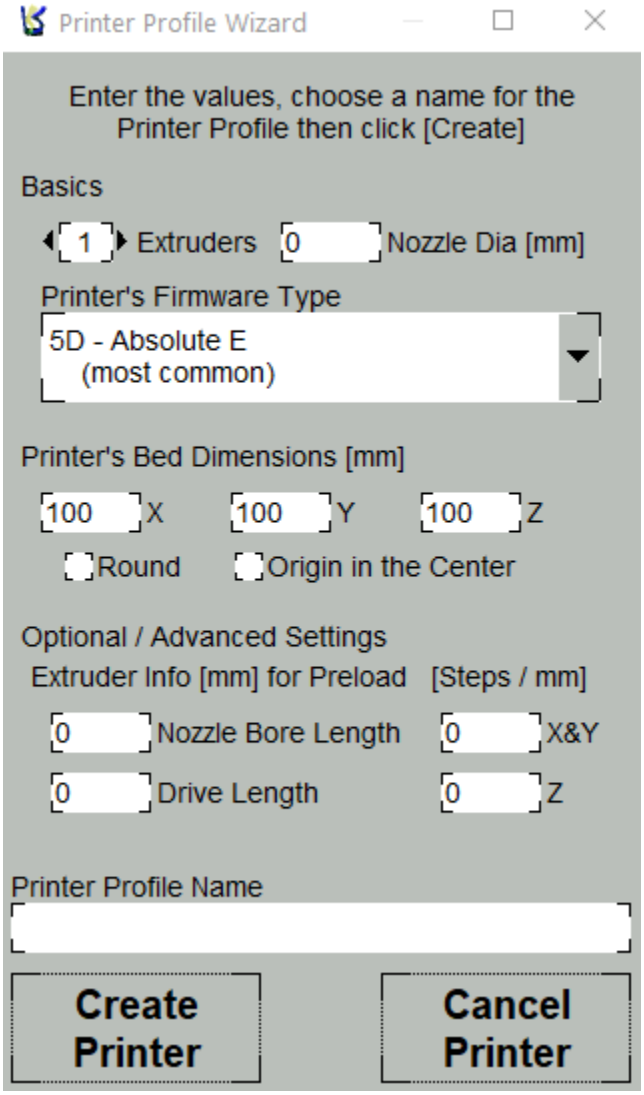


Figure 2 - Folder organization within the KISSlicer main folder.

3.2 Wizards

The wizards are designed to walk you through the creation of profiles for all aspects of the printing process.

3.2.1 Printer Wizard



3.2.2 Material Wizard

3.2.3 Style Wizard

3.2.4 Support Wizard

4.0 KISSlicer Interface Overview

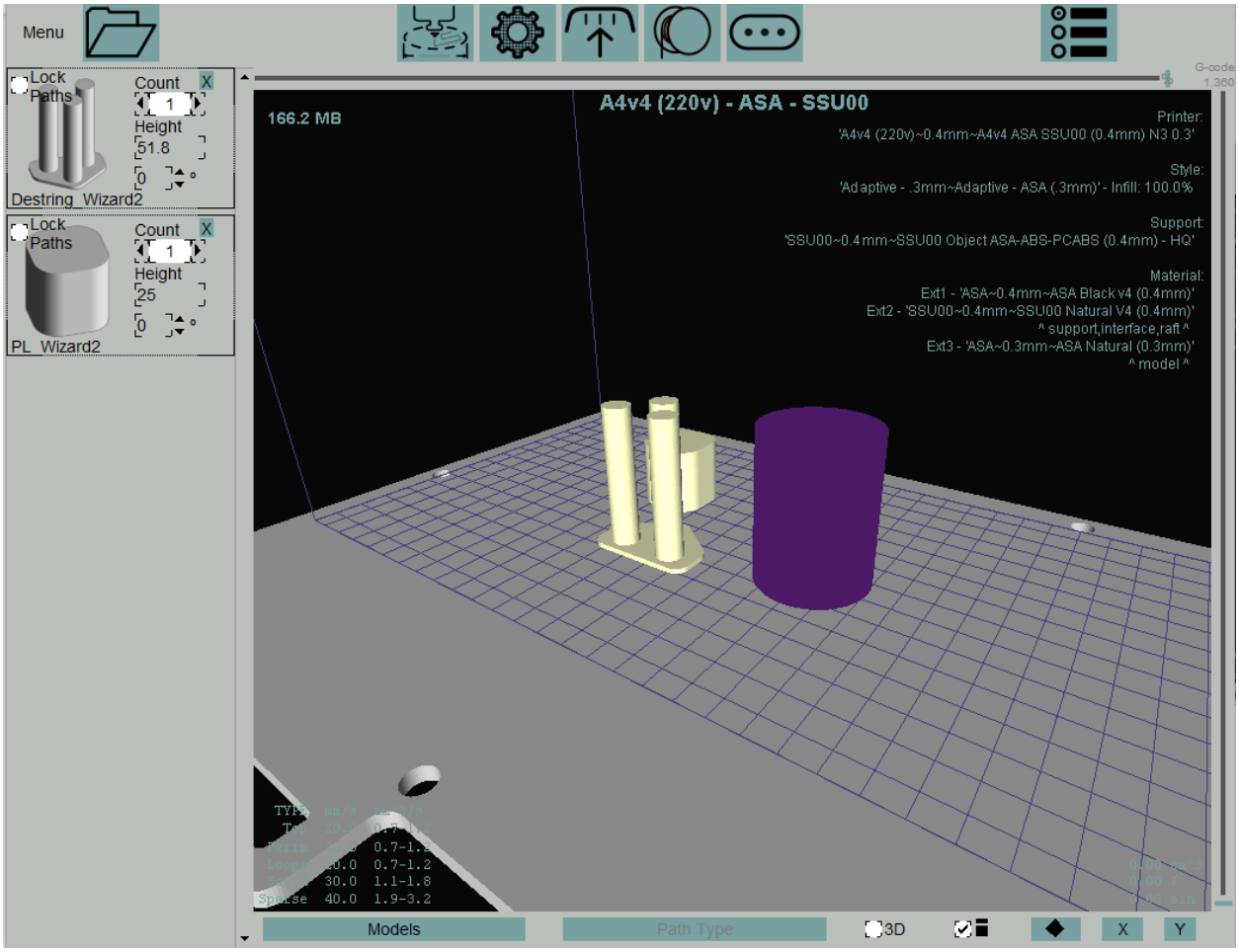


Figure 3 - KISSlicer main window.

4.1 Menu and Open

Clicking on “Menu” shows the main menu.

Clicking on the picture of the folder (right hand side) brings up the “Open STL” window.

Figure 4 - Menu and Open buttons from main window.

4.2 Print Settings

Figure 5 - From left to right: Printer Profile, Style Profile, Support Profile, Material Profile, and Miscellaneous Settings.

4.3 Slice and Save G-Code

The button shown in Figure 6 will change to the button shown in Figure 7 once the models have been sliced.

Figure 6 - Slice Models button from main window.

Figure 7 - Save G-Code button from main window. Only displays once the models have been sliced.

4.4 Model List

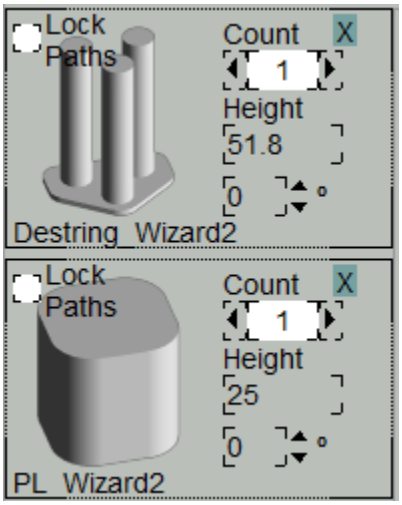


Figure 8 - List of all models currently open in KISSlicer.

4.5 Viewport

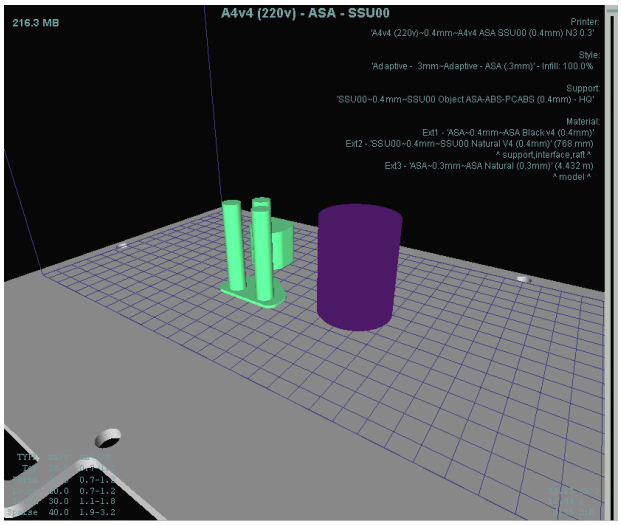


Figure 9 - Viewport that shows the models on the build plate. Also shows sliced models and all supports generated during slicing.

4.6 Viewport Settings

Figure 10 - Controls what is displayed in the viewport.

5.0 Menu

When you click on the menu button the following drop-down menu will appear:

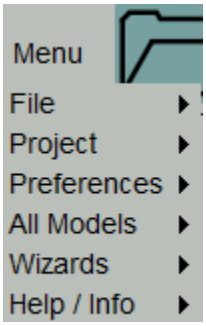


Figure 11"- "M"nu" drop-down menu.

5.1 File

The File menu has the following options:

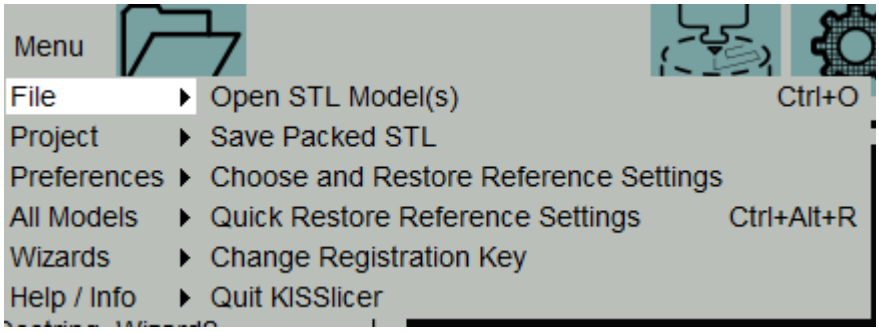


Figure 12 - File menu.

- Open STL Model(s)
  - Allows the user to open one (or more) STL model(s).
  - The top two models shown in Figure 13 were opened individually.
  - The bottom file shown in Figure 13 is the result of opening the two models together.
    - Multiple models will be opened as one item in the model list and the parts will be placed according to their individual coordinate system.
    - This behaviour can be changed in the preferences menu

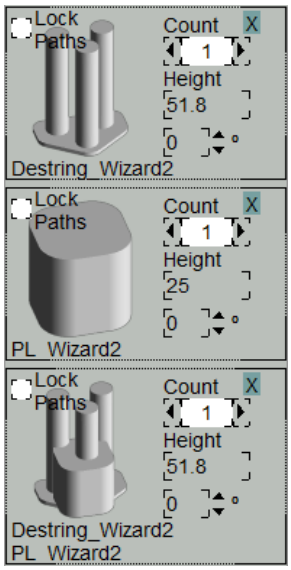


Figure 13 - The top two parts were opened individually, and the bottom part is made up of two files opened together.

- Save Packed STL
  - Saves the entire build plate layout as a single STL file.
- Choose and Restore Reference Settings Settings
  - Replaces all profiles with the ones found in user selectable folder.
- Quick Restore Reference
  - Replaces all profiles with the ones found in the Reference folder using shortcut keys
- Change Registration Key
  - Enter a license key to upgrade or change the current license level

- Quit KISSlicer
    - Closes the software

5.2 Project

The Project menu has the following options:

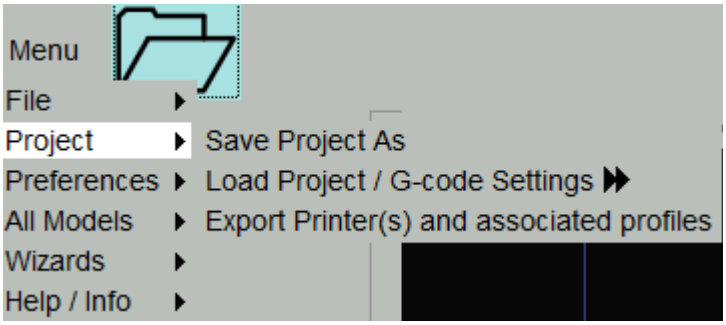


Figure 14 - Projects menu.

- Save Project As
    - Saves all profiles active in the current project to a *.ksp project file
- Load Project/G-Code Settings
    - Loads profiles from *.ksp file or a G-Code file
        - To load settings from the G-Code files “Include Comments” checkbox must be checked under the Firmware section of the Printer Profile
- Export Printer(s) and associated profiles
    - Allows the export of all profiles “Linked” to a specific printer or group of printers
        - Style, Support, and Material profiles can all be linked to a printer profile

5.3 Preferences

The Preferences menu has the following options:

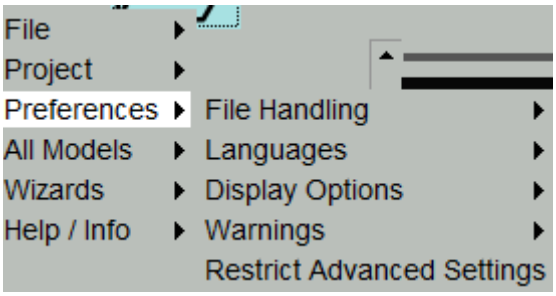


Figure 15 - Preferences menu.

5.3.1 File Handling

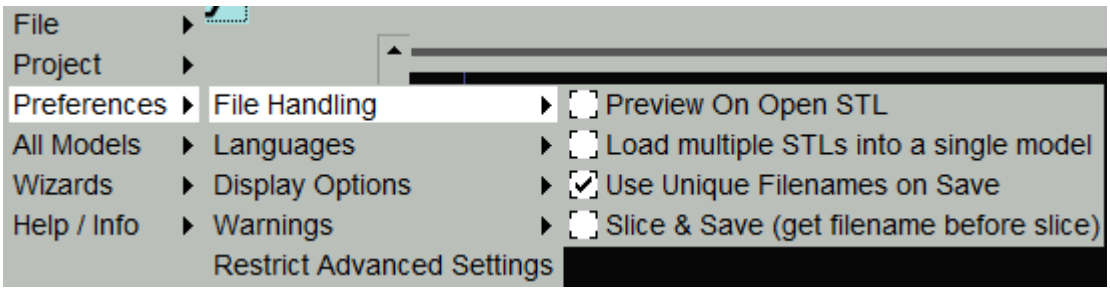


Figure 16 - File handling sub-menu under Preferences.

- Preview On Open STL
  - When checked KS shows a preview of the STL file selected in the "Open STL Model(s)" command.
- Load multiple STLs into a single model
  - When check multiple STL files will be grouped into a single item, as shown at the bottom of Figure 13, when opened.
  - If unchecked the files will all open as separate items.
- Use Unique Filenames on Save
  - When check this setting will add (0), (1), (2), etc to the filename on saving if another file of the same name already exists.
- Slice & Save (get filename before slice)
  - Opens the save dialog before slicing.

5.3.2 Languages

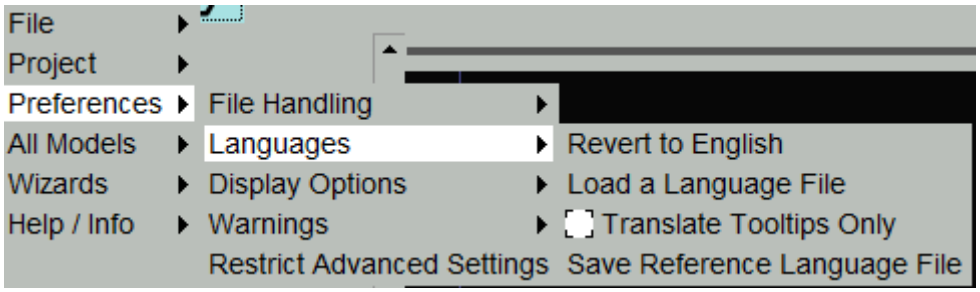


Figure 17 - Languages sub-menu under Preferences.

- Revert to English
  - Returns the software to its original English language interface.
- Load a Language File
  - Allows the user to load a translation file that will change the interface language.

- Translate Tooltips Only
    - Checking this box will only apply the translation to the tooltips that pop up when a user hovers their mouse over a setting.
- Save Reference Language File
    - Saves a reference file that can be used to create a “Language File” that can be imported by other users.

5.3.3 Display Options

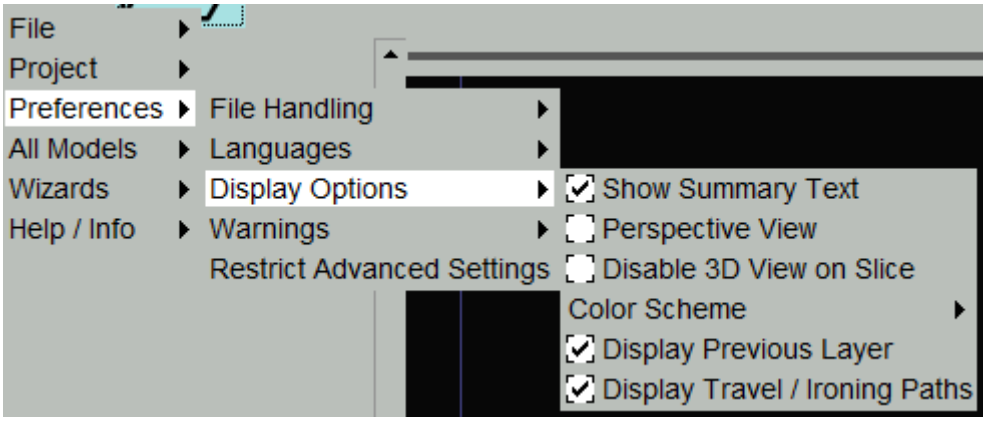


Figure 18 - Display options sub-menu under Preferences.

- Show Summary Text
    - Images below show the viewport with Summary Text enabled (left) and Summary Text disabled (right).

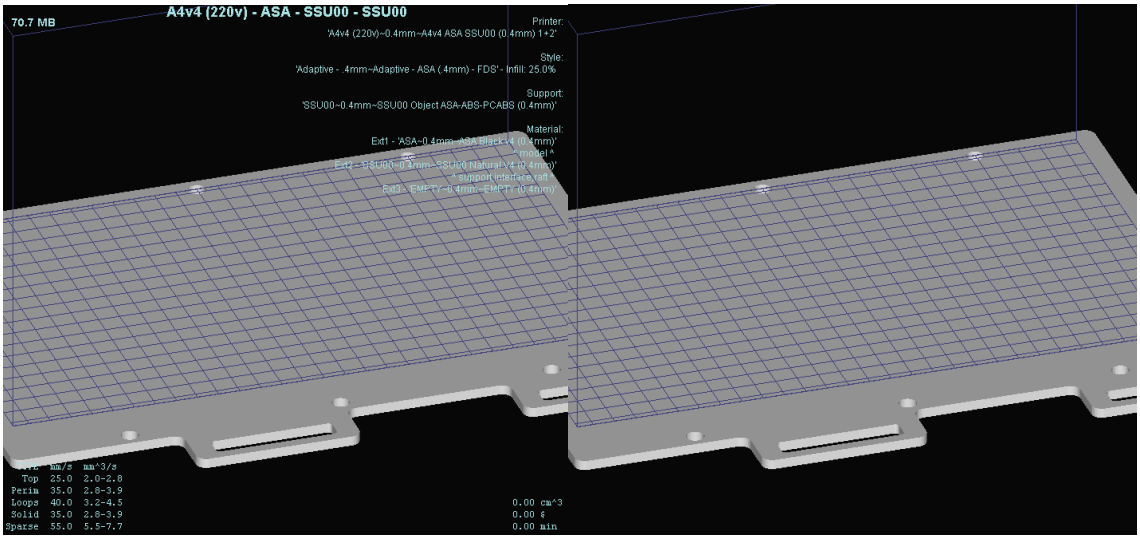


Figure 19 - Summary text enabled (left) and disabled (right).

- Perspective View
  - Toggles perspective view for the viewport.
  - Copies of objects further from the screen will appear smaller when enabled.
  - All copies of an object will appear the same size regardless of position when disabled.
- Disable 3D View on Slice
  - Turns off the viewport while slicing the models.
- Colour Scheme
  - Changes the appearance of the KS window.

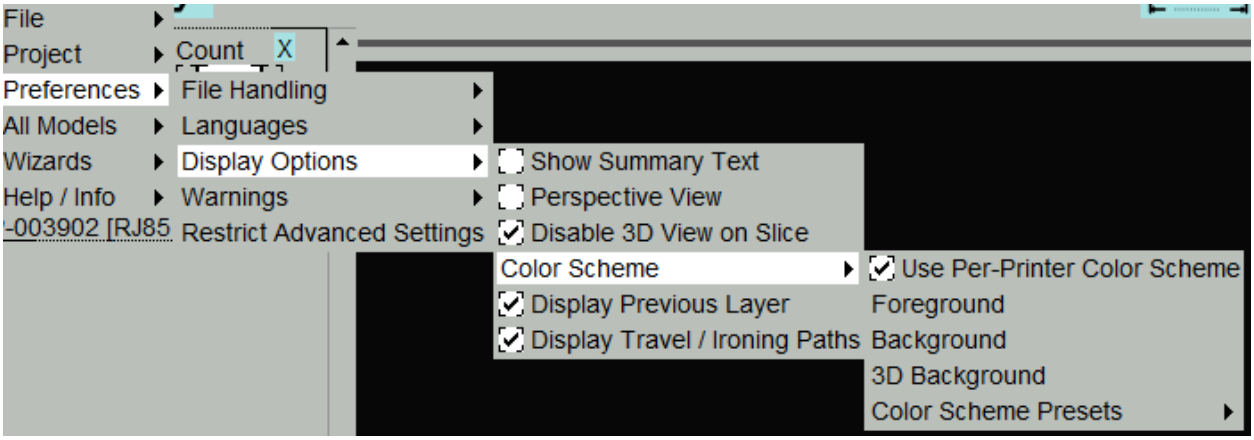


Figure 20 - Color Scheme Menu.

  - Use Per-Printer Color Scheme
    - The color scheme can be set for each printer (can be a useful visual reminder of which printer is currently active).
  - Foreground
    - Sets the color of all icons as well as the summary text in the main viewport.
  - Background
    - Sets the color of the main window background. Useful for visually differentiating different printers.
      - Only works if “Use Per-Printer Color Scheme” is not selected.
  - 3D Background
    - Sets the color of the viewport background.
  - Color Scheme Presets
    - A selection of predefined color schemes that can be applied.
- Display Previous Layer
  - Shows the current layer as well as the previous layer (in red below) when previewing the toolpaths with this option selected.

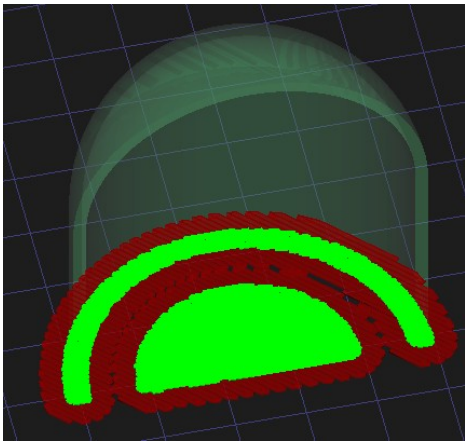


Figure 21 - Current layer and previous layer visible.

- Display Travel/Ironing Paths
    - Shows toolpaths used for traveling (non-printing moves) and ironing when previewing the toolpaths with this option selected.

5.3.4 Warnings

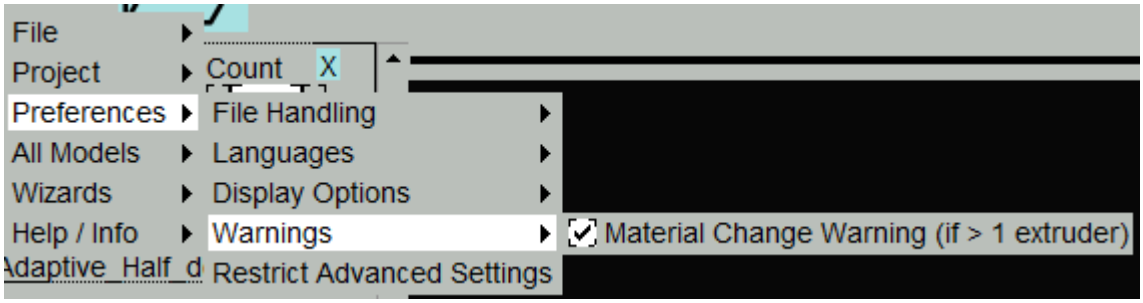


Figure 22 - Warning sub-menu under Preferences.

5.3.5 Restrict Advanced Settings

- Clicking menu option the first time will allow you to set a password to restrict access to the Advanced Settings.
    - More detail on Advanced Settings is available in section 10.0 Miscellaneous Settings.

5.4 All Models

The All Models menu has the following options:

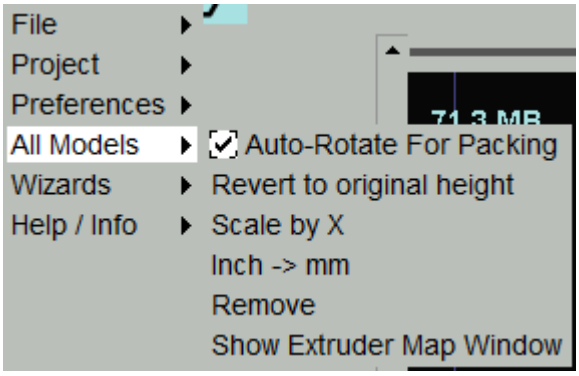


Figure 23 - All Models menu.

- Auto-Rotate For Packing
  - Models will be rotated to increase packing efficiency when checked.
- Revert to original height
  - Restores the original height of any models in the Model List have had their height altered.
  - More detail on the Model List is available in section 11.0 Models List
- Scale by X
  - Scales all models on the build tray by a factor.

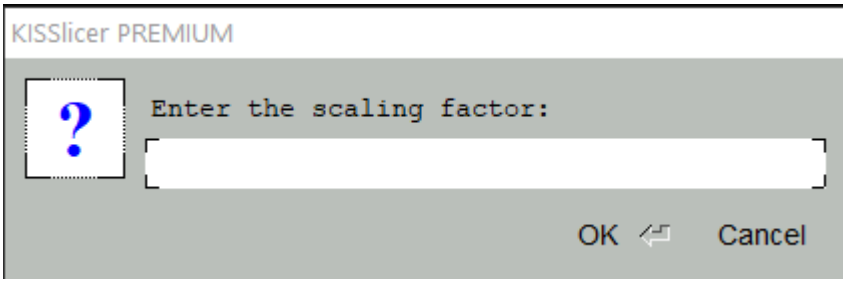


Figure 24 - Scale factor window.

- Remove
  - Removes all models from the build tray.
- Show Extruder Map Window
  - Brings up the Extruder Map Window.

5.4.1 Extruder Mapping

- Can be used to assign non-standard extruders to specific meshes/models.

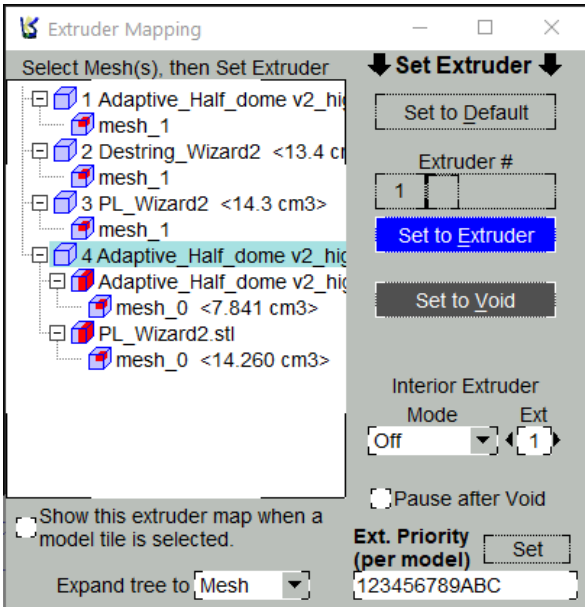


Figure 25 - Extruder Mapping Window.

- Select a mesh from the list and then set the extruder.
  - Select the extruder number and press the “Set to Extruder” button.
  - Select “Set to Void’ if a mesh should not be sliced and printed.
  - An extruder can be set for the interior of a model or mesh.
    - Mode can be set to:
      - Off – No extruder specified for interior.
      - Sparse – The specified extruder will be used for sparse infill only.
      - NonSkin – The specified extruder will be used for any toolpaths that are not designated as skin.
      - NonPerim – The specified extruder will be used for any toolpaths that are not designated as perimeter.
  - “Pause after Void” will pause the print after printing the top layer of a “Void” mesh to allow the user to insert an object mid-print.
  - Extruder priority is only used when multiple meshes, assigned to multiple extruders, overlap.
    - The extruder with the highest priority will be printed.

5.5 Wizards

The Wizards menu has the following options:

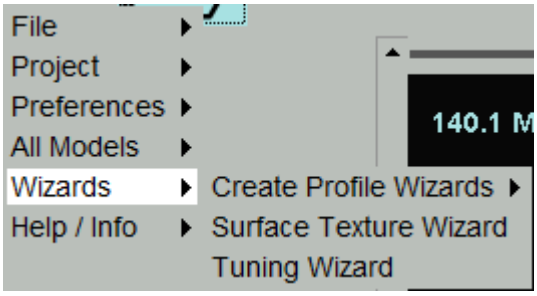


Figure 26 - The Wizards menu.

5.5.1 Create Profile Wizards

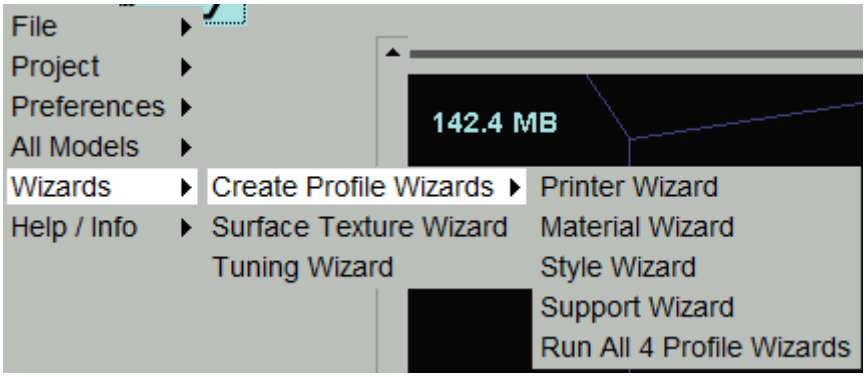


Figure 27 - Create Profile Wizards Menu

- Printer Wizard
  - Uses user input to create a printer profile (see 6.0 Printer for more details).
- Material Wizard
  - Uses user input to create a material profile (see 9.0 Material for more details).
- Style Wizard
  - Uses user input to create a style profile (see 7.0 Print Style for more details).
- Support Wizard
  - Uses user input to create a support profile (see 8.0 Support Style for more details).
- Run All 4 Profile Wizards
  - Runs all four profile wizards.
  - Generally used the first time KS is used with a printer.

6.0 Printer

Printers in KS are more than just individual devices. The same physical printer can have multiple printers set up in KS. The most common reasons to create multiple KS printers are:

- Different nozzle sizes
- Different materials

- Different extruder combinations

The Printer menu is accessed by clicking the icon below:

The menu appears on the right hand side of the screen:

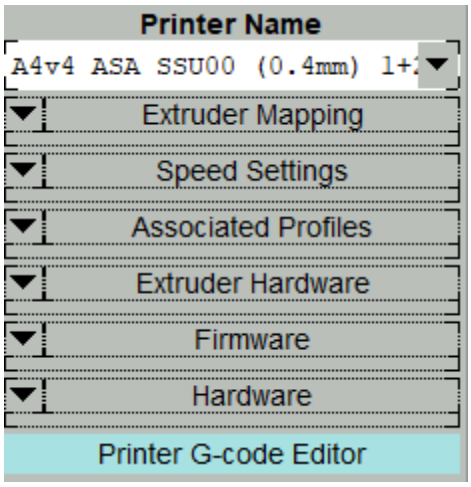


Figure 28 - Printer Menu

6.1 Printer Name

The Printer Name menu can be used to select from a list of printers. Setting up multiple printers for different material combinations, nozzle sizes, etc can help avoid configuration mistakes when frequently switching printer configurations.

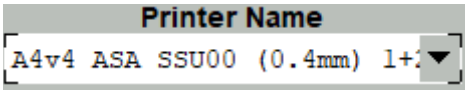


Figure 29 - Printer Name menu.

The menu can have multiple levels which can be used to sort the printers by nozzle size, printer type, or any other variable. The format for the name is shown below with each level separated by a ~:

Top Level~1st Sub Level~2nd Sub Level~…~Profile Name

An example is shown below and the outcome is shown in Figure 30.

A4v4 (220V)~0.4mm~A4v4 (220v) ASA SSU00 (0.4mm)

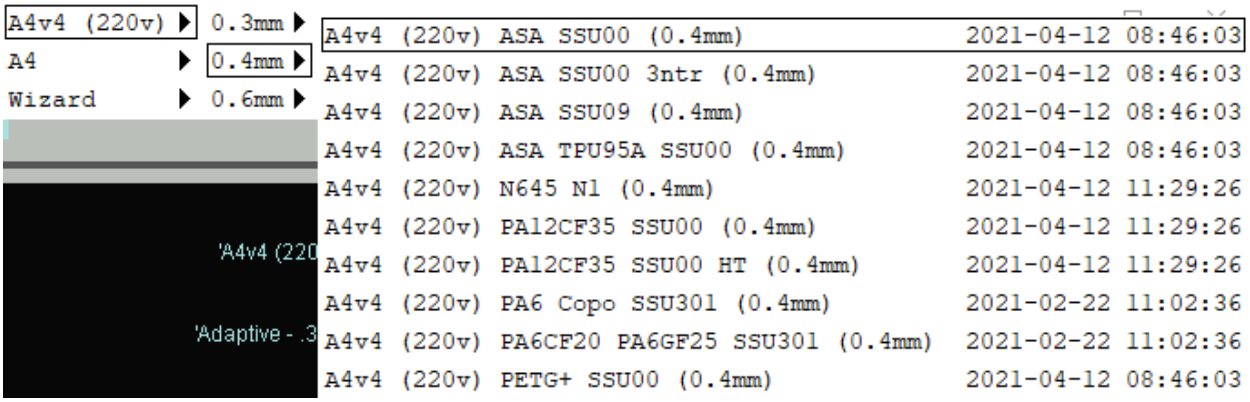


Figure 30 - Printer Name example.

6.2 Extruder Mapping

This menu allows a material profile to be assigned to an extruder.

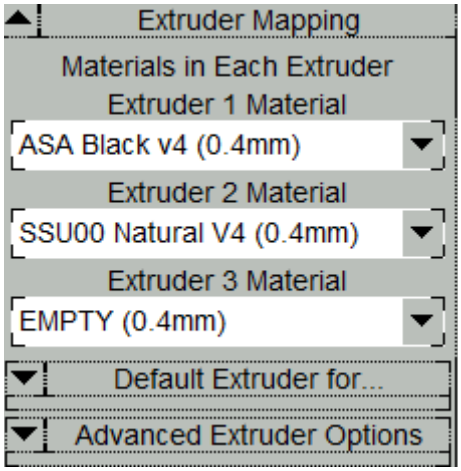


Figure 31 - Extruder Mapping menu.

6.2.1 Default Extruder for…

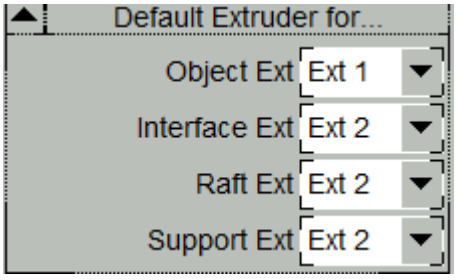


Figure 32 - Default extruder selection.

The Default Extruder for… menu sets the default extruder used for various parts of the print:

- Object Ext – The extruder used to print the model.

- Interface Ext – The extruder used to print the direct interface layer(s) between the support and the model.
- Raft Ext – The extruder used to print the raft on the build plate.
- Support Extruder – The extruder used to print any support material that is not in direct contact with the build plate or the model.

6.2.2 Advanced Extruder Options

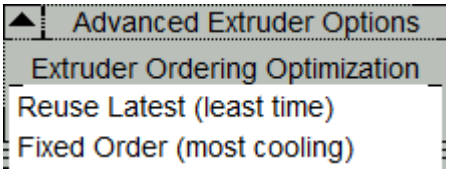


Figure 33 - Advanced extruder options menu.

The Advanced extruder options menu can be used to set the order in which extruders are used.

- Reuse Latest (least time) – The current extruder is reused for the next layer (Example extruder order 1-2-2-1-1-2-2)
- Fixed Order (most cooling) – Extruders are alternated in a fixed order (Example extruder order 1-2-1-2-1-2)

6.3 Speed Settings

The speed setting menu sets the printer maximum speeds for various moves as well as the minimum and maximum print speeds.

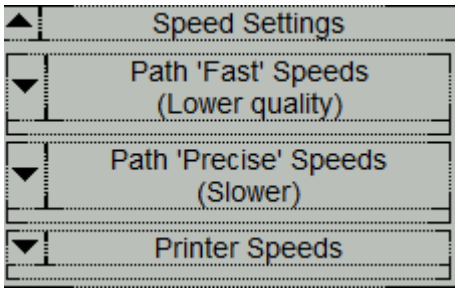


Figure 34 - Speed settings menu.

6.3.1 Path 'Fast' Speeds (Lower quality)

The settings in this menu are used as the highest speed allowed for printing. The units for these speeds are set in Section 6.6.11.

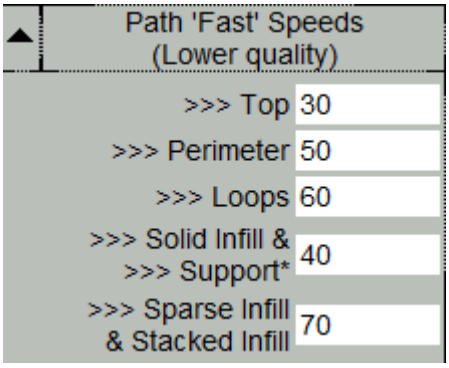


Figure 35 - Path 'Fast' Speeds menu.

- Top – Print speed that will be used to print the top surfaces of the model.
- Perimeter – Print speed that will be used to print the perimeter of the model.
- Loops - Print speed that will be used to print the loop toolpaths.
- Solid Infill & Support - Print speed that will be used to print solid infill and support toolpaths.
- Sparse Infill & Stacked Infill - Print speed that will be used to print sparse infill and stacked infill.

6.3.2 Path 'Precise' Speeds (Slower)

The settings in this menu are used as the lower speed allowed for printing. The units for these speeds are set in Section 6.6.11.

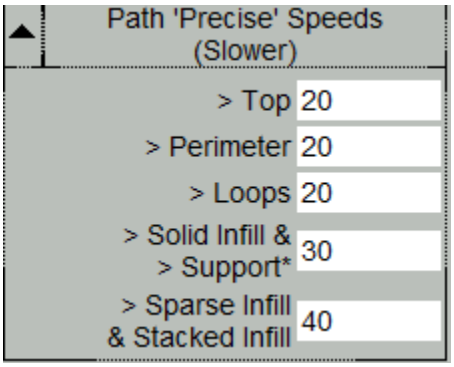


Figure 36 - Path 'Precise' Speeds menu.

- Top – Print speed will be used to print the top surfaces of the model.
- Perimeter – Print speed that will be used to print the perimeter of the model.
- Loops - Print speed that will be used to print the loop toolpaths.
- Solid Infill & Support - Print speed that will be used to print solid infill and support toolpaths.
- Sparse Infill & Stacked Infill - Print speed that will be used to print sparse infill and stacked infill.

6.3.3 Printer Speeds

The settings in the menu are used as the absolute maximum speeds for the toolpaths. The units for these speeds are set in Section 6.6.11.

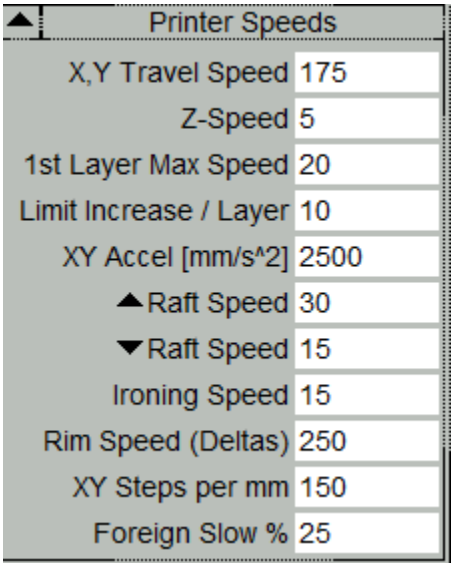


Figure 37 - Printer Speeds menu.

- X,Y Travel Speed – The maximum speed the printer can move in the X and Y axis.
- Z-Speed – The maximum speed the printer can move in the Z direction.
- 1st Layer Max Speed – The maximum speed allowed for a first layer toolpath.
- Limit Increase/Layer – The first layer max speed will increase by this amount every layer.
- XY Accel [mm/s^s] – The maximum allowed acceleration for any toolpath.
- ↑ Raft Speed – The maximum speed to travel for the raft top layer.
- ↓ Raft Speed – The maximum speed to travel for the raft bottom layer.
- Ironing Speed – The speed used for ironing toolpaths.
- Rim Speed (Deltas) – Used only for Delta-style 3D printers. Speed of head while not extruding.
- XY Steps per mm – The step pulses required to move the head 1mm in either X or Y. NO LONGER USED.
- Foreign Slow % - The percentage the head will slow if the toolpath from one extruder sits on the toolpath from another.

6.4 Associated Profiles

Useful for the export printer function. Any associated profiles will be exported with the printer.

6.5 Extruder Hardware

The settings in this menu are used to configure the extruder(s) on the selected Printer Profile. The main menu is used to configure Extruder #1 and the sub-menus are used to configure additional extruders.

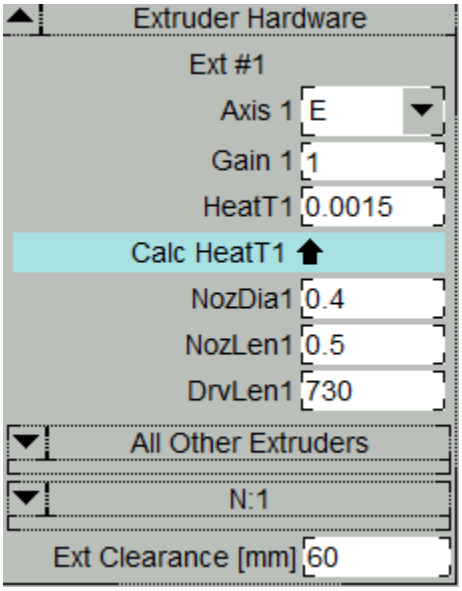


Figure 38 - Extruder Hardware menu.

- Axis 1 – Drop down menu for selecting the axis letter used in the G-Code.
- Gain 1 – Normally set to 1. Can be used to adjust for new gear ratios, etc.
- HeatT1 – Coefficient for estimating warm up time. Calculated using the button below.
- Calc HeatT1 – Brings up the window below.
    - Enter the desired setpoint in the "Extruder Temperature"
    - Enter the minutes the printer took to reach the desired "Extruder Temperature".
    - Click "Compute the Warm-Up" button to compute and enter the HeatT1 coefficient.

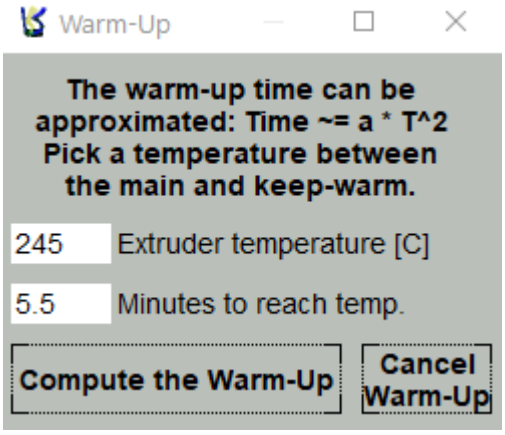


Figure 39 - Window that can be used to calculate the HeatT1 coefficient.

- NozDia1 – The nozzle diameter for Extruder #1.
- NozLen1 – The length of the nozzle throat.

- The throat is the length of the nozzle that is at the printing diameter.

- DrvLen1 – The distance from the drive wheels in the extruder to the nozzle.
- Ext Clearance [mm] – Clearance around the extruders used for sequential prints and the safe bed region for multi-material prints.

6.5.1 All Other Extruders

The settings in this menu are used to configure any additional extruders available on the selected Printer Profile.

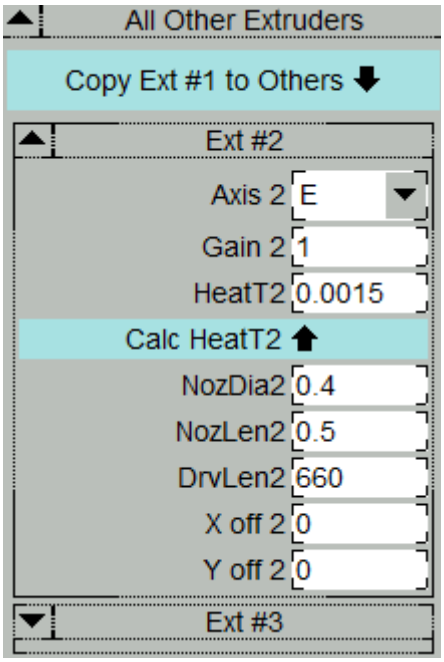


Figure 40 - All Other Extruders menu.

- Copy Ext #1 to Others – Copies the settings from Extruder #1 to all other extruders.
- Ext #2 – All of the same settings shown in Extruder Hardware 6.5 but with 2 additional settings.
  - X off 2 – The X offset of Extruder #2 relative to Extruder #1.
  - Y off 2 – The Y offset of Extruder #2 relative to Extruder #1.
- All additional extruders are the same as Ext #2.

6.5.2 N:1

Settings used for printers with mixing nozzles (multiple extruders feeding one nozzle).

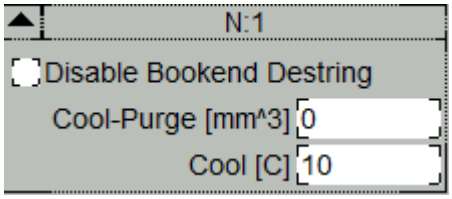


Figure 41 - N:1 menu.

- Disable Bookend Destring – Eliminating the prime and retract when swapping extruders for a single nozzle system.
- Cool-Purge [mm^3] – Heater turned off/down and material is purged. Helps eliminate stringing and is essential for systems like the Prusa MMU.
- Cool [C] – Temperature difference from standard printing temperature used in the Cool-Purge.

6.6 Firmware

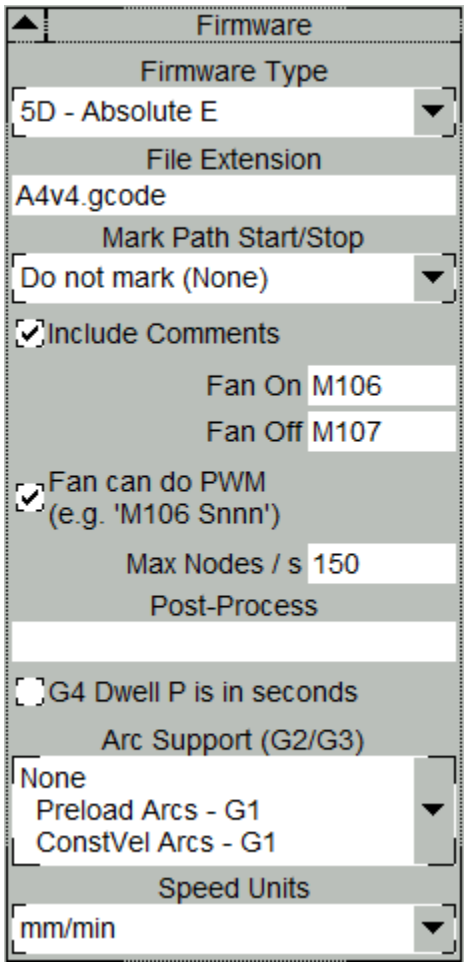


Figure 42 - Firmware Menu.

6.6.1 Firmware Type

The drop-down menu shows the various firmware types supported by KS.

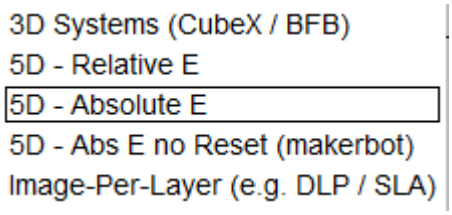


Figure 43 - Firmware Type drop-down menu.

- 3D Systems (CubeX/BFB)
  - Settings for firmware specific to the 3D Systems CubeX and BFB printers.
- 5D – Relative E
  - Settings specific to standard firmware with relative extruder moves (each extruder move is relative to the last move)
- 5D – Absolute E
  - Settings specific to standard firmware with absolute extruder moves (each extruder move is relative to start of the G Code.
- 5D – Abs E no Reset (makerbot)
  - Settings specific to firmware specific to Markerbot hardware.
- Image-Per-Layer (e.g. DLP/SLA)
  - Settings specific to Digital Light Processor and Stereo Lithography hardware.

6.6.2 File Extension

The file extension used for all G Code files created by KS.

6.6.3 Mark Path Start/Stop

Some printer firmware may require the start and stop of paths to be marked.

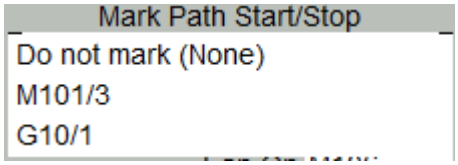


Figure 44 - Mark Path Start/Stop drop-down menu.

6.6.4 Include Comments

This checkbox adds comments to the G Code files.

- Allows the G Code settings to be imported back into KS (see Section 5.2).
- Makes the code much easier to read.

6.6.5 Fan On and Fan Off

Enter the M Codes used by the printer firmware that turn the part cooling fan on and off.

6.6.6 Fan can do PWM

Pulse width modulation (PWM) can be used to control the speed of the part cooling fan. Check this box to enable this feature if the printer firmware supports it.

6.6.7 Max Nodes/s

The number of points per second that the printer controller can process. Paths will be slowed so they do not exceed this limit. A value of '0' disables the limit.

6.6.8 Post-Process

A script file can be specified if post processing of the KS generated G Code files is required or desired.

6.6.9 G4 Dwell P is in Seconds

Check this box if the printer firmware interprets G4 dwell commands in seconds. The G Code standard is that G4 commands are in milliseconds.

6.6.10 Arc Support (G2/G3)

Traditionally 3D printer firmware has not supported true arcs. Typically, arcs are approximated by hundreds of tiny straight lines. This drop-down menu should be set to match the level of arc support in the printer firmware.

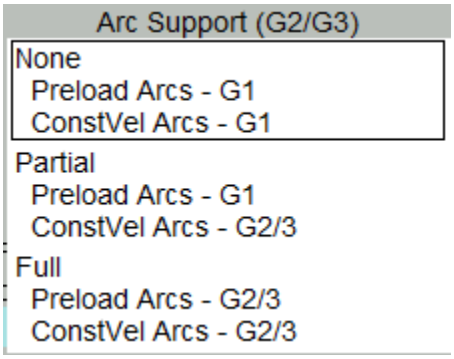


Figure 45 - Arc Support drop-down menu.

- None – The firmware does not support arcs at all. All move commands will be issued with G1.
- Partial – The firmware has limited support for arcs. Some move commands will use G2 and G3.
- Full – The firmware has full support for arcs and all curved paths will use G2 and G3 commands.

6.6.11 Speed Units

The speed unit combination used by the printer firmware.

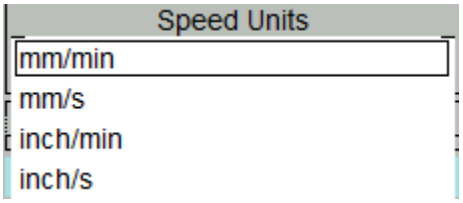


Figure 46 - Speed Units drop-down menu.

- mm/min – millimetres per minute
- mm/s – millimetres per second
- inch/min – inches per minute
- inch/s – inches per second

6.7 Hardware

Hardware settings for the selected printer.

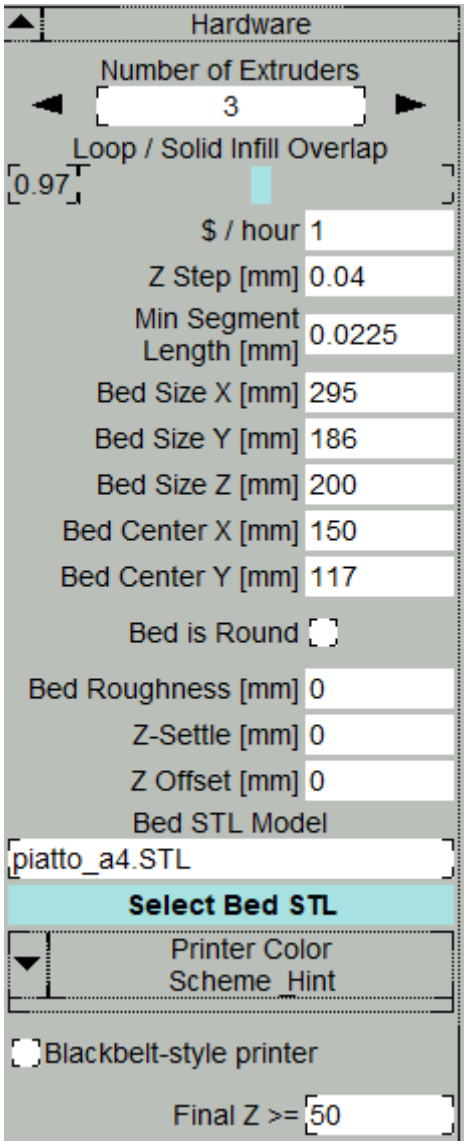


Figure 47 – Hardware Menu.

6.7.1 Number of Extruders

The number of physical extruders installed on the 3D printer.

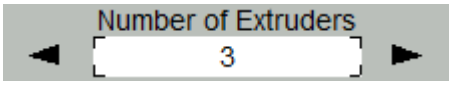


Figure 48 - Number of Extruders.

6.7.2 Loop/Solid Infill Overlap

The amount of overlap between the innermost loop and the sparse or solid infill.

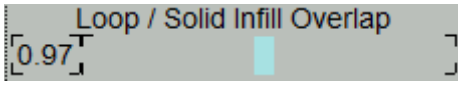


Figure 49 - Loop/Solid Infill Overlap.

A setting of 0 means the paths are just touching and a value of 2 is for maximum overlap.

6.7.3 $/Hour

The cost of running the machine for one (1) hour.

6.7.4 Z Step [mm]

The distance that a single stepper motor step will move the Z axis.

6.7.5 Min Segment Length [mm]

If the controller has a minimum segment length this setting will allow KS to only output usable segment lengths. A good starting point is 2x(a single stepper motor step distance).

6.7.6 Bed Size X [mm]

The usable size of the print bed in the X direction.

6.7.7 Bed Size Y [mm]

The usable size of the print bed in the Y direction.

6.7.8 Bed Size Z [mm]

The usable size of the printer in the Z direction.

6.7.9 Bed Center X [mm]

KS treats the center of the bed as the origin (0,0) but most printers do not. This parameter sets the X coordinate of the bed center. For example: A bed with a 300mm usable X dimension would have a Bed Center X of approximately 150mm.

6.7.10 Bed Center Y [mm]

KS treats the center of the bed as the origin (0,0) but most printers do not. This parameter sets the Y coordinate of the bed center.

6.7.11 Bed Is Round

Checkbox for Delta-style printers. Affects display of print bed as well as model packing.

6.7.12 Bed Roughness [mm]

The difference between the lowest point and the highest point on the print bed. The 1st layer will be thickened to compensate for the roughness.

6.7.13 Z-Settle [mm]

Overshoots on Z moves by this amount. Generally used for printers where the bed moves down (instead of the head moving up). Reverses the direction of the Z axis so that the last move is against gravity to eliminate backlash in the drive mechanism.

6.7.14 Z Offset [mm]

Adds a constant Z offset value to the G Code files.

6.7.15 Bed STL Model

The STL model used for display in the viewport.

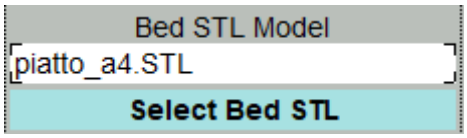


Figure 50 - Bed STL Model selection.

6.7.16 Printer Color Scheme Hint

Color scheme of the main window and viewport used to give a reminder of which Printer Profile is currently selected.

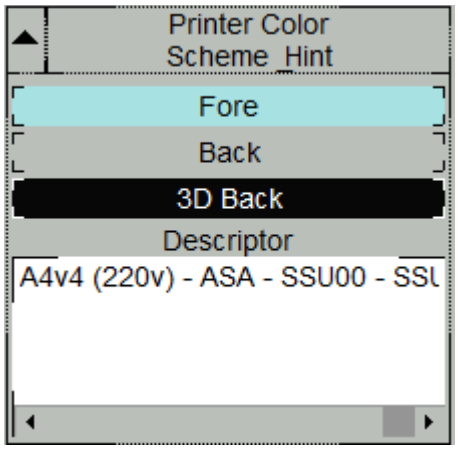


Figure 51 - Printer Color Scheme Hint menu.

- Fore – Color of most buttons.
- Back – Color of the main window and menu background.
- 3D Back – Color of the background of the 3D viewport.
- Descriptor – The description of the printer.

6.7.17 Blackbelt-Style Printer

Checkbox used for “infinite” length belt-style printers.

6.7.18 Final Z

The build plate will move to this value if the final Z height of a print job is less than the selected height.

6.8 Printer G-Code Editor

Entry for various printer functions.

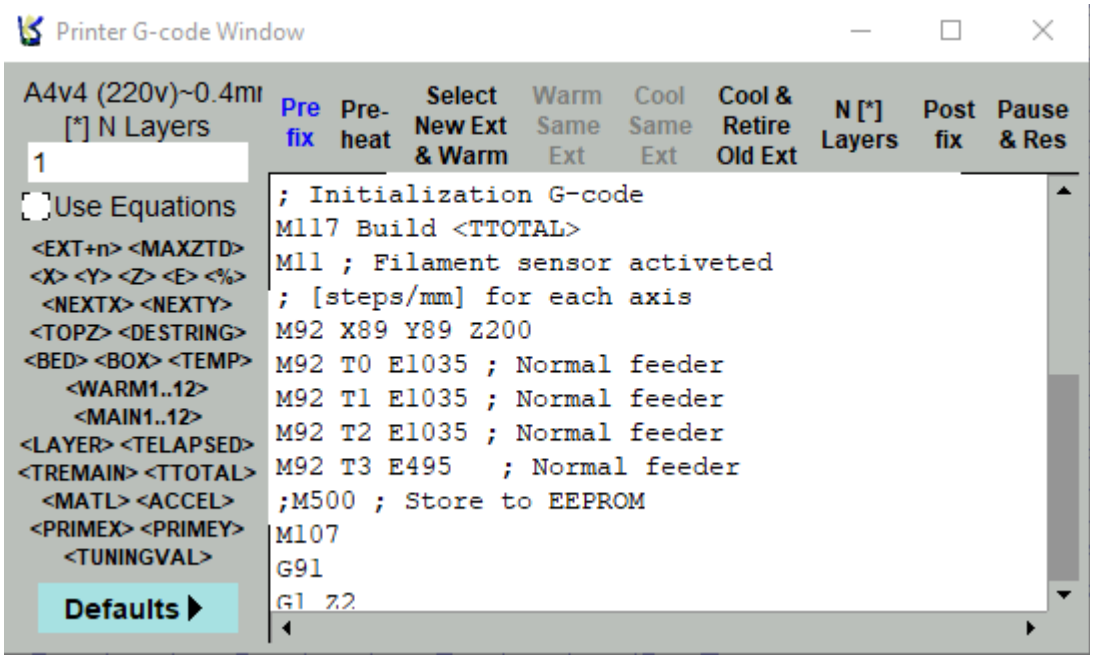


Figure 52 - Printer G-Code window.

6.8.1 Prefix

G-Code to be added to beginning of each print job.

6.8.2 Pre-Heat

G-Code inserted to pre-heat an extruder.

6.8.3 Select New Ext & Warm

G-Code inserted to select a new extruder and warm it to the selected temperature.

6.8.4 Warm Same Ext

G-Code inserted to warm the current extruder.

6.8.5 Cool Same Ext

G-Code inserted to cool the current extruder.

6.8.6 Cool & Retire Old Ext

G-Code inserted to cool and retire an extruder. Usually set to a lower temperature for keep-warm.

6.8.7 N [*] Layers

G-Code inserted every 'N' layers.

6.8.8 Postfix

G-Code to be added at the end of each print job.

6.8.9 Pause & Res

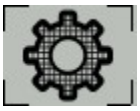
G-Code used any time a Pause is triggered.

7.0 Print Style

Print Styles in KS are typically used to keep track of settings for different:

- Nozzle sizes
- Material types
- Infill
- Layer thicknesses

The Print Styles menu is accessed by clicking the icon below:



The menu appears on the right hand side of the screen:

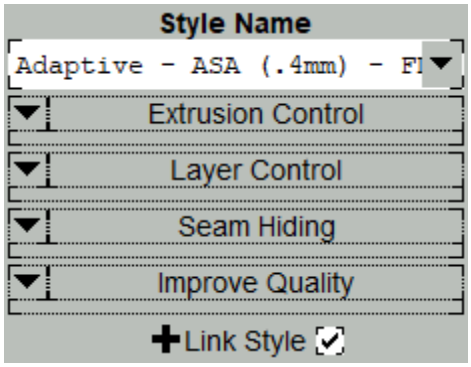


Figure 53 - Print Styles menu.

7.1 Style Name

The Style Name menu can be used to select from a list of print styles that can be organized similarly to the Printer Name.

The menu can have multiple levels which can be used to sort the styles by nozzle size, layer settings, or any other variable. The format for the name is shown below with each level separated by a ~:

Top Level~1st Sub Level~2nd Sub Level~…~Profile Name

An example is shown below and the outcome is shown in Figure 54.

Adaptive - .4mm~Adaptive - ASA (.4mm)

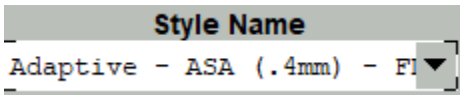


Figure 54 - Style Name menu.

7.2 Extrusion Control

The extrusion control menu provides the basic control of some simple print parameters.

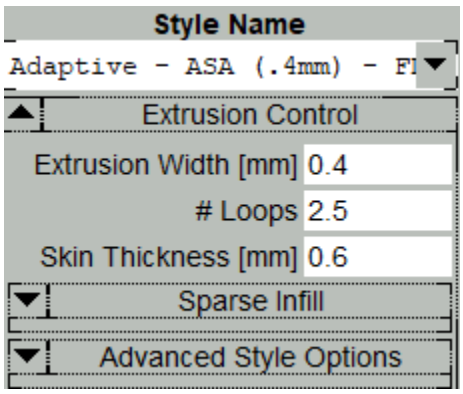


Figure 55 - Extrusion Control menu.

7.2.1 Extrusion Width [mm]

The width of the extruded path. This value MUST be equal to, or wider than, the nozzle size. It is also recommended to keep this value larger than the layer thickness.

7.2.2 # Loops

The number of loops around the permitter of the printed part. Decimal values will periodically insert another loop to increase the strength of the bond between infill and perimeter.

7.2.3 Skin Thickness [mm]

The minimum thickness of solid infill and perimeter loops on all sides of the part (including top and bottom). Solid infill will be added if [# loops]x[Extrusion Width] is less than the skin thickness.

7.2.4 Sparse Infill

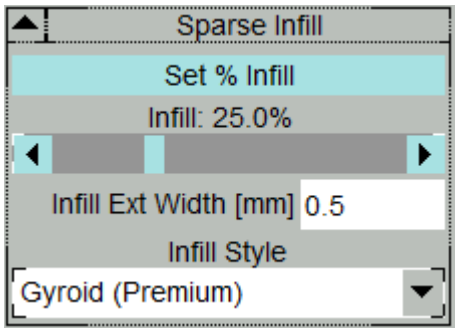


Figure 56 - Sparse Infill submenu.

- Set % Infill – Sets the percentage of the interior of the part that will be filled with extruded plastic.
  - Can be set from Hollow (no infill) to 100 (solid infill).
  - Vase infill is hollow and removes the top surface of the part.
- Infill Ext Width [mm] – Set the extrusion width to be used for all infill toolpaths.
  - A good default is to use the same value as the extrusion width.
  - Thinner reduced warping and internal part stress.
  - Thicker increases bond strength between external loops and infill.
- Infill style – The toolpaths used for part infill.
  - Straight – The infill is a series of crossing straight paths.

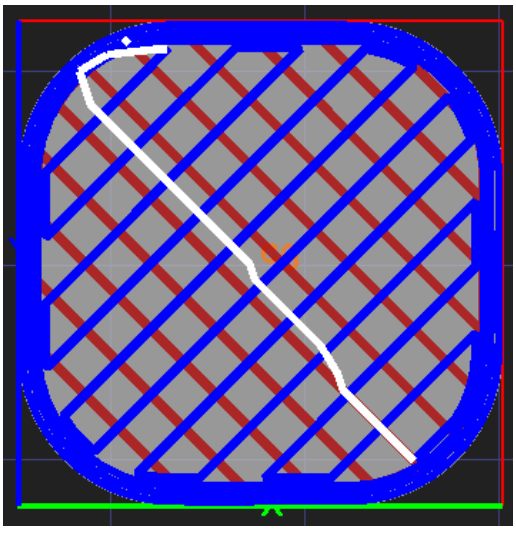


Figure 57 - Straight infill example.

  - Octagonal – The infill is made up of octagonal toolpaths.

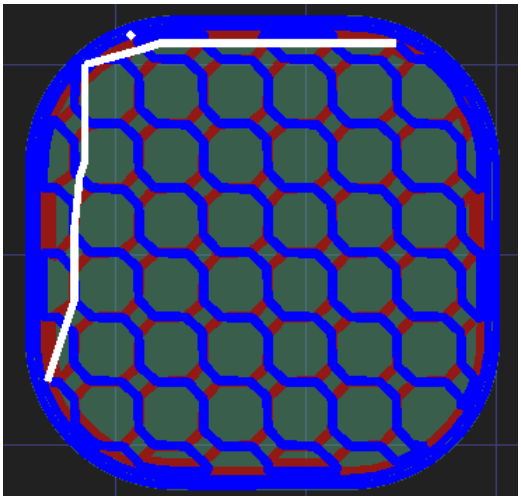


Figure 58 - Octagonal infill example.

- Rounded – The infill is made up of circular toolpaths.

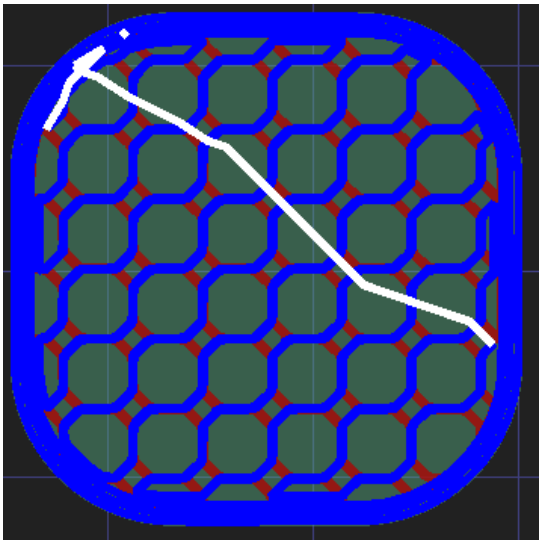


Figure 59 - Rounded infill example.

- Cellular – The infill is made up of toolpaths that change with each layer to create cells like a sponge or foam.
  - This infill style can be combined with flexible materials to achieve greater elasticity than the material alone can achieve.

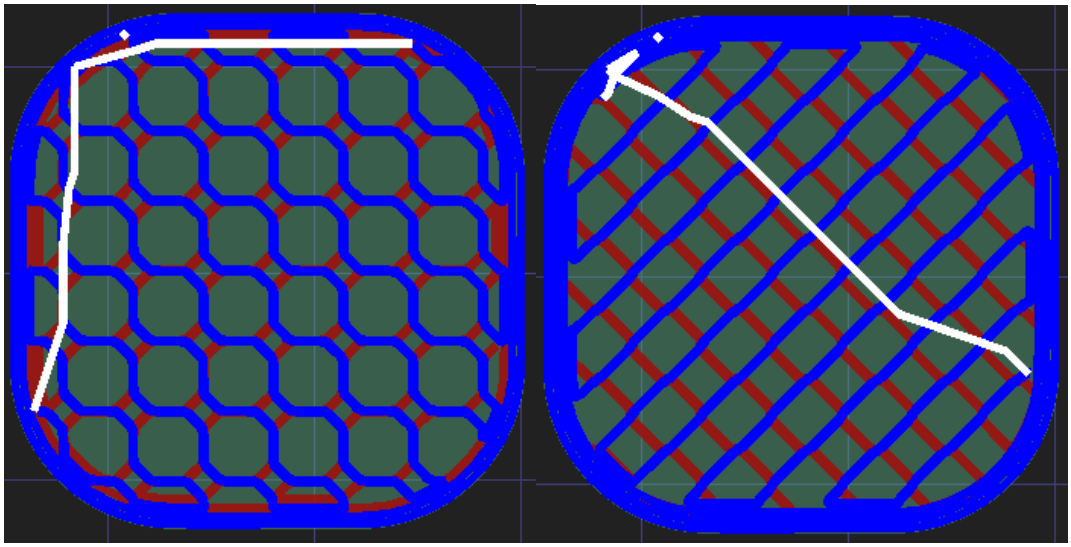


Figure 60 - Cellular infill example.

- Gyroid (Premium) – The infill is made up of toolpaths that change with each layer to create a more isotropic (equal properties in all directions) part.

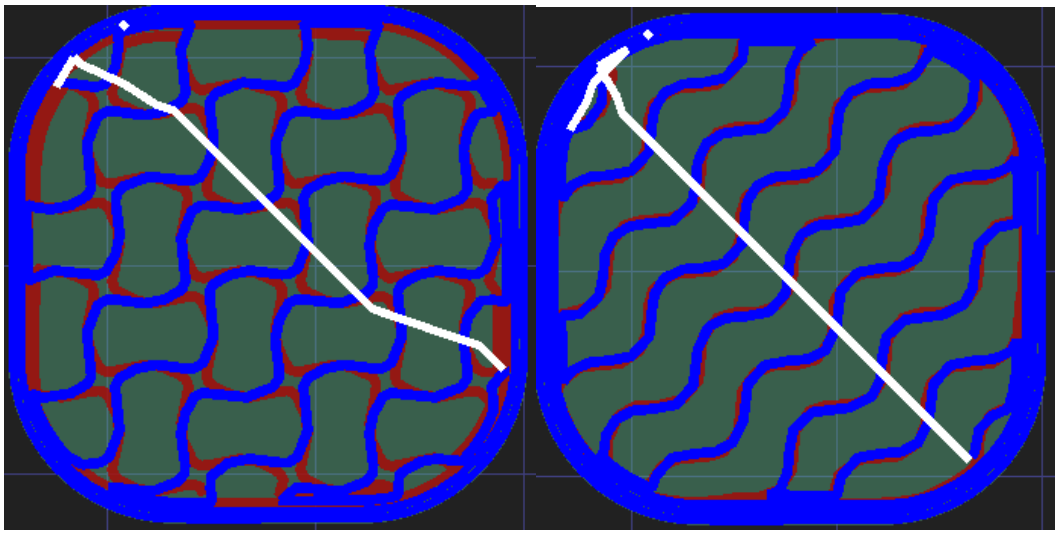


Figure 61 - Gyroid (Premium) infill example.

- Dynamic (Premium) – The infill density changes based on a grayscale gradient that can be applied to the part. (See section 11.0 Models list for more details.)

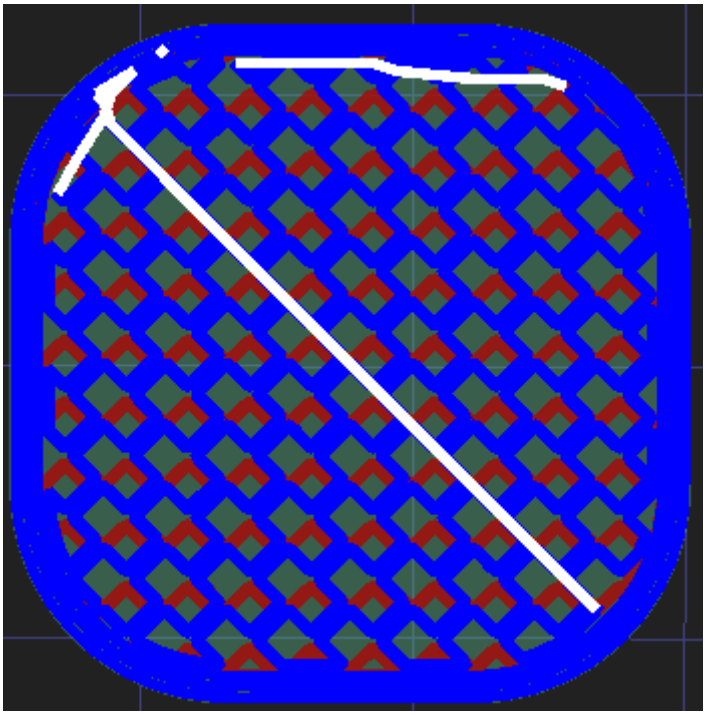


Figure 62 - Dynamic (Premium) infill example.

7.2.5 Advanced Style Options

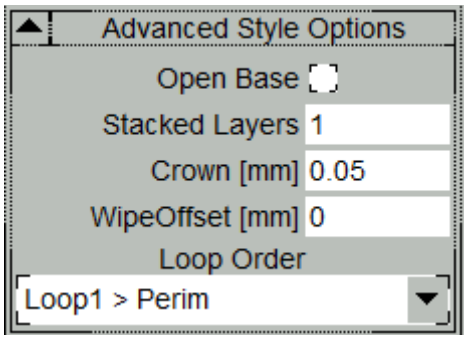


Figure 63 - Advanced Style Options Menu.

- Open Base – When checked KS will not put a solid skin on the base of the part. This will leave the infill exposed on the base.
- Stacked Layers – Infill and support are only extruded every "N" layers. Figure 63 shows N=1 which extrudes support and infill every layer. N=2 would extrude support and infill every other layer.
- Crown – Crowning tries to fill in small voids and every thin walls. Crowning toolpaths must be greater than the entered value to be generated. Negative values will disable this feature.
    - Very small values (like the one in Figure 63) can be combined with very small nozzles to create very intricate parts at the expense of greatly increased build times.

- WipeOffset – KS will offset wipe toolpaths inward by this value.
- Loop Order – Changes the order in which the outer skin of the part is printed:
  - Permi 1st – Prints the perimeter first for the best dimensional accuracy but can lead to sloped outer walls of the parts.
  - Loop1 > Perim – Prints the first inner loop followed by the outer perimeter and then any other inner loops. This setting is a good compromise between dimensional accuracy and minimal slope.
  - Perimeter Last – Prints all the inner loops first and then the outer perimeter. Minimal outer wall sloping but worst dimensional accuracy of the part.

7.3 Layer Control

The layer control menu provides fine grained control over the various layer related settings.

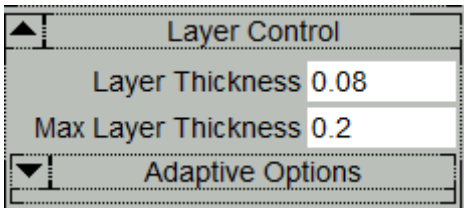


Figure 64 - Layer Control Menu.

7.3.1 Layer Thickness

The minimum distance the Z axis will move for each printed layer. This value works best as a multiple of the minimum Z step (see 6.7.4) and a good default value is less than the extrusion width.

7.3.2 Max Layer Thickness

The maximum distance the Z axis will move for each printed layer when trying to match the stepover values specified in 7.3.3.

7.3.3 Adaptive Options

Adaptive Z changes the layer thickness between the “Layer Thickness” value and the “Max Layer Thickness” to maintain the desired overlap between layers. The Adaptive Z settings allow for optimal speed when printing a part that has both angled/curved surfaces and vertical walls.

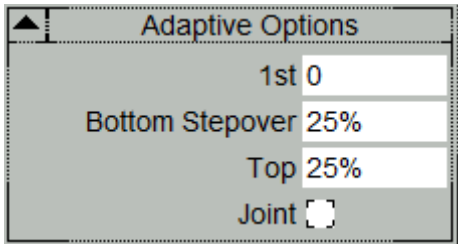


Figure 65 - Adaptive Options Menu.

- 1st – The minimum thickness of the first layer.
- Bottom Stepover – The amount that each layer should overlap the previous one on the bottom side of the part. A value of 50% or half of the extrusion width should be used to avoid the need for support material.
- Top - The amount that each layer should overlap the previous one on the bottom side of the part.
- Joint – Force all models to use the same layer heights. This setting is not recommended unless the printer uses 1 nozzle to print multi-extruder prints.

7.4 Seam Hiding

The Seam Hiding menu provides settings to minimize the appearance of seams on the exterior of the parts. The seam is where the outer perimeter loop starts and ends.

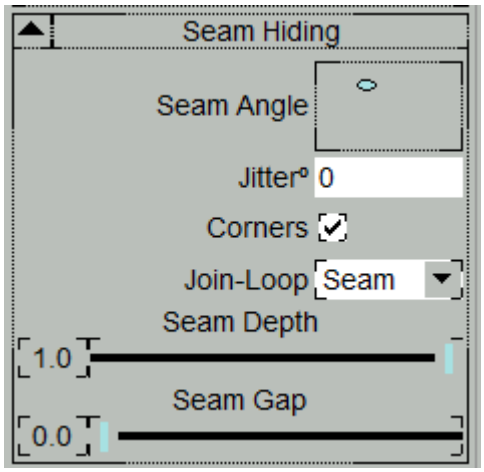


Figure 66 - Seam Hiding menu.

7.4.1 Seam Angle

The seam angle settings can be used to position the seam around the parts on the build tray. The seam position can be changed by either dragging the small dot or by clicking on the desired position.

7.4.2 Jitter

The Jitter angle can be used to avoid linear seams. The seams for each layer are randomly distributed over the angle entered.

7.4.3 Corners

The Corners checkbox will place the seam on a corner when one is available.

7.4.4 Join-Loop

The Join-Loop settings has 3 options:

- Off – Prints the perimeter using perimeter settings and the 1st loop using loop settings.
- Seam – Prints the perimeter using perimeter settings and continues onto the 1st loop only near the seam.
- Full – Prints the perimeter and the 1st loop with perimeter settings in one continuous toolpath.

7.4.5 Seam Depth

Unknown

7.4.6 Seam Gap

The Seam Gap setting can be used to eliminate any excess material at the start and stop points of the outer perimeter. The slider adjusts the value between 0 and 4 with 0 being no gap and 4 being a gap that is 4x the extrusion width. A seam gap of 0 is the ideal but can only be achieved with a properly calibrated preload setting (specific to each material type) 9.5.

7.5 Improve Quality

The Improve Quality menu has settings that directly affect the overall quality of the final printed part.

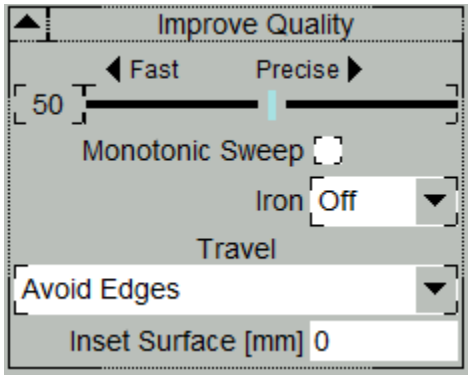


Figure 67 - The Improve Quality menu.

7.5.1 Speed Slider

The speed slider selects the speed of the print and can vary the print speed between Fast and Precise. A setting of 50 is an average between Fast and Precise. The Fast print speed was set in 6.3.1 and the Precise speed was set in 6.3.2.

7.5.2 Monotonic Sweep

The Monotonic Sweep checkbox forces the toolpaths of the top and bottom surfaces to sweep only in one direction. This setting can make some complex surfaces look better but at the cost of much longer print times.

7.5.3 Iron

Ironing can improve the appearance of flat top surfaces but running the hot nozzle over the already extruded part without extruding material.

- Off – No surfaces are ironed
- Sweep – The ironing toolpaths cross the entire part, even areas that cannot be ironed.
- Group – The ironing toolpaths are limited to the areas that can be ironed.

7.5.4 Travel

Travel moves are rapid moves that are used to position the head from the end of one toolpath to the beginning of another.

- Simple – Travel moves will take the shortest path from point to point.
- Stay Inside - Travel moves will attempt to stay within the footprint of the printed part.
    - Internal voids are ignored.
- Avoid Edges – Travel moves will attempt to within the printed area of the printed part.
    - Internal voids are recognized and avoided whenever possible.

7.5.5 Inset Surface

Offsets the outer surface of the part inward in the X and Y axis only (does not affect the Z height of the part). This value should be 0 for a properly calibrated printer but can be set as high as half the extrusion width.

7.6 Link Style

The Link Style checkbox associates the current style with the currently selected Printer Profile.

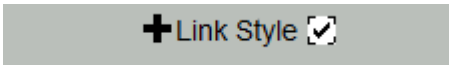


Figure 68 - Link Style checkbox.

8.0 Support Style

Support styles in KS are used to keep track of settings for different:

- Nozzle sizes
- Support Material Types
- Densities and Spacing
- Rafts

The Print Styles menu is accessed by clicking the icon below:

The menu appears on the right hand side of the screen:

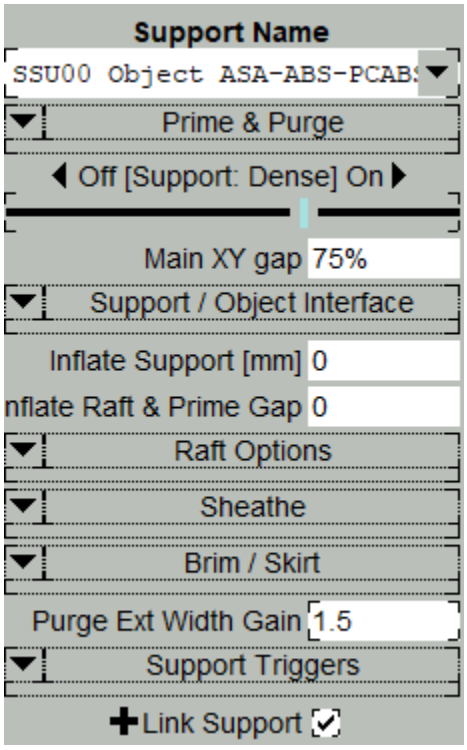


Figure 69 - Support Styles menu.

8.1 Support Name

The Support Name menu can be used to select from a list of support styles that can be organized similarly to the Printer Name and Print Style Name.

The menu can have multiple levels which can be used to sort the styles by nozzle size, layer settings, or any other variable. The format for the name is shown below with each level separated by a ~:

Top Level~1st Sub Level~2nd Sub Level~…~Profile Name

An example is shown below and the outcome is shown in.

SSU00~0.4mm~SSU00 Object ASA-ABS-PCABS (0.4mm) – HQ

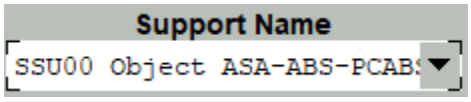


Figure 70 - Support Name menu.

8.2 Prime & Purge

The Prime and Purge menu controls the height and position of the prime pillar.

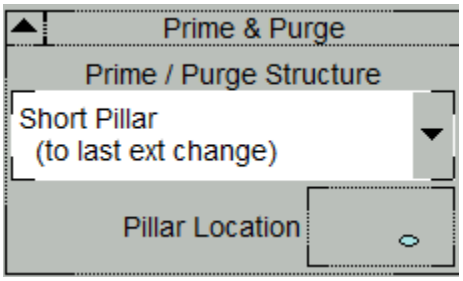


Figure 71 - Prime and Purge menu.

8.2.1 Prime/Purge Structure

The menu allows for the selection of prime structure and location:

- No Prime Structure – No prime structures are printed.
- Single Pillar – A separate cylindrical part is added to the build plate and the prime structure is printed at every layer regardless of extruder changes.
- Short Pillar – A separate cylindrical part is added to the build plate and the prime structure is printed only as high as the final extruder change.
- Short Wall – A thin wall is added around each part on the build plate. The prime structure is printed only as high as the final extruder change.
- Wall – A thin wall is added around each part on the build plate. The prime structure is printed at every layer regardless of extruder changes.
- Purge Octagon – A separate octagonal structure is added to the build plate and material is purged by volume rather than layers.
    - This setting would typically be used by mixing nozzles where multiple extruders feed a single nozzle.

8.2.2 Pillar Location

The prime pillar location can be set by moving the small dot into any of the four corners.

8.3 Support Density

The slider sets the density of the support structure. Supports can also be turned off using the slider.

- A higher density support structure reduces part warping at the expense of longer print times and more difficulty removing supports from the printed parts.
- A lower density support structure is much easier to remove from the printed parts but reduces the ability to resist warping.

Figure 72 - Support Density slider.

8.4 Main XY Gap

The Main XY gap sets the horizontal gap between the support structure and the model. The value can be set in mm or in % of extrusion width.

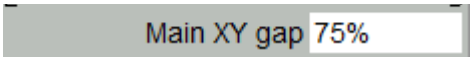


Figure 73 - Main XY gap setting.

8.5 Support/Object Interface

The Support/Object Interface menu controls the contact between support material and model material.

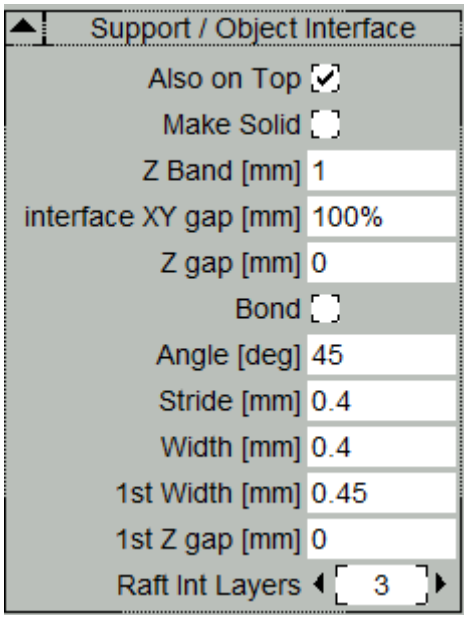


Figure 74 - Support/Object Interface menu.

8.5.1 Also on Top

The interface support style is used on top of the part (where required) when this checkbox is selected.

8.5.2 Make Solid

This checkbox will make all interface support solid when checked. If unchecked the interface support is 50% dense.

8.5.3 Z Band

The support structure will transition from regular support to interface support this far below the interface between the support material and the model material.

8.5.4 Interface XY Gap

The Interface XY Gap is like the Main XY Gap except it only applies to the interface support material. The value can be set in mm or in % of extrusion width.

8.5.5 Z Gap

The gap between the support material and the model material. Limited to less than the smallest layer thickness.

8.5.6 Bond

The first layer of model material follows the same paths as the top layer of support to increase the bonding between support and material. This setting should not be used unless the model and support materials do not adhere well.

8.5.7 Angle

The angle of the interface support layers relative to the standard support paths.

8.5.8 Stride

The horizontal spacing of the interface paths.

8.5.9 Width

The width of the interface extrusion path.

8.5.10 1st Width

The width of the interface extrusion path on the first layer only.

8.5.11 1st Z Gap

The vertical gap between the top of the raft interface and the first layer of the model material.

8.5.12 Raft Interface Layers

The number of interface support layers to be placed on top of the raft before model material is printed.

8.6 Inflate Support

Increases the size of the support in all directions by the specified amount.

8.7 Inflate Raft & Prime Gap

Increases the size of the raft (or skirt) in all directions by the specified amount to increase adhesion to the bed.

8.8 Raft Options

The Raft Options menu allows for the fine tuning of the raft toolpaths.

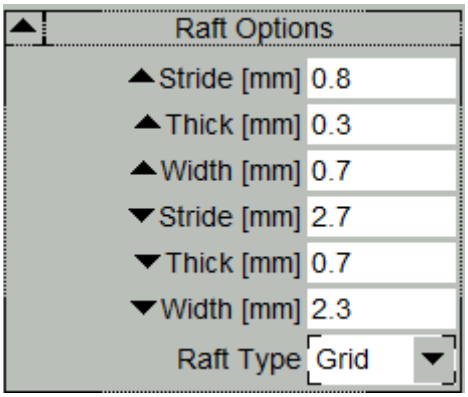


Figure 75 - Raft Options menu.

8.8.1 Upper Stride

The horizontal spacing of the upper raft toolpaths.

8.8.2 Upper Thick

The thickness of the upper raft toolpaths.

8.8.3 Upper Width

The width of the upper raft toolpaths.

8.8.4 Lower Stride

The horizontal spacing of the toolpaths directly on the build plate.

8.8.5 Lower Thick

The thickness of the lower toolpaths directly on the build plate. A thicker lower toolpath can compensate for minor variations in bed flatness and level.

8.8.6 Lower Width

The width of the lower toolpaths directly on the build plate. A wider lower toolpath can increase bonding to the build plate.

8.8.7 Raft Type

- Off – No raft is printed.
- Grid – The raft is made up of a series of linear toolpaths.

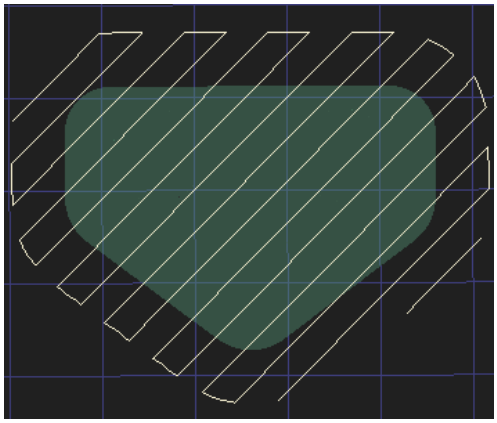


Figure 76 - Grid raft style example.

- Pillar – The raft is made up of many small pillars.

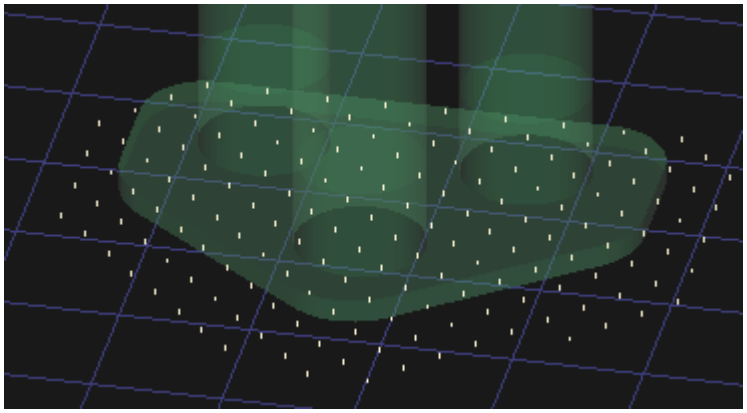


Figure 77 - Pillar raft style example.

8.9 Sheathe

The Sheathe menu gives the option to create stronger support structures by putting a solid perimeter around the standard sparse support structure. Sheathe should NOT be used when support will be recessed inside a part.

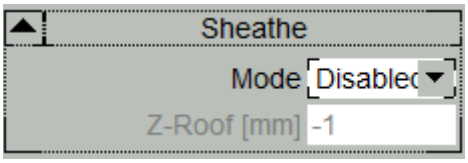


Figure 78 - Sheathe menu.

8.9.1 Mode

- Disabled – No sheathed support. Standard support is used.
- Sparse – Puts the solid perimeter around only the sparse support layers only.
- Interface – Puts the solid perimeter around the interface support layers only.

- Both – Puts the solid perimeter around both the sparse supports and the interface support.

8.9.2 Z- Roof

No solid perimeter will be generated above this Z height. A value of less than 0 will disable the Z-Roof setting and will sheathe all Z layers (if Sheathe is enabled).

8.10 Brim/Skirt

The Brim and Skirt options can help the part adhere to the build plate or the raft.

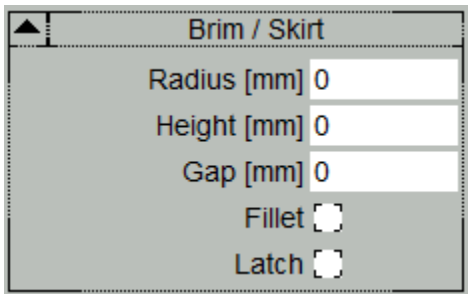


Figure 79 - Brim/Skirt menu.

8.10.1 Radius

Increases the size of the first layer(s) of model material by this amount. The expansion is done using extra loops.

8.10.2 Height

The height of the brim or skirt.

8.10.3 Gap

If the Gap value is greater than 0 then the brim is separated from the part by the Gap distance and becomes a skirt.

8.10.4 Fillet

The Fillet checkbox rounds the outer corners of the brim or skirt.

8.10.5 Latch

The Latch checkbox captures the brim with raft or interface paths. The brim needs to be at least 2 layers thick and at least 2 loops wide. This setting will decrease the chances of the part warping.

8.11 Purge Ext Width Gain

The Purge Extruder Width Gain sets a multiple of the extrusion width for use with the Purge Octagon setting (8.2.1).

8.12 Support Triggers

The Support Triggers menu controls the top, bottom, and overhang angles for which support will be generated.

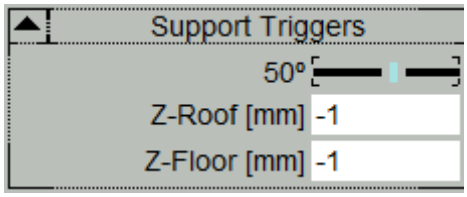


Figure 80 - Support Triggers menu.

8.12.1 Angle

The Angle slider is used to set the angle from vertical for which supports will be generated.

8.12.2 Z-Roof

The Z-Roof setting is the highest Z height for which supports will be generated regardless of the other support settings. Some supports will still be generated for structures that cannot self-support.

8.12.3 Z-Floor

The Z-Floor setting is the lowest Z height for which supports will be generated regardless of the other support settings. Some supports will still be generated for structures that cannot self-support.

8.13 Link Support

The Link Support checkbox associates the current support style with the currently selected Printer Profile.

Figure 81 - Link Support checkbox.

9.0 Material

Materials in KS are used to keep track of settings for different:

- Nozzle sizes
- Material combinations
- Shrinkage
- Pre-heat
- Preload

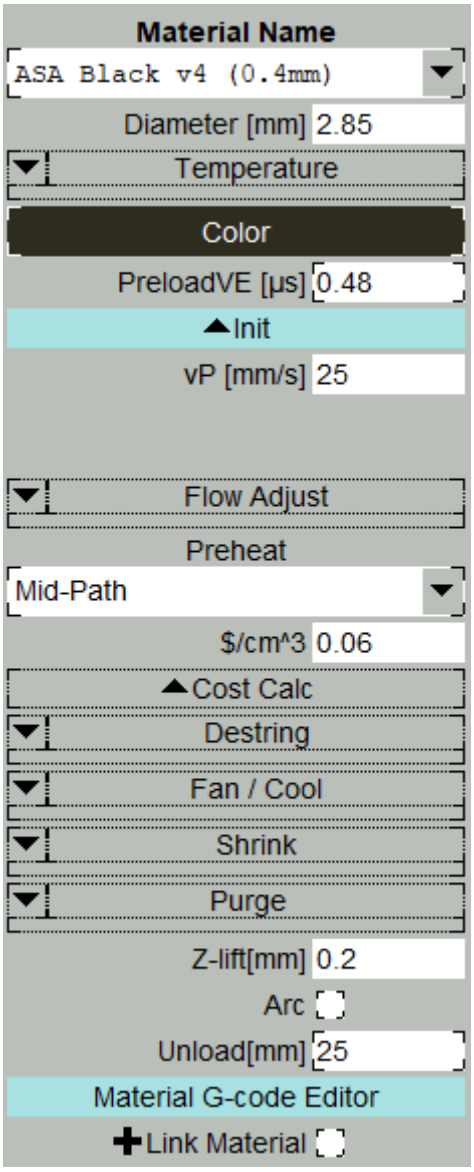


Figure 82 - Material Profile menu.

9.1 Material Name

The Material Name menu can be used to select from a list of model and support materials that can be organized similarly to the Printer Name, Print Style Name, and Support Style Name.

The menu can have multiple levels which can be used to sort the materials by nozzle size, material type, or any other variable. The format for the name is shown below with each level separated by a ~:

Top Level~1st Sub Level~2nd Sub Level~…~Profile Name

An example is shown below and the outcome is shown in Figure 83.

ASA~0.4mm~ASA Black v4 (0.4mm)

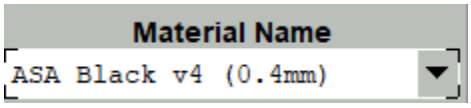


Figure 83 - Material Name menu.

9.2 Diameter

The exact filament diameter as measured with callipers or a micrometer. This diameter value is used to calculate the volume of material extruded for each mm of filament length.

9.3 Temperature

The temperature menu contains all the temperature settings associated with the material.

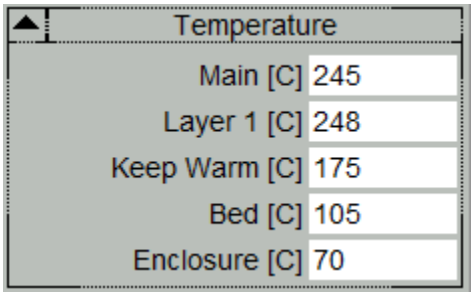


Figure 84 - Temperature menu.

9.3.1 Main

The extruder temperature while printing.

9.3.2 Layer 1

The extruder temperature for the 1st layer. A higher temperature for the first layer can improve adhesion to the build plate and support material.

9.3.3 Keep Warm

The Keep Warm temperature is used when the extruder is not active during a print job. This setting would be typically be used for a multi-extruder print. A lower temperature will reduce oozing and power consumption. A good stating value is 65% of the main temperature.

9.3.4 Bed

The temperature for the print bed during the print job. Can be referenced using the <BED> token in the G-Code.

9.3.5 Enclosure

The temperature of the printer enclosure (for printers that have a heated enclosure). Can be referenced using the <BOX> token in the G-Code.

9.4 Color

The colour of the filament. Clicking the word “Color” brings up the Color menu which can be used to select the filament color.

Figure 85 - Filament color button.

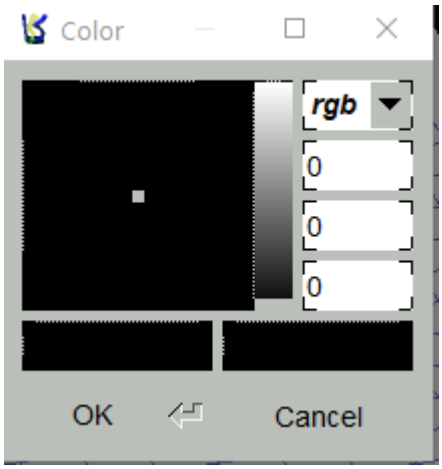


Figure 86 - Filament color menu.

9.5 PreloadVE

The preload viscoelasticity is a material constant that should be calculated using the “Init” button. This setting compensates for the elasticity in the material to help control the exact start and stop points for extrusion which leads to a cleaner print with minimal stringing.

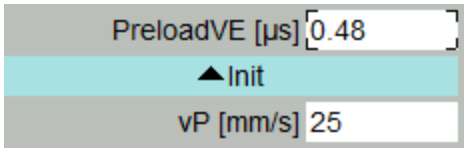


Figure 87 - Preload Viscoelasticity setting.

Clicking the “Init” button brings up the initialization selection menu.

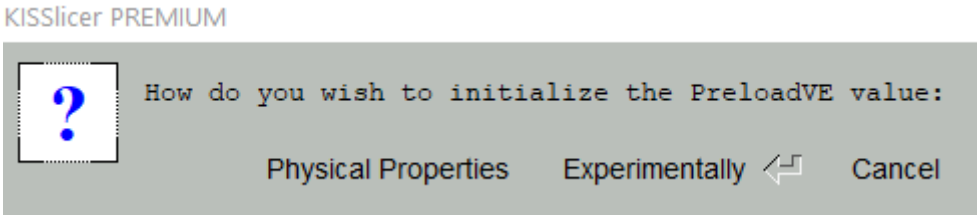


Figure 88 - Initialization selection menu.

9.5.1 Physical Properties

The viscoelastic properties of the material, if known, can be directly entered as a good starting point for further calibration.

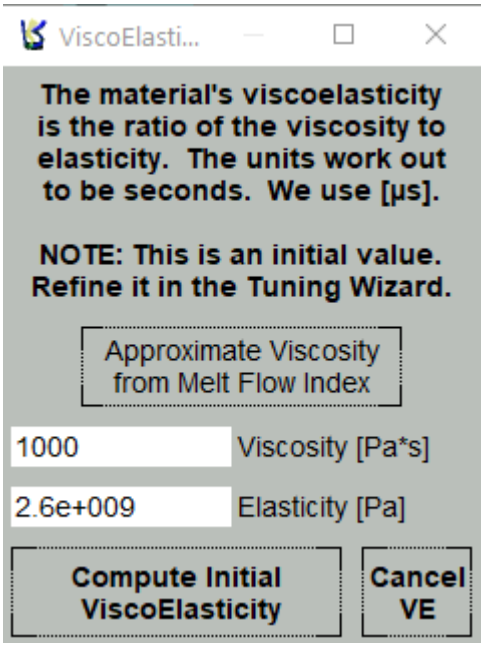


Figure 89 - Physical properties entry window.

9.5.2 Experimentally

The most accurate to determine the viscoelastic properties of the material is experimentally.

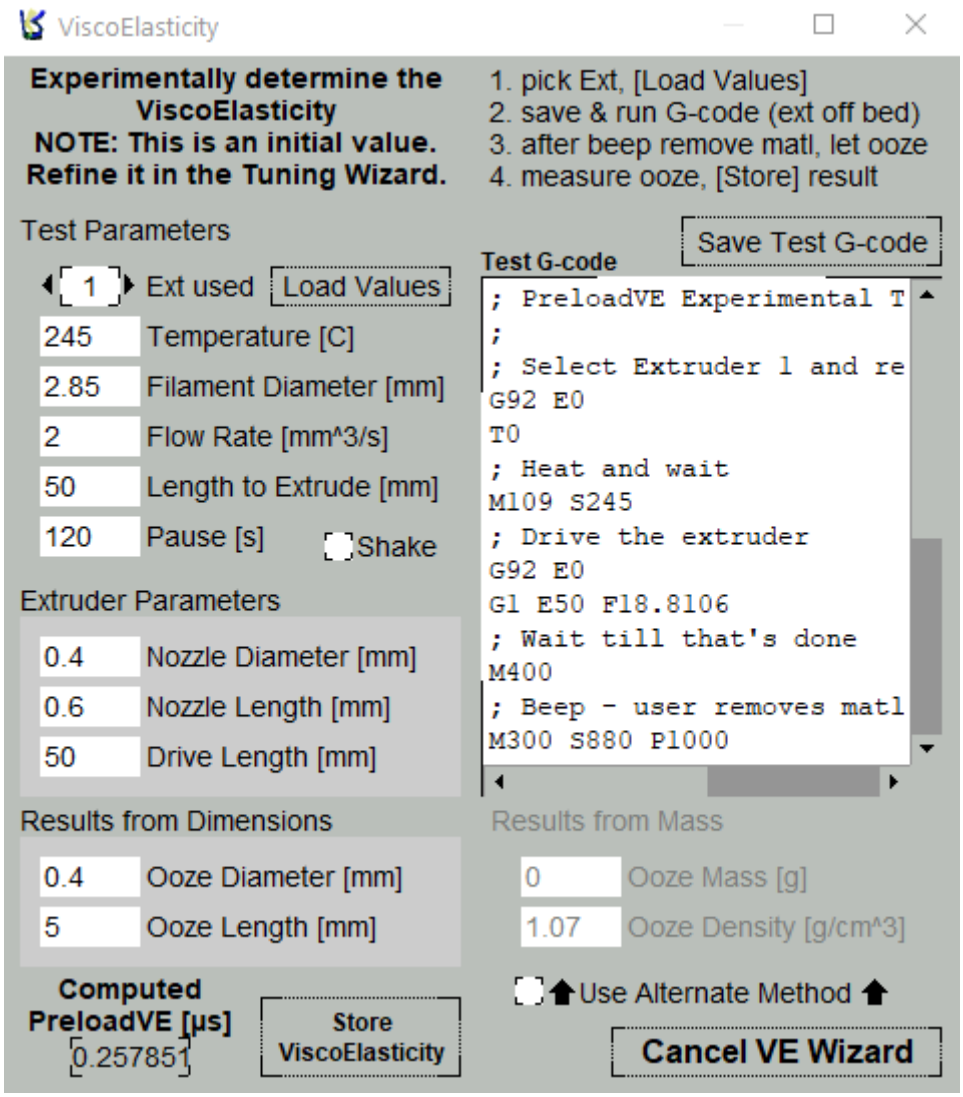


Figure 90 - Viscoelasticity calulation wizard.

The viscoelasticity is determined by:

1. Picking the desired extruder and loading the material values using the “Load Values” button.
2. Enter the extruder parameters.
    a. Nozzle Diameter – The nozzle extrusion diameter.
    b. Nozzle Length – The length of the nozzle that is at the nozzle diameter.
    c. Drive Length – The length of filament between the drive gear and the nozzle tip.
3. Enter or modify the G-code to suit the printer. Press the “Save Test G-code” button.
4. Run the code and wait for the machine to beep and remove the extruded material from the nozzle.
    a. Use the “Shake” checkbox if the machine is unable to beep.
5. Wait for any additional material to ooze out of the nozzle as it cools.
6. Measure the ooze using either the standard method or the alternate method:

a. Standard – Measure the diameter and length of the material that has oozed out of the nozzle.
b. Alternate – Weigh the ooze and enter the material density.

7. Press the “Store ViscoElasticity” button.

You can cancel the wizard at any time using the “Cancel VE Wizard” button.

9.5.3 vP

The filament velocity used during preload feeding.

9.6 Flow Adjust

The settings in the flow adjust menu set limits on the extrusion parameters to ensure that the extruder is always capable of following the G-code commands.

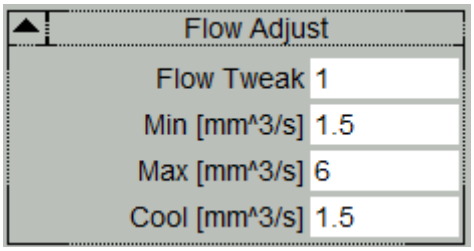


Figure 91 - Flow adjust menu.

9.6.1 Flow Tweak

The Flow Tweak setting can be used to compensate for slight differences between the material settings and the actual material.

If the Preload Viscoelasticity is set correctly then the Flow Tweak setting should be set to 1.

9.6.2 Min

The minimum volumetric flowrate (volume/second) that the extruder can extrude for the selected material. KS will increase the flow rate and the head speed to maintain the minimum flow rate.

9.6.3 Max

The maximum volumetric flowrate (volume/second) that the extruder can extrude for the selected material. KS will decrease the flow rate and the head speed to maintain the maximum flow rate.

9.6.4 Cool

The maximum volumetric flowrate (volume/second) that the part cooling fan can cool. This setting is only used for unsupported overhangs. KS will decrease the flow rate and the head speed to maintain the cool flow rate.

9.7 Preheat

The Preheat menu determines when the next extruder will be pre-heated in a multi-extruder print job.

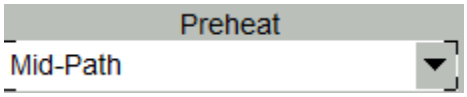


Figure 92 - Preheat dropdown menu.

Figure 93 - Preheat options.

9.7.1 None

The next extruder will not be heated until the current extruder is cooling down.

9.7.2 Safe

The next extruder will be preheated after the last path is completed for the current extruder

9.7.3 Early

The next extruder will be preheated before the start of the last path for the current extruder.

9.7.4 Mid-Path

The next extruder will be preheated based on the estimated heat time set in section 6.5 regardless of the state of the current extruder. The next extruder can even be preheated during an extrusion move for the current extruder.

9.8 Cost ($/cm^3)

KS already calculates the volume of material needed to complete a print job. The Cost per unit volume is used to calculate the cost of the print job.

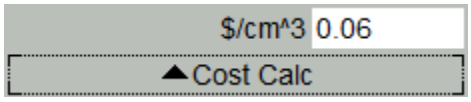


Figure 94 - Material Cost setting.

9.8.1 Cost Calculator

The cost can be entered directly into the field, or the Cost Calculator can be used.

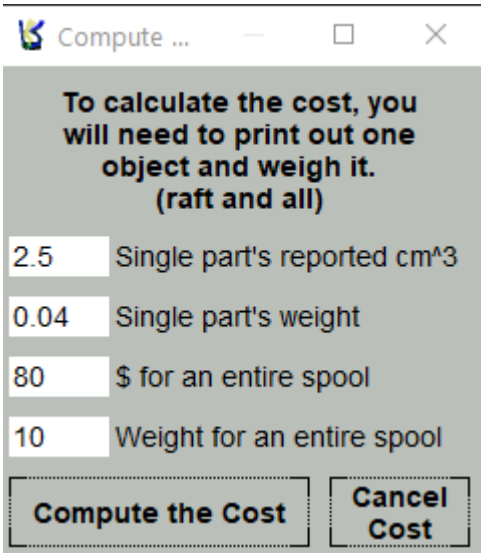


Figure 95 – Material cost calculation window.

Follow the steps below to calculate the cost per unit volume for the current material:

1. Select a part and slice it for a single extruder print.
2. Enter the reported volume of the part.
3. Enter the weight of the part in whatever units work best.
4. The cost of the entire spool.
5. The weight of the entire spool (using the same units use above).
6. Press “Compute the Cost”.

9.9 Destring

The destring menu contains the settings that can reduce material stringing when the nozzle stops extruding and moves to another location.

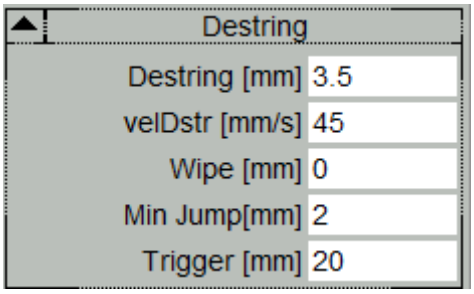


Figure 96 - Destring menu.

9.9.1 Destring

The distance in mm the filament should be retracted at the end of the extrusion path.

9.9.2 velDstr

The speed in mm/s at which the filament should be retracted during a Destring move.

9.9.3 Wipe

Before the nozzle leaves the current extrusion path it will backtrack (or continue over the loop) by the distance entered.

9.9.4 Min Jump

Any jump (stop extrusion, move, start extrusion) less than the entered value will not Destring or Wipe. A value of 0 will never disable the Destring or Wipe.

9.9.5 Trigger

Any jump (stop extrusion, move, start extrusion) greater than the entered value will force a Destring or Wipe. A value of 0 will always force a Destring or Wipe on all travels.

9.10 Fan/Cool

The Fan and Cool settings control when the part cooling fan will engage and at what percentage of the maximum speed. The fan speed is typically controlled by Pulse Width Modulation (PWM).

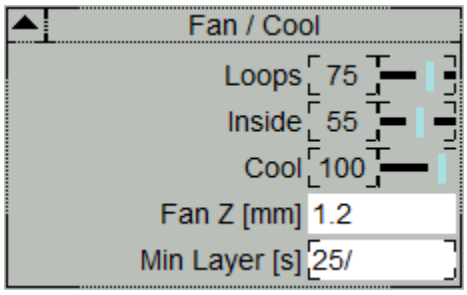


Figure 97 - Fan and Cool menu.

9.10.1 Loops

The Loops setting controls the fan speed for the perimeter, loops, and crown toolpaths.

9.10.2 Inside

The Inside setting controls the fan speed for all other toolpaths.

9.10.3 Cool

The Cool setting is used if the minimum layer time is less than the Min Layer setting (9.10.5).

9.10.4 Fan Z

The fan settings will be ignored below the entered Z value. KS compensates for any Z offset but this setting can be made negative to allow the cooling fan to be active while printing the raft.

9.10.5 Min Layer

The Min Layer setting is used to ensure that all printed layers have enough time to cool and solidify. Any layer that will take less than the entered value will be slowed, and the print will be paused if required.

9.11 Shrink

The Shrink setting is used to compensate for the shrinkage that happens when the extruded plastic cools. These settings are best determined by printing several identical samples and averaging the measurements.

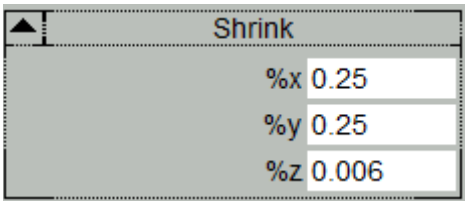


Figure 98 - Shink menu.

- %x – The amount that the model will shrink in the X dimension after cooling.
- %y – The amount that the model will shrink in the Y dimension after cooling.
- %z – The amount that the model will shrink in the Z dimension after cooling.

9.12 Purge

The Purge settings are used for multi-extruder printers with a single nozzle.

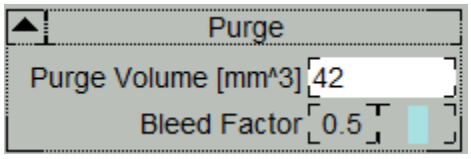


Figure 99 - Purge menu.

9.12.1 Purge Volume

The base volume of material that should be purged before switching to this material.

9.12.2 Bleed Factor

The Bleed Factor is a measure of how likely this colour will bleed into other colours.

- Darker colours will have higher bleed factors.
- Lighter colours will have lower bleed factors.

9.13 Z-Lift

The Z-Lift setting is used to raise the Z height after a destring (if enabled) and before a travel move.

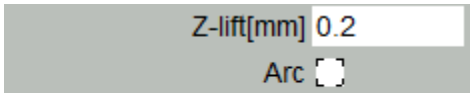


Figure 100 - Z-Lift Setting.

9.13.1 Arc

The Arc checkbox moves the head in an arc for the Z-Lift. This setting is intended to be used on Delta-style printers and is only functional when the travel mode is set to Simple.

9.14 Unload

The unload setting slows the last X mm of an island toolpath (an isolated portion of the toolpath on a given layer) to perimeter speed.

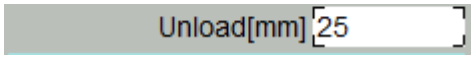


Figure 101 - Unload setting.

9.15 Material G-code Editor

Clicking the material G-code Editor button brings up the Material G-Code Window.

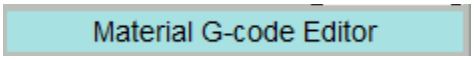


Figure 102 - Material G-code Editor button.

The Material G-code Window is used to enter any custom G-code that should be used for the material. Typically this would include the bed temperature token, the heated chamber token, and any other material specific G-codes that should be run.

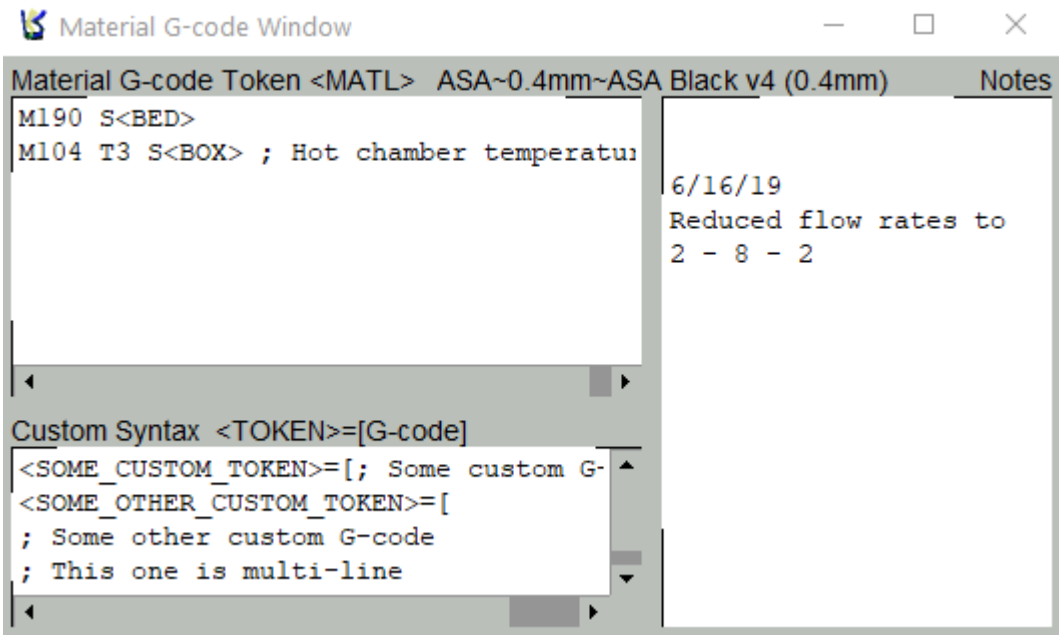


Figure 103 - Material G-code Window.

9.16 Link Material

The Link Material checkbox associates the current material with the currently selected Printer Profile.

Figure 104 - Link Material checkbox.

10.0 Miscellaneous Settings

The Miscellaneous Setting menu is where many of the settings that don't fit into another category.

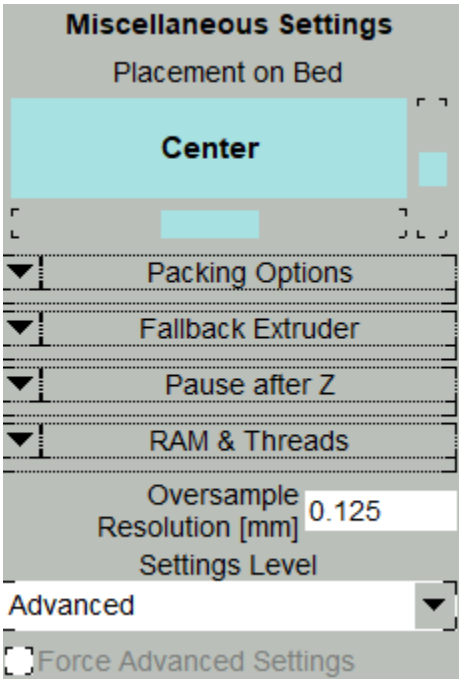


Figure 105 - Miscellaneous Settings menu.

10.1 Placement on Bed

The horizontal and vertical sliders can be used to shift the placement of the parts on the build tray. The "Center" button can be used place the parts back in the center of the build tray.

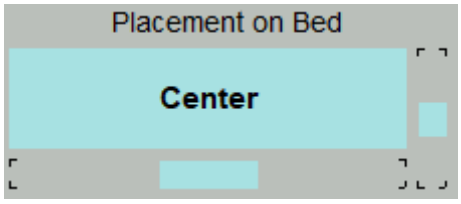


Figure 106 - Placement on Bed settings.

10.2 Packing Options

The packing options can be used to change how the parts are organized on the build tray.

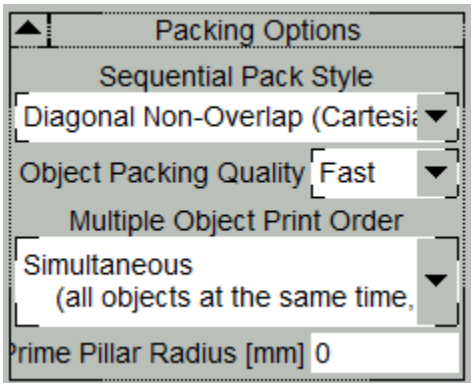


Figure 107 - Packing Options menu.

10.2.1 Sequential Pack Style

The Sequential Pack Style is used when multiple parts are printed in sequence instead of simultaneously.

- Diagonal Non-Overlap – Used on Cartesian style printers.
- Extra Space – Used on Delta style printers.

10.2.2 Object Packing Quality

The Object Packing Quality setting is used to balance the part placement time against the packing efficiency.

- Fast – Parts are the furthest apart on the build plate but this setting requires the least processing time.
- Balanced – Parts are positioned more accurately but take longer to process.
- Precise – Parts are positioned as close to each other as possible at the expense of significantly longer processing times.

10.2.3 Multiple Object Print Order

The Multiple Object Print Order setting controls the print order for multiple objects on the build plate.

- Simultaneous – Prints all objects at the same time and the print order is determined by KS.
- Sequential – Prints one object at a time from shortest to tallest. This setting reduces the usable build space significantly since the print head for any future objects needs to clear the previously printed object.
- Model Tab Order – Prints all objects at the same time and the print order is determined by the model list.

10.2.4 Prime Pillar Radius

The prime pillar is used to purge each nozzle before it is used to extrude a model. The pillar radius can be adjusted smaller to allow for more parts to be fit on the build plate or larger to purge more material before each nozzle is used.

10.3 Fallback Extruder

The fallback extruder menu can be used to switch extruders partway through a print job. This can be useful if there is not much material left on a spool or if the print job requires more than a single spool of material.

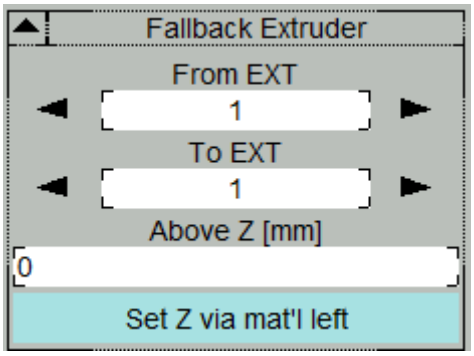


Figure 108 - Fallback Extruder menu.

10.3.1 From EXT

The From Extruder setting is the starting extruder for the print job.

10.3.2 To EXT

The To Extruder setting is the second extruder for the print job.

10.3.3 Above Z

The primary extruder will switch from the "From EXT" to the "To EXT" above the set Z height.

- Set Z via mat'l left – Clicking on the button brings up a window where the user can enter the filament length remaining on the From EXT spool (in meters) and KS will determine the optimal Z setting to switch extruders.

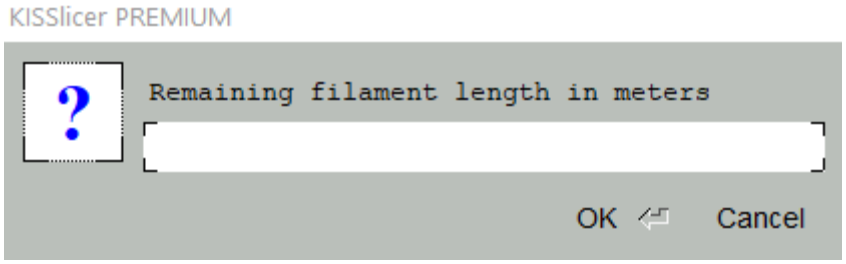


Figure 109 - Remaining filament setting.

10.4 Pause After Z

The Pause After Z menu can be used to pause the print job at one or multiple model Z heights.

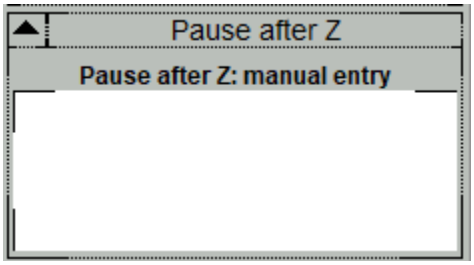


Figure 110 - Pause After Z menu.

- The Z heights are based on the model height and not the G-code heights.
  - Do not include any support rafts when picking the heights.
- Multiple values can be entered.
  - Values should be separated with a space or each value should be put on a new line.
- The information entered in this field is not saved.

10.5 RAM & Threads

The RAM and Threads menu can be used to set the allowable RAM and processor threads for slicing.

10.5.1 RAM Warning

The RAM Warning can be set to any value up to the total available RAM on the system. KS will show a warning pop-up where the user can continue or cancel slicing.

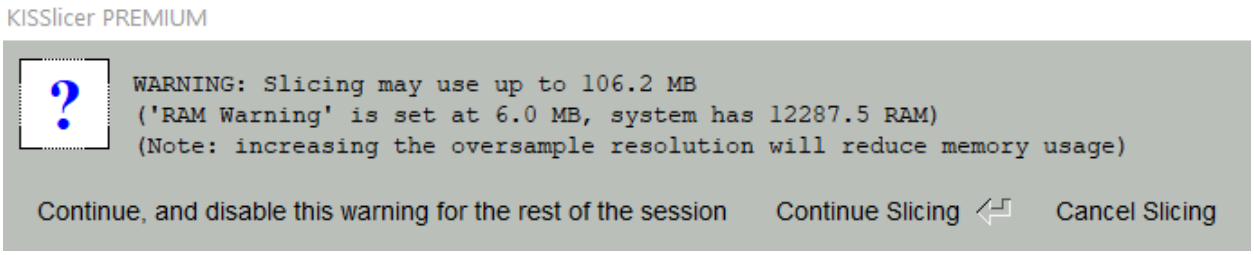


Figure 111 - RAM usage warning window.

10.5.2 Slicing Threads

The slicing threads setting settings the number of processor cores that will be used for slicing. More cores will increase the slicing speed.

10.6 Oversample Resolution

The Oversample Resolution setting controls the resolution that KS uses for internal calculations. A larger value will reduce RAM usage at the cost of lower resolution parts.

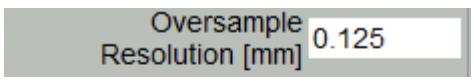


Figure 112 - Oversample Resolution setting.

10.7 Settings Level

The Settings Level can be changed to three different levels:

- Production – Limits the settings to only those necessary for production type tasks.
- Intermediate – Allows more complex settings to be modified.
- Advanced – Allows all settings to be modified.
- Force Advanced Settings – Forces the default settings for any setting not enabled in Production and Intermediate settings levels.

11.0 Models List

The Models List is on the left-hand side of the main KS window. Every model loaded into KS will appear as an item in the list. If multiple files are loaded as one item (see 5.3.1) they will appear as a single item in the Models list.

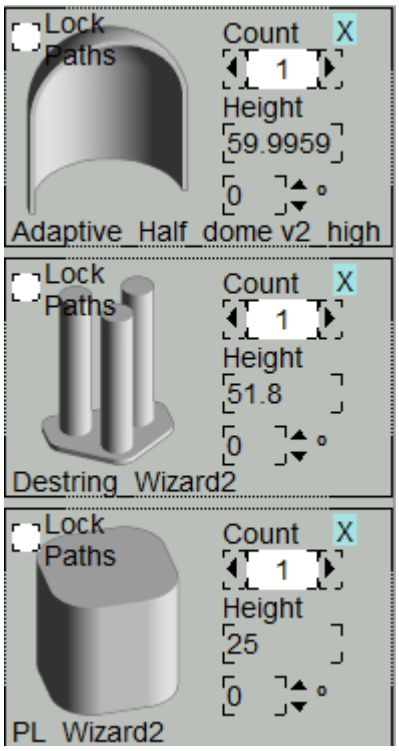


Figure 113 - Models list with three (3) models loaded.

Each model (or group of models) appears as below.

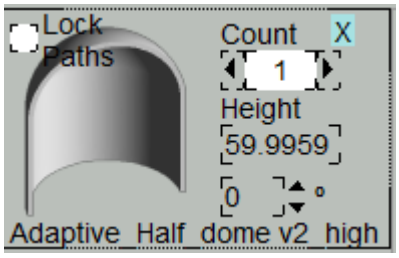


Figure 114 - Model Item.

- Lock Paths – The Lock Paths checkbox will freeze the current toolpaths for the model. This setting is useful when multiple models are being printed and each requires a different Print Style. (Pro Feature)
- Count – The quantity of the model on the build tray.
- Height – The Z height of the model in the current orientation.
- Angle – The angle of rotation of the model about the Z axis (0 in the above image).
- Model Name – The file name of the model (Adaptive Half done v2 high in the above image).

11.1 Right Click Menu

The right click menu is accessed by right clicking on a model in the list.

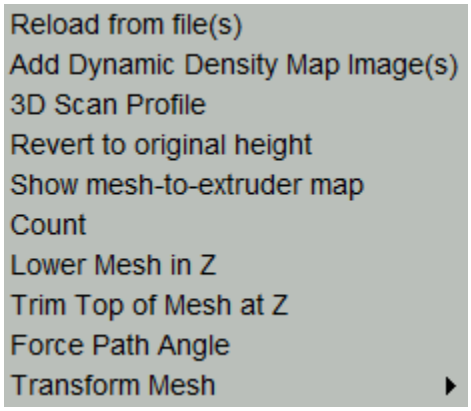


Figure 115 - Right click model menu.

- Reload from file(s) – Reloads the model (or models) from the original file.
    - Reverts the file to the original state when it was first opened.
- Add Dynamic Density Map Image(s) – The infill density can be set using a grayscale image.
- 3D Scan Profile – Unknown
- Revert to original height – Reverses any change to the “Height" in section 11.0.
- Show mesh-to-extruder map – Brings up the mesh-to-extruder map window (5.4.1).
- Count – Sets the count for the selected model.

- Lower Mesh in Z – KS typically aligns all models with the build plate. The Lower Mesh in Z can be used If the model should be positioned lower.
- Trim Top of Mesh at Z – The model will be trimmed at the Z value entered and anything above the Z value will not be printed.
- Force Path Angle – Infill and support paths will be forced to the entered angle.
- Transform Mesh – Sub-menu with options to control the size, orientation, and position of the selected model.

11.1.1 Transform Mesh

The Transform Mesh sub-menu has options to resize, move, and reorient the selected model.

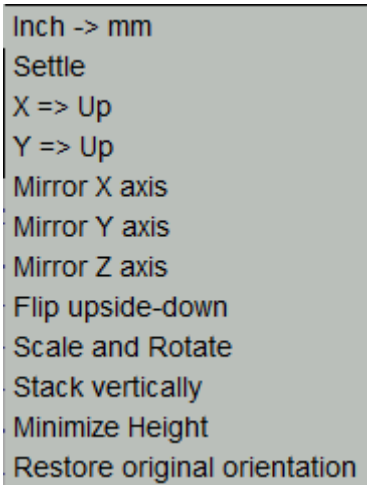


Figure 116 - Transform Mesh sub-menu.

- Inches -> mm – Converts the selected model units from inches to mm (multiplies units by 25.4).
- Settle – Reorients the part so that the nearest flat surface is placed parallel to the build plate.
  - The “Settle" command is best used once the part has been roughly aligned using the “Scale and Rotate command.
- X => Up – Orients the X axis of the part up relative to the build plate.
- Y => Up – Orients the Y axis of the part up relative to the build plate.
- Mirror X axis – Mirrors the part about the X axis. Plane normal to Axis.
- Mirror Y axis – Mirrors the part about the Y axis.
- Mirror Z axis – Mirrors the part about the Z axis.
- Flip upside-down – Flips the part so that the top is facing the build plate.
- Scale and Rotate – Brings up a window that can be used to rotate the part and change the scale.

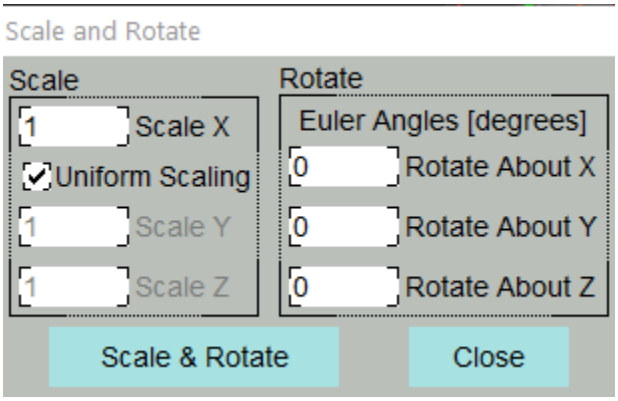


Figure 117 - Scale and Rotate window.

- Scale – A uniform scale can be entered (scales equally in all 3 directions) if the checkbox is checked. Otherwise, a separate scale should be entered for all three axis.
- Rotate – The part can be rotated about each of the three axis.
- The scale and rotations will only be applied once the "Scale & Rotate" button is pressed. Once the part is oriented correctly, press the "Close" button.

- Stack Vertically – Parts can be stacked vertically to fill more of the build volume. This feature should only be used with either breakaway or soluble support.

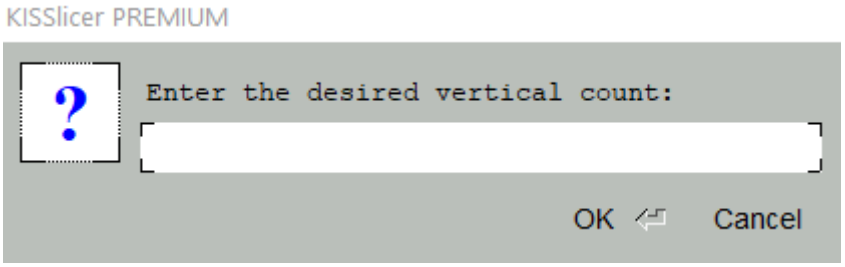


Figure 118 - Vertical count window.

- Once the number of vertically stacked parts is entered another window will appear.

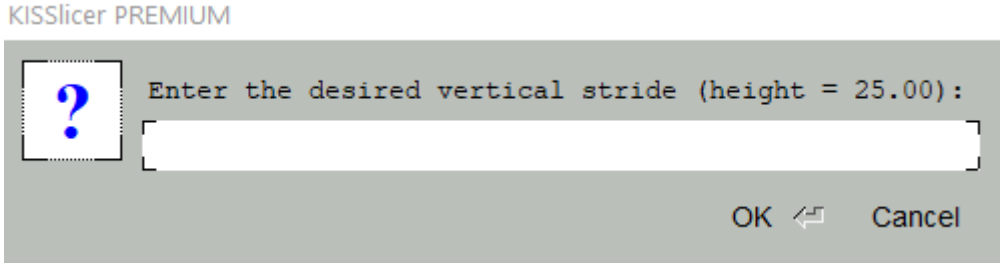


Figure 119 - Vertical stride window.

- The vertical stride is the height at which the next part starts. KS will show the height of the current part. The space between the parts will be filled with support material.

- Minimize Height – KS will attempt to find the orientation for the part that produces the smallest possible Z height.
    - This feature is currently experimental and is best combined with the “Scale and Rotate” command.
- Restore original orientation – Removes all transformations and restores the original orientation of the part.

12.0 Viewport

The Viewport is where the build plate, models, and toolpaths. Basic information about the print job is also displayed.

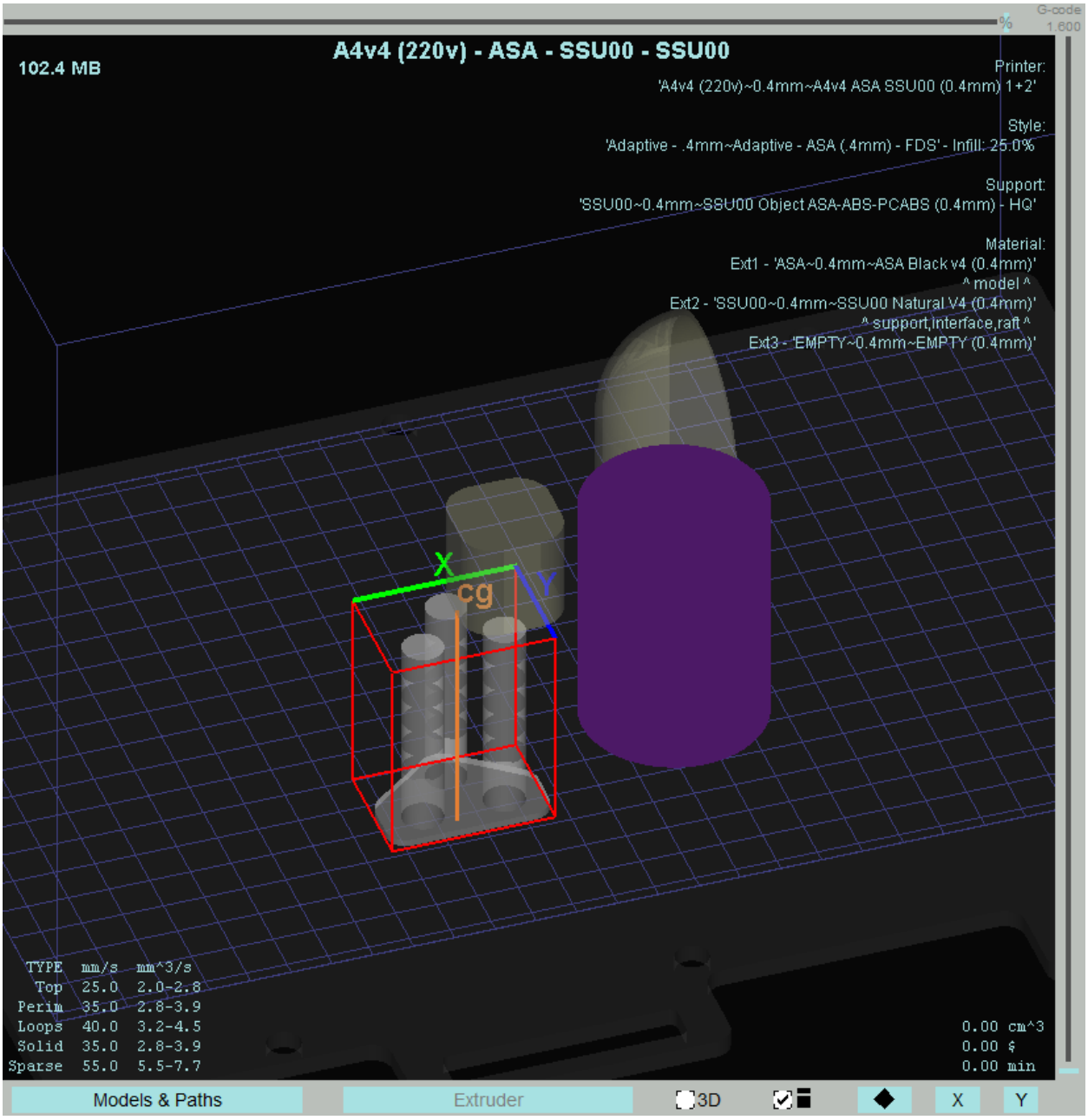


Figure 120 - Viewport.

12.1 Sliders

The sliders on the top and right-hand side control which layers are visible and the portion of the toolpath on that layer.

12.1.1% Slider

The top slider is the Layer % slider. Sliding it from left to right will show how the current layer will be printed.

12.1.2 Layer Slider

The right-hand slider selects the currently visible layer. Sliding it from bottom to top shows each layer as it would be built during the printing process.

12.2 RAM Usage

Ram usage is displayed in the top left-hand corner.

Figure 121 - RAM usage.

12.3 Printer Name and Materials

The printer name and the material selected for each extruder is displayed across the top of the viewport.

Figure 122 - Printer name and materials.

12.4 Slicing Parameters

The currently selected slicing parameters are displayed in the top right-hand corner.

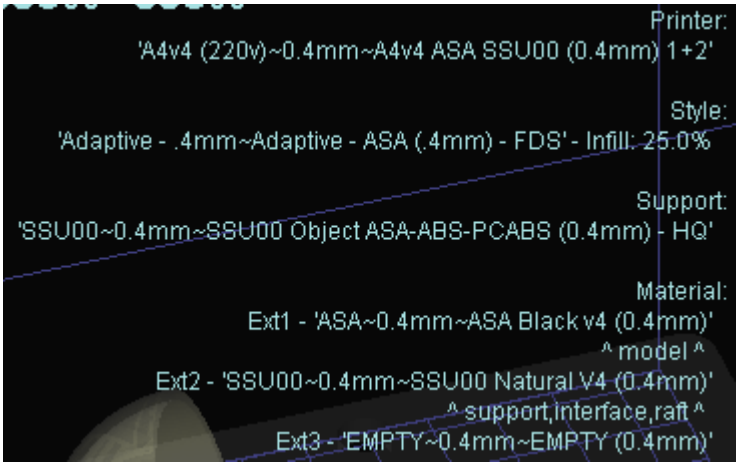


Figure 123 - Slicing parameters.

12.5 Printing Speeds and Volumes

The currently selected print speeds for the various path types and the corresponding extrusion volumes are displayed in the bottom left-hand corner.

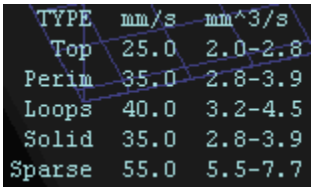


Figure 124 - Printing speeds and volumes.

12.6 Print Cost, Volume, and Time

The print volume, cost and total print time are displayed in the lower right-hand corner.

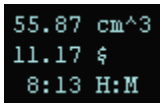


Figure 125 - Print volume, cost, and time.

12.7 Display Options

The display options are shown across the bottom of the viewport.

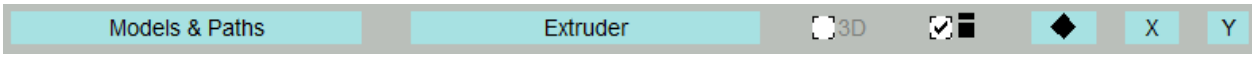


Figure 126 - Display options.

12.7.1 Model Display

- Model – Displays the model only with no toolpaths.
- Model & Paths – Displays the model and the toolpaths together.
- Paths – Displays the toolpaths only with no model.

12.7.2 Toolpath Colours

- Path type – Toolpaths are colour-coded by the type of path.
- Extruder – Toolpaths are colour-coded by extruder.
- Material – Toolpaths are colour-coded by material type.

12.7.3 3D Display

The 3D checkbox is used to render the toolpaths in 3D.

12.7.4 Layers Checkbox

The Layers Checkbox is used to draw all layers below the selected layer when checked. When unchecked the viewport will show only the current layer.