Post Processor ReadMe file

(Search for the post processor name to jump to the section)

Add Cooling Profile (Advanced Cooling Fan Control)

* Multiple instances can be run. Part of a print can be by “Layer Number” and another by “Feature Type”.
* If the fan speed is RepRap (0 to 1) the post processor will note the setting in Cura and adjust.
* This plugin supports up to 4 extruders (T0 to T3) and 4 fan circuits (P0 to P3)
* Layer numbers are according to the Cura Preview.
* If a fan speed is not 0% then the minimum fan speed is 12%.
* Settings:
  + Cooling Control By:
    - By Layer Number
      * Remove M106 lines prior to inserting new
        + For the first instance of this post processor, this should be checked so it removes any M106 lines that Cura would have inserted. For any subsequent instances this should be UN-checked or it will remove the M106’s inserted by the previous instance.
      * Layer / % #1 thru Layer / % #8:
        + Enter the Layer number you wish to make the change at followed by a “/” and then the fan percentage (0 to 100).
      * Enable Raft Cooling:
        + Check if you want the fan on at the beginning of the Raft.
        + You can select the fan % for the raft.
    - By Feature Type:
      * Remove M106 lines (See above).
      * Starting layer:
        + Enter the layer number to start making fan speed changes. Changes start at the beginning of the layer.
      * Ending Layer:
        + Enter the layer number to end the changes. Enter “-1” for the entire file.
        + If you enter a layer number then “Final Fan %” is available and that will be the speed from the end of the Ending Layer to the end of the file.
      * Feature Names:
        + These are the names that Cura uses in a gcode file and so they are searchable. Enter the fan % for each feature.
      * Fan Off During Combing:
        + When checked the fan speed will be set to 0% for combing moves.
      * Final %:
        + Available if you name an Ending Layer.
      * Enable Raft Cooling:
        + See above.

Add Cura Settings

* Adds the Cura settings to the end of the gcode. The settings have been filtered slightly.
  + All Categories
    - All the Cura settings (Machine, Quality, Speed, etc.).
  + Select Categories:
    - Pick and choose which groups to add to the gcode.
  + If post processors are running then the settings for the post processors will be listed (per All or Select).

Alter Z-hops layer to layer:

* Allows you to change the Zhop height across a range of layers.
  + Multiple instances can be run.
  + Zhops must be enabled in Cura.
  + Multiple extruders with different Z hop heights are supported.
  + Setting the new Zhop height to “0” will negate the Z hops but the gcode lines will remain in the file.

Bridge Temperature Adjustment:

* Change the hot end temperature for bridges.
  + Ignore Bridge Walls
    - Bridge walls are often a single line of Gcode. This gives an option to not pause and wait for the nozzle temperature to adjust for every bridge line.
  + Bridge Temperature Command
    - M109 R will wait for the temperature to either rise or fall to the new set point.
    - M104 S will set the new temperature and continue the print without waiting.
  + Bridge Temperature
    - Enter the temperature you want the bridging to print at. If there are multiple bridge layers they will all be affected.
  + Resume Temperature Command:
    - M109 R or M104 S
  + Resume Print Temperature:
    - The Temperature to continue the print after the bridging.
  + Park the Head at the:
    - The four corners or the midpoint (based on the size of the build plate) as entered in Cura).
    - This option is not available if both the Bridge Command and the Resume Command are M104 S.
  + Z Lift before Travel:
    - Provides a Z hop prior to the head parking.
    - This option is not available if both the Bridge Command and the Resume Command are M104 S.

Change at Z:

* Allows you to change a variety of settings during a print.
  + Enabled
    - Turn the post processor on or off.
  + Trigger
    - By Height or by Layer
      * NOTE: Sometimes By Height can be fooled by Zhops.
  + Change Height or Change Layer
    - Either the Height or Layer number to make the change at.
    - Layer numbers are Base0 so subtract 1 from the layer in the Cura preview.
  + Apply To
    - The target layer/height, or to the end of the file.
  + Output to Display
    - Adds an M117 line to send the info about the change to the LCD display. If you have other post processors running that send info to the display then they will over-write each other and only the last one will remain on the display.
  + Change settings:
    - The list of settings you may make changes to and the settings values.

Create Thumbnail:

* Adds a coded image to the gcode file. You can pick the size of the image in pixels.

Display Info on LCD:

* Combines two separate post-processors “Display Layer and Filename” and “Display Progress”.
* File Name and Layer example: M117 Printing Layer 6 of 169 CE3E3V2\_Pendant 5-30
* Display Progress Example: M117 54/250 | ET 1h38m
  + LCD Display Option
    - File Name and Layer
      * Scroll enable small layers:
        + If “Scroll long filenames” is enabled in your firmware you can select this setting.
      * Text to Display
        + You can add a short note that will be added to every M117
        + Leaving the box empty will insert the File Name into the message.
      * Initial Layer Number
        + “1” will number the layers to match the Cura preview (base1). “0” will match the layer numbering of the Gcode (base0).
      * Display Max Layer
        + Formats the message as “Layer 123 of 500”.
      * Add Prefix “Printing”
        + Results in “Printing “ added before the other options.
      * Add M118 line
        + M118 will contain the same information as M117. M118 will bounce the message back through the USB to a print server.
    - Display Progress
      * Display Total Layers
      * Display Remaining Time
      * Add M118 line
      * Time Fudge Factor
        + Error correction for the Cura time estimate. If your printer generally finishes a print in 75% of the time that Cura estimated then enter 75 in the box. ActualTime / Cura Estimate / 100 = Fudge Factor.

Fast First Infill:

* Adjust the speed of the Infill on the Initial Layer

Filament Change:

* Uses M600 to configure the printer firmware parameters for a Filament Change at a Layer.
  + Enable
    - Turns the post processor on and off.
  + Layer
    - The layer to make the change at. The change will occur at the start of the layer number you enter per the Cura preview. If you want the change at the start of layer 10 then enter 10.
  + Use firmware configuration
    - Will enter M600 into the gcode with no parameters. The firmware will handle all other settings.
  + Initial Retraction
    - The initial retraction about.
  + Later Retraction
    - The Unload amount
  + X position
    - The Park Head X location
  + Y position
    - The Park Head Y location
  + Z position
    - The Zhop height prior to parking the head.
  + Retract Method
    - Firmware dependent
  + Enable Gcode before
    - Allows you to enter a gcode command that will run prior to M600. Example: M300. For multi-line insertions delimit with a comma.
  + Enable Gcode After
    - Enter a gcode command that will run after M600. For multi-line insertions delimit with a comma.

Insert at Layer Change:

* Adds a command at Layer Changes.
  + How often to Insert
    - A single instance only, or every layer, every 2nd, 3rd, 5th, 10th, 25th, 50th, 100th.
  + Start Layer
    - The layer number to start the insertions at. Use the Cura preview number. For a single insertion enter the layer number.
  + Enable End Layer
    - Allows you to enter an Ending Layer number. Use -1 for the entire file.
  + Gcode to insert
    - Enter the command. For multi-line insertions delimit with a comma. All commands are converted to upper case.

Limit the X-Y Accel/Jerk:

* For bed-slinger printers. This allows you to change the acceleration and jerk at a layer. If you enter a value that is lower than the Cura setting then both Travel and Print accel will be affected.
  + X Max Acceleration
    - The number is pulled from Cura when the Post Processor loads. Enter the maximum X Accel you want
  + Y Max Acceleration
    - The number is pulled from Cura when the Post Processor loads. Enter the maximum Y Accel you want.
  + Change the Jerk
    - Enables separate Jerk values for the X and Y axes. If the Jerk is set to 0 then that axis will remain whatever it was set to in Cura with Travel and Print jerk as it was set. If a hard number is entered then it will be the number used for both Travel and Print. The initial numbers are pulled from Cura.
  + From Start of Layer
    - Enter the layer you wish to make the change at.
  + To End of Layer
    - Enter the layer you want to end the change at. Enter -1 to keep the change until the end of the file.
  + Gradual Accel Change will gradually alter the Accel across a range of layers. The Accel will remain at the final under until the end of the file. If Jerk is enabled then the Jerk will change at the start layer and continue to the end of the file. It is not gradual.
  + Gradual From Layer
    - Enter the starting layer per the Cura preview.
  + Gradual To Layer
    - Enter the layer for the final Accel.

Little Utilities:

* Four post processors that may occasionally be helpful.
  + Remove Comments (this can help speed up the gcode transfer to a printer)
    - This removes everything to the right of a semi-colon. That includes the entire line “;TIME\_ELAPSED:12345” and the comment portion of “G28 ; Auto-Home”.
    - You can exclude the Gcode opening section, the Cura StartUp gcode section, you can choose to leave the “;LAYER:” lines, and you can exclude the Ending Gcode.
  + Renumber or Revert
    - For One At A Time prints you can re-number the file to All At Once style. That allows greater use of Pause At Height and Filament Change. A side effect is that the gcode file will preview correctly in Cura. A second instance of the post processor can be used to Revert to One At A Time numbering.
  + Add extruder end gcode
    - A bug in Cura kept the final tool change Ending Gcode from being inserted at the end of the print. This inserts it.
  + Add Data Headers
    - A debugging tool, this adds a line to show the splits in the data received from Cura during post-processing.

MultiBrim:

* Adds additional layers to the brim. They don’t show in the Cura preview until you load the Gcode file.

Multi Extruder Color Mixer:

* For “multiple in one out” hot ends. Supports up to 3 mixing extruders as on the Geeetech AxxT printers. Allows color mixing using the M163 and M164 command. Multiple instances can be run on different layer ranges. The firmware must support the commands.
  + Mix Style
    - Constant
      * A fixed mix ratio. You set the ratio for each extruder. The sum of the percentages must equal 100.
    - Gradient
      * A gradual mix ratio from one set of percentages to another. Each included extruder will have a starting ratio and an ending ratio. The Start Ratios must add up to 100 and the ending ratios must add up to 100.
  + Start Layer
    - The layer to start the mixing at
  + End Layer
    - The layer to complete the mixing at. Enter -1 to continue to the end.
  + Ext 1 in the mix
    - Include extruder 1. Allows you to set the ratio. If “Gradual” is selected then you also set the Ending Ratio.
  + Ext 2 in the mix
    - Include extruder 1. Allows you to set the ratio. If “Gradual” is selected then you also set the Ending Ratio.
  + Ext 3 in the mix
    - Include extruder 1. Allows you to set the ratio. If “Gradual” is selected then you also set the Ending Ratio.
  + Park and Purge (when selected it occurs before the mixing extruder is used)
    - Enter the X and Y park location. Enter the amount of filament to purge to change the color.

Pause at Height:

* Use to change filament or to insert nuts, magnets, etc. into a print.