Cura Post Processors

Contents:

Open Post Processor ReadMe File (PDF):

- Opens this file in the default viewer. If the post processor is left in the active list then this file will open on Cura startup.

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.

(Back to Contents)

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.) including all settings for any post processors running.
    - Select Categories:
        - Pick and choose which groups to add to the gcode.

- General, Machine, Quality, Wall, Top/Bottom, Infill, Material, Speed, Travel, Cooling, Support, Build Plate Adhesion, Dual Extruder, Mesh Fixes, Special Modes, Experimental, and Post Processors.

(Back to Contents)

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.

(Back to Contents)

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.

(Back to Contents)

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.

(Back to Contents)

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.
    - Countdown To Pauses
        - Changes the "ET" to "TP" and the time counts down to the next pause.

(Back to Contents)

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.

(Back to Contents)

Insert at Layer Change:

- Adds command(s) 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.

(Back to Contents)

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 acceleration value 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.

(Back to Contents)

Little Utilities:

- Some 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.
  - Lift Head Parking

        - This will add a movement to get the nozzle away from the print during the G4 pause. The movement is calculated as the shortest orthogonal move to the skirt/brim/raft rather than all the way to a corner.
- Change Printer Settings
    - Adds changes to the Max Feedrate, Max Accel, Home Offsets, and Steps/MM to a gcode file. An additional option can save the settings to make them the new defaults in the printer.
        - Change Max Feedrate (X, Y, Z, E axes)
            - The default values are what is listed in Cura Printer Settings and should be the defaults from the printer definition file.
            - Black input boxes mean no change to that axis.
        - Change Max Accel (X and Y axes)
            - The defaults are from Cura Printer Settings.
            - Blank input boxes mean no change to that axis.
        - Change Home Offsets (X Y Z axes)
            - Change any or all three axes. This will move a print on your build plate or can allow you to print an "origin at center" gcode file on any printer.
        - Change X Y Z E Steps/MM
            - Will add an M92 line to change the steps/mm of any axis.
            - Blank input boxes mean no change to that axis.
        - Save All Changes (M500)
            - Sets the defaults in the printer to any new values you enter here. If you don't save then these settings are only effective until the printer is turned off (the printer would restart with the previous defaults).
            - If you do save the settings, then the "Printer Settings" in Cura will update as well.
- Fan-Only ToolPath
    - Creates a zigzag toolpath at the end of a layer that the print head will follow. The print head will z-hop 1mm and run back and forth across the print with the layer cooling fan on. There are retractions and primes when necessary. After completion the print resumes.
        - End of which layer(s)
            - Enter a layer number using the Cura preview. The cooling path will be inserted at the end of the layer. For multiple layers delimit with a comma. For ranges of layers delimit with a dash ("-").
            - Example of single layers mixed with ranges of layers:
                - 24,30,101,115-119,151,175-200
            - Example of a single layer range:
                - 25-30

        - Example of a single layer
        - 13
    - Tool Path speed mm/sec
        - Enter the speed you want the print head to run the path.
    - Tool Path Cooling Fan %
        - Enter the speed percentage for the fan.
    - Add Y zigzag path
        - If enabled a second path will be added. This doubles the cooling effect and the time to complete the moves.

(Back to Contents)

MultiBrim:

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

(Back to Contents)

Multi Extruder Color Mixer:

- For "multiple in one out" hot ends. Supports up to 3 mixing extruders as on the Geeetech AxxT printers. (You can use the post with a 4 extruder mixing hot end but only extruders 1, 2, and 3 can be mixed.) Allows color mixing using the M163 and M164 command. Multiple instances can be run on different layer ranges. The firmware must support the commands. Multiple instances can be run so different mixes can occur at different layers for a variety of effects.
    - Prime Towers are not allowed.
    - The post removes all Tool Changes from the Gcode so any changes must be accomplished with the post processor.
    - The post also removes all temperature lines beyond the startup gcode. That means your Initial Layer Print Temperature continues throughout the print rather than changing to Print Temperature at the end of LAYER:0. I intend to revisit that.
    - 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. The ratio changes at layer changes.
    - 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.

(Back to Contents)

Pause at Height:

- Use to change filament or to insert nuts, magnets, etc. into a print. Multiple instances can be run on a print. (Note that using *By Height* can be fooled by Zhops).
    - Pause At:
        - Choose Height or Layer Number.
    - Pause (Height or Layer depending on the above)
        - If a height can't be hit exactly then the next height will be used.
        - If you use Layer Number then the pause will occur at the END of the layer you enter when looking at the Cura preview. Example: You enter "20" as the layer number. Per the Cura preview the pause would occur between layer 20 and 21.
    - Method
        - The command that is inserted into the Gcode. It is usually specific to the Firmware Flavor.
    - Keep Motors Engaged
        - If the stepper motors are allowed to disable they will lose their position so the printer no longer knows where the nozzle is. The default "time out" on most printers is 120 seconds (2 minutes). The max time you can enter might be as high as 14400 seconds (4 hours) but it is firmware dependent. Some printers will accept "0" seconds to mean "never disable" and for other firmware it means "disable now".
        - When selected - the steppers will not lose then position for 30 minutes (1800 seconds).
        - When un-checked the "Disarm Timeout" box is available and you can enter a custom time at the end of which the steppers will disable.
    - Park Print

        - When enabled will allow you to pick an X and Y location for the nozzle to travel to where it will wait during the pause. It should be a corner where the nozzle won't ooze onto the print.
    - Retraction
        - The length of filament to pull back before the pause.
    - Retraction Speed
        - The speed you want to use for the retraction
    - Extrude Amount
        - The length of filament you want the printer to extrude after the pause. This is the purge amount.
    - Extrude Speed
        - The speed for the extruder to prime at in mm/sec.
    - Re-do Layer
        - This can help get the filament flowing correctly again.
    - Use M109 or M104 for the resume temperature.
        - M109 will often impose a 10 second delay even if the temperature is the same as the standby temperature. It must be used when the temperature is different from the standby.
        - M104 has no wait period and printing can start up faster. This is meant to be used when the resume temperature is equal to the standby temperature.
    - Stand-by Temperature
        - The temperature to use while the print is paused.
    - Display Text
        - The message to be displayed on the LCD during the pause.
    - Beep at Pause
        - The length of the beep is dependent on Beep Length in milliseconds.
    - Gcode Before
        - The gcode to be inserted prior to the pause line. For multi-line insertions delimit the code with a comma. Ex: M220 S1,M999,M115
    - Gcode After
        - Same rules as above.

(Back to Contents)

Reduce Z Brim:

- Reduce the layer height of just the brim on the initial layer.

(Back to Contents)

Retract Continue:

- Instead of a retraction-then-move, this spreads the retraction across any following combing moves to reduce stringing.

(Back to Contents)

Search and Replace:

- Replace text in the file with new text. Multi-line capable replacements when separated by the python" \n" newline character.
    - Search
        - Enter the string to search for. This is case sensitive. You can also enter regular expressions. Ex Search: ;LAYER(\d*) will find all the layer number lines. ;LAYER:1\n with regular expressions checked will find only ;LAYER:1 and not LATER:11, or LAYER:100.
    - Replace
        - The replacement string. It is not case sensitive so if you are putting a command in like M117 then you should capitalize the M or G as some firmware does not understand lower case letters.

(Back to Contents)

Stretch:

- A 6 year old post processor to correct dimensional problems with cylinders, holes, and curves. It may be obsolete do to new capabilities in Cura.

(Back to Contents)

Time Lapse:

- Usually inserts M240 to signal a camera.
    - Camera Trigger Command
        - Usually M240 but you may enter the command of your choice.
    - How Often?
        - Every layer, every 2^{nd}, 3^{rd}, 5^{th}, 10^{th}, etc.
    - Pause Length
        - Enter the number of milliseconds to pause.
    - Park Print Head
        - When enabled you will enter the coordinates that you want the nozzle to travel to and wait for the pause.
    - Retract when required
        - Checks to see if the pause happened during a retraction. When enabled, if there was no retraction then a retraction and prime will be added.
    - Z-Hop height when parking
        - The height you want the nozzle to raise from the print during the photo.

(Back to Contents)

Timed Cool Down:

- Slow down the cooling of the bed to provide "annealing" for certain materials that require it. If there is a heated build chamber, and if you include it in the cool down
    - Hold the temp for the:

        - Bed
        - Bed and Chamber
    - Hold time @ Bed/Chamber Temp
        - Holds at the current bed (and Chamber) temperature for the specified time in decimal hours.
    - Shut Off Temp
        - The lowest temperature to "time-cool" the bed to. When it is reached the bed will turn off. The minimum value is 30°
    - Cool Down Span
        - The length of time (in decimal hours) to stretch the cool down across. The temperature will drop in 3 degree increments from the Print Bed Temperature to the Shutoff Temperature.
    - Park head and Max Z
        - Will move the nozzle to Max Y and Max Z in case you want to put a cover over the print.
    - Beep
        - Annoying noise from the printer when the bed shuts off. You can select the duration of the beep in milliseconds.

(Back to Contents)

Use Previous Probe Measurements:

- Will remove the G29 command and insure that previous probe measurements are loaded and enabled.

(Back to Contents)