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 / Fan%" #1 thru "Layer / Fan%" #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 Fan %:
        - 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 but there are over 300 is you select "All".
    - 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.
    - Max/Min Speeds in the Gcode
        - This collects the Initial Layer Speeds and then the rest of the speeds in the file and determines the max and min for both travel and printing. The data gets added to the beginning of the Gcode file and also at the end. There is a pop-up information window that comes up after saving a Gcode file.

(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 (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. This will normally be the regular printing temperature.
    - 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 because there is no reason to wait.
    - 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 (Cura Script):

- 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 (Cura script):

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

Display Info on LCD:

- File Name and Layer example: M117 Printing Layer 6 of 169 CE3E3V2_Pendant 5-30
  - 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.
      - Time Fudge Factor (see below)

        - Enabled when “Enable End-Time Message” is selected or “Display Progress” is selected.
    - Enable End-Time Message (see below)
- Display Progress options:
    - 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. This is also used when calculating the “End Time” of a print.
    - Countdown To Pauses
        - Changes the “ET” to “TP” and the time counts down to the next pause. Little Utilities must below all PauseAtHeights / PauseAtLayers/M600 Filament Changes in the post processor list for this to work correctly.
    - Enable End-Time Message
        - After a slice is completed a message will pop up to inform the user when the print would end if they were to start the print right now.
        - There is also an option to enter a start time (for example it’s 4pm and you want to know the finish time if the print starts at 8am tomorrow).

(Back to Contents)

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. The first letter of all commands is 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. It is possible to limit the Accel at the beginning of a print and then allow it to increase at a point in the print, or gradually across a range of layers.
    - 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. If the firmware is “RepRap” then M566 is used for Jerk.

- 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. Jerk is not gradual because it has no limit setting.
- 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.
    - Note that the last section of a Cura gcode is "SETTINGS_3:". It is not present during post processing and so cannot be deleted.
  - Add last extruder ending gcode
    - A bug in Cura kept the final tool change Ending Gcode from being inserted at the end of the print. This inserts it.
  - 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.
  - Very Cool FanPath
    - 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 of the cooling path 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 at 90° to the first path. This doubles the cooling effect and the time to complete the moves.

- Renumber Layers
    - For One-at-a-Time prints you can re-number the file to "All-at-Once" style. That allows greater use of Pause At Layer 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.
- 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.
- Add Data Headers
    - A debugging tool, this adds a line to show the splits in the data received from Cura during post-processing.
- Add line numbers to the Gcode

        - Some firmware requires line numbering in a Gcode file. The numbers are sequential. The options are:
            - Add a Prefix (Ex: "N1234").
            - Pick a starting number.
            - Skip comment lines.
    - Create a debugging file.
        - Pick a range of layers and the gcode will be edited to remove all extrusions and "M" code commands. The resulting Gcode can be run without any heating of the bed or hot end so a toolpath can be checked.
            - Enter your Auto-home command (usually G28).
            - Starting Layer Debug File. Enter the layer you want to start from. All layers previous to the start layer will be deleted.
            - Ending Layer Debug File. Enter the end layer. All layers after the end layer will be deleted. Use "-1" to indicate the end of the file.
    - Disable ABL for small models.
        - Enter the minimum area of a model and if the print is below that area then "G29" and "M420" will be disabled in the startup.
    - One-at-a-Time Final Z Move.
        - When printing One at a Time, if the last model is shorter than the others then the final Z move of Cura will be 5mm above that model, but can be below the tops of other models. An XY movement in the ending gcode can then crash the nozzle in to a taller print. This is a fix for that and adds a Z move to "transit height" before the ending gcode runs.
    - Adjust Per Model Print Temperature
        - Also for One-at-a-Time prints, you can change the print temperature for each model.
    - Custom Script
        - You can open the Little Utilities file and add your own script here. As it is, there is one setting written and the command just pops up a message.

(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" mixing hot ends. Supports up to 4 mixing extruders. Allows color mixing using the M163 and M164 command. Multiple instances can be run on different layer ranges. With multiple instances different mixes can occur at different layers for a variety of effects.
    - The firmware must support the commands.
    - Prime Towers are not allowed.

- The post removes all Tool Changes from the Gcode so any and all changes must be accomplished with this 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.
- 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 Layer:

- Use to change filament or to insert nuts, magnets, etc. into a print. Multiple instances can be run on a print.
  - 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.
  - Pause Command
    - The command that will actually pause the printer. It is usually specific to the Firmware Flavor. You can choose "Custom" to enter a command that might not be in the list.
  - Reason for Pause

- "Filament Change" will enable the park and unload and load settings. Printers can have a built-in safety to avoid over-long extrusions. The unload and reload amounts are broken into 150mm chunks to avoid that. A user shouldn't notice but it does show up in the gcode.
  - Unload Amount is the amount of filament to pull back for the change.
  - Reload Amount is the amount of filament to push forward towards the nozzle. The first 80% will be at your Max E speed and the final 20% will be at a reduced speed to the filament doesn't ram into the back of the nozzle and cause missed steps.
  - Purge Amount is the amount to purge to change colors or material.
- "All Others" disables those settings as they aren't usually required if you are just making insertions.

- 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 the number of minutes you enter into the box.
  - When un-checked the firmware setting for "disarm timeout" will be used.
- 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. You can also enter a z-hop height. The printer should park in a corner where the nozzle won't ooze onto the print. The park position should also allow access to the print for any insertions you might be making.
- Standby Temperature
  - The temperature to hold the hot end at during the pause.
- Resume
- 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.
- Resume Temperature Command
  - Use M104 if the resume temperature is the same as the standby temperature. If they are different they use M109.

        - Multi-extruder printers have an option to enter the resume temperature or to allow the program to pick the temperature of the current tool.
    - Resume Temperature
        - This will be the temperature for the new filament. If there are temperature lines later in the file then this could be overridden.
    - Message to LCD
        - 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.
    - Re-do layer
        - Some folks like to do the previous layer over again to get the new color flowing. If you select this then the Flow setting will be available. After the "redo" layer the flow is reset to 100%.
    - Gcode Before
        - The gcode to be inserted prior to the pause line. For multi-line insertions delimit the code with a comma. Ex: M220 S100,M999,M115
    - Gcode After
        - Same rules as above.

(Back to Contents)

Retract Continue:

- Instead of a retraction-then-move, this spreads the retraction across any following combing moves to reduce stringing. This doesn't work with Relative Extrusion.

(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.
    - Use Regular Expressions
        - Allows Regex replacement searches.
    - Enable Search within a Layer Range
        - You can limit the search to a section of the file.
    - Replace first instance only

        - Once a single match and replacement is made the script exits.
    - Ignore StartUp
        - Don't make any replacements within the startup gcode.
    - Ignore Ending
        - Don't make any replacements within the ending gcode.

(Back to Contents)

Support Interface Material Change:

- The provides the ability for a single extruder printer to apply a different material to the top layer of a "Support-Interface". This is excellent for large flat support surfaces like the underside of a box top that needs to be printing right-side up. Using a non-compatible material for the interface and setting the "Top Distance" to 0 and the "Top Interface Density" to 100% will result in support that comes off with very little fuss, and a surface of the "first layer over support" that rivals a top layer. Using PETG for the interface of a PLA print or vice versa works very well. Each material change requires 2 pauses. Trying to do all the interfaces of print can get very tedious very fast but for one or two it works very well. NOTE: There must be a "TYPE:SUPPORT-INTERFACE" line in each "layer of interest". There is a message that will popup to inform the user whether or not adding the pause code was successful.
    - Enable this script.
        - It must be enabled to run. Disabling allows the script to remain in the Post Processor list and be dormant until enabled.
    - Pause Command
        - The command to insert into the gcode that will cause the printer to pause so the filament can be changed. There are several options.
    - G4 Dwell time (in munutes)
        - If G4 is selected as the Pause Command then this setting is available to set the length of the pause.
    - Layer Number(s) for material change
        - These are the "Layers of Interest". Use the numbers from the Cura preview. You may enter a single layer, multiple layers delimited with commas, and/or a range of layers delimited with a dash.
            - Entering 10,12,15-18,25 would indicate that pause should occur at layers 10, 12, 15, 16, 17, 18, 25. Since each layer receives two pauses this particular example would be really annoying.
    - Model Material (msg to LCD)
        - Enter the material type of the main model
    - Model Print Temperature
        - The printing temperature of the model material.
    - Interface Material (msg to LCD)
        - The material being used for the interface.
    - Interface Material Print Temperature
        - The temperature that you with the Interface material to be printed at

- Interface Flow Rate
  - The % flow rate of the interface (M221)
- Interface Feed Rate
  - The % feed rate to be used for the interface (M220)
- Unload Filament Amount
  - The amount of filament to unload to back the filament out of the extruder. Set to 0 to disable.
- Load Filament Amount
  - The amount of filament to reload. I have found that 75 to 100mm less than the unload amount works well as the filament must be started past the extruder gear.
- Enable purge after each change
  - Purging is critical as the two materials must be incompatible. When reverting from the Interface material back to the Model material don't skimp. The purge from model material to interface material is done in one step. Purging from Interface material to the Model material is done in three steps with retractions and 1 second pauses in between in an effort to clean all the interface material from the nozzle.
    - Model Material Purge Amount
    - Interface Material Purge Amount
- Park head for changes
  - Park X location
  - Park Y location
- Beep at Pauses
- Add M118
  - Sends the same messages that are sent to the LCD but back to a print server like OctoPrint.

(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 2nd, 3rd, 5th, 10th, 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)

Time Lapse Camera

- Another script similar to Pause at Layer.
  - Camera Trigger Command
    - Typically M240 you can enter what works with your system.
  - How Often
    - Every layer or every 2nd, 3rd, 5th, 10th, 25th, 50th, or 100th.
  - Pause Length in milliseconds
    - How long to wait for the camera to complete the shot.
  - Park Head
    - Pick the location to park at.
  - Retract When Required
    - When checked the script will add retractions if there wasn't one before the pause. When un-checked then the script will not add retractions.
  - Z-hop when parking
    - The relative height to move the nozzle above the print prior to parking.

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 at MaxY and MaxX
    - Will move the nozzle to Max X and Max Y to insure the nozzle is away from the print.
  - Move to MaxZ
    - Moves the Z to provide room for adding 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)