

Fabricator Interface User Manual

Contents

0.1	About the Fablicator Interface	1
0.2	Reporting issues	1
1	Basic Usage	3
1.1	Starting a print	4
1.1.1	Loading files	4
1.1.2	Printing	4
1.2	Manual movement controls	5
1.3	Manual temperature controls	5
1.4	Manual extrusion	6
2	While printing	7
2.1	Stopping and resuming prints	8
2.1.1	Pausing prints	8
2.1.2	Stopping prints	8
2.1.3	Restarting prints	8
2.2	GCode viewer	9
2.3	Changing print speed and flow rate	9
2.4	Excluding parts	10
2.5	Auto-off feature (Added Jan. 3, 2020)	11
3	Settings	13
3.1	Overview of settings	14
3.2	Viewer settings	15
3.2.1	Switching between 2D and 3D viewer	15
3.2.2	Using a light 3D view	15
3.3	Interface layout	16
3.3.1	Interface mode	16
3.3.2	Controls mode	16
3.3.3	Temperature graph	16
3.4	Defining macros and creating buttons	17
3.4.1	Creating a macro	17
3.4.2	Creating a button	18

0.1 About the Fablicator Interface

The Fablicator Interface is a derivative of the open source project **Pronterface**. This program is meant to communicate with and control 3D printers (and similar CNC machines). Our version slightly modifies the user interface (UI) to work better with our printers.

0.2 Reporting issues

If you find a bug with this interface, please report it to: **technicalsupport@fablicator.com**

It helps to include screenshots of the interface and, if possible, a screenshot of any errors that pop up in the terminal!

1. Basic Usage

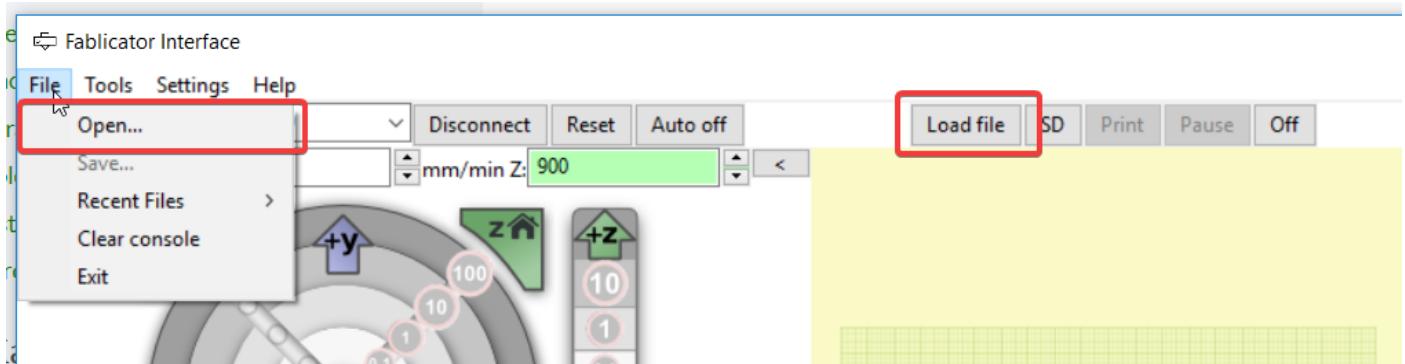
This section will teach you how to:

- Load files and start prints
- Home the axes
- Move the extruders and bed
- Heat up the extruders and bed
- Extrude filament

1.1 Starting a print

Loading files

In the interface, we can either use the "Load file" button or go to File > Open... to open a new file.

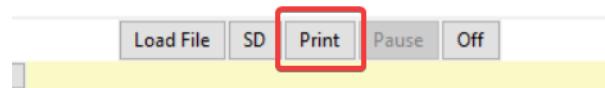


This will open the explorer so that you can select your file.

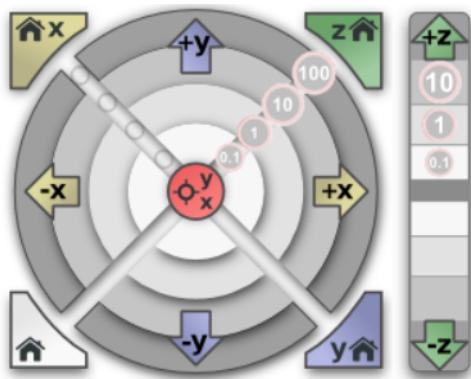
If you want to print a file that you've recently printed. It will be under the Recent Files drop-down menu shown in the picture above. Files can also be dragged and dropped into the main window of the interface. Double-clicking a .gcode file will open it in the current window(... or open a new window if the Fablicator program isn't open).

Printing

After opening the file, the program will parse the GCode and generate a preview. Start the print by pressing the print button above the GCode Viewer.



1.2 Manual movement controls



The control wheel pictured above is used to move the printer.

The buttons in the top-left, top-right, and bottom-right are used to home the individual axes. The button on the bottom-left is used to home all axes. **This should always be done before moving the axis, because it zeros the axis locations and tells the printer where its parts are. Failure to do this may cause parts of the printer to collide with each other, causing damage.**

The directional buttons on the wheel are used to jog the head in the X- and Y-axis and the slider is used to move the Z-axis. The numbers on each control corresponds to the distance to move per click (in mm). The red button in the center moves the X- and Y- axis to the center.



On MX machines, the head can be selected on the tool selector shown above. This makes the X-axis buttons move that extruder instead. **0 L** corresponds to the left extruder and **1 R** corresponds to the right extruder.

1.3 Manual temperature controls

The temperatures of the bed and extruders can be set using the temperature dropdown menus.



You can also manually type into this box to set a custom temperature. After setting the temperature in the dropdown, click **Set** to tell the machine to set the temperature.

To turn off the heating element, click the off button to the left of the dropdown.

Reminder: When running the GCode file, the temperatures of the nozzles and bed will be set to whatever was selected in the slicer.

On MX machines, the tool selector can be used to select the extruder to be heated.

1.4 Manual extrusion

Length:	Speed:
10.0	mm @ 1000.0
<input type="button" value="▲"/>	<input type="button" value="▲"/>
<input type="button" value="▼"/>	<input type="button" value="▼"/>
	mm/ min

The extruders can be controlled by the **Extrude** and **Reverse** buttons to the right of the tool selector. Below that we can specify the **Length** of filament to extrude (usually 60 mm) and the **Speed** at which we extrude the filament (usually around 100 mm/min).

2. While printing

This section will teach you how to:

- Stop and resume/restart prints
- Orient and read the GCode viewer
- Use the autoshutdown feature
- Adjust printing speed
- Exclude parts

2.1 Stopping and resuming prints

Pausing prints

In some cases you may want to stop a print to do something with the printer or part that it's printing. For example, if you were printing hollow paper weight, you may want to print up to a certain point and pour sand into it before the print finishes.

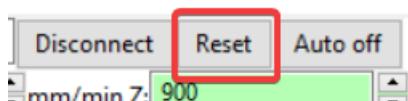
In this case, you must use the **pause button** located at the top of the window.



The pause button doesn't immediately stop the print, but pauses after it finishes the moves it has planned. When it is pressed it's replaced by a resume button allowing you to resume the print.

Stopping prints

In some cases, you may require the printer to stop what it's doing immediately. This is where you would use the **reset button**.



The reset button literally resets the serial connection which causes the controller to reboot and stops the print. The print cannot be resumed after a reset.

Restarting prints

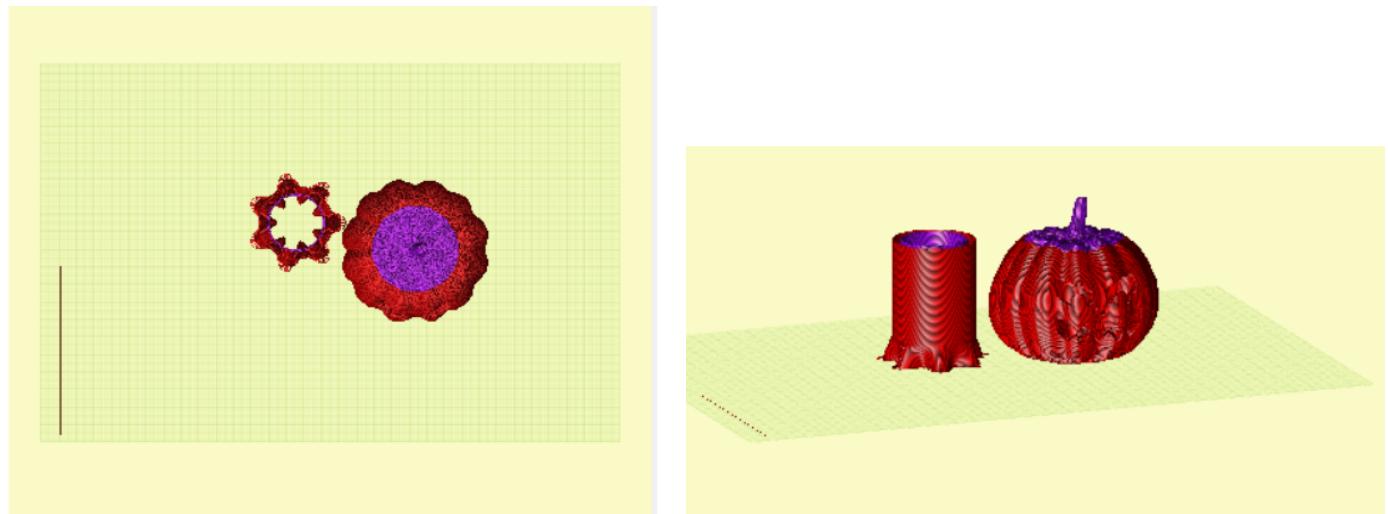
To restart the print, press **pause** or **reset**, home all axes, prep the bed, then press **restart** or **print** depending on what button is available.

Do not press restart without clearing the bed! Pressing **restart** causes the printer to immediately start the print from the beginning, allowing little time to prepare the print again.

Failure to clear the bed before printing will cause damage to the bed and extruders!

2.2 GCode viewer

The GCode viewer allows you to preview and track the progress of your print. By default, the viewer renders your GCode in 3D.



- The viewer can be **rotated** by dragging in the viewer using the **left mouse button**.
- You can **pan** around the viewer by dragging with the **right mouse button**.
- You can **zoom in** by **scrolling** the mouse wheel in the viewer.

In the above pictures, you can see that preview of the part has multiple colors. By default, red and purple correspond to the left and right extruders respectively.

While printing, the green paths indicate moves that were already sent to the printer.

2.3 Changing print speed and flow rate

Be careful! Changing the speed and flow rate of the printer may:

- Create uneven surface quality
- Overrun motion planner (for high detail parts), causing rough stepping

The print speed and flow rate can be changed by adjusting the sliders above the temperature gauges.



These two sliders are capped at 150% for safety.

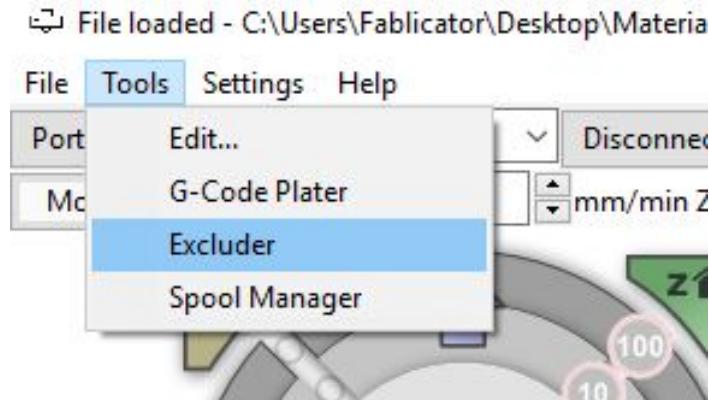
Changing the print speed changes the speed of all moves being sent to the controller. The speeds and accelerations of the moves are still controlled by the firmware settings

Changing the flow rate causes the printer to over- or under-extrude filament. This could be useful for creating water tight prints.

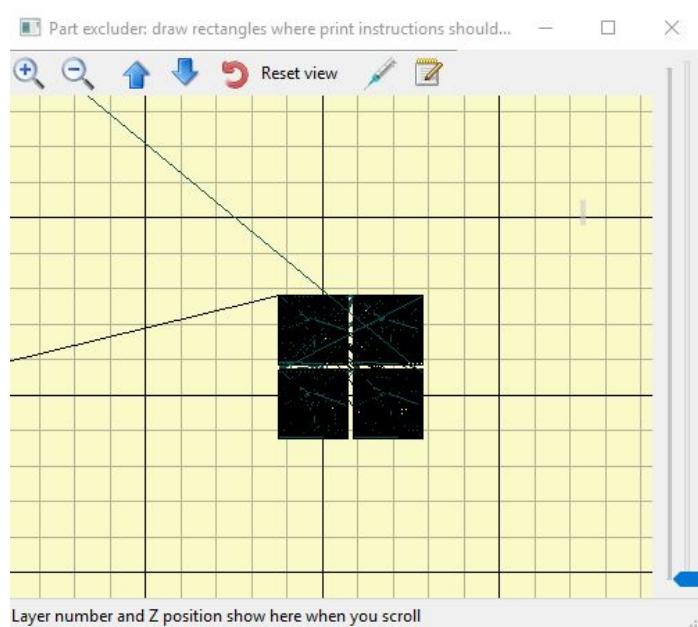
2.4 Excluding parts

If you print a group of parts and one part fails it may cause problems printing the other parts. The interface allows you to stop printing in a section of the bed using the part exclusion tool.

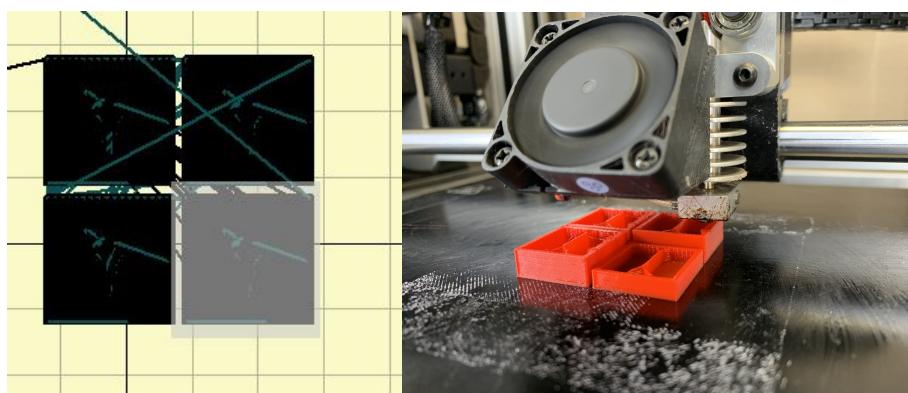
To access the Part Exclusion tool, click the tool drop down menu in the top left corner and click **Excluder**.



Once you open the Excluder menu you will see a 2D view of your print on the current layer.

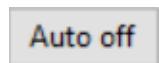


Drag a box around what you want to exclude from the print.

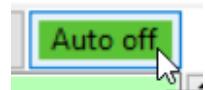


Once you exclude something you cannot make it print in that area again. In order to make it print normally again, the file must be reloaded.

2.5 Auto-off feature (Added Jan. 3, 2020)



The **Auto off** button located to the right of the reset button toggles whether the printer will shut down after printing or not.



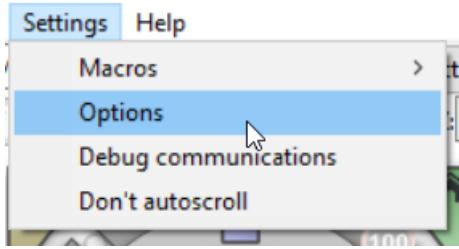
Pressing the button will make it turn green, which means that the machine will shut down after the print is finished. **To disable this, just press the button again so that it turns grey again.**

3. Settings

This section will teach you how to:

- Modify look of GCode viewer
- Modify interface layout
- Define macros and make buttons

3.1 Overview of settings



You can access the settings by going to **Settings > Options**. This will open a window with the following tabs:

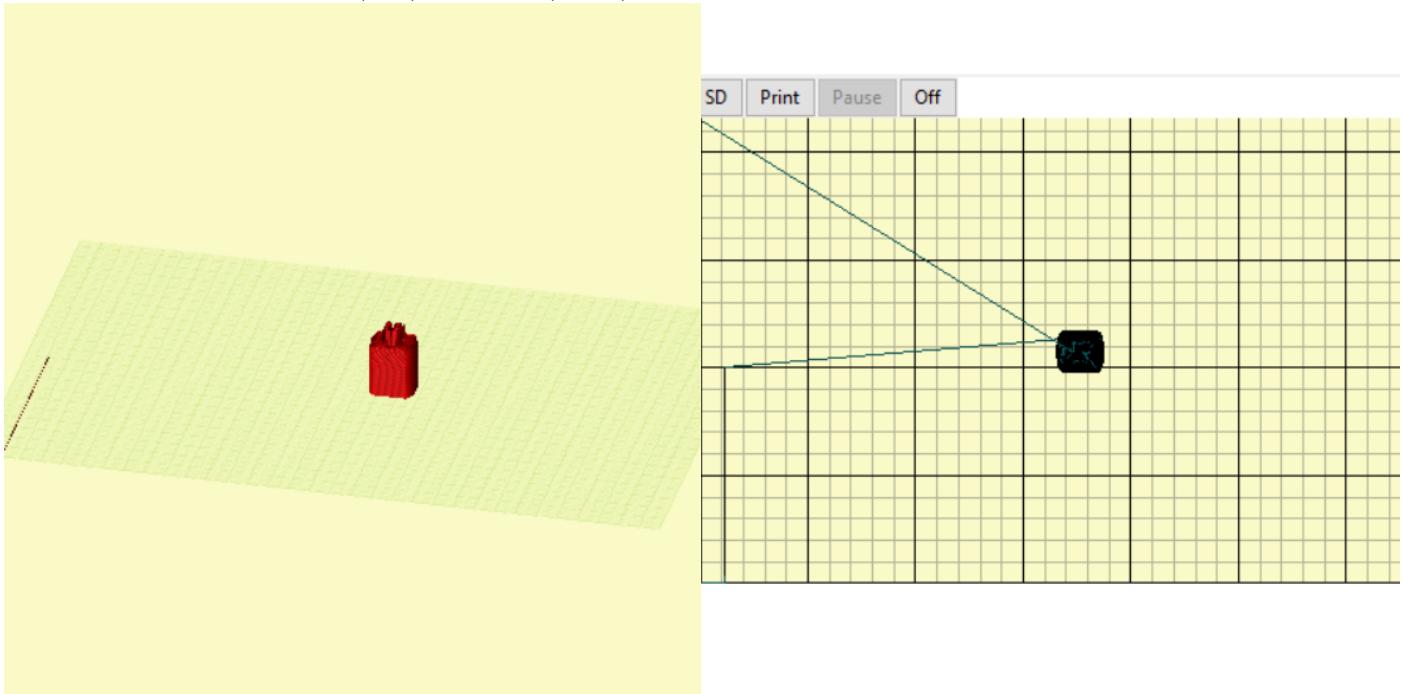
- **Printer settings:** Change information about the printer such as default movement speeds, default temperatures, build dimensions, etc
- **User interface:** Change the interface layout and temperature display
- **Viewer:** Switch between 2D and 3D viewer and customize graphics performance settings
- **Colors:** Change colors displayed in the GCode viewer
- **External commands:** Set functionality for when an STL is opened in this program.

3.2 Viewer settings

Switching between 2D and 3D viewer

We can switch between 2D and 3D view by changing the **Main Visualization** option. After Selecting the viewer that you want, you must restart the interface.

Below we can see the 3D (left) and 2D (right) viewer:

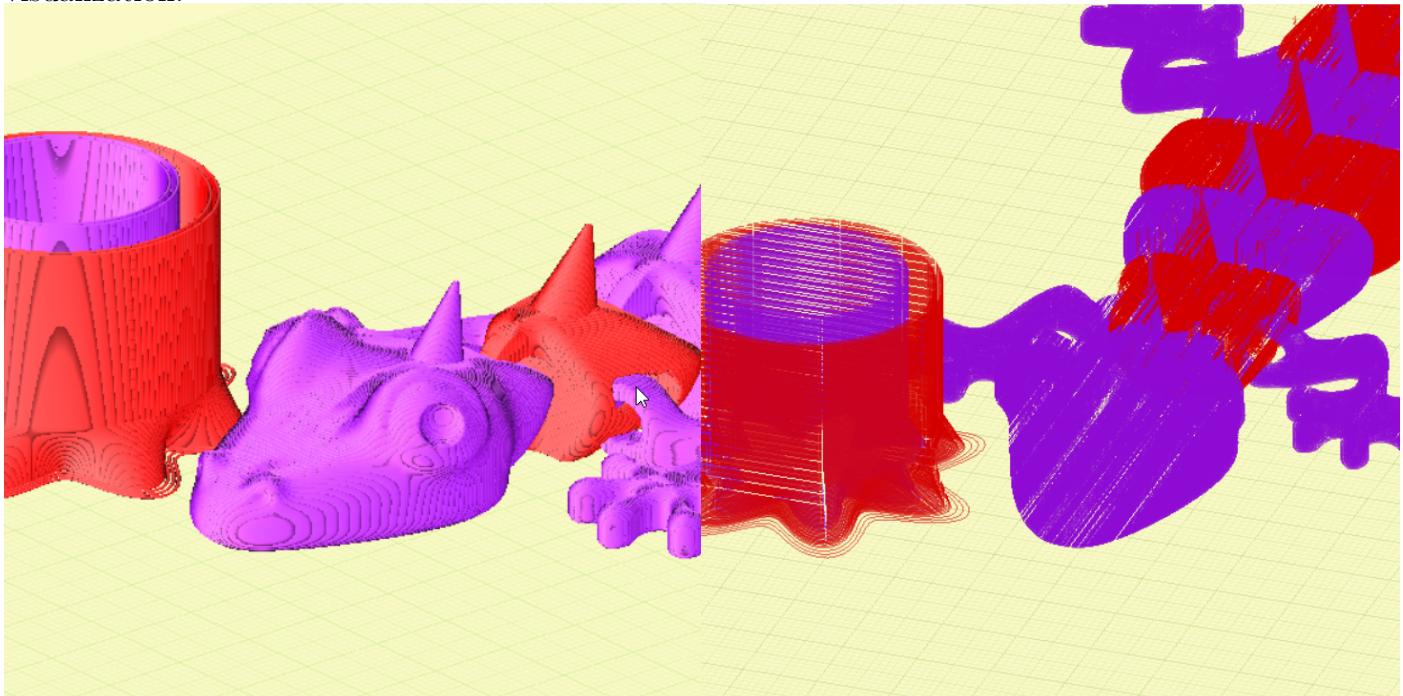


By default the external GCode Viewer (accessible by double-clicking the GCode Viewer) uses the 2D view. You can change this by checking the option **Use 3D in GCode viewer window**.

Using a light 3D view

By default, the 3D viewer draws thick lines to create a more realistic model.

For large models, this may slow down the viewer. To make the 3D view run a bit faster, check **Use a lighter 3D visualization**. Below you can see the difference between normal (left) and light (right) visualization:

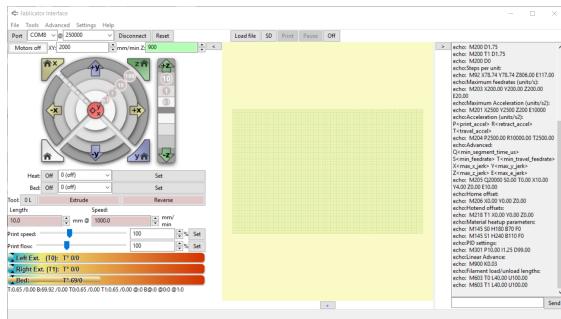


3.3 Interface layout

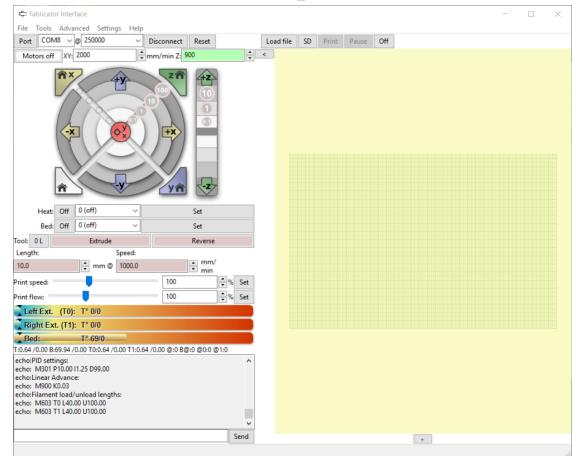
Interface mode

This changes the location of the terminal which can make the interface more compact when combine with the miniature controls.

Standard

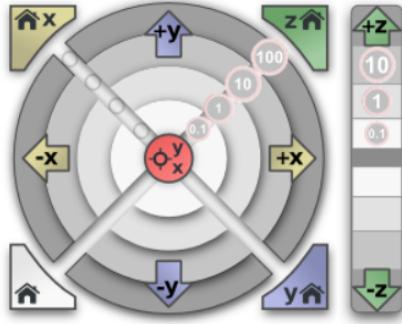


Compact

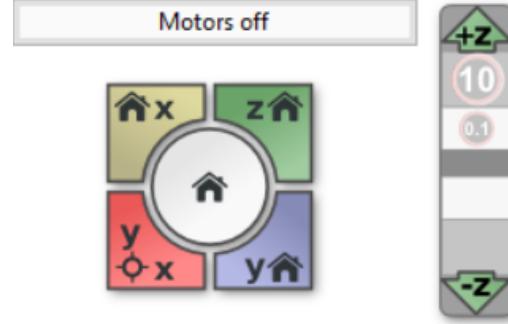


Controls mode

Standard



Mini



Temperature graph

Checking the "Display temperature graph" box will put a graph of the temperatures over time near the temperature gauges. This could be useful for doing manual PID tuning.

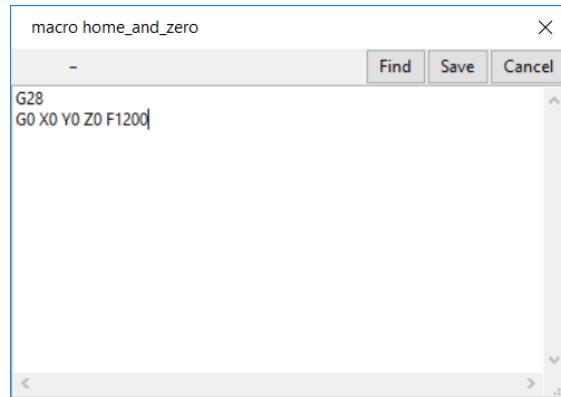
3.4 Defining macros and creating buttons

Creating a macro

Macros can be defined by going to **Settings > Macros** and clicking **<New...>**.



This will prompt you to enter a name for the macro and bring up a window where you can enter GCode the macro will run. For example, a macro called *home_and_zero* can be created to home the printer and move the head to the origin (this is shown below):



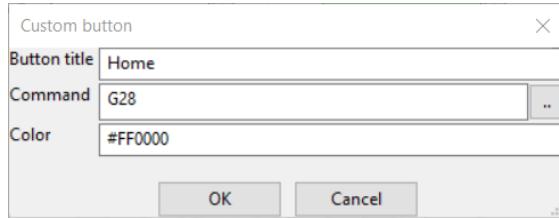
The list of GCode commands can be found at: <http://marlinfw.org/meta/gcode/>

We can run the macro by typing it into the terminal or binding it to a button.

Macros can be deleted by going back to macros and removing all of the text.

Creating a button

Buttons can be created by clicking the  button beneath the GCode viewer. This will bring up the following menu:



Button title specifies the label that the button will have.

Command is where you either type a single GCode command or the macro that you created.

Color will let you specify a button color. This is given as an HTML style color code.

There are a number of color codes at: https://www.w3schools.com/colors/colors_names.asp

Right-clicking on a button will allow you re-define the button, shift it left or right, and delete it.