Extruder flow calibration

This is a simple guide that takes you through why and how you should calibrate your extruder flow.

What is flow calibration?

The extruder is driven by a stepper motor, that must rotate certain number of steps for every millimetre of filament pushed through the hot end. This is defined in the printer controller as steps/mm.

Why do this?

If the extruder is feeding too much filament (over extruding), you can get blobs (extruder bogies) being dropped into the print.

If the extruder is feeding too little filament (under extruding), you can get gaps or strings in the print as the extruded filament is stretched too much.

What do I need to do this?

- A pencil (to mark the filament)
- A ruler or calliper (to measure the moved filament)
- A Computer and Terminal program (to send commands to the printer)

How do I do this?

Before the full step by step guide, I want to explain what you will be doing.

Your printer uses a language called GCODE to print, which is created by the slicer tool you use. There are lots of GCODE commands to set up and configure the printer too and you will be sending these commands one at a time using a simple terminal program. (Termite is a good free example)

To calibrate the flow, you will need to do the following:

1) Connect your printer to a PC and send a command using the terminal program
2) Read the current steps/mm setting from the printer using the terminal program
3) Make a mark on the filament near the extruder
4) Send commands to the printer to heat up the extruder, then feed 20mm of filament
5) Measure the amount of filament actually extruded
6) Calculate the difference between the 20mm you asked for and what was actually fed.
7) Calculate the new steps/mm number using the answer above
8) Set the new steps/mm on your printer using the terminal program
9) Save the settings so your printer remembers the new calibration when you turn it off.
10) Congratulations! Your feed rate is now calibrated!!

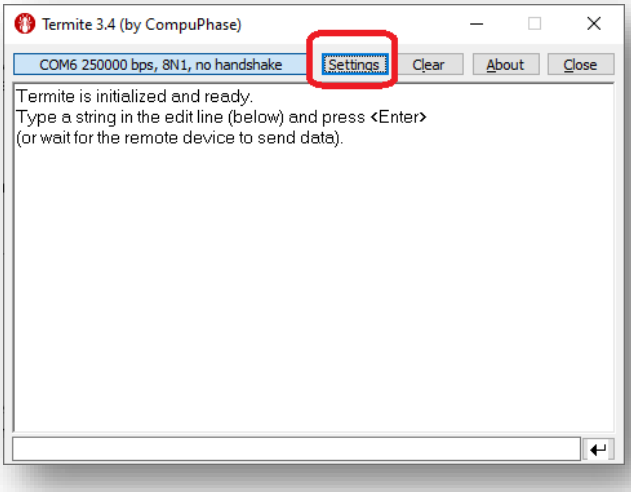
Calibrating the Anycubic Chiron Extruder

NOTE:
Make sure Cura or any other print slicing tool is closed, or you won't be able to send commands to your printer.

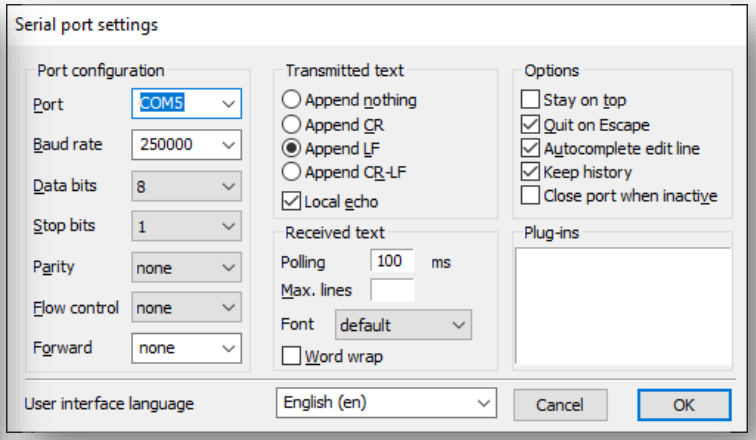
Step 1 Talk to the printer

Turn on the printer and connect the USB cable to your PC.

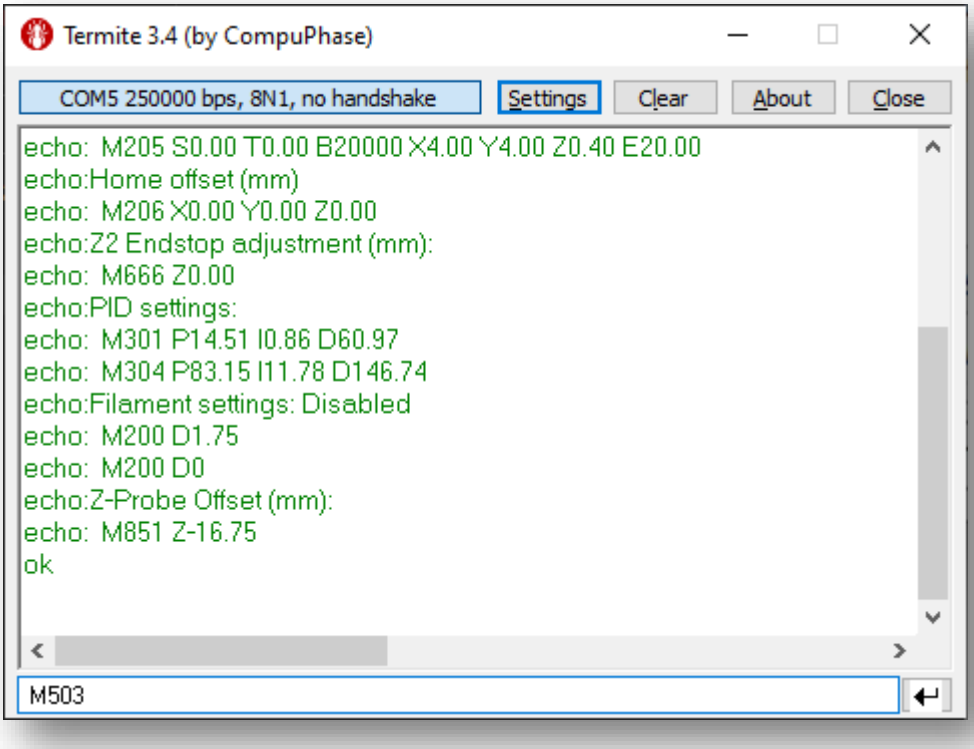
Open the Termite program and click the settings Button.



Select the Port for your printer (probably 5 or 6) set the Baud rate to 250000 then click OK.



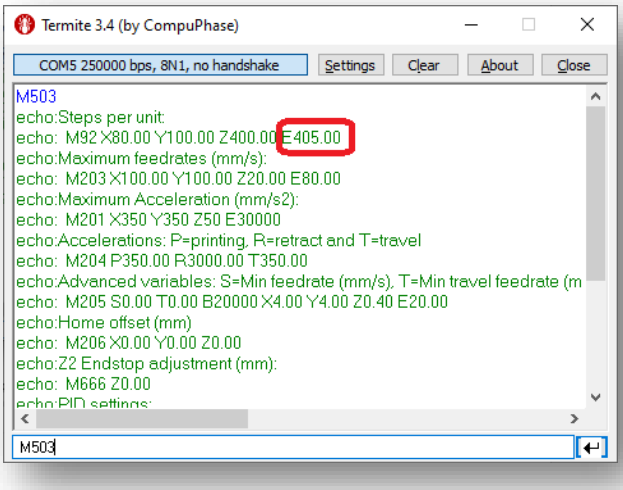
Now let's talk to the printer!!
In the box at the bottom Type `M503` then press enter.



You have just asked the printer to report all settings, which hopefully it has!
If nothing happens, set a different COM port number and try again.

Step 2 Read the current Steps/mm

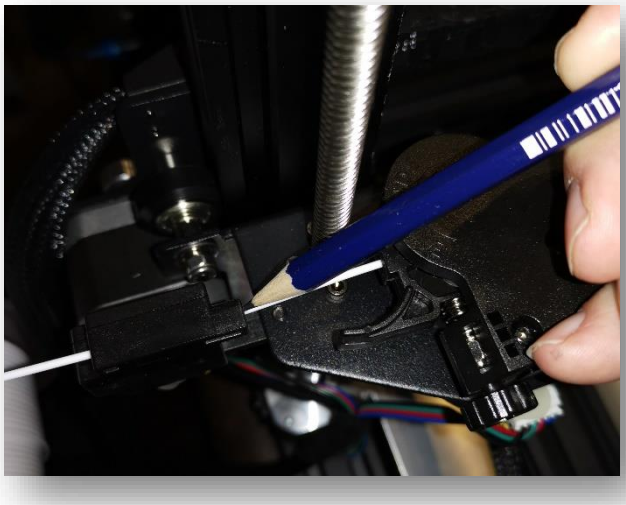
If you got the answer from your printer, scroll up and look for the extruder steps per unit setting E

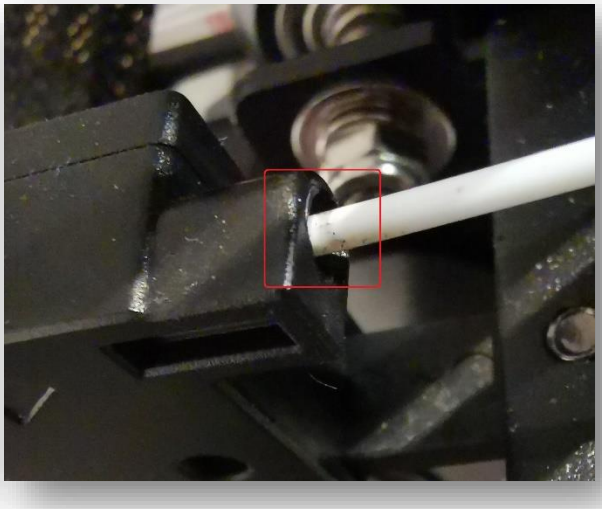


Write it down as the Start Steps = 405

Step 3 Mark the filament

Use a pencil to mark the filament as close to the runout sensor as you can.





Step 4 Extrude 20mm of filament

Either use the front panel or in the terminal set the hotend to 200 degrees by sending

`M104 S200` then press enter.

The printer should answer OK.

Now send these commands one at a time

`G91`	then press enter.	Sets absolute extrusion mode
`G92 E0`	then press enter.	Sets the current extruder position to 0
`G1 E20 F200`	then press enter.	Feed 20mm of filament at 200mm/minute
`M104 S0`	then press enter.	Turns off the extruder heater

Step 5 Measure the extruded filament

Use a ruler and measure the distance between the pencil mark and the run out sensor.
If the calibration is correct it should be exactly 20mm

My measurement is 18mm, so my printer is under extruding!

Step 6 Calculate the error

We wanted 20mm and got 18mm.

18mm / 20mm = 0.9
0.9 x 100 = 90%

So, my printer is under extruding by 10%

Step 7 Calculate the new steps/mm value

So our current steps per unit value from step 2 is 405.

To calculate the new value simply divide by the answer above.

So, 405 / 0.9 = 450

Which makes sense as I need the stepper motor to do 10% more steps than it is right now.

If your printer is over extruding then the new number will be smaller.

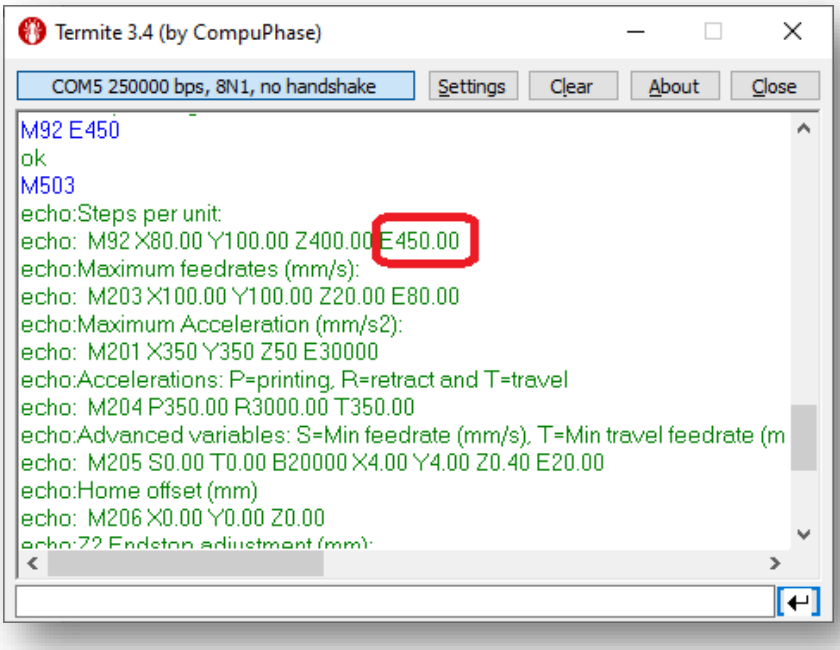
Step 8 Send the new value to your printer

Using the terminal send the following commands.

M92 E450 then press enter. This sets the new step value using 450 answer from step 7

M503 then press enter. This asks for all settings again.

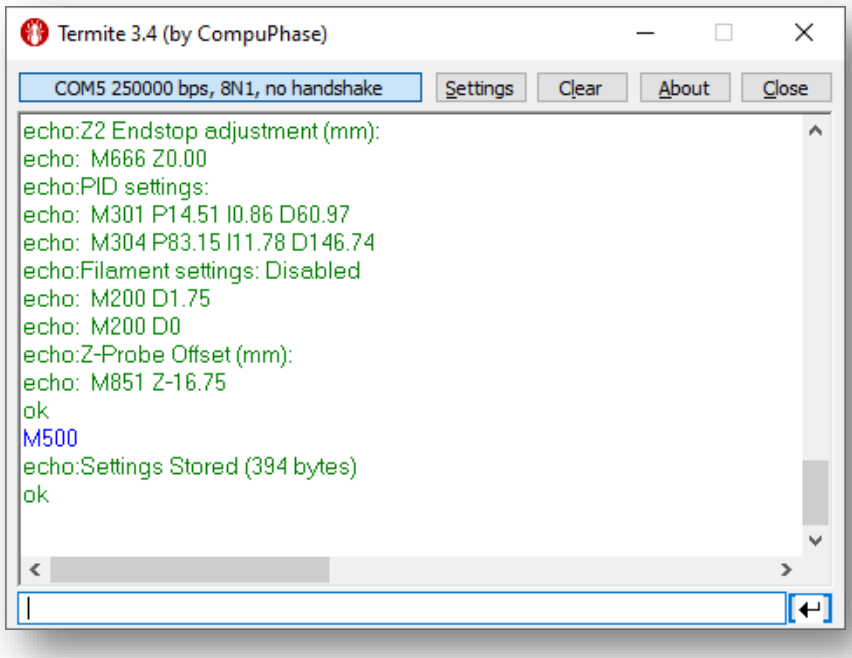
You should now see your new value for the extruder steps.



Step 9 Save the settings to your printer

Using the terminal send the following commands.

M500 then press enter. This saves the changed settings to non-volatile memory



Step 10 That's it your extruder is now calibrated!