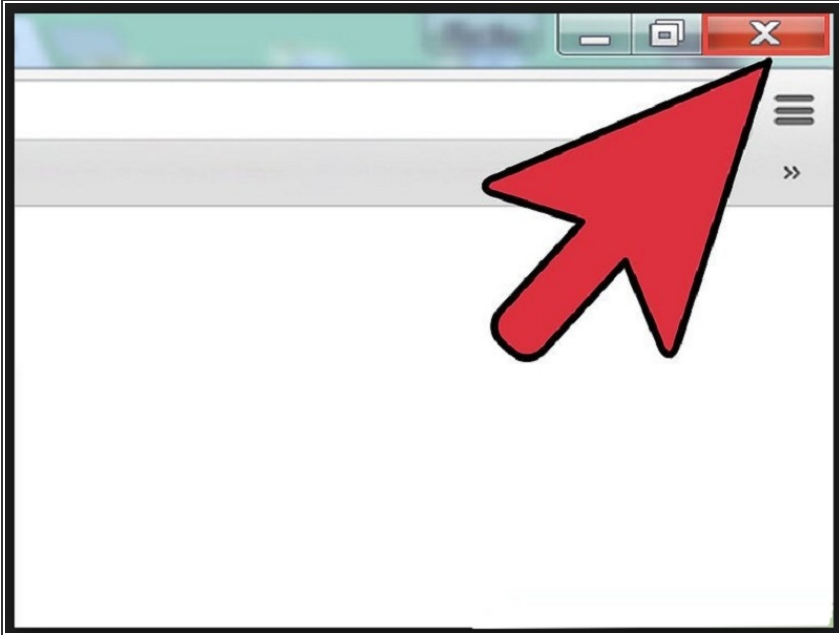# SeeMeCNC Guides

Configuring Artemis

Configuring your Artemis printer. Connecting it to your network and getting everything dialed in for successful printing.

Written By: SeeMeCNC

Step 1 — Close Other APPs

⚠️ Close other applications that may use the USB com port.

ⓘ This printer is NOT designed to run from the USB port. Only connect the USB cable during setup of the printer.

ⓘ The printer is designed to run from either (a) browser window over your intranet or (b) by addition of touchscreen with SD card reader

● Close Cura, Mattercontrol, or other slicers you may have open.

● Close any non-3D related programs, like Arduino or other programs that may use the USB com port.

Step 2 — STOP! READ THIS STEP

⚠️ If you purchased a fully assembled printer, an RTP (ready to print) model, DO NOT install or update new firmware. Your new printer is ready to use. Follow steps to configure the WiFi connection, then proceed to the last step of this guide to get started with the slicing software.

Step 3 — Download Duet Driver Files
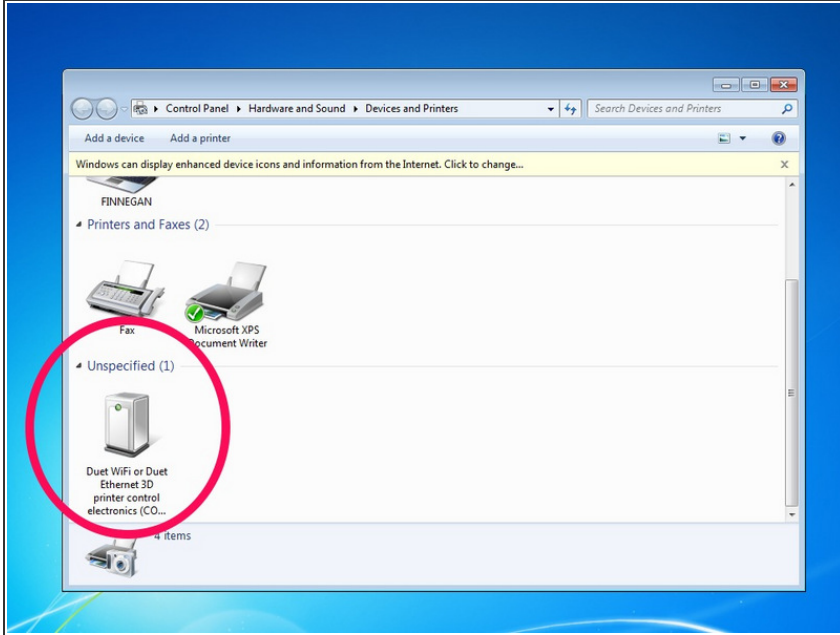
- If you are running windows, click here to download the Duet Wifi Drivers. If you are running Mac or Linux, you won't need them - skip this step. Unzip the file and save the drivers somewhere you'll be able to find them. On windows 10, just double click the "duet.inf" file and in will install the driver. On windows 7, you'll need to find the duet in Device Manager and click update driver.

- If you don't already have a program installed to send commands to a 3D printer, we recommend YAT (Download here: https://sourceforge.net/projects/y-a-ter...). Once it's downloaded you shouldn't need to change any settings, just connect to the printer.

- ⚠ If you are using Simplify 3D, Pronterface, or MatterControl to connect to the printer, make sure it's not converting all your text to uppercase when you're setting up your wifi in the next step.

Step 4 — WiFi Setup - MAC
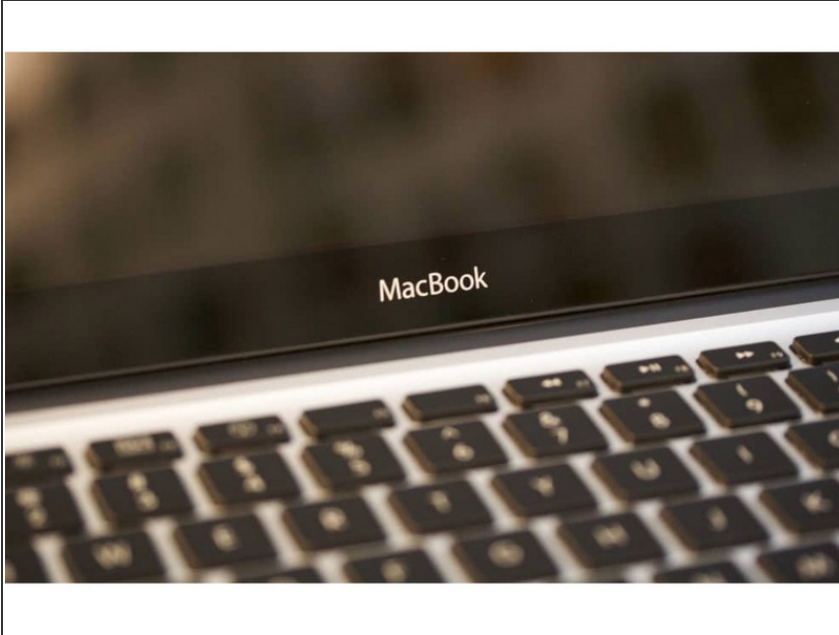
(i) If you are not using a MAC, proceed to the next step.

- Mac OSX Users: Open Applications > Utilities > Terminal .
  - Run ls /dev/tty.*
  - Copy the result that has USB in the name. For example; /dev/tty.usbmodem1411
  - Run screen /dev/tty.usbmodem1411 115200. Replace "usbmodem1411" with the result you received. You should see a blank screen. Hit enter and you should see OK returned. You are connected successfully.

⚠ If you get an error, Sorry, could not find a PTY. unplug the USB, plug it back in and try again.

Step 5 — Wifi Setup - Windows
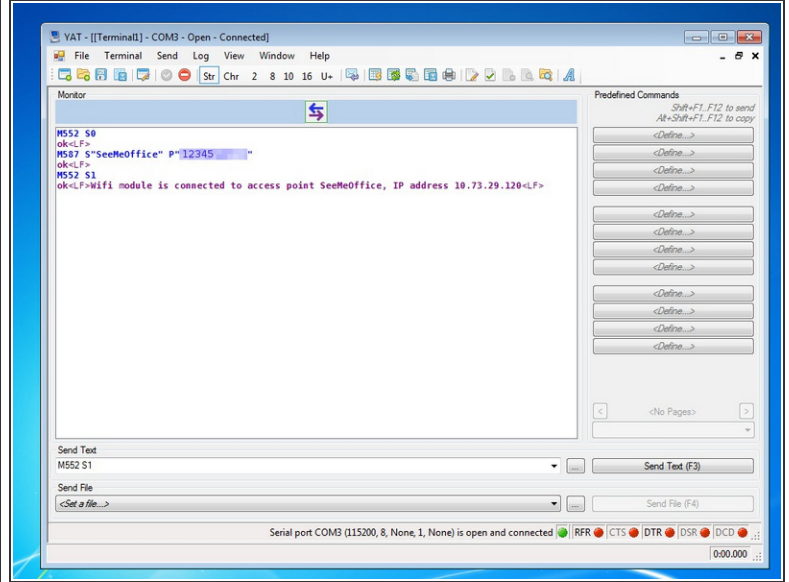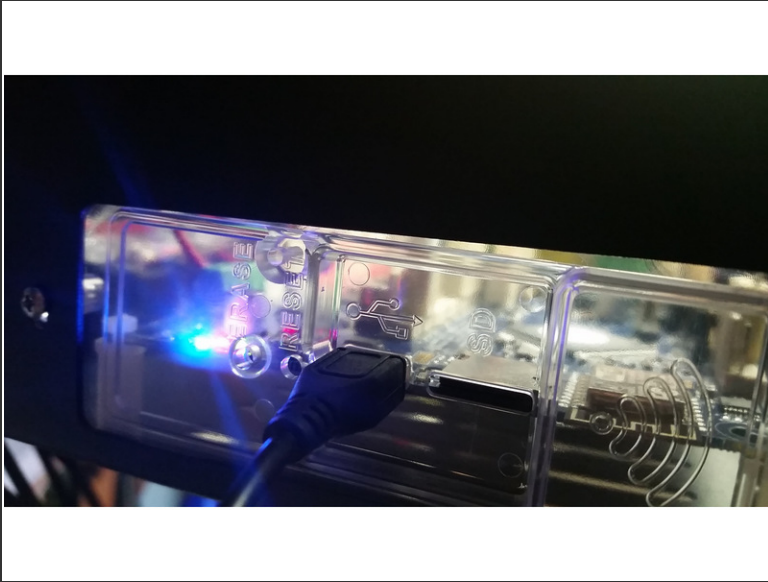
- Plug the micro USB cable into the USB slot on the (top) left side of your printer.

- Connect to the printer with YAT, S3D, MatterControl, or carrier pigeon. Note that you'll need to set the baud rate to 115200 in order to connect. Then send the following commands:
  - M552 S0
  - M587 S"yourssid" P"yourpass"
  - M552 S1

- After sending M552 S1 , you should see a confirmation, and the IP address of your printer. Write the IP address down, you'll use it in the next steps

- Make sure your printer is close to your Wifi Router. If you a getting a lot of AJAX disconnected errors, it's probably because the printer can't talk to your wifi reliably. Keep the distance between them to a minimum.

- (i) The SD card can become partially unseated in shipping. It is highly recommended to remove it (push it in until you hear the click, then it'll come out) and put it back in (push until you hear the click) before you get too far into the setup.

Step 6 — Initial Setup
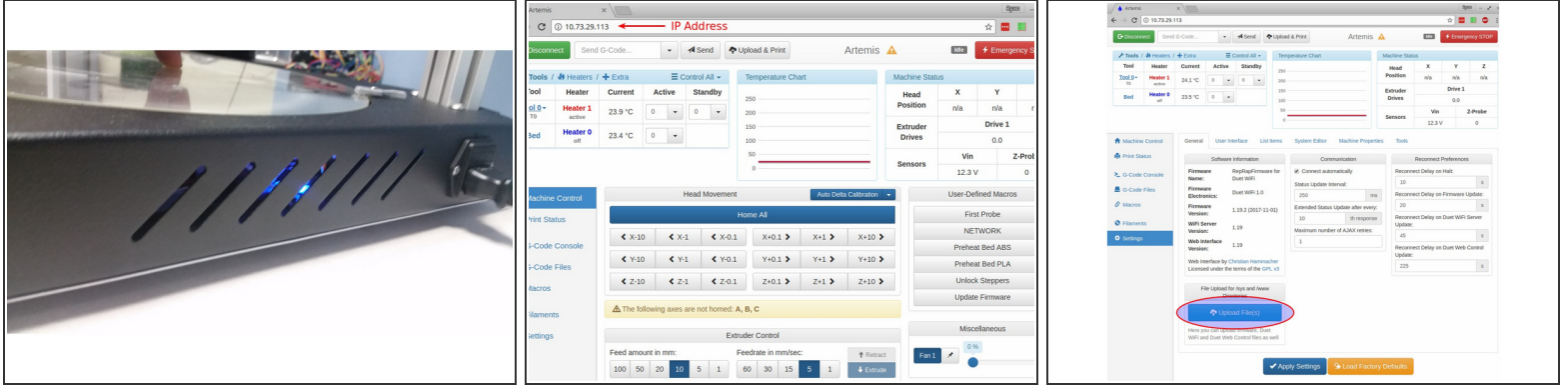
- Check that your FSR's are working properly. You should see a blue light through the slots in the side of the base with the power cord. Push down on the bed in a few different places and make sure the lights flash/blink - you shouldn't need to push very hard. If they don't blink, contact support.

- Open a web browser and download the newest config files here:
  https://github.com/seemecnc/Artemis

- Open a new tab and enter the IP Address of your printer. When the page loads, click on "Settings" in the lower left of the screen. Click on the "Upload Files" button, and upload the config.zip file that you just downloaded.

- Once it's uploaded, you will be prompted to reboot the Duet. Click yes. After about 10 seconds, click the Refresh button on your browser and your printer will be running with the new config files.

Step 7 — Calibrating your printer
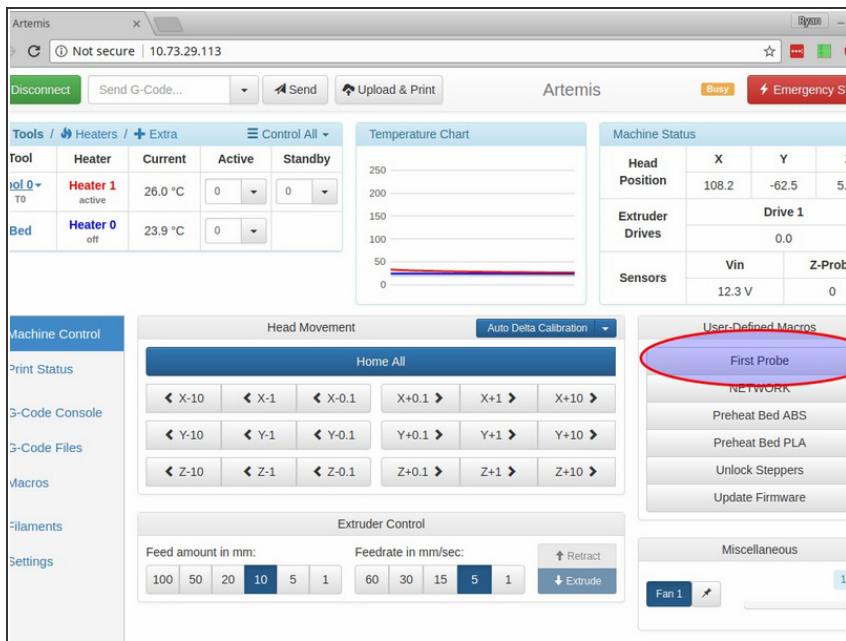
- We have setup a custom macro to take care of all the heavy lifting for your first calibration. On the "Machine Control" tab, you can click "First Probe" under "User-Defined Macros" to do your first calibration.

- The macro will do an initial calibration that starts extra high off the bed to account for slight variances in build height. Once it's completed, it will do a second calibration, double checking each height reading, and save the final results to your SD card. Don't worry, future calibrations go much faster than the first one.

- Test your calibration by putting a sheet of paper on the bed and running: "G0 Z0" in the gcode console. Your nozzle should just touch and pinch the paper so it's hard to move. If it's too loose, run the following GCode: "G31 P500 X0 Y0 Z-0.25", then run "G32" to recalibrate, then "M500" To save all your current calibrations and settings.

- If it's too tight, you want to try "G31 P500 X0 Y0 Z-0.35", then run "G32". If it's still too tight, make the number after Z a larger negative number. If it's still too loose, make it a smaller negative number.

- After your initial calibration, you can calibrate by clicking the "Auto Delta Calibration" button, or by sending "G32" in the gcode console. We highly recommend probing with the bed cold. FSR's get finicky with temperature increases and the thermal expansion of the bed is so minimal as to be non-existent.

- (i) You might see a line or two that says the Height Error Difference was above the maximum tolerance. Don't worry about them unless you see a lot of them.

Step 8 — Loading Filament

- You can load filament by heating the hot end up to 220 degrees and manually feeding the filament in through the EZR and boden tube, or by using the "Filament Load" macro

- If your printer didn't come with the macro, you can find the gcode with our Artemis setup files on github, and setup the macro on your printer.

- To use the load filament macro, hold in the red button on the EZR extruder and press the filament in until it is just peeking out the side with the boden tube. Then click "Filament Load" under User-Defined Macros, and click OK.

- (i) Click here for the guide on creating macros.

- (i) Click here to go to the list of SeeMeCNC Macros.

Step 9 — Printing!

- Upload and print your first gcode file. You can do this two different ways. You can either click "Upload & Print" at the top of the page, or click on G-Code files at the side of the page, then "Upload G-Code File(s)", then select your file from the list.

- If you'd like a pre-sliced file for your first print, FIRST PRINT

- NOTE: On the first heatup of your bed, you may notice a small amount of smoke from the bed / base of the printer. This is nothing to worry about, this is excess residue on the silicone bed from the manufacturing process. We cycle every bed at SeeMeCNC before shipping to reduce / eliminate this issue, but some are more determined than others.

Step 10 — Start Printing Successfully
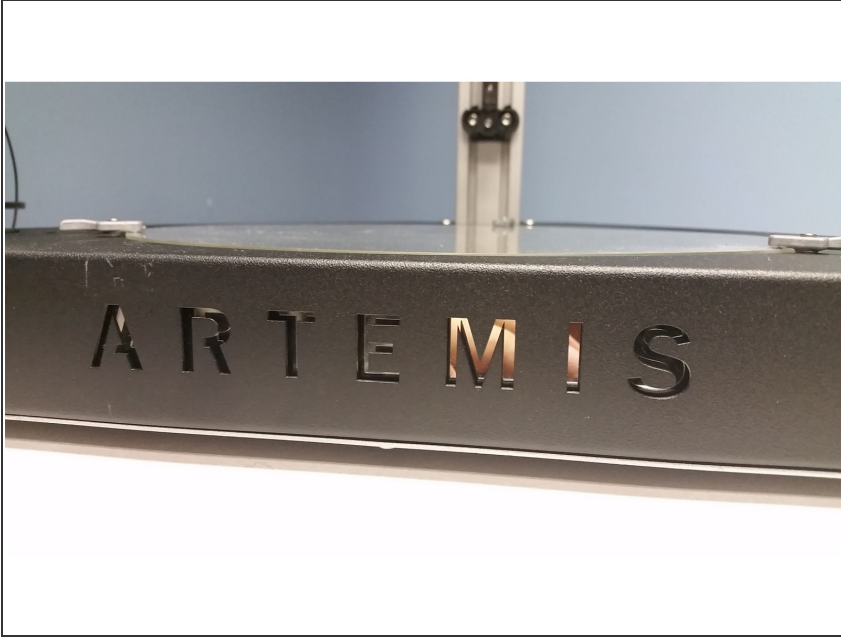
- We recommend using Cura for slicing. All v3.2 testing is performed with Cura and our slicing profiles for our printers will get you going fast! [Click here to see our guide for installing Cura](#) and downloading our profiles for your new printer.

This document was last generated on 2018-06-05 09:43:48 AM.