

## Full Lightburn Control Guide

### (by Skreelink)

This guide will allow you to do your entire project in Lightburn, and (if connected via USB) run it directly from Lightburn letting you skip Luban or using the USB drive entirely.

The assumption with this guide is that you have some experience with the Machine. The set-up will work with both the 1.6W and the 10W lasers.

#### **Step 0: Accurate and Repeatable.**

This guide is an extension of my laser origin guide ([Accurate, Repeatable Laser Guide](#)) and requires the guide to be followed in order for you to find your true origin and laser height.

If you intend to use the laser as Snapmaker intended (Touchscreen/Luban) and eyeballing the positioning of your material skip this step and Step 4, however Step 7 is required.

#### **Step 1: Inline Control**

This requires the installation of the “inline control” version of the firmware that can be found on (public) GitHub. This is NOT included in the official Snapmaker (private) releases so you MUST load this ‘custom’ firmware. **Warning: *I’m not liable for any damage to your machine, material, fire, or loss of sanity.***

[Skreelink's Snapmaker2 Controller V4.5.34](#) (latest as of writing this guide)

This is modified and compiled by me and is what I use on both of my machines, an F350 with 10W laser, and A350 with 1.6W laser.

1. Make sure to update your machine and modules to the latest official firmware
2. Download and unzip my firmware to your USB drive.
3. With your Snapmaker switched **OFF**, insert the USB drive
4. Power on the machine
5. From the console navigate to <files>, <USB> select the file (.bin). Should work like this [<Youtube link>](#). This will flash and enables the inline features to the Marlin firmware, allowing you to use dithering without the stop/go stutter that standard software does. In addition it provides true grayscale, which varies the power of the laser to create gradients. This requires Lightburn, however, and is not currently a feature of Luban.
6. Check the “About Machine” on the Console that it has taken the patch

If you switch tool heads back to Printing or CNC you **DO NOT** have to reinstall the official firmware to use them.

#### **Step 2: Install LightBurn and set-up your Snapmaker**

To start, you require... **LIGHTBURN** imagine that, a guide that specifically states a software, you need it.

There is a trial version which can be used free for 30 days before a purchase is required. You can download from here: [Lightburn Software](#)

And a very active support forum found at: [Lightburn forum](#)

#### **Step 3: Set-up your Snapmaker**

Now here's the special part, you need to setup your machine type as **GRBL-M3** instead of Snapmaker or Marlin. If you already have your Snapmaker setup in Lightburn, you can click devices (on **Laser** Tab) > select your machine > edit. If not, add the machine new. It will scan and should identify it as GRBL-M3 (1.1e or earlier), if not select from the list.

Afterwards, it's mostly normal, except uncheck the “Auto home” during device setup.

Selecting GRBL-M3 solves a few problems with using Marlin as the machine type (mostly how it turns the laser on/off, and had required some minor GCode edits before being run).

## Step 4: Change Basic Settings

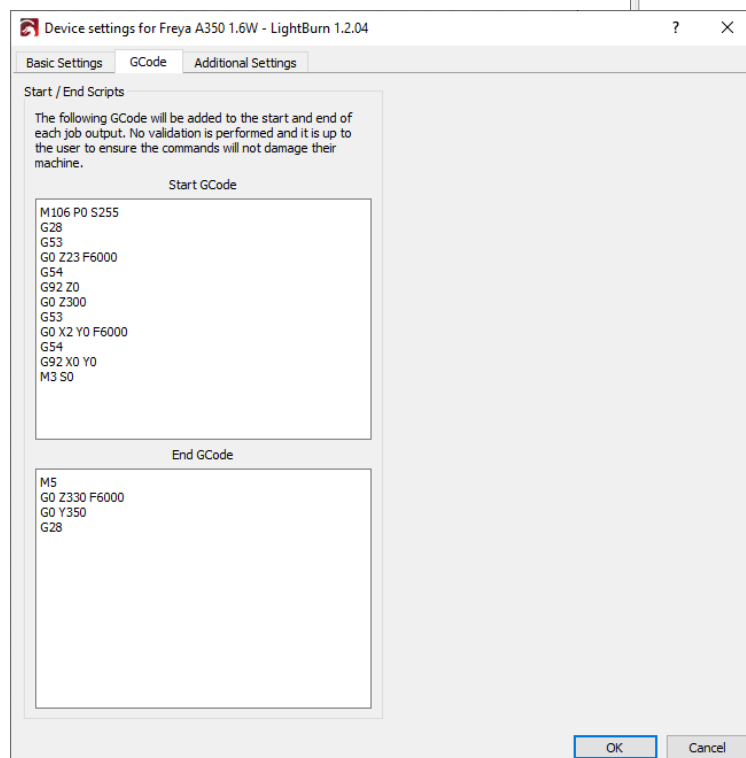
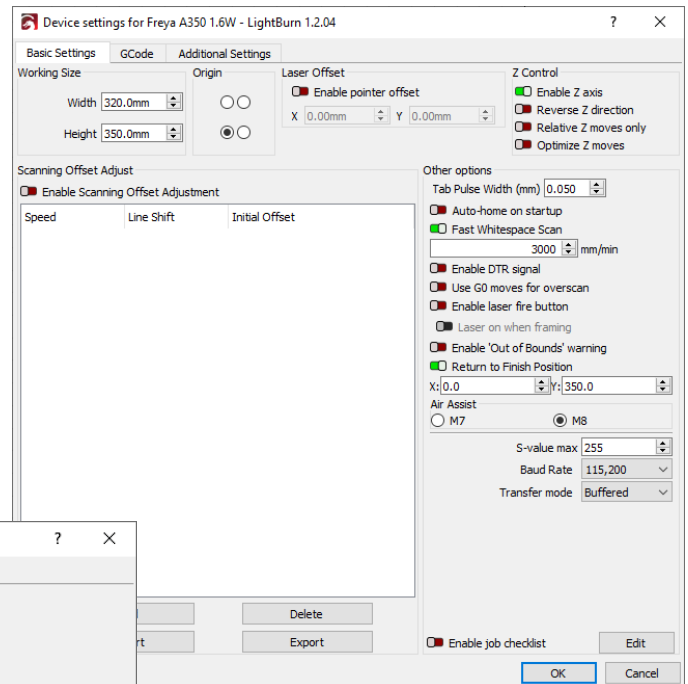
Now after setting your machine to the type that it's certainly not, open the device settings, as there's a few specific things we need to do. In the <edit>, <Device Setting> **Basic Settings** tab:

**Tick "Enable Z axis" and Untick "Relative Z moves only".** The other settings you can change as you need, but ensure "*S-value max*" is set to 255.

## Step 5: GCode for Accurate and Repeatable Lasering

This is one of the more important parts and only required if you have done Step 0: Accurate and Repeatable, if not go to Step 5.

You **WILL** already have found your true origin and have noted your laser height.



In the **GCode** tab, we're setting up all the parameters that Lightburn doesn't know and can't pull from the machine like Luban does. Such as laser focus, origin offsets, and a few safety lines. Keep in mind these are the settings for **MY MACHINE** while it may match yours fine, I encourage you to do your own tests for proper settings. I'll paste my settings with notes on things you need to change.

## Explanation of GCode (Remove comments if copying and pasting):

### Start GCode:

M106 P0 S255 ; This line is spit out by Luban and Snapmaker settings in Lightburn, so I include it.

G28 ; Safety home, in case you forgot after power on.

G90 ; set to Absolute Positioning – See FAQs. Z axis height wrong after restarting a failed job

G53 ; Changes to Machine co-ordinates.

G0 **Z23** F6000 ; **IMPORTANT**, the Z value here is your **LASER HEIGHT** on the touchscreen.

G54 ; Changes back to Work Co-ordinates

G92 Z0 ; This sets Z Work Origin to 0, making your laser height as 0.

G0 Z300 ; Moves the tool head back up out of the way.

G53 ; Back to machine co-ordinates!

G0 **X2** Y0 F6000 ; **IMPORTANT**, this is the offset you found following my laser guide!

G54 ; Oops, we're back in work co-ordinates.

G92 X0 Y0 ; Yep, we gotta set 0,0 on X,Y as well!

M3 S0 ; This is a safety line, it brings the laser online at 0 power, basically a ready state.

## End GCode:

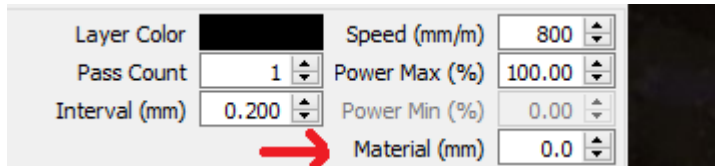
M5 ; Safety line, ensures the laser is offline and actually off.

G0 Z330 F6000 ; Custom line to rapid toss the tool head up out of the way.

G0 Y350 ; Custom line to rapid the bed forward.

## Step 6: Determining Material Thickness

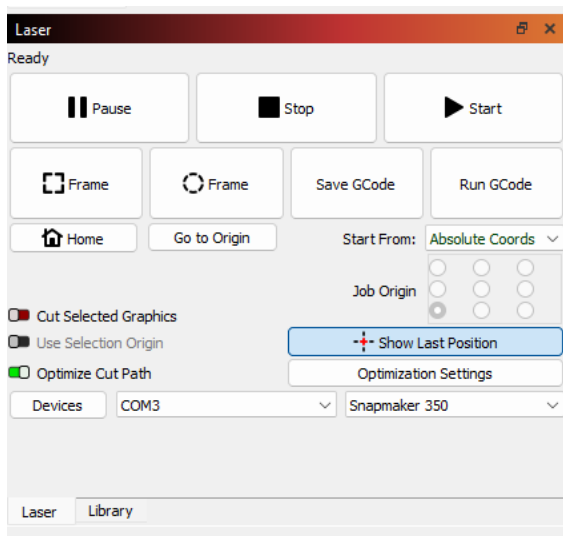
Now with these settings and custom GCode in place, you're ready to setup a project! The last, very important note, is you now have a **Material** option in Lightburn. It only shows up after you place your first image, svg, shape, whatever.



This is where you put your material thickness. This is where the difference between the 1.6W and 10W comes in! As long as you've homed your machine, you can point any browser at the web API and initiate the thickness measurement on the 10W laser. This will give you the value to slap into the material box, instead of having to measure with callipers.

[http://XXX.XXX.XXX.XXX:8080/api/request\\_Laser\\_Material\\_Thickness?x=150&y=75&feedRate=6000](http://XXX.XXX.XXX.XXX:8080/api/request_Laser_Material_Thickness?x=150&y=75&feedRate=6000) 98

Replace the XXX with the IP of your Snapmaker and adjust the x and y values to move the tool head around the bed. It's a little offset, so you might have to play with it to get it where you want. This is what I use for a general measure on objects around 150mm.



As a final check, from the **Laser window** ensure that "Start From" is set to **Absolute Coords**. The reason for this, is **User Origin** adds a G91 before the cut sequence, this is Relative Movements, which means instead of using the work origin we set in the header, it's trying to move 4mm in Z from the *current* position, which is at the top of the gantry.

Lastly, here's a video of how the process looks after setting up your project. A single click and go!

YouTube Video: [Snapmaker A350: Full Lightburn Control](#)

## Step 7: GCode for no-repeatable origin

If you intend to run the code "as intended" (Snapmaker's way) then you only need the safety M3 S0 in the header, and the M5 in the footer.

All the rest is relevant **ONLY** to set your repeatable origin and material height or basically, the full automation parts. Just make your project in Lightburn, click save GCode, then load it into Luban.

## FAQs and Problem solving

### Why select GRBL-M3 as our device rather than Snapmaker or Marlin?

The reason for the *GRBL-M3* profile is to take advantage of the firmware modifications. Since all the movement code is understood properly by Marlin for all the inline functions. The “\$” codes are all GRBL lines to ask information from the machine such as size, buffer, etc. Since the Snapmaker doesn’t understand these, they’re simply ignored with an unknown command response which you can see being returned in the **Console window**. For Example “M9” which switches on the air assist will return to the console *echo:Unknown command: "M9"*

### Can’t get Lightburn to Connect to my Snapmaker



Ensure you have the Snapmaker’s Driver installed (Windows only):  
<[CH34X Driver v3.4](#)>

Try various USB cable if the one Snapmaker supplied doesn’t work connected to the mini-USB connection on the controller.

Make sure your serial port is configured, within Device Manager, for 115200, 8, N, 1 and also in Lightburn settings.

Try this: Lightburn help: [Troubleshooting Connection](#)

### The Lightburn Home button doesn’t work!

This is because Lightburn is trying to use a GRBL “\$h” command to home and because Snapmaker runs on Marlin it doesn’t recognise it and returns an *Unknown command: “M2”*. If you want to home the machine via Lightburn, either type “G28” in the console, or make a macro button and name it home that contains “G28”.

This is also true for the Left, Right, Up, Down buttons within the **Move window**.

### Can’t adjust the laser height/Z axis

Check in the <edit>, <Device Setting> **Basic Settings** tab that you have “*enable Z axis*”. See Step 4.

### Z axis height wrong after restarting a failed job

**Issue:** When re-running a job (or a new one) after the previous job was stopped mid-run the toolhead goes all the way to the top of the Z axis and stays up there and will try to engrave as if that is the new Z axis height, even if the GCode specifies the correct height. Closing Lightburn and restarting resolves the issue, however it reappears if I again stop the job before then end.

**Reason:** Within Lightburn all main moves uses relative motion. It only uses absolute at the start, and swaps back at the end. However, if you stop the program in the middle the machine is *still in relative mode*. So even if you start over, it’s not until the first *G90* that it swaps back into absolute mode. A remedy would be to add another safety line in the header. Place a *G90* after the *G28*. Thus after it homes, it immediately swaps back to absolute movements.

### Control buttons run twice

Any control button seems to run twice. Be it framing, jog, etc. Even if you hand-type a command in the console it runs it twice. Why? I don’t know probably because Lightburn is expecting GRBL, but the Snapmaker runs Marlin. So likely it’s some kind of buffer or waiting for specific machine response so it repeats the command. I’m not sure.

### Transferring files via Wi-Fi without using Luban

Guide can be found here: [Lightburn: Guide to automatically start via Drag and Drop](#).

### I have a Snapmaker A250 what do I do different?

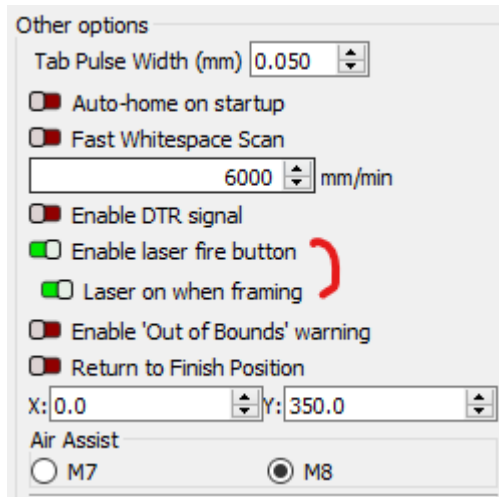
The firmware is universal between all the 2.0 machines, however the GCode start code need to have the Z and Y movements reduced by 100. i.e. Return height for Z is 230 and Y is 250

### The Laser Material API call returns a negative number

This is usually caused by an error in the triangulation that somehow slips by. Usually it'll just say error, but sometimes it freaks out and I guess catches a reflection, usually when scanning a glossy surface. I saw this when I was coming up with an auto bed scanner and it just so happened to land on the screws of the laser platform and I got like, -80mm.

### How to turn on the laser on low power to do a boundary check?

Ensure these two options are enabled in the <edit>, <Device Setting> **Basic Settings** tab. Then set the power under the **Move** tab.



### **Useful Links**

[Is there a way to skip the auto-focus and keep the same origin for when you are laser engraving the same material](#) Found on the forum at Snapmaker.com

[Lightburn Rotary Guide](#)

[Snapmaker 2.0 A350T and Lightburn](#) On Snapmaker forum

[Finding a material's best engraving settings](#). Found on Darkly Labs