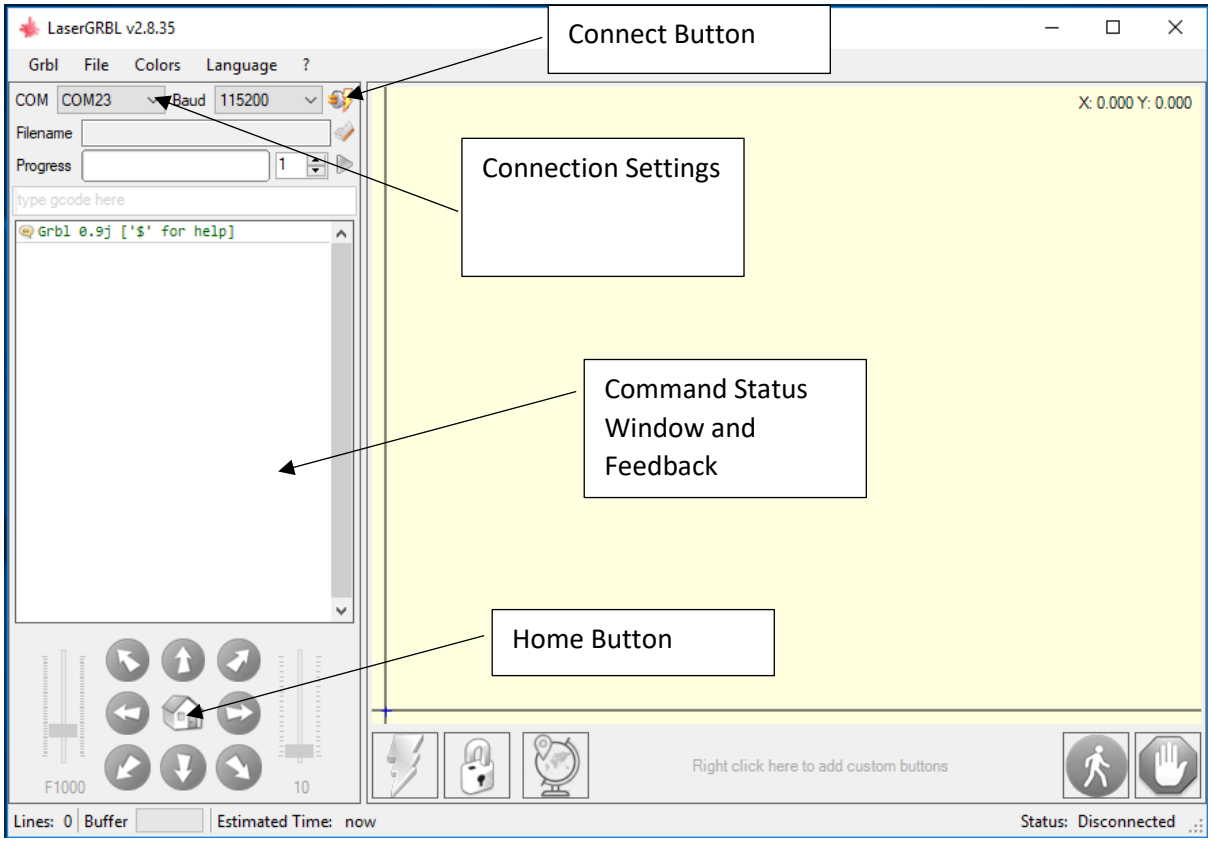
LaserGRBL

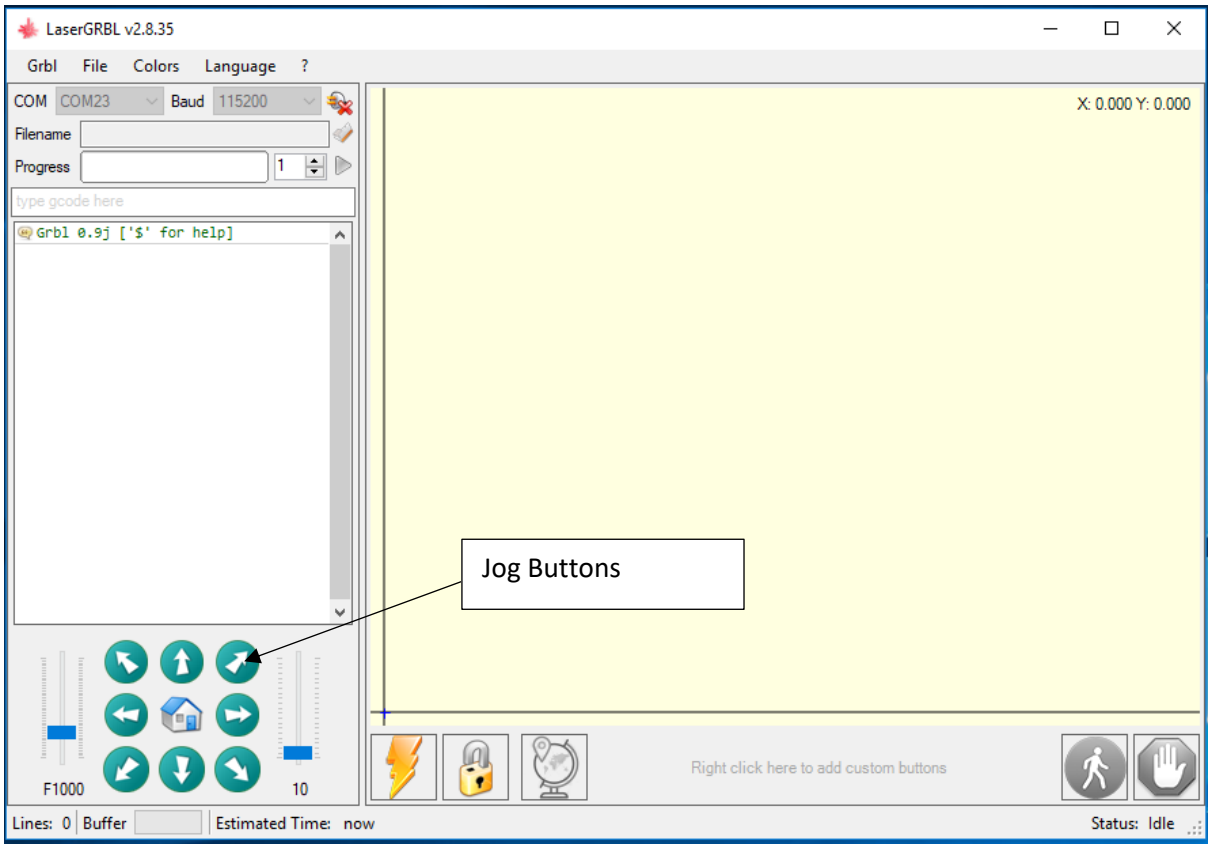
LaserGRBL is a popular program for Windows that streams GCode to DIY laser engravers. GCode is the language that many CNC machines interpret to produce items. GCode incorporates positioning movements, speeds and control of various tools. Whilst there are a number of different tools you could use to stream G-Code to your CNC machine, we will use LaserGRBL as an example, because it also incorporates the ability to turn pictures into vectored images that the CNC machine can attempt to draw.

Once you have installed LaserGRBL, before starting the program ensure that your machine has been 'zeroed' to its home position. Our CNC machine does not have limit switches to help it do this automatically, so to do this, turn off the CNC from all of its power and manually move each axis to its home location (you must ensure the machine is fully disconnected from all power sources when you do this! Otherwise, you risk damaging the stepper motors). When you plug the machine back in, the controller assumes whatever the current physical position is "0". As long as you do not power cycle your CNC machine, you can now use the 'home' button for the machine to return to this position.

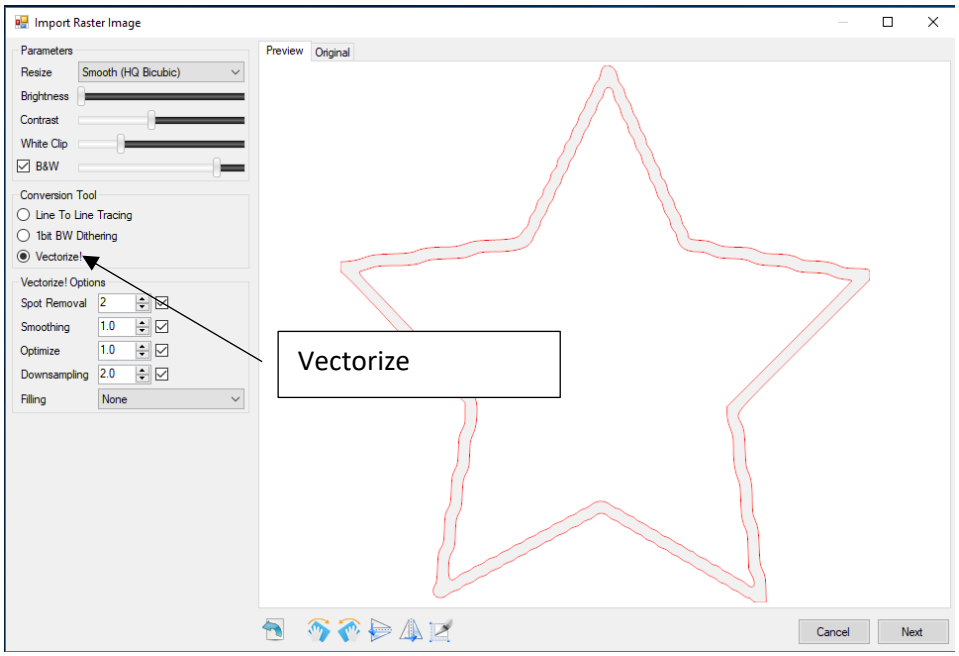


When you start LaserGRBL, you will be presented with the above interface. We first need to connect to your CNC machine. To do this, select the correct COM port for your CNC machine from the COM menu, then click the "Connect Button" (the icon with the plug with the power symbol). If the machine successfully connected, the other buttons on the screen should become enabled (non-greyed) and a message appear in the Command Status Window "GRBL xxxx ['$ ' for help]". You can

use the buttons marked with arrows to manually control and move your CNC around - this is known as ‘jogging’.

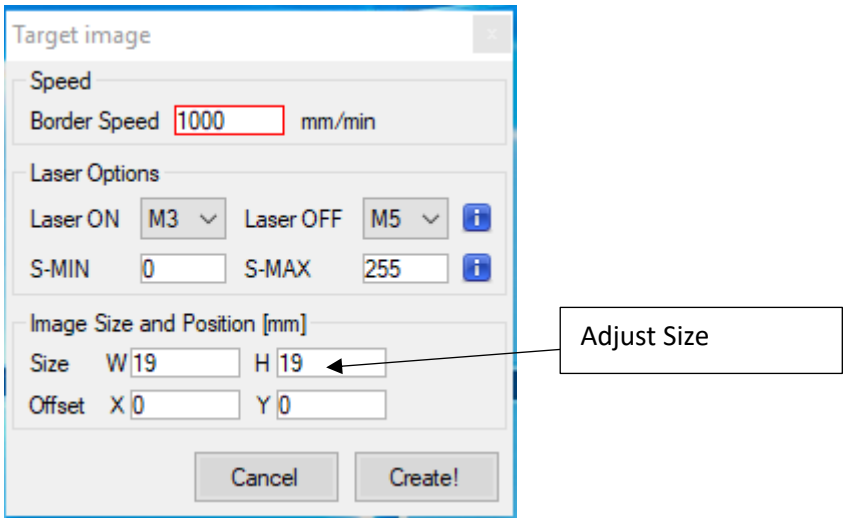


We are now going to import an image for out CNC machine to draw. Click ‘File’ from the top menu bar and ‘Open’. This interface supports a number of different file types; such as, pre-computed G-Code and images. We will select a simple raster image (png) – it is best to use very simple images with this machine.



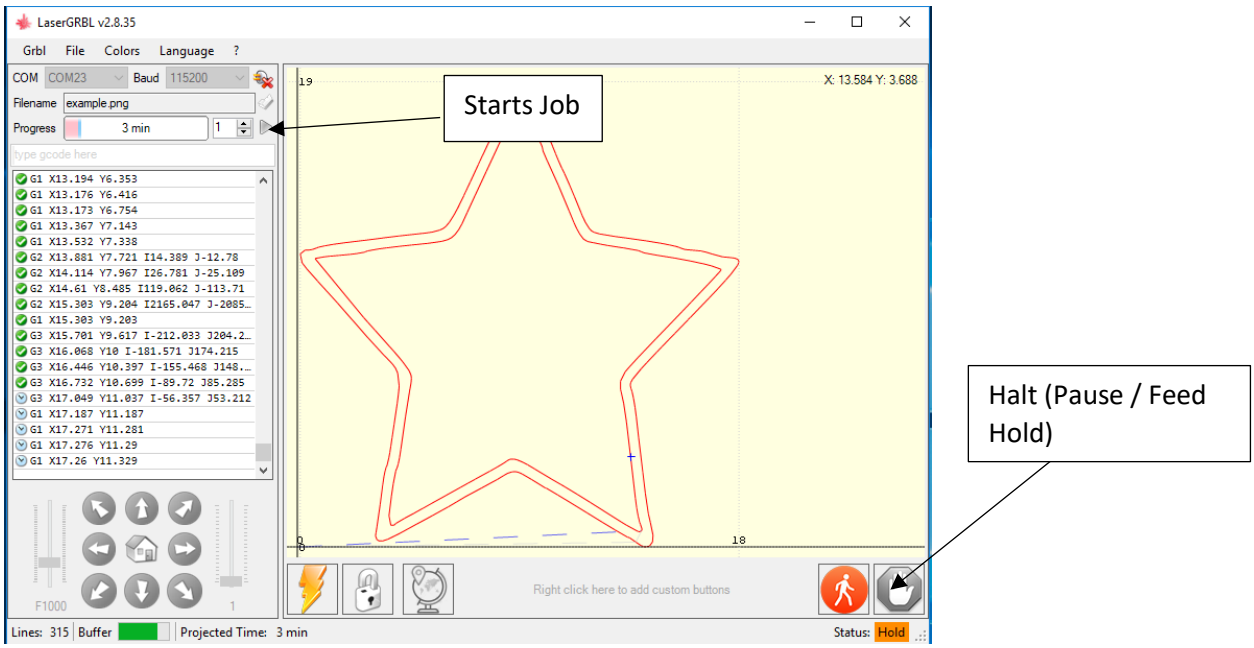
A new window will open labelled ‘Import Raster Image’. This attempts to trace our image into something that our CNC machine can understand. The conversion tool selected should be “vectorize!”. The other settings will require some manual tweaking to best suit your image. I regularly use “B&W”, because my images use simple black lines. I also choose “Smooth (HQ Bicubic)” for “resize” because this helps create a smooth continuous image for our machine to draw. Avoid choosing anything for “filling”, remember our CNC machine is quite basic and cannot lift the pen of the paper, so filling would be quite hard and will employ a lot of strain on the pen. When you are content with your tracing, click “next”.

Another window will open with options.



Adjust the image size and position to something our CNC machine can handle, e.g. W19 H19. Our CNC uses metric, so all measurements are in “millimetres”. Finally click “Create!”.

You should find the program returns now to the main application window, but this time with our image shown in the work window. This window provides a preview of what the machine will attempt to produce and, when running, how far through the job it is.



Before you start the job, ensure the machine is powered, both axis are in the right starting location and that nothing is capable of being trapped in the machine (including you!).

To start the job, click the icon that looks like a play program, or go “File -> Send to machine” from the top menu bar. You should see text begin to scroll in the Command status Window, this is the different GCode commands that are currently being sent to the machine. Should you need to halt the machine, you can press the icon with the hand.

When the machine has finished, ensure it is fully powered off before you attempt to take the paper away from the machine / do anything with it.

Hopefully you have found this guide useful in getting started with producing things with your little CNC machine - remember it is best to use simple graphics when first starting with your machine.