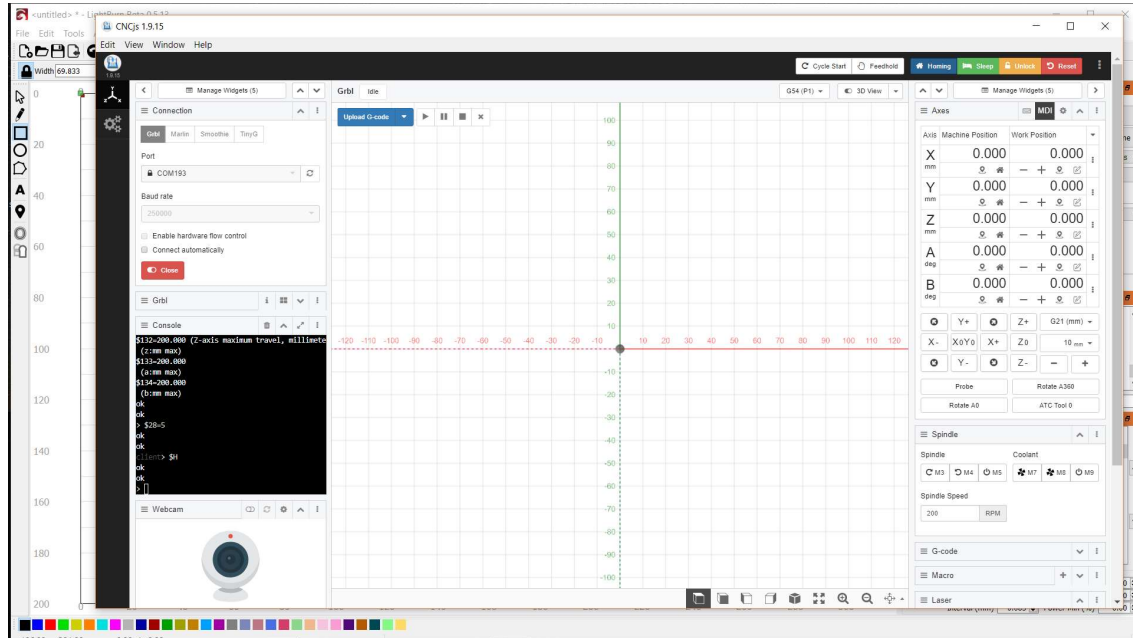


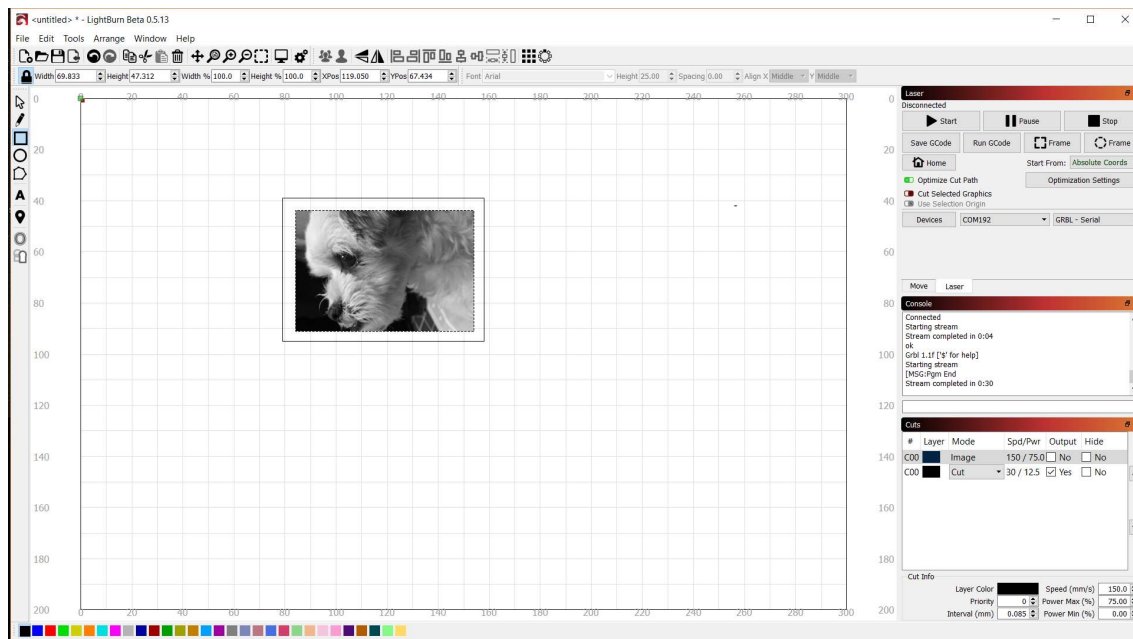
## LightBurn setup

Check connectivity first via a G-Code sender (CNCjs depicted below but any will do like UGS).  
Connect via Grbl profile, 250.000 Baud, refresh the ports (tiny button beside the COM port selection). Click on the drop down list and select the COM port.

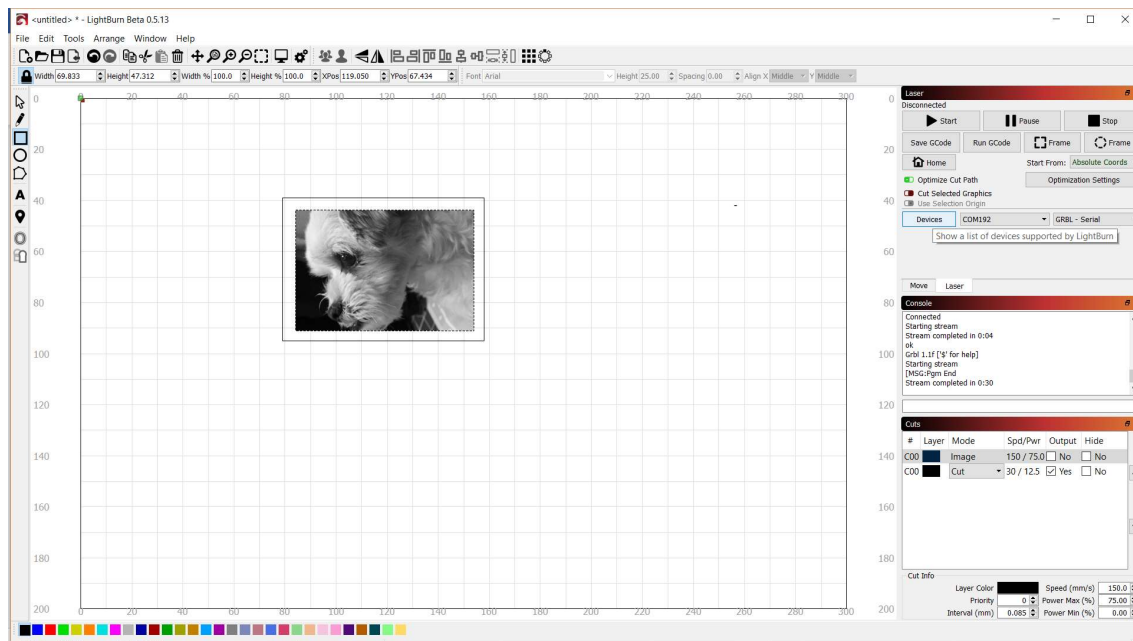


## Install LightBurn

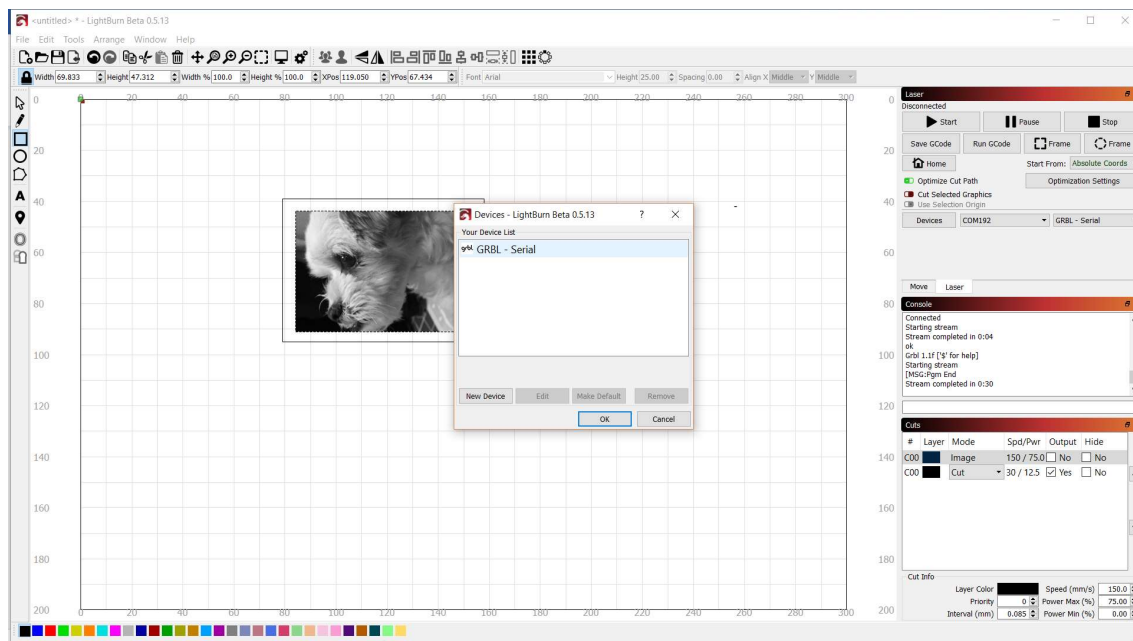
Import a small test image and draw a square around it (left ribbon, square icon).



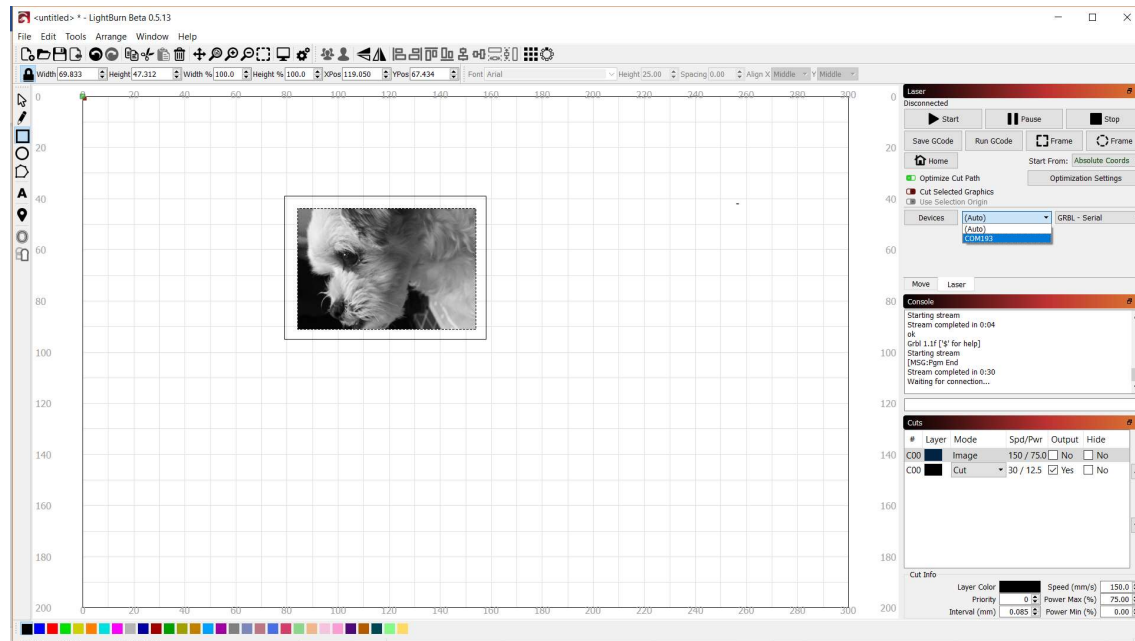
Click on Devices in the right scroll section (“Show a list of devices supported by LightBurn”)



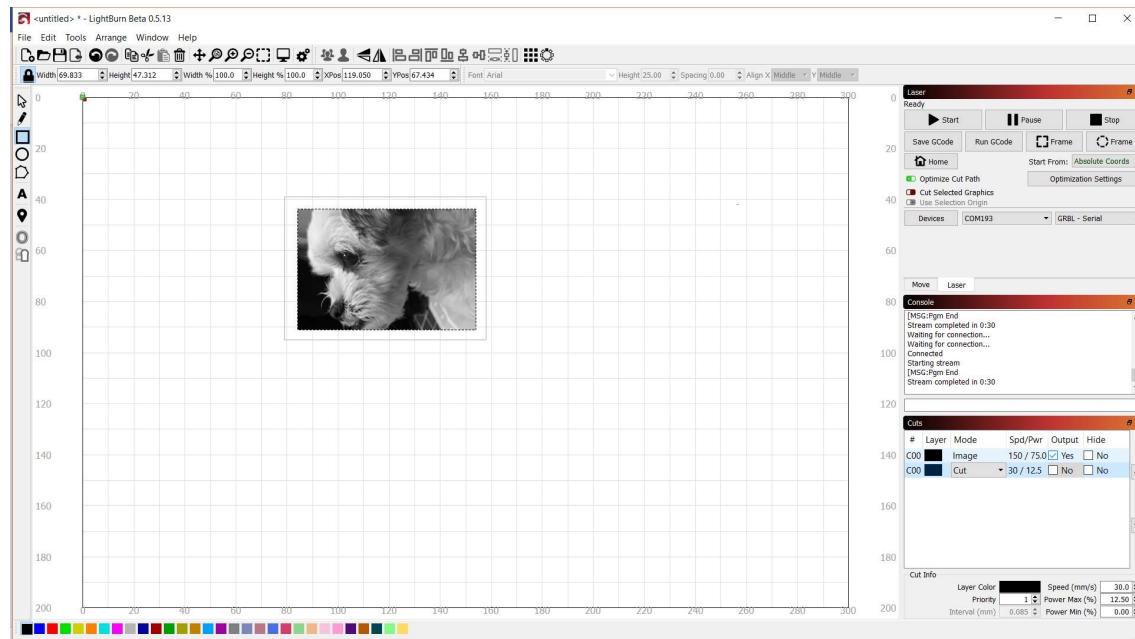
Click new Device and select GRBL – Serial and follow the buttons next and accept the defaults.



Click in the right scroll section of the screen, next on the auto dropdown selection list. Select the COM port that you see. This should be the same com port number you saw before in the G-Code sender. Ensure you disconnected the G-code sender before connecting in LightBurn. Click Home in the top left under start and check whether the machine homes correctly,



Select in the right scroll section of the screen, the lower 3<sup>rd</sup> Part called Cuts. Check the Output check box to test the raster-ing. Makes sure that the travelling speed is 150 and power 75%. Click on start and the raster starts running. Adjust the power if needed. After that uncheck the output checkbox and check the cut output. Select speed 30 and power 12.5% and click Run.

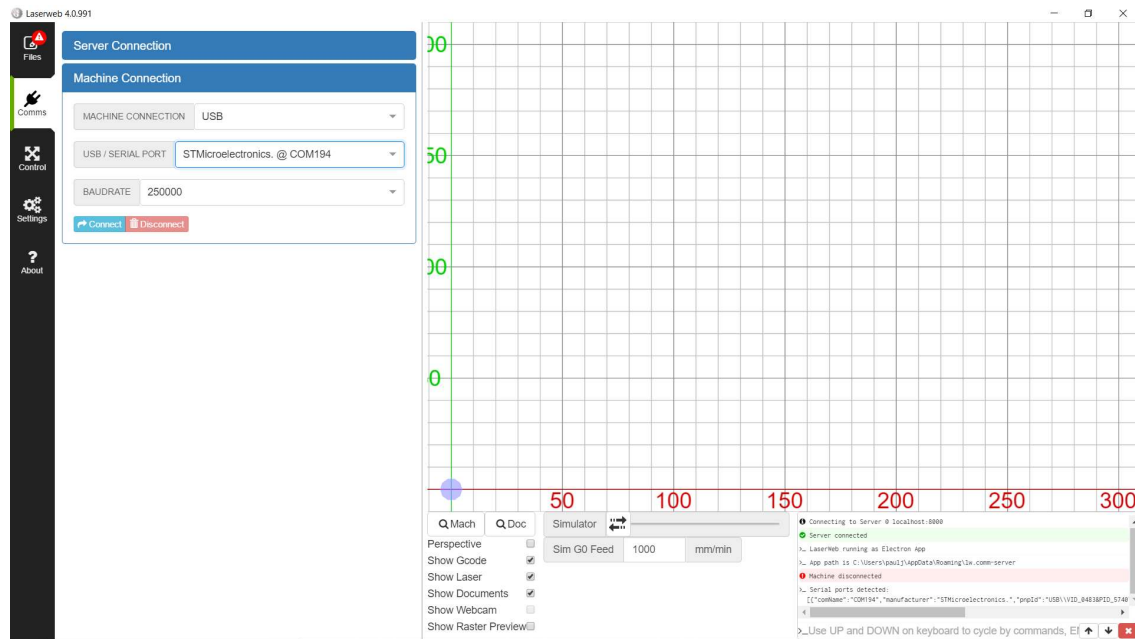


## LaserWeb setup

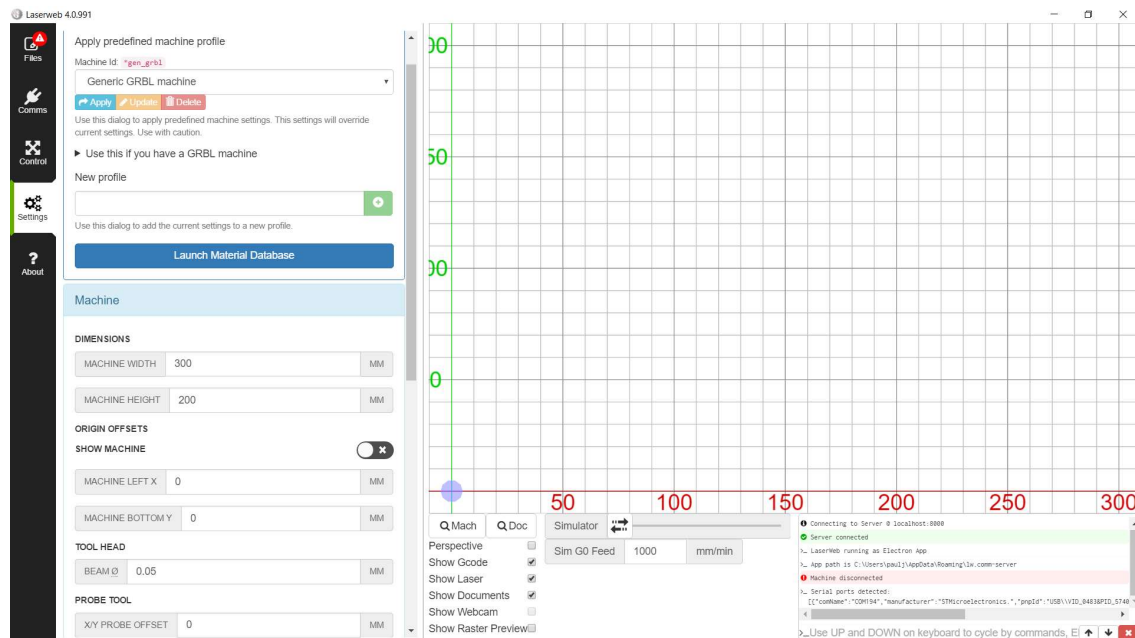
Check connectivity first via a G-Code sender (CNCjs depicted below but any will do like UGS). Connect via Grbl profile, 250.000 Baud, refresh the ports (tiny button beside the COM port selection). Click on the dropdown list and select the COM port.

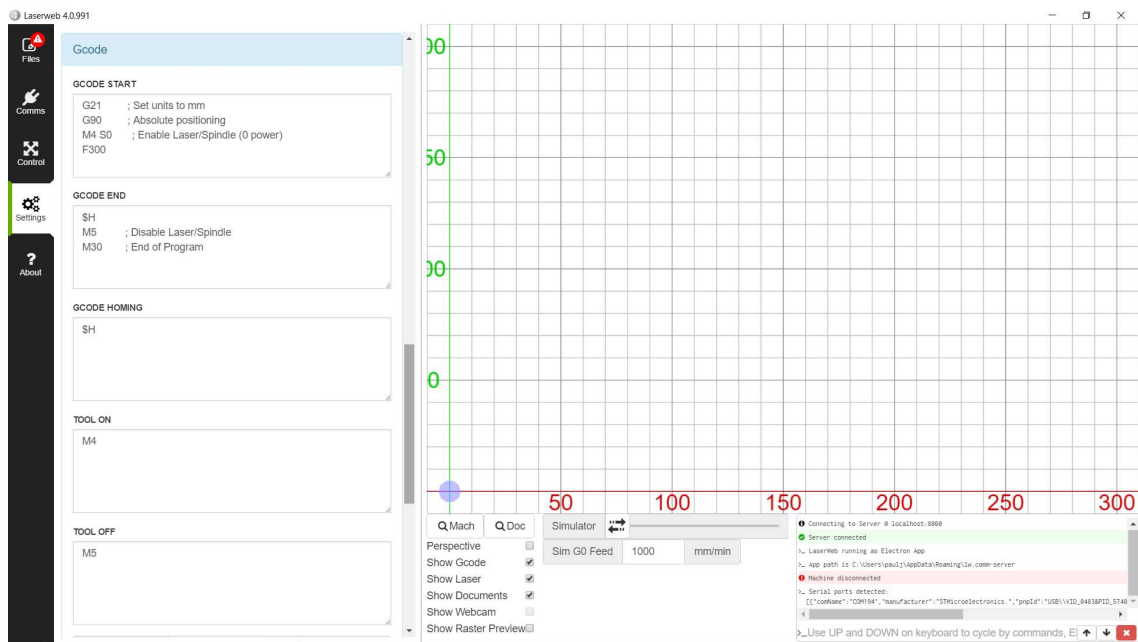
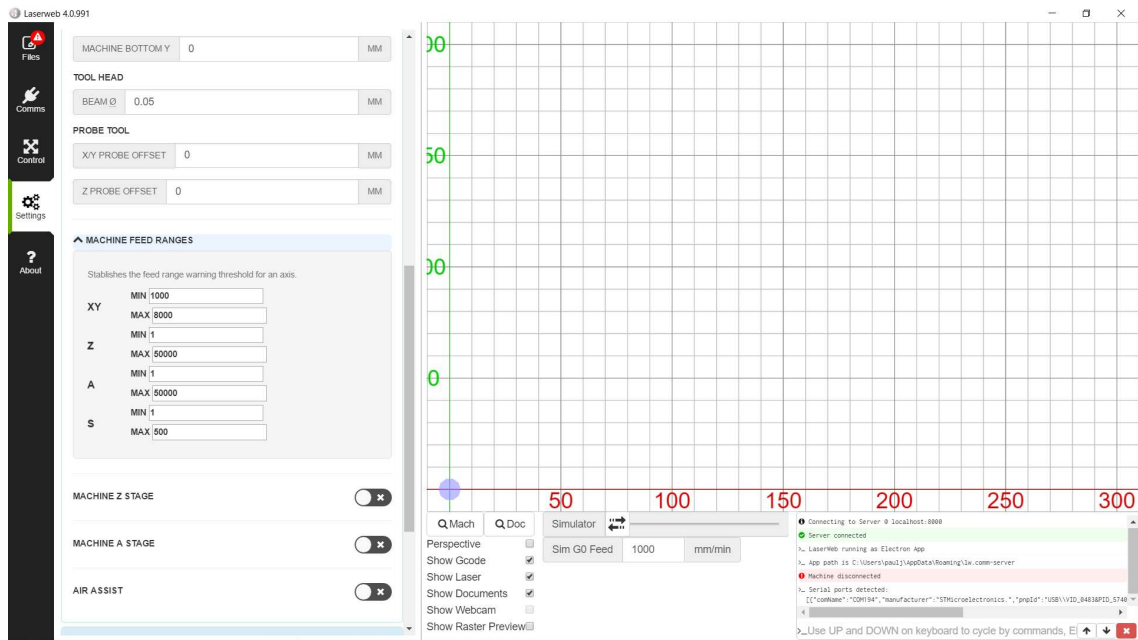
After that disconnect and install LaserWeb.

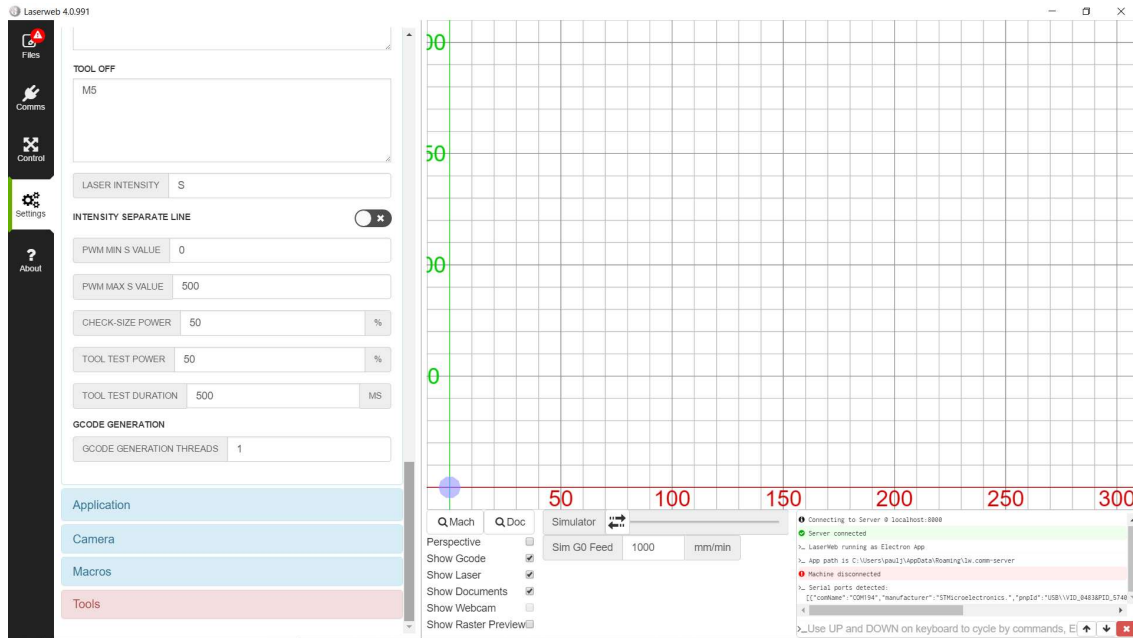
Click on the left ribbon “Comms” and select machine connection details as depicted below. The com port number should be the same you found previously when testing via the G-Code sender.



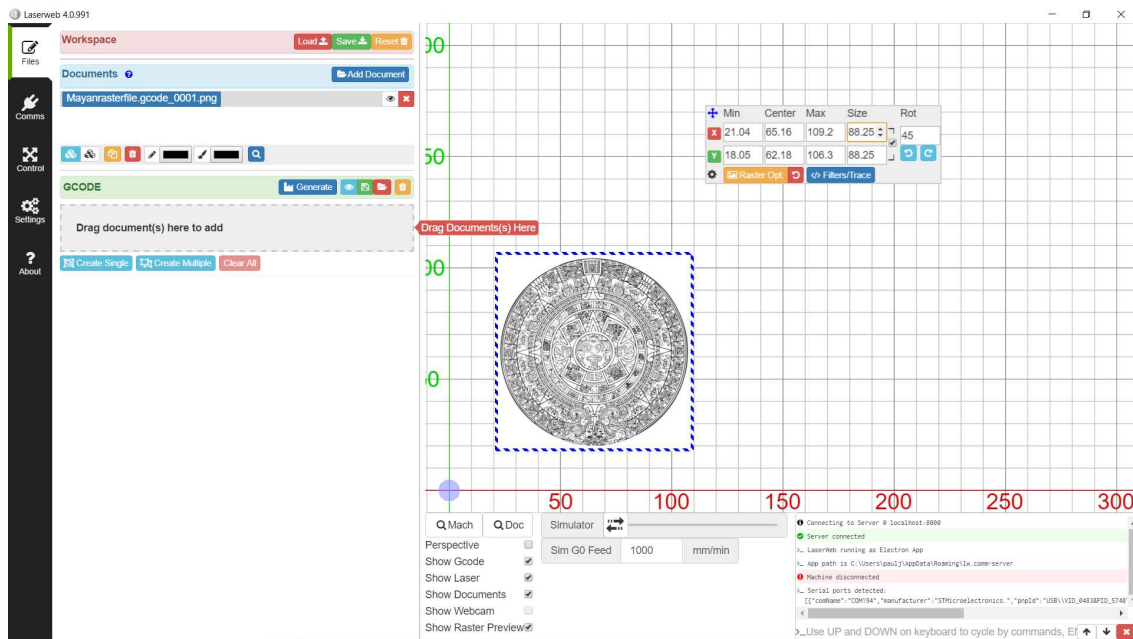
Click on the left ribbon, “Settings” and select the Generic GRBL machine profile. Enter the details as depicted below.





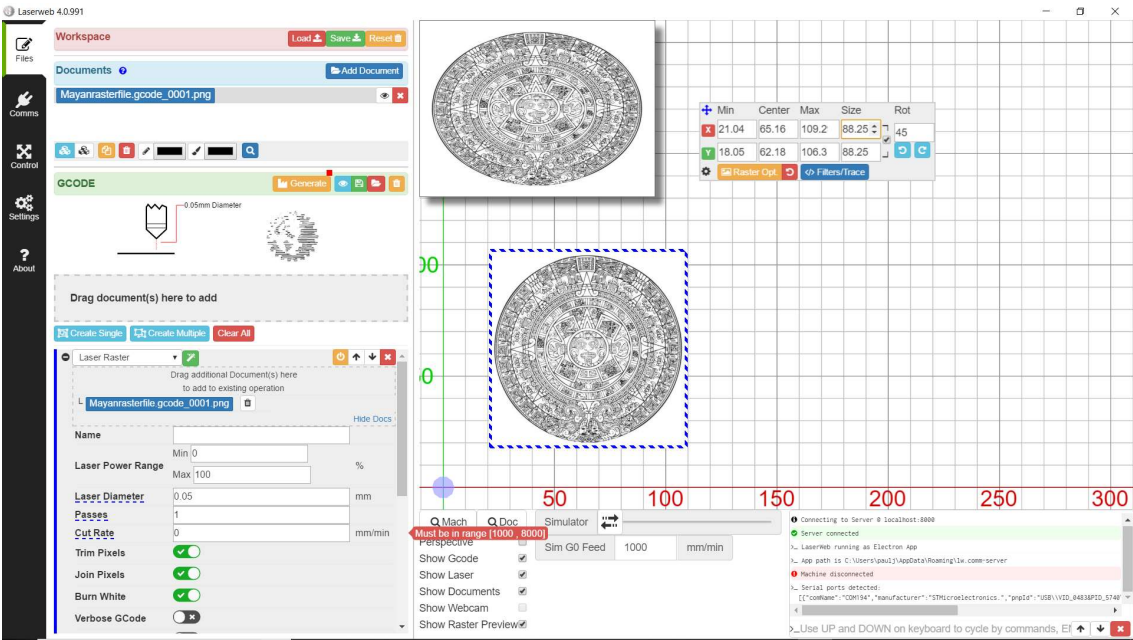


Now it is time to test the K40 with the program. Click “AddDocument” and the picture appears in LaserWeb (here a Mayan Calendar). Drag the document into the field below called “Darg document(s) here”.

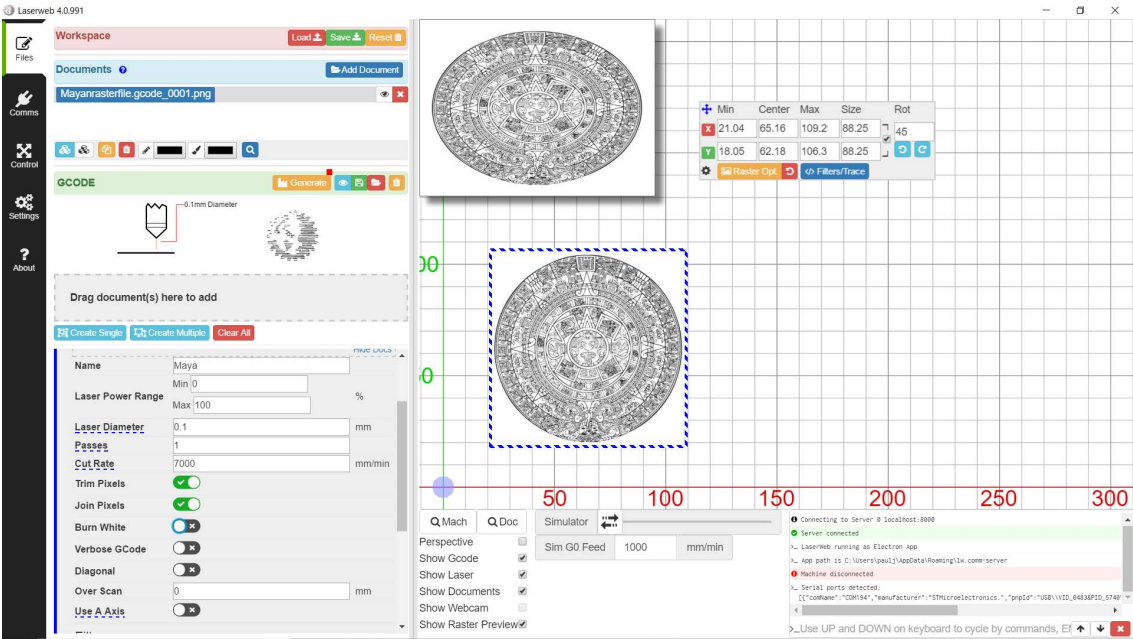




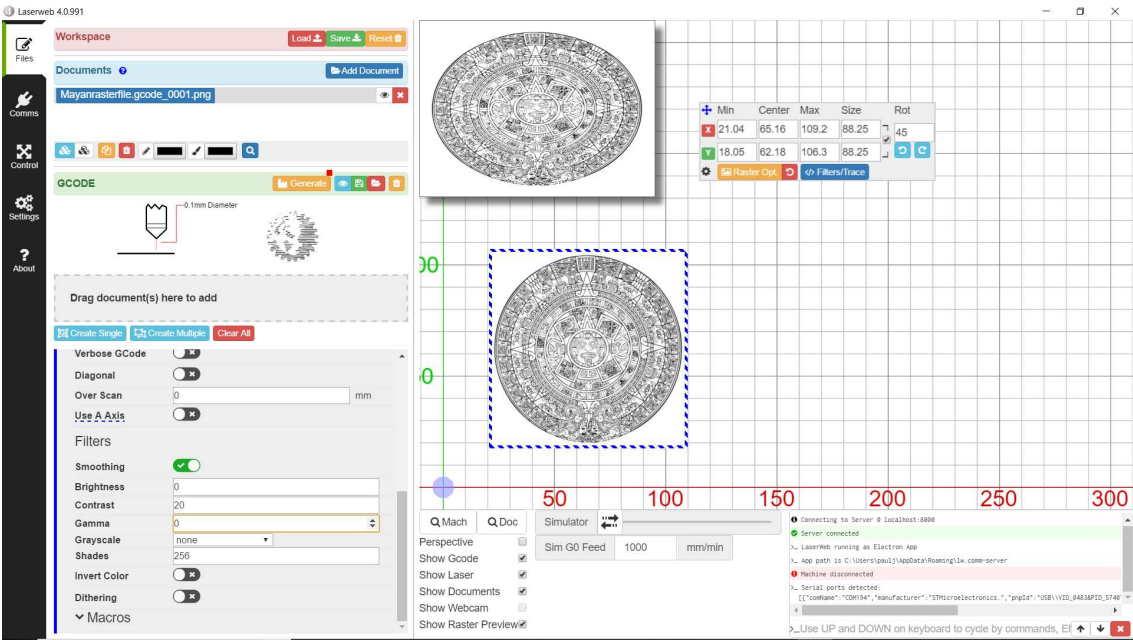
Enter the details depicted below like the name, laser diameter 0.1mm, uncheck “Burn White”



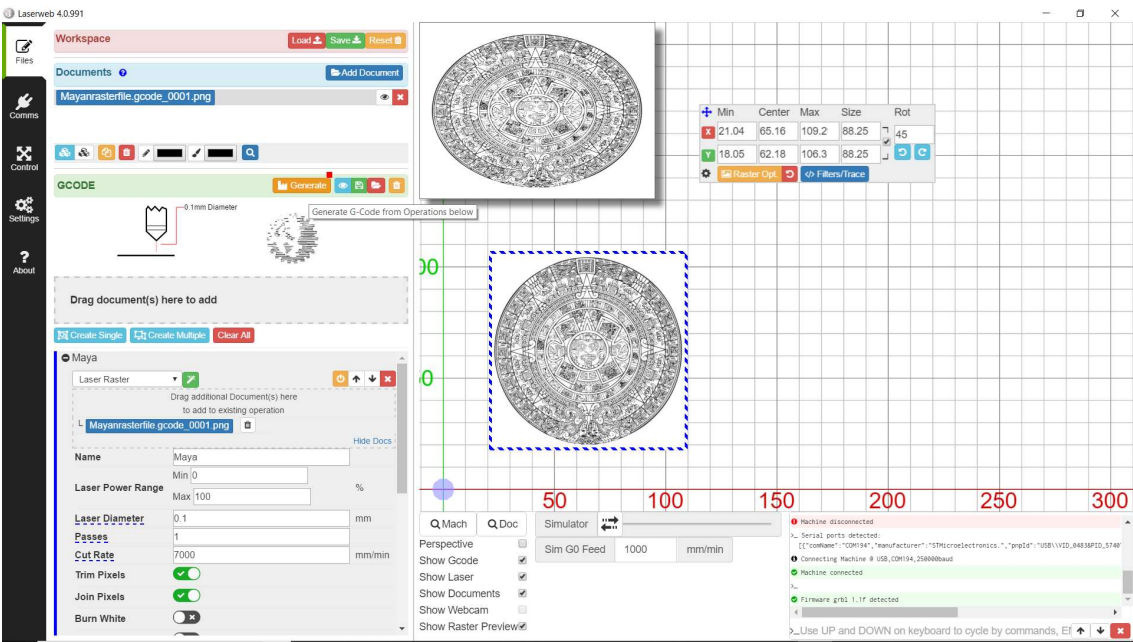
Enter the cut rate like 7000 to start with



Click the “Smoothing” check box.

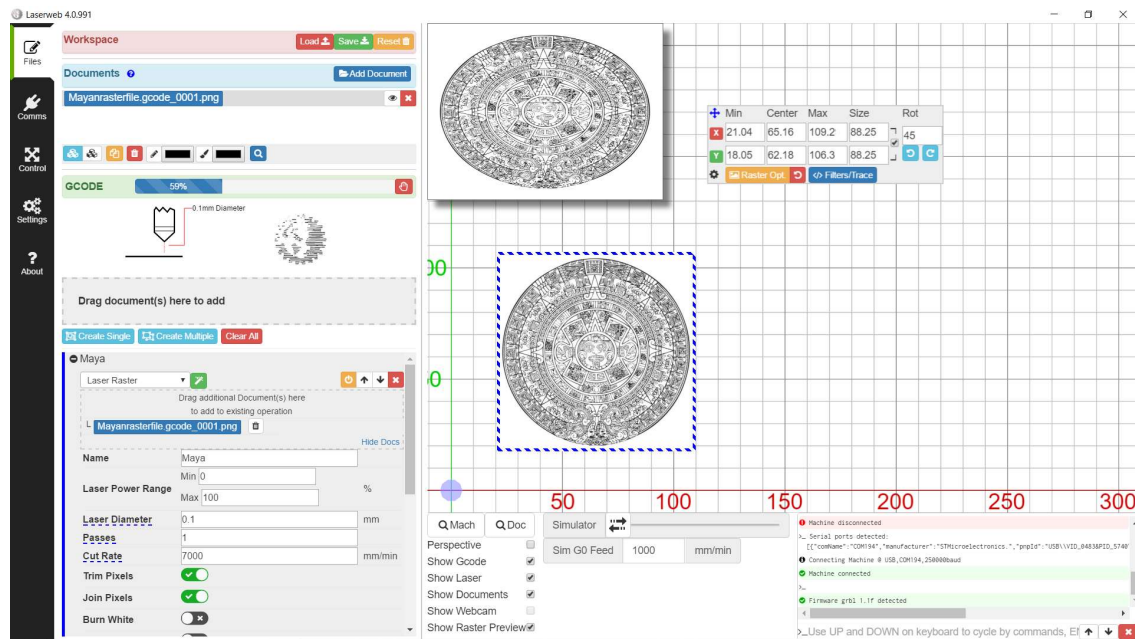


Click on the “Generate” button in the G-Code section





You see now a status bar appearing in the G-Code section, wait till it is finished. Start with small files since it can take a while.



After finishing the G-Code generation, click on the left ribbon, “Control” and home the machine. Click on “run – job” to start the engraving process. Note that the co-ordinates and image are mirrored so you might need to mirror the image first before uploading into LaserWeb.

