

1. Download LightBurn software from

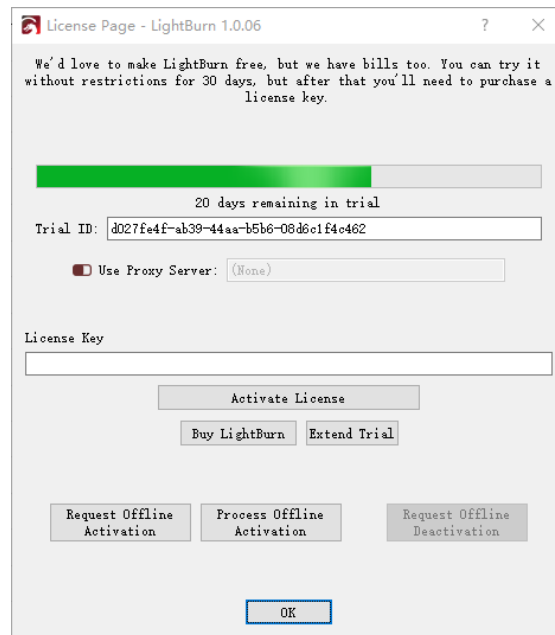
<https://lightburnsoftware.com/pages/trial-version-try-before-you-buy>

Notes: The free trial period of the software is 1 month. Visit

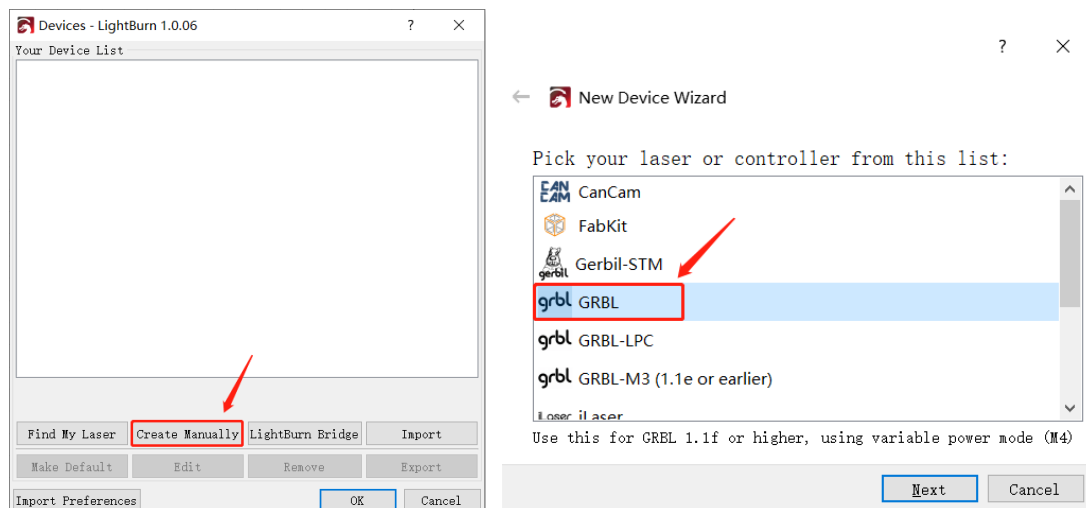
<https://lightburnsoftware.com/collections/frontpage/products/lightburn-gcode>

for permanent serial number.

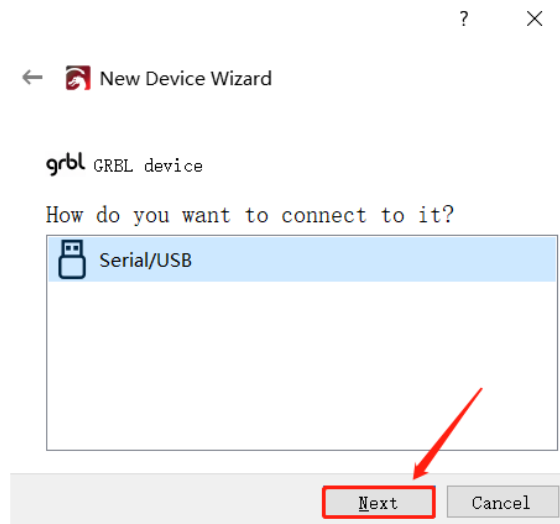
2. Open LightBurn after installation. Click “OK” to close the License Page.



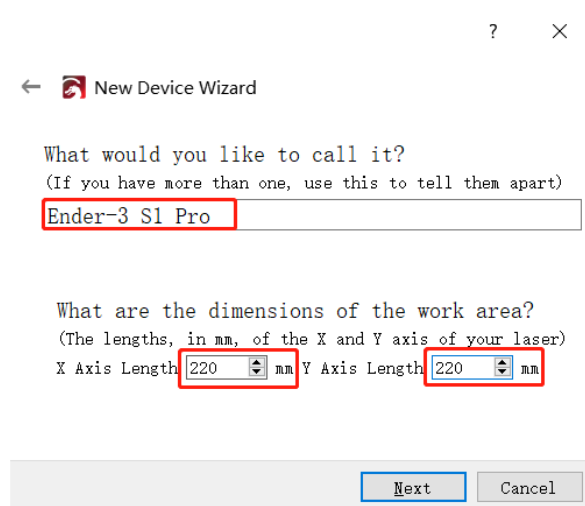
3. Click “Create Manually”, choose “GRBL” and click “Next”.



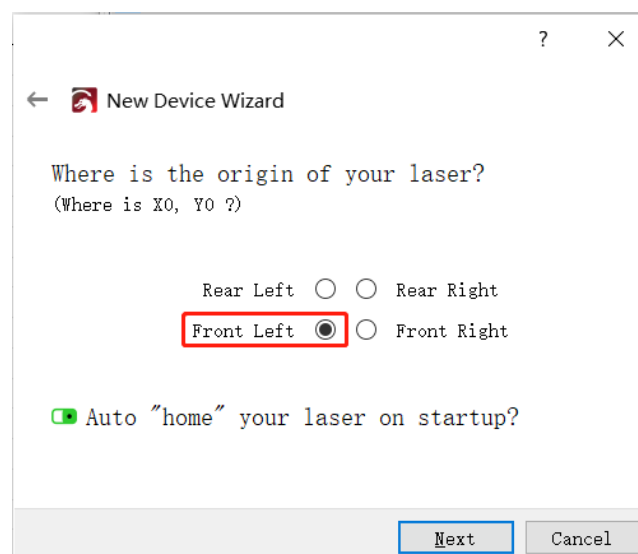
4. Click “Next”.



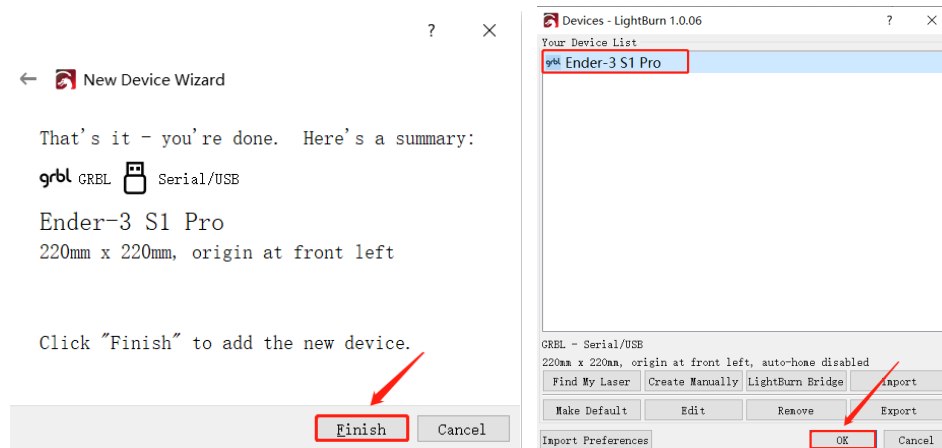
5. Take a name and set the maximum engraving size. Then click "Next".
Take Ender-3 S1 Pro as an example:



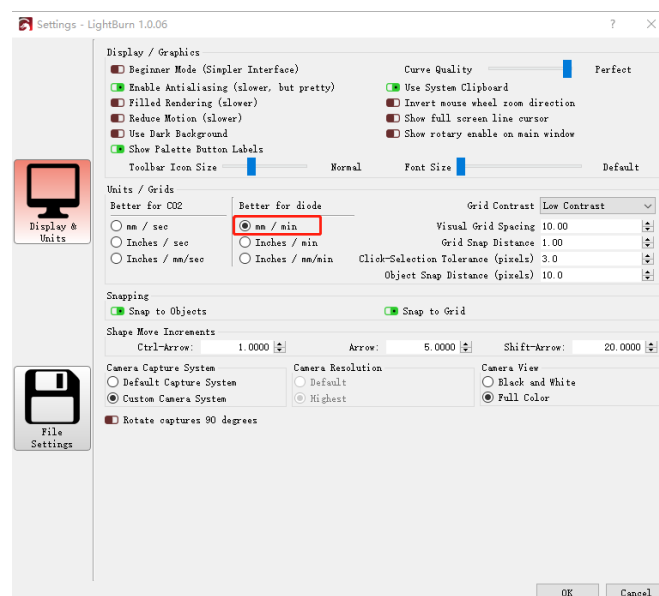
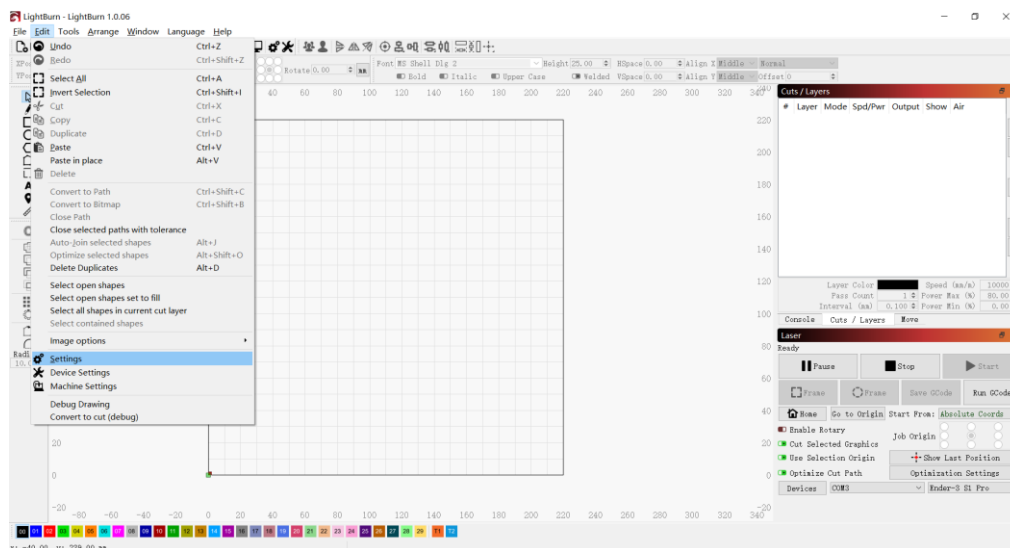
6. Set the origin to "Front Left" and click "Next".



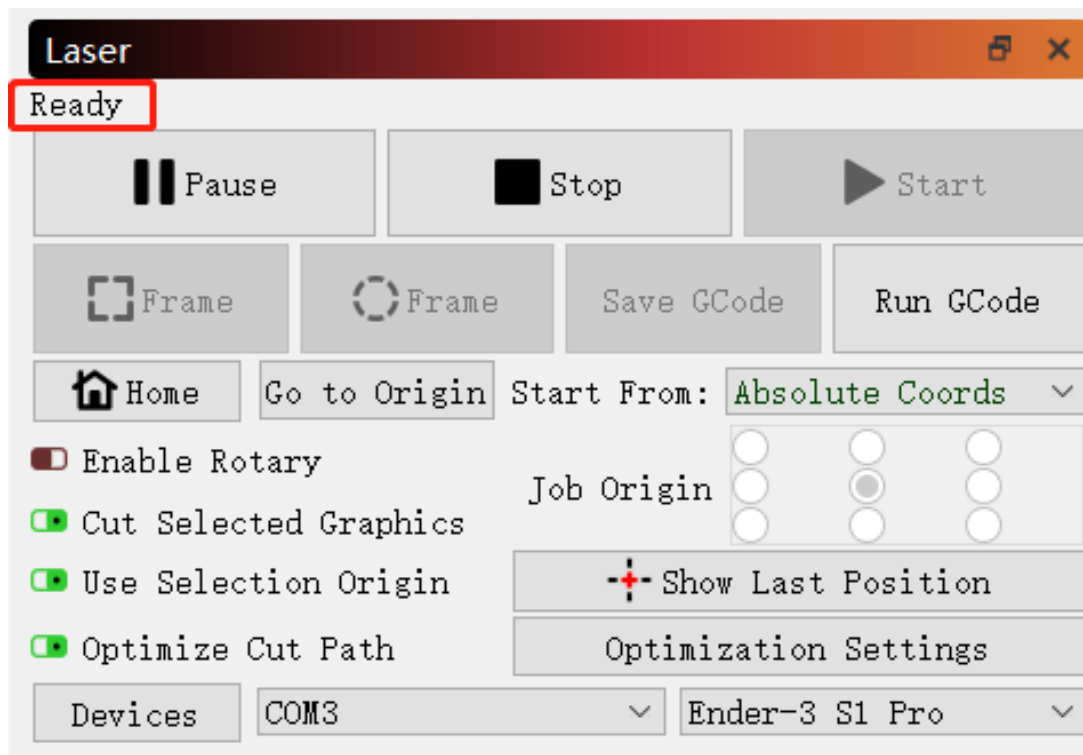
7. Click “Finish” finally. There is a corresponding machine in your device list. Select the machine and click OK to configure successfully.



8. Click “Edit-Settings”, choose “mm/min” and click “OK”.



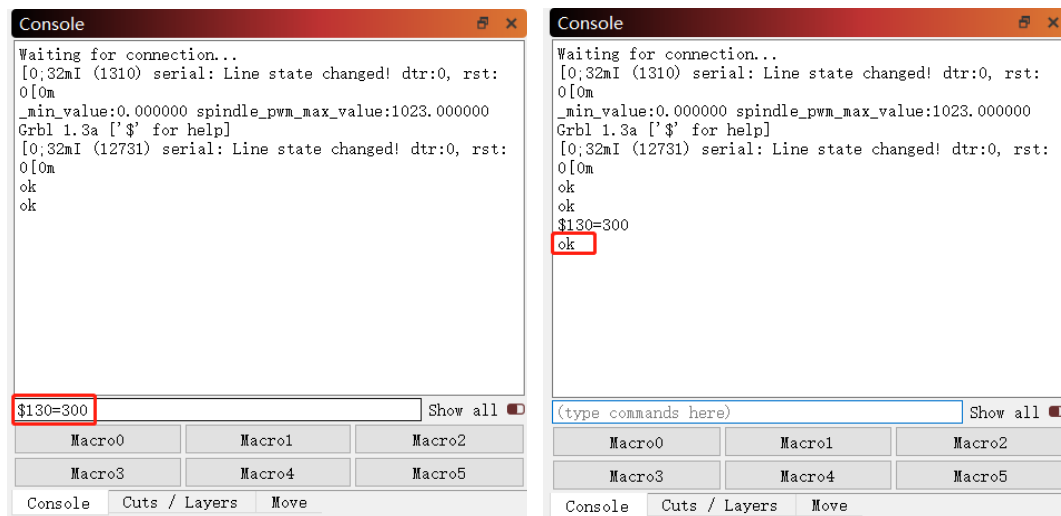
9. Use the Type-C data cable to connect the control box to the computer. Turn on the switch on the control box and select the corresponding port. You can see "Ready" if the connection is successful.



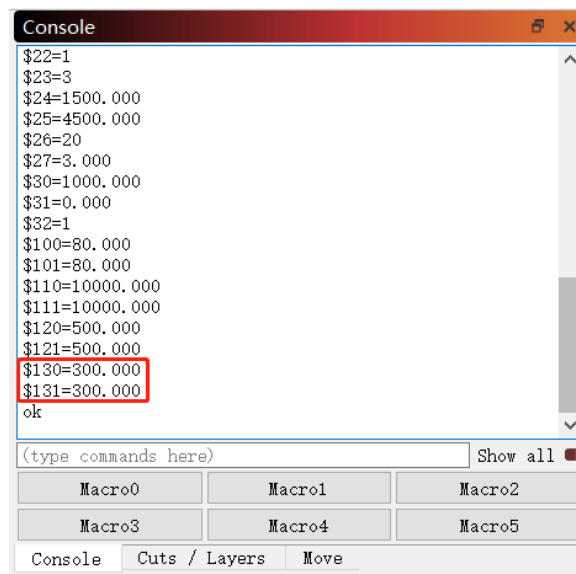
10. For different printers, if the X and Y print size is not 220mm, you need to modify two configuration parameters of \$130 and \$131 in the console. Please refer to the TF card file "User manual for LightBurn" for details. The following table shows the configuration parameters corresponding to different printers:

Configuration parameter machine model	\$130	\$131
Ender-2 pro	165	165
Ender-3	220	220
Ender-3 Pro	220	220
Ender-3 neo	220	220
Ender-3 max neo	300	300
Ender-3 s1	220	220
Ender-3 s1 pro	220	220
Ender-3 s1 plus	300	300
Ender-3 V2	220	220
Ender-3 v2 neo	220	220
CR-10	300	300
CR-10 mini	300	220

Take CR-10 as an example, enter \$130=300 in "Console" and press Enter and the value of \$130 is successfully modified:

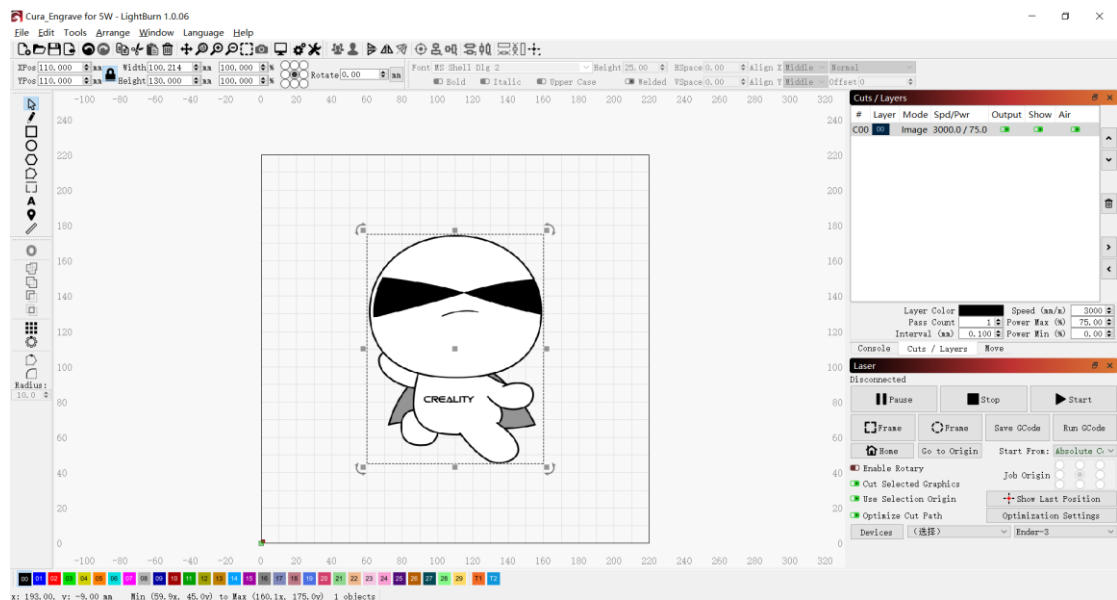
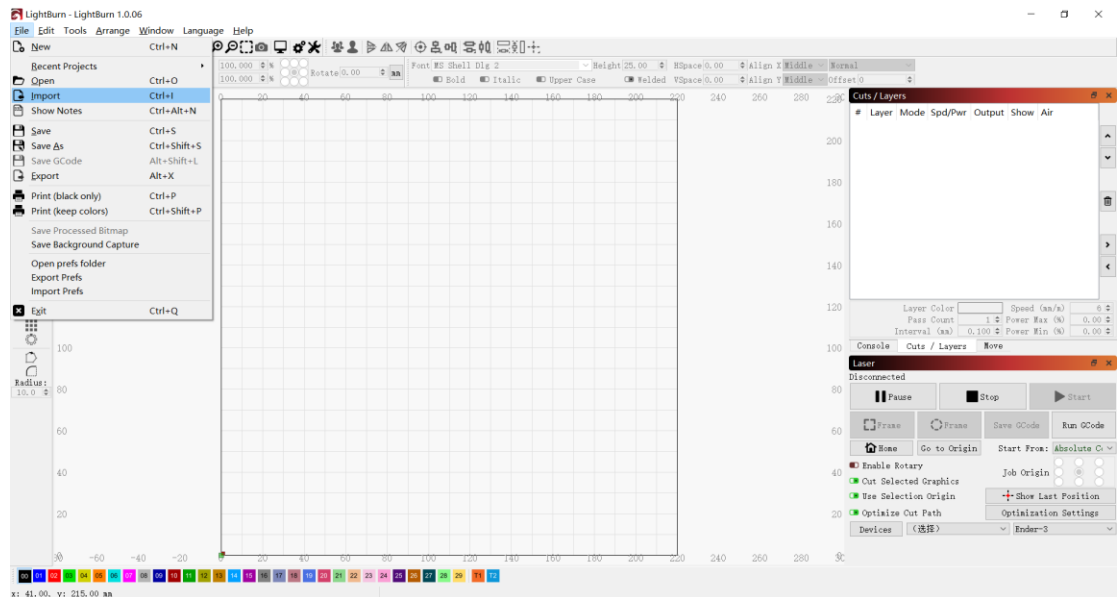


Similarly, if you want to modify the value of \$131, enter \$131=300 in "Console" and press Enter. After the modification is completed, enter \$\$ in "Console" and press Enter to confirm whether the parameters have been modified successfully. Please restart the control box after the modification is successful

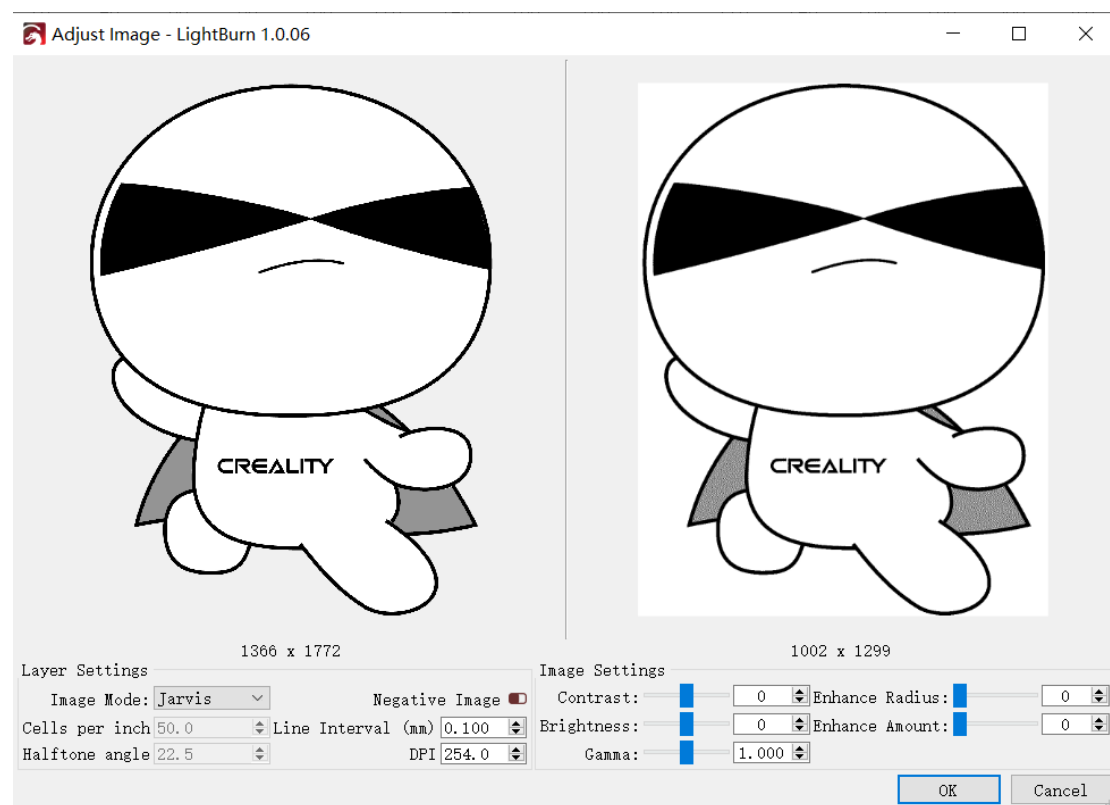
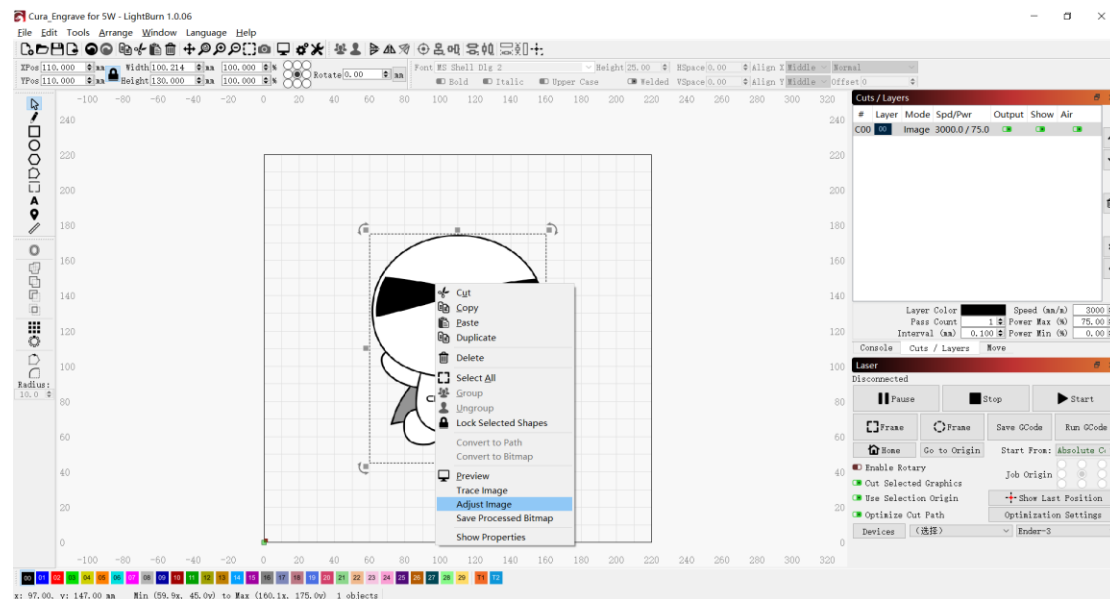


11. Click "File-Import" to import the picture. Set the position, size and rotation angle of the picture.

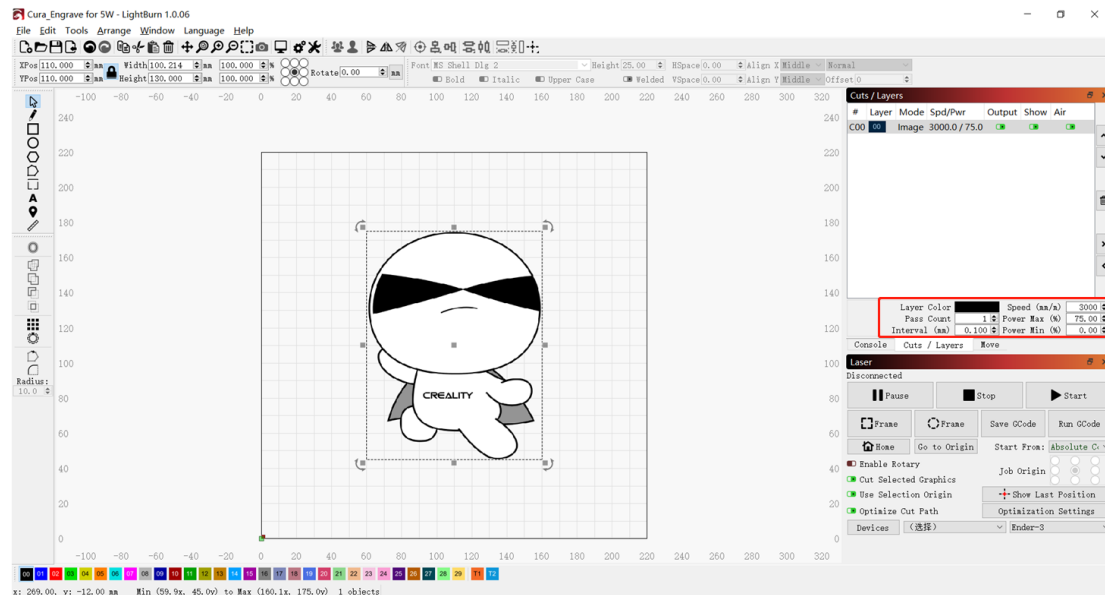
Tips: Usually import "dxf" or "svg", and set "Mode" to "Line" in the Cuts/Layers window for cutting.



12. Right click on the image. Adjust Image to set the effect and click OK.



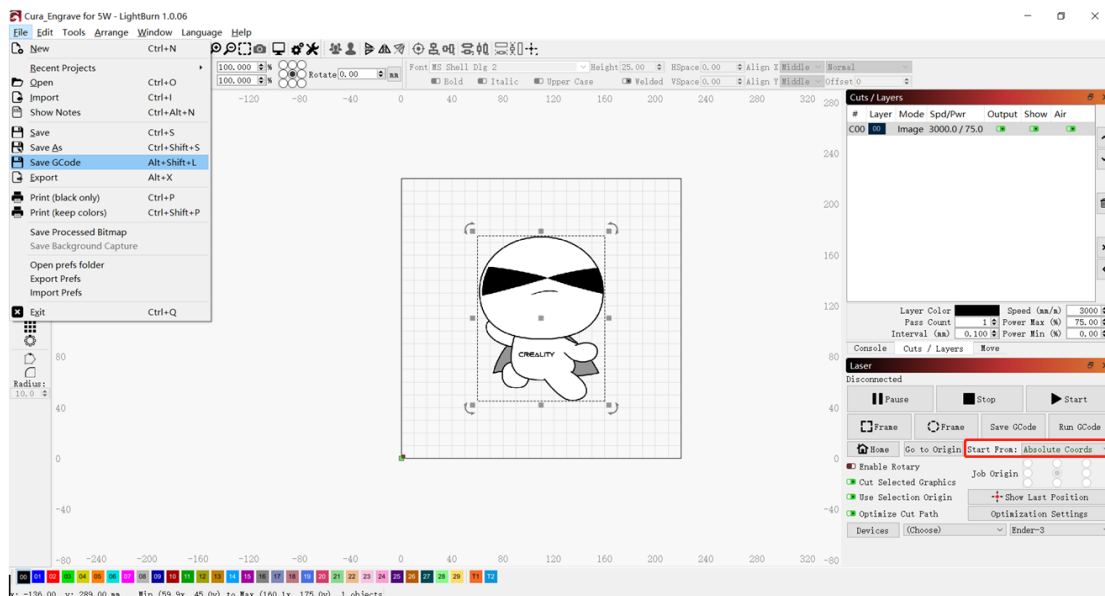
13. Set the appropriate Power %, Speed and Pass Count.



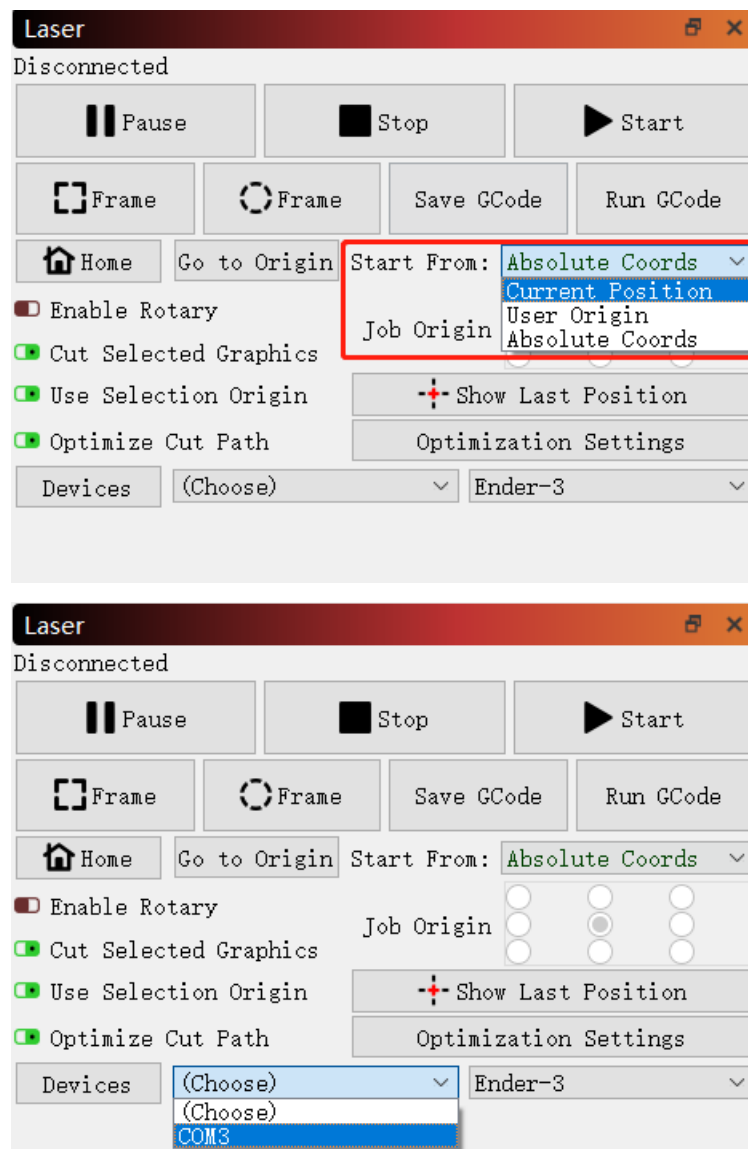
NOTES: Higher power % or slower speed make deeper effect; Lower power % or faster speed make shallower effect. Generally we just adjusted "Power Max". Please refer to the TF card for recommended parameters.

14. You must "Start From Absolute Coords" during offline-operation. Finally, click "File-Save GCode" to save the engraving file to the directory of TF card. Follow the quick start guide to start engraving.

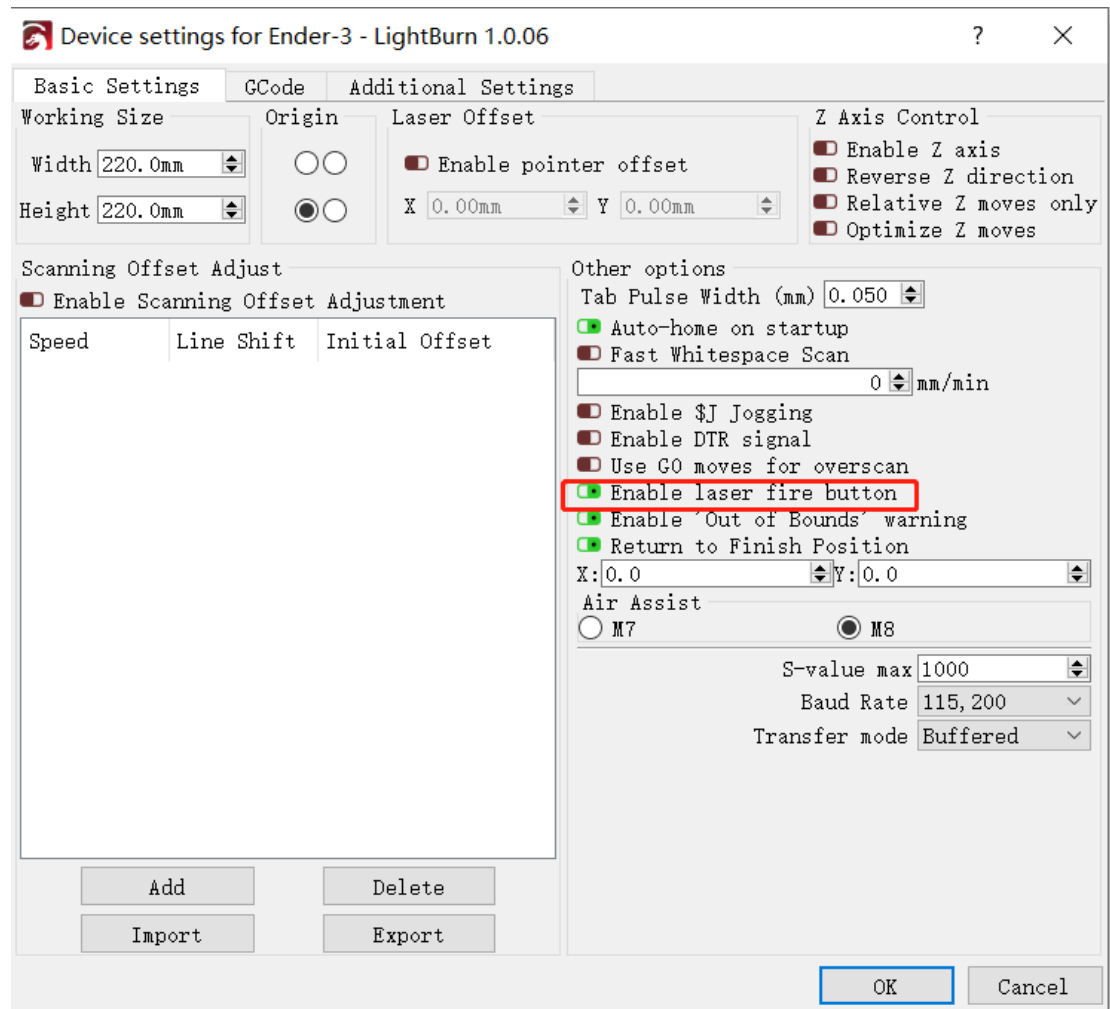
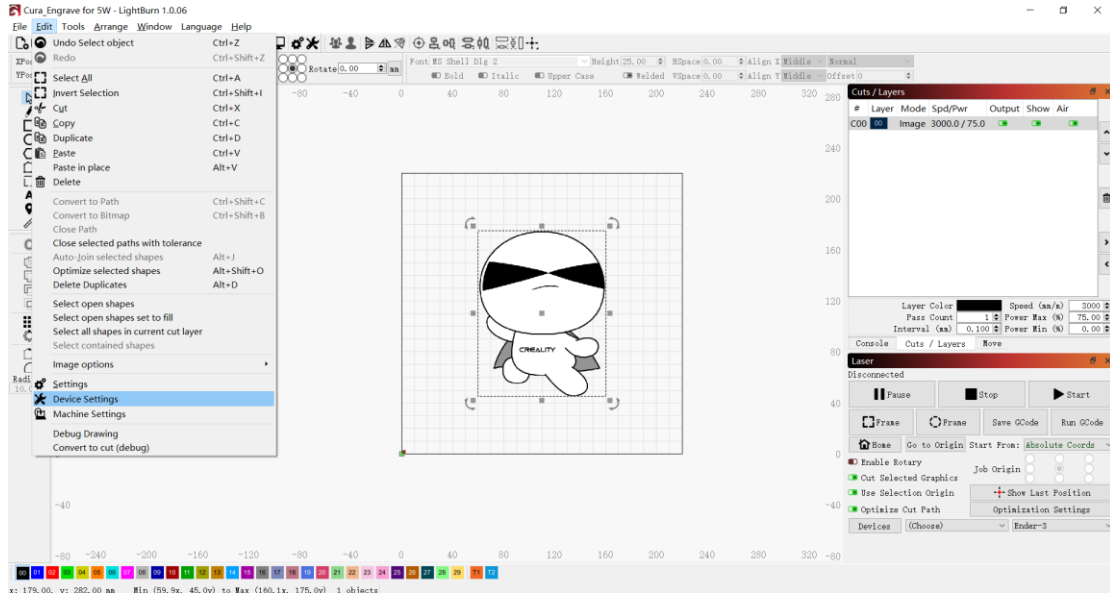
Note: Make sure that the format of exported file is ".gcode/.gc/g/.nc/.ngc". Other formats can not be recognized by the machine.

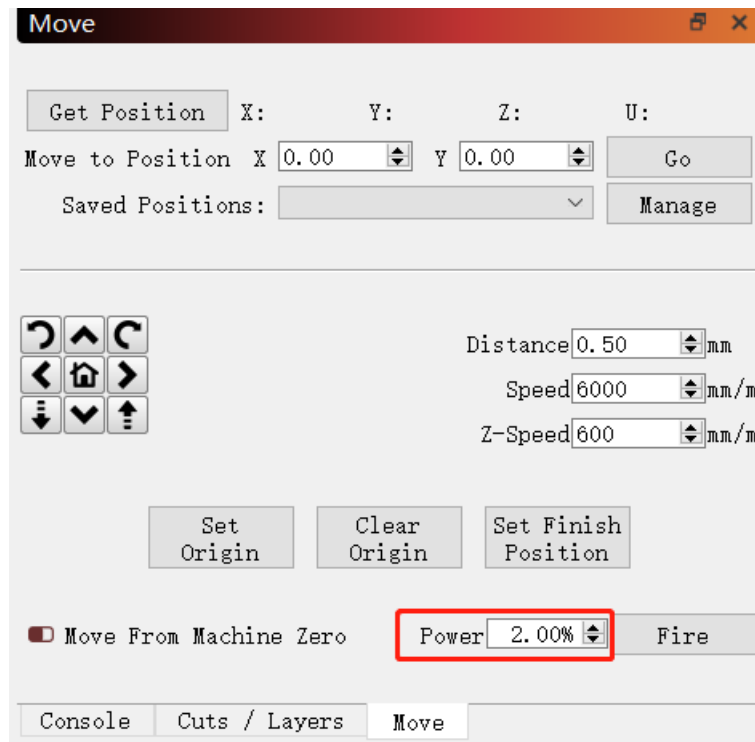


15. You can also connect the engraving machine through Type-C data cable to engrave in real time and you should select the corresponding Com port. Choose "Start From" according to personal habits. If you choose "Absolute Coords", please click "Home" before engraving.



Notes: Before click “Frame”, you need to enable the laser fire button in “Edit-Device Settings”, and then restart LightBurn. After setting the preview power in “Move”, press “Shift” and click “Frame” to preview.





Refer to the website <https://lightburnsoftware.github.io/NewDocs/> for detailed software tutorials.

Others

1. If the line cannot be closed, you can set "Overcut" in Cut Settings Editor.

