Guide to Laser Cutting the T-Rex

Hardware

Custom CNC (MPCNC with Ramps 1.4 running Marlin)
Endurance 3.5 (3500mW) Watt Laser
LASER PROTECTION EYEWEAR (Yes, this is essential, A Laser will blind you, do not use your laser without the appropriate eyewear)

Software

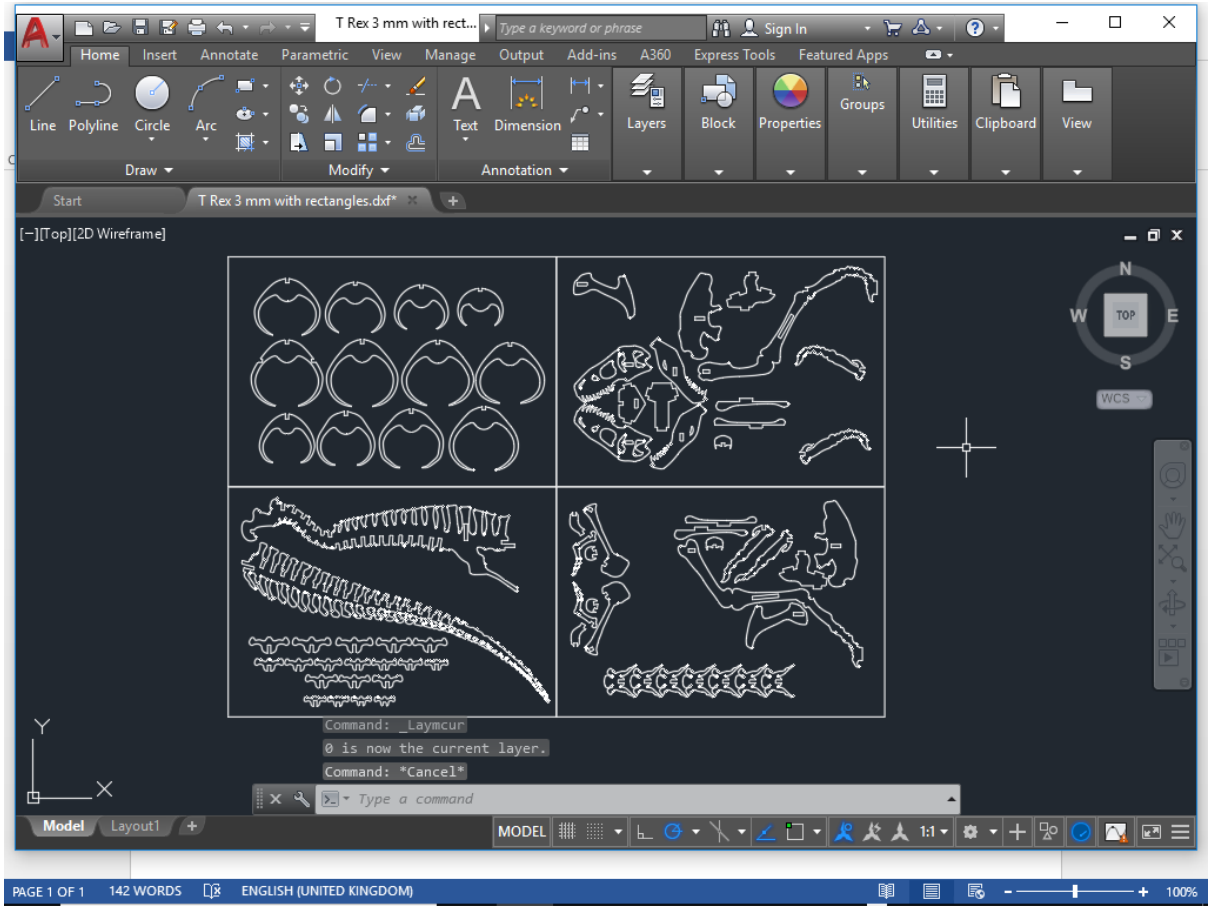
AutoCAD 2018
Estlcam V11
Repetier V2.05Host
Notepad++

The plans I used are available at www.makecnc.com, this is a good resource for Hobby CNC users, please note I have no affiliation with this company, I came upon them after a google search.

The plans come in several formats, for me I used DXF so I could import them into AutoCAD.

My first steps was to modify the plans for use with the current wood stock I had available which was sheets of 3mm – 3.4mm Plywood at 300mm x 210mm.

In AutoCAD I drew a rectangle of 300mm x 210mm to represent the wood stock, I then placed all of the components into one of these rectangles, in my case I needed 4 rectangles.

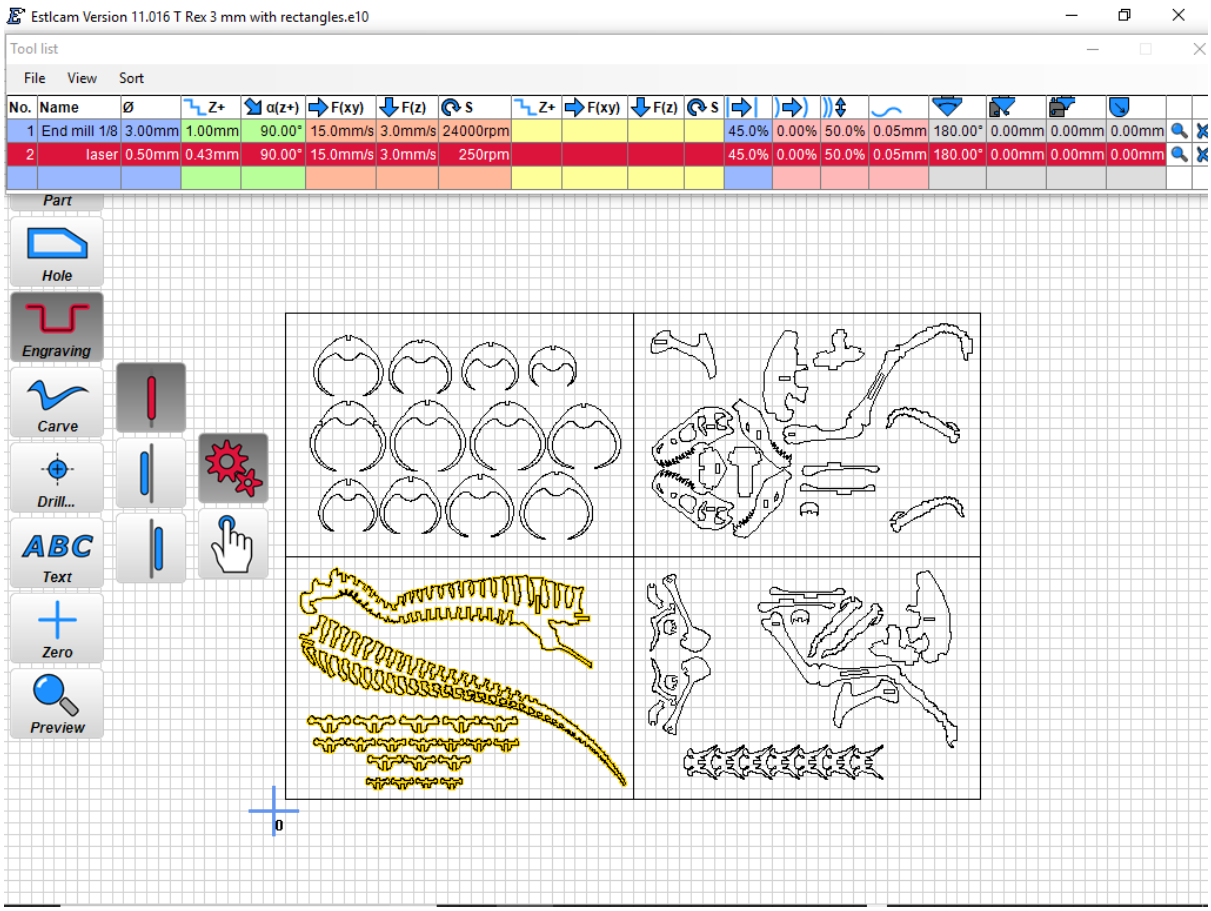


When I was happy with that I saved this as a DXF.

I then imported this into Estlcam

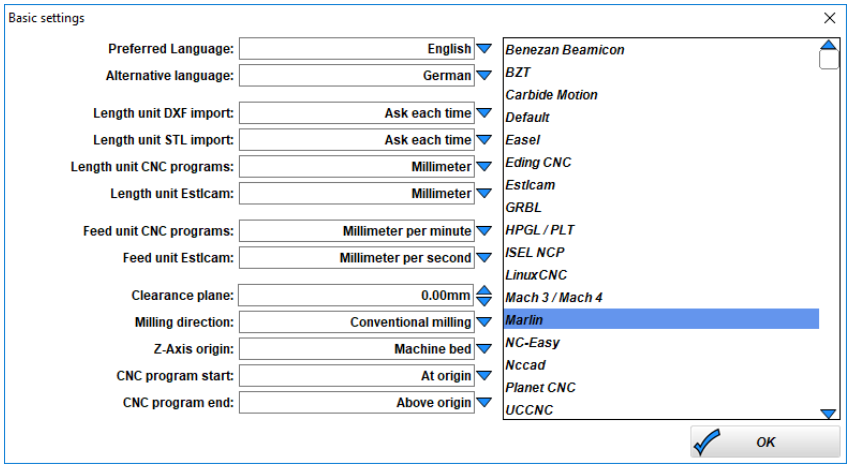
In Estlcam I used the Engraving Tool set on Centred with Automatic Object Detection and clicked on each object I wanted to create a cutting path for, please note I did not cut the rectangles, they are only used for reference.

I zeroed my CNC 10mm x 10mm from the corner of the work piece, each rectangle represents a single work piece so these are done individually, I could have done all 4 at the same time, but decided to do one piece at a time.



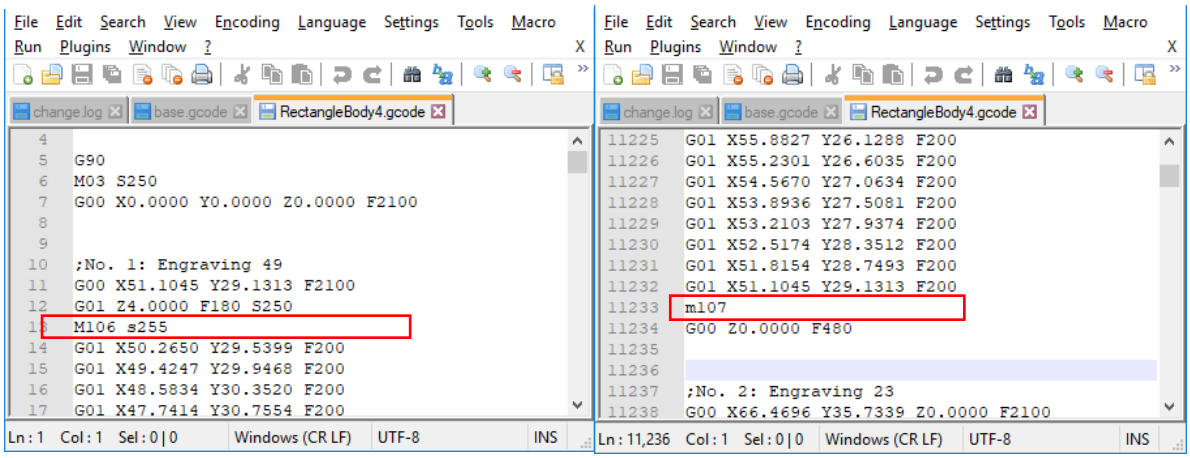
My tool path depth is set at 3.4mm and my Laser Depth per Pass is set at 0.43, which will work out at 8 passes per object dropping my Z Axis height 0.43 mm each pass.

My Estlcam parameters are set as follows when using my laser.



When I was happy with my Estlcam setup I then saved the CNC program and Estlcam converts everything into GCODE.

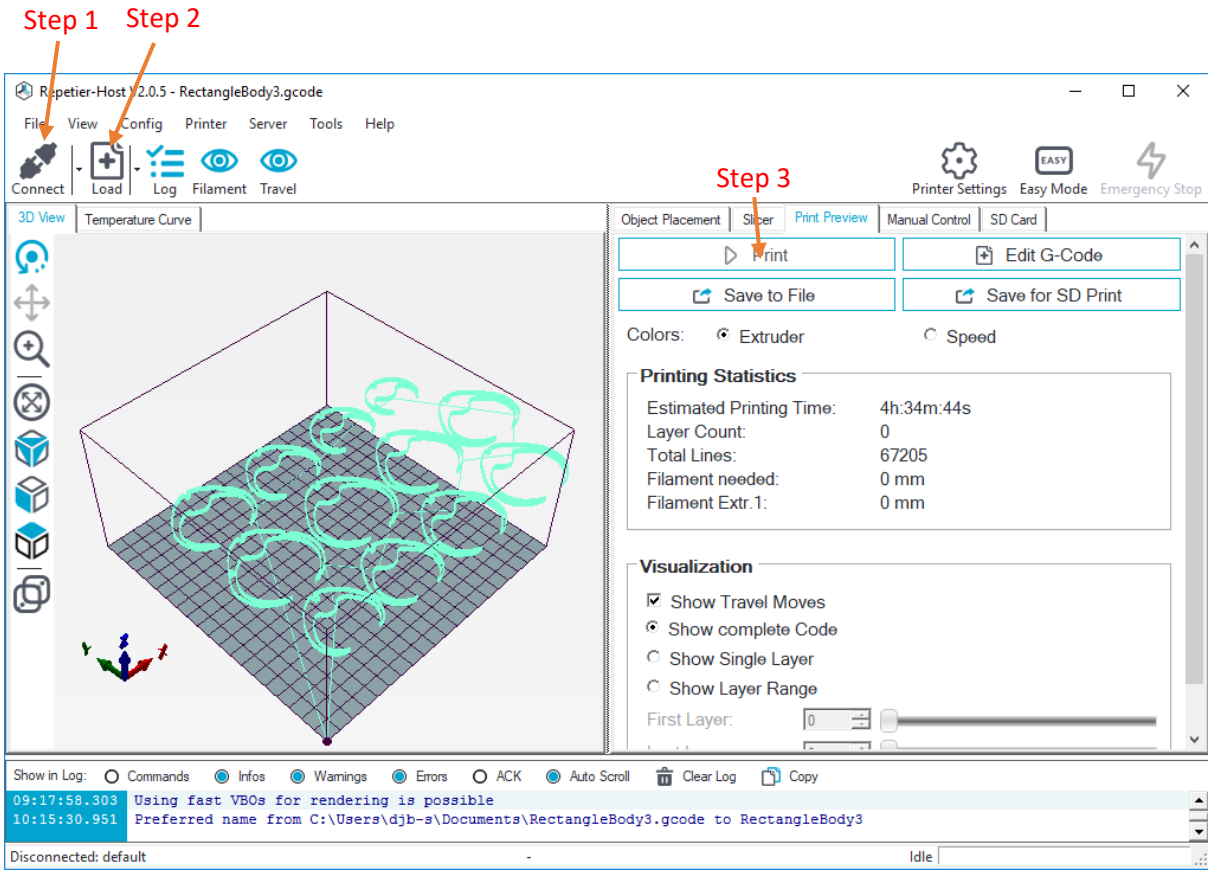
This next step may or may not be needed, I am just new to Estlcam and if this option exists great, if not my way works. I cannot find out how to turn a laser on and off within Estlcam so I put the codes in manually by editing the GCODE within Notepad++



So for each engraving I insert my Laser Start and End Codes, for me using Marlin and Ramps 1.4 my Laser On is Code M106 S*** *** represents the power to the laser 0 = off, 255 = full power and Laser Off Code is M107. Please see above, these codes are placed at the start and beginning of each Engraving Pattern.

Lastly within Notepad++ I do a find and Replace on my movement speed and change this from F*** (F*** being whatever speed it's set to in the GCODE) to F200. Again being new to Estlcam I may be missing this option as it was easier for me to manually do the Feed rate than trying to work it out.

Once happy I save this and open up Repetier V2.05 Host. To be honest this is the easiest part, only requires 3 Steps. First I click on Connect to connect to my CNC Machine, then Load to Load the GCODE file we saved earlier, then Print to Start the Cutting.



Step 1 – Connect
Step 2 – Load
Step 3 – Print

Some info about my laser focus.

For me I found my laser was focused best at 65mm from top of cutting surface, so I would manually adjust my Z height to take into account the thickness of the Work Piece, in this case 3.4mm so I adjust my Z Height to 61.6mm, then the GCODE moves the Z Height back up to 65mm during operation, with a Z height decrease of 0.43mm each pass I found the focus to work great.

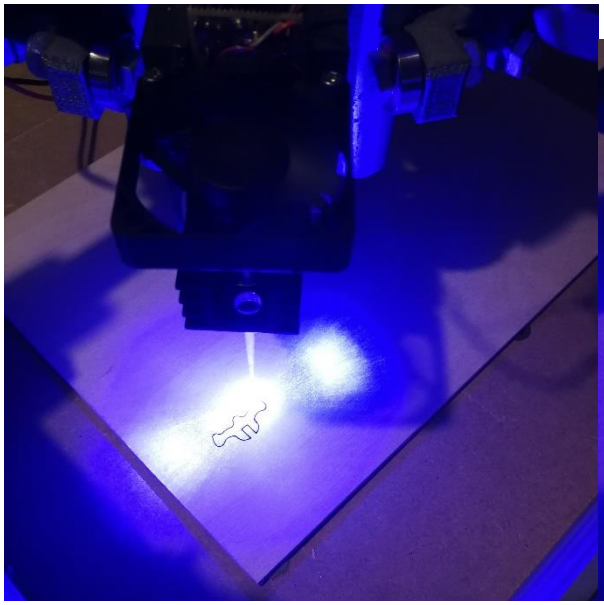
The only real issues I had with the whole cutting process was any kind of warping of the wood, since the focus is so precise warping does have impact on the cutting process where I found sometimes not all cuts passed fully through the wood.

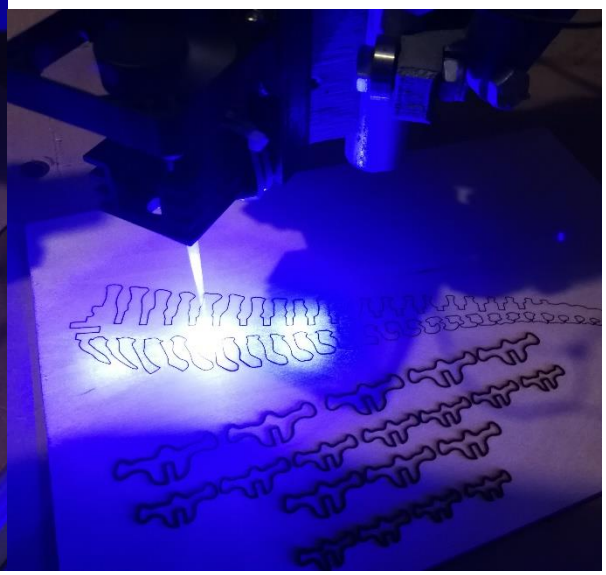
Please see some pictures below.

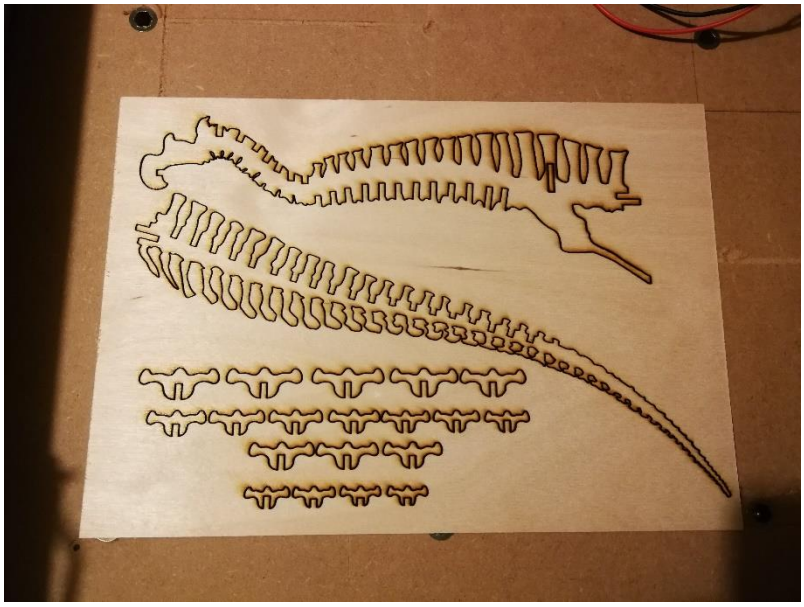
My first real laser cutting project, meet the T-REX

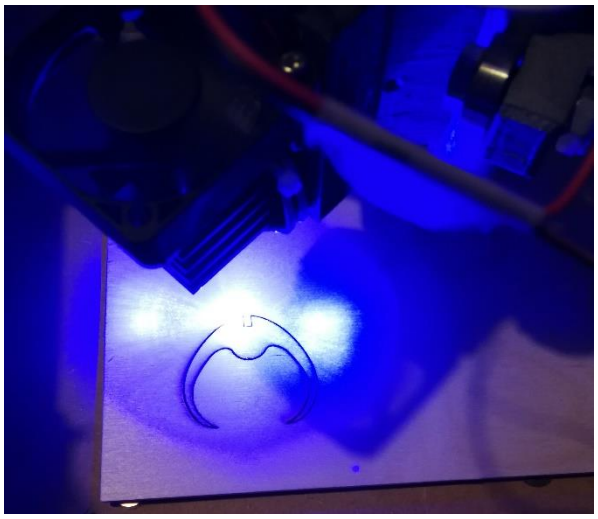
Please note that I like the surface discolouration of the wood, makes it more aged and bone like, but to prevent this I found blue painters tape stuck to the work surface removes the surface discolouration.

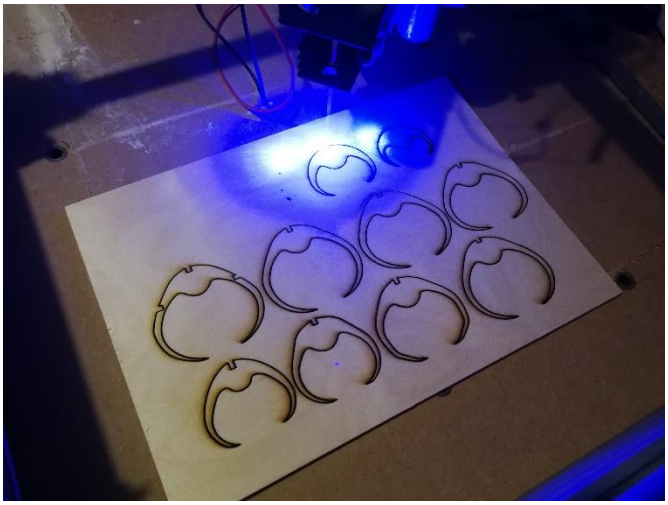
Some pictures of the cutting process.

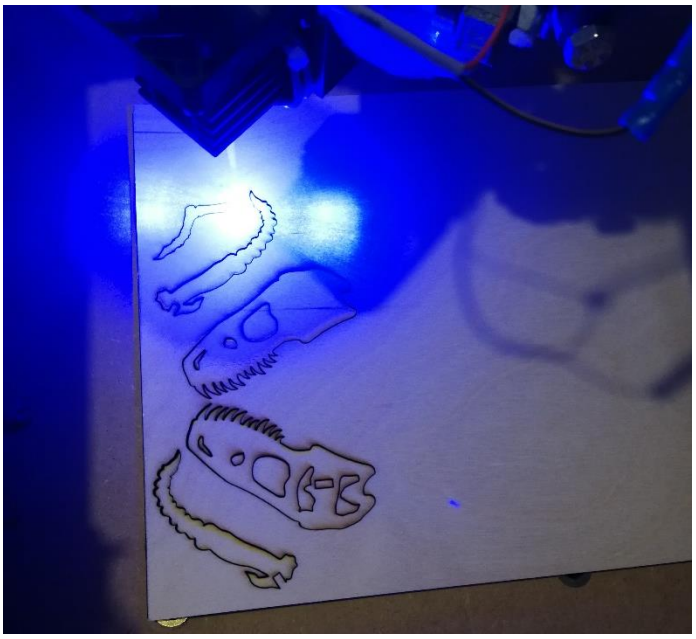


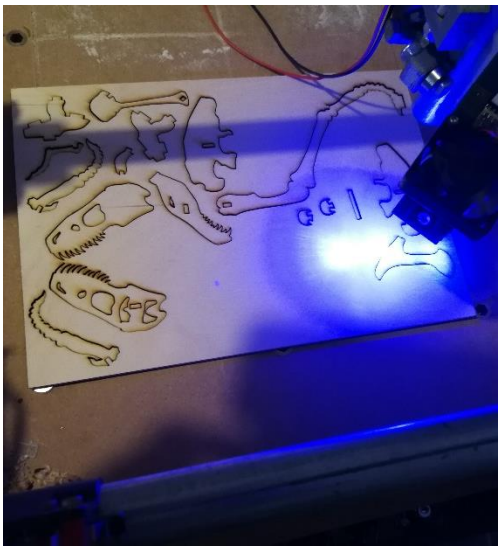












I ended up combining my last 2 rectangles into 1 to save on wood.

I dedicated my base to Endurance Lasers