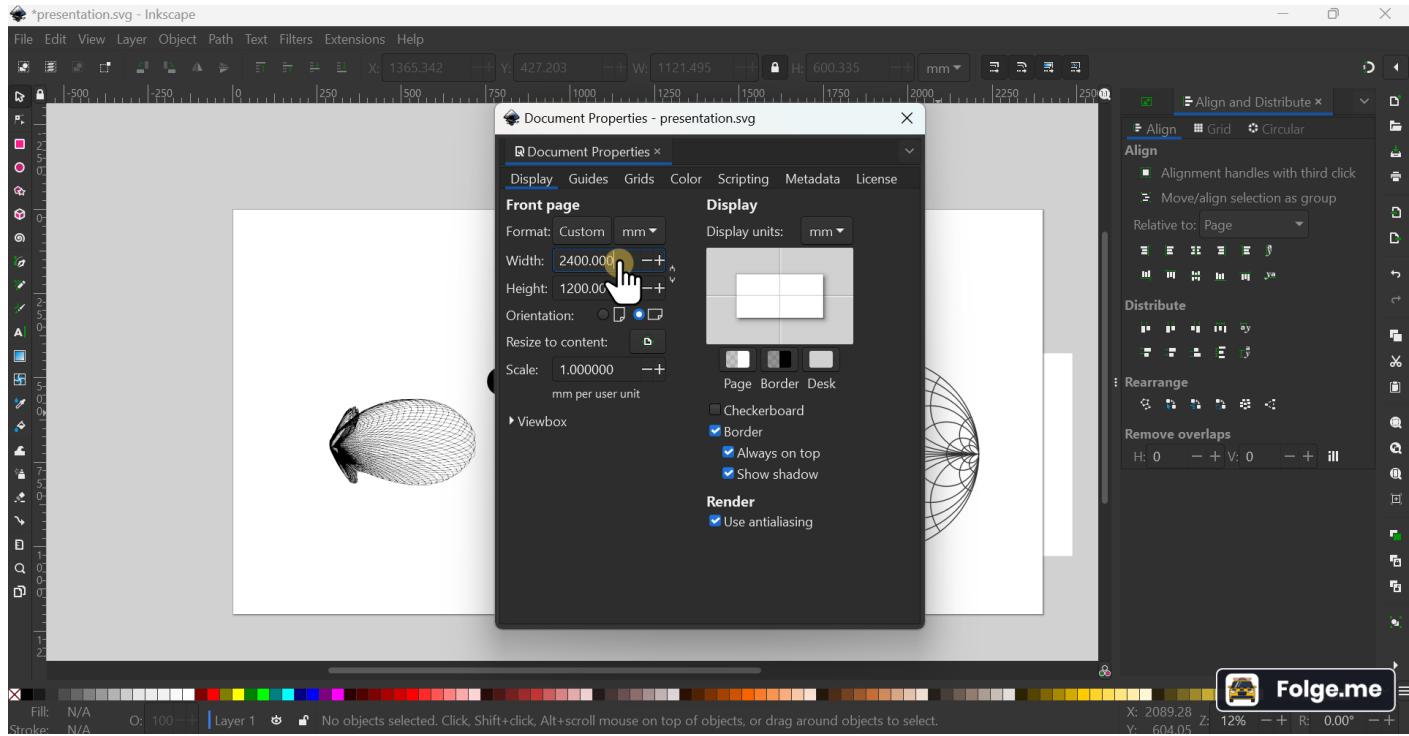
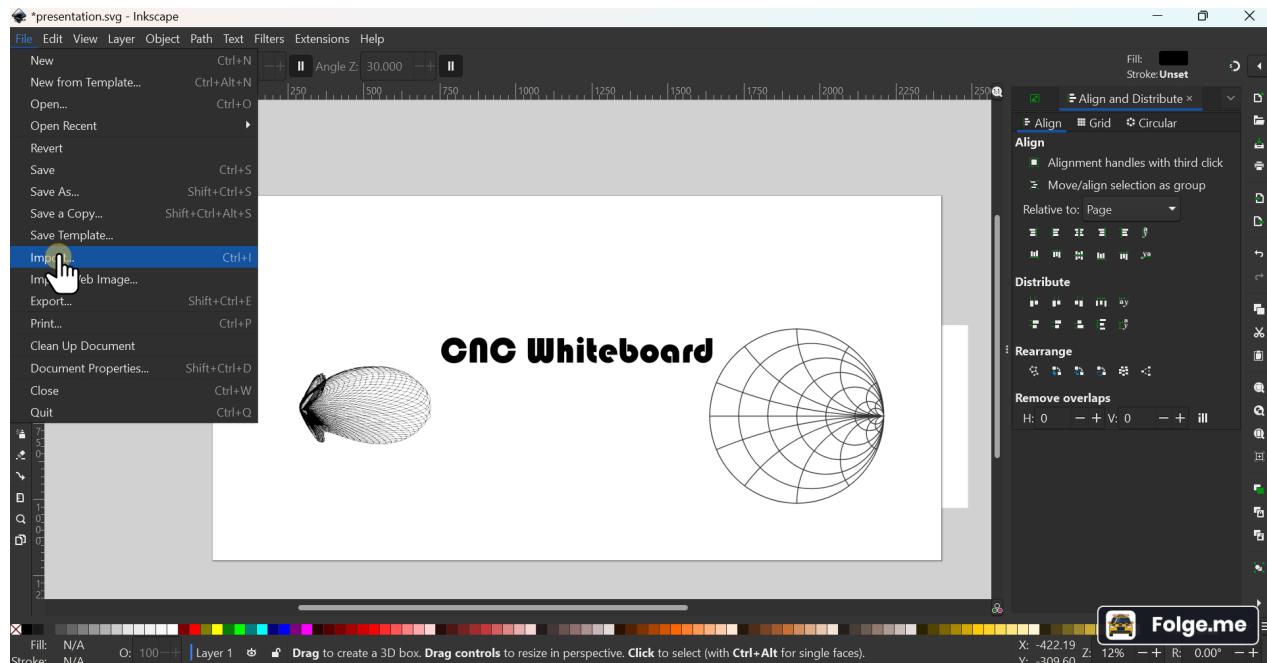


Convert images to Gcode using Inkscape

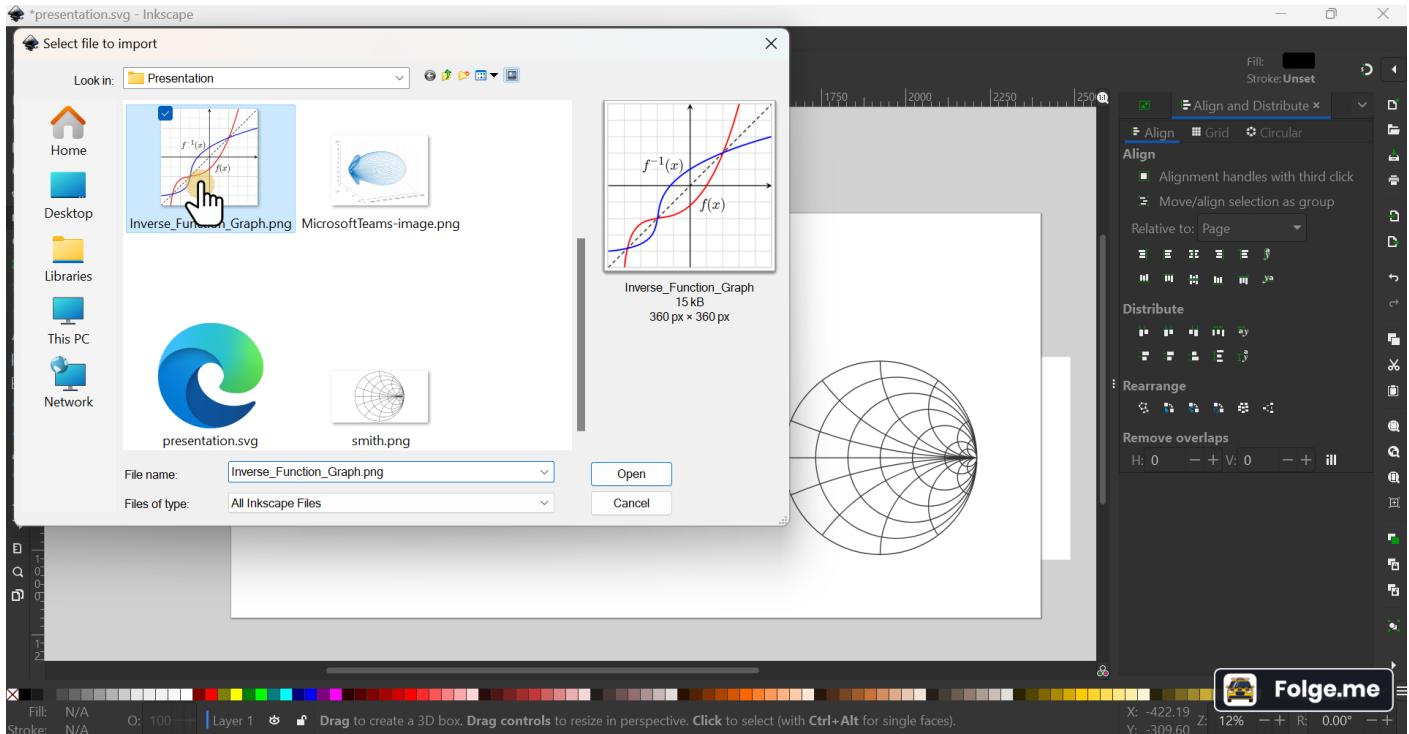
1 Ctrl+Shift+D: Set correct page size corresponding to board size



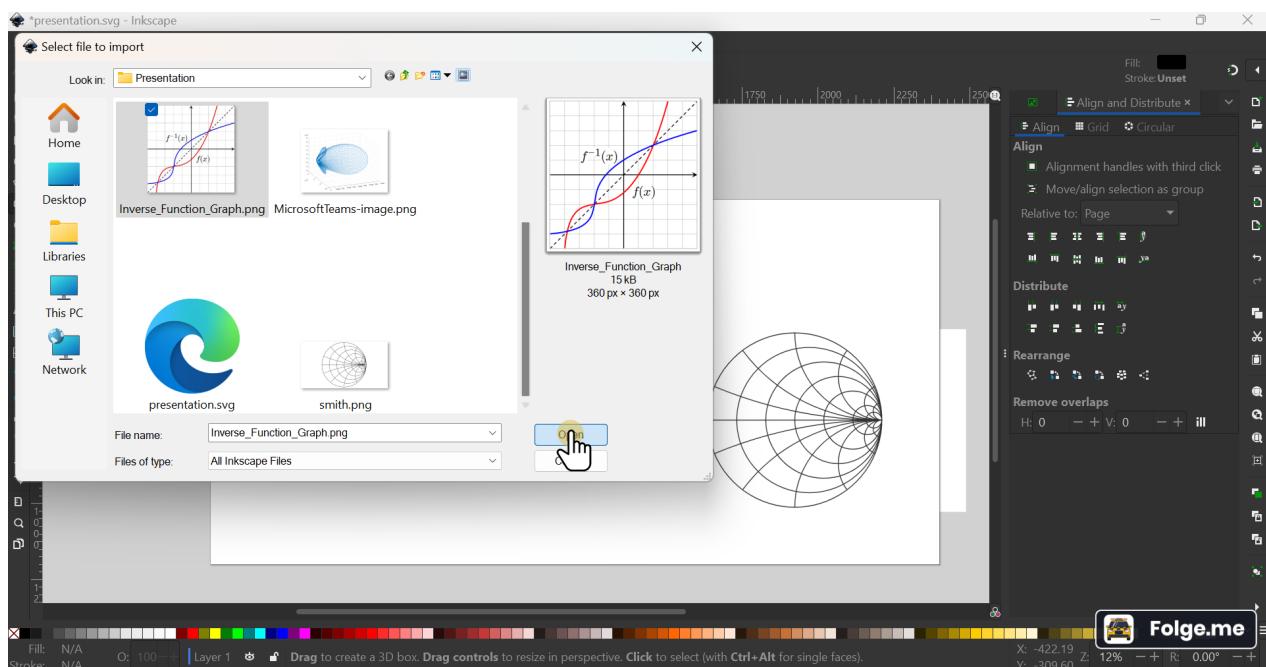
2 Import the image you want



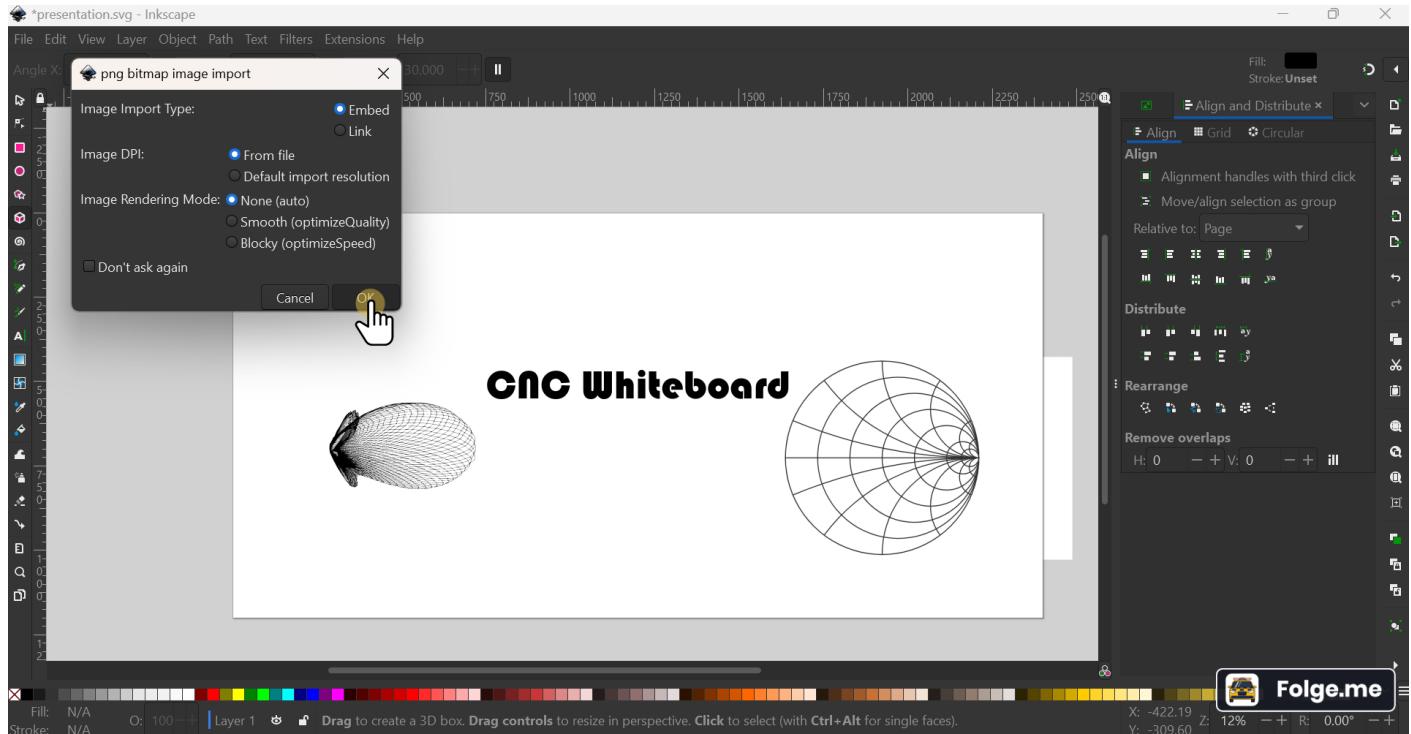
3



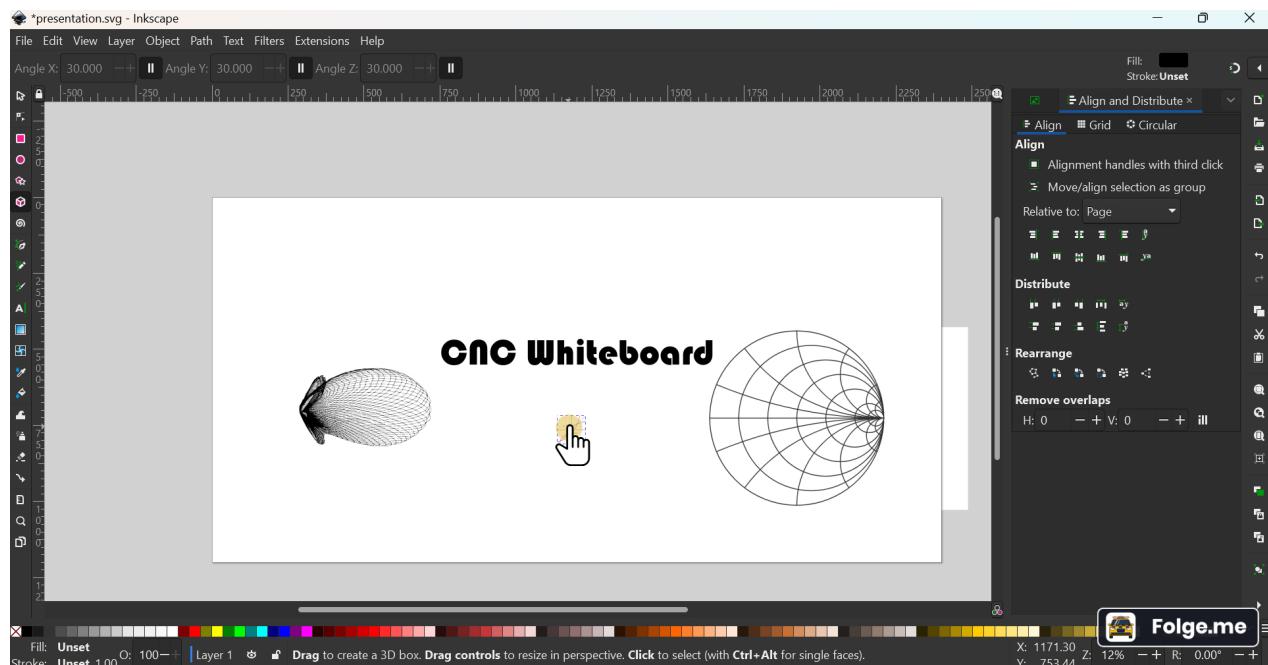
4



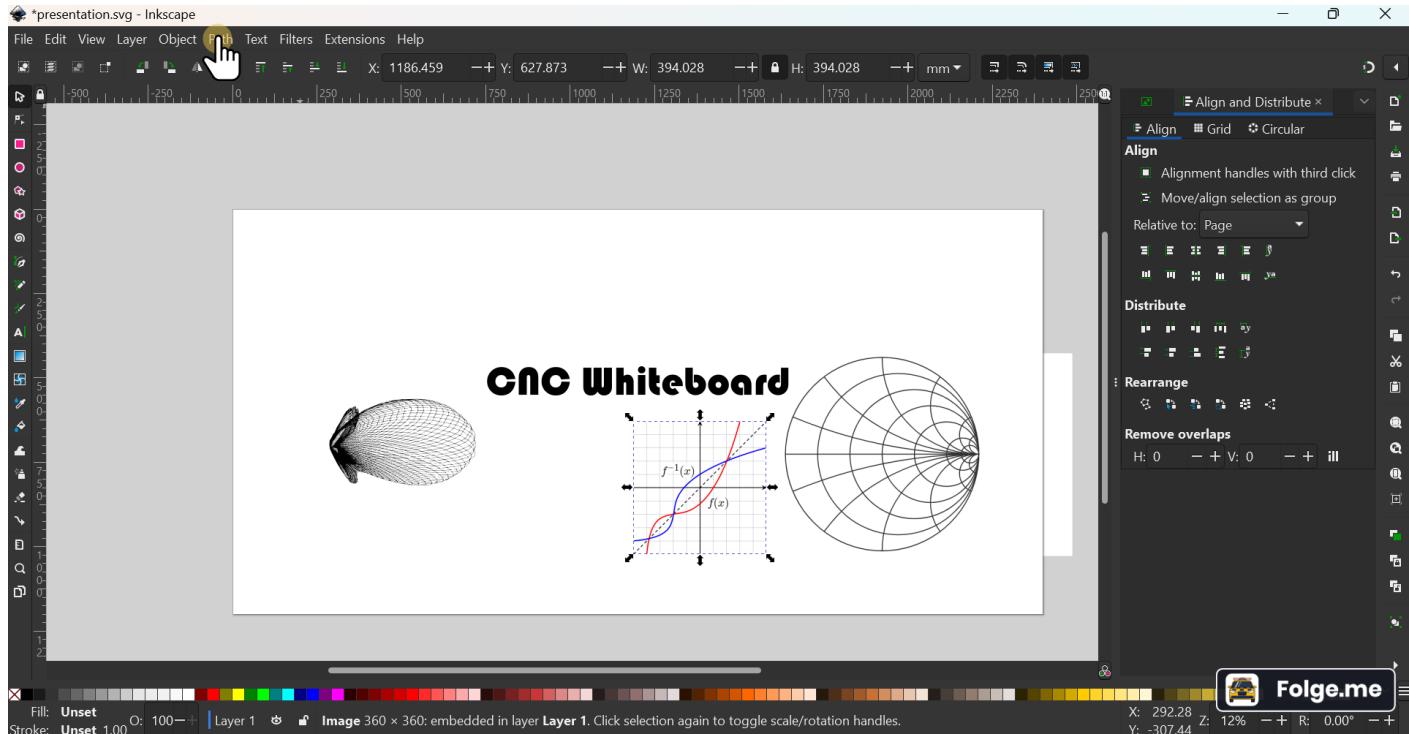
5



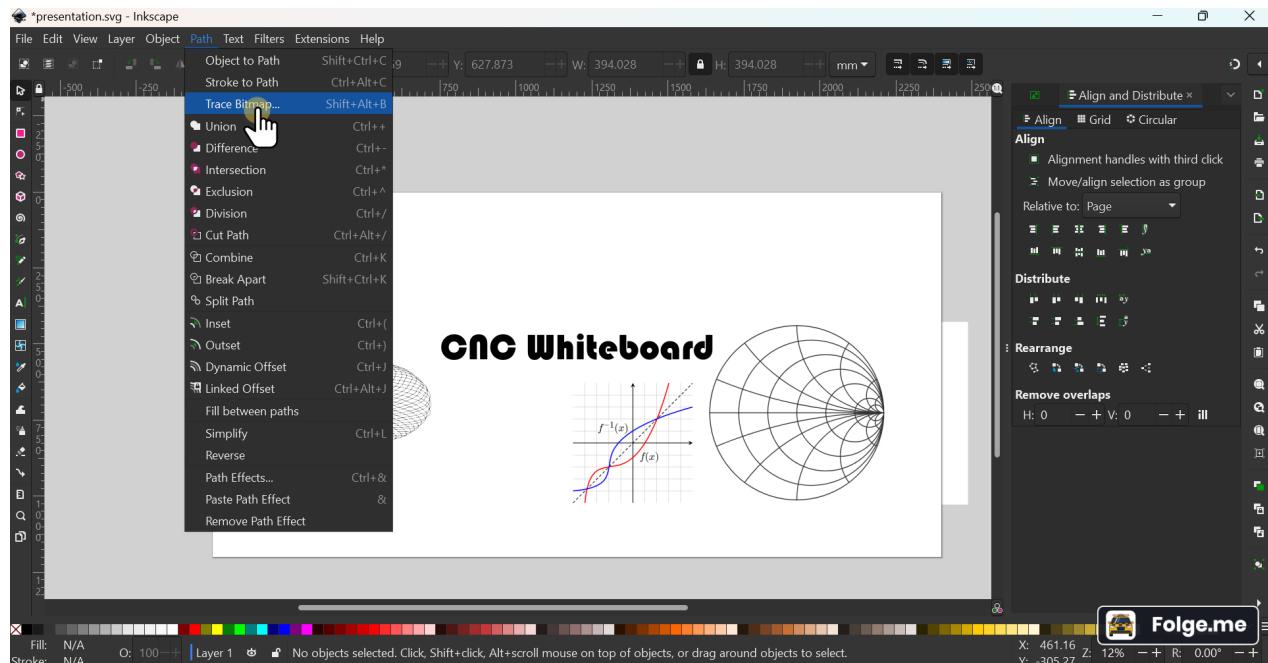
6



7 Resize and move to correct spot

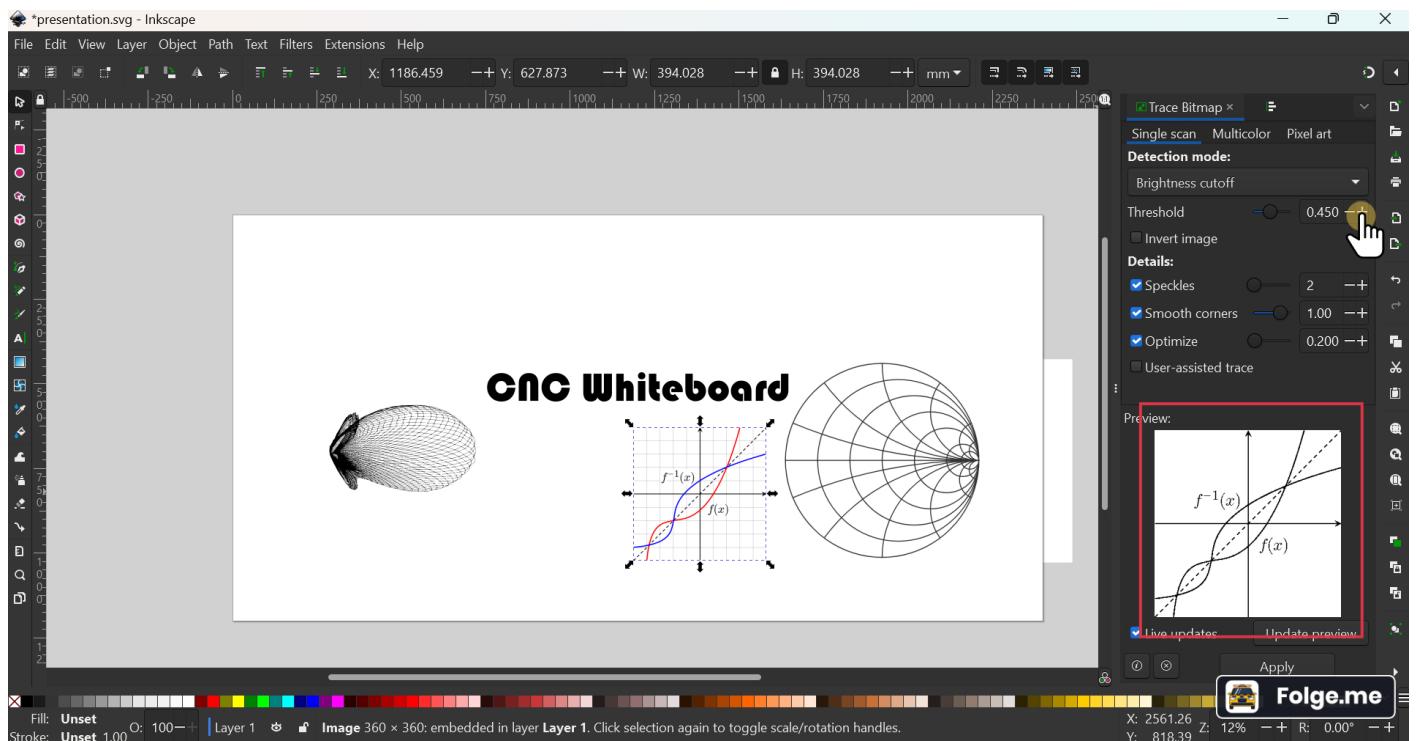


8 Path -> Trace Bitmap

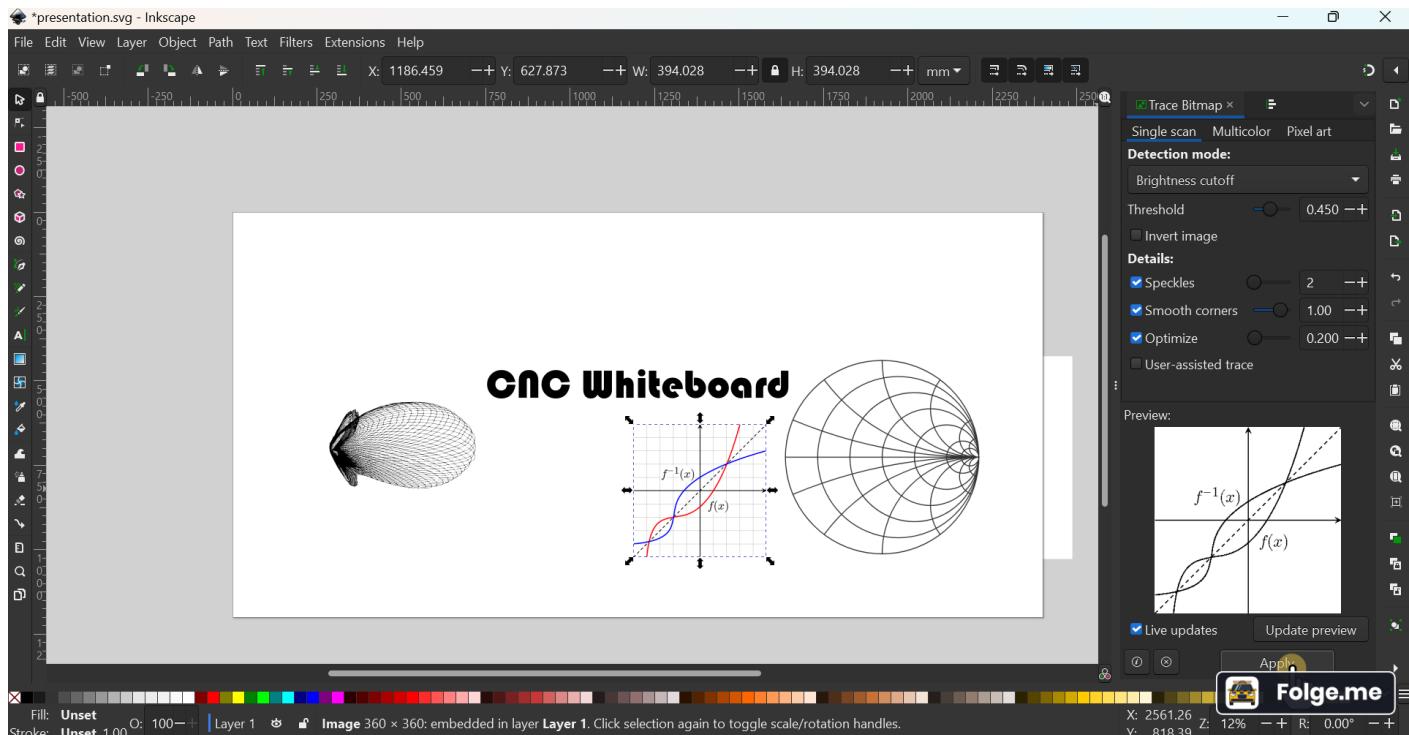


9 Select threshold to get preview you want

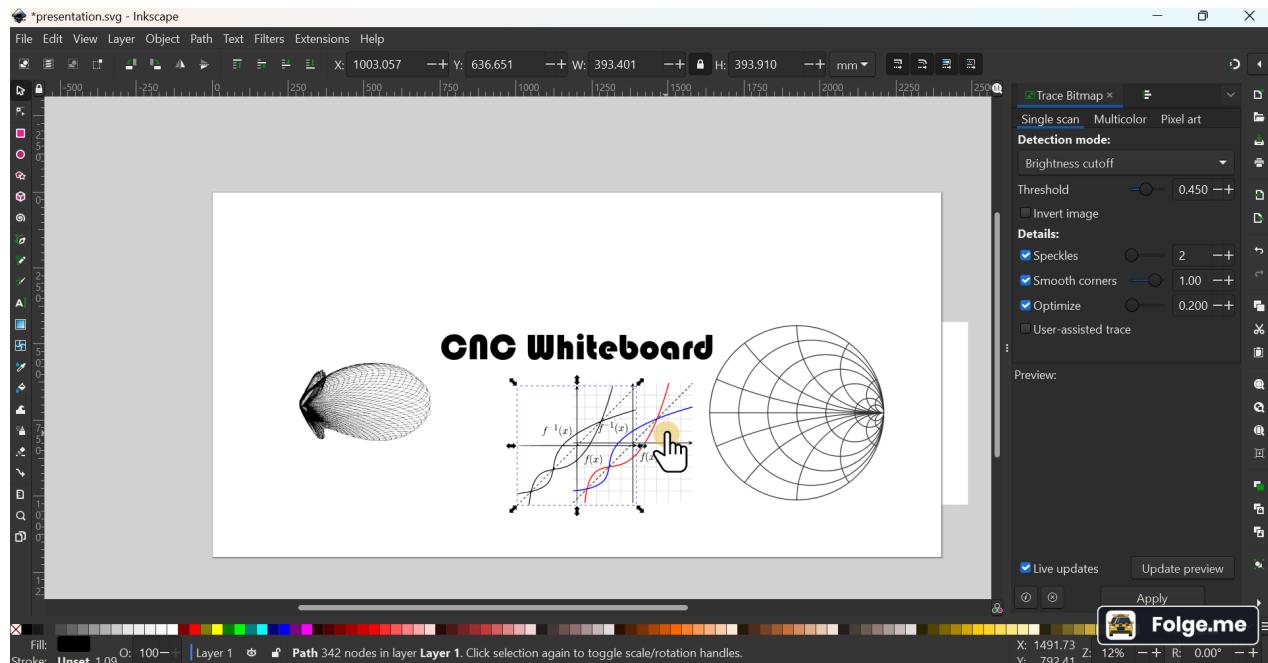
Title
Description



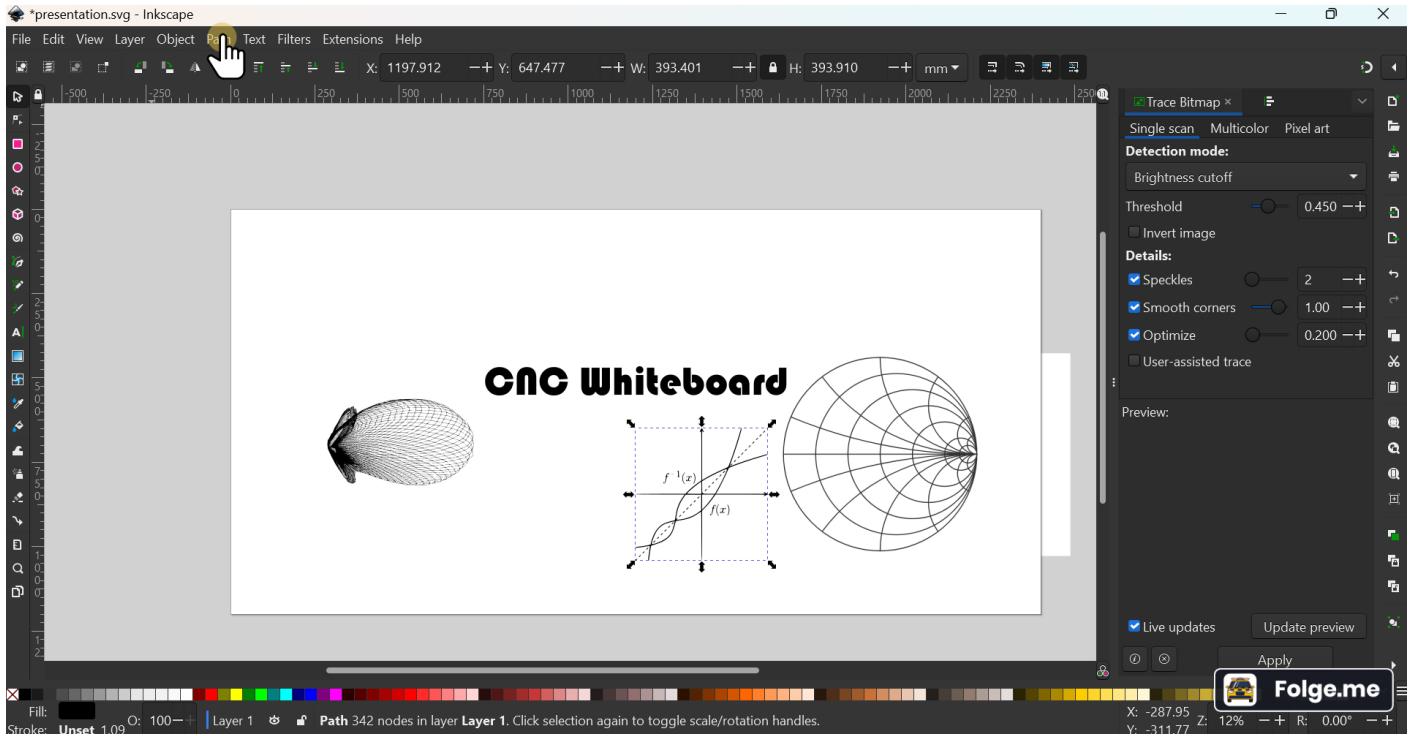
10 Press apply



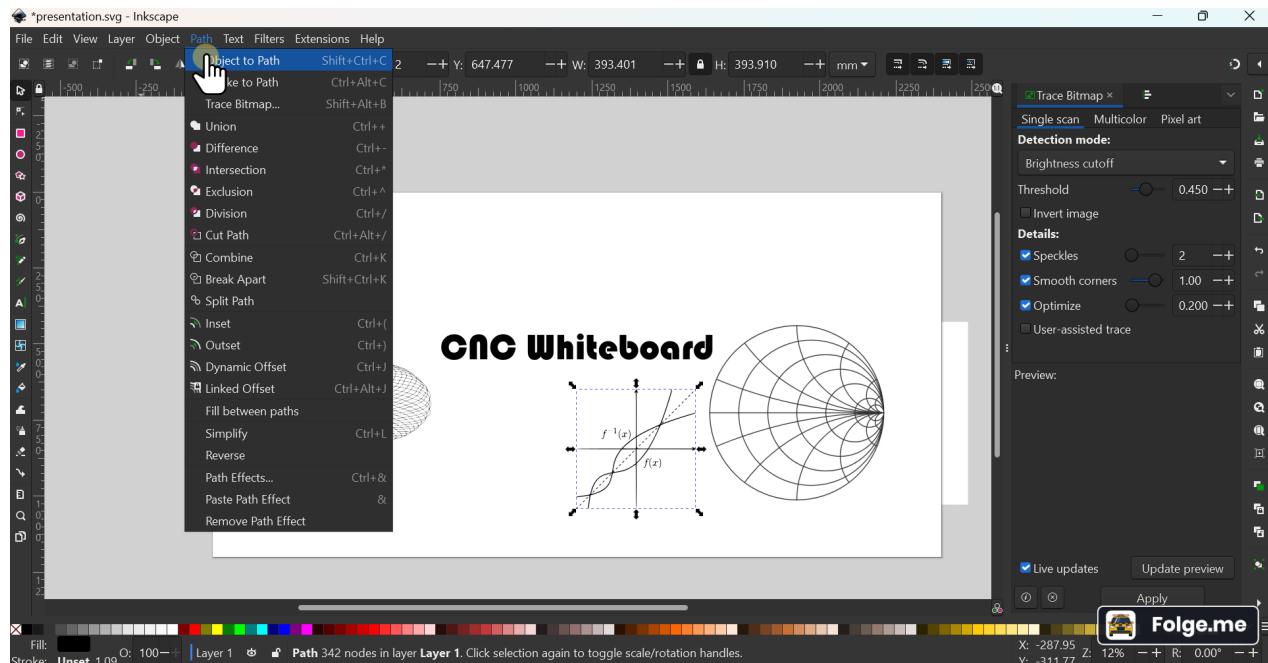
11 Move bitmap slightly and delete OG image



12

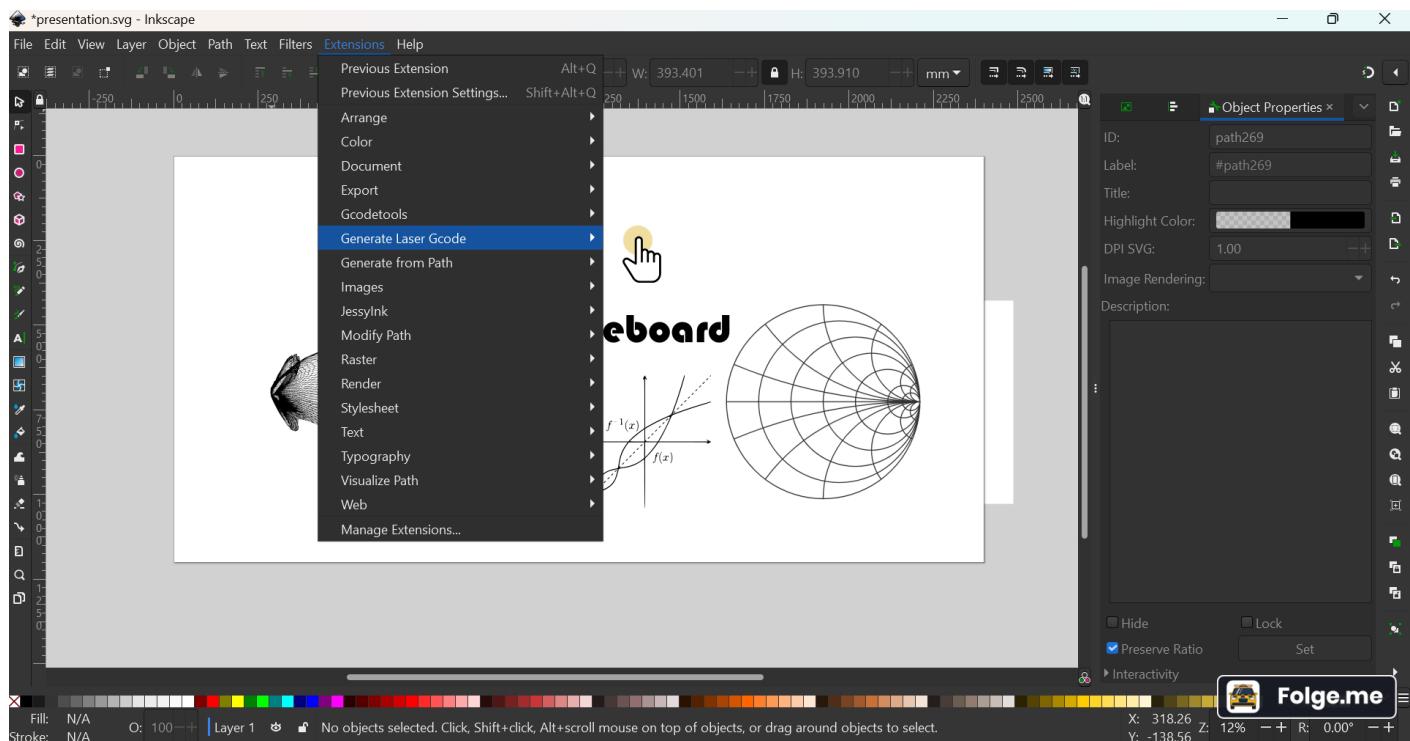


13 Path -> Object to Path



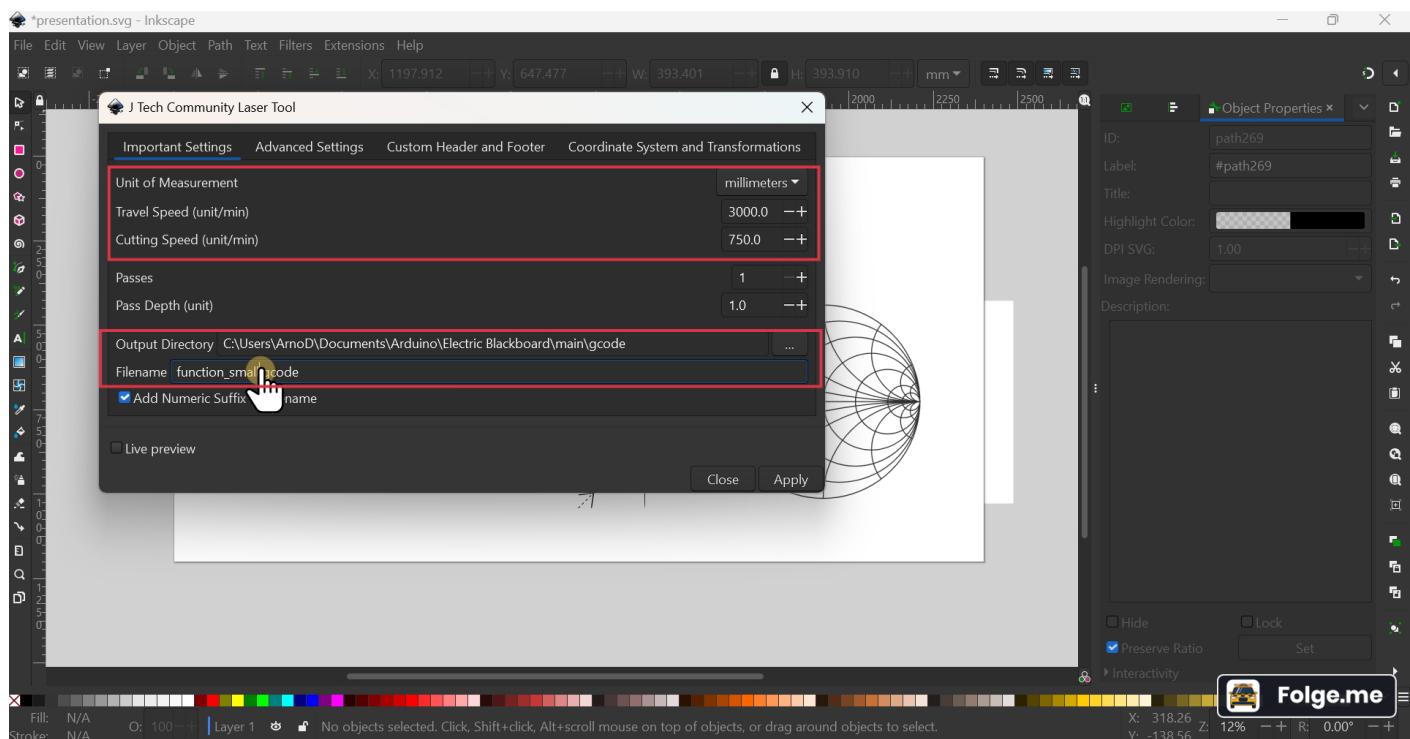
14 Once all images are a path

Repeat for all desired images. Once all finished, go to the newly installed extension: J Tech Community Laser Tool



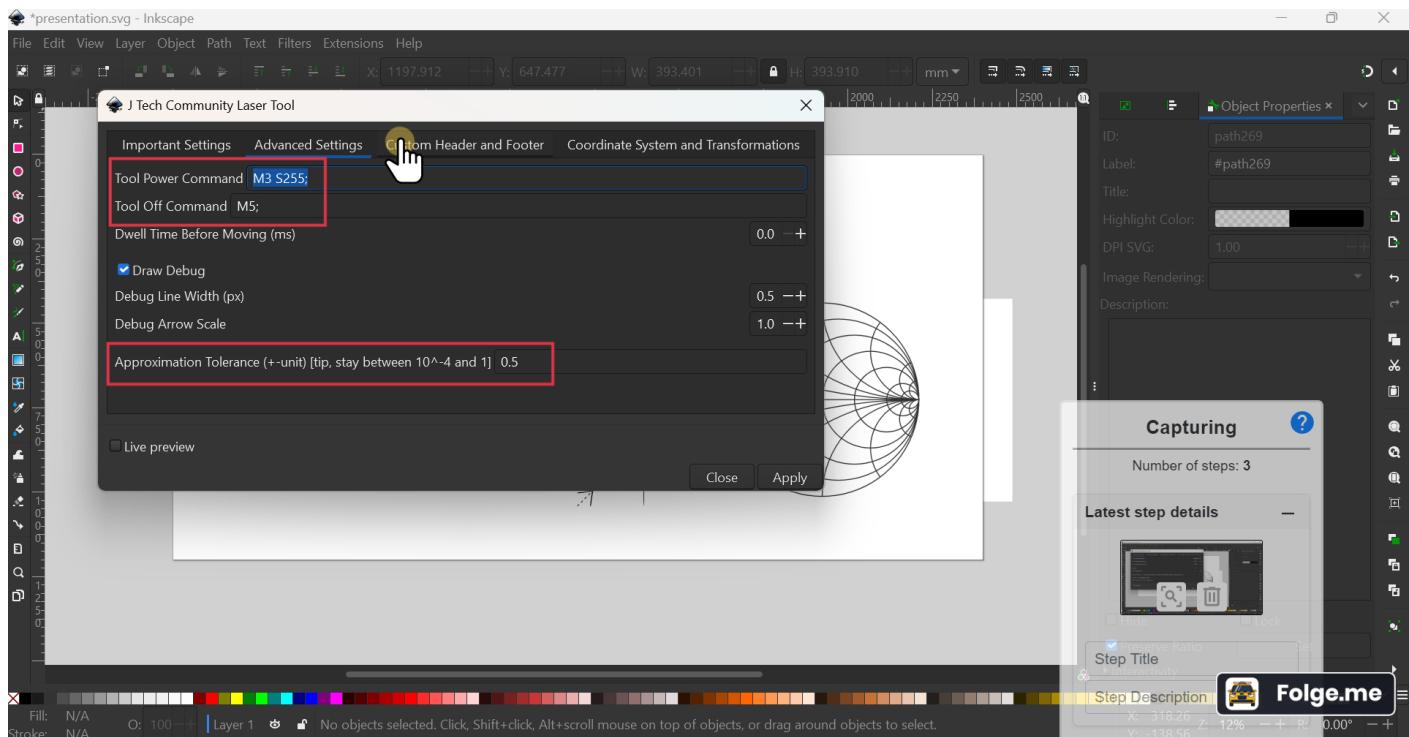
15 Important settings

This extension is made for lasercutting, so many settings aren't important. Travel speed and cutting speed determine the speeds in steps/s of the writing/non-writing movement. Choose the desired file name and location (for the Gcode).



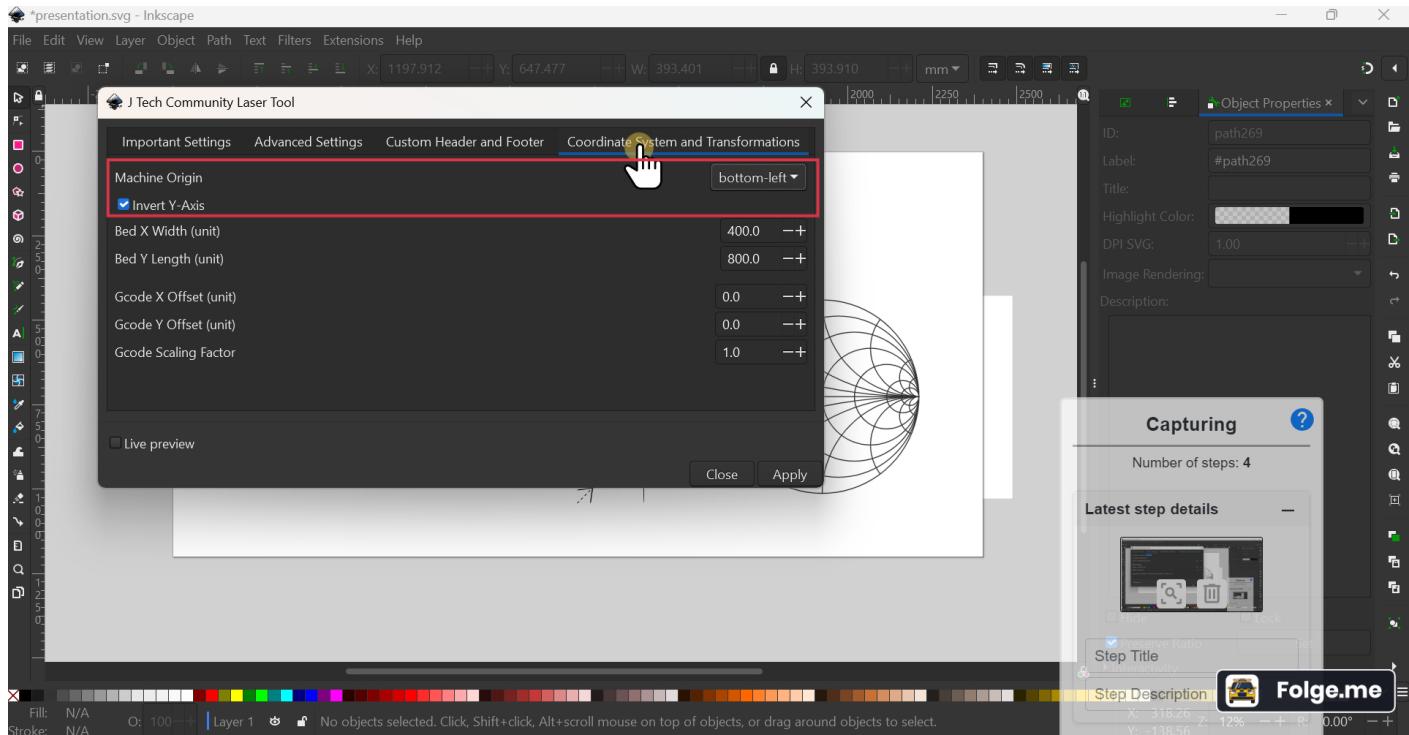
16 Advanced settings

Important to name the tool power command M3 S255; and the tool off command M5; The approximation tolerance is to decide the 'resolution' of the lines that approximate the figure. The smaller this value, the more detail and the longer it will take to draw.



17 Coordinate system and transformations

Machine origin: bottom left. Invert Yaxis



18 Press apply

You're done!

