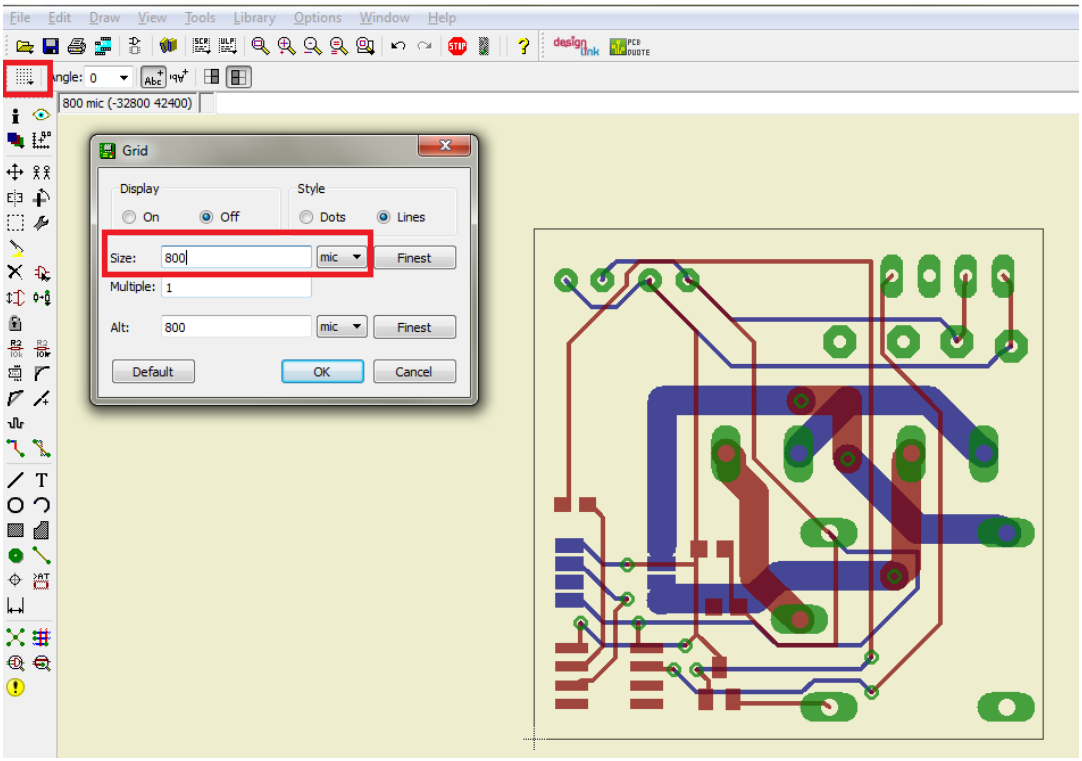
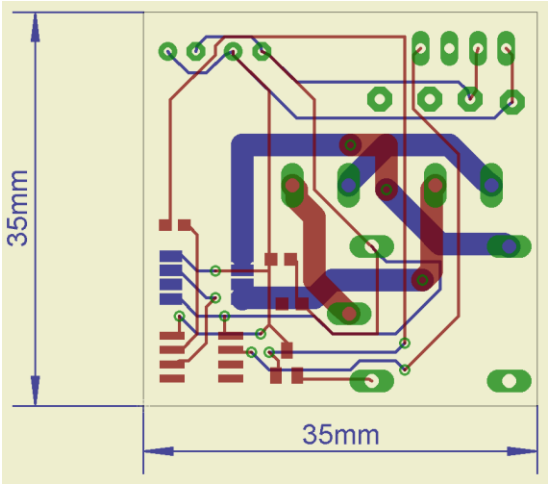
OpenCdNC - How to do PCB

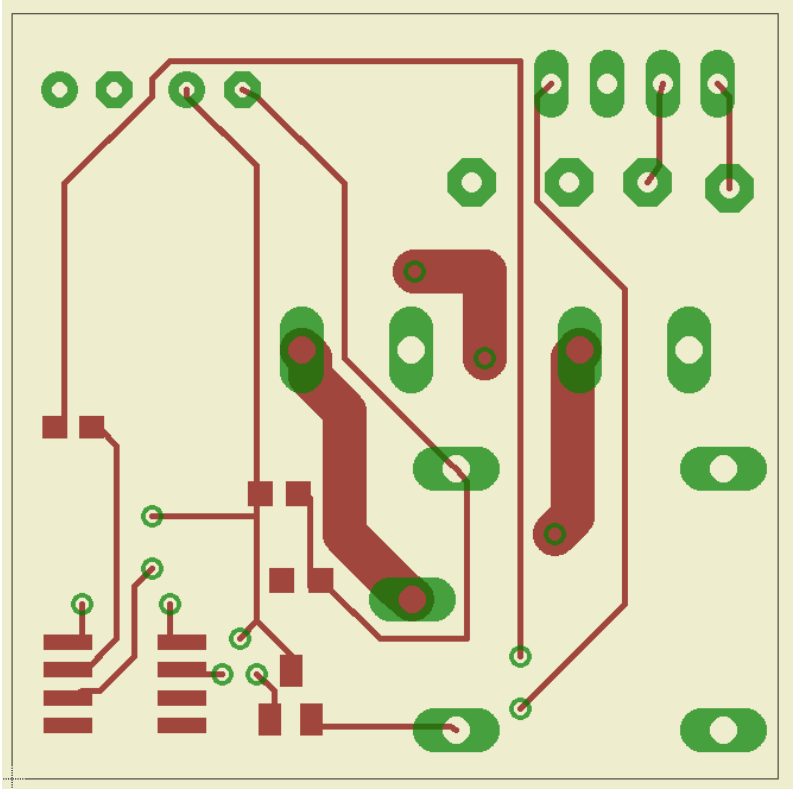
Open Eagle PCB, load your project, place the components inside a rectangle with your CNC's work area size, set grid to 'mic' unit and size to something above your CNC's precision and route it:



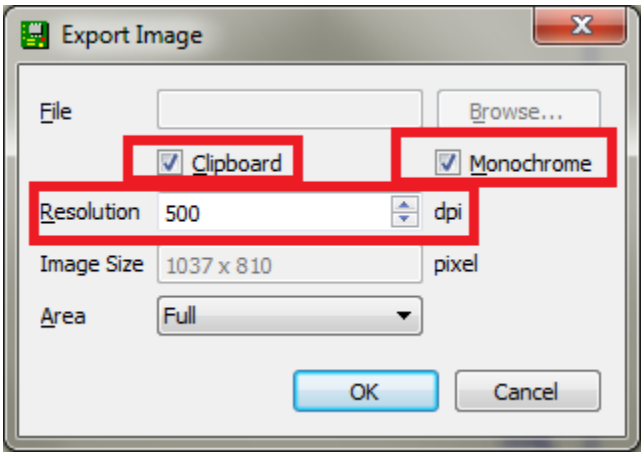
Remember, my CNC's work area is 35x35mm (do not exceed the work area of your CNC!):



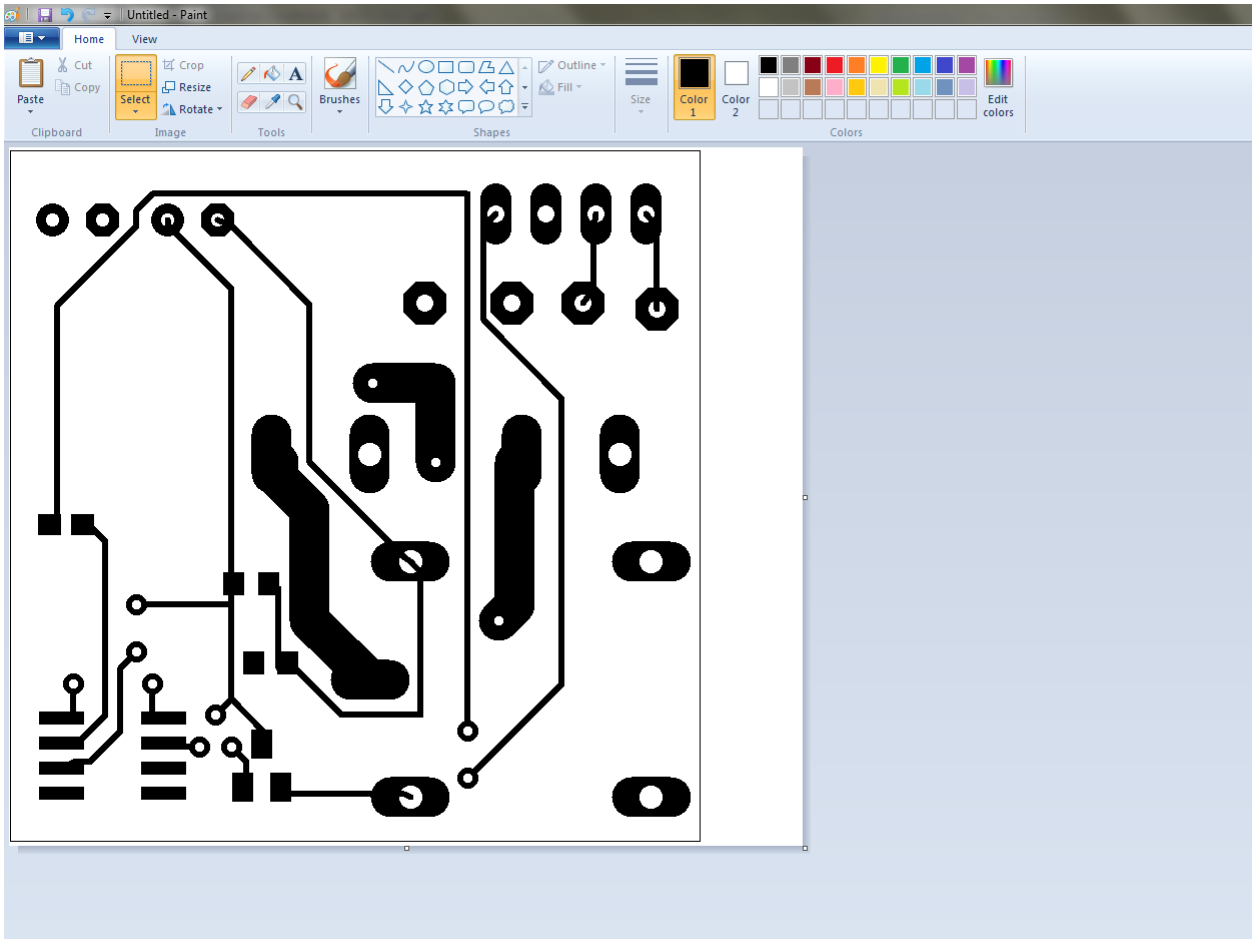
Let's print just vias, pads, dimension and top layer (choose from layer option):



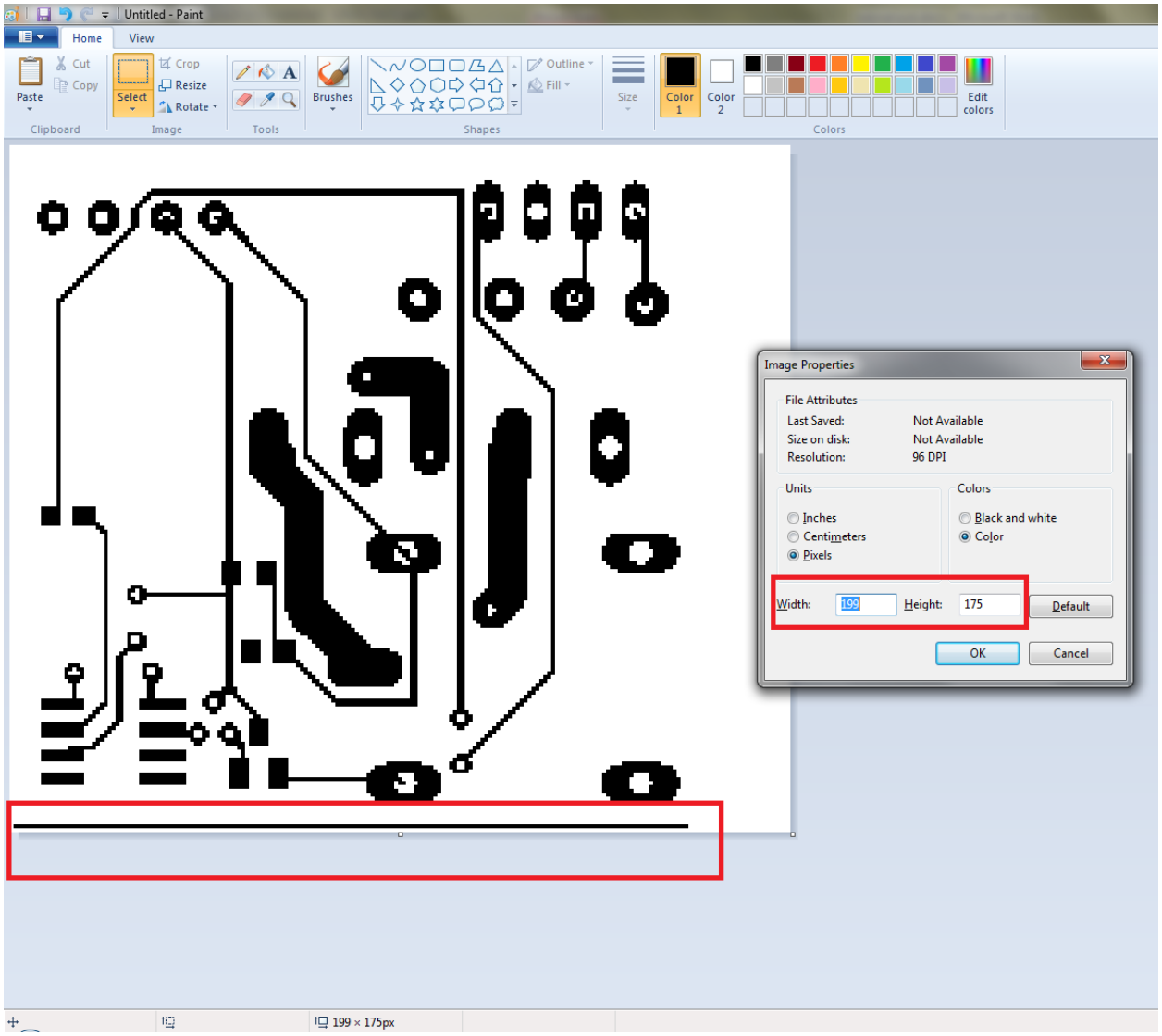
Now go to 'file'->'export'->'image', increase the resolution and set these checkbox:



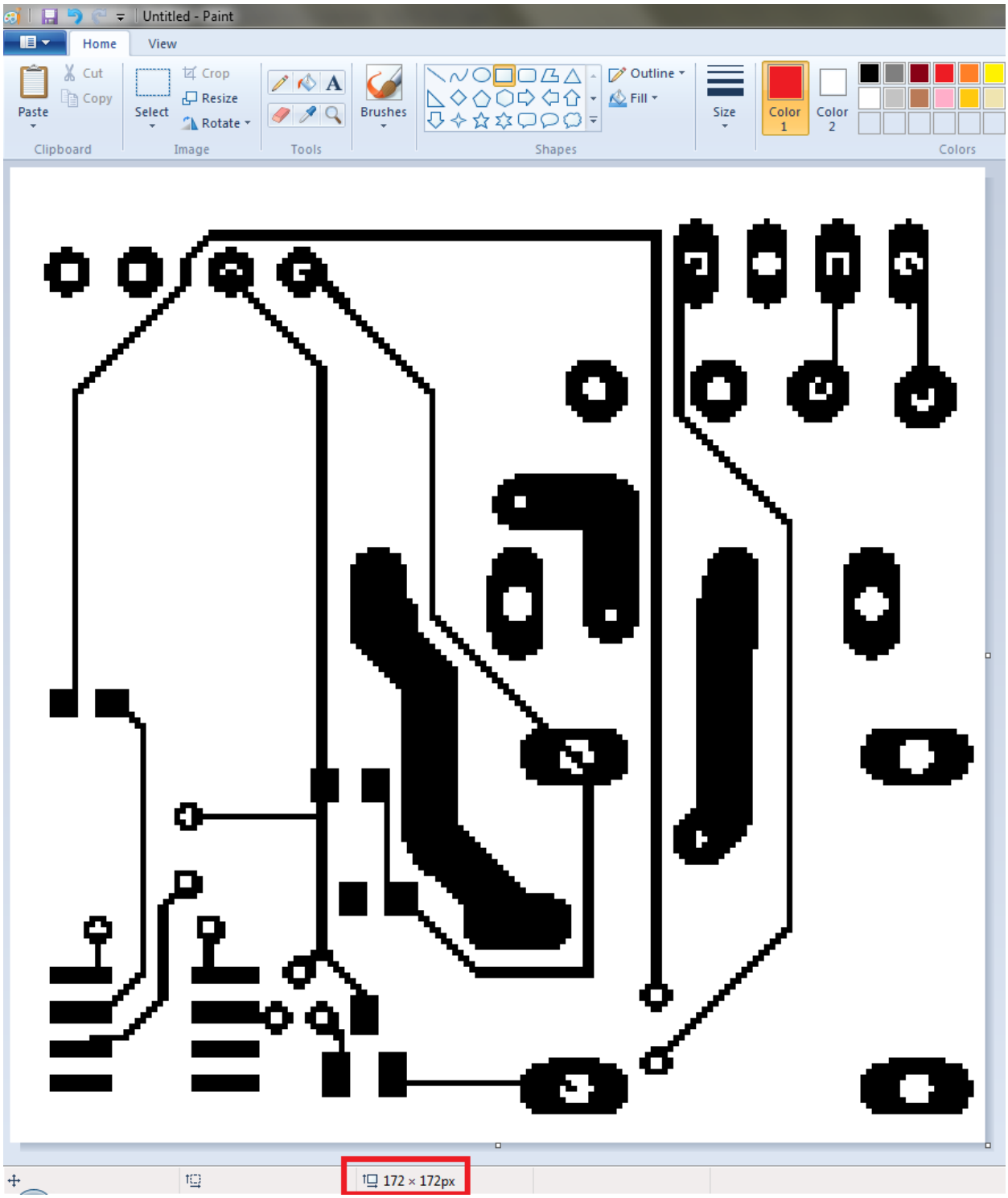
Now click 'ok', open MS Paint and paste:



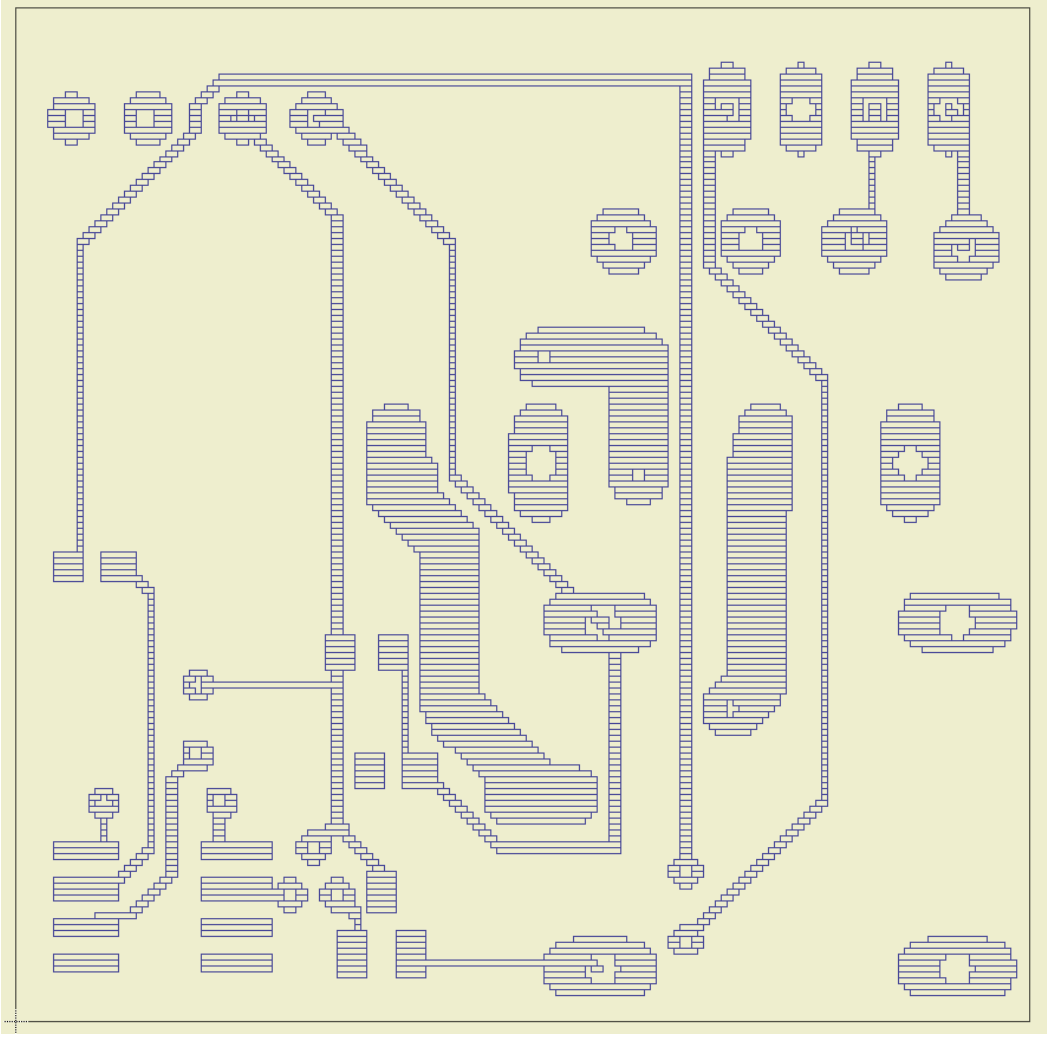
At this point we lost the reference of how many pixels is X mm, this is why we keep the work area rectangle around the board. As you remember from the 'how to print images' tutorial, we need reduce the image size to something lower than 300x300 pixels:



As we can see, the 35 mm horizontal reference still there, the image resolution was reduced to 199x175 pixels. Now resize the picture to fit exactly the horizontal/vertical reference dimension:



Now we have a 172x172 pixels image, just save as monochromatic bitmap, follow the 'How to print images' tutorial and you will have your board:



Due the CNC's precision, maybe you will have trouble at the distance between tracks, with experience and time you will know better your CNC's limitations and how to do a proper routing. Of course a really thin marker/pen and a solid CNC construction help a lot. You can use a laser module with a photoresist film too.

Another way to do PCB is using directly the 'wire' tool, then you can draw above the image layer (with horizontal and vertical lines manually), this will result in a faster and better print, because the pen/marker/laser will follow your wire just once, as a path, instead of draw the whole board with 'small rectangles'. After you print, just etch your board as usual!

https://github.com/themrleon/OpenCdNC