

FAB LAB TRAINING SESSION

PCB Designing

Aim:

To design a PCB which demonstrates the working of Astable Multivibrator using IC555 Timer.

Objectives:

- To implement a working astable multivibrator using IC555 Timer Circuit using PCB Designing.
- Main objective is to learn how PCBs are designed and produced from scratch and how circuits are implemented.
- To demonstrate students with the software required for PCB designing: EAGLE by AutoDesk and FLATCAM.
- Giving a tutorial to students how the PCB will be carved using the CNC Carving Machine.
- To implement the soldering procedure required for constructing the circuit.

Software requirements:

EAGLE by Autodesk and Flatcam should be installed on the students' laptop/PC.

Hardware requirements:

- 2 LEDs of different colour
- 4 Resistors of 1k Ohm each
- IC555 Timer IC
- Capacitor
- 9V Batteries
- Connectors

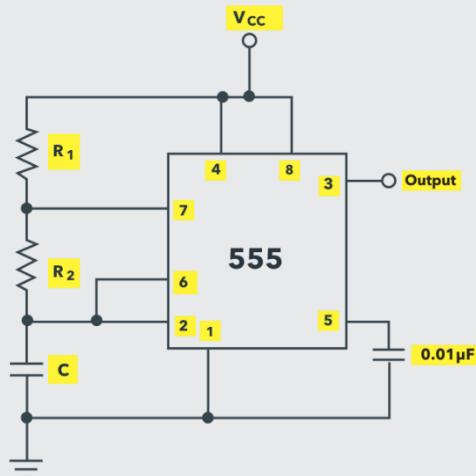
Apart from these we need Copper boards where we can engrave our PCB Design and CNC Carving Machine.

Theory:

IC 555 Timer:

Astable multivibrator is also called as Free Running Multivibrator. It has no stable states and continuously switches between the two states without application of any external trigger. The IC 555 can be made to work as an astable multivibrator with the addition of three external components: two resistors (R1 and R2) and a capacitor (C). The schematic of the IC 555 as an astable multivibrator along with the three external components is shown below.

Astable Multivibrator Using 555 Timer IC



Electronics Hub

The pins 2 and 6 are connected and hence there is no need for an external trigger pulse. It will self trigger and act as a free running multivibrator (oscillator). The rest of the connections are as follows: pin 8 is connected to supply voltage (VCC). Pin 3 is the output terminal and hence the output is available at this pin. Pin 4 is the external reset pin. A momentary low on this pin will reset the timer. Hence, when not in use, pin 4 is usually tied to VCC.

The control voltage applied at pin 5 will change the threshold voltage level. But for normal use, pin 5 is connected to ground via a capacitor (usually 0.01 μF), so the external noise from the terminal is filtered out. Pin 1 is ground terminal. The timing circuit that determines the width of the output pulse is made up of R₁, R₂ and C.

PCB

PCB or Printed Circuit Board is the traditional name for the bare board of which you supply us with the layout data and which you use to mount your components on once we have delivered it to you.

A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces etched from copper sheets laminated onto a non-conductive substrate.

When the board has only copper tracks and features, and no circuit elements such as capacitors, resistors or active devices have been manufactured into the actual substrate of the board, it is more correctly referred to as printed wiring board (PWB) or etched wiring board.

Applications of Astable Multivibrator:

- Square Wave Generation
- Pulse Position Modulation

- Pulse Train
- Frequency Modulation using Astable Multivibrator

Pre-Requisites:

The prior knowledge required for designing is knowledge of schematic symbols of electronic components and the basic understanding of how connections of wire are made. Awareness about the co-ordinate system in the CNC machine is also needed.

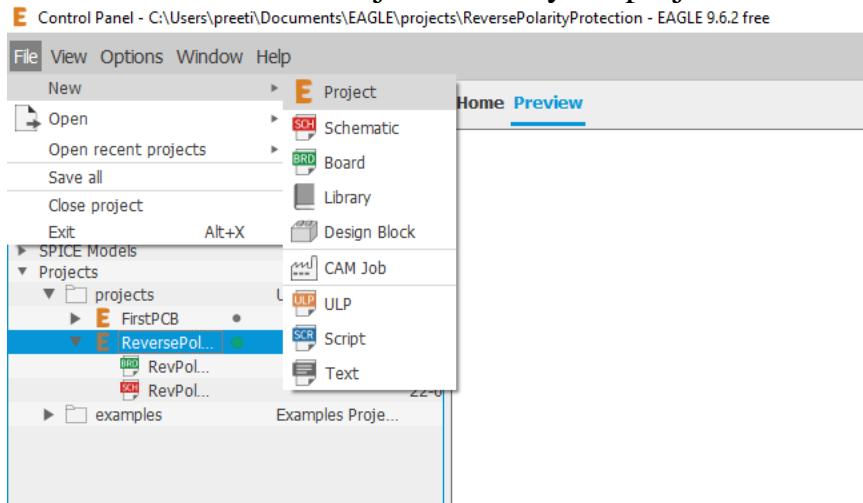
PRECAUTIONS:

While designing the PCB on software and also on hardware certain precautions are to be ensured.

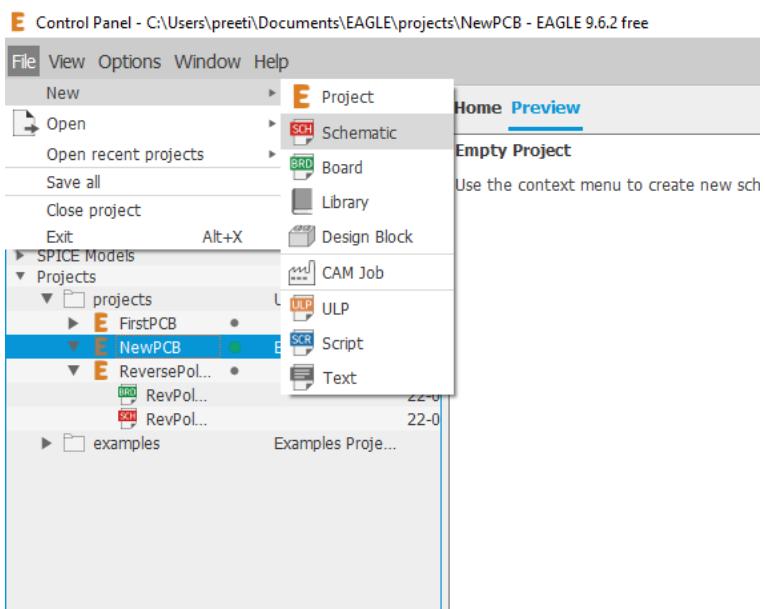
- All the wire connections made should not be placed too closely.
- The wires should never make a right angle in the design as it is not considered as ideal practice.
- All the components in the designed should be placed neatly so as to avoid any confusion.
- If a power/voltage source is depicted in the circuit, replace it with the connector component. Also, ignore the ground if it is shown in the circuit.
- No electronic component should touch the surface of PCB while placing them, it leads to heating up of the board due to Antenna Effect. It may damage our PCB.

Procedure:

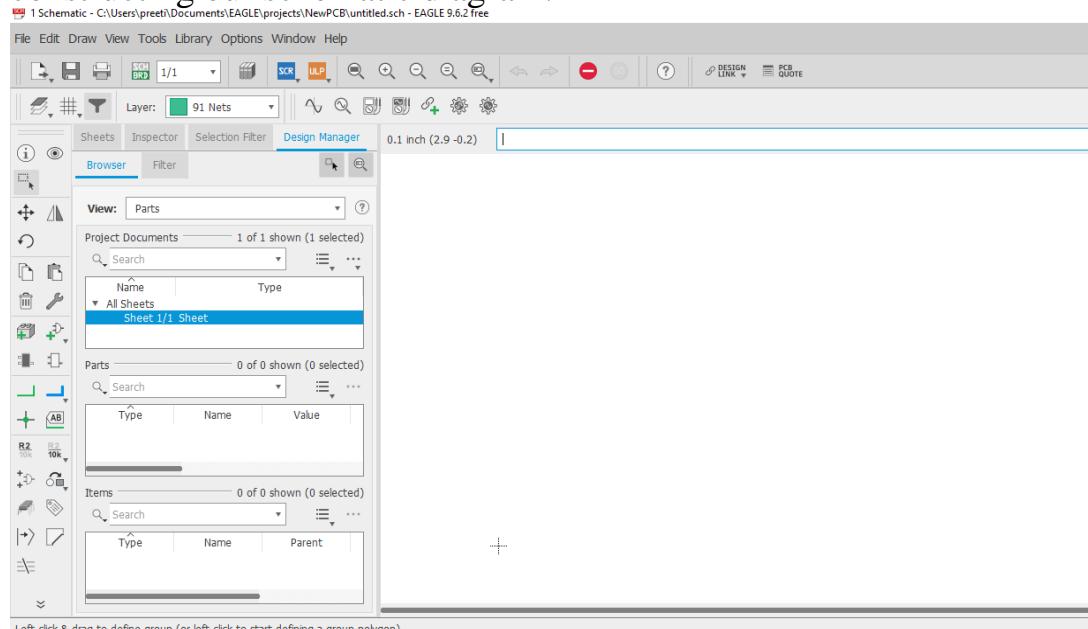
1. Open the EAGLE Software. Go to File in the top left corner and from the menu Select New and then Select Project. Name your project accordingly.



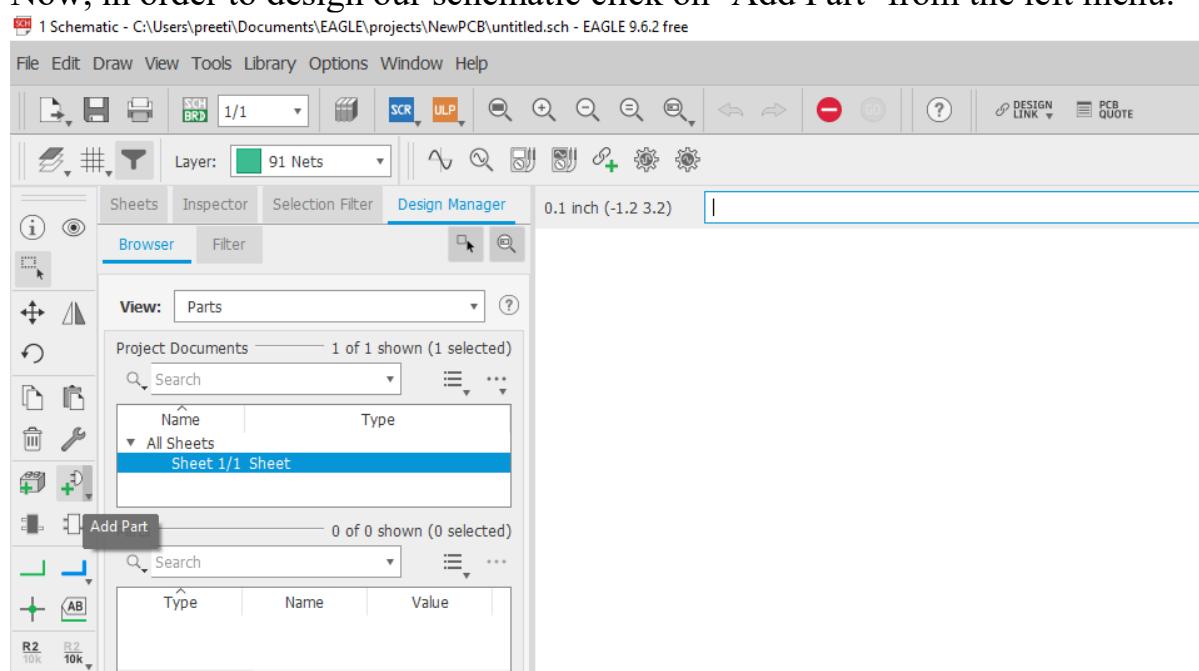
2. An empty Project is created. Again go to File and Select New. This time select Schematic from the menu and name it accordingly.



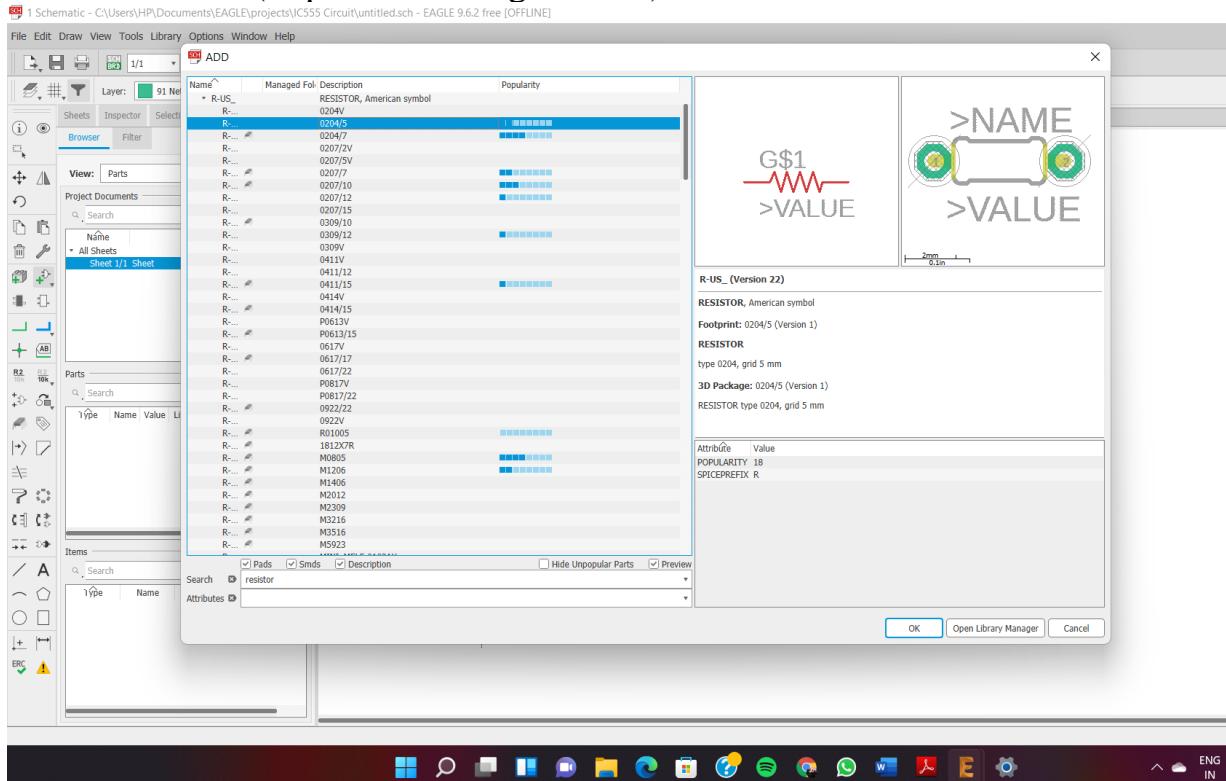
3. A new window like this appears. The blank area is the place where we will be constructing our schematic diagram.



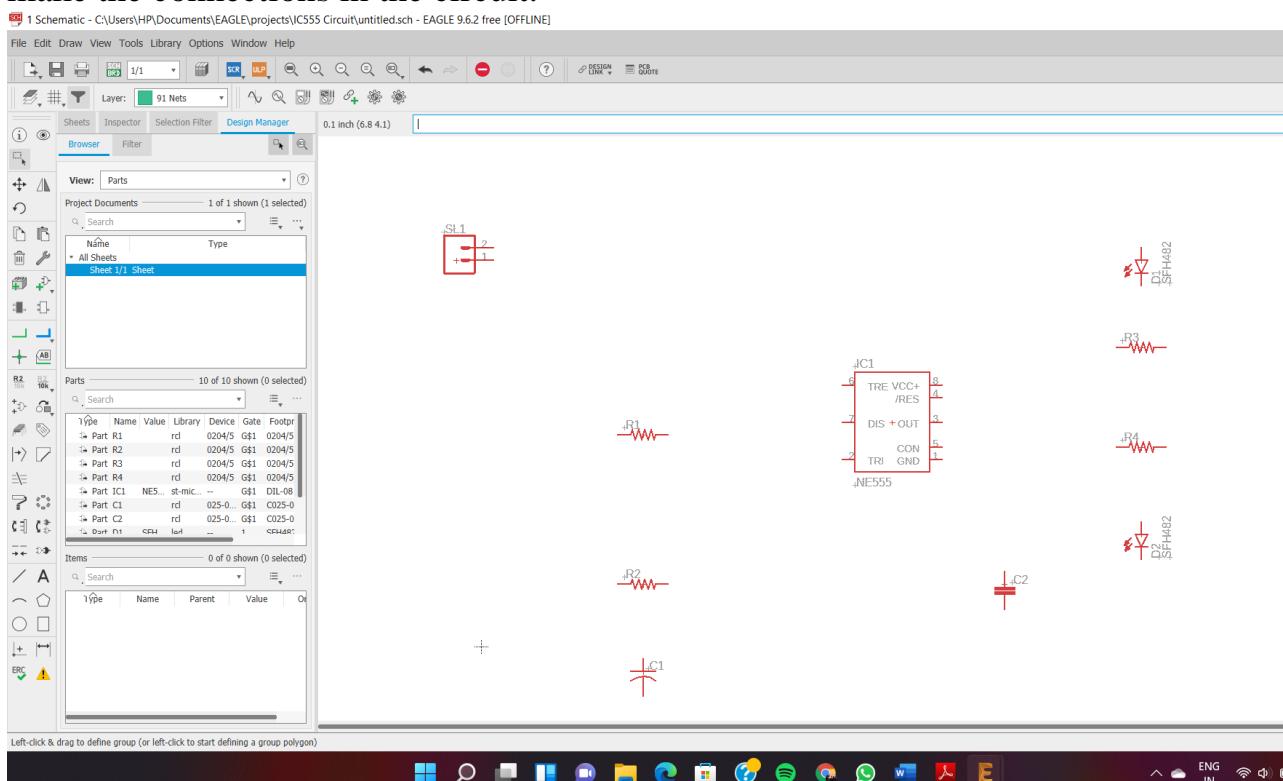
4. Now, in order to design our schematic click on 'Add Part' from the left menu.



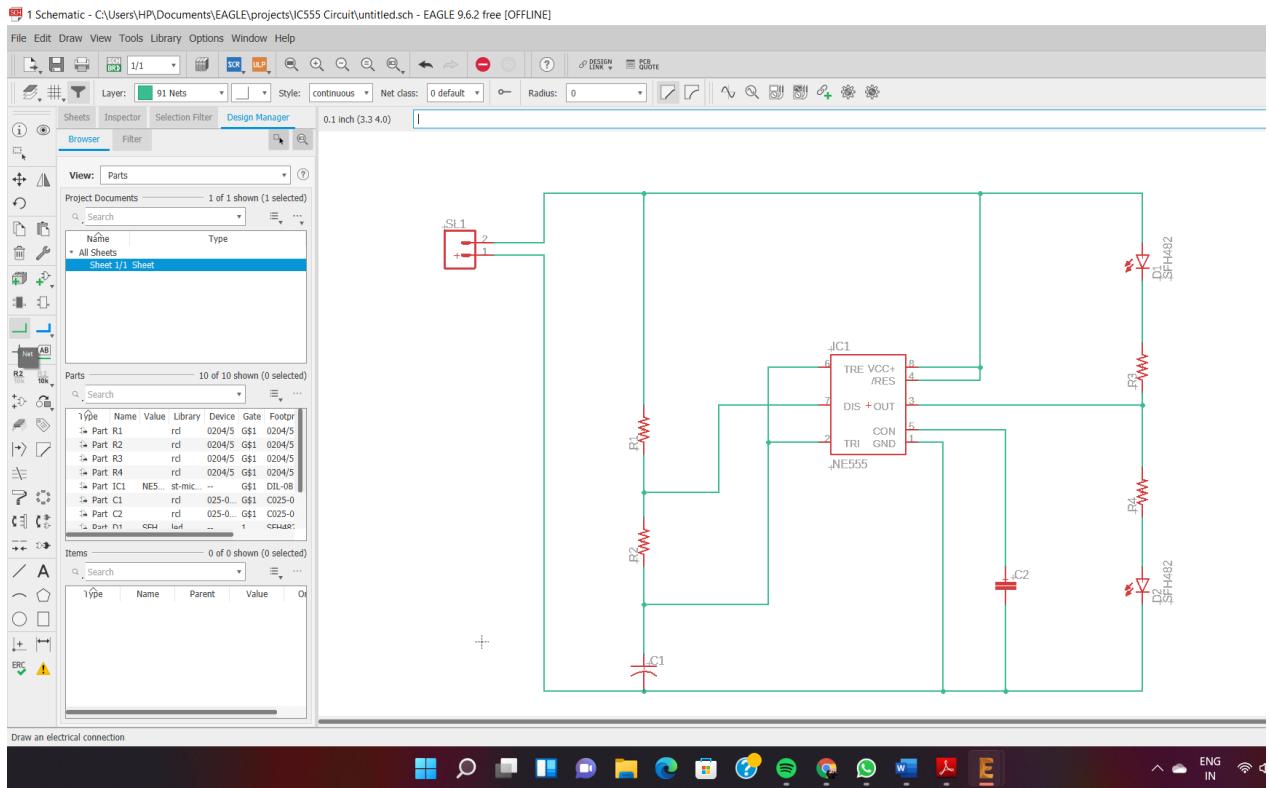
5. A new dialog box will appear. Search for the components which are required in the circuit. In this case, we require an IC 555, Two LEDs, Two Capacitors, four resistors and a connector.(In place of voltage source)



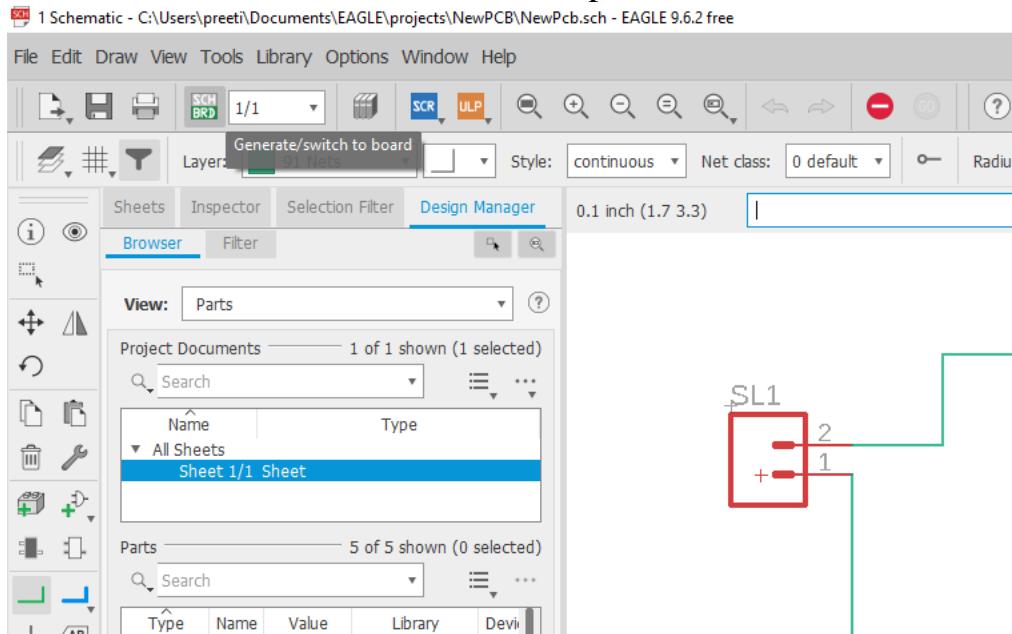
6. After Selecting all the components, place them according to the circuit. Now, we can make the connections in the circuit.



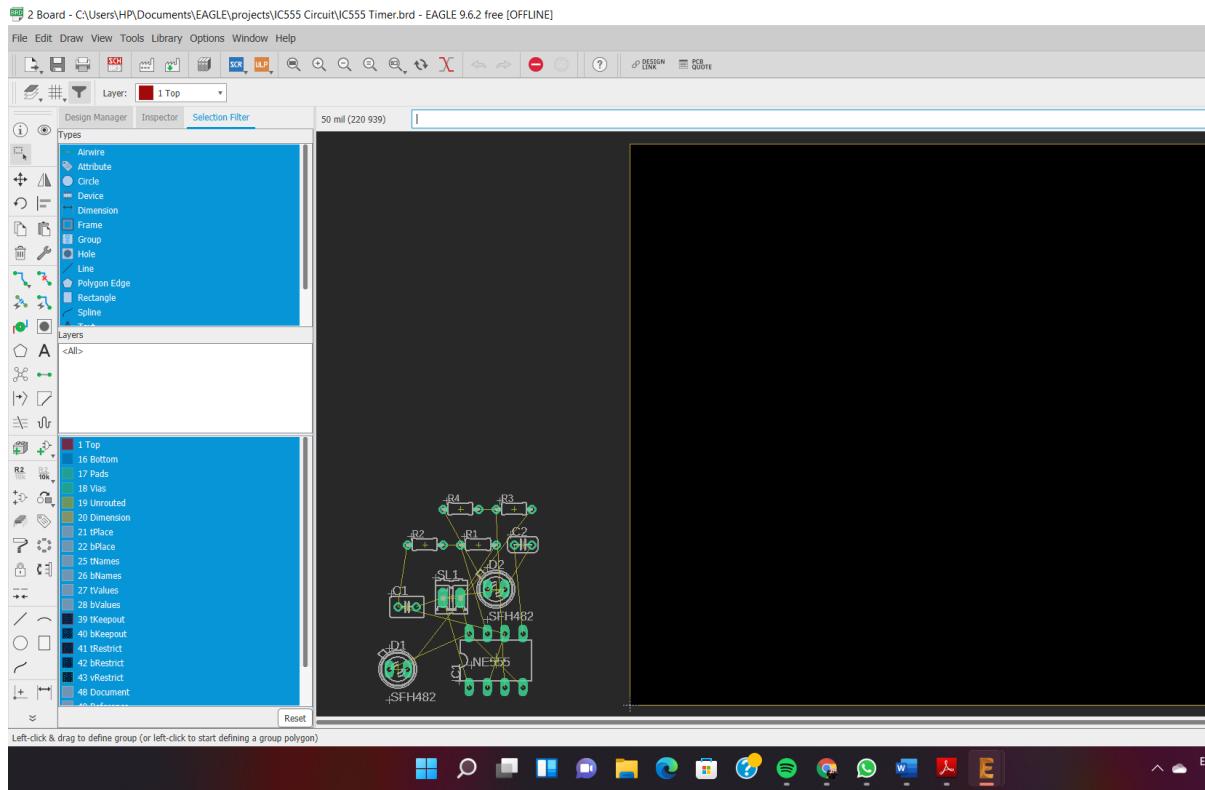
7. Now select Net from the left menu and make the connections between the components according to the circuit.



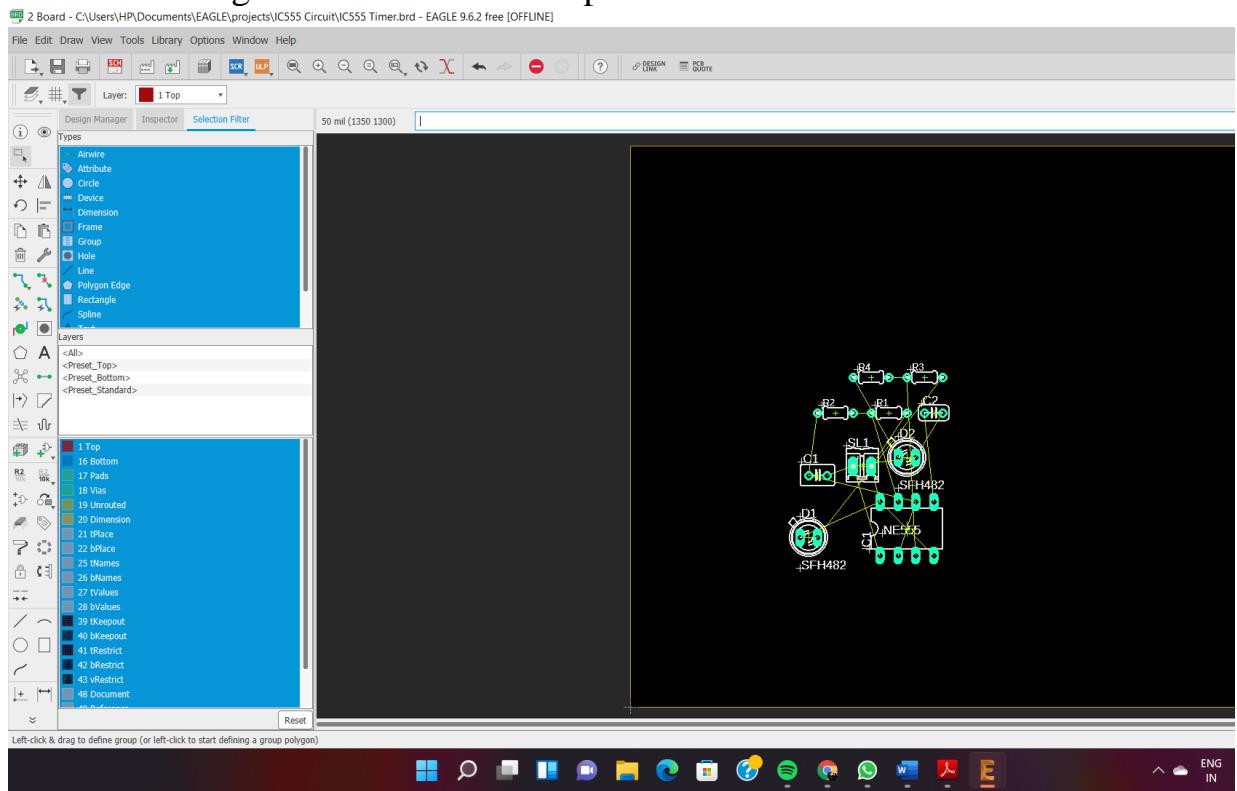
- Now, your schematic board is ready. Save it accordingly and then click on ‘Generate/Switch to board’ from the top menu.



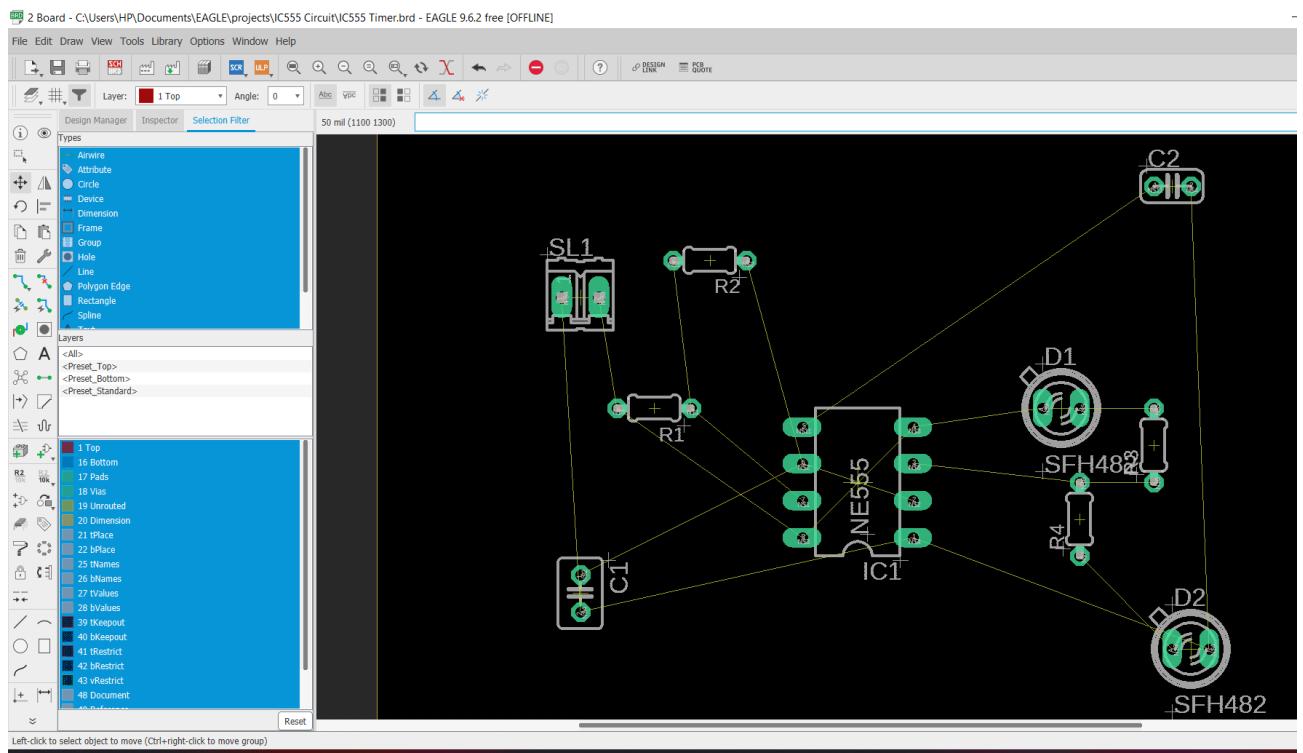
- Click on YES and now your screen will look like this:



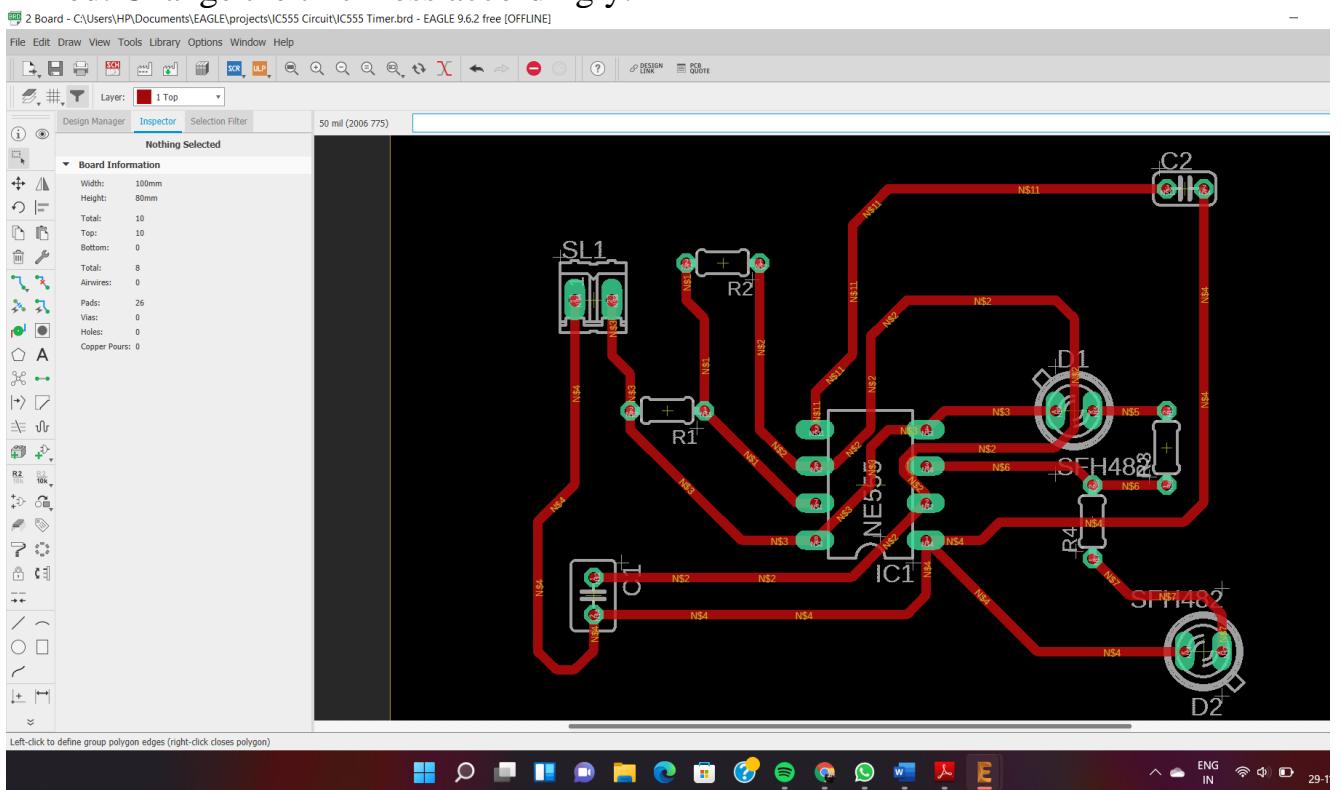
10. Select the Design from the corner and place it in the black area.



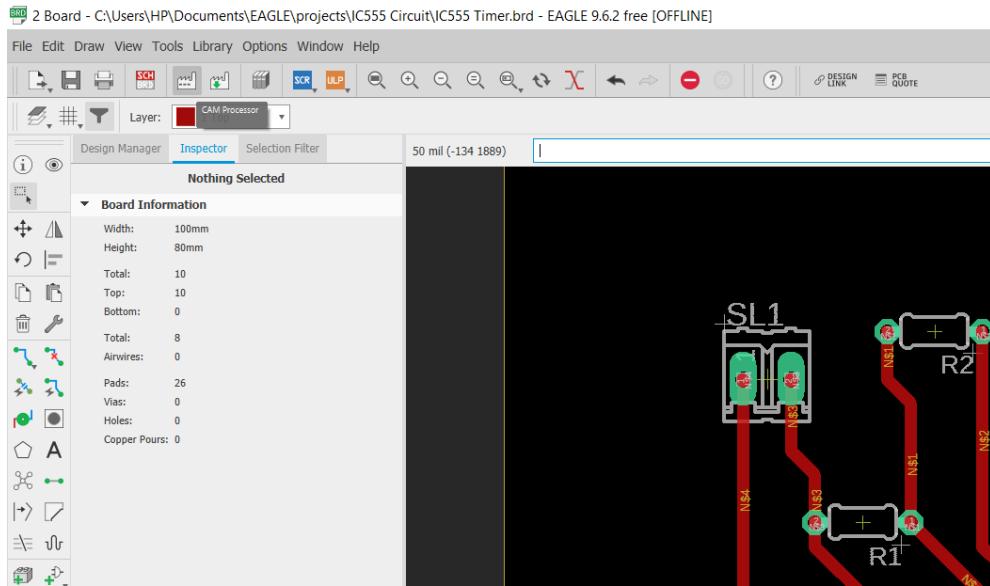
11. The software has generated the board on its own. Arrange all the components neatly and as close as possible. The design should be as compact and simple as possible. Make sure that the wires don't make any right angles.



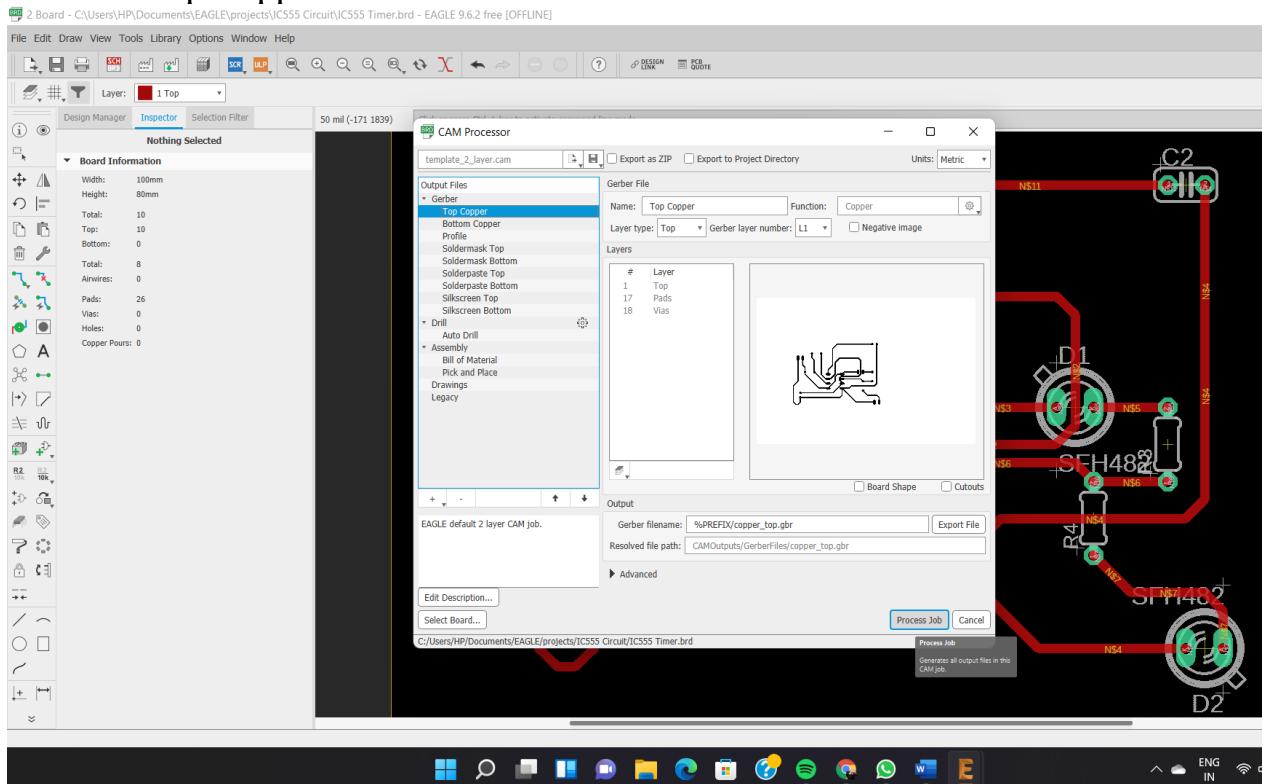
12. Now Select ‘Route Airwire’ to make final connections. These wire connections will be red. Change the thickness accordingly.



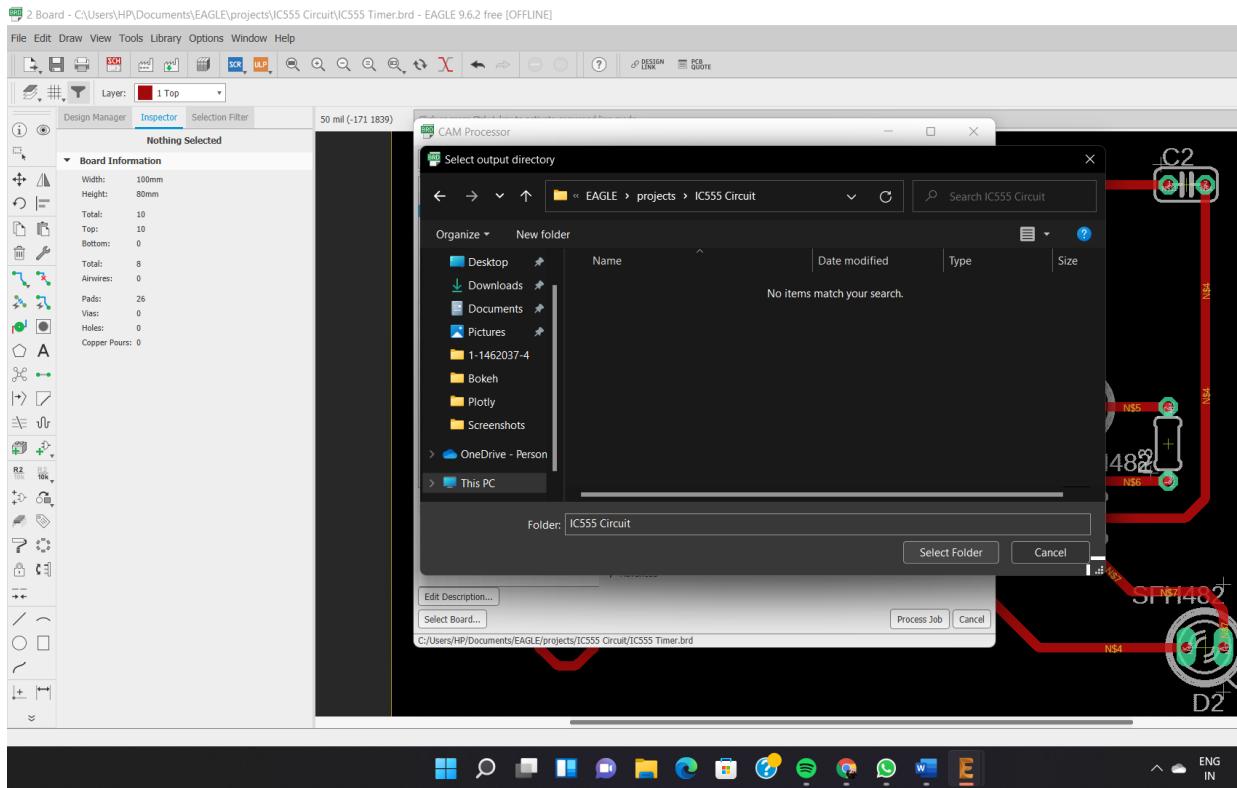
13. Click on Cam Processor



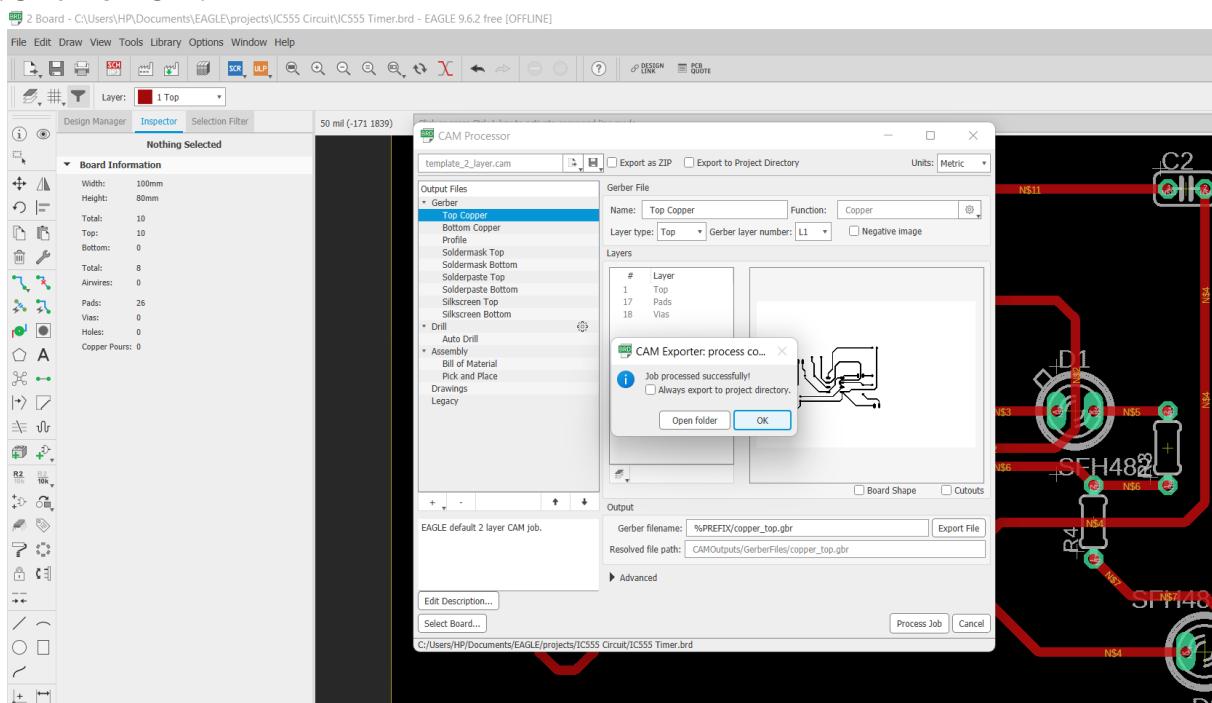
14. Click on Top Copper and Process Job



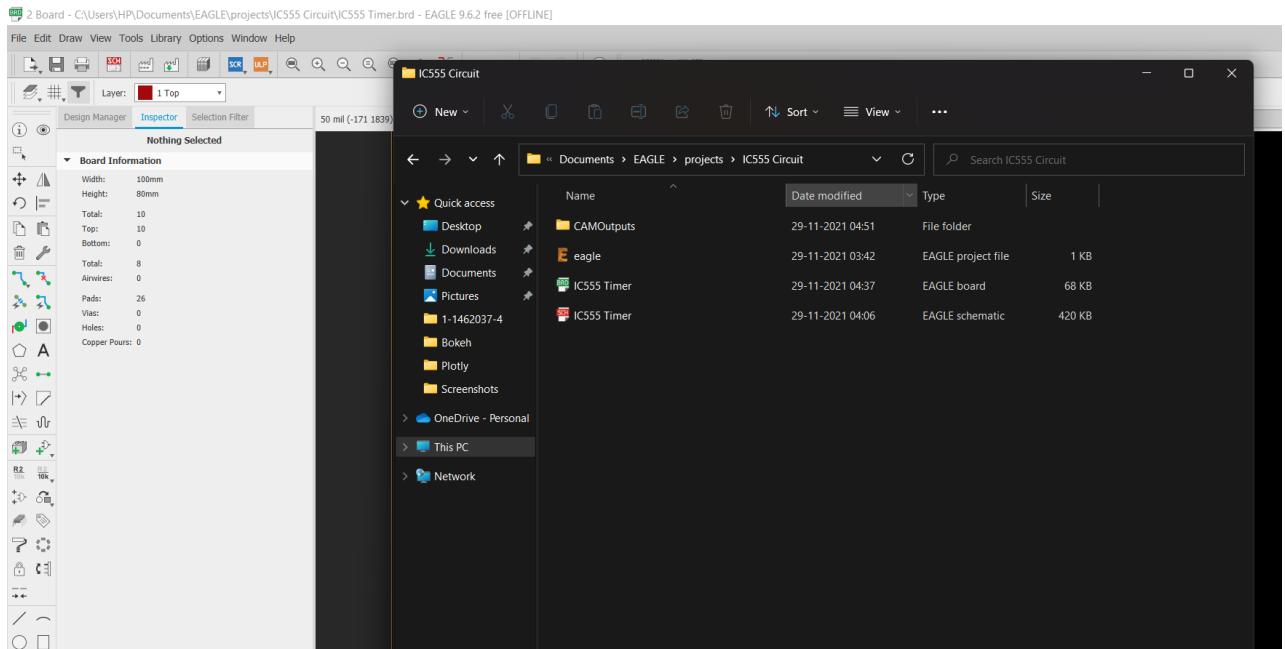
15. Select the Folder in which you want to Save.



16. Click on OK.

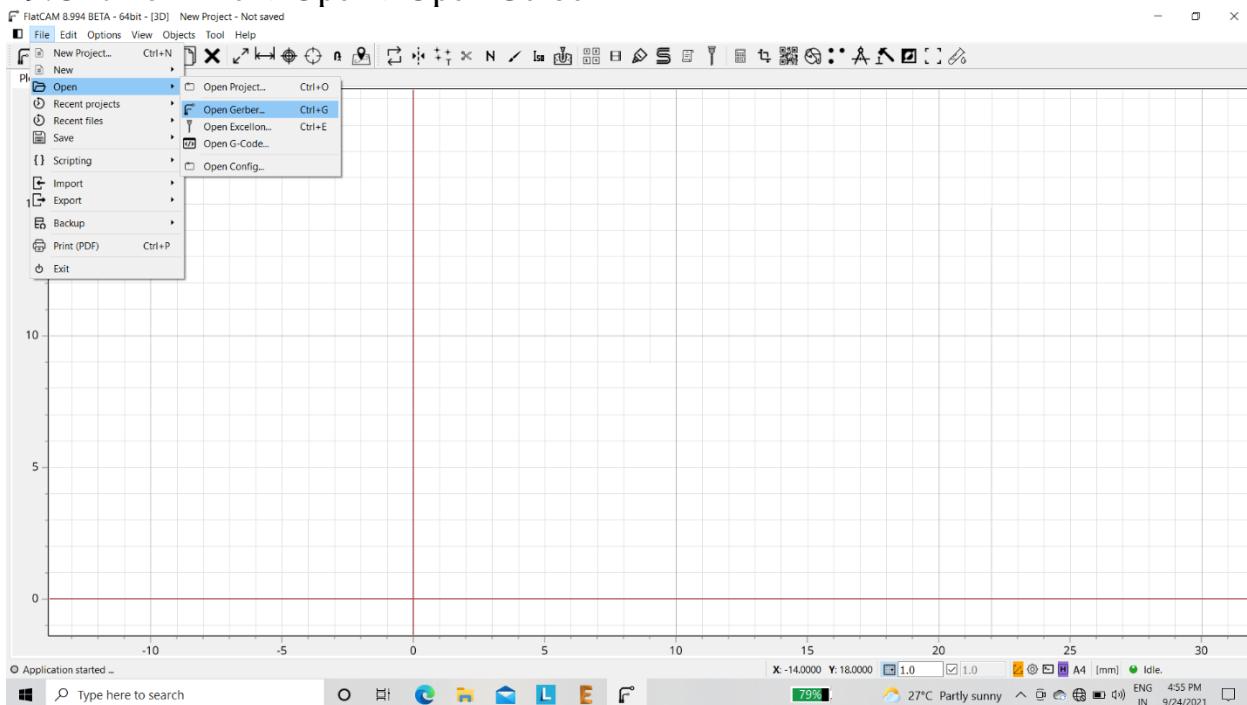


17. CAMOutputs Folder is generated.

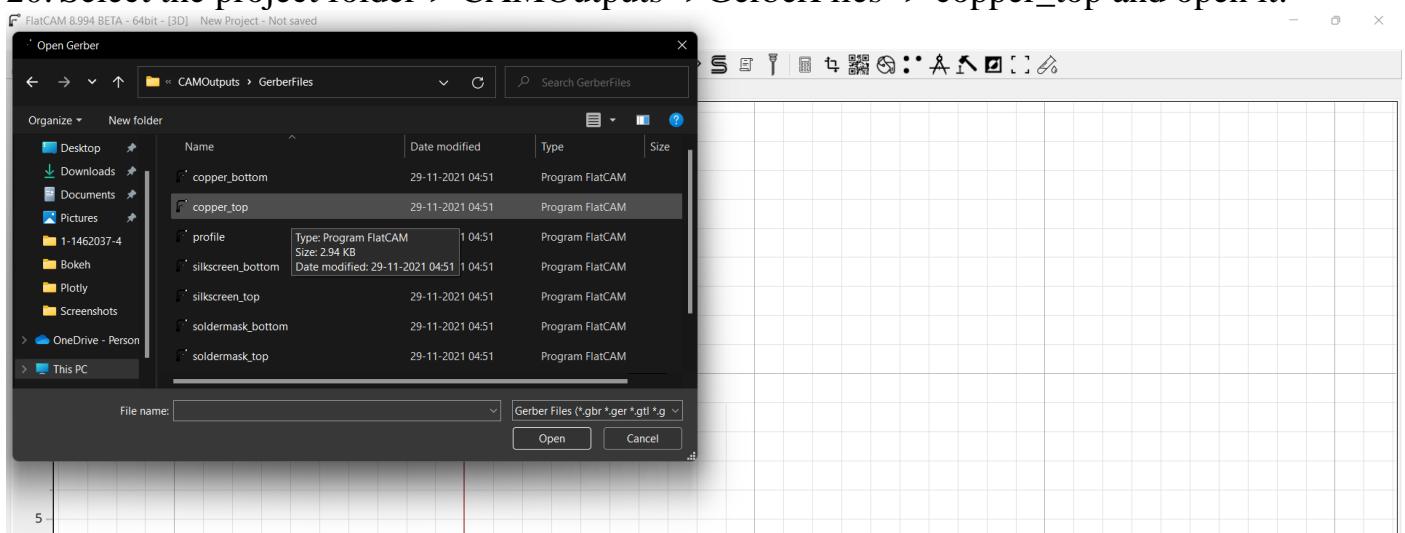


18.Close EAGLE and Open FlatCam software.

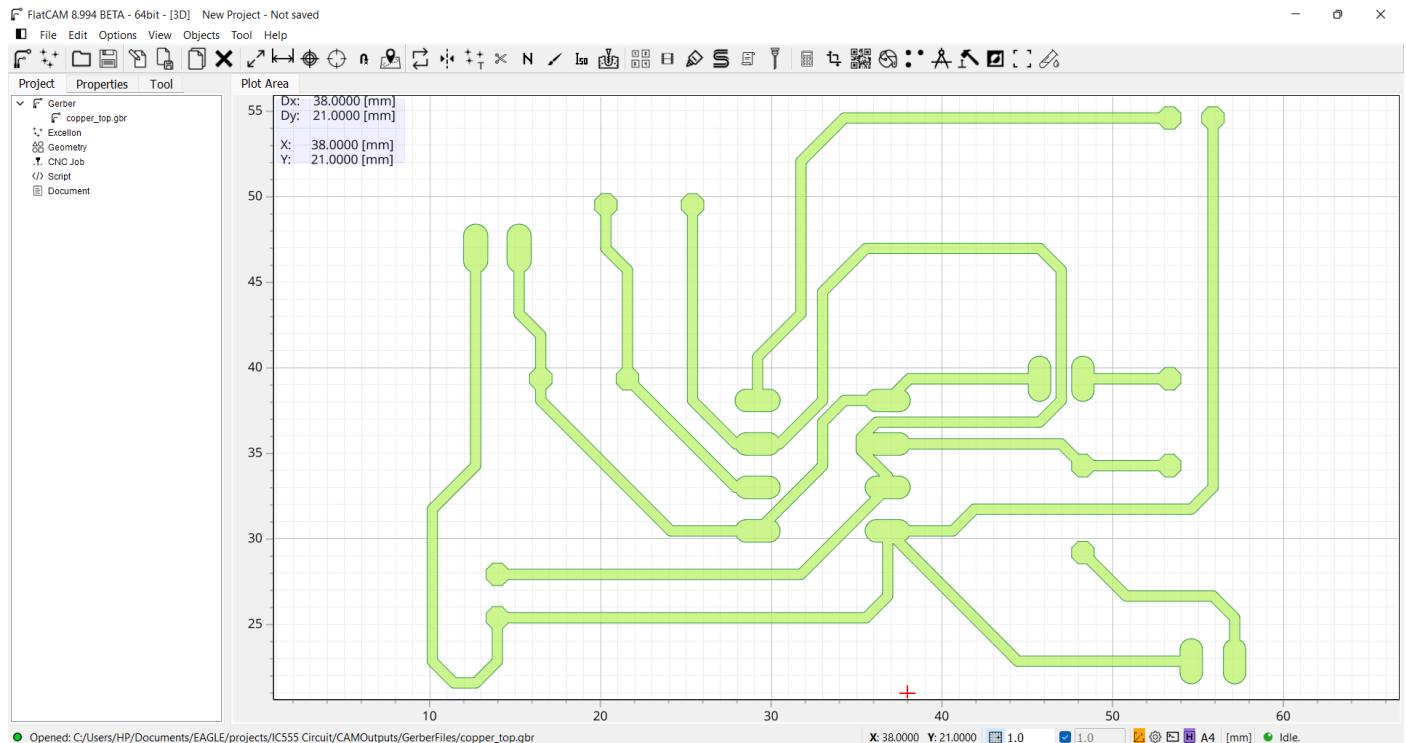
19.Click on File ->Open->Open Gerber



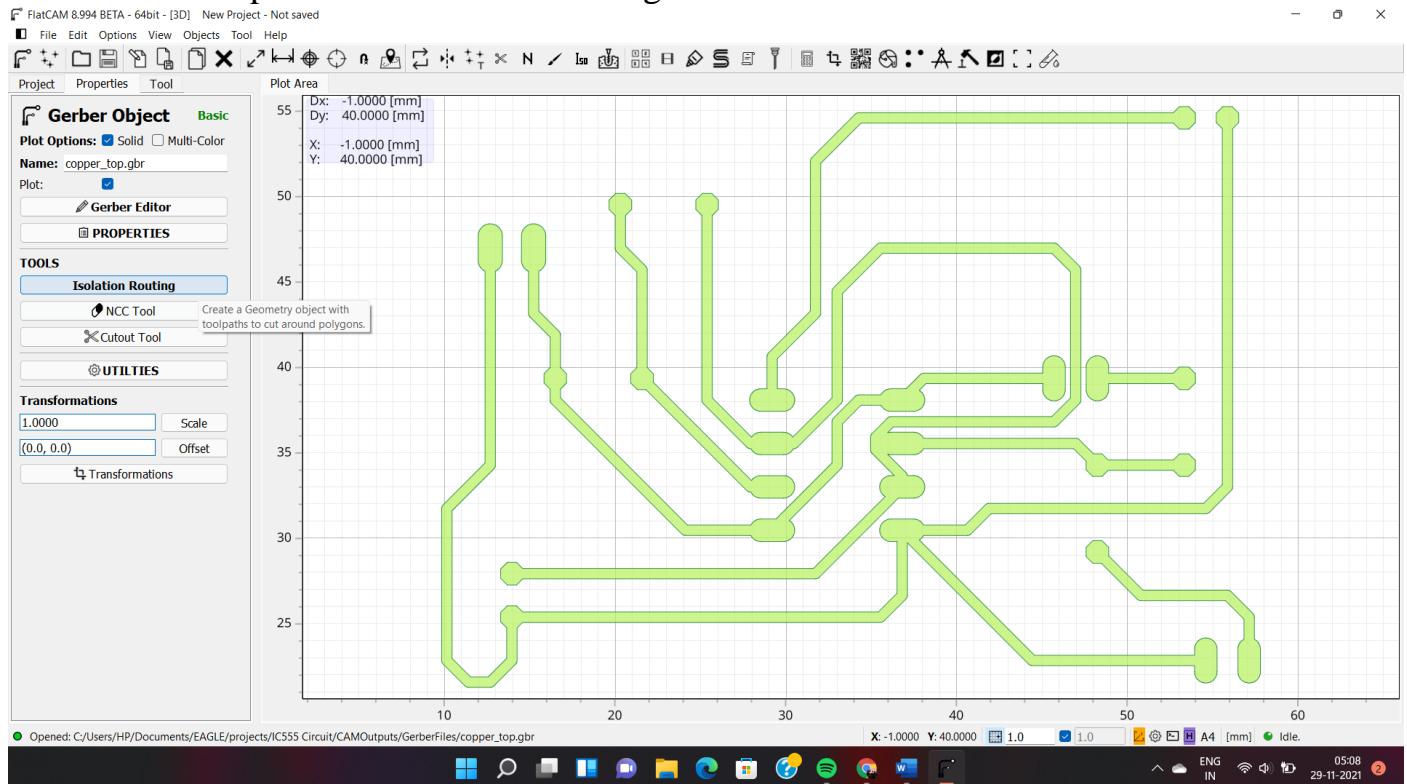
20. Select the project folder-> CAMOutputs ->GerberFiles -> copper_top and open it.



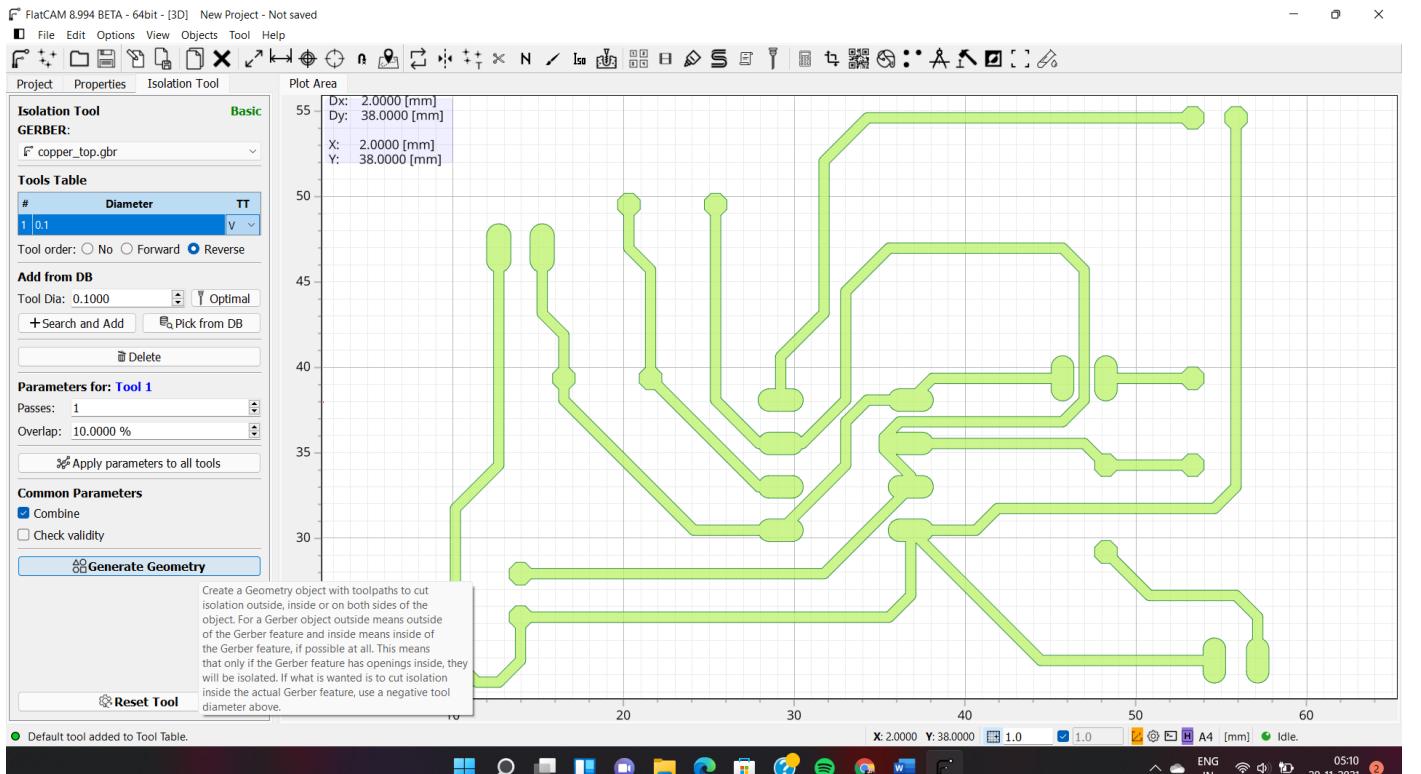
21. Click on copper_top.gbr under Gerber on the left under project section, your design will be visible.



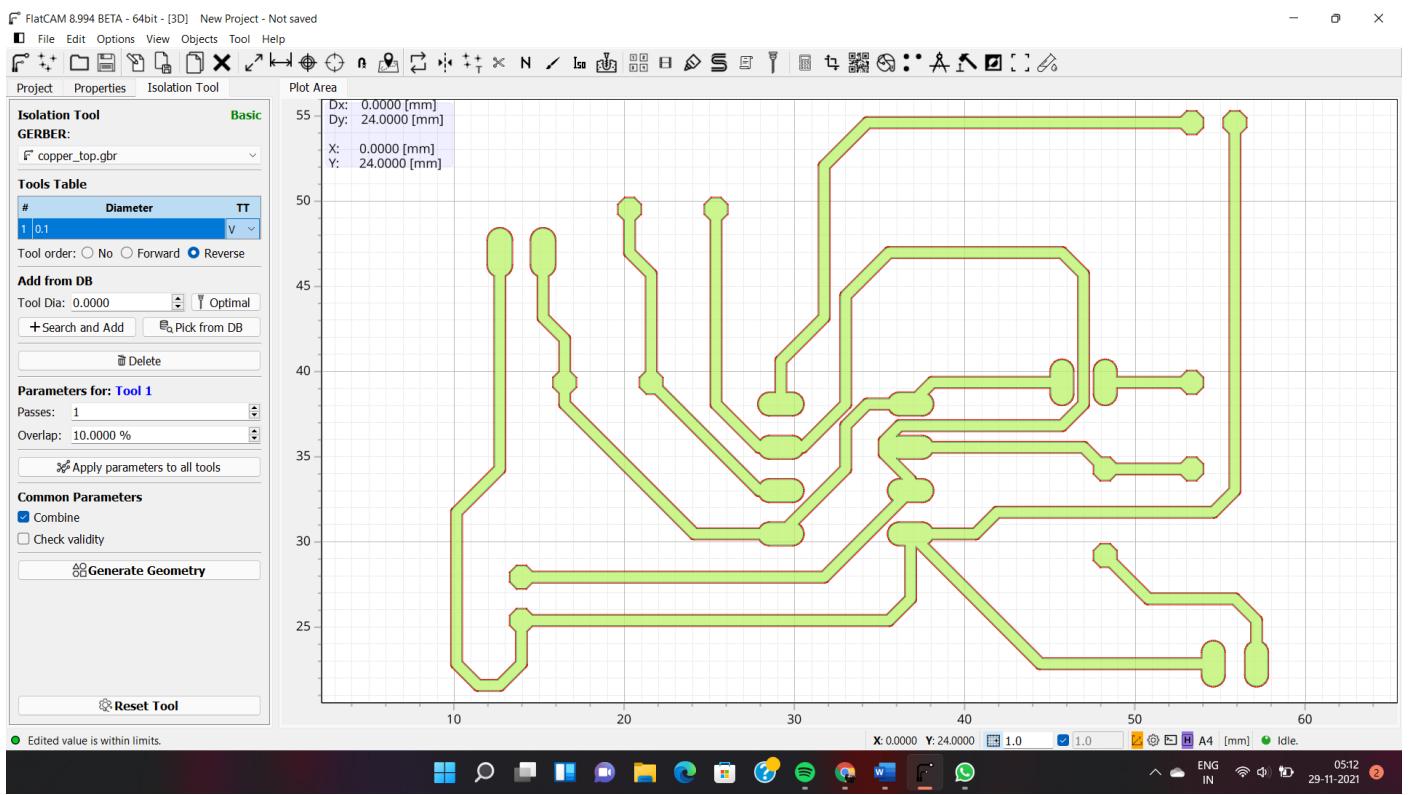
22. Click on Properties -> Isolation Routing



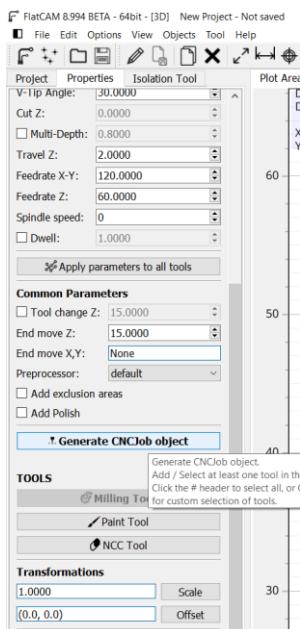
23. Click on Generate Geometry.



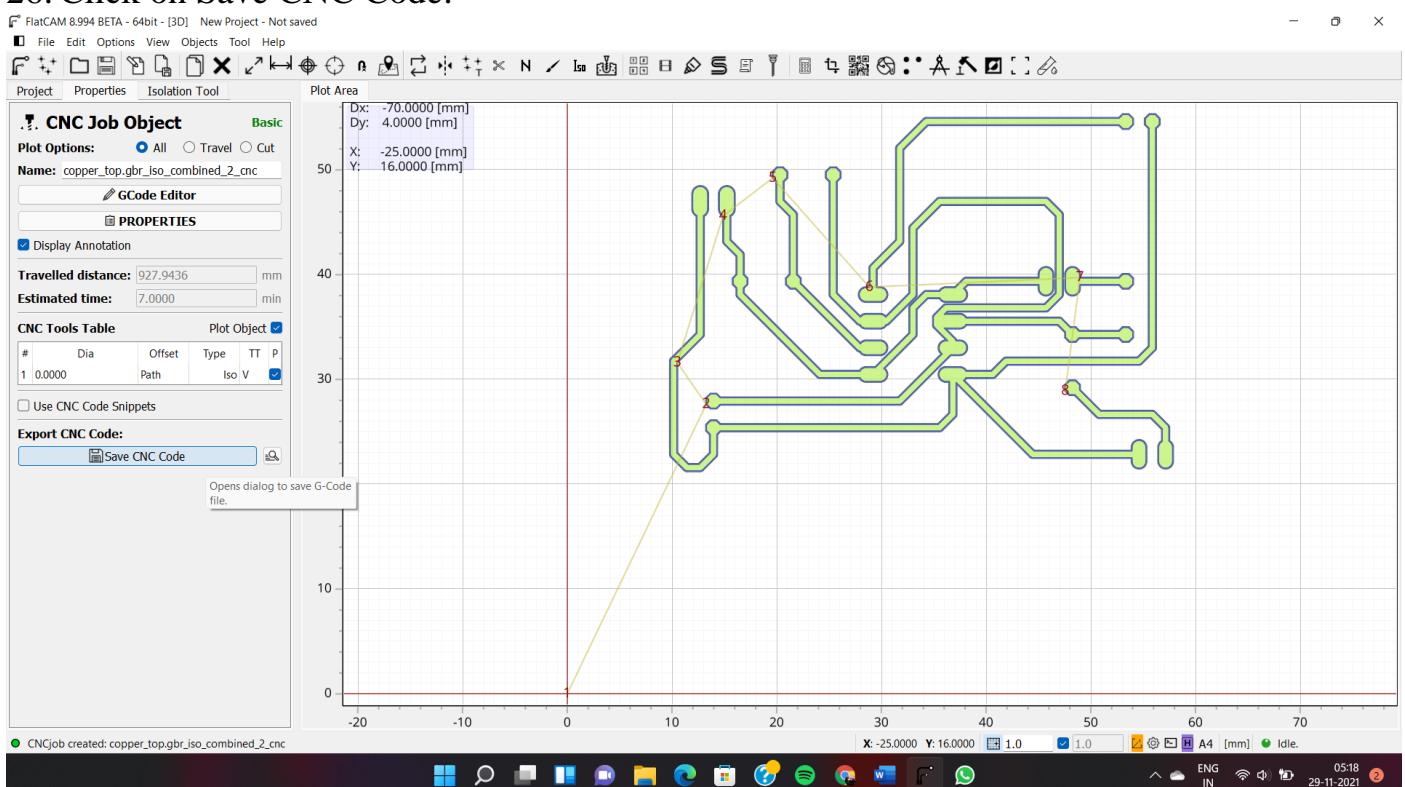
24. Click on Isolation Tool->Set the Diameter to zero->Click on Generate Geometry.



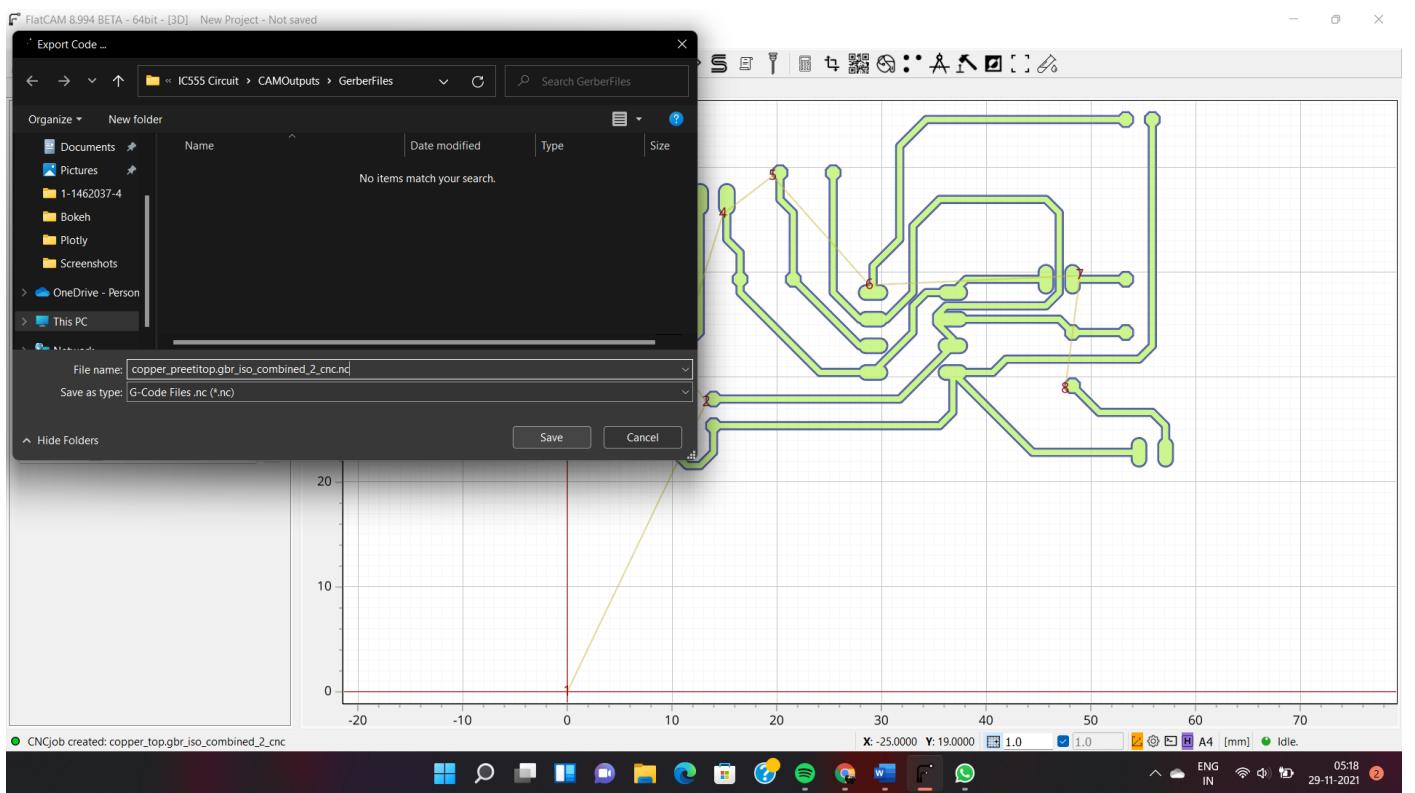
25. Click on Generate CNCJob object under Properties section.



26. Click on Save CNC Code.

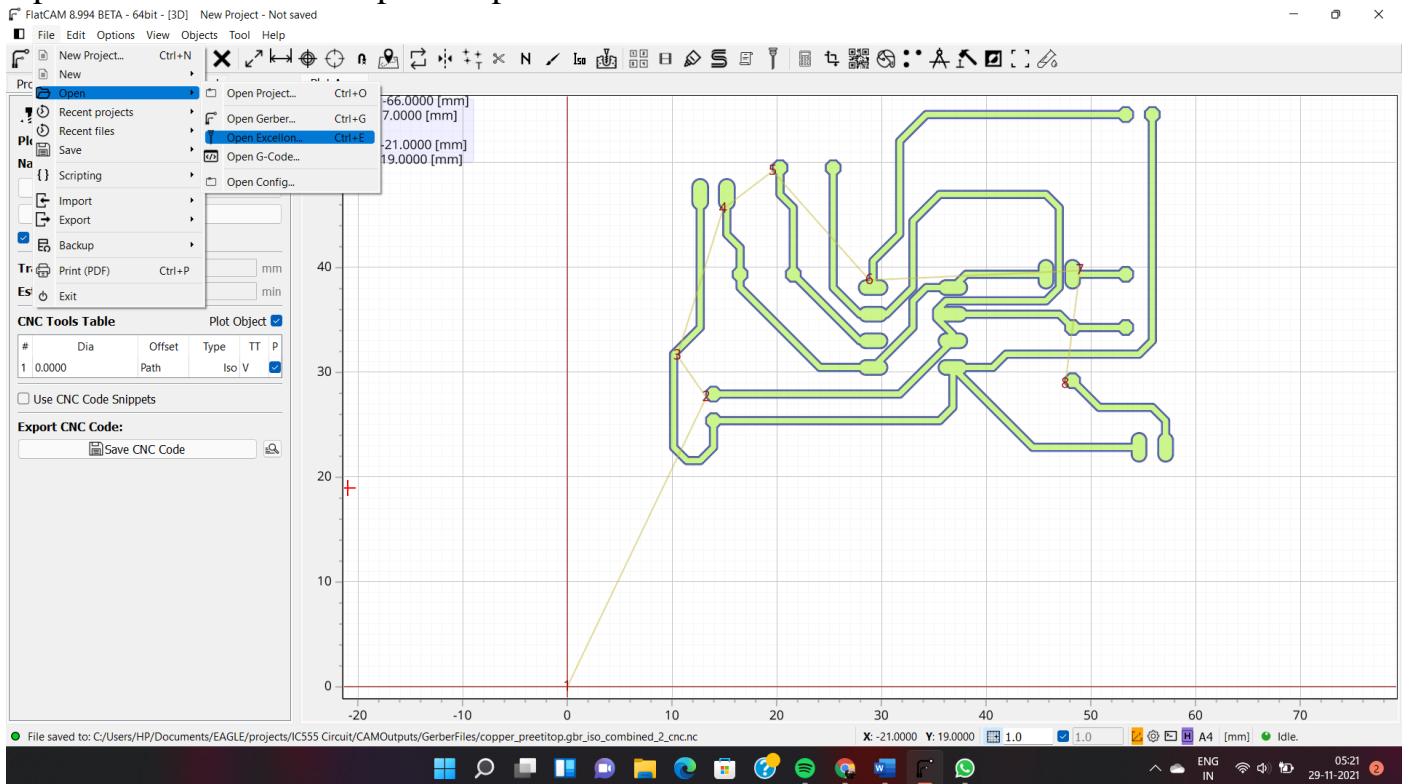


26. Name the file with .nc extension and Save it.



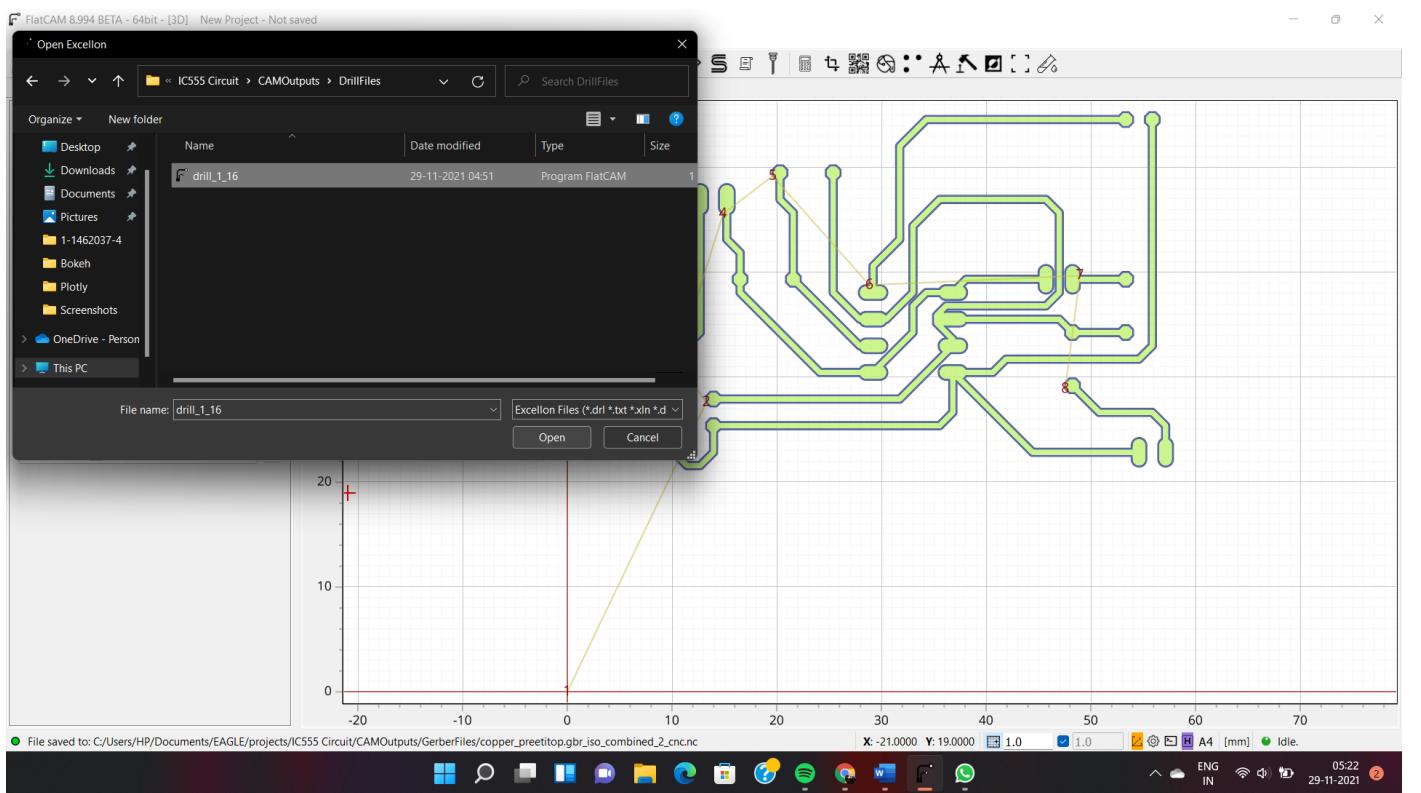
27. Your gerber cnc file is generated which includes geometry design. Now let's generate drill file.

Open FlatCam->File->Open->Open Excellon

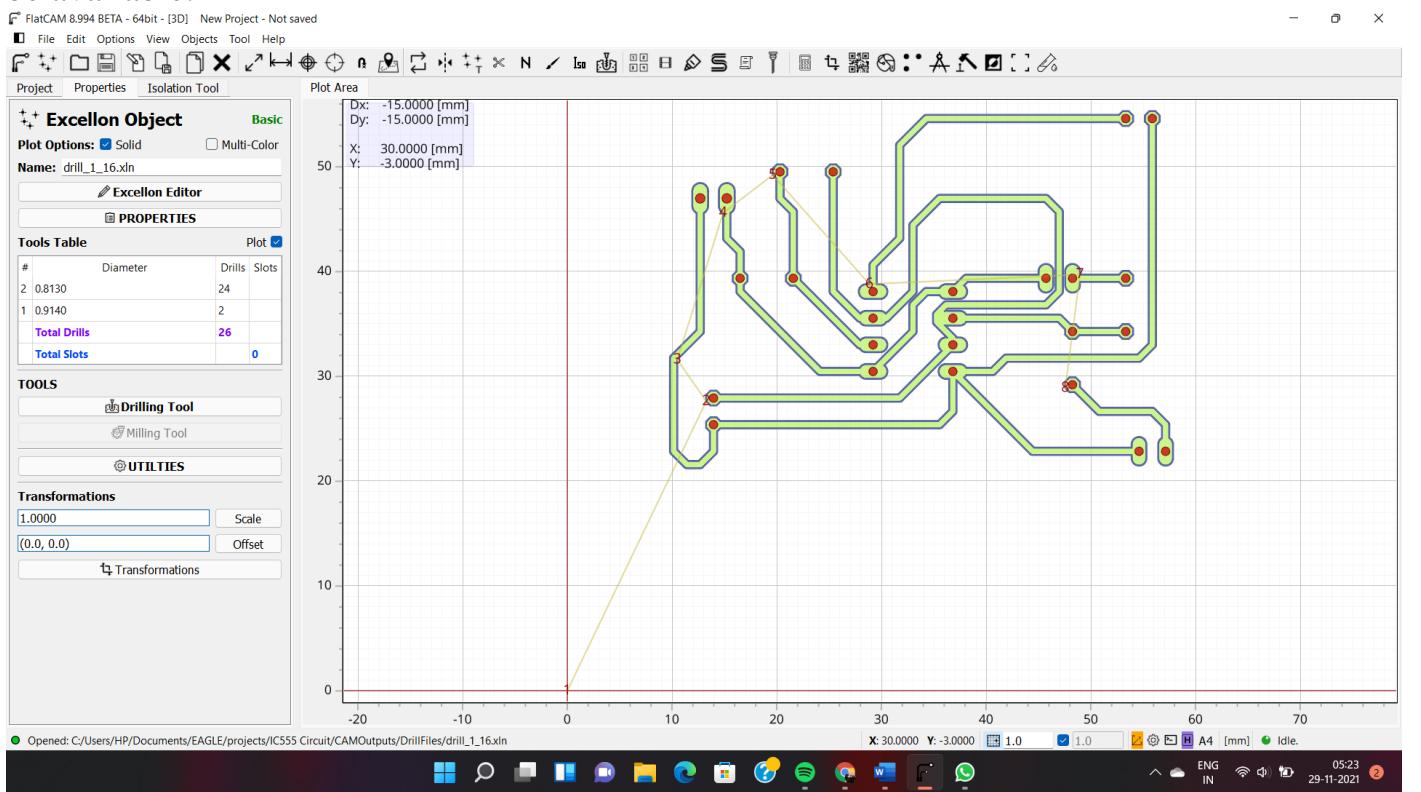


28. Select the Folder and Open it.

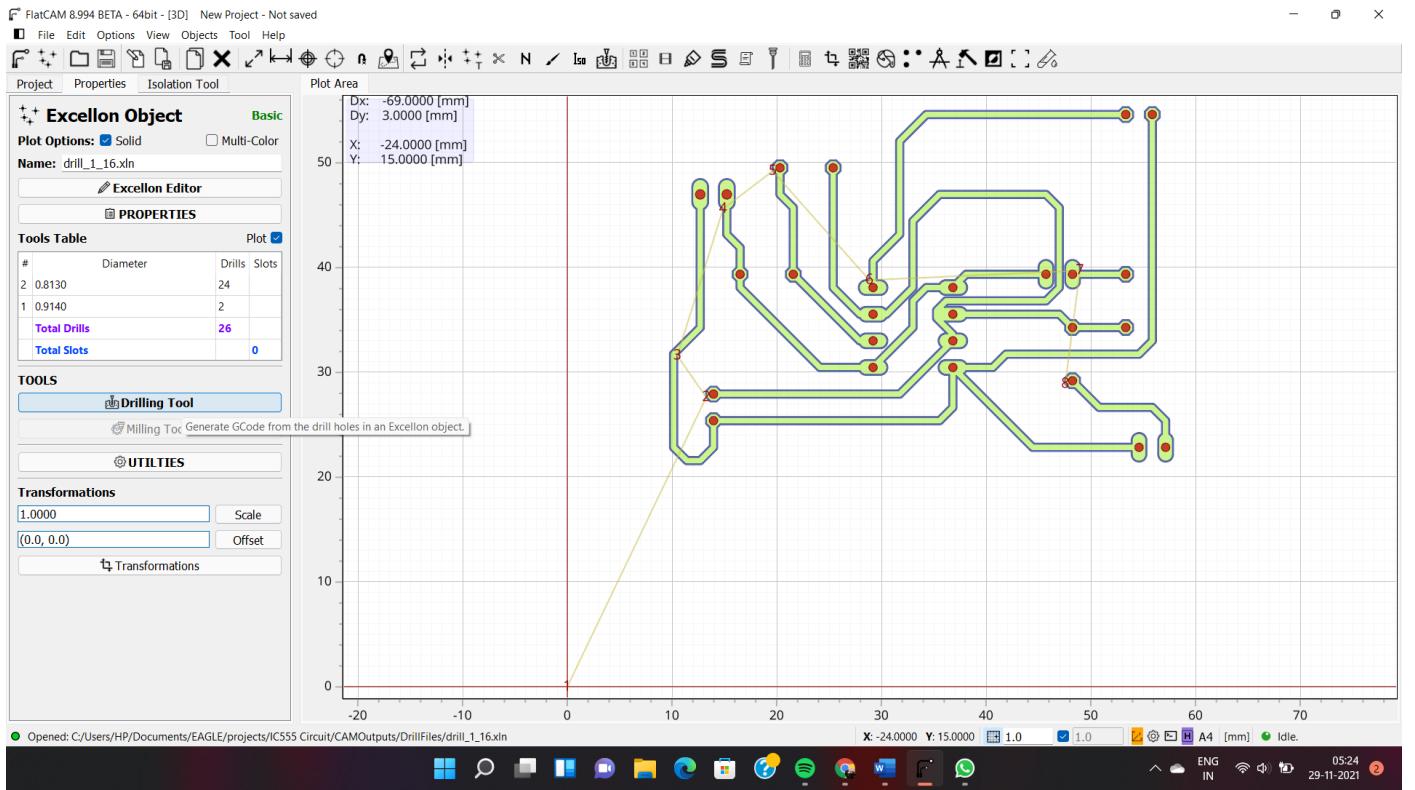
The drill file is present in projects->your_project_name->CAMOutputs->DrillFiles.



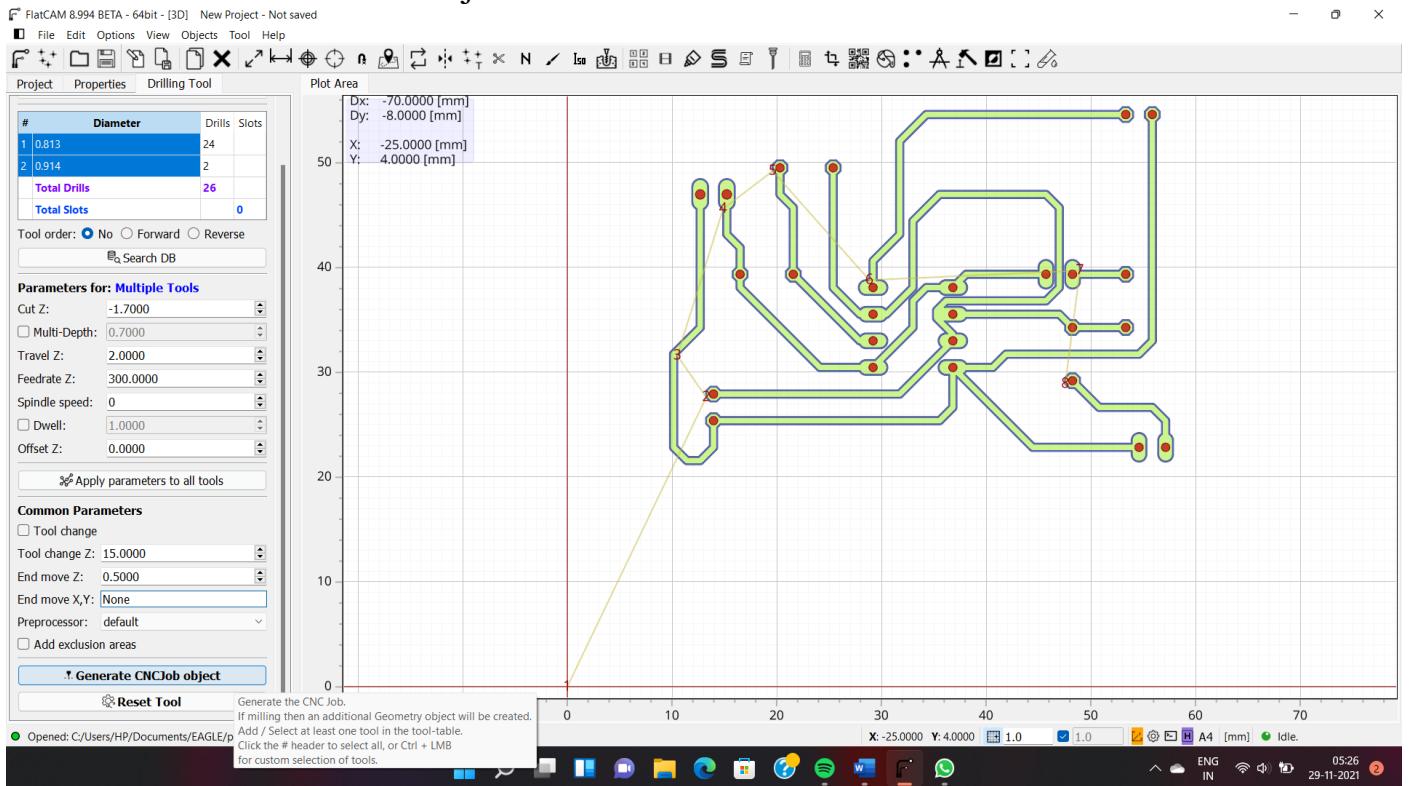
29. Click on the drill file under Excelon in Project section. Your design with drill holes will be available.



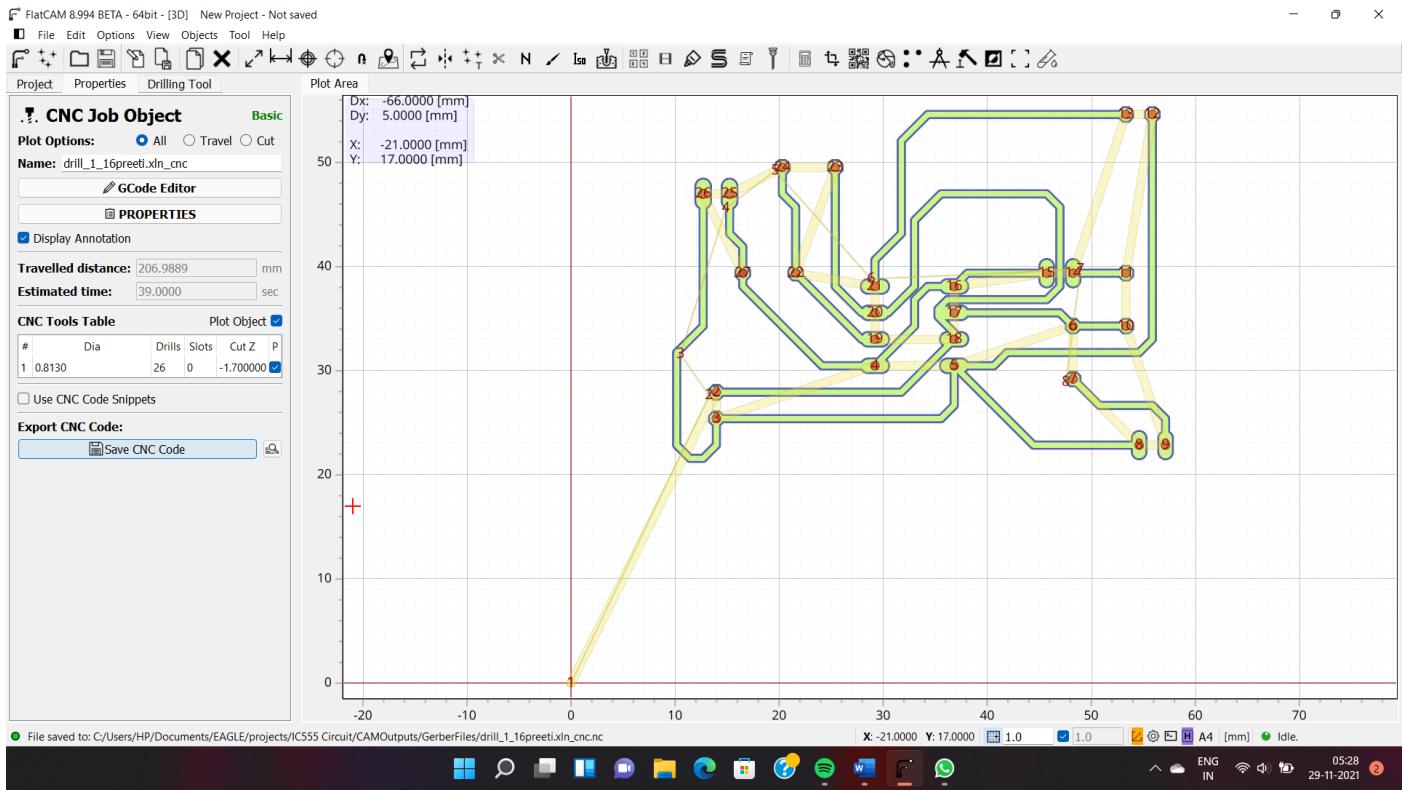
30. Click on Drilling Tool.



31. Click on Generate CNC object.



32. Click on Save CNC Code and save it with .nc extension.



33. Your cnc file for drilling design is ready.
 Copy the gerber cnc file and drill cnc file to pendrive
 Insert the pendrive in the machine
 Set the coordinates and select file and start.
 Your PCB design will be printed on board.

After the design is printed on the board, we have to manually drill the holes on the board.
 These holes is where our electronic components will go.
 After placing the components at their places, solder them carefully.