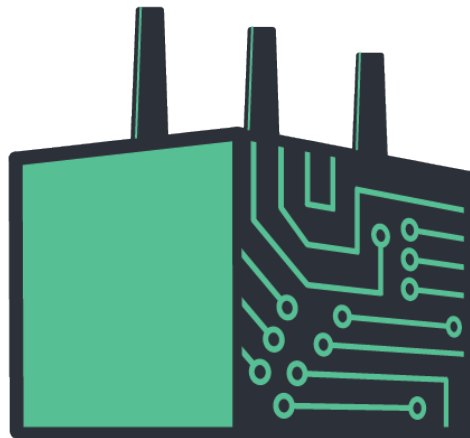


**EXPORTING GERBER FILES WITH:**



**CIRCUITMAKER**

**PRESENTED BY:**

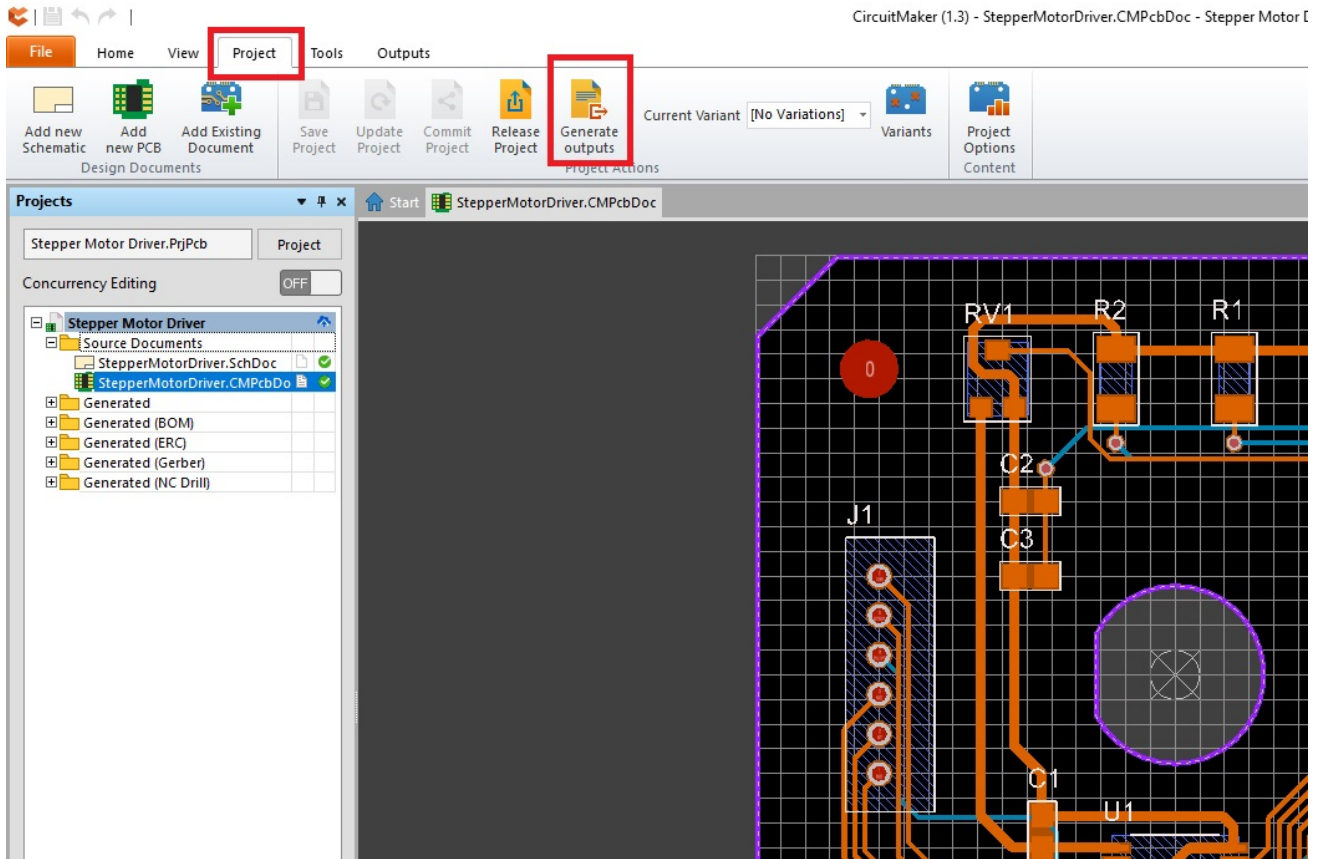


**THE FACTORY**

<https://thefactory.mcgilleus.ca>

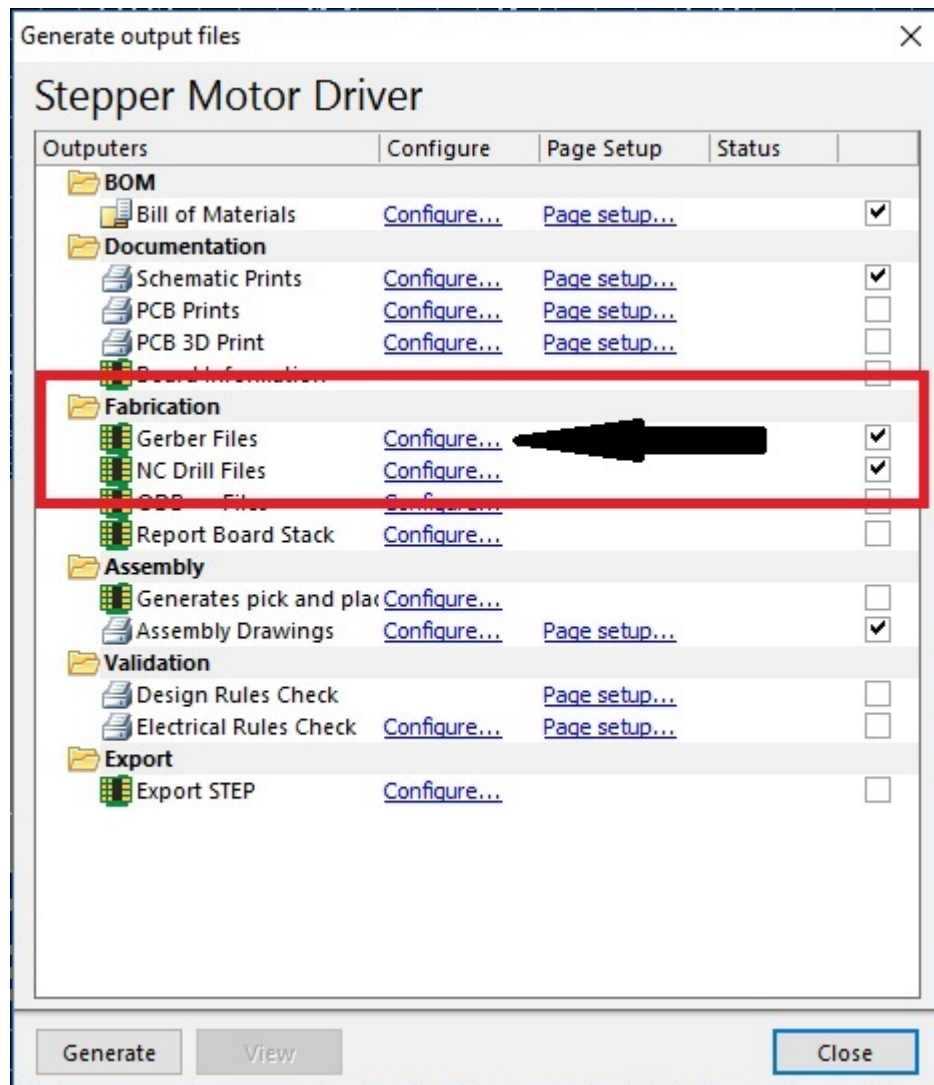
## STEP 1:

Having selected your PCB project, go to the “Project” tab on the upper left corner of the window and click on “Generate outputs”



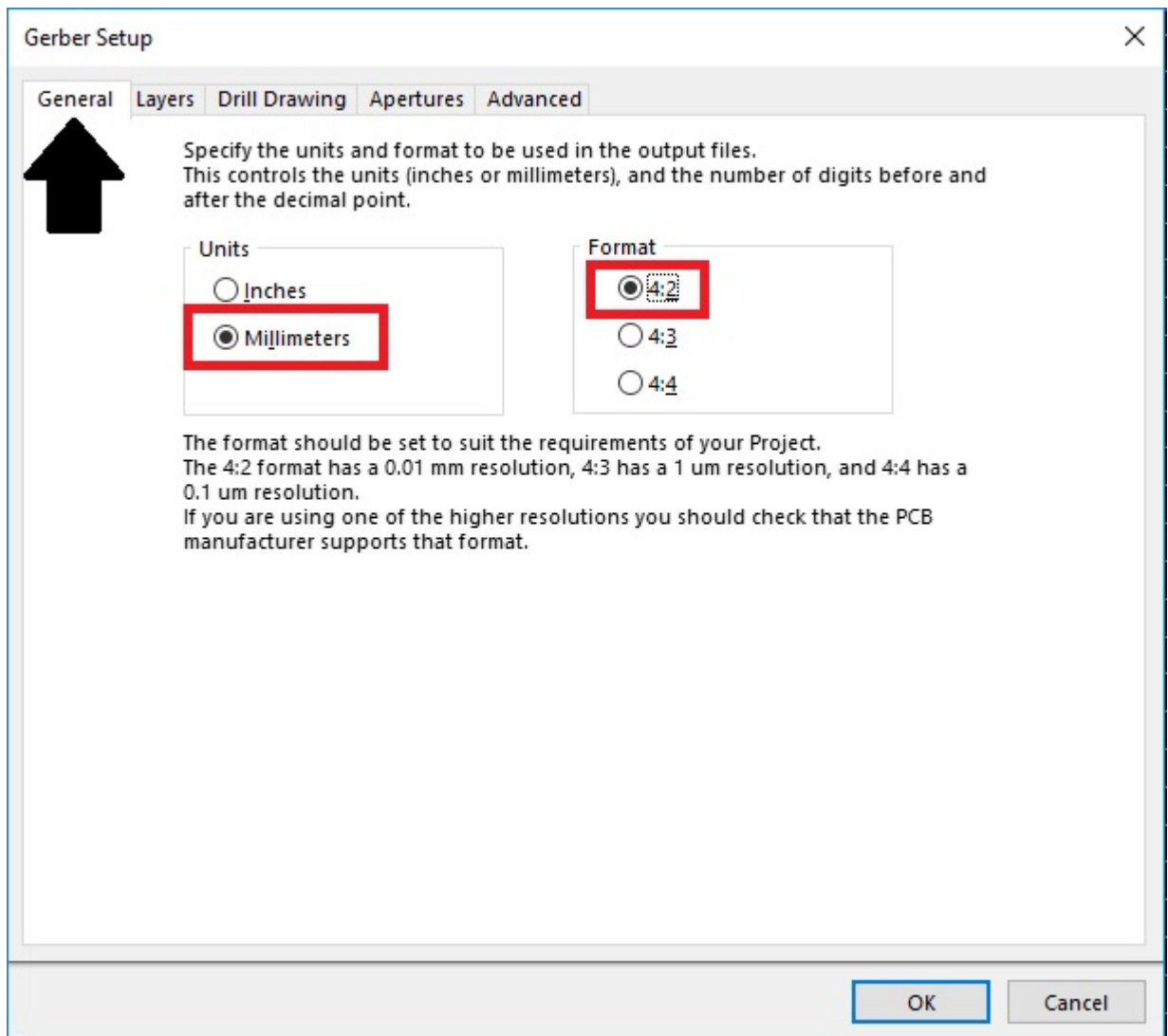
## STEP 2:

The following window will pop-up. Make sure that the “Gerber Files” option and the “NC Drill File” option are selected. Once selected click on the Gerber Files “Configure” option.



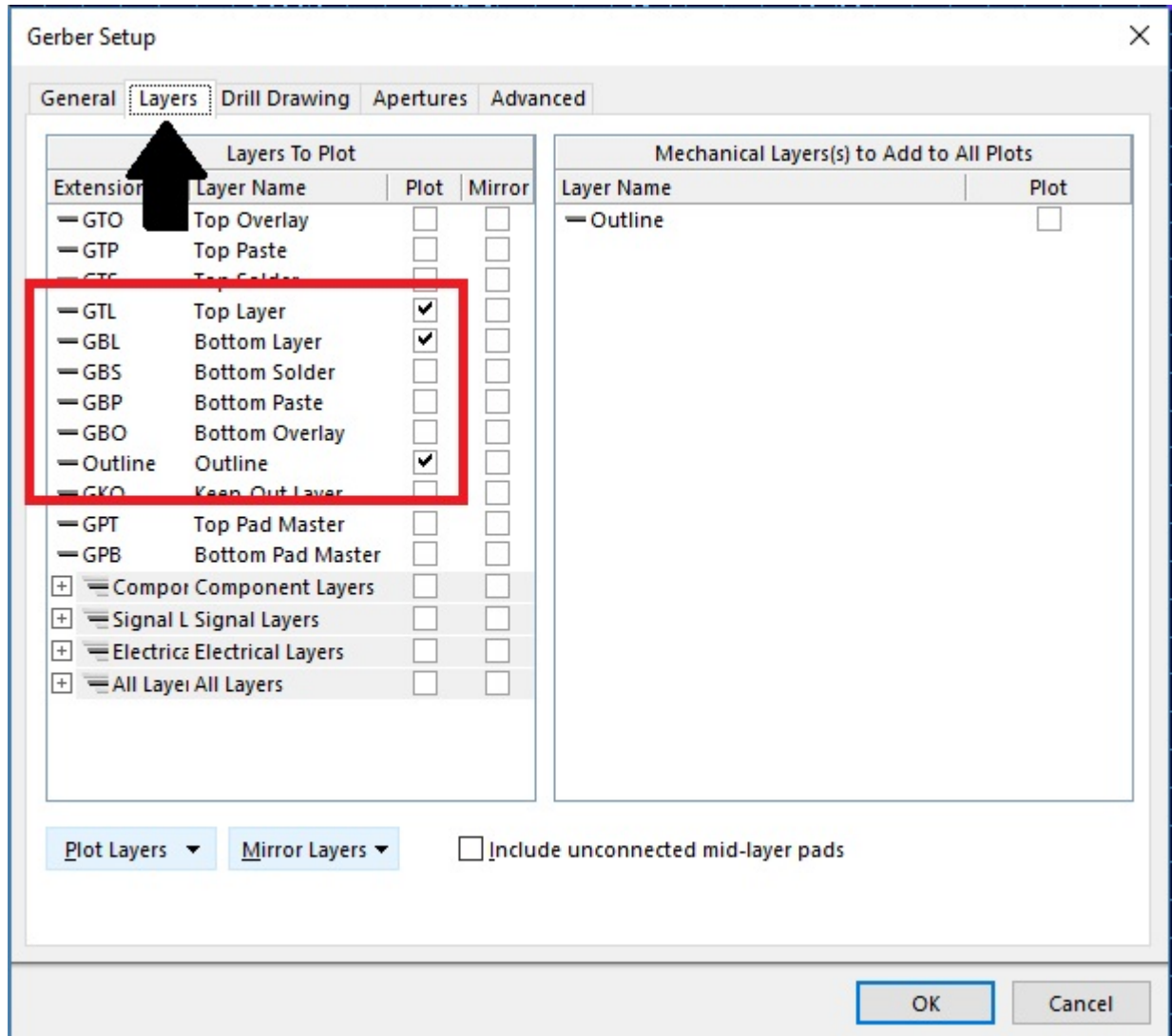
### **STEP 3:**

The following configuration window will pop-up. On the “General” tab, select **Millimeters** as your units and select **4:2** as your Format.



#### STEP 4:

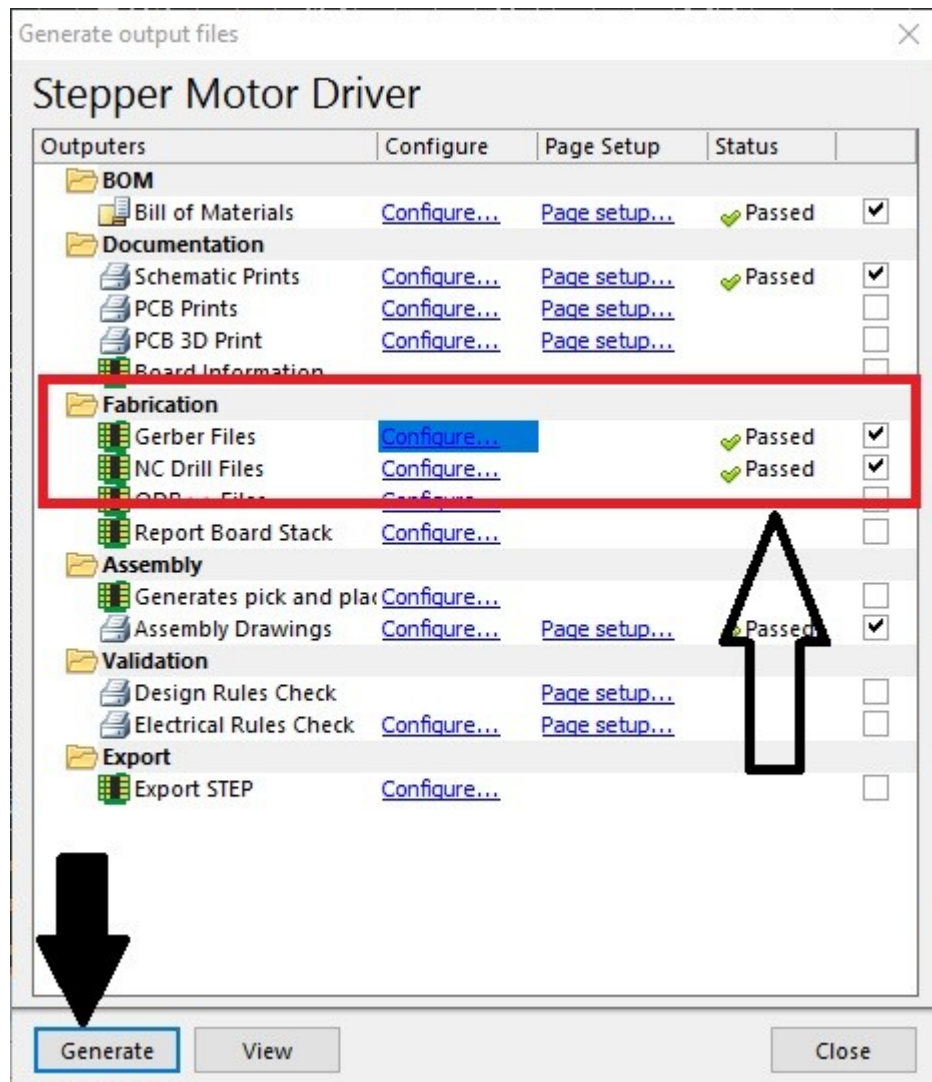
Go to the “Layers” tab and select the “Top Layer”, “Bottom Layer” and “Outline” plots as shown below. Then Click “OK” at the bottom right of the window.



## STEP 5:

Click on “Generate” and make sure that the “Gerber Files” and “NC Drill Files” options have been generated (denoted by the “Passed”). If one of those doesn’t pass make sure your PCB is correct and that you’ve followed steps 1-4 correctly.

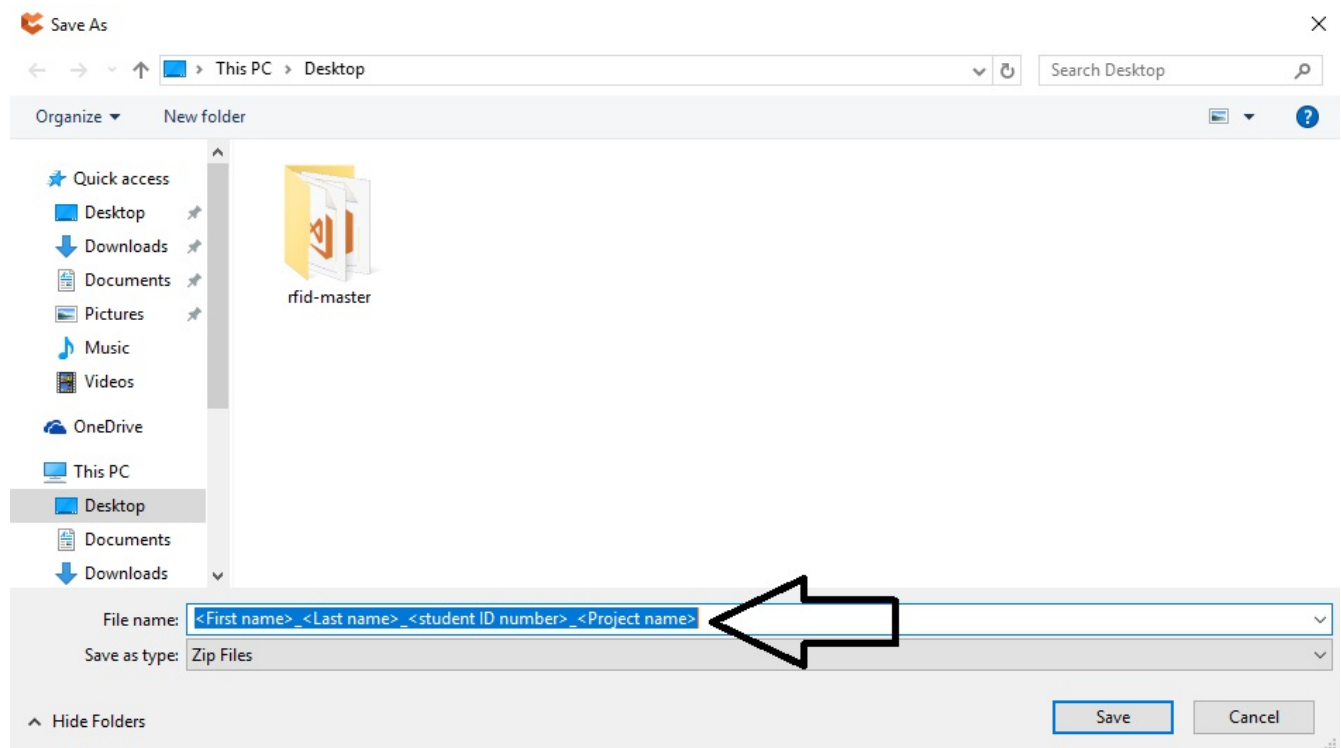
Once Passed a new window will open.



## STEP 6:

The new window will prompt you to enter a location and name for the output files. Name the .zip file that will be generated in the following way:

<First name>\_<Last name>\_<Student ID number>\_<Project name>



## STEP 7:

Send an e-mail to [thefactory@mcgilleus.ca](mailto:thefactory@mcgilleus.ca) with Subject: PCB Request <Name> <Student ID number>. Don't forget to attach the generated Zip file to your e-mail.