

# PRESENTED BY:

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"

<b>ジ</b> 目目 ち た I		Ci	rcuitMaker (1.3	) - StepperMotorDri	ver.CMPcbDoc - Stepper Motor [
File Home View Project Tools	Outputs				
Add new Add Add Existing Schematic new PCB Document Design Documents	Update Commit Release Ge Project Project Project P	Current Variant [No Variations] -	Variants	Project Options Content	
Projects 🗸 🖛 🛪	🕻 🏫 Start 🏢 StepperMotorDrive	er.CMPcbDoc			
Stepper Motor Driver.PrjPcb Project					
Concurrency Editing OFF					
Stepper Motor Driver				RV1	
StepperMotorDriver.SchDoc			•		
Generated (BOM)					
Generated (ERC)					
E Generated (Gerber)				40	
			1		
				03	
				01	

# <u>STEP 2:</u>

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.

Outputers	Configure	Page Setup	Status	
	configure	rage setup	Status	
Bill of Materials	Configure	Page setup		~
Schematic Prints	Configure	Page setup		<b>~</b>
PCB Prints	Configure	Page setup		
PCB 3D Print	Configure	Page setup		
Pabrication				
Gerber Files	Configure			✓
NC Drill Files	Configure	1999		
Para di Para di Charle	C			
Report Board Stack	Configure			
Generates nick and ni	e Configure			
Assembly Drawings	Configure	Page setup		-
P Validation	<u>comparent</u>	- age betaprin		
Design Rules Check		Page setup		
Electrical Rules Check	Configure	Page setup		
Export				
Export STEP	Configure			

# 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.

Layers To Plot		Mechanical Layers(s) to Add to All Plots		
Extension	Layer Name Plot Mirror	Layer Name	Plot	
<ul> <li>GTO</li> <li>GTP</li> <li>GTI</li> <li>GBL</li> <li>GBS</li> <li>GBP</li> <li>GBO</li> <li>Outline</li> <li>GFO</li> <li>GPT</li> <li>GPB</li> <li>÷ ■ Compo</li> <li>÷ ■ Signal</li> <li>÷ ■ Electric</li> <li>† ■ All Lay</li> </ul>	Top Overlay   Top Paste   Top Layer   Bottom Layer   Bottom Solder   Bottom Paste   Bottom Overlay   Outline   Keen Out Laver   Top Pad Master   Bottom Pad Master   Bottom Pad Master   Sottom Pad Master   Signal Layers   Et All Layers	- Outline		
<u>P</u> lot Layers	▼ <u>M</u> irror Layers ▼ □ Inclu	de unconnected mid-layer pads		

# 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 with Subject: PCB Request <Name> <Student ID number>. Don't forget to attach the generated Zip file to your e-mail.