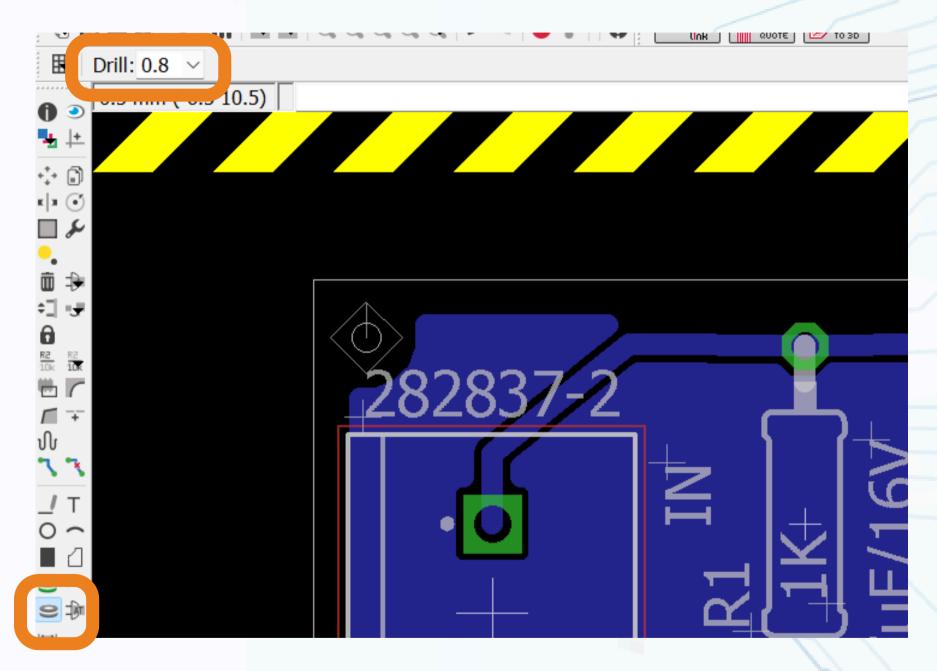
# UNIT Y

Finalization and Export



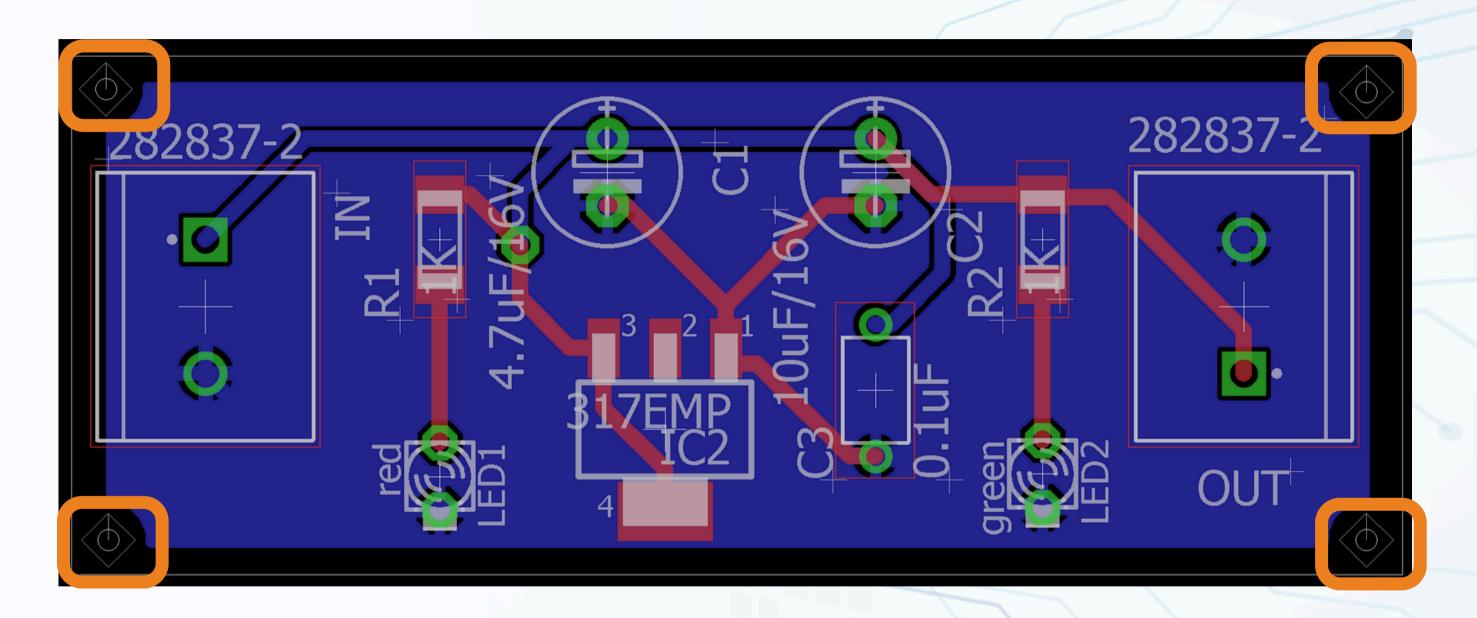
# Mounting Holes



**Select > Holes > Drill size** 



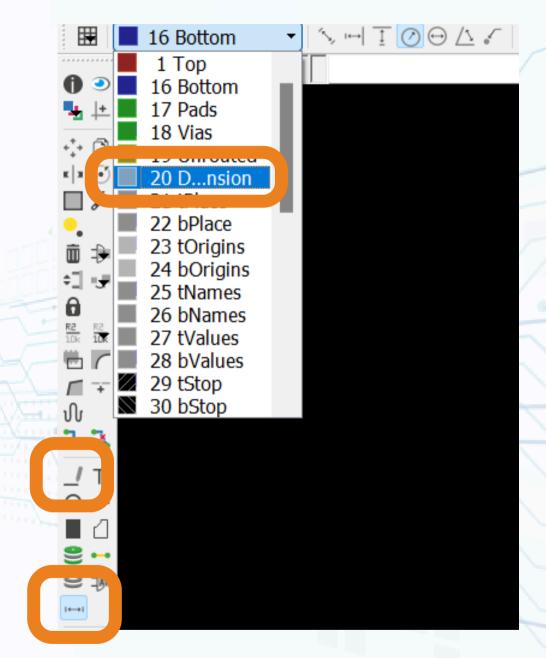
# Mounting Holes



Place the mounting holes on the board corners



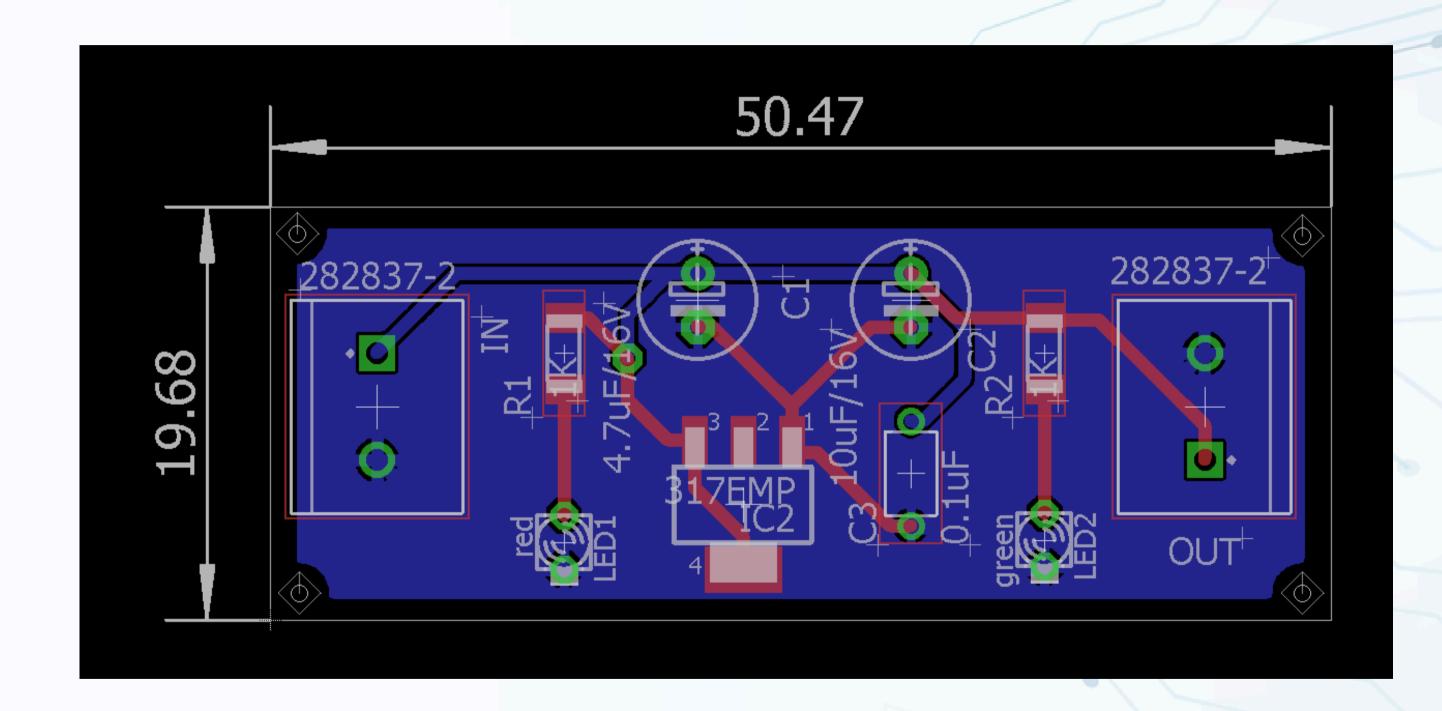
# **Board Dimenstions**



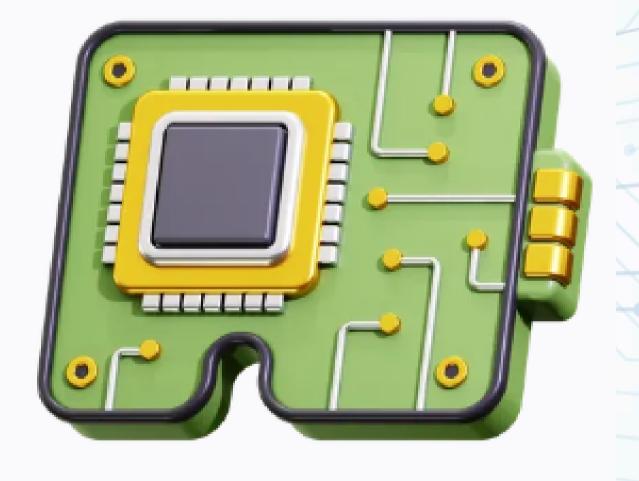
**Select > wire > Dimension** 



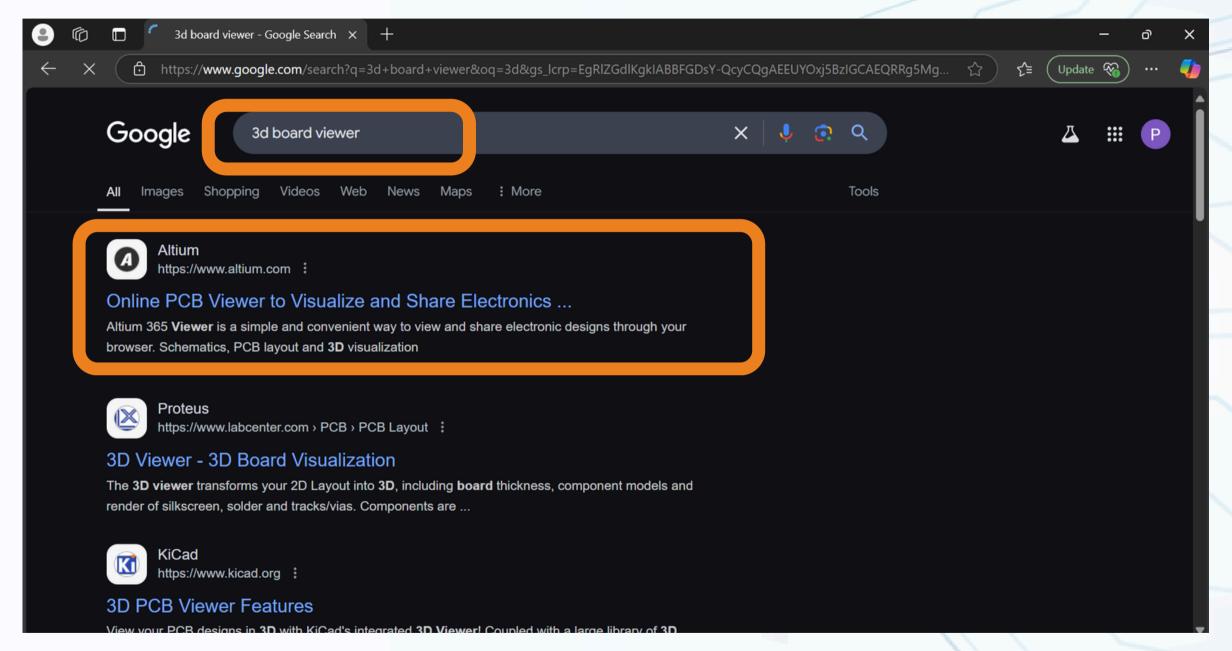
## **Board Dimenstions**





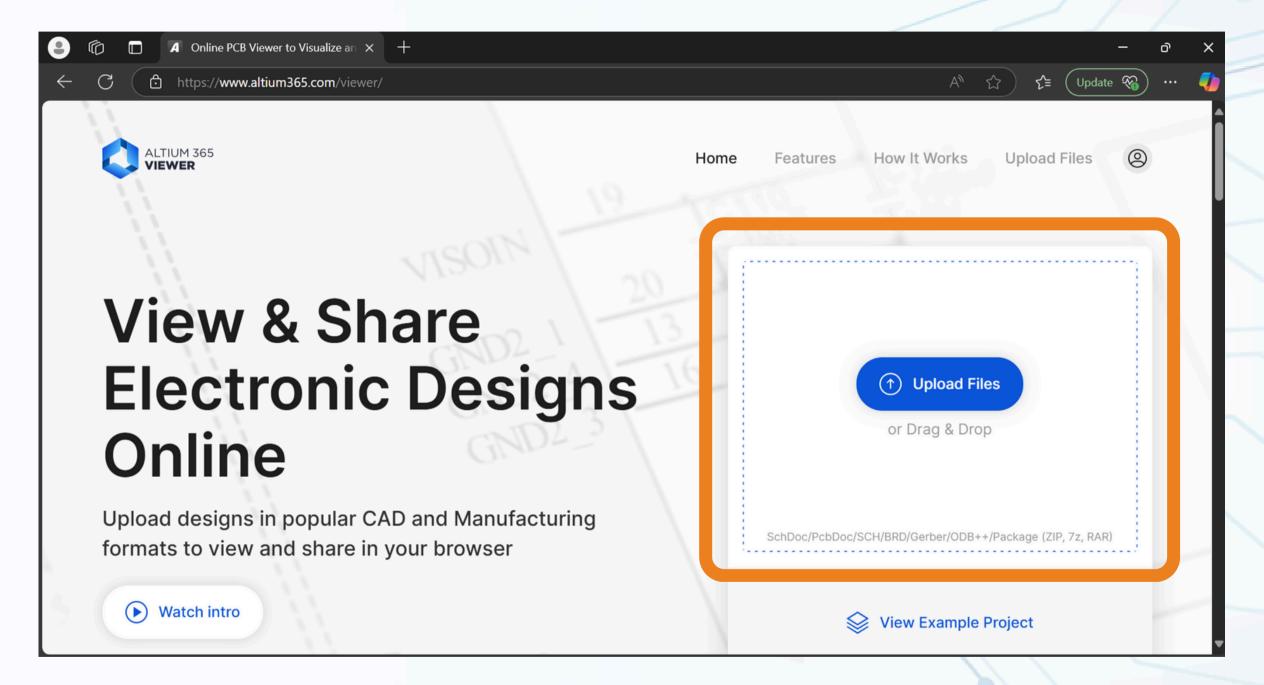






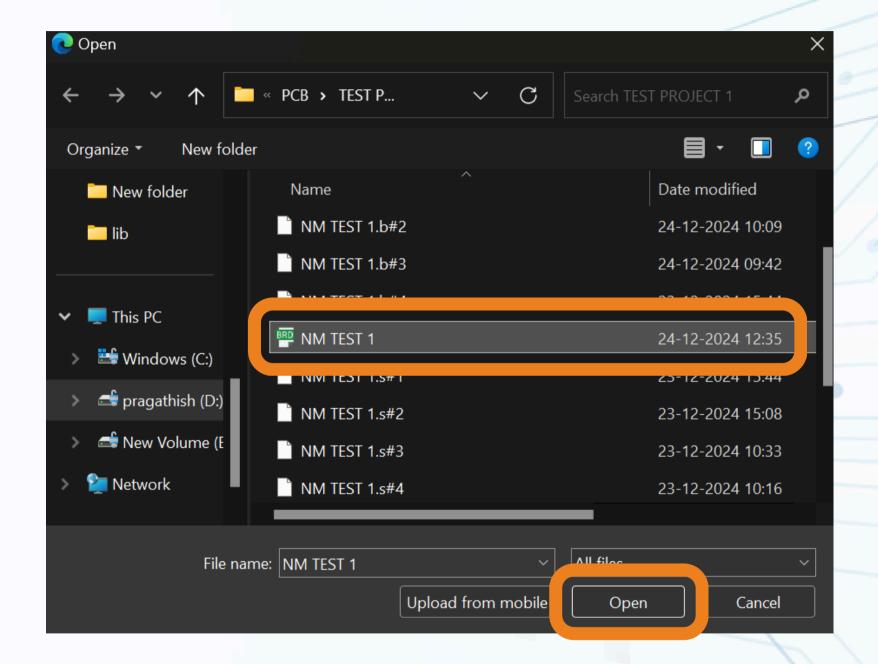
Search > 3D Board Viewer > **Altium 3D Viewer** 





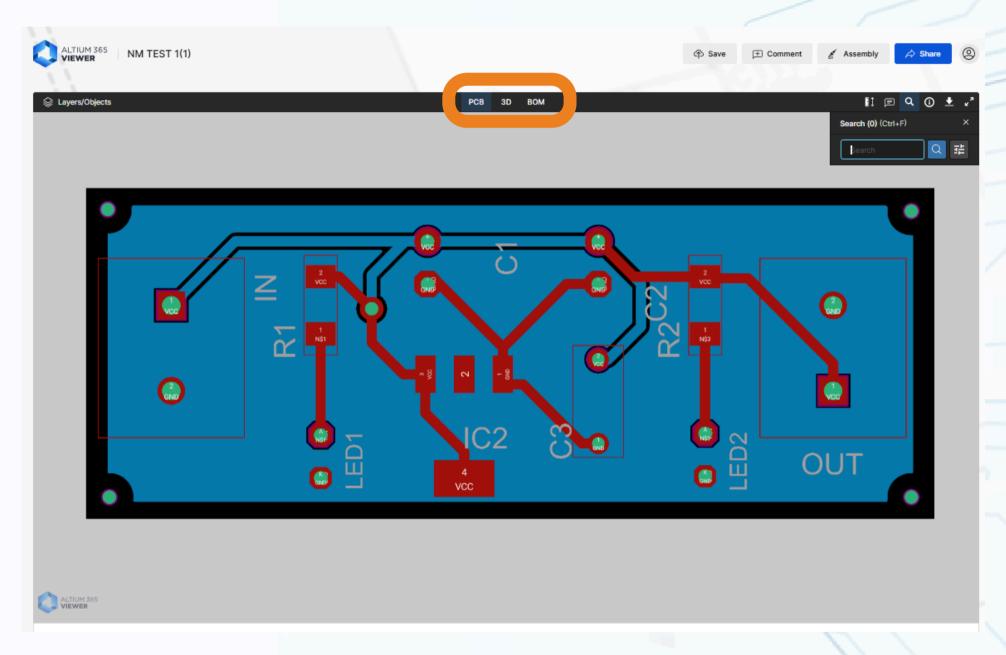
Upload the Board file





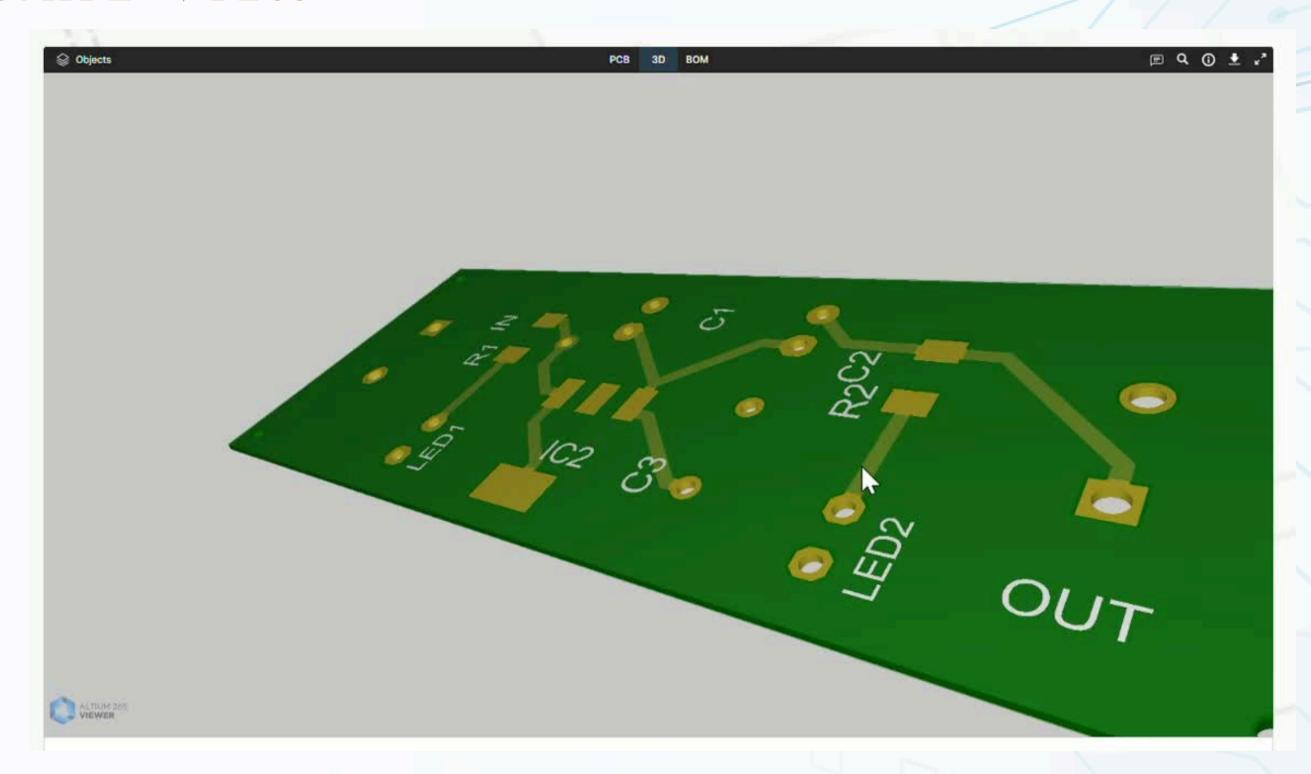
**Select > Board file >** Click Open





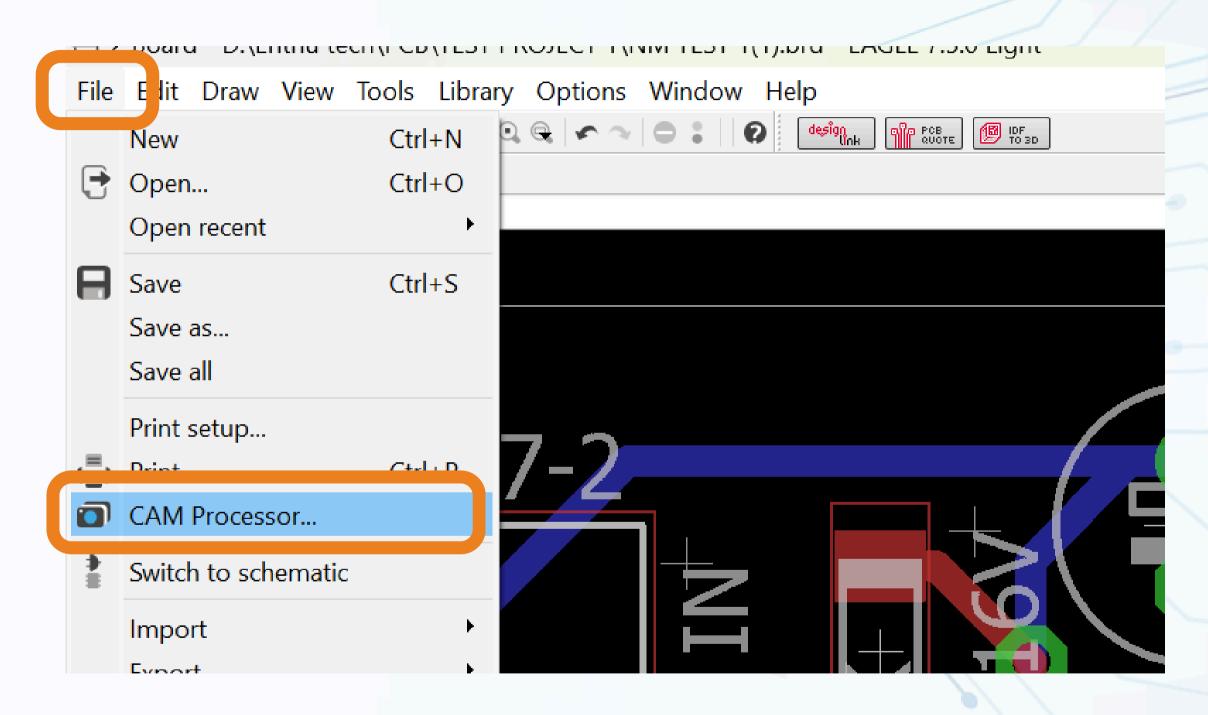
**Seleact > 3D Option** 







### GERDER file GENERATION



**Seleact** > **File** > **CAM Processor** 

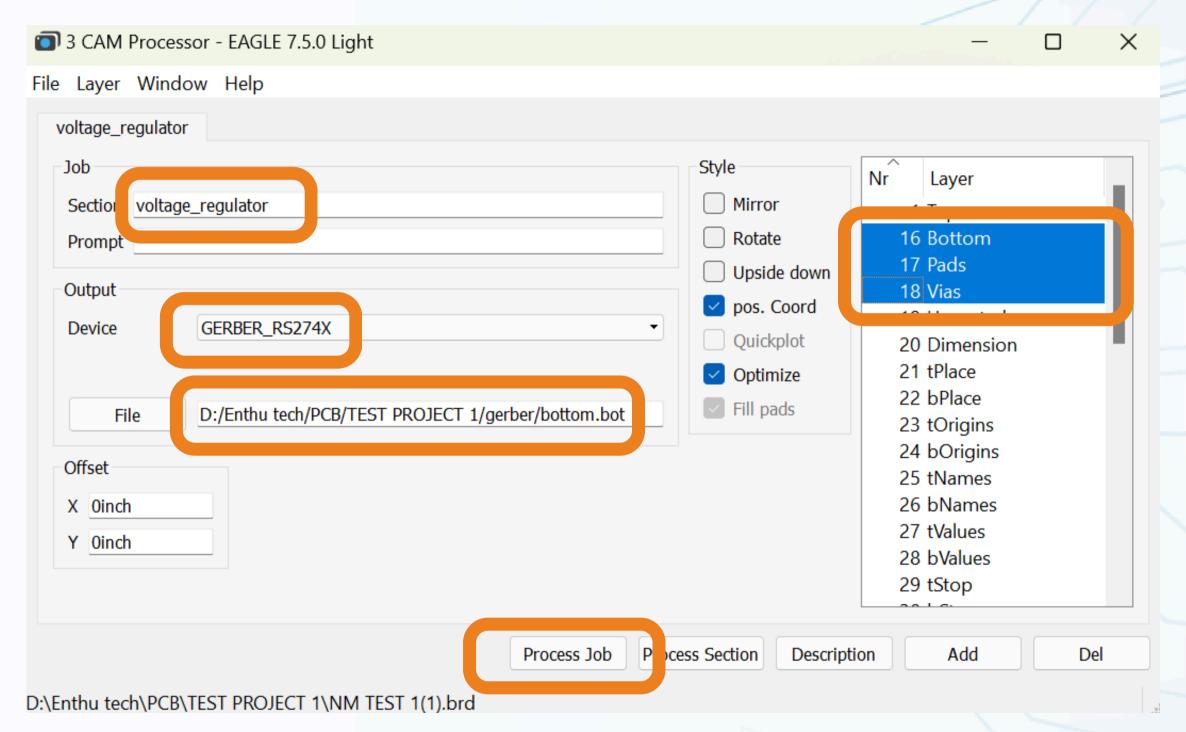


3 CAM Processor - EAGLE 7.5.0 Light		_		×
File Layer Window Help				
voltage_regulator				
Job Section voltage_regulator Prompt  Output Device GERBER_RS274X  File D:/Enthu tech/PCB/TEST PROJECT 1/gerber/top.cmp	Style  Mirror  Rotate  Upside dow  pos. Coord  Quickplot  Optimize  Fill pads	1 Top 16 Bottom 17 Pads 18 Vias 20 Dimension 21 tPlace 22 bPlace 23 tOrigins		
Offset X Oinch Y Oinch Process Job Pr	cess Section Description	24 bOrigins 25 tNames 26 bNames 27 tValues 28 bValues 29 tStop	D	)el

Top layer



### GERDER file GENERATION



**Bottom Layer** 



3 CAM Processor - EAGLE 7.5.0 Light		<b>–</b> (	⊃ ×
File Layer Window Help			
voltage_regulator			
Section voltage_regulator  Prompt  Output  Device GERBER_RS274X  File D:/Enthu tech/PCB/TEST PROJECT 1/gerber/cutting.cut  Offset  X Oinch  Y Oinch	Style  Mirror  Rotate  Upside down  pos. Coord  Quickplot  Optimize  Fill pads	Nr Layer  1 Top 16 Bottom 17 Pads 18 Vias 19 Officulted 20 Dimension 21 tDlace 22 bPlace 23 tOrigins 24 bOrigins 25 tNames 26 bNames 27 tValues 28 bValues 29 tStop	
D:\Enthu tech\PCB\TEST PROJECT 1\NM TEST 1(1).brd	ss Section Descripti	on Add	Del

**Cutting Layer** 



3 CAM Processor - EAGLE 7.5.0 Light				×
voltage_regulator  Job Section voltage_regulator  Prompt	Style  Mirror  Rotate	Nr Layer 37 tTest 38 bTest		
Output Device EXCELLON  File D:/Enthu tech/PCB/TEST PROJECT 1/gerber/drill.ncd	<ul><li>Upside down</li><li>✓ pos. Coord</li><li>Quickplot</li><li>✓ Optimize</li><li>✓ Fill pads</li></ul>	39 tKeepout 40 bKeepout 41 tRestrict 42 bRestrict 44 Drills 45 Holes		
Offset X Oinch Y Oinch		47 Measures 48 Document 49 Reference 51 tDocu 52 bDocu		
D:\Enthu tech\PCB\TEST PROJECT 1\NM TEST 1(1).brd	ess Section Descript	ion Add	Del	

**Drill Layer** 

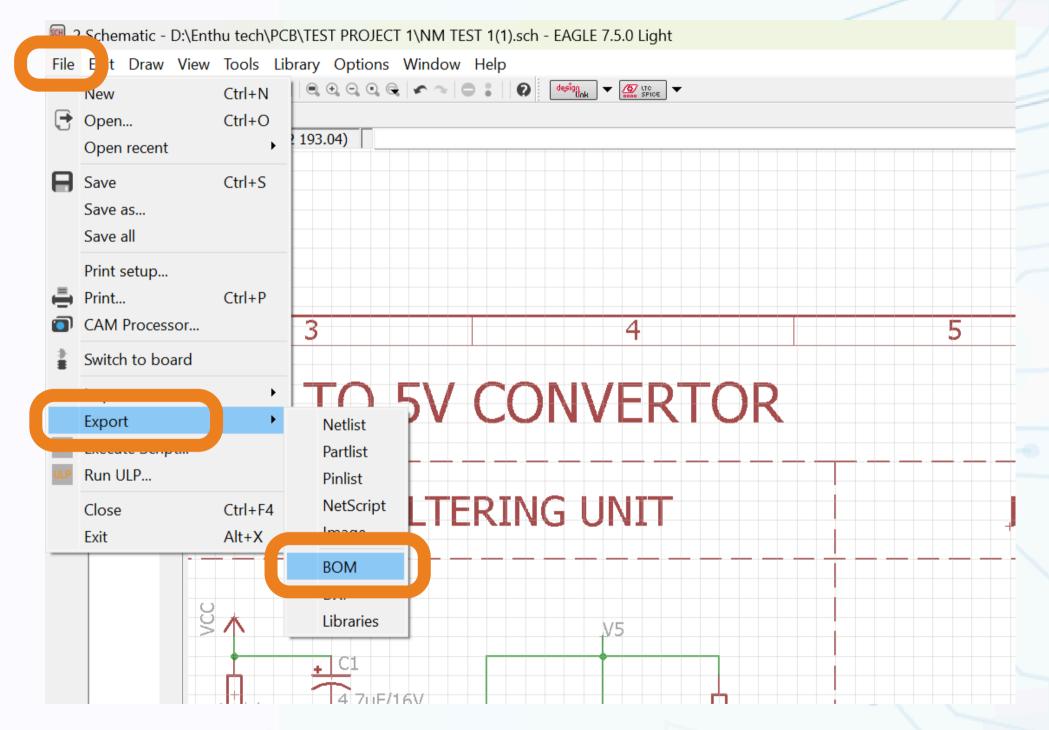


3 CAM Processor - EAGLE 7.5.0 Light			) ×
File Layer Window Help voltage_regulator			
Job Sec on voltage_regulator Prom,  Output Device GERBER_RS274X  File D:/Enthu tech/PCB/TEST PROJECT 1/gerber/silk.plc	Style  Mirror  Rotate  Upside down  pos. Coord  Quickplot  Optimize  Fill pads	Nr Layer  1 Top  16 Bottom  17 Pads  18 Vias  20 Dimension  21 tPlace  23 tOrigins	
Offset X Oinch		25 tNames	
Y Oinch		27 tValues 28 bValues 29 tStop	
Process Job roces D:\Enthu tech\PCB\TEST PROJECT 1\NM TEST 1(1).brd	ss Section Description	on Add	Del

Silk Layer



### Bill Of Materials



Select File > Export > BOM



## Bill Of Materials

