
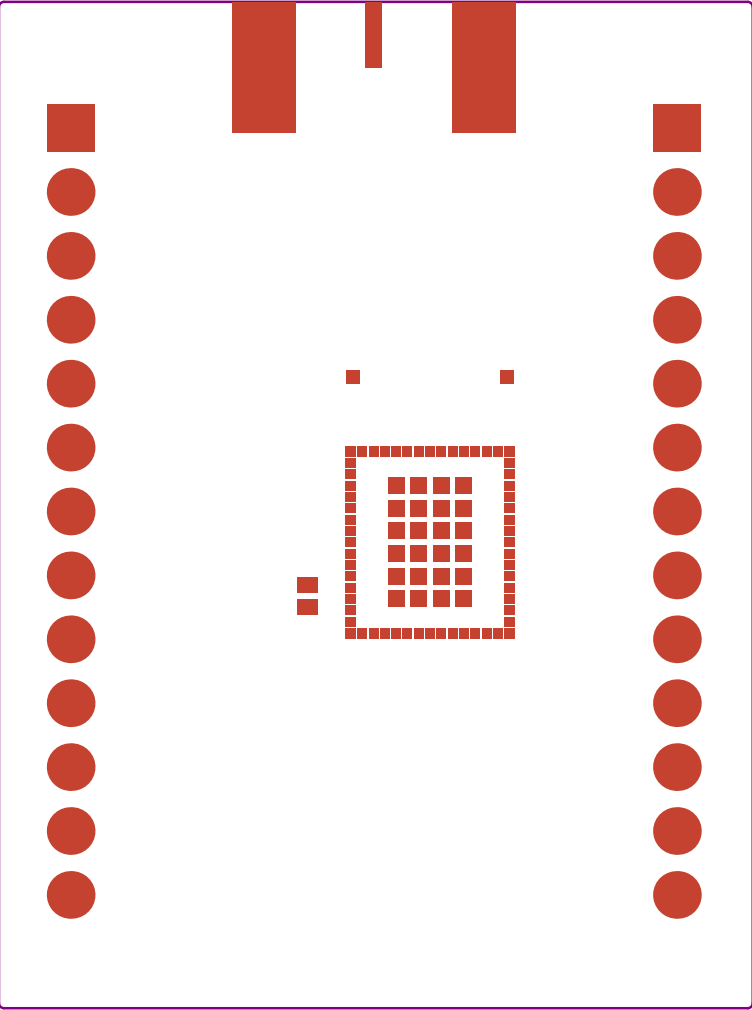
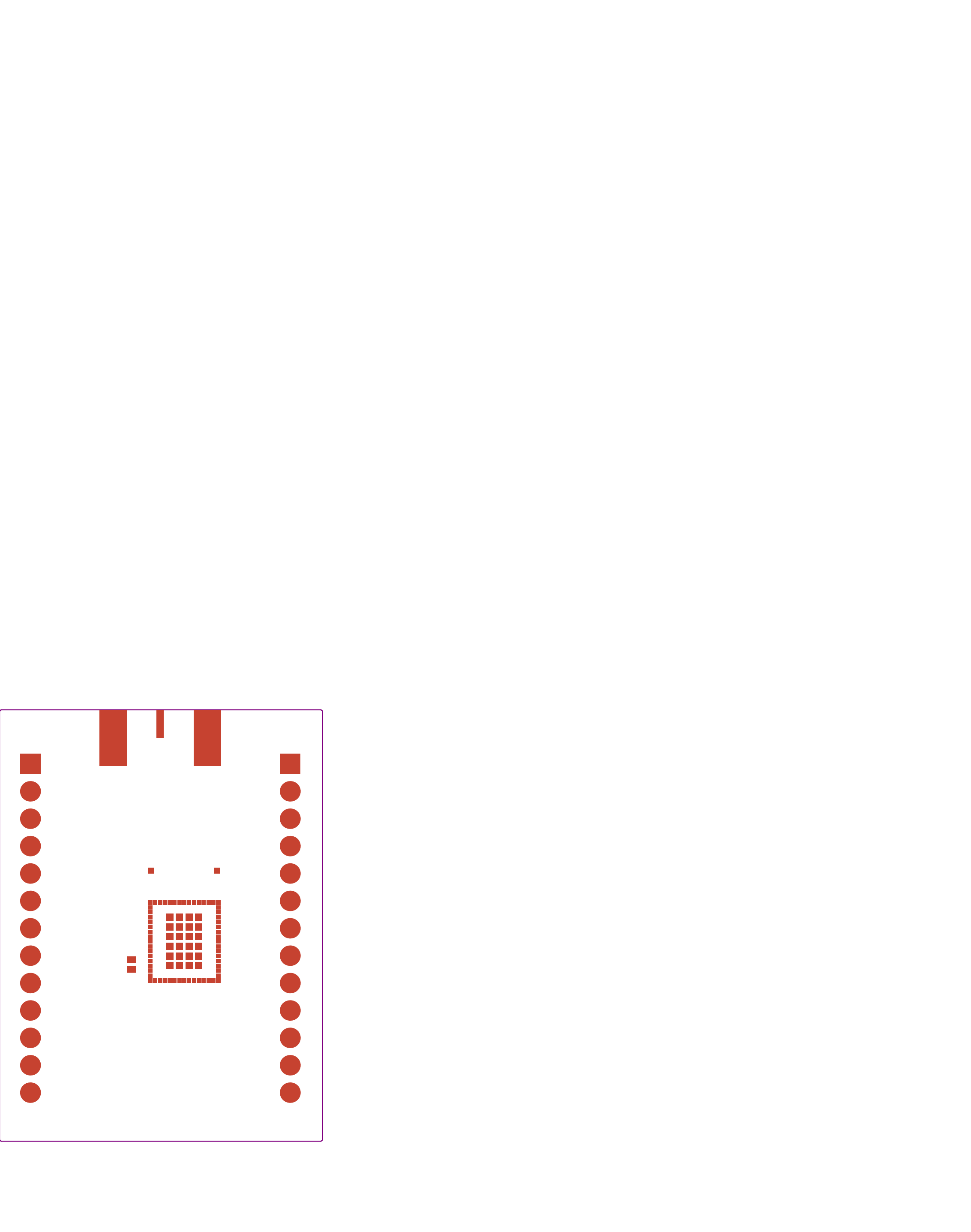

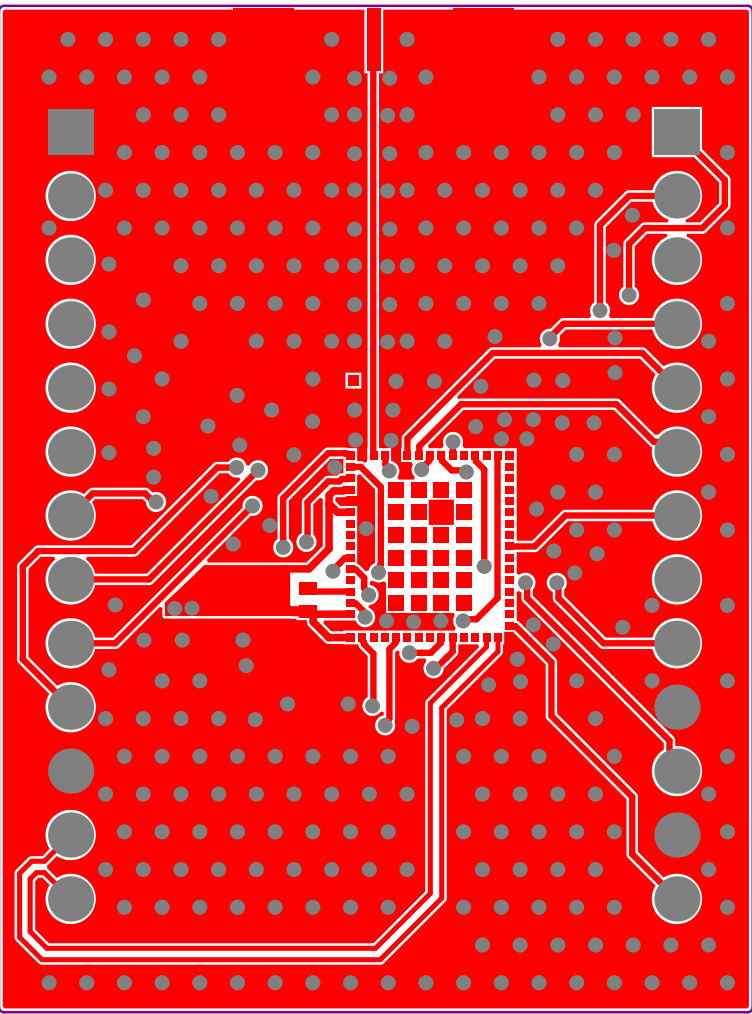



Project: Module_WB5M_SPIDER_ML		
Layer: Top Overlay	Gerber: .GTO	
Variant: [No Variations]	Ref: MB2390	
Date: 15-jan-25	Rev: A	



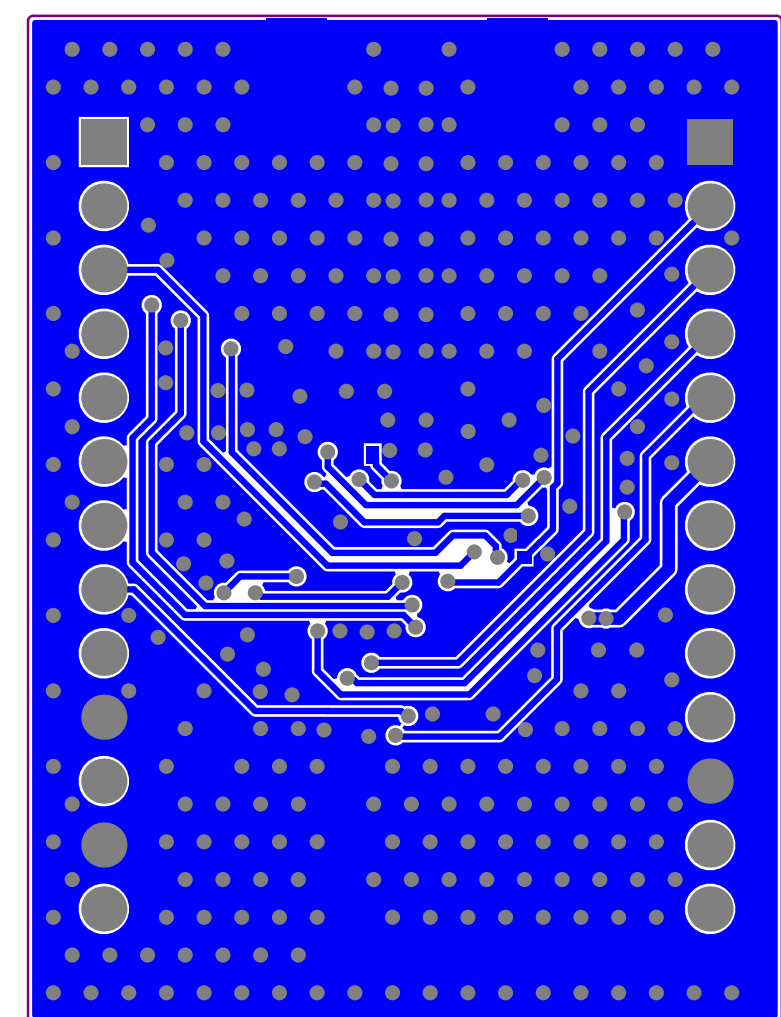
Project: Module_WB5M_SPIDER_ML		
Layer: Top Solder	Gerber: .GTS	
Variant: [No Variations]	Ref: MB2390	
Date: 15-jan-25	Rev: A	



Project: Module_WB5M_SPIDER_ML		
Layer: Top Layer	Gerber: .GTL	
Variant: [No Variations]	Ref: MB2390	
Date: 15-jan-25	Rev: A	

Gerber File List

Gerber files are used to define the physical layout of a printed circuit board (PCB). They are a standard format for communicating PCB design data to manufacturing. The Gerber files are organized into a list, where each entry represents a specific layer or feature of the PCB. The list includes the file name, the layer or feature it represents, and the file format. The Gerber files are typically generated from a PCB design file (e.g., a CAD file) and are used to create the physical PCB. The Gerber files are a critical part of the PCB manufacturing process, as they provide the manufacturing house with the necessary data to produce the PCB accurately. The Gerber files are also used for quality control and verification purposes, ensuring that the manufactured PCB matches the design specifications.



Project: Module_WB5M_SPIDER_ML	
Layer: Bottom Layer	Gerber:.GBL
Variant: [No Variations]	Ref: MB2390
Date: 15-jan-25	Rev: A



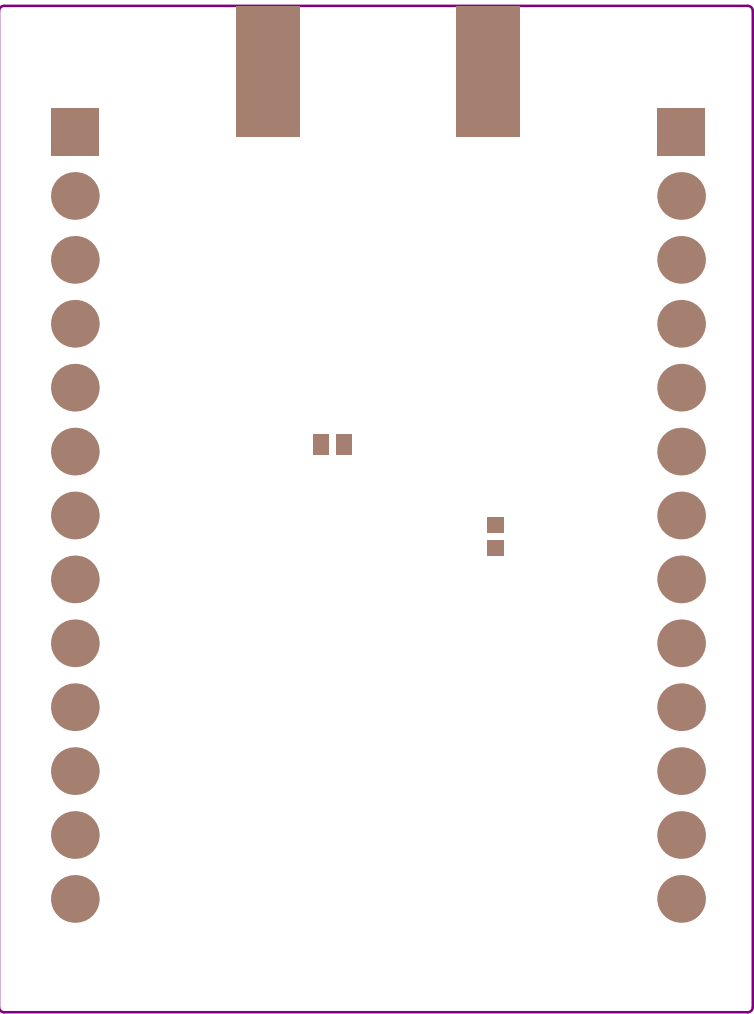
Gerber File List


Project: Module_WB5M_SPIDER_ML
Layer: Bottom Solder
Variant: [No Variations]
Date: 15-jan-25

Gerber:.GBS

Ref: MB2390

Rev: A

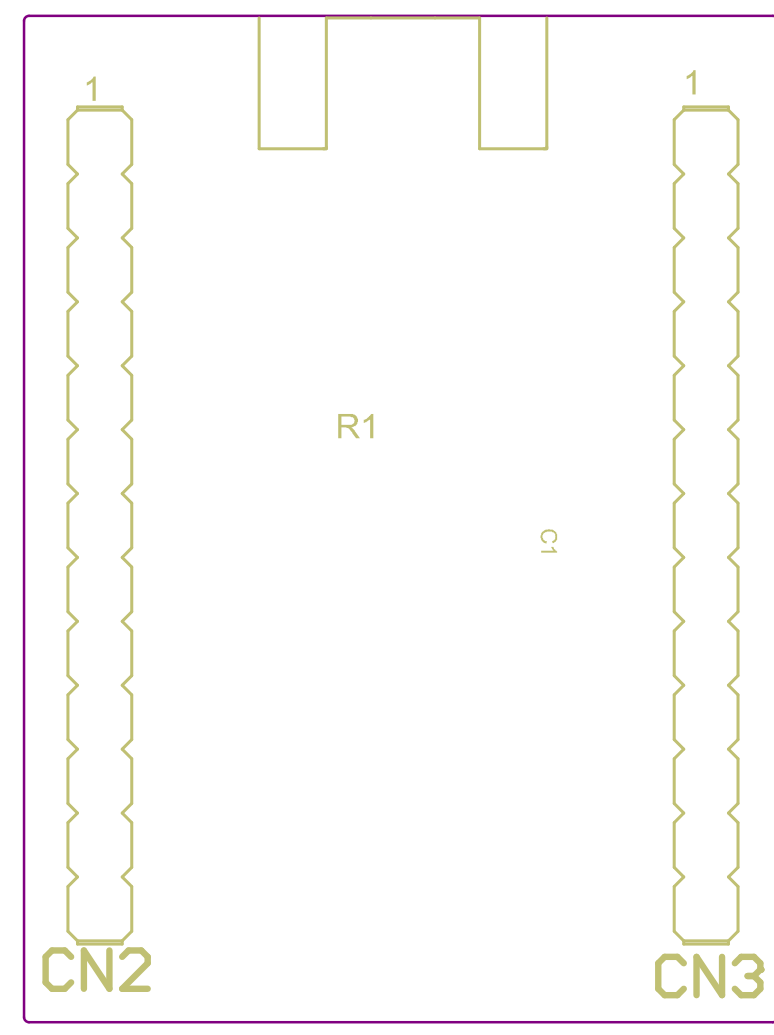


Project: Module_WB5M_SPIDER_ML		
Layer: Bottom Solder	Gerber:.GBS	
Variant: [No Variations]	Ref: MB2390	
Date: 15-jan-25	Rev: A	

Gerber File List

Project: Module_WB5M_SPIDER_ML
Layer: Bottom Overlay
Variant: [No Variations]
Date: 15-jan-25

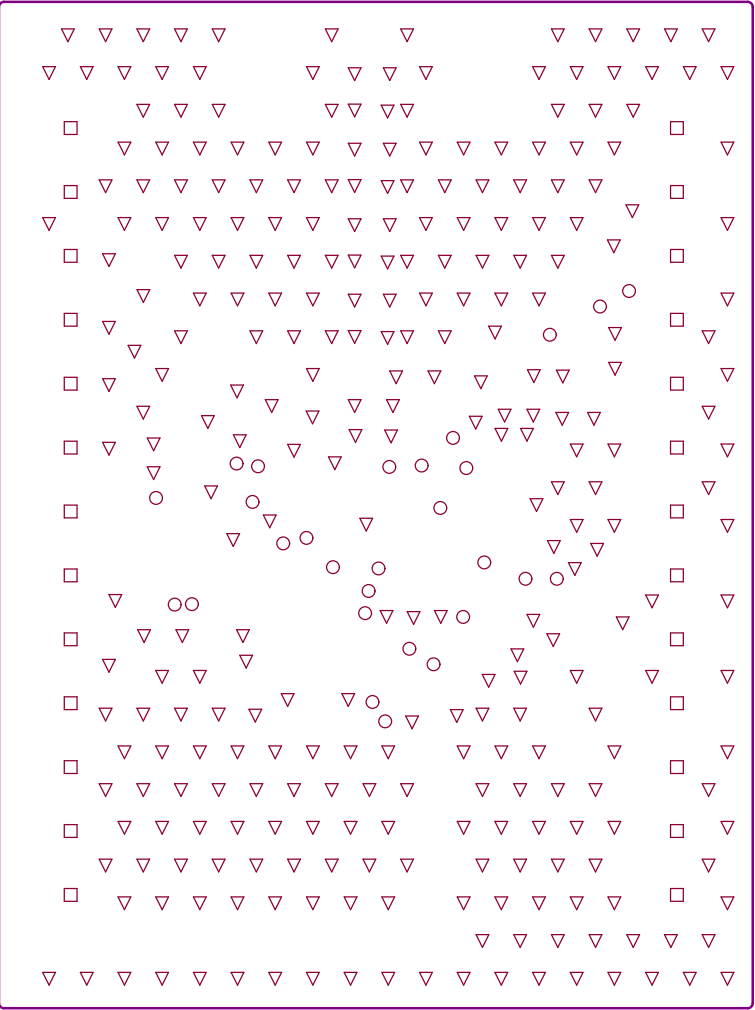
Gerber: .GBO
Ref: MB2390
Rev: A

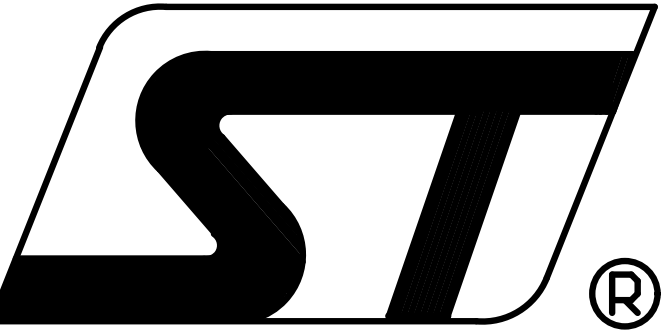


Project: Module_WB5M_SPIDER_ML	
Layer: Bottom Overlay	Gerber: .GBO
Variant: [No Variations]	Ref: MB2390
Date: 15-jan-25	Rev: A



Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad shape	Template	Description	Hole Tolerance (+)	Hole Tolerance (-)
□	26	1.100mm <43.31mil>	PTH	Round	Top Layer - Bottom Layer	Pad	<Mixed>	<Mixed>			
○	28	0.300mm <11.81mil>	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v60h30m0m0<Tel1-2>_564437741	General	0.010mm <0.39mil>	0.020mm <0.78mil>
▽	301	0.300mm <11.81mil>	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v60h30m0m0_564437741			
	355 Total										



Project: Module_WB5M_SPIDER_ML		
Layer: Drill Drawing	Gerber: .DRL	
Variant: [No Variations]	Ref: MB2390	
Date: 15-jan-25	Rev: A	