



RELEASED UNDER THE CREATIVE COMMONS ATTRIBUTION SHARE-ALIKE 4.0 LICENSE
[HTTPS://CREATIVECOMMONS.ORG/LICENSES/BY-SA/4.0/](https://creativecommons.org/licenses/by-sa/4.0/)
DESIGNED BY: L. Buechley, N. Seidle, P. Lewis, T. Klopfenstein
REVISION BY: Elias Santistevan

ORDERING INSTRUCTIONS:			
(ADD THESE COMMENTS WHEN UPLOADING GERBERS.)			
Board Thickness: 0.032" Purple Solder Mask			
		Sheet:	
		File: kicadBoard.kicad_pcb	
		Title:	
		Size: A4	Date:
		KiCad E.D.A. pcbnew (6.0.9)	Rev: Id: 1/1