



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