



Overview

While the BuckPuck is an efficient driver, it will still need to dissipate a small amount of heat. Although no external heatsink is required, using ground plane construction to create at least 1 square inch of copper underneath the BuckPuck is recommended. If a 2-layer PCB is being created, ground plane construction should be used on both layers. 0.1µF decoupling capacitors should be placed across the power input and output pins, as close to the BuckPuck as possible, when the traces to the load or to the power source are more than 6" long. This will also reduce EMI. When placing multiple BuckPucks, there should be at least .25" between buckpucks to allow for heat dissipation, especially when using BuckPucks over 500mA.

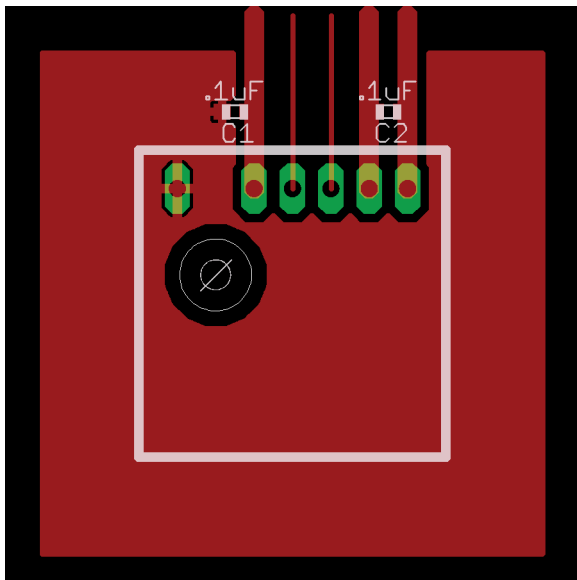


Figure 1.

Example PCB layout with ground plane construction and decoupling capacitors (TOP view)

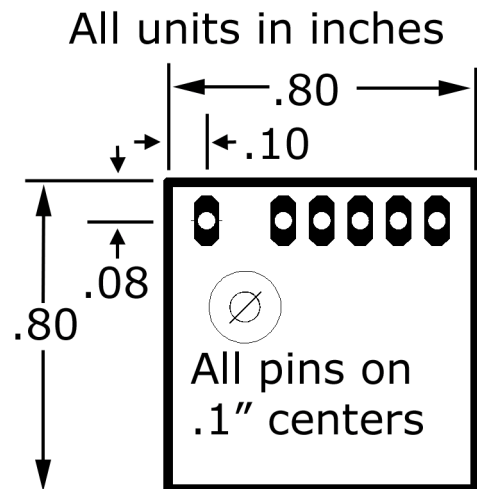


Figure 2.

PCB Footprint (TOP view)

When using I version BuckPucks (internal 40-110% current adjustment), a 3/16" hole should be placed as indicated in Figure 3. This will allow in-circuit adjustment of the current.

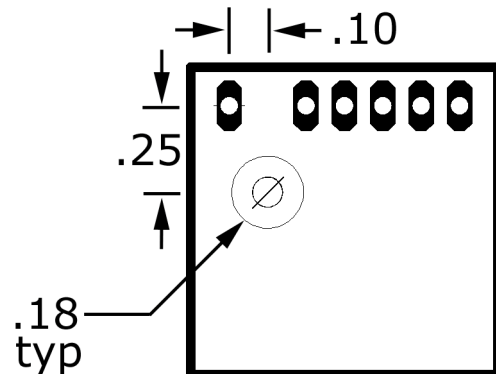


Figure 3.

Adjustment hole location (TOP view)

