

2.2 Layout Guidelines

The layout is optimized for excellent electrical and thermal performance. However, when designing a PCB, use the following general practices:

- Position the input and output capacitors close to the IC. Both input and output currents can be discontinuous in a buck-boost converter; therefore, it is important to place both the input and output capacitors as close as possible to reduce loop areas. This reduces radiated EMI emissions.
- Keep the ground connections of the input and output capacitors as short as possible and placed on the component layer to avoid problems caused by high-switching currents flowing through PCB vias. If it is necessary to use the vias, use multiple vias to minimize the effective trace inductance.
- It is strongly advised that the second layer be a clean GND to mitigate problems that arise from long GND traces and subsequent parasitic inductive components. Also, a clean GND shields the intermediate layers from high power traces on the top layer.
- After placing short input and output loops, place an inductor as close as possible to the IC. While being cautious of any EMI concerns, ensure that the switch node traces (from LX1 and LX2 to the inductor) are short and wide.
- Route the EN, VSEL, SDA, and SCL traces away from high energy and high dV/dt traces to prevent mistriggering. These traces can be routed through intermediate layers.

3. Ordering Information

Part Number	Description
RTKA236105DE0010BU	DRV package evaluation board

CONFIDENTIAL
IN REVIEW

4. Revision History

Revision	Date	Description
1.01	Nov 14, 2024	Updated input quiescent and Shutdown current
1.00	Sep 25, 2024	Initial release