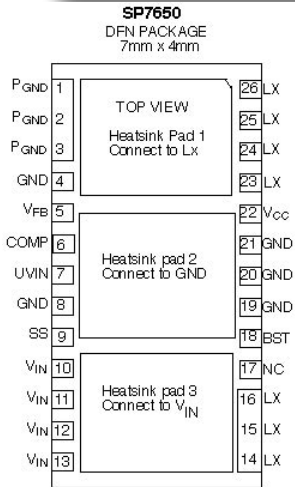


# Thermal Resistance on SP765X Devices by Brian Kennedy



The SP765X series of devices is a family of synchronous step-down switching regulators optimized for high efficiency, with industry leading power density. The parts are designed to be especially attractive for dual supply, 12V or 24V distributed power systems step down with 5V used to power the controller. This lower V<sub>CC</sub> voltage minimizes power dissipation in the part and is used to drive the top switch. The SP765x family is designed to provide a fully integrated buck regulator solution using a fixed frequency, PWM voltage mode architecture. Protection features include UVLO, thermal shutdown and output short circuit protection. The SP765X are available in the space saving and robust DFN package. This application note explains the thermal resistance characteristics of the family of devices and the implications for usage.

Available in Lead Free Packaging

### SP765X Thermal Resistance

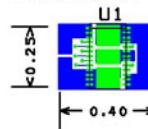
**4 Layer Board:**  
 Top Layer 3oz copper  
 GND Layer 1oz copper  
 Power Layer 1oz copper  
 Bottom Layer 3oz copper

**Minimum Footprint: 44°C/W**  
 Top Layer: 0.1 sq in.  
 No Vias to other 3 Layers

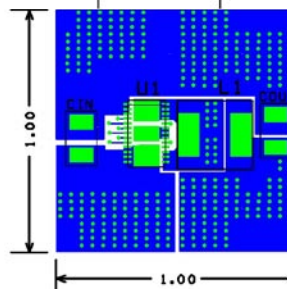
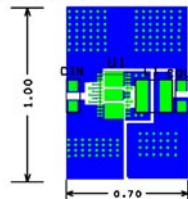
**Medium Footprint: 36°C/W**  
 Top Layer: 0.7 sq in.  
 Vias to other 3 Layers

**Maximum Footprint: 36°C/W**  
 Top Layer: 1.0 sq in.  
 Vias to other 3 Layers

MINIMUM FOOTPRINT = 0.10 SQ IN.



MEDIUM FOOTPRINT = 0.70 SQ IN.



MAXIMUM FOOTPRINT = 1.0 SQ IN.

## SP765X Thermal Resistance Application Note

The SP765X family has been tested with a variety of footprint layouts along with different copper area and thermal resistance has been measured. The layouts were done on 4 layer FR4 PCB with the top and bottom layers using 3oz copper and the power and ground layers using 1oz copper.

For the Minimum footprint, only about 0.1 sq in. of 3oz copper was used on the top or footprint layer, and this layer had no vias to connect to the 3 other layers. For the Medium footprint, about 0.7 sq in of 3oz copper was used on the top layer, but vias were used to connect to the other 3 layers. For the Maximum footprint, about 1.0 sq in. of 3oz copper was used on the top layer and many vias were used to connect to the 3 other layers.

The results show that only about 0.7sq in. of 3oz copper on the top layer and vias connecting to the 3 other layers are needed to get the best thermal resistance of 36°C/W. Adding area on the top beyond the 0.7 sq in. did not reduce thermal resistance.

Using a minimum of 0.1 sq in. of 3oz copper on the top layer with no vias connecting to the 3 other layers produced a thermal resistance of 44°C/W. This value of 44°C/W is only 22% more than the best thermal resistance of 36°C/W. This indicates that a minimum footprint of 0.1sq in. if used on a 4 layer board can produce 44°C/W thermal resistance, which is still very good if used in a space constrained design.

The following page shows the footprint layouts from an ORCAD file. The thermal data was taken for still air, not with forced air. If forced air is used, some improvement in thermal resistance would be seen.

---

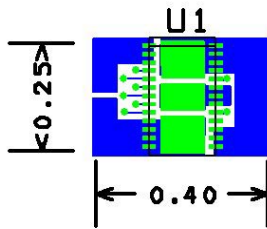
ANALOG EXCELLENCE



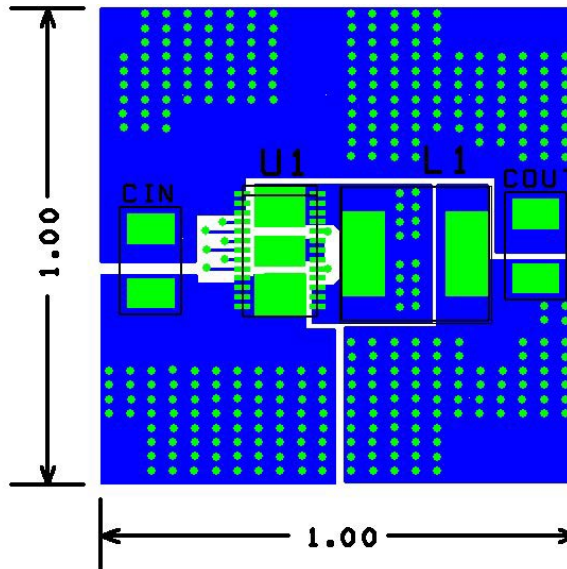
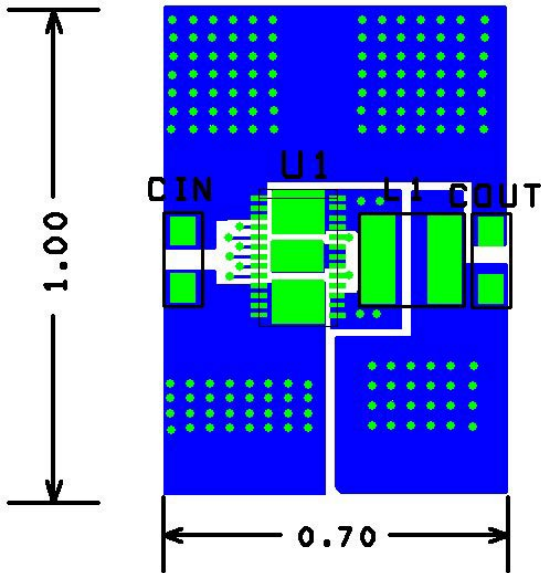
**Sipex Corporation**  
**Headquarters and**  
**Sales Office**  
233 South Hillview Drive  
Milpitas, CA 95035  
TEL: (408) 934-7500  
FAX: (408) 935-7600

Sipex Corporation reserves the right to make changes to any products described herein. Sipex does not assume any liability arising out of the application or use of any product or circuit described herein; neither does it convey any license under its patent rights nor the rights of others.

**MINIMUM FOOTPRINT = 0.10 SQ IN.**



**MEDIUM FOOTPRINT = 0.70 SQ IN.**



**MAXIMUM FOOTPRINT = 1.0 SQ IN.**