

The Golden Rule of Board Layout for SMPS

One factor tops all others in optimizing board layout, and it has to do with designing traces, especially in terms of their length.

This article explains the basis for achieving an optimized board layout, a critical aspect in the design of switch-mode power supplies (SMPS). A good layout ensures stable functioning of the switching regulator and minimizes radiated interference as well as conducted interference (EMI)—widely known by electronics developers. However, what’s not generally known is how an optimized board layout for a switch-mode power supply should look.

Figure 1 shows the circuit of an **LT8640S** evaluation board. It’s a step-down (buck) switching regulator that can tolerate input voltages of up to 42 V and is designed for output currents up to 6 A. All components have been placed very compactly.

Placing the components as closely together as possible on the board is a general recommendation. Though this statement isn’t false, it’s also not particularly suitable if the goal is to obtain an optimized board layout. In Figure 1, quite a few (11) passive components surround the switching regulator IC.

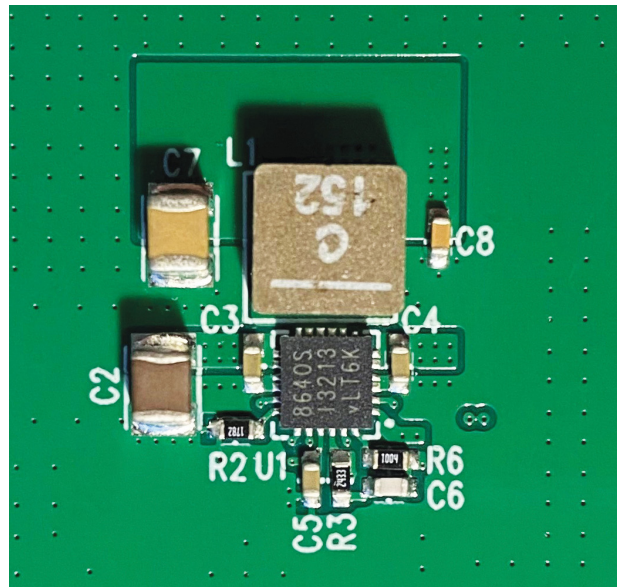
Which of these passive components have priority over the others in the placement and why?

In a switching-regulator PCB design, the most important rule is to route the traces that carry high switched currents to be as short as possible. Successfully implementing this rule will properly address a large part of the board layout for a switching regulator.

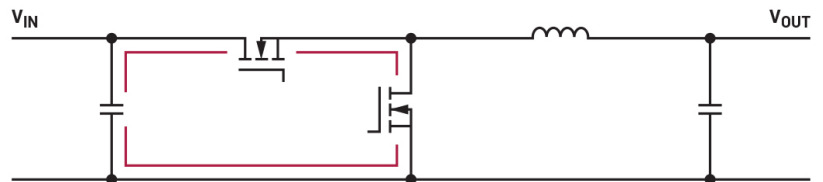
What’s the easiest way to implement this golden rule in the board layout?

The first step is to find out which paths in a switching-regulator topology are critical. In these paths, the current flow changes with the switch transitions.

Figure 2 shows a typical circuit for a step-down converter (buck topology). The critical paths are shown in red. They are connecting lines in which either the full current or no current flows, depending on the states of the power switch-



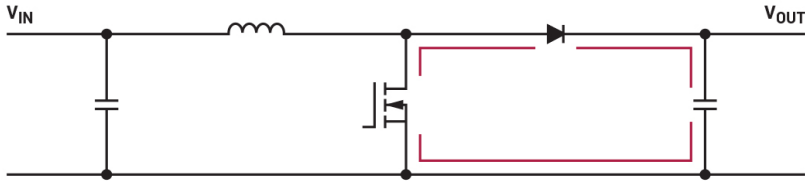
1. Shown is the board of an **LT8640S** switching regulator with closely spaced components, thus creating a compact board layout.



2. This is a schematic of a step-down switching regulator and paths with rapidly changing currents shown in red.

es. These paths should be as short as possible. For a buck converter, the input capacitor should be situated as close as possible to the V_{IN} pin and GND pin of the switching regulator IC.

Figure 3 illustrates a basic schematic diagram of a circuit with a boost topology. Here, a low voltage is converted to a higher voltage. Once again, the current paths in which the



3. Here's a schematic of a step-up switching regulator and paths with rapidly changing currents shown in red.

current flow changes with switching of the power switches (shown in red).

Interestingly, the placement of the input capacitor isn't critical at all; rather, it's the placement of the output capacitor. It must be as close as possible to the flyback diode (or the high-side switch) as well as to the ground connection of the low-side switch.

After that, any other switching-regulator topology can be examined to yield information on how the current flow changes when the power switches are switched. The classic method involves printing out the circuit and drawing the current flow, using three differently colored pens.

One color is used to indicate the current flow during the on-time—that is, when the power switch is conducting current. The second color shows the current flow during the off-time—that is, when the power switch is switched off. And finally, the third color is used for all paths marked either only in the first color or only in the second color. The critical path, in which the current flow changes with the switching of the power switches, can then be clearly identified.

Inexperienced circuit designers often consider the board layout for a switching regulator to be black magic. The most important rule is to design the traces, in which the current flow changes with the switch transitions, to be as short and compact as possible. This can be explained easily, follows logical relationships, and is the basis for an optimized board layout in a switch-mode power supply design.

Frederik Dostal is a power-management expert with more than 20 years of experience in this industry. After his studies of microelectronics at the University of Erlangen, Germany, he joined National Semiconductor in 2001, where he worked as a field applications engineer, gaining a lot of experience in implementing power-management solutions in customer projects. During his time at National, he also spent four years in Phoenix, Arizona (USA), working on switch-mode power supplies as an applications engineer.

In 2009, he joined Analog Devices, where since then he held a variety of positions working for the product line and European technical support, and currently brings in his broad design and application knowledge as a power-management expert. Frederik works in the ADI office in Munich, Germany.