

Timothy Hegarty

Today, in an era of intense competition, the challenge for product designers is to stay ahead of the pack and not merely walk in lockstep with it. This ups the ante for system designers to innovate with differentiated products.

One important way to innovate is with [high-density](#) designs. In the push for smaller-footprint solutions, power system designers are now focusing on the question of power density – the output power per unit of area or volume of a power converter circuit.

The most visible example of DC/DC converter printed circuit board (PCB) layout for high density relates to power stage component placement and routing. Careful layout can coincide with better [switching performance](#), lower component temperatures and reduced electromagnetic interference (EMI) signatures. Consider the power stage layout and schematic in [Figure 1](#).

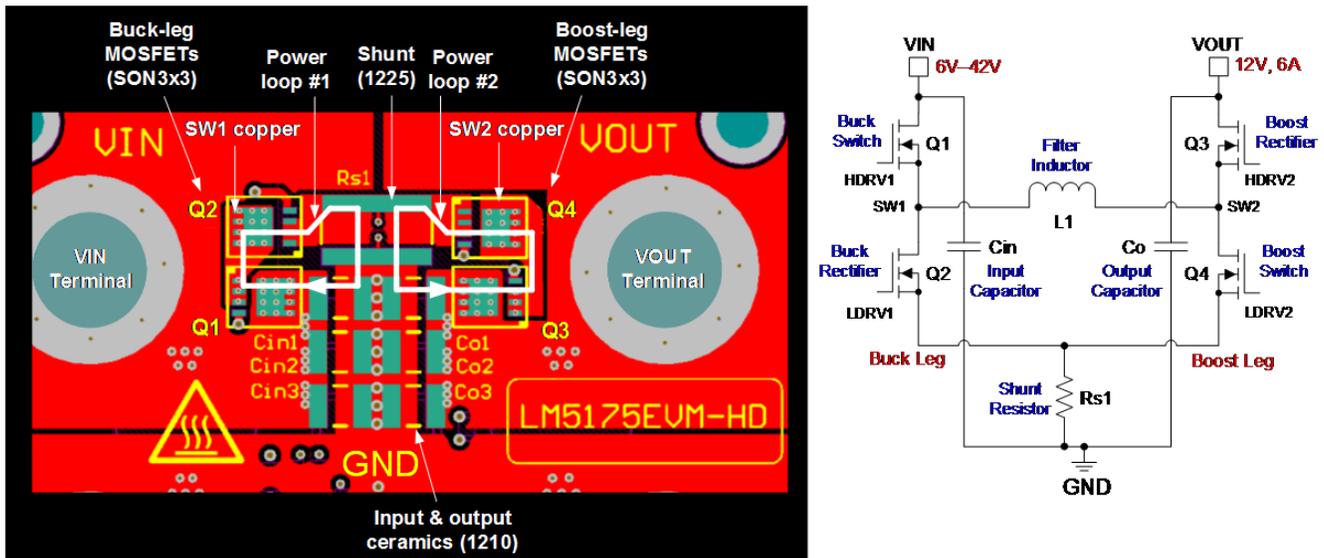


Figure 1. Four-switch Buck-boost Converter Power Stage Layout and Schematic

As I see it, these are the challenges when designing high-density DC/DC converters:

- Component technology.** Advancements in component technology are key to reducing overall power dissipation, particularly at higher switching frequencies vital for filter passive component [size reduction](#). For example, power MOSFETs have seen consistent advances in silicon and packaging, most notably with the introduction of gallium nitride (GaN) power devices with very low parasitics. Meanwhile, magnetic component performance has advanced independently, albeit at a rate arguably lagging that of power semiconductors. Prudent layout of the control IC – with integrated, adaptive gate [drivers](#) close to the MOSFETs – in many cases obviates the need for switch-node voltage slew-rate adjustment using power-dissipating snubber or gate resistor components.
- Thermal design.** While a high-density layout is generally positive for conversion efficiency, it may create a thermal performance bottleneck. The same power dissipation in a smaller footprint becomes untenable. Increased component temperatures amplify concerns of higher failure rates and reliability. Lower-profile power MOSFETs placed on the top side of the PCB – not airflow-shadowed by taller components like the inductor and [electrolytic](#) capacitors – help thermal performance with convective airflow. For the converter

in [Figure 1](#), the inductor and electrolytics are purposely located on the bottom side of the multilayer PCB, because they would impede heat transfer if placed on the top.

- **EMI performance.** EMI regulatory compliance is a key milestone in a product's design cycle. A high-density design typically has little space available for EMI filtering. However, tight layout improves [radiated emissions](#) as well as immunity to incoming disturbances. Two essential steps are to minimize loop areas containing high di/dt currents (see the white current paths in [Figure 1](#)) and reduce surface areas with high dv/dt voltages (see the SW1 and SW2 copper polygons in [Figure 1](#)).
- **High-density PCB design flow.** Clearly, it's important for power system designers to develop and hone their PCB design skills. Even though the layout duties are often delegated to layout specialists, engineers still bear the ultimate responsibility to [review](#) the design and sign off on it.

With these challenges in mind, I recently wrote a three-part series for EDN titled "[DC/DC Converter PCB Layout](#)" that delves into PCB layout in detail. It includes a list of PCB layout guidelines structured as a checklist to aid designers during layout. The essential steps in the PCB design flow for DC/DC converters are:

1. Choose the PCB structure and stack-up specification.
2. Identify the high di/dt current loops and high dv/dt voltage nodes from the schematic.
3. Perform power stage component layout and placement.
4. Place the control IC and complete the control section layout.
5. Perform critical trace routing, including MOSFET gate drive, current sense, and output-voltage feedback.
6. Design the power and GND planes.

In [part 2](#) of this blog series, I'll unveil a high-density buck-converter layout – a 25A point-of-load design in a 20mm-by-11mm footprint.

Additional Resources:

- Read parts [1](#), [2](#) and [3](#) of "DC/DC Converter PCB Layout" on EDN.
- Watch a [video](#) featuring the high-density [evaluation module \(EVM\)](#) for the [LM5175](#) four-switch synchronous buck-boost controller.
- Study the design equations for a [buck-boost](#) converter.
- Examine the schematic, layout and test data of these high-density [buck-boost designs](#) from the [PowerLab™](#) reference design library:
 - [Four-Switch Buck-Boost Converter with Adjustable Output Voltage for Military Radio Applications TI Designs reference design.](#)
 - [LED PWM Dimming Reference Design Using High Efficiency Buck-Boost Converter TI Designs reference design.](#)
- Start a design now with [WEBENCH® Power Designer](#).

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to [TI's Terms of Sale](#) or other applicable terms available either on [ti.com](https://www.ti.com) or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

TI objects to and rejects any additional or different terms you may have proposed.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2023, Texas Instruments Incorporated