Place DC/DC Regulators on Backside of PCB to Clear Room on the Topside for Digital ICs

By Afshin Odabaee, Power Module Business Manager
Sam Young, Applications Engineer
Linear Technology

Faster IC-to-IC data communication benefits us all with more capable systems from higher quality video streaming to more capable networking equipment. This flow of more data in shorter time periods exerts pressure on system designers to tackle the everlasting challenge of data quality by minimizing unintended induced errors especially those stemming from PCB layouts, trace impedance, cross talk, EMI and other factors. So, it makes sense to shorten distances between IC that are in constant communication with each other and add more layers to the PCB which constitute a denser circuit boards. Doing so starts another battle: the fight to fit as many ICs and connectors on top of PCB without increasing the cost of the end-product. The cost is in large percentage associated with the size and thickness of the PCB. For smaller size boards and fewer PCB layers, sometimes the question becomes which components need to be removed or which functions must be eliminated to reduce the manufacturing cost albeit, the resale value of the device may be in jeopardy. It's a fight to fit performance and features.

The top side of system boards such as PCIe cards are densely populated with FPGA/ASIC/µP, transceivers, connectors, memory ICs and DC/DC regulator circuits of various heights up to a few centimeters. However, the back side often has a height restriction where only components with less than 2.3mm tall packages such as capacitors are allowed and much of real estate is unoccupied. What if this empty PCB area could be used by DC/DC regulator circuits uncluttering the top side allowing use of more memory ICs, for example, adding to the system's capability?

The top side of system boards such as PCIe cards are densely populated with FPGA/ASIC/uP, transceivers, connectors, memory ICs and DC/DC regulator circuits of various heights up to a few centimeters. However, the back side often has a height restriction where only components with less than 2.3mm tall packages such as capacitors are allowed and much of real estate is unoccupuied. What if this empty PCB area could be used by DC/DC regulator circuits uncluttering the top side allowing use of more memory ICs, for example, adding to the system's capability?

The LTM4622 is a dual 2.5A or by tying the outputs in current sharing, single 5A, step-down µModule® (power module) regulator in 6.25mm × 6.25mm × 1.82mm ultrathin LGA package. At nearly the height of a soldered down 0805 case size capacitor, its ultralow height allows it to be mounted on the backside of a PCB, freeing space on the topside for digital ICs. The thin profile allows it to meet demanding height restrictions such as those required by PCIe and Advanced Mezzanine Cards in embedded computing systems.

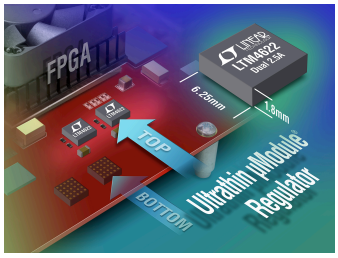

Figure 1. LTM4622 Tiny Dual Supply in Ultrathin Package

Mirrored Layout for Higher Power, Smaller PCB

The LTM4622's pin configuration is organized symmetrically, so for higher current applications where two LTM4622s can be parallel, one device can be on the top side of PCB and the other mirrored on the bottom side, minimizing PCB area while increasing output power and power density.

Flexible Dual Supply in a Simple < 0.5cm^2 Footprint

The LTM4622 has a wide input voltage range of 3.6V to 20V, ideal and it can also be configured to operate down to 3.1V for operation from a 3.3V input supply. It regulates two voltages for a compact multi-rail solution where each output can supply up to 2.5A (3A peak) and is capable of precisely regulating between 0.6V to 5.5V to with 1.5% maximum total DC output voltage error over line, load and temperature. For higher output current up to 5A, simply tie the outputs for current sharing.

The LTM4622 requires only 3 ceramic capacitors and 2 resistors to complete a solution occupying less than 1cm^2 single-sided or 0.5cm^2 on a double-sided PCB.

Figure 2 shows the LTM4622 circuit in a typical dual output application, also illustrating its compact solution size. Efficiency and power loss for the circuit operating at a 12V input are shown in Figure 3.

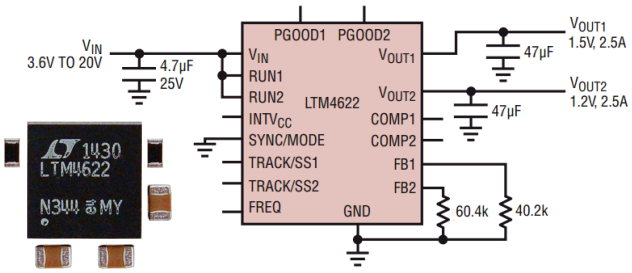

Figure 2. Typical Application: 1.5V/2.5A, 1.2V/2.5A Dual

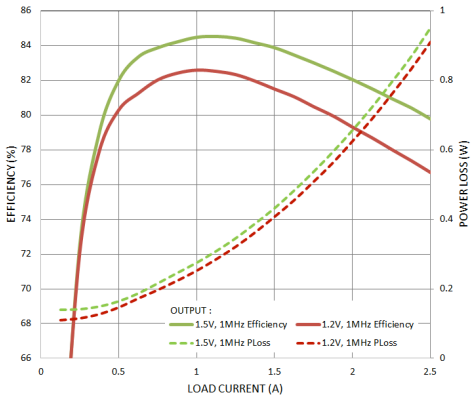

Figure 3. Efficiency, Loss at 12V Input (Fig. 2)

Reliable High Performance Regulation

The LTM4622 has a controlled on-time current-mode architecture for a fast transient with good loop stability over a wide voltage range. It provides system protection such as short-circuit, overvoltage and overtemperature as well as ensures monotonic output voltage ramping with tracking, soft-start and the ability to start into a pre-biased output. It also has no limitation on input supply slew rate.

Figures 4 and 5 demonstrate the fast transient and pre-bias start-up performances for the 1.5V output rail of Figure 2 circuit.

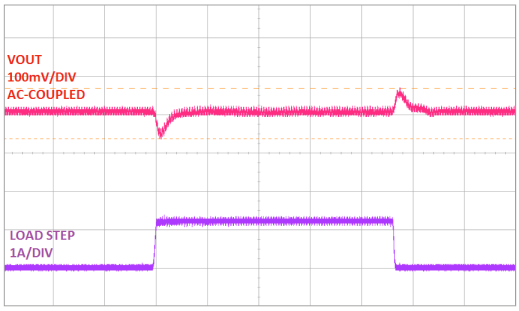

Figure 4. $12V_{IN}$, $1.5V_{OUT}$ 1.25A-2.5A Load Step (Fig. 2)

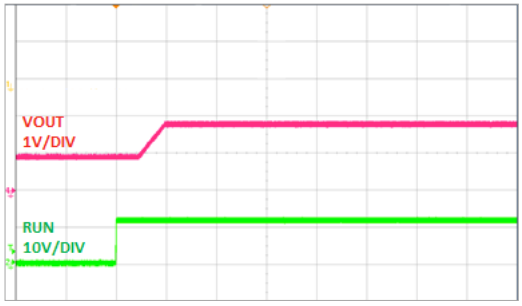

Figure 5. $12V_{IN}$, $1.8V_{OUT}$ Start-up into Pre-Biased Output

Parallel Operation for Higher Current Applications

The LTM4622's current-mode architecture gives it good and reliable cycle-by-cycle current monitoring allowing for its outputs to be paralleled together to support load currents of up to 5A.

Figures 6 and 7 demonstrate the good thermal and current sharing performance of the LTM4622 while configured for a two outputs current sharing, generating a 3.3V at 5A from a 5V input.

Figure 6. V_{IN} = 5V, V_{OUT} = 3.3V / 5A, T_A = 25C Thermals

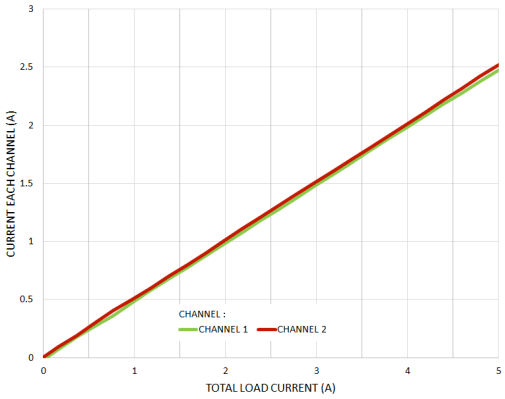

Figure 7. V_{IN} = 5V, V_{OUT} = 3.3V / 5A Current Sharing

Conclusion

The ultrathin LTM4622 provides a high performance regulator for single and multi-rail applications. Its wide operating range, features and compact solution size make it a highly flexible and robust solution capable of fitting into the tightest spaces on topside and backside of a PCB.