

Single-Layer Aluminum PCB Power Supply Design Noise Optimization Guide



Jingguo Wang

China DCP Team

Abstract

This document was translated from a simplified Chinese source. ([ZHCAG96](#))

Single-layer aluminum PCBs are widely used in LED lighting (such as streetlights and tube lights), automotive electronics, and high-power power supply modules due to their excellent heat dissipation performance and system cost advantages. As competition in the consumer electronics market intensifies, an increasing number of application scenarios are beginning to adopt single-layer aluminum PCBs in product designs. At the same time, however, power supplies of this type are highly susceptible to various noise sources because of the single-sided board design, leading to problems such as power supply abnormalities and output oscillation. Therefore, appropriate measures must be taken during the development stage to avoid these issues.

This article first introduces the advantages of single-layer aluminum PCBs in power supply design, then analyzes the challenges they may bring in circuit design and typical noise sources based on the LM5145, and finally proposes targeted optimization measures to ensure circuit reliability and stability.

Overview of Single-Layer Aluminum PCBs

Metal Core PCBs (MCPCBs) are printed circuit boards attached to a metal substrate with better thermal conductivity, enabling more efficient heat conduction through the metal core and thereby improving the board's heat dissipation performance. Among these, aluminum PCBs are one of the most widely used types of MCPCBs, providing efficient heat dissipation and cooling for electronic components and thereby enhancing overall product performance and reliability. Compared with conventional FR4 fiberglass PCBs, aluminum PCBs typically have a thermal conductivity of $1\text{--}3\text{W}/(\text{m}\cdot\text{K})$, with some high-performance products reaching above $8\text{W}/(\text{m}\cdot\text{K})$, far superior to the roughly $0.3\text{W}/(\text{m}\cdot\text{K})$ of FR4 material. This allows heat generated by power devices to be quickly conducted to the substrate, effectively reducing device junction temperature and improving system thermal reliability. At the same time, aluminum PCBs offer higher mechanical strength, with greater rigidity and impact resistance, so they can serve the dual role of heat dissipation and structural support in high-power-density designs.

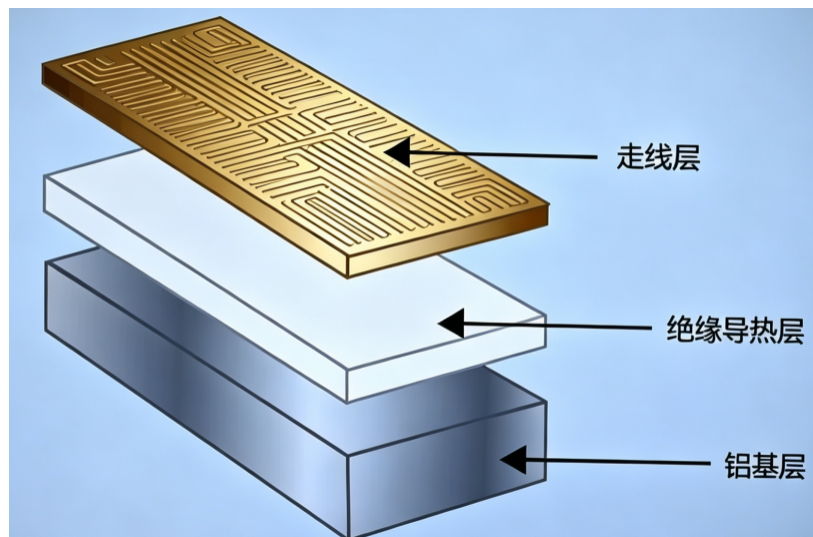


Figure 1. Schematic diagram of a single-layer aluminum PCB structure

In terms of cost, the raw material and processing costs of single-layer aluminum PCBs are typically higher than those of traditional FR4 boards. FR4 boards have mature processes and a well-established supply chain, resulting in a lower price per unit area; aluminum PCBs, however, require special insulating and thermally conductive layer materials and processing techniques, so their board cost is generally 2–3 times that of FR4 boards of the same specifications. Despite the higher board cost of aluminum PCBs, their overall cost advantage is quite clear from a system-level perspective. Aluminum PCBs can reduce or even eliminate the need for additional heat sinks, and the simplified thermal structure helps reduce assembly steps and lower manufacturing costs. Therefore, in certain suitable application scenarios, the system cost is often more competitive, especially in applications with high heat dissipation requirements and limited space.

Challenges in Single-Layer Aluminum PCB Power Supply Design

Single-layer aluminum PCBs offer clear advantages in system cost and heat dissipation, but their single-layer structure lacks dedicated internal power and ground layers, and the metal substrate readily forms capacitive coupling, making them highly prone to significant power supply noise, which poses challenges for power supply design. The causes of noise mainly include the following aspects. First, high-frequency switching and transient currents: when the driver circuit or switching devices produce rapidly changing currents, the parasitic inductance and resistance in the power path cause sharp voltage fluctuations, generating ripple and ground bounce noise. Second, the lack of a continuous ground plane (GND): routing on a single-layer board must be done on the top layer, so the return path is lengthened or broken. Larger current loops generate electromagnetic radiation and contaminate sensitive signal grounds. Finally, the interlayer capacitance effect of the aluminum substrate: only a thin thermally conductive insulating layer exists between the copper foil and the aluminum substrate, forming a relatively large distributed capacitance. High-frequency noise is coupled to the aluminum substrate through this capacitance, causing severe common-mode radiation. When radiated to the application side, the power supply exhibits abnormal phenomena such as unstable output or even output oscillation. The following sections analyze the noise sources and provide commonly used debugging approaches and solutions to avoid these problems during the development stage.

Noise Source Analysis and Debugging Optimization Approaches

Next, the LM5145 will be used as an example in a single-sided aluminum PCB design to analyze possible noise problems and common solutions. The LM5145 is a 75V synchronous buck controller suitable for high-voltage input power supplies or input rails with high-voltage transients, and it can effectively reduce the need for external surge suppression components. Its high-side switch minimum on-time is only 40ns, supporting large step-down ratios and enabling direct conversion from a 48V nominal input voltage to a low-voltage rail, thereby reducing system complexity and overall solution cost. In addition, the LM5145 can continue operating even when the

input voltage drops to 6V, with the duty cycle approaching 100% when necessary, making it an ideal choice for high-performance industrial control, robotics, data communications, and RF power supply applications.

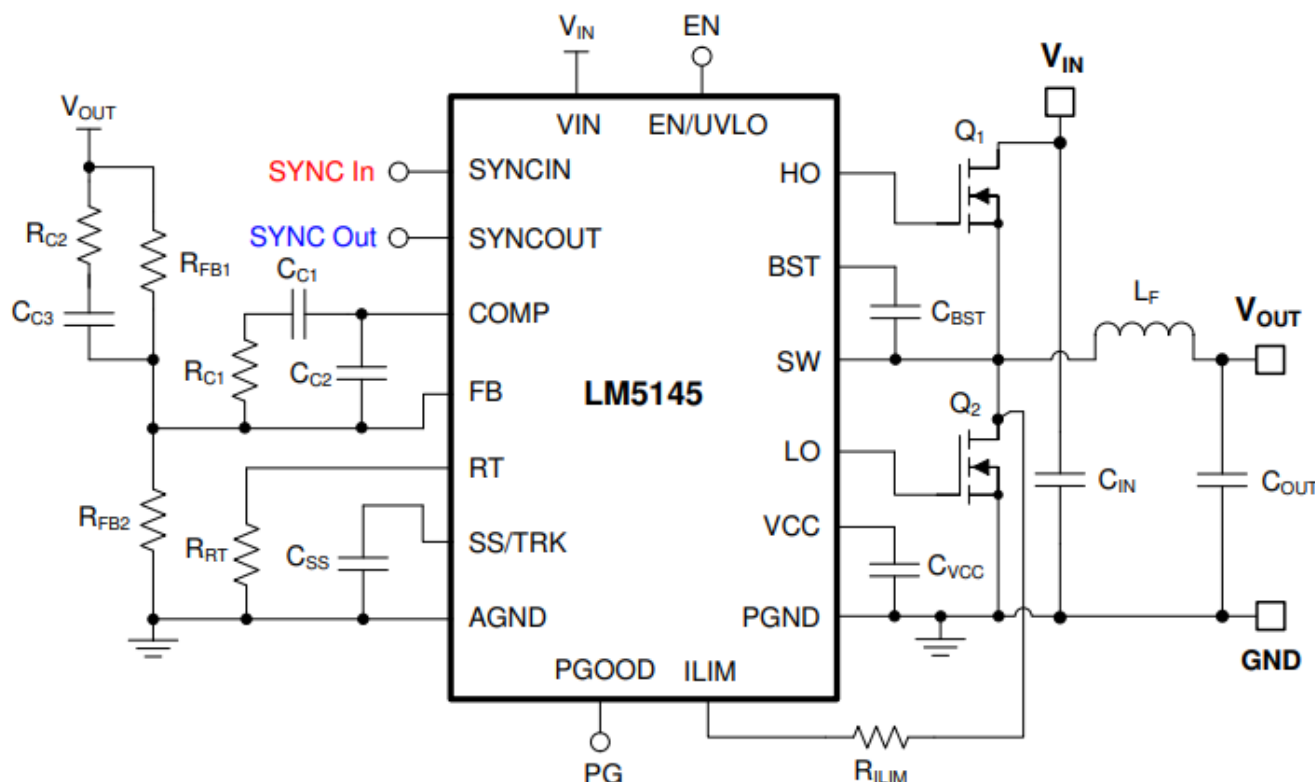


Figure 2. Typical application circuit of the LM5145

The noise sources of a single-layer aluminum PCB mainly include: ground bounce and power supply ripple caused by power/ground loop inductance, coupling of digital switching transient currents, crosstalk and radiation due to the lack of a complete reference plane, thermal noise, and external EMI coupling. Because the single-layer structure cannot provide the low-impedance power/ground planes and controlled return paths of a multilayer board, its noise suppression capability is significantly limited. Among these, the noise between power and ground is dominant. An aluminum PCB typically has only one conductive layer (copper traces) and no separate power/ground plane. The fast di/dt of switching devices (such as DC-DC converters and MOSFETs) or high-speed digital ICs generates significant voltage fluctuations ($\Delta V = L \cdot di/dt$) in trace loops with high loop inductance, manifesting as power supply ripple and ground bounce. The following sections analyze the common abnormal phenomena and solutions for this type of noise.

First, the first common problem in such designs is excessive output ripple caused by switching node noise, including both dynamic and steady-state ripple. Ringing in switching power supplies is a common issue, but due to the interlayer capacitance effect between the single-sided routing layer and the aluminum substrate, the ringing can become excessive and lead to excessive output ripple. Therefore, the most direct solution is to add local filtering: an RC snubber circuit between the SW node and GND (typical values $R = 2\text{--}10\Omega$, $C = 100\text{--}1000\text{pF}$, with parameters tuned after measuring the actual ringing frequency); and a π -type LC filter at the output (a small inductor plus multiple ceramic capacitors in parallel, such as $10\mu\text{F}/0.1\mu\text{F}/0.01\mu\text{F}$), while paying attention to the inductor DCR and saturation current. Second, the hot loop area should be minimized as much as possible in the PCB layout.

In addition to switching node filtering, in COT-controlled DC-DC products, proper design of the feedforward capacitor can also optimize output ripple, especially dynamic ripple. The feedforward capacitor is connected in parallel with the feedback divider resistor, coupling the switching ripple of the output voltage (without DC attenuation) directly to the feedback voltage, thereby increasing loop bandwidth and phase margin and, in particular, preventing oscillation when low-ESR ceramic capacitors are used. There is no unified analytical

formula for selecting the feedforward capacitor value; it is usually chosen empirically to "provide sufficient AC ripple injection to maintain loop stability", with typical values of 10–100pF. If an estimate is needed, it can be approximated based on the feedback divider network and the crossover frequency: $C_{ff} \approx 1 / (2\pi \times R1 \times f_c)$, where R1 is the upper divider resistor and f_c is the target crossover frequency (often taken as 1/5–1/10 of the switching frequency). However, final verification of loop stability through measurement is required. TI also has an application note that provides detailed simulation explanations on this topic, ZHCA058, which can be referenced. The following is a dynamic ripple test based on the TI TPS548B27 EVM under the following conditions to illustrate the effect of the feedforward capacitor on output dynamic ripple: 2ms/4ms, 3A/ μ s, 0.015 to 10A dynamic load, with and without the feedforward capacitor. It can be seen that a suitable feedforward capacitor can greatly optimize dynamic ripple performance.

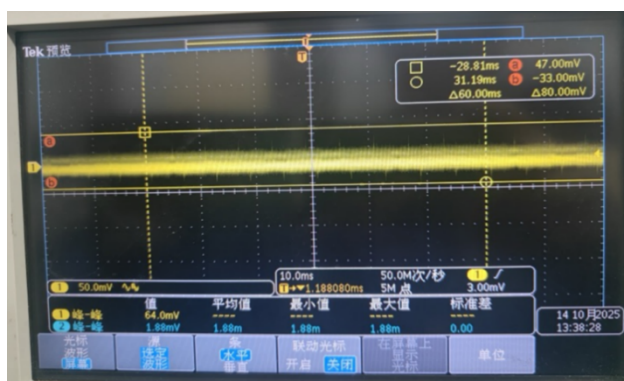


Figure 3. TPS548B27 with 120pF feedforward capacitor (left)

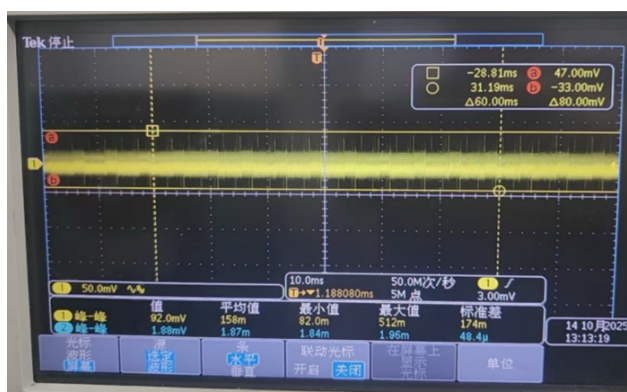


Figure 4. TPS548B27 without 120pF feedforward capacitor (right)

The second common problem in such designs is abnormal output voltage, the most direct manifestation of which is a significant error between the output voltage and the designed value. This problem is generally caused by power supply noise being introduced into the chip's FB pin, resulting in abnormal output voltage. This problem also needs to be optimized from both the PCB itself and the circuit application. On the PCB itself, the lower divider resistor at the FB pin should be placed as close to the chip as possible and connected to the nearest GND as much as possible, thereby reducing noise sources.

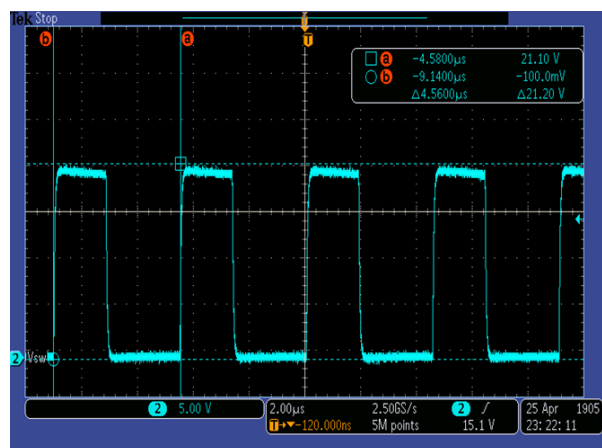


Figure 5. 20Vin, 8.4Vout with 1A load, CH2 = SW, 33pF

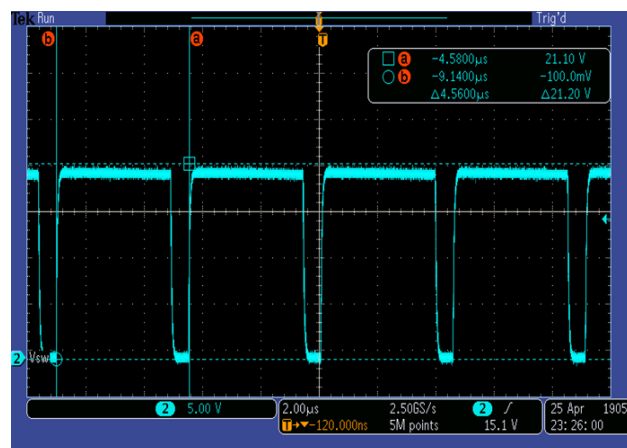


Figure 6. 20Vin, 17.12Vout with 1A load, CH2 = SW, 150pF

At the application level, filtering capacitors should be added to filter out FB noise. Note, however, that the FB-to-ground capacitor must not be too large, as an excessively large value may affect the loop and cause more serious problems. In engineering practice, it is generally necessary to start with 10–100pF, then gradually

increase the value while observing the experimental behavior. The above results are based on tests with the LM5145 at a 20V input, a 17.1V output setting, and a 1A load. It can be seen that at 33pF, the output voltage is abnormal, and when increased to 150pF, the output voltage returns to normal.

Finally, in addition to clear application-side problems such as abnormal output, noise can also cause many issues such as abnormal loop behavior. Layout optimization is also a very important aspect. In addition to the switching node and FB point layout considerations mentioned above, great care must also be taken in the design of the overall ground plane. First, AGND and PGND must be separated. Taking the LM5145 as an example, RT, SS, SYIN, and FB should be connected to AGND, while FB, VIN, ILIM, and EP need to be connected to PGND. Second, the switching current loop should be compressed as much as possible—place the input ceramic capacitor, switching devices, and inductor as close together as possible to minimize the loop area (if necessary, use a "return ground island" with localized copper pour to create a low-inductance loop); place decoupling capacitors at multiple points and as close as possible—equip the IC power pins with 0.1–1μF X7R ceramic capacitors (within <5mm distance); keep magnetic components (inductors/transformers) away from sensitive analog circuits, and add copper shielding (grounded) if necessary to suppress radiation; if permitted, locally slot or cut out the non-insulating layer side (metal substrate) of the aluminum PCB to avoid eddy current losses or coupling beneath the power loop (insulation layer withstand voltage must be confirmed).

Summary

This article uses the application of the LM5145 on a single-layer aluminum PCB as its background and focuses on the core advantages of single-layer aluminum PCBs in terms of thermal performance, as well as the challenges they face in noise suppression. Targeted solutions are proposed for noise issues at the FB, SW, and GND nodes, and the effectiveness of these solutions is verified with actual test results.

References

1. LM5145 datasheet <https://www.ti.com/product/LM5145>
2. Complete Guide to the Use of Aluminum PCB Substrates. <https://www.jiepei.com/design/3287.html>
3. TPS548B27 datasheet <https://www.ti.com/product/TPS548B27>

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), 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 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](#), [TI's General Quality Guidelines](#), or other applicable terms available either on [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. Unless TI explicitly designates a product as custom or customer-specified, TI products are standard, catalog, general purpose devices.

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

Copyright © 2026, Texas Instruments Incorporated

Last updated 10/2025