

How to Lay Out a PCB for High-performance, Low-side Current-sensing Designs



Tim Claycomb

In my previous [blog post](#), I discussed how low-side current sensing helps control a motor and then provided three steps to design a low-side current-sensing circuit for cost-sensitive applications. In this post, I'll discuss how you – with the proper printed circuit board (PCB) layout techniques – can design an accurate, low-cost, low-side current-sensing circuit with a single tiny [operational amplifier](#) (op amp).

[Figure 1](#) displays the schematic of the low-side current-sensing circuit in my previous blog, shown here using the [TLV9061](#) tiny op amp.

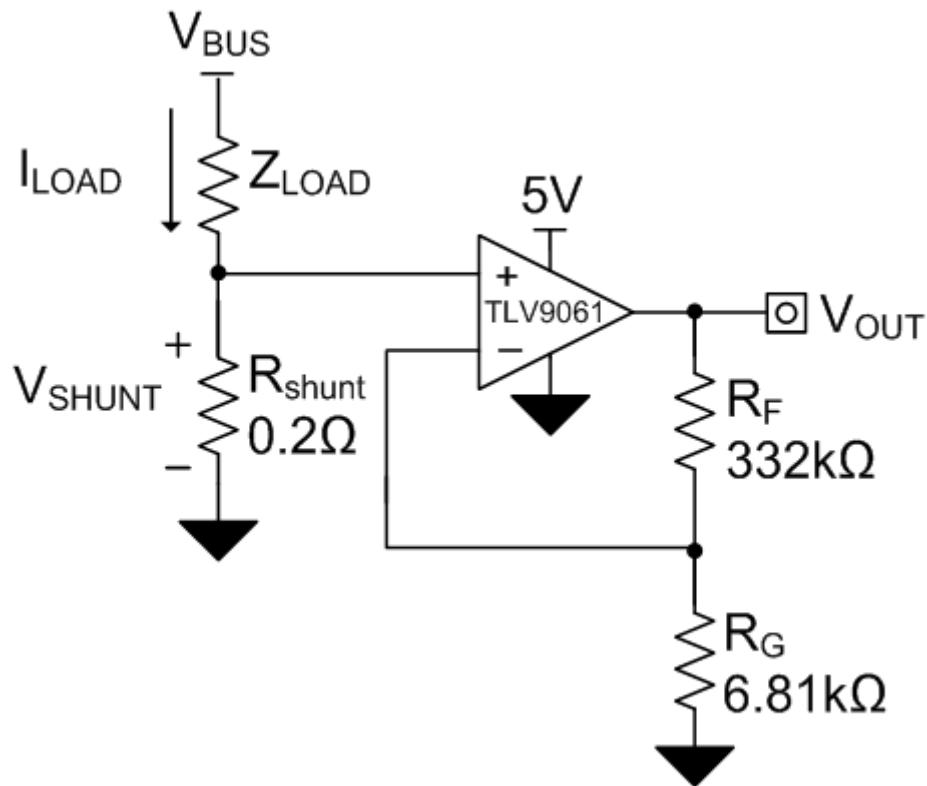


Figure 1. Low-side Current-sensing Schematic

Equation 1 calculates the transfer function of the circuit shown in [Figure 1](#):

$$V_{out} = I_{LOAD} \times R_{SHUNT} \times Gain \quad (1)$$

$$\text{where } Gain = 1 + \frac{R_F}{R_G}$$

There are two main PCB design requirements for an accurate low-side current-sensing design. The first is to ensure that the connections from the shunt resistor are directly connected to the noninverting input of the amplifier and the ground connection of R_G . This is typically referred to as a “Kelvin connection.” Not using a Kelvin connection creates a stray resistance in series with the shunt resistor, which causes a gain error in the system. [Figure 2](#) shows the location of the stray resistance in the system.

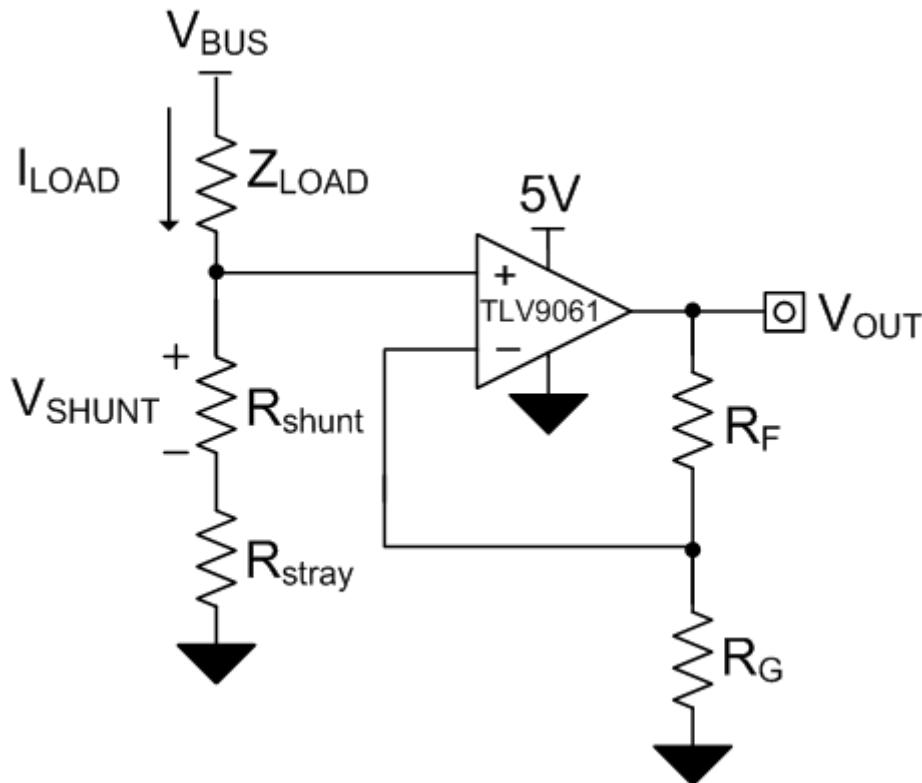


Figure 2. Stray Resistance in Series with R_{shunt}

Equation 2 calculates the transfer function of the circuit in [Figure 2](#):

$$V_{out} = I_{LOAD} \times (R_{SHUNT} + R_{STRAY}) \times Gain \quad (2)$$

The second design requirement is to place the ground connection of resistor R_G as close to the ground connection of the shunt resistor as possible. As current flows through the ground plane of the PCB, voltage drops develop across the ground plane, which cause voltage differences in the ground plane at different locations on the PCB. These voltage differences cause an offset voltage error in the system. In [Figure 3](#), a voltage source in place of the ground symbol connected to R_G represents a difference in ground potential.

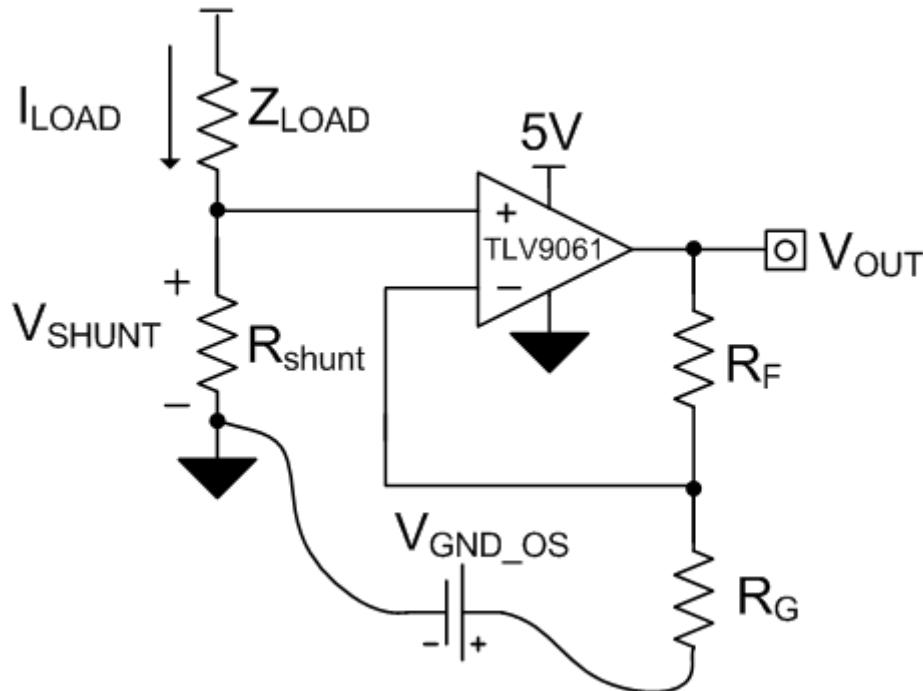


Figure 3. Voltage Differences on the Ground Plane

Equation 3 calculates the transfer function of the circuit shown in [Figure 3](#):

$$V_{out} = I_{LOAD} \times R_{SHUNT} \times \left(1 + \frac{R_F}{R_G}\right) - V_{GND_OS} \times \frac{R_F}{R_G} \quad (3)$$

[Figure 4](#) shows a schematic representation of a proper PCB layout.

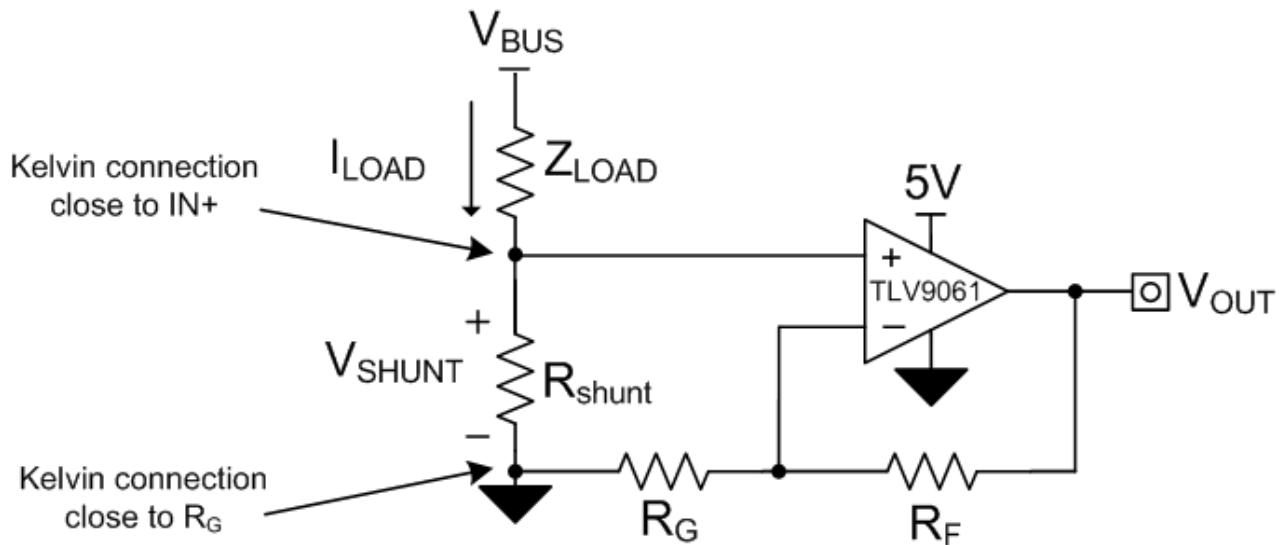


Figure 4. Schematic Representation of a Proper Layout

[Figure 5](#) illustrates proper PCB layout for a low-side current-sensing design following the recommendations I provided earlier. The top layer is red and the bottom layer is blue. R5 and C1 are included in the PCB layout in order to show where you should place a load resistor and decoupling capacitor.

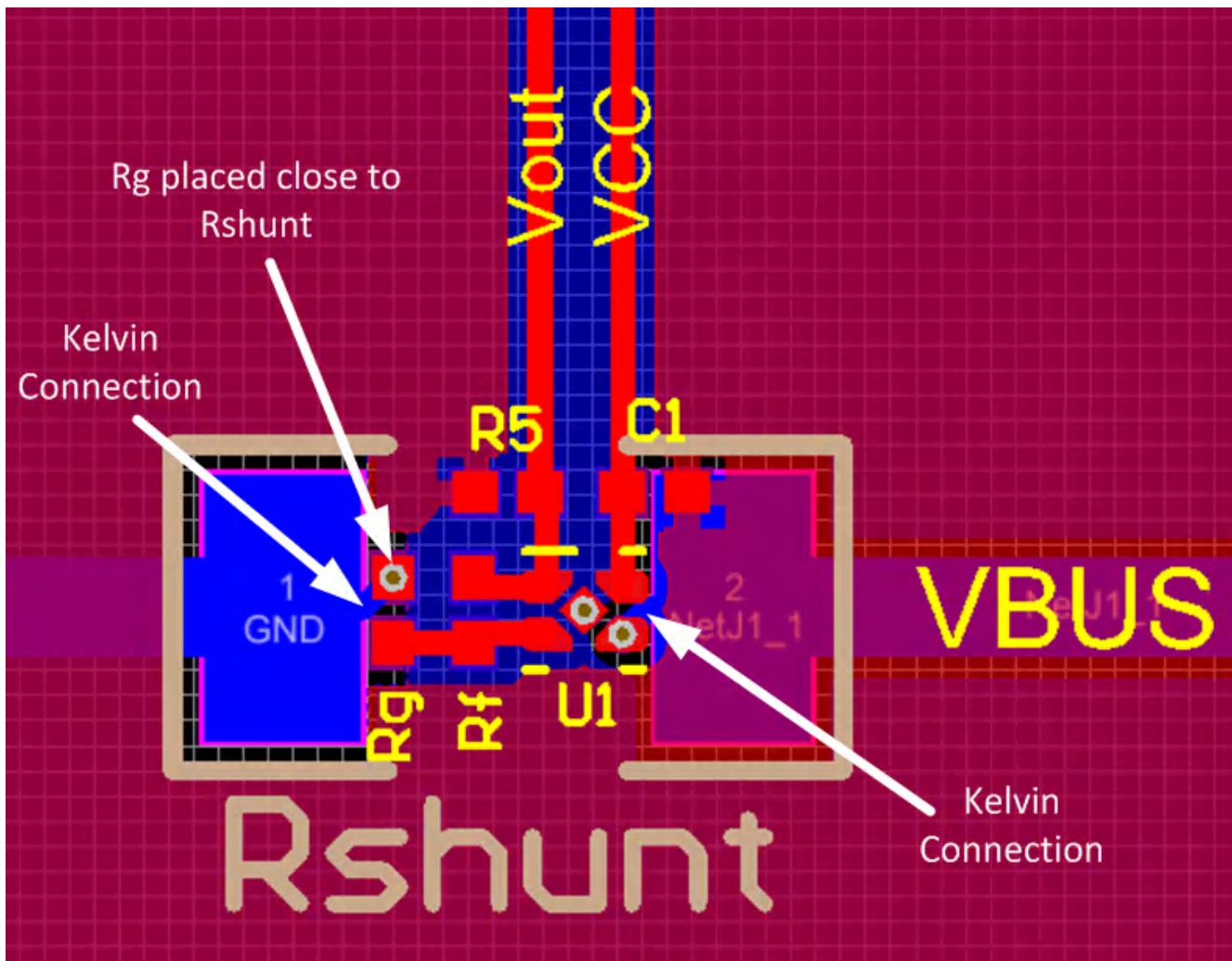


Figure 5. Proper PCB Layout for Low-side Current Sensing

Notice that the traces coming off of Rshunt use a Kelvin connection and RG is placed as close to Rshunt as possible. The TLV9061 op amp's five-pin extra small outline no-lead (X2SON) package (0.8mm by 0.8mm) enables you to place all passive components in between the two pads of the shunt resistor on the top layer. From there, you can easily use vias to route the connections from the shunt resistor on the bottom layer to the noninverting pin and RG on the top layer.

The next time you lay out a PCB for a low-side current-sensing design, be sure to follow these guidelines to reduce any potential errors in your design:

- Use Kelvin connections on the shunt resistor.
- Place RG as close to the shunt resistor ground as possible.
- Place decoupling capacitors as close to the power-supply pins as possible.
- Pour at least one solid ground plane.

To learn more about designing a PCB layout with the X2SON package take a look at the application report [“Designing and Manufacturing with TI’s X2SON Packages.”](#)

Additional Resources

- For additional information on PCB layout, read these blog posts:
 - [“The basics: How to layout a PCB for an op amp.”](#)
 - [“How to layout a PCB for an instrumentation amplifier.”](#)
 - Find common analog design formulas in the [“Analog Engineer’s Pocket Reference.”](#)
 - Read about issues related to decoupling capacitors in amplifier expert Art Kay’s blog post, [“The decoupling capacitor ... is it really necessary?”](#)
 - Learn more in this Planet Analog article, [“When GND Isn’t GND, Single-Ended Circuits Become Differential.](#)

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](#) 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