

DEM-OPA-SOT-1A Demonstration Fixture

1 Description

The DEM-OPA-SOT-1A demonstration fixture is a generic, unpopulated printed circuit board (PCB) designed to provide an optimized layout when evaluating wideband operational amplifiers in the tiny SOT-23 package. [Figure 1](#) shows the package pinout for this PCB. For more information on these op amps, as well as good PCB layout techniques, see the individual amplifier data sheets.

Only a few components are necessary to be assembled onto the board to set up the operational amplifier in either an inverting or noninverting configuration. In order to avoid parasitic component inductance the board only accepts surface mount resistors and capacitors. The actual values of the components, especially the resistors, depend on the type of operational amplifier used and the intended mode of operation. The board was designed to accommodate a variety of amplifiers: voltage feedback types or current feedback types. Furthermore, the evaluation board not only handles op amps in the existing 5-lead SOT-23 package but also the evaluation platform for 6-lead SOT-23 package. Therefore, the layout includes a sixth pad for the IC (U1) located on the opposite side of pad 2. The sixth lead may be an optional function pin for those amplifiers featuring a disable function. As shown in [Figure 3a](#) (top layer view), the user has the option to either hardwire this function pin to the positive supply by shortening the solder switch A, or connecting it to connector J3 soldering the switch in position B.

For a detailed discussion on the individual op amp and the appropriate selection of the recommended feedback and gain resistors, see the respective data sheets. As a reminder, the frequency response performance of a current feedback amplifier is determined by the feedback resistor value. Also note that the feedback resistor compensates for the package parasitics, and therefore its value usually varies for different packages.

All high-frequency devices have a common sensitivity to component and layout parasitics that may alter the AC performance. Therefore, slight adjustments of the feedback resistor (R2) and/or the gain resistor (R6) might be necessary to obtain the final desired frequency response. For current feedback amplifiers, increasing R_F from its nominal value will reduce the achievable bandwidth, while decreasing R_F will cause the frequency response to peak more. However, at higher gains, it might be advantageous to decrease R_F in order to optimize for bandwidth.

For voltage feedback amplifiers, changing the impedance level (increasing or decreasing R_F and R_G) can also control the amount of peaking.

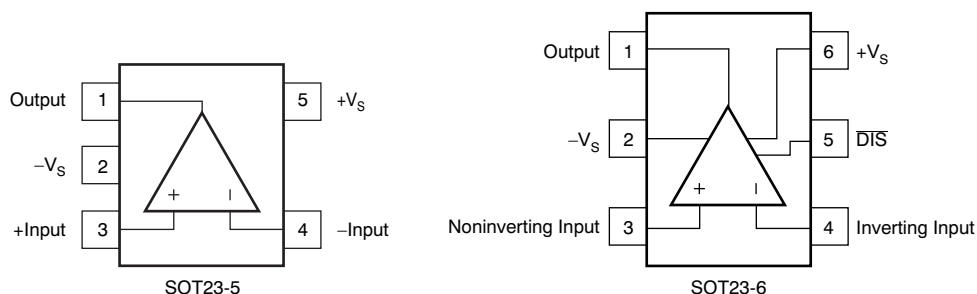
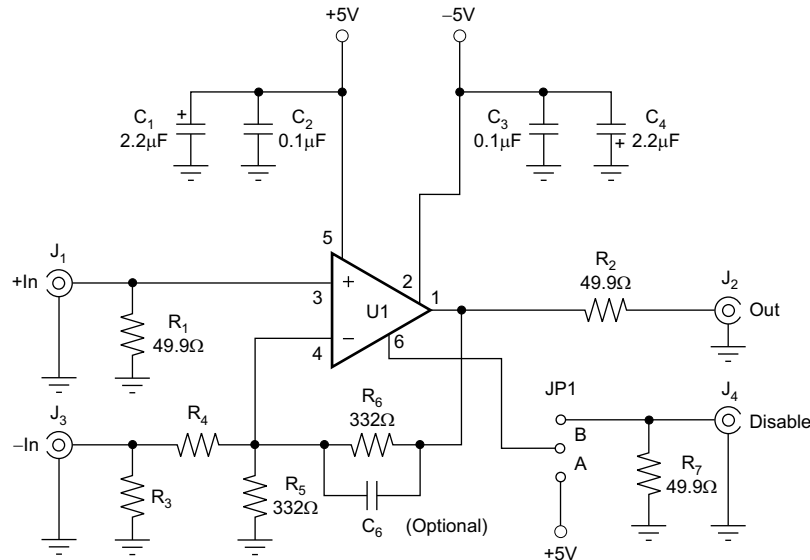


Figure 1. SOT23 Package Pinout, Top View

2 Circuit

The circuit schematic in [Figure 2](#) shows the connections for all possible components. Each configuration uses only some of the components. The power-supply bypassing capacitors are crucial components for high-frequency amplifiers and should not be omitted during evaluation. Typically, each supply should be bypassed by a parallel combination of a ceramic 0.1µF capacitor and a tantalum 2.2µF capacitor. Those bypass capacitors are located on the board such that the return path to the power connector is very short. Any larger transient currents will have a short loop, and will not reflect to the input and cause an increase in distortion.



NOTE: Values for G = +2, OPA658. R3, R4, R7, C6 not assembled.

Figure 2. Schematic for DEM-OPA-SOT-1A

3 Components

Components that have RF performance similar to the ones in [Table 1](#) may be substituted.

Table 1. Component Descriptions

PART	DESCRIPTION
C ₁ , C ₄	2.2µF, 16V, Size 3548
C ₂ , C ₃	0.1µF, 50V, Size 1206
C ₆	Feedback capacitor (optional); depends on application (not used on current feedback op amps).
R ₁ , R ₂ , R ₇	Typically 50Ω
R ₄ , R ₅ , R ₆	Depends on application
JP1	Power Connector (On-Shore Technology ED555/3DS)
J ₁ – J ₄	SMA or SMB Connectors
R ₃	Set to get R3 R4 = desired input impedance for inverting operation.

4 Board Layout

This demonstration fixture is a two-layer PCB with the power traces on the bottom layer. Even though both sides have a ground plane, a window has been opened up around the DUT and its surrounding components. The purpose of this window is to reduce the parasitic capacitances between sensitive nodes and the ground planes. The footprint of the SMA connectors were designed to use straight connectors in either a vertical or horizontal mounting position. Note that the center conductor of the SMA must be on the top side of the board when mounted horizontally.

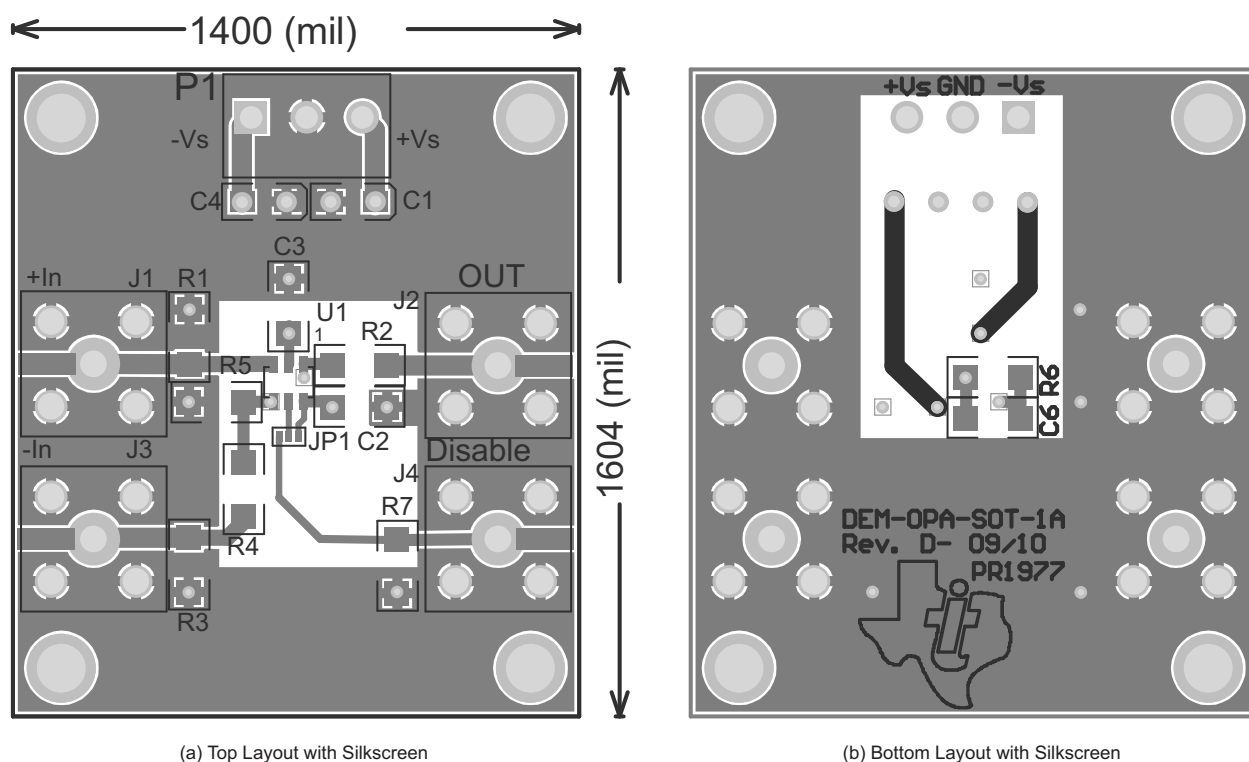


Figure 3. DEM-OPA-SOT-1A Demonstration Board Layout

5 Measurement Tips

This demonstration fixture, with the component values shown, is designed to operate in a 50Ω environment; most data sheet plots are obtained this way. It is easy to change the component values for different input and output impedance levels. However, do not use high-impedance probes; they represent a heavy capacitive load to the op amp, and will alter the amplifier response. Instead, use low-impedance ($\leq 500\Omega$) probes with adequate bandwidth. The probe input capacitance and resistance set an upper limit on the measurement bandwidth. If a high-impedance probe must be used, place a 100Ω resistor on the probe tip to isolate its capacitance from the circuit.

FCC Warning

This evaluation board/kit is intended for use for **ENGINEERING DEVELOPMENT, DEMONSTRATION, OR EVALUATION PURPOSES ONLY** and is not considered by TI to be a finished end-product fit for general consumer use. It generates, uses, and can radiate radio frequency energy and has not been tested for compliance with the limits of computing devices pursuant to part 15 of FCC rules, which are designed to provide reasonable protection against radio frequency interference. Operation of this equipment in other environments may cause interference with radio communications, in which case the user at his own expense will be required to take whatever measures may be required to correct this interference.

EVALUATION BOARD/KIT IMPORTANT NOTICE

Texas Instruments (TI) provides the enclosed product(s) under the following conditions:

This evaluation board/kit is intended for use for **ENGINEERING DEVELOPMENT, DEMONSTRATION, OR EVALUATION PURPOSES ONLY** and is not considered by TI to be a finished end-product fit for general consumer use. Persons handling the product(s) must have electronics training and observe good engineering practice standards. As such, the goods being provided are not intended to be complete in terms of required design-, marketing-, and/or manufacturing-related protective considerations, including product safety and environmental measures typically found in end products that incorporate such semiconductor components or circuit boards. This evaluation board/kit does not fall within the scope of the European Union directives regarding electromagnetic compatibility, restricted substances (RoHS), recycling (WEEE), FCC, CE or UL, and therefore may not meet the technical requirements of these directives or other related directives.

Should this evaluation board/kit not meet the specifications indicated in the User's Guide, the board/kit may be returned within 30 days from the date of delivery for a full refund. THE FOREGOING WARRANTY IS THE EXCLUSIVE WARRANTY MADE BY SELLER TO BUYER AND IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED, IMPLIED, OR STATUTORY, INCLUDING ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE.

The user assumes all responsibility and liability for proper and safe handling of the goods. Further, the user indemnifies TI from all claims arising from the handling or use of the goods. Due to the open construction of the product, it is the user's responsibility to take any and all appropriate precautions with regard to electrostatic discharge.

EXCEPT TO THE EXTENT OF THE INDEMNITY SET FORTH ABOVE, NEITHER PARTY SHALL BE LIABLE TO THE OTHER FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES.

TI currently deals with a variety of customers for products, and therefore our arrangement with the user is **not exclusive**.

TI assumes **no liability for applications assistance, customer product design, software performance, or infringement of patents or services described herein**.

Please read the User's Guide and, specifically, the Warnings and Restrictions notice in the User's Guide prior to handling the product. This notice contains important safety information about temperatures and voltages. For additional information on TI's environmental and/or safety programs, please contact the TI application engineer or visit www.ti.com/esh.

No license is granted under any patent right or other intellectual property right of TI covering or relating to any machine, process, or combination in which such TI products or services might be or are used.

EVM WARNINGS AND RESTRICTIONS

It is important to operate this EVM within the input voltage range of $\pm 15\text{V}$ and the output voltage range of -15V to $+15\text{V}$.

Exceeding the specified input range may cause unexpected operation and/or irreversible damage to the EVM. If there are questions concerning the input range, please contact a TI field representative prior to connecting the input power.

Applying loads outside of the specified output range may result in unintended operation and/or possible permanent damage to the EVM. Please consult the EVM User's Guide prior to connecting any load to the EVM output. If there is uncertainty as to the load specification, please contact a TI field representative.

During normal operation, some circuit components may have case temperatures greater than $+50^\circ\text{C}$. The EVM is designed to operate properly with certain components above $+50^\circ\text{C}$ as long as the input and output ranges are maintained. These components include but are not limited to linear regulators, switching transistors, pass transistors, and current sense resistors. These types of devices can be identified using the EVM schematic located in the EVM User's Guide. When placing measurement probes near these devices during operation, please be aware that these devices may be very warm to the touch.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2009, Texas Instruments Incorporated