

PCB GUIDELINES FOR TELEPHONE LINE INTERFACE

This application note is intended to assist DAA users with their PCB layout considerations. Most users of DS2249 family devices will be using the transferable Dallas Semiconductor FCC Part-68 registration, and will need no further approvals. However, since telephone signals will be routed through a user's circuit board prior to reaching the DS2249 device, certain layout precautions should be taken. If these guidelines are followed, the user will remain in compliance. Later in this note, European guidelines will be given for users of the DS2249EU.

U.S. GUIDELINES

The DS2249 Data Access Arrangement has four telephone network signals that can be connected through the user's PCB. The DS2249 family of devices incorporate protection for the user's equipment or protected devices. However, the user's board must carry these network signals. Connection to the telephone line should be made via standard RJ-11C jacks. The PCB traces used by TIP, RING, TIPO, and RINGO should be separated from each other by 0.1 inches. These traces should also be separated from other protected traces by 0.2 inches. These traces should also be separated from other conducting material, such as the chassis, by 0.2 inches. TIP, RING, TIPO, and RINGO should be etched using traces of 0.02 inches in width or greater. These traces should be as short as possible.

Other considerations in layout are the noise sensitivity of the analog signals that run between the modem and DAA. These traces should also be as short as possible

to prevent noisy telephone connections. It is likely that wire-wrapped prototypes will experience noisy phone connections.

EUROPEAN GUIDELINES

An isolation barrier is a region of physical space which separates devices on the telephone network side from those on the equipment side. The only devices that cross the barrier are those designed for isolation, such as opto-couplers and transformers. The DS2249EU DAA for Europe is designed with such a barrier. Since the network traces will be passed through the user's board, it will also need to incorporate a barrier. Most of Europe and Australia require a 6 mm barrier between network and equipment (protected and unprotected) components and PCB trace. No other conductive parts can cross the barrier. Also, no traces can enter the barrier region on their way to another part of the board. No power or ground planes should be passed into this region, even if they are on different layers from the phone signals. The barrier must be an uninterrupted physical space. Also note that since the phone jack is probably at the edge of the board, the barrier also extends to the board edge, including an edge connector if one is present. Any unprotected traces should remain across the barrier from the phone jack.

If a design is being done for the U.S. but may be shipped into Europe in the future, it would be prudent to incorporate the barrier in advance. Since the U.S. and European versions of the DAA are pin-compatible, it will then be simple to replace the U.S. DAA with the European one.

CIRCUIT BOARD BARRIER REGION Figure 1

