IRFIMI.®

by Allan Coon
Director, Filter Product Marketing
RF Monolithics, Inc.
1999

Introduction

The performance of surface acoustic wave (SAW) filters depends on a number of external factors. These include source and load impedances presented to the filter by external matching networks, the quality of the connections to the filter, the proximity of other circuitry and conducting structures, and the layout of the printed circuit board (PCB) that the filter is soldered to. This application note addresses the later from a practical point of view. The quality of the PCB design can affect critical passband characteristics as well as the ultimate rejection of a SAW band-pass filter. This tutorial outlines the basic principles of PCB design required to obtain the best performance from SAW filters. It is provided as a guide especially for the RF circuit designer with little or no experience in applying SAW filters or in SAW filter PCB layout. Knowledge of appropriate general PCB design rules and standard RF layout principles is assumed.

Background

Many SAW filters have excellent ultimate rejection characteristics inherent to their fundamental designs. These characteristics are shown to best advantage in test fixtures used for production electrical tests. These fixtures are available from SAW filter manufacturers for the purpose of correlating electrical characteristics between the filter manufacturer and the filter customer. These fixtures, usually complete with external impedance matching to 50 ohms, are ideal for correlation purposes. However, these fixtures often contain a great deal of isolation between the coaxial input and output. The typical commercial application requires a much simpler and less costly layout, with no shielding and often in tight spaces. Consequently performance of the SAW filter in the end application is sometimes not as good as in the test fixture or as advertised.
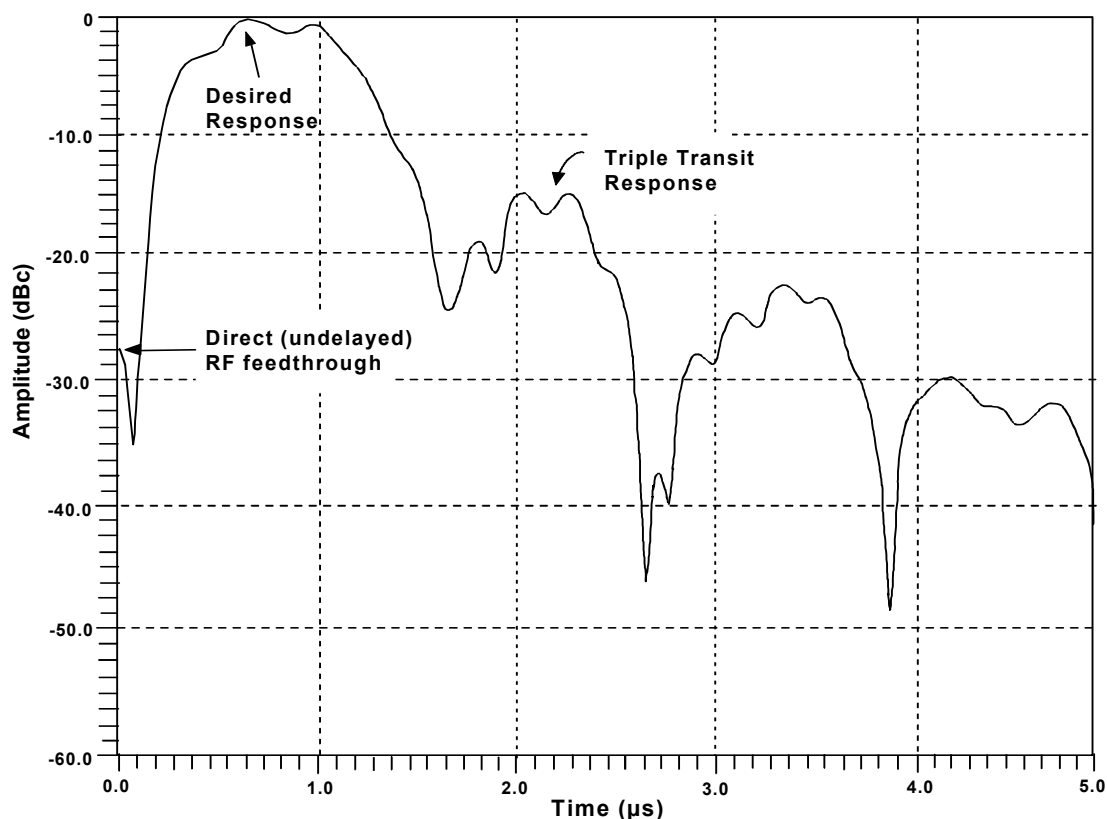

Figure 1
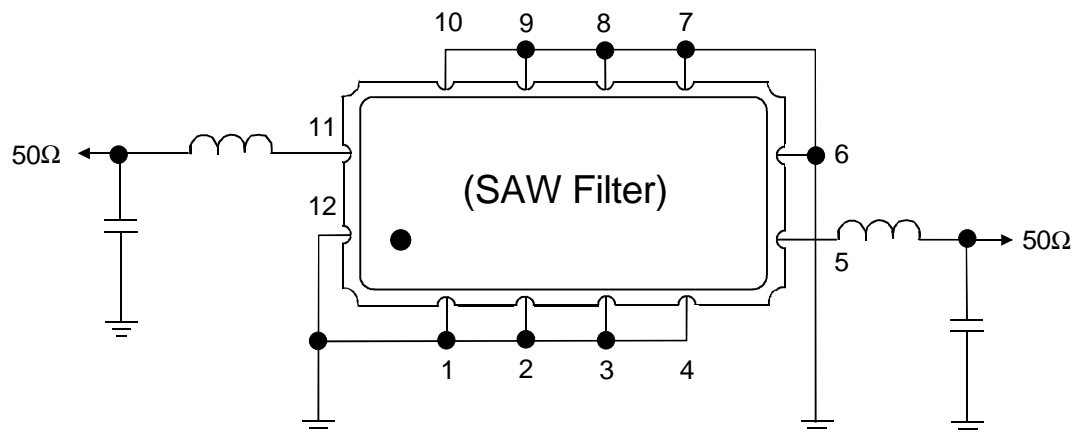Representative Impulse Response

Figure 2
Typical External Connections

Another consideration of test-fixture versus PCB soldered performance involves the quality of the ground connection. A production test fixture does not result in the lowest impedance ground connections and may have inferior ultimate rejection compared to the final application circuit. The point is that actual circuit performance may differ from that in a test fixture or even a demo board. In any case, the PCB layout should be designed to maximize performance. In most cases, the observation of a few simple rules can result in better filter performance for no additional real estate or cost.

There are two key aspects of circuit design that have a very significant impact on the performance of most SAW filters. One is external impedance matching. This includes centering component values, observing minimum component Q and maximum tolerances, and also variations in the source and load impedances of adjacent circuits. The other key consideration is the physical design of the PCB and sometimes also the enclosure.

SAW filters typically have propagation delays of several hundred nanoseconds to several tens of microseconds. Consequently there is a significant difference in the time delay of the directly conducted electromagnetic signal as compared to those that are conducted at a lower speed through the device as acoustic waves. Figure 1 shows the time impulse response of a typical, small, inexpensive coupled-resonator SAW filter used in consumer devices. There are three key features of this response: First, the direct RF feedthrough is due to the RF signal essentially bypassing the SAW filter. This signal is due to direct coupling internal to the filter and also external to the small case. In this example, the

direct RF feedthrough is 27 dB below the desired (delayed) signal. Second, after a nominal delay comes the desired signal. This is followed by the triple transit spurious time response and subsequent responses from internal acoustic reflections in the filter.

These spurious time responses cause constructive and destructive interference with the desired signal in the passband. This results in degradation to amplitude ripple, phase linearity, and group delay deviation in the passband. Outside of the passband, direct feedthrough bypasses the SAW filter entirely, resulting in degraded ultimate rejection. In the example of Figure 1, triple transit response far exceeds the direct feedthrough because of the inherently resonant nature of this particular coupled-resonator design. However, typical triple transit rejection is on the order of 50 to 60 dB in transversal SAW filters. For that class of filter, direct RF feedthrough can easily become a limiting factor in filter performance.

There are three principle sources of RF feedthrough external to the filter package: Electrostatic coupling between input and output circuits, mutual inductance between input and output matching inductors, and ground currents shared between input and output circuits. (See Figure 2 for typical connections and external matching.) All of these issues must be considered by the SAW filter and package designers to optimize performance of the filter. However, whether the end result in the practical circuit takes advantage of the filter's capability depends on how the circuit designer handles these issues.
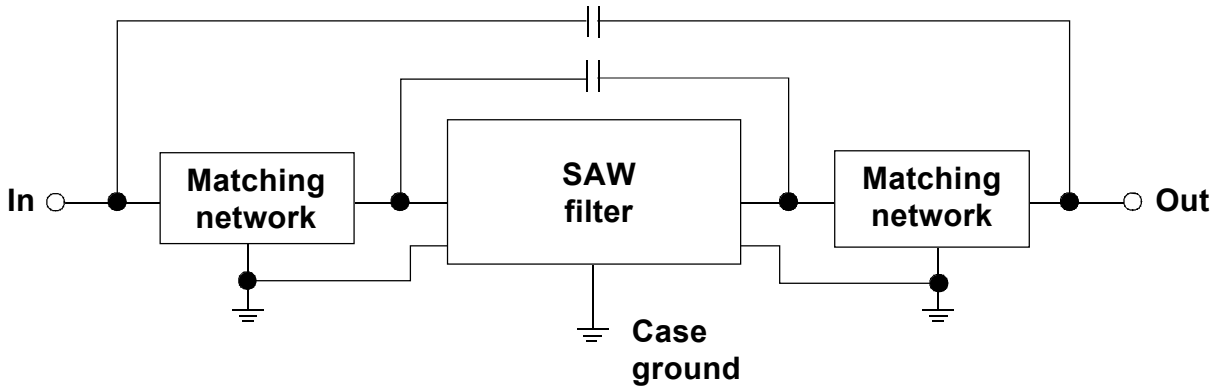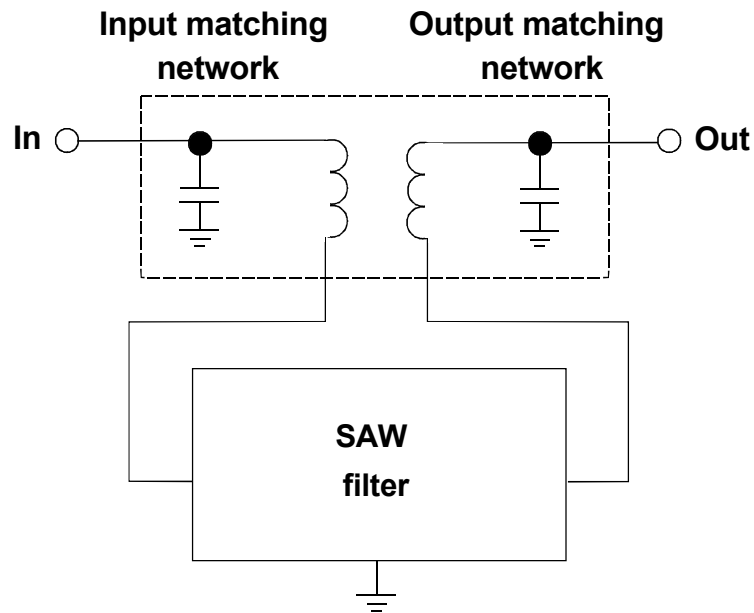
Figure 3a
Feedthrough Model of Capacitive Coupling

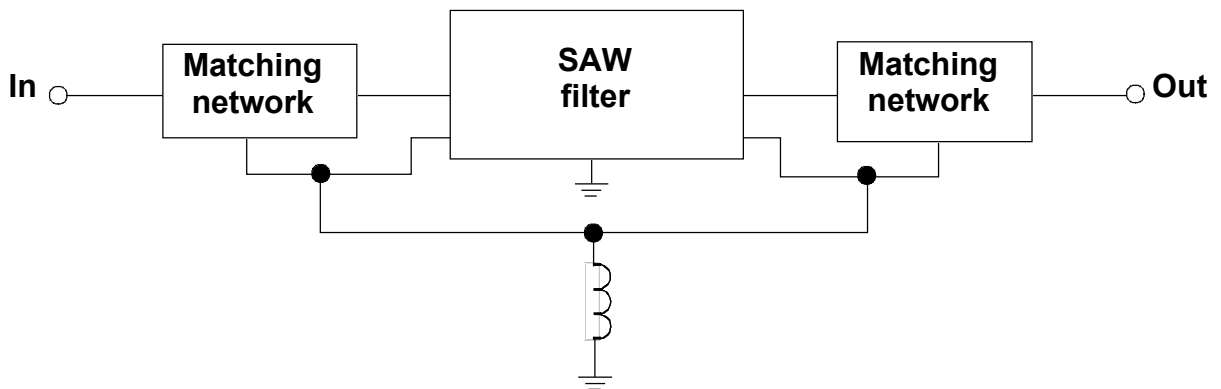*Figure 3b*
Feedthrough Model of Mutual Inductance

Figure 3c
Feedthrough Model of Ground Paths

Electrostatic Coupling

This mode of undesired coupling from input to output is often referred to as "electrostatic" because it can be measured at low frequency and modeled as a static capacitance. However the adverse effect on filter performance is obvious at RF, especially at frequencies above the passband. The nature of this coupling is illustrated in Figure 3a. This model is very simplified and is shown only as an illustration of the concept. Input-to-output coupling is actually distributed and occurs among terminals of the SAW filter, PCB traces, impedance matching components, and circuits connected to the filter.

Standard RF layout practices are critical. These include minimizing trace and lead lengths and also keeping input circuits as far as possible from output circuits. External matching components, especially inductors, often couple capacitively regardless of orientation. Unfortunately, the goals of placing components as close to the filter as possible and as far from each other as possible are contradictory. There are at least five strategies for combating this engineering trade-off problem: First, the most reliable option is shielding that isolates input circuits from output circuits. In its simplest form a metal shield covers either the input or output circuit, or both. The technique most often used in high-performance applications and in test fixtures consists of placing these circuits inside milled cavities in aluminum or brass housings. The later option is inappropriate in consumer portable devices due to cost. The former can usually be avoided with good PCB layout, but is sometimes necessary in portable devices due to the cramped space.

Second, a common strategy to accomplish a healthy degree of shielding without introducing costly shields, is to place input and output circuits on opposite sides of the PCB. This solution is often impractical, but should always be kept in mind as a possible solution.

Third, alternative matching topologies may be an option. Review the impedance-matching alternatives. Different matching network topologies may permit the elimination of or reduction in the number of inductors, which are especially prone to parasitic capacitive coupling with their surroundings. This is often impractical because typical SAW filters present capacitive impedances at their terminals. However, a frequently used strategy is to use some distributed matching in the form of a section of microstrip between the filter and induc-

tors. This puts the inductors farther apart when the extra real-estate is available.

Fourth, inductors shielded with metal on the exterior sometimes solve the problem nicely. However, this option usually reduces Q, often adds size and cost, sometimes reduces the flexibility to change inductance values, and does not always work. Note that some "shielded" coils use ferrite cores to contain magnetic flux, but have no shielding. These are not recommended due to lower Q and the introduction of unecessary non-linearity to the circuit.

Finally, internally-matched SAW filters are available from some SAW filter vendors. This shifts the burden of matching-network parasitic coupling to the filter manufacturer. Cost will be somewhat higher and a larger package may are may not be required.

Matching Network Mutual Inductance

Mutual inductance between input and output matching inductors is another mode of coupling around the SAW filter. (See Figure 3b.) Any of the strategies described above for moving inductors farther apart or eliminating them from the matching networks may be useful. However, one strategy that should generally be assumed at the time of layout is the orthogonal positioning of input and output inductors with respect to each other.

Figure 4 is an excellent example of the impact of matching inductor orientation. The two plots are for the exact same filter in exactly the same layout and with exactly the same matching components. The only difference between the two is a 90 degree rotation of one matching inductor!

Orthogonal orientation sometimes conflicts with the goal of all components oriented in one direction for automated assembly. Also, it sometimes does not have much difference in filter performance. In the event of this conflict, be sure to confirm the need for orthogonal inductor orientation before committing to it.

Ground Currents

The interaction of input and output ground currents can be modeled as shown in Figure 3c. The degradation to SAW filter performance can be just as significant as from the other two filter layout problems. Unfortu-
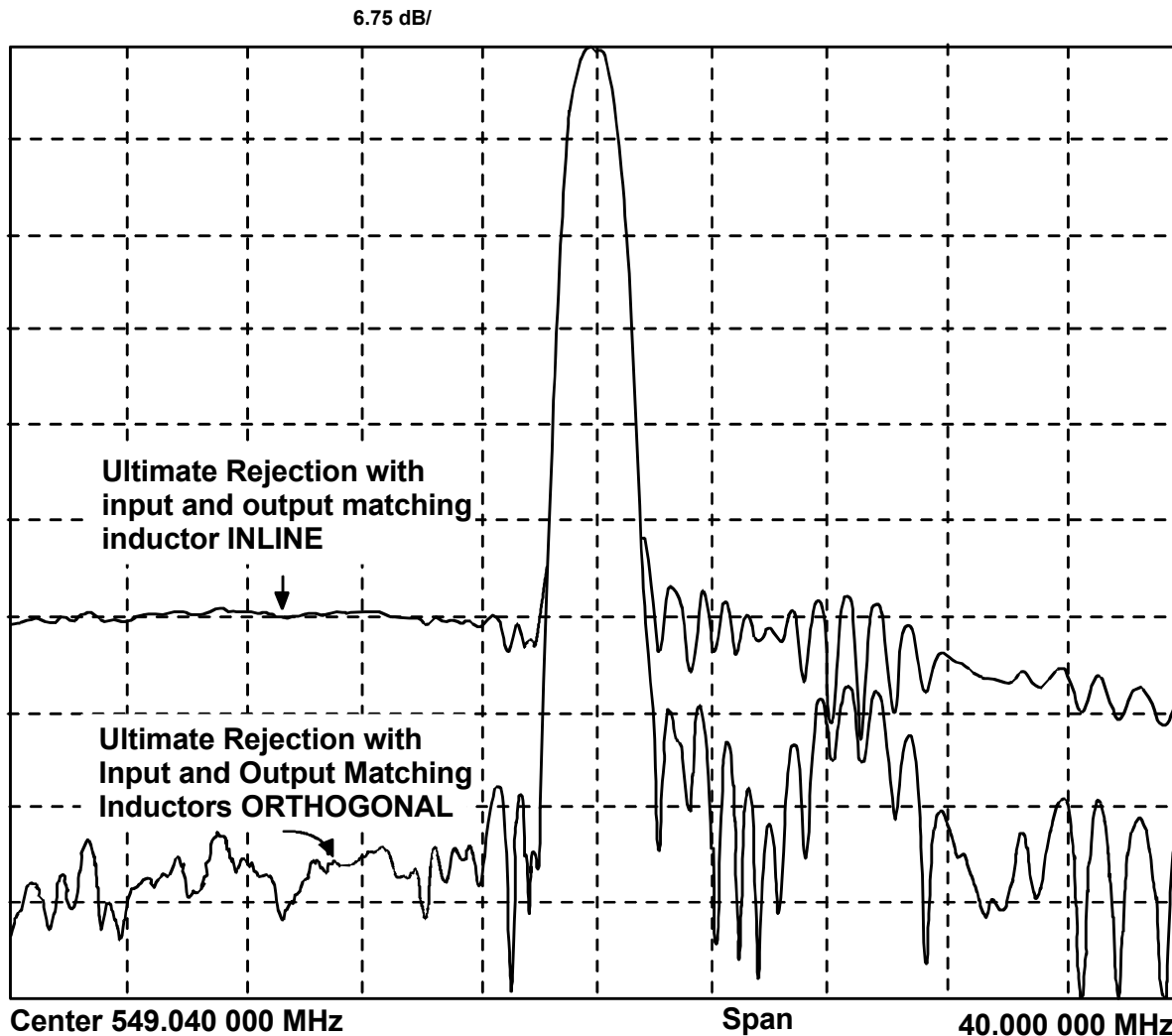
6.75 dB/

Ultimate Rejection with input and output matching inductor INLINE

Ultimate Rejection with Input and Output Matching Inductors ORTHOGONAL

Center 549.040 000 MHz   **Span**   **40.000 000 MHz**

Figure 4
Inline vs Orthogonal Inductors

nately, this issue is frequently overlooked until there is a problem.

At least one layer of solid ground plane directly under the filter, on the filter side, is essential. Multiple ground planes, usually two, can reduce ground-current resistance and inductance, but must be very solidly connected to each other. Pay very careful attention to all RF ground current paths and keep input and output ground currents as separate from each other as possible.

Many SAW filters have specified "return" terminals. Make ground paths between matching circuit grounds and specified return terminals as short and direct as possible. If no return connection is specified, return the

ground current to the terminal nearest the appropriate signal terminal, unless specified otherwise by the filter manufacturer. In Figure 2, terminals 6 and 12 are ground returns, and in Figure 5, Pins 2 and 9 serve this purpose.

Many applicatons require differential, or balanced inputs or outputs. SAW filters can be designed for this option. In that case, there are no ground returns and it is obvious where the "return" currents are traveling. All the same basic rules apply for unbalanced inputs and outputs except that it is easier to confine the "return" RF currents.

In some cases, slots in the ground plane may also be useful in minimizing the interaction of input and output
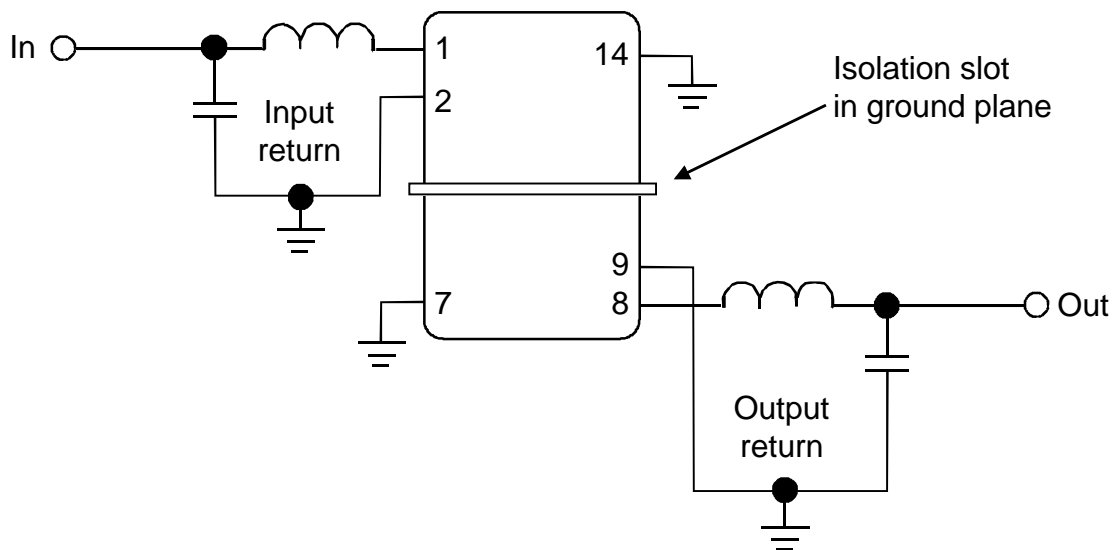
Figure 5
Typical Location of Ground Slot

ground currents by separating them. If more than one ground plane is used, the slot should be in all ground planes, with each plane solidly connected to the other(s) both at the slot and in the vicinity of the filter. A common technique for maximizing slot performance is to cut the slot into the dielectric and plate through the slot to connect two ground planes. Isolation slots are usually more useful for SAW filters in DIP packages than surface-mount packages. The location of an isolation slot is illustrated in Figure 5 for a SAW filter in a DIP package.

Matching Components

Design of matching networks is beyond the scope of this application note. However, there are some points that involve PCB layout that need to be mentioned. Most commercial applications use either no matching compo-

nents or fixed-value LC. These components may be as small as available as long as the minimum component Q is no less than 50. If size needs to be minimized (at the expense of insertion loss) then a minimum Q of 20 may be acceptable. These are issues that should be addressed at the brassboard stage, before the first PCB layout is done.

For many SAW filters, a 2-element LC match is all that is required. However, time to market is best minimized by doing the PCB layout prior to the availability of custom SAW filters. In that case, if there is sufficient real estate on the PCB, the matching networks can be layed out to accomodate a PI match consisting of typically 2 parallel shunt C's, one or two series L's, and two parallel shunt C's. In production, this board layout would be overkill. However, this can be a very useful technique for getting the fastest results from the prototype. This is illustrated in Figure 6.
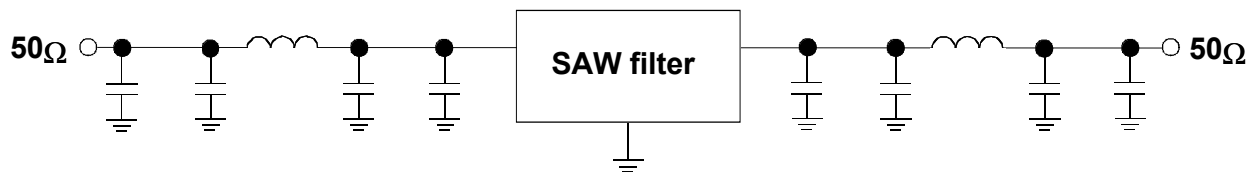

Figure 6
Typical Prototype Board Layout

Conclusion

The requirements for optimizing PCB layout for best in-band and ultimate rejection performance of SAW filters can be summarized in three simple rules:

1. Keep input and output circuits as far apart as possible, within the constraint of keeping those same components as near to the filter connections as possible. Various shielding strategies can be used as a last resort.

2. Orient input and output matching inductors orthogonally to each other to minimize the mutual inductance between them.

3. A solid ground plane under the filter is necessary. Pay very careful attention to ground current paths and keep input and output ground currents as separate from each other as possible. Make the shortest ground path to the specified "return" terminal if the filter manufacturer specifies any. If no return connection is specified, return the ground current to the terminal nearest the appropriate signal terminal. In some cases, slots in the ground plane may also be useful for maximum performance.

Following these rules at the time of the PCB design results in the best possible performance from the SAW filter, reduces the number of design iterations, and reduces the cost and time to market. Ignoring these rules can result in both passband ripple and ultimate rejection problems that may be impossible to solve until the PCB is redesigned.

References

1. P. Dufilié, J. Desbois, "Modeling of Feedthrough and Ground Loops in SAW Filters," Proceedings IEEE Ultrasonics Symposium, 1993, pp 223-226.

2. A. Coon, "Optimizing Circuit Boards for SAW Filters," Portable Design, May 1997.