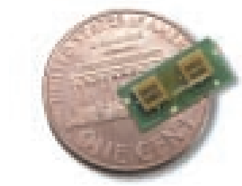
HPMD-7904

FBAR Duplexer for US PCS Band

Data Sheet



General Description

The HPMD-7904 is a miniaturized duplexer designed for US PCS handset, designed using Avago Technologies' Film Bulk Acoustic Resonator (FBAR) Technology. The HPMD-7904 features a very small size: it is less than 2 mm thick and has a footprint of only 5.6 x 11.9 mm^2.

The HPMD-7904 enhances the sensitivity and dynamic range of CDMA receivers, providing more than 50 dB attenuation of transmitted signal at the receiver input, and more than 40 dB rejection of the transmit-generated noise in the receive band. Typical insertion loss in the Tx channel is only 1.8 dB, minimizing current drain from the power amplifier. Typical insertion loss in the Rx channel is 2.2 dB, improving receiver sensitivity.

Avago's thin-Film Bulk Acoustic Resonator (FBAR) technology makes possible high-Q filters at a fraction their usual size. The excellent power handling of the bulk-mode resonators supports the high output powers needed in PCS handsets, with virtually no added distortion.

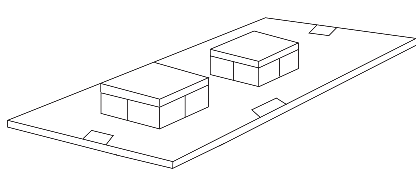
Features

- Miniature size: less than 2 mm high;
  5.6 x 11.9 footprint
- Rx Band: 1930-1990 MHz
  Typical performance:
  Rx noise blocking: 42dB
  Insertion loss: 2.2 dB typical
  3.0 dB band edge
- Tx Band: 1850-1910 MHz
  Typical performance:
  Tx interferer blocking: 54dB
  Insertion Loss: 1.8 dB typical
  2.5 dB band edge
- 30 dBm Tx power handling

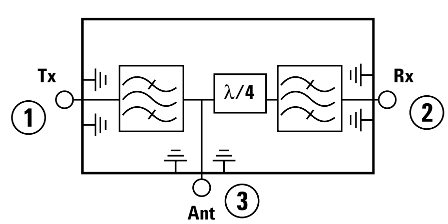
Applications

- Handsets or data terminals operating in the US PCS frequency band

Board Diagram



Functional Block Diagram



port numbers are circled

HPMD-7904 Electrical Specifications, $Z_0 = 50\Omega$, T_C[1] as indicated

Symbol	Parameters	Units	+25°C[1,3] Min	Typ	Max	+85°C[1,2,3] Min	Typ	Max	–30°C[1,2,3] Min	Typ	Max
f_{RX}	Receive Bandwidth	MHz	1930.6	—	1989.4	1930.6	—	1989.4	1930.6	—	1989.4
S23 Rx	Attenuation in Transmit Band (1850.6 – 1909.4 MHz)	dB	50	54	—	50	52	—	50	52	—
S23 Rx	Typical Insertion Loss in Receive Band (1935 – 1985MHz)	dB	—	2.2	3.5	—	2.2	3.8	—	2.2	3.8
S23 Rx	Insertion Loss at Edges of Receive Band (1930.6-1935 MHz and 1985-1989.4 MHZ)	dB	—	3.0	3.5	—	3.0	3.8	—	3.0	4.5
Δ S23	Ripple in Receive Band	dB	—	1.5	—	—	1.5	—	—	2.0	—
S22 Rx	Return Loss in Receive Band	dB	8.0	10	—	8.0	10	—	8.0	10	—
f_{TX}	Transmit Bandwidth	MHz	1850.6	—	1909.4	1850.6	—	1909.4	1850.6	—	1909.4
S31 Tx	Attenuation in Receive Band (1930.6 – 1935 MHz)	dB	40	42	—	40	42	—	37	42	—
S31 Tx	Attenuation in Receive Band (1935 – 1989.4 MHz)	dB	40	42	—	40	42	—	40	42	—
S31 Tx	Insertion Loss in Transmit Band (1855 – 1905 MHz)	dB	—	1.8	3.0	—	1.8	3.5	—	1.8	3.5
S31 Tx	Insertion Loss at Edges of Transmit Band (1850.6-1855 MHz and 1905-1909.4 MHz)	dB	—	2.5	3.0	—	3.0	3.8	—	3.0	3.6
Δ S31	Ripple in Transmit Band	dB	—	2.0	—	—	3.0	—	—	2.0	—
S11 Tx	Return Loss in Transmit Band	dB	8.0	10	—	8.0	10	—	8.0	10	—
S33 Ant	Return Loss, Tx and Rx bands	dB	8.0	—	—	—	—	—	—	—	—
S21	Tx-Rx Isolation, 1850.6 – 1909.4 MHz (Transmit Band)	dB	50	54	—	50	54	—	50	54	—
S21	Tx-Rx Isolation, 1930.6 – 1935 MHz (Receive Band)	dB	40	42	—	40	42	—	38	42	—
S21	Tx-Rx Isolation, 1935 – 1989.4 MHz (Receive Band)	dB	40	42	—	40	42	—	40	42	—

Absolute Maximum Ratings[4]

Parameter	Unit	Value
Operating temperature[1]	°C	-30 to +85
Storage temperature[1]	°C	-30 to +100

Notes:

1. T_C is defined as case temperature, the temperature of the underside of the duplexer where it makes contact with the circuit board.
2. Specifications are given at operating temperature limits and room temperature. To estimate performance at some intermediate temperature, use linear interpolation.
3. Specifications are guaranteed over the given temperature range, with the input power to the Tx port equal to +30 dBm (or lower) over all Tx frequencies. The application of input power levels in excess of +30 dBm will not destroy the duplexer, but its performance may exceed the specification limits given above.
4. Operation in excess of any one of these conditions may result in permanent damage to the device.

The plots below provide typical performance obtained from samples of the HPMD-7904 duplexer.

In order to obtain the best performance from the HPMD-7904 duplexer, refer to Design Note D007, which is available from your Avago Technologies technical support or sales departments.

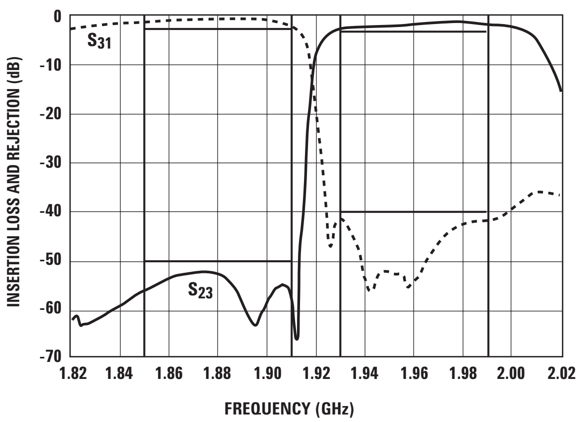


Figure 1. Tx and Rx Port Insertion Loss (typical).

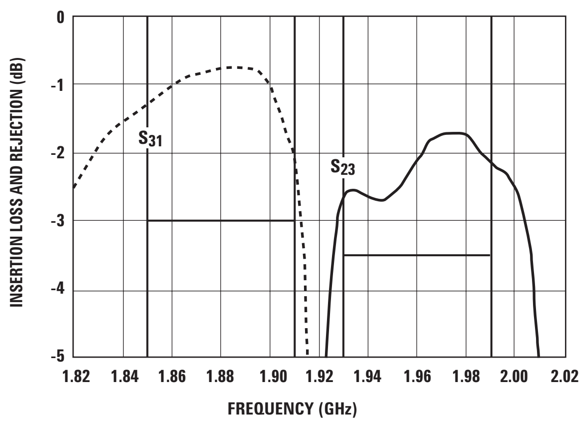


Figure 2. Insertion Loss Detail (typical).

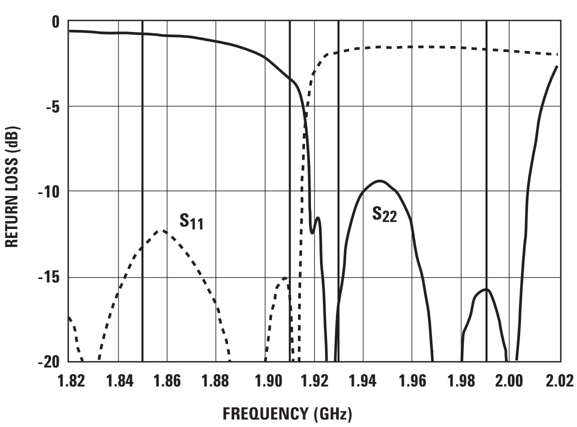


Figure 3. Tx and Rx Port Return Loss (typical).

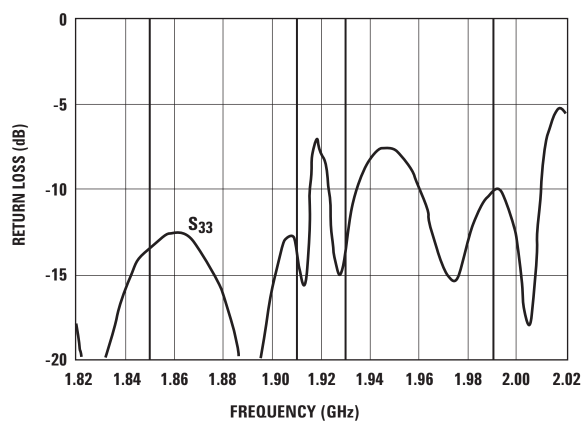
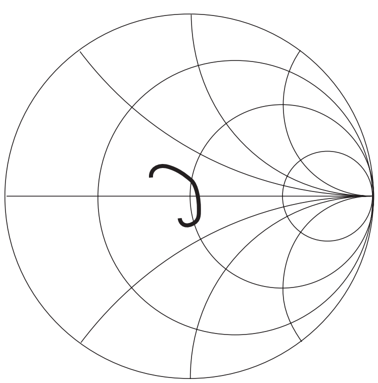
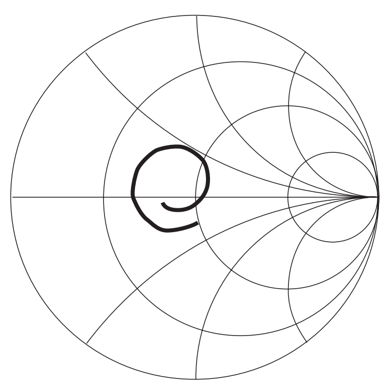


Figure 4. Ant Port Return Loss (typical).



freq (1.850 GHz to 1.910 GHz)

Figure 5. S11, Tx Port Impedance (typical).



freq (1.930 GHz to 1.990 GHz)

Figure 6. S22, Rx Port Impedance (typical).

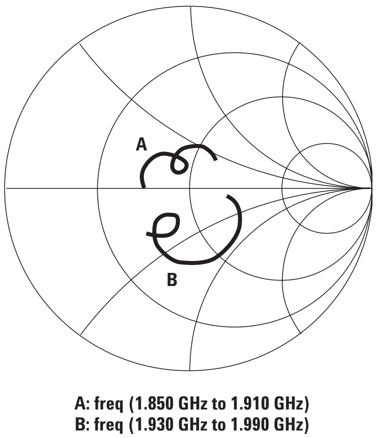


Figure 7. S33, Ant Port Impedance.

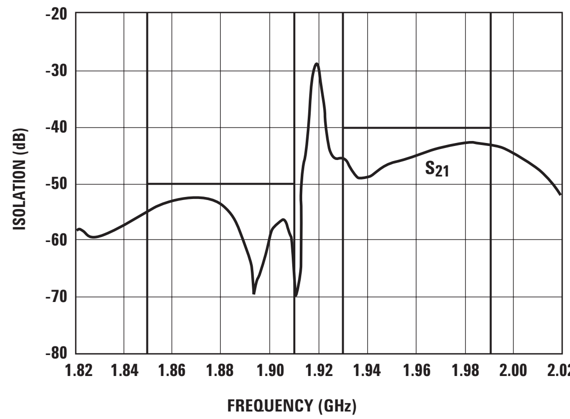


Figure 8. S21, Isolation (typical).

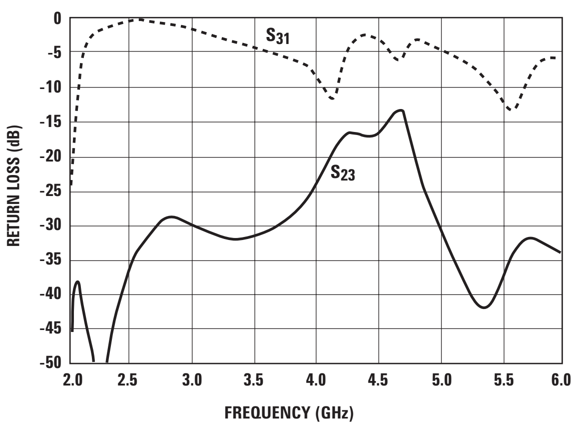


Figure 9. Wideband Insertion Loss (typical).

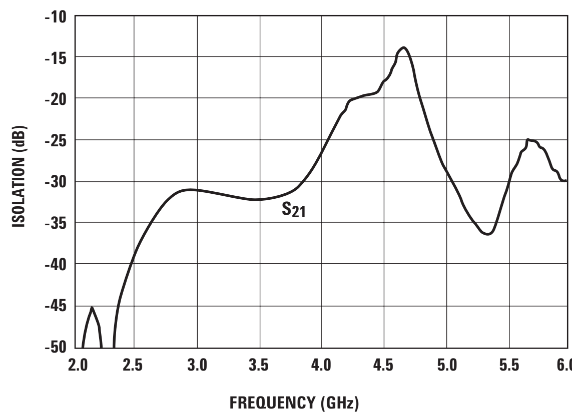


Figure 10. Wideband Isolation (typical).

Note that the specifications given on page 2 are guaranteed when the duplexer is mounted on a ground surface with a hole pattern like that one shown in Figure 11. See Design Note D007, which is available from your Avago Technologies technical support or sales departments.

Note that it is important that proper heat sinking be provided in order to remove the heat generated in the Tx filter by the handset's power amplifier. Failure to do so may result in excessive losses, especially at the top end of the Tx band.

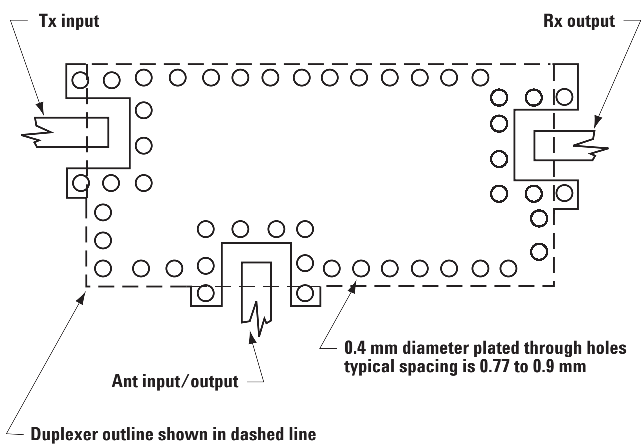


Figure 11. Mounting (grounding) Pattern.

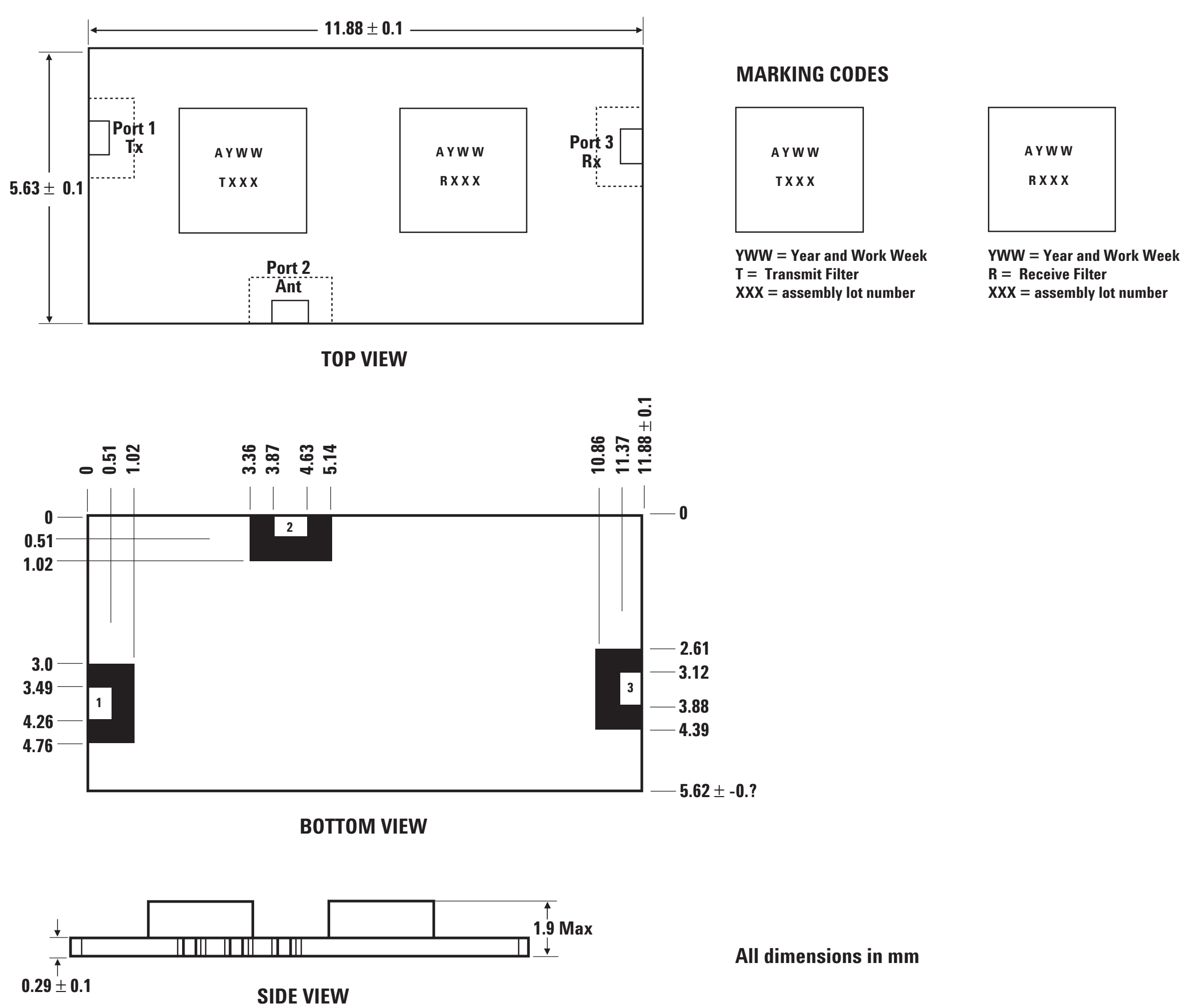


Figure 12. Outline Drawing.

Using the HPMD-series CDMA Duplexers

Avago's FBAR (Film Bulk Acoustic Resonator) duplexers provide high RF performance in a very small package. However, in order to achieve all the performance available from the duplexer, care must be taken in the design of the board onto which it is mounted. The purpose of this Design Tip is to provide Avago's recommendations on the design of that board (called the motherboard in this note).

Areas where care in design must be observed are **thermal ground**, **RF ground**, **in/out connection design**, and **solder mask/solder stencil design**. These four design areas, which are sometimes interrelated, will be considered one at a time below (along with comments on demo boards and S3P S-parameter files).

Thermal Ground

FBAR resonators have a negative temperature coefficient of frequency—as temperature goes up, the frequency response of the filter shifts down in frequency. See Figure 13. Typical coefficients are 57 KHz/°C for the Tx filter and 40 KHz/°C for the Rx filter. In Figure 14, the same data are presented with the scale narrowed down to the upper end of the Tx band. Note that all these data are taken at low input power levels (+10 dBm).

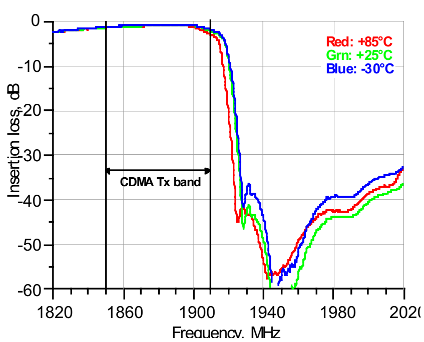


Figure 13. Tx Filter Response with Temperature.

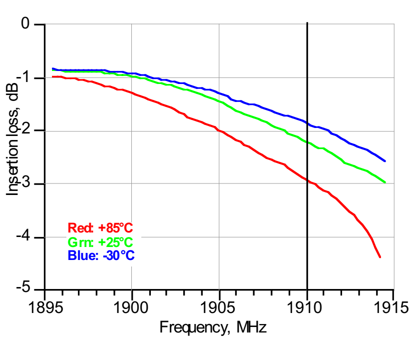


Figure 14. Tx Filter Response with Temperature (expanded).

When input power is +29 or +30 dBm, heating in the Tx filter due to RF losses causes the filter membranes to heat up beyond 85°C. This, in turn, causes the filter response to shift further left (down in frequency), resulting in increased insertion loss at the high end of the Tx band (1910 MHz). The only place where a problem may develop is at the top end of the Tx band. Avago Technologies takes this into account in the manufacture and final test of the duplexer—all specifications for insertion loss (and other parameters) will be met at the specified input power level and motherboard temperature. Note that high power/high temperature testing done at Avago is performed with the duplexer soldered down to a test board having a very good heat sink.

The motherboard must be designed to remove heat from the duplexer with the lowest possible thermal resistance. Mount the duplexer on a large surface of _ ounce copper (as shown in Figure 15), to enable the heat to be removed in all directions. Via holes, necessary for RF grounding, should be filled with copper plating to further remove heat from the duplexer's Tx filter and dump it into a second ground plane located in a lower layer of the motherboard.

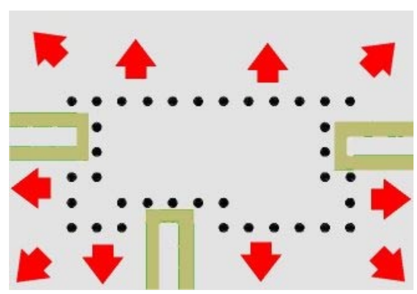


Figure 15. Thermal and Electrical Ground Plane.

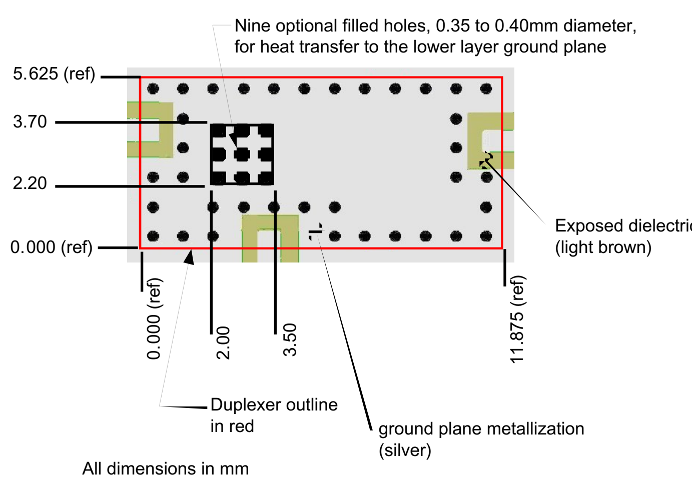


Figure 16. Thermal Via Holes.

An additional set of nine thermal holes (copper filled) may be added to remove heat from the Tx filter as shown in Figure 16.

FBAR duplexers have extremely low thermal mass and must be properly heat sunk, as well as isolated from external sources of heat (such as a nearby power amplifier). Failure to provide an adequate thermal design to cool the duplexer may result in degradation of insertion loss at the high end of the Tx band, or other performance problems.

RF Ground

Many of the same considerations which apply to the thermal ground plane also apply to the RF ground plane. A large area, as shown in Figure 3, provides data sheet performance for the duplexer. In addition, a series of 36 (or more) plated and plugged via holes to a lower ground plane (0.35 to 0.40 mm in diameter) should be provided around the perimeter of the duplexer, as shown in Figure 16.

If a continuous ground plane cannot be provided on the top surface of the motherboard, a ground patch which extends 0.5 mm (or more) out from each side of the duplexer can be used. However, in this case the 36 (or more) plugged via holes connecting to a lower level ground are essential for good performance.

In/out Connection Design

High isolation between ports is a characteristic of Avago FBAR duplexers, and values often exceed 50 dB. In order for this performance to not be degraded by coupling within the user's motherboard will require 60dB or more of isolation between each pair of the three input/output lines of that motherboard. This is measured as shown in Figure 17, using copper pins to short out all three of the lines to ground. Isolation is then measured between the Tx, Rx and Ant lines, with 60 dB as a minimum acceptable level.

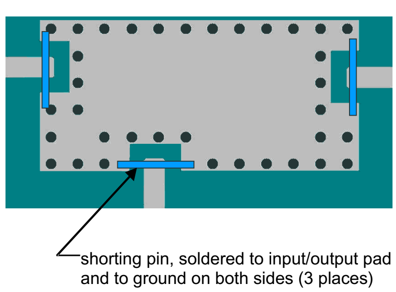
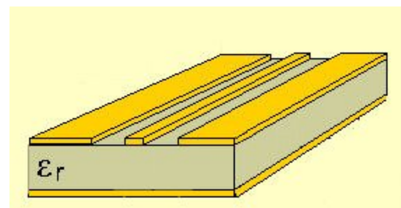


Figure 17. Motherboard Isolation Test Method.

In order to achieve 60 dB of isolation in the motherboard itself using microstrip transmission lines, extreme care must be taken in the design due to the poorly contained field lines in microstrip. Ground areas between the lines and other good techniques will have to be used. The use of coplanar waveguide over a groundplane (CPWG) provides for higher line-to-line isolation.



Design guidelines for CPWG transmission lines can be found in AppCAD, the design/analysis software found on the Web at http://www.semiconductor.avagotech.com/cgi-bin/morpheus/wirelessDesignTool/utility.jsp?flag=App. Figure 18 provides the recommended dimensions for a CPWG realization. Shown in the figure is a solder mask (green)—this will be dealt with in detail below.

Better performance can be obtained using stripline, where a buried conductor has aground plane on layers above and below. Such transmission lines have excellent shielding, and line-to-line values in excess of 80 dB can easily be achieved using this transmission line. The details of this transmission line realization are shown in Figures 19 and 20. Small in/out

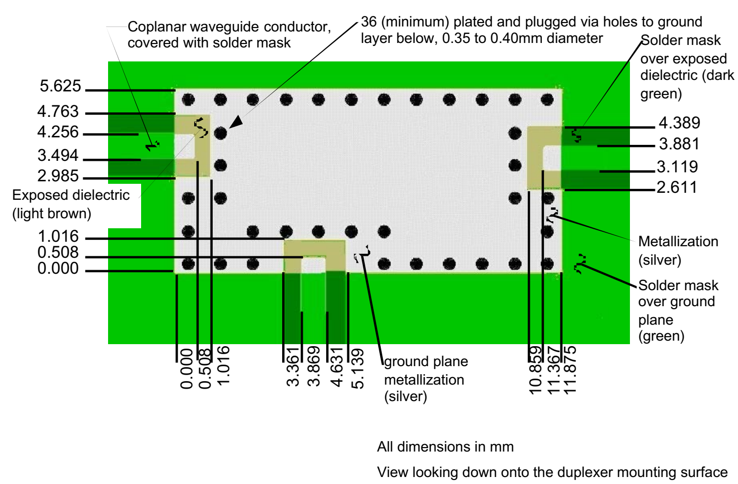
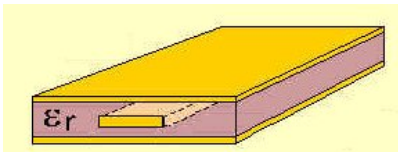


Figure 18. Recommended mounting for CPWG realization.

pads are provided on the top surface, extending beyond the edges of the duplexer (Figure 19). These are connected to the buried stripline conductors by means of two plated and plugged via holes (each).



In this buried layer, the three lines are isolated by large ground areas, connected to the ground plane above and below. This provides for the best containment of transmission line fields.

Solder Mask and Solder Stencil

Solder mask is used on all motherboards, to prevent solder from adhering to places where it is not desired. In mounting the duplexer, it serves as an aid to alignment during wave soldering.

The motherboard solder mask (shown in green in Figures 18 and 19) should correspond to the edges of the duplexer. This will contain the solder during reflow, and prevent the duplexer from rotating or slipping out of alignment.

A solder stencil is used to print a pattern of solder paste onto the motherboard, with the duplexer placed upon this paste before the reflow process begins. The design of the solder stencil is critical to obtaining good yields in reflow soldering.

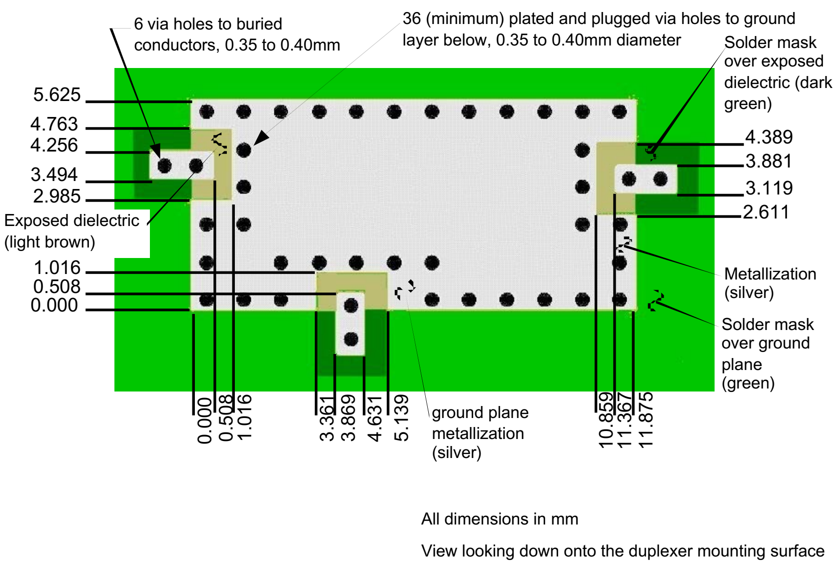


Figure 19. Top layer for stripline realization.

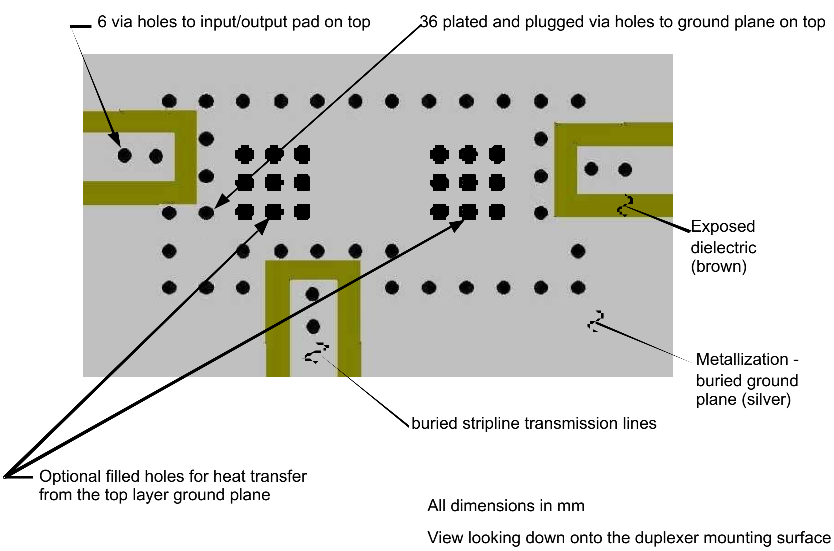


Figure 20. Buried layer for stripline realization.

It is the nature of PBC (printed circuit board) products that there is some warp in them, especially in the long dimension. The FBAR duplexer will exhibit a small amount of lifting on the two narrow faces (where the Tx and Rx pads are located) when it is placed upon a flat surface. See the illustration above. In order to achieve data sheet performance, the Tx pad and two ground connections on each side, as well as the Rx pad with two ground connections on each side, must be soldered to mating lands on the motherboard. Thus it is necessary to screen down a pattern of solder paste that will allow the center of the duplexer to be closer to the motherboard, bringing the Tx and Rx contacts down to their solder paste pads. See Figures 21, 22 and 23. These stripes provide approximately 50% coverage—that is, 50% of the ground pad is covered with solder paste. This is equivalent to laying down a continuous layer of solder paste at half the thickness of the in/out pads and their ground pads.

Solder Materials

The recommended solder profile for the FBAR duplexer is shown in Figure 24. Guidelines are given in red, and a typical profile is shown in blue. This typical profile was tested on ten samples of the duplexer, each of which was subjected to the profile six times without effect upon the mechanical or electrical characteristics of the device. Solder temperatures and times in excess of those given in the guidelines of Table 1 may result in damage to the duplexer or changes in its characteristics.

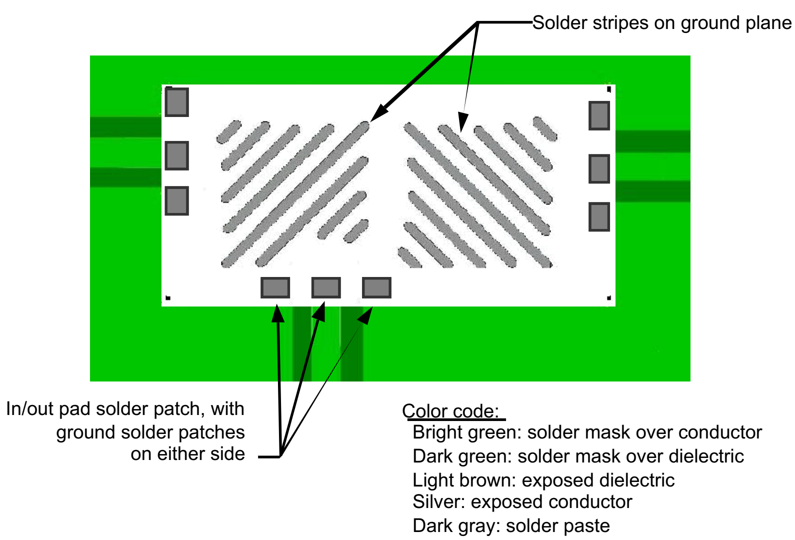


Figure 21. Duplexer solder paste pattern.

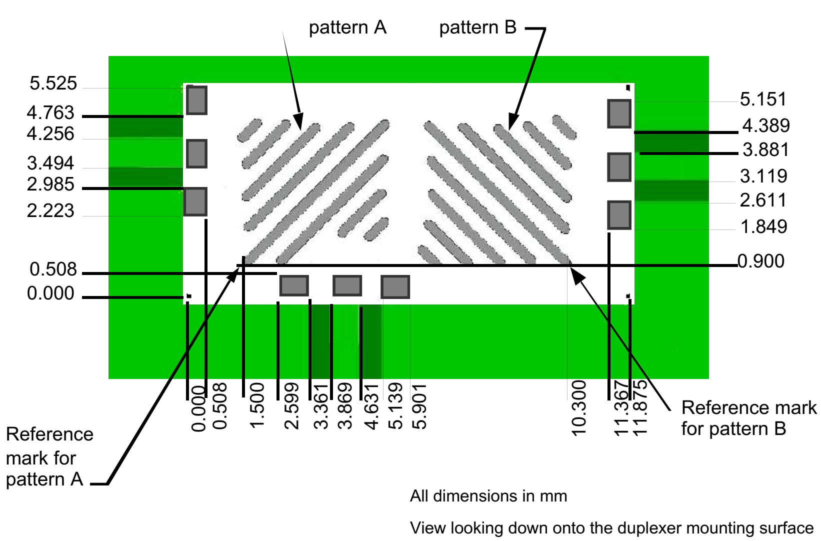


Figure 22. Duplexer solder stencil dimensions.

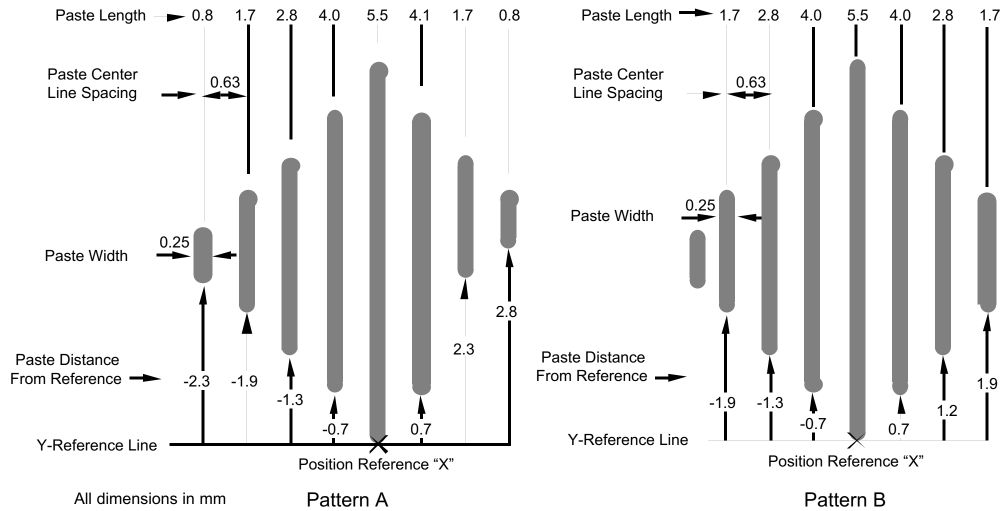


Figure 23. Solder stencil details.

Table 1. Solder Compositions.

Alloy type	Melting temp. (°C)	Recommended working temperature (°C)	Comments
Sn42Bi58	138	160 – 180	Lead free
Sn43Pb43Bi14	144 – 163	165 – 185	Contains lead – some customers prohibit it.
Sn63Pb37	183	200 – 240	Contains lead – some customers prohibit it.
Sn60Pb40	186	200 – 240	Contains lead – some customers prohibit it.
Sn91/Zn9	199	200 – 240	May have oxidation problems
Sn96.2Ag2.5Cu0.8Sb0.5	216	235 – 255	Popular lead free composition
Sn95.8Ag3.5Cu0.7	217	235 – 255	Other alloy ratios are available
Sn96.5Ag3.5	**221**	**240 – 260**	**Used in the assembly of duplexers**
Sn100	232	260 – 280	Too hot – will melt the duplexer
Sn95Sb5	235	260 – 280	Too hot – will melt the duplexer
Sn97Cu3	240	260 – 300	Too hot – will melt the duplexer

Solder Recommendations

The HPMD-7904 FBAR duplexer (and its variants) is an assembly consisting of two LCC ceramic packages, containing the Tx and Rx filters, mounted to a small circuit board. Both packages on the circuit board are mounted in place using Sn96.5/Ag3.5 solder (shaded in Table 1).

The recommended solder profile for the FBAR duplexer is shown in Figure 16. Guidelines and a typical profile are both shown. This typical profile was tested on ten samples of the duplexer, each of which was subjected to the profile six times without effect upon the mechanical or electrical characteristics of the device. Solder temperatures and times in excess of those given in the guidelines of Table 1 may result in damage to the duplexer or changes in its characteristics.

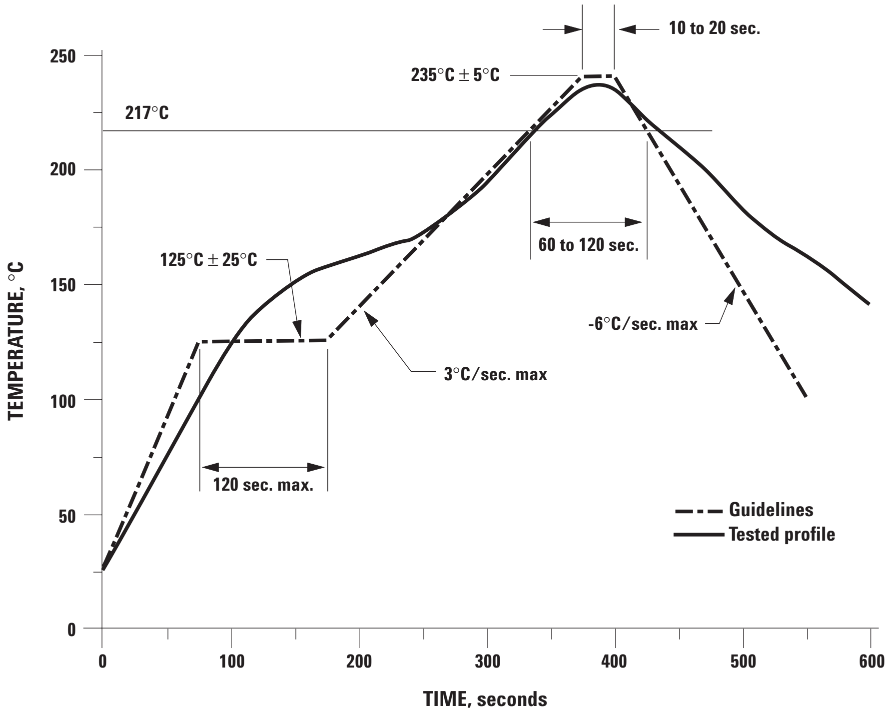


Figure 16. Recommended Solder Profile.

Duplexer Demo Board

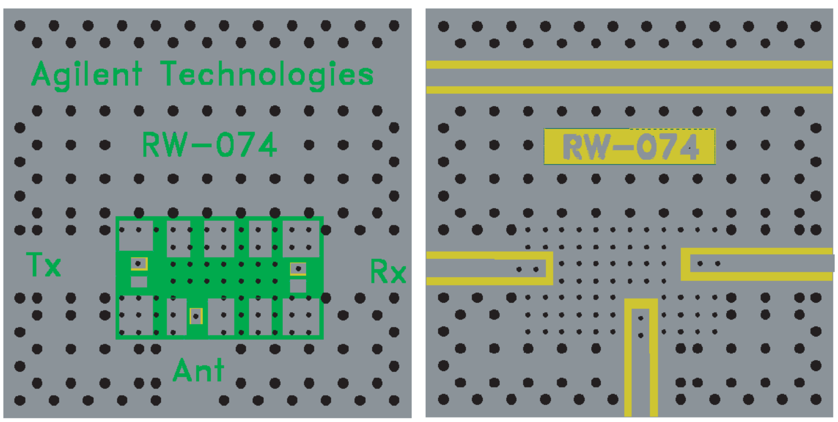
The RW074A duplexer demo board (Figure 25) utilizes a CPWG transmission lines for high isolation between the three ports. Ground plane spacing is 0.032" (0.81 mm). It is designed for use with E.F. Johnson 142-0701-881 end launch SMA connectors (see Figure 26).

This demo board illustrates the design concepts described earlier in this note. It uses via holes (two per connection) to connect the CPWG transmission line on the underside of the board to the duplexer mounting pads on top, with ground planes separating all lines and pads.

The board also has a 50Ω reference line, which can be used to calibrate out the insertion loss of the board, and which will provide the user with information on the quality of connector mounting. The return loss of that reference line should be higher than 20 dB over the 1.5 to 2.5 GHz range. If it is not, the connectors were improperly mounted.

In order to obtain the best performance from this demo board, care must be taken to mount the connectors and duplexer properly. The first step is to reflow solder the duplexer into position on the demo board, using a solder paste with a melting temperature of 190°C or less. Reflow the solder using a preheated hotplate. As soon as the solder paste flows, remove the board and allow it to cool to room temperature.

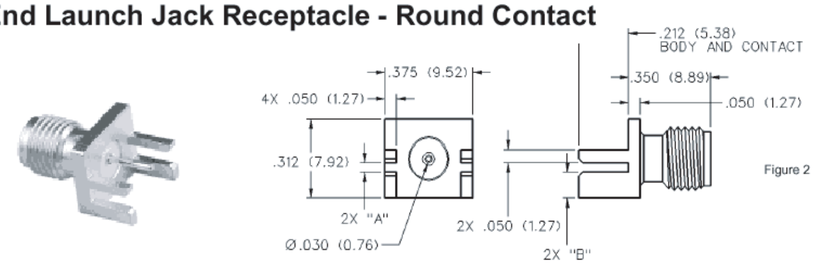
Prepare the connectors by trimming the pins down to 1.0 to 1.5 diameters (0.050" to 0.75") in length. Mount the connectors with the board upside down (duplexer down) and clamped in a small vise to assure that the connector body is firmly pressed against the edge of the board. Solder the pin to the transmission line, and then solder the two ground fingers to the ground planes on either side of the line. See Detail A in Figure 27.



Top (duplexer) side Bottom (connector) side

Figure 25. RW074A demo board.

End Launch Jack Receptacle - Round Contact



VSWR & FREQ. RANGE	BOARD THICKNESS	GOLD PLATED	NICKEL PLATED	"A"	"B"	FIGURE
VSWR: N/A 0-18 GHz	.062 (1.57)	142-0701-801	142-0701-806	.068 (1.73)	.073 (1.85)	1
	.042 (1.07)	142-0701-831	142-0701-836	.048 (1.22)	.093 (2.36)	1
	.031 (0.79)	142-0701-881	142-0701-886	.037 (0.94)	.104 (2.64)	2
	.047 (1.19)	142-0711-871	142-0711-876	.053 (1.35)	.088 (2.24)	1

Figure 26. EF Johnson connector.

Turn the board over and solder the connector fingers to the board's top ground plane, taking care to flood the area with sufficient solder as to completely cover the small crescent of exposed teflon. See Detail B in Figure 27.

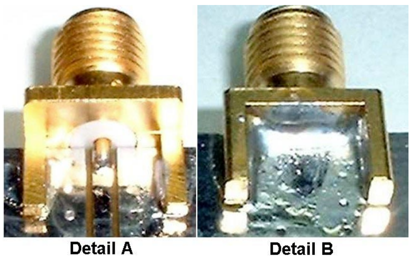


Figure 27. Connector details.

To obtain a demo board, contact your Avago Technologies sales representative.

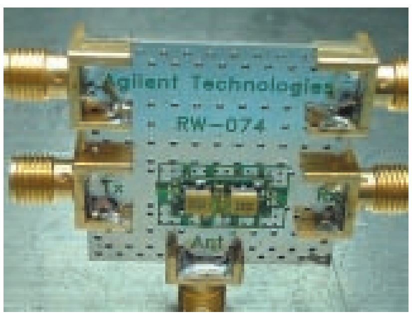


Figure 28. Assembled demo board.

Duplexer S-parameter Files

Three port S-parameter files are available for the Avago FBAR duplexer, both in narrow band form (1820 to 2020 MHz in 0.5 MHz steps) and wideband form (1 to 6 GHz). These files are in Touchstone format, compatible with ADS and other simulation software. They are useful in performing simulations of the duplexer under various conditions, with external matching networks, parasitics, etc.

However, care must be exercised when considering an external impedance matching network for the duplexer. Consider the simple L-C network simulated in ADS and shown in Figure 29. This network was added to the Tx port in an attempt to increase the return loss over the 1850 to 1880 MHz portion of the Tx band. Figure 30 shows the effect of this matching network, with "before" given in red and "after" in blue. The goal of return loss improvement is made. However, an examination of Figure 31 will show that a small reduction in rejection over the upper half of the Rx band resulted from the addition of this impedance matching network. In Figure 32, it will be seen that Tx to Rx port isolation was slightly improved at the low end of the Rx band (1935 MHz) and reduced over the high end of the Rx band, with no substantial change over the Tx band.

Bear in mind that the addition of any external impedance matching network, or parasitic reactance (such as from poor quality connectors) will impact the duplexer's rejection and isolation performance.

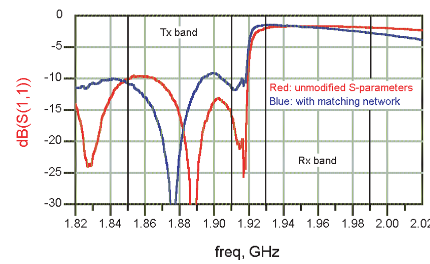


Figure 29. Network simulated.

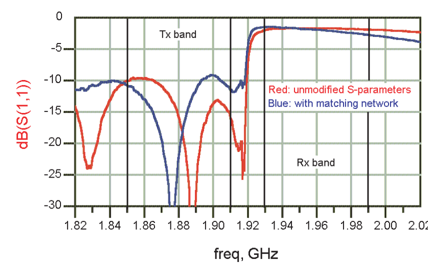


Figure 30. Tx port return loss.

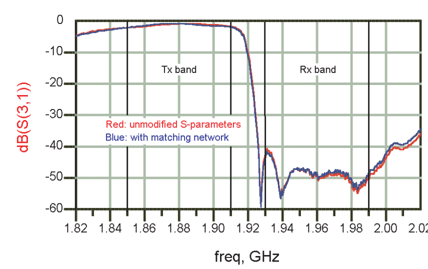


Figure 31. Tx to Antenna insertion loss.

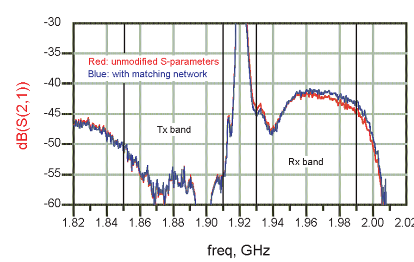
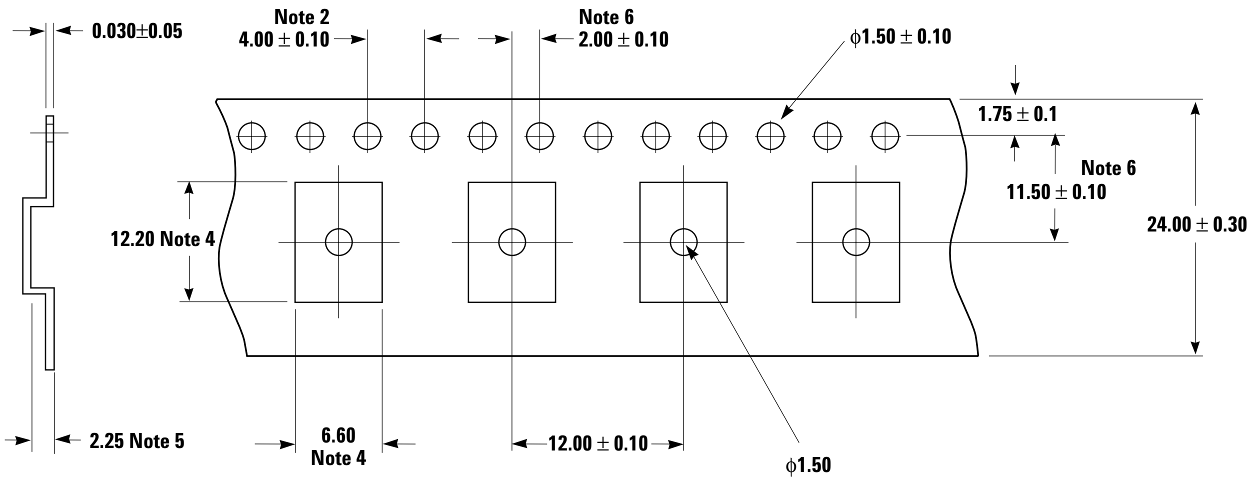
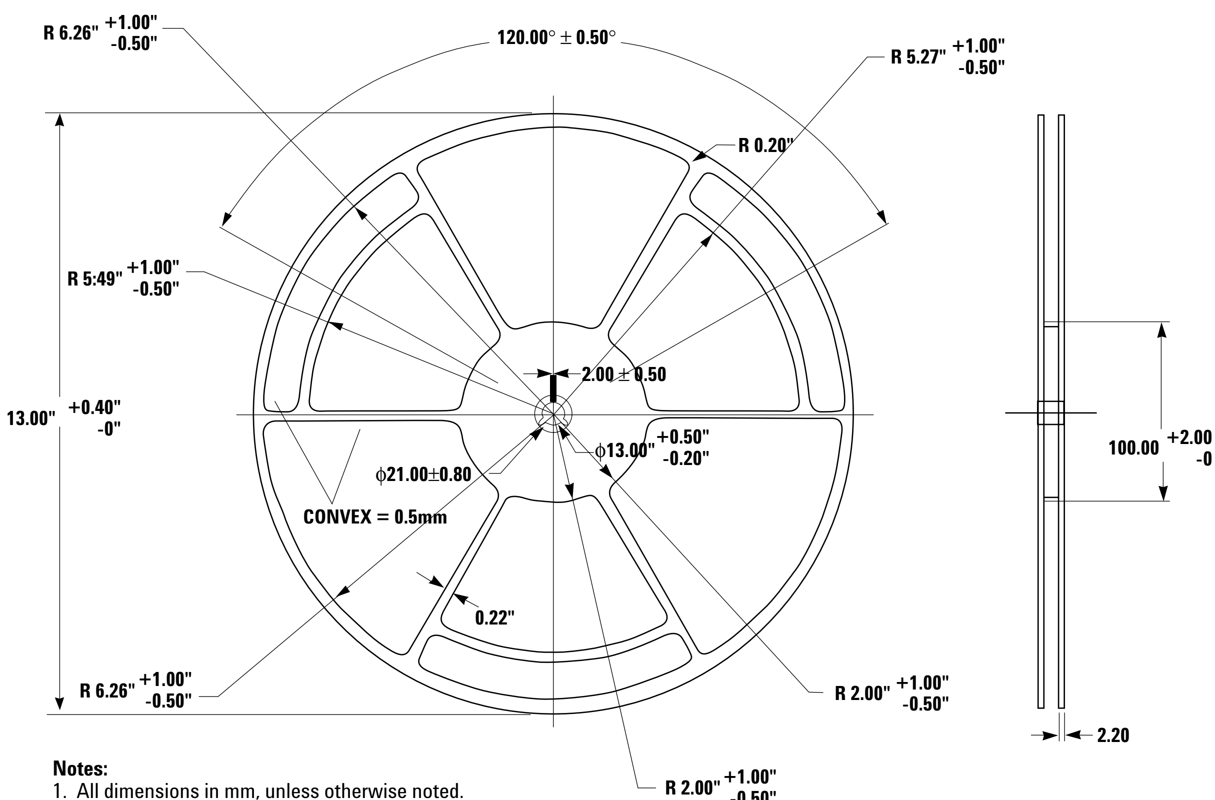


Figure 32. Tx to Rx isolation.



Notes:
1. All dimensions in mm.
2. 10 sprocket hole pitch accumulative tolerance ± 0.10 mm
3. Camber not exceed 1 mm in 250 mm
4. Pocket dimensions measured on a plane 0.3 mm above the bottom of the pocket.
5. Pocket depth measured from a plane on the inside bottom of the pocket to the top surface of the carrier.
6. Pocket position on relative to sprocket hole measure as true position of pocket, not the pocket hole.

Figure 33. Tape Drawing.



Notes:
1. All dimensions in mm, unless otherwise noted.
2. Material: polystyrene
3. Surface resistivity: 1x10^9 ohm-mm^2

Figure 34. Reel Drawing.

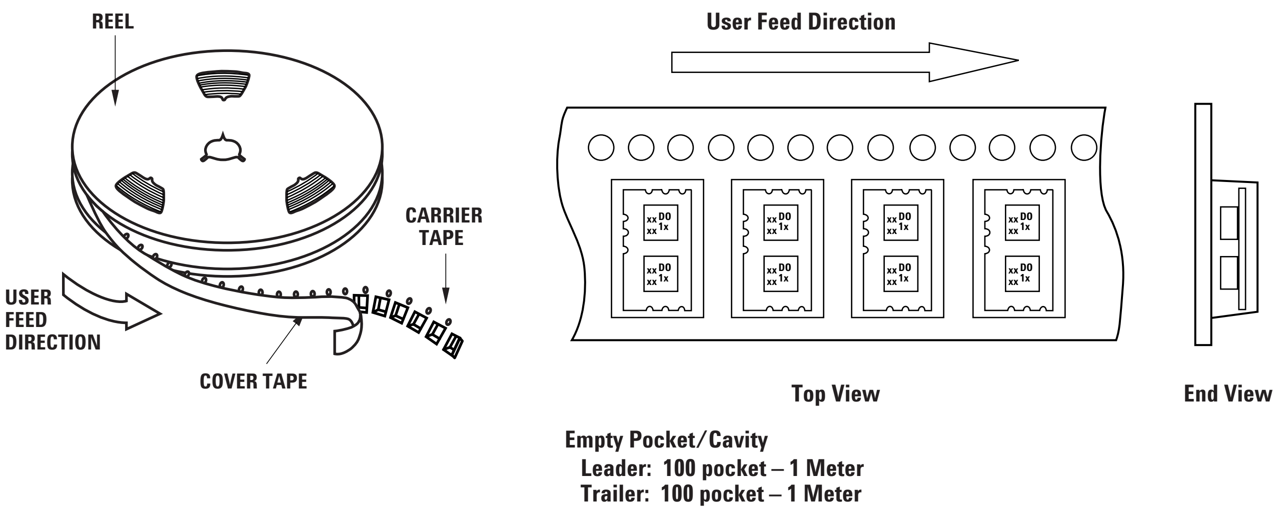


Figure 35. Device Orientation.

Ordering Information

Part Number	No. of Devices	Container
HPMD-7904-BLK	25	Anti-static bag
HPMD-7904-TR1	1000	13” Reel

For product information and a complete list of distributors, please go to our web site:
www.avagotech.com

5988-6610EN May 23, 2006