cannot resolve by moving your existing routes around, then your placement isn't right yet. In that case, you *must* rip up *all* routes in the area, fix the placement, and then start the routing again. Don't be tempted to ignore any routing problem you know about, no matter how insignificant. A single trace, via, or component in the wrong place has put many an RF board into the scrap heap, and caused many more board revisions. It's much more work to do the board over than to do it right the first time!

Special RF Routing Techniques: Microstrip

When routing high-frequency RF signals, the traces themselves become an important part of the circuit. Remember the high-speed digital circuit boards discussed above? The traces on those boards are sometimes called *transmission lines*, because of the reflections that happen at their ends. The solution there was to terminate the lines with resistors to try to reduce or eliminate these reflections. A different solution is often used on RF transmission lines: the lines themselves are designed to match the terminating impedance inherent in the RF circuitry. Two ways to achieve this are with the use of *microstrip* or *stripline*. These are types of *controlled-impedance* traces. They are easy to design and build, even for the home hobbyist. We will discuss microstrip first and then stripline in the next section.

Which RF traces should be treated as transmission lines? In general, all high frequency or long traces should be considered transmission lines. But how high is high or how long is long? A rule of thumb to use is that any trace that is longer than 1/16 of a *wavelength* will act as a transmission line. To compute a signal's wavelength, use the formula:

$$\lambda = \frac{11232}{F}$$

where λ is the wavelength (in inches) and F is the signal's frequency (in MHz). This equation includes an approximation to account for the fact that your signal will travel slightly slower on a circuit board than it would in free space.

For example, a 915-MHz signal has a wavelength of about twelve inches, so a transmission line would be any trace longer than about 0.8 inch. Note that this doesn't mean that you should carefully route a two-inch trace as a transmission line while carelessly routing a 0.7-inch trace. Once you have one controlled impedance transmission line on your board, it is just as easy to make all of your RF traces into transmission lines. See Table 3-1 for a list of frequencies, wavelengths, and transmission line lengths. The table also includes the length of a quarter-wave dipole antenna for each frequency. A quarter-wave dipole is a very common type of antenna; it consists of a quarter-wave conductor mounted over a ground plane. You can see that, at high frequencies, circuit board traces can easily become efficient antennas!

Table 3-1: Signal Frequencies, Wavelengths, and Transmission Line Lengths

Signal Frequency	Wavelength		Minimum Transmission Line Length	Dipole Antenna Length
(MHz)	(inches)	(feet)	(inches)	(inches)
1	11232.0	936.0	702.0	2808.0
10	1123.2	93.6	70.2	280.8
20	561.6	46.8	35.1	140.4
50	224.6	18.7	14.0	56.2
100	112.3	9.4	7.0	28.1
500	22.5	1.9	1.4	5.6
1000	11.2	0.9	0.7	2.8

Once you have identified which traces on your board are to be treated as transmission lines, then it is time to decide on the type of transmission line to use. There are many different types to choose from. You are already very familiar with the most common type of transmission line, the coaxial cable. It certainly works well, as millions of cable TV viewer will attest to. Unfortunately, coaxial transmission lines are difficult to fabricate on circuit boards. As previously noted, the two most common types of transmission line used on circuit boards are microstrip and stripline.

Microstrip is the most common type of transmission line. It simply consists of a copper trace of a known width a certain distance above an infinite ground plane. A dielectric (insulating) material is between the trace and the ground plane, and nothing is above the trace. Figure 3-6 shows a cross-section drawing of a microstrip.

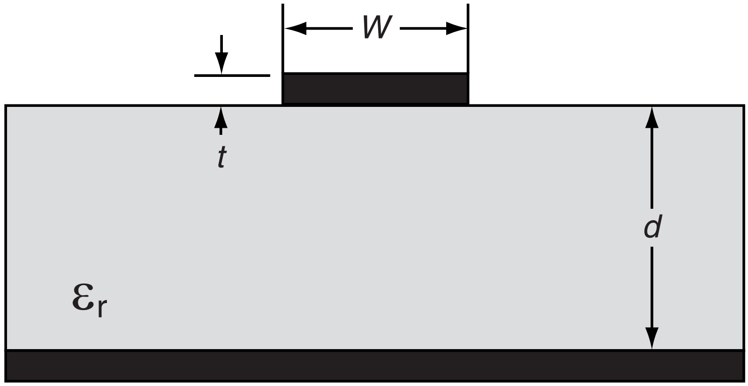

Figure 3-6: Cross-section of a microstrip

Note that any trace on a printed circuit board can be viewed as a microstrip. In fact, all circuit board traces are microstrips, but only some are designed that way. The complete analysis of a microstrip is a bit complex but a few approximations will help. The first approximation is that the ground plane is finite in size, since it cannot really be infinite, but it must be much larger in all directions than the transmission line. Another approximation concerns the free space above the trace. Most circuit boards are used in an enclosure of some type; if a microstrip is used on the circuit board then any metal cover must be kept far above the traces. This will keep the electric field lines above the microstrip from becoming distorted by the presence of a cover.

These approximations allow one to write a simple formula for the characteristic impedance of the microstrip. This equation is:

$$Z_0 = \frac{87 \ln\left[\frac{5.98d}{(0.8w + t)}\right]}{\sqrt{\varepsilon_r + 1.41}}$$

where w is the width of the microstrip trace (in mils, which are 0.001 of an inch), d is the distance to the ground plane (again, in mils), t is the thickness of the trace (in mils) and ε_r is the relative *dielectric constant* of the dielectric material used in the circuit board. The result is Z_0, which is the characteristic impedance of the transmission line in ohms. The term *characteristic impedance* is simply a way of describing the effects the transmission line will have on the signals it carries. For the purposes of nearly all circuit boards, your goal as a board designer is to produce transmission lines of a known characteristic impedance. This will allow the circuit board to "match" to the components on it, resulting in a minimum of transmission line reflections.

What is the dielectric constant (ε_r), you ask? It is a material property of the circuit board you have chosen to use, and it is usually outside of the board designer's control. Nearly all (99%) of circuit boards are built on "FR-4" woven fiberglass circuit boards, including the ones used in this manual. Exotic high frequency, high power, or low-loss circuit boards are sometimes built on more advanced (and much more expensive) materials. Table 3-2 lists material properties for common circuit board materials. Air is listed for reference only (it's very hard to build circuit boards in air!).

Table 3-2: Properties for Circuit Board Materials

Circuit Board Material	Dielectric Constant	Loss Tangent
FR-4	4.5	0.02
Ceramic (a.k.a. alumina)	9.8	0.0001
Teflon (a.k.a. duriod)	2.1	0.0003
Air	1	0

In addition to dielectric constant, the *loss tangent* is also given for each circuit board type. This is related to the loss an RF signal will experience when it passes through circuit boards made with each dielectric. A lower loss tangent means less loss. Notice how the least expensive material (FR-4) has the highest loss tangent, while the more exotic materials have lower loss tangents.

The process of computing the loss on a transmission line from the loss tangent is even more complex than the computations for characteristic impedance, as it involves the frequency of the signal. Table 3-3 lists a few representative losses of different types of microstrip traces. Notice how the microstrip width is the same for any signal frequency; only the RF loss rate changes with frequency.

Special RF Routing Techniques: Stripline

Stripline is another type of transmission line that can be easily built on a circuit board. It is identical to microstrip, but with ground planes both above and below the trace. Figure 3-7 shows a cross-sectional diagram of stripline. Stripline offers much improved isolation over microstrip, but at the cost of increased RF loss. Striplines are most often used for either high- or low-level RF signals requiring isolation from surrounding circuitry.

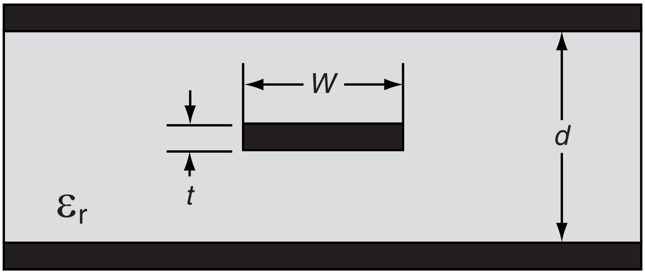

Figure 3-7: Cross-section of a stripline

As with microstrip, several simplifying approximations allow one to write a simple formula for the characteristic impedance of the stripline. This equation is:

$$Z_0 = \frac{60 \ln\left[\dfrac{1.9d}{(0.8w+t)}\right]}{\sqrt{\varepsilon_r}}$$

where w is the width of the stripline trace (in mils, which are 0.001 of an inch), t is the thickness of the trace (in mils), d is the total distance between ground planes (again, in mils), and ε_r is the *dielectric constant* of the dielectric material used in the circuit board.

Once again, this equation can be re-written to solve for the stripline width, given a desired characteristic impedance:

$$w = 1.25\left[(1.9d)e^{-\left(\frac{Z_0\sqrt{\varepsilon_r}}{60}\right)} - t\right]$$

Striplines behave identically to microstrip, but with the added benefit that the RF signal is surrounded top and bottom by ground. The ground planes provide a high degree of isolation, so external signals are less likely to interfere with the RF signal on the stripline. The reverse is also true; RF signals on the stripline will radiate much less energy due to the shielding effect of the ground planes. The downside to stripline is increased RF loss. This is due to the fact that the dielectric (insulating) material is now on both sides of the trace, and tends to absorb more of the RF. Table 3-5 lists stripline widths and decibel loss rates for a few stripline designs.

Table 3-5: RF Losses for Various Striplines

(1/16 inch thick each layer, d=1/8 inch)	50Ω Microstrip Width (mils)	Signal Frequency (MHz)	RF Loss Rate (dB/inch)
FR-4	53	1	0
FR-4	53	1000	0.11
FR-4	53	10000	1.06
Ceramic	21	10000	0.15
Teflon	103	10000	0.05

Table 3-6 lists stripline line widths for the most common applications. As with microstrips, you should discuss your stripline requirement with your fabrication house for the most accurate line width to use for a given characteristic impedance.

Table 3-6: Common Stripline Widths

Board Material	Thickness Between Groundplanes (inch)	50Ω Microstrip Width (mils)	75Ω Microstrip Width (mils)
FR-4	1/4	110	42
FR-4	1/8	53	20
FR-4	1/16	25	9

Striplines are most easily constructed on the inner layers of multi-layer printed circuit boards. If you are making a multi-layer board, then building a stripline layer is easy. First define two ground layers (one on either side of your stripline layer), and then define your stripline layer in the middle. Remember to set the default routing width on the stripline layer to the stripline's design width. You will have to use vias or through-hole device pins to connect to and from the stripline layer.

Note that the stripline layer can also be used to route other (non-stripline) signals. Just remember to keep all signals far from the striplines. In fact, it is best to place copper (ground) areas around your striplines, right on the stripline layer. This will enclose the RF trace in ground on all sides, and will provide the highest degree of isolation available. The resultant stripline trace is very similar to a coaxial cable, with the signal in the center, completely surrounded by ground. Remember when placing the ground areas to keep them at least five line widths away from the stripline. Use plated-through vias to "tie" the ground areas to the ground planes above and below the stripline layer. Figure 3-8 shows an example stripline layer with ground isolation around each stripline. Notice in the figure how the RF striplines are completely protected by ground areas which are kept far from the stripline trace. Also note that there are other ground areas which are around non-RF traces. These ground areas are allowed to come much closer to the traces, because they are not controlled impedance transmission lines. The circles represent vias, in which signals are crossing through this layer from other layers.

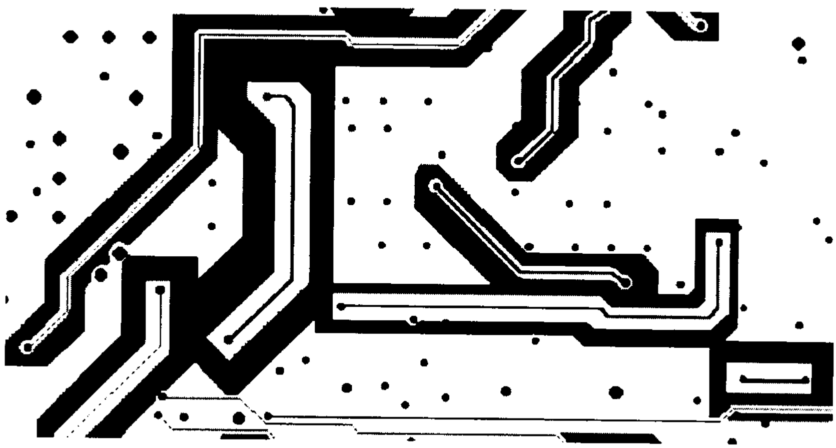

Figure 3-8: Stripline layer with ground isolation areas

Just because you are building your own two-layer circuit boards does not mean that you cannot use striplines. Simply etch the stripline onto one side of a board, with a ground plane on the other side. Then use another single-sided board (of the same thickness) as the ground plane for the other side of the stripline. Place the two boards together, with the stripline traces touching the empty (non-copper) side of the single-sided board. Drill many holes through the boards in the ground areas (don't drill through your striplines!), and then use press-in vias or small wires to solder the two ground planes together. Finally, wrap copper tape around the outer edges of the circuit board and solder the tape to both ground planes. Now you've made your own stripline circuit board!