

# 73 Amateur Radio Today

JUNE 1993  
ISSUE #393  
USA \$2.95  
CAN \$3.95

A WGI Publication  
International Edition

**PC BOARD  
LAYOUT TIPS**

**Portable  
Packet  
Digipeater**

**Easy Coax  
Connectors**

**73 Reviews**

**MFJ Antenna  
Analyzer**

**US Digital's  
Simplex  
Repeater**



# 16 Tips For Using PCB Layout Software

*You can do things differently when you design a PCB on your computer.*

by Jim Tarchinski N8PTG

Fewer and fewer printed circuit boards (PCBs) are laid out by hand these days. The reason is simple: More and more hams are using computers to create the artwork for their boards.

There are many reasons for this trend away from hand-taped artwork, but there are two main ones. First, prices for PCB layout packages are only a fraction of what they were just a few years ago. In fact, there are even a few shareware programs floating around that allow you to lay out a board on your computer (see the sidebar). Prices are definitely in a range that is affordable to most casual PCB creators.

The second reason layout programs are becoming so popular is that they can shorten the time it takes to design a board from scratch. This is especially true if the software can automatically route the traces, or check your traces against a network connection list (netlist) generated by a schematic drawing program.

Like any other change, however, going from hand-taped to computer-created artwork requires you to change your thinking somewhat. There are some things that can be done, or even should be done, differently. I've learned a few tricks in the year or so that I've been using a layout program, so I'd like to share some of them with you. This article offers 16 tips to help you make

a smooth transition to creating computer-based PCB layouts.

## Tip #1

Before you even begin to lay out a board, give some thought to how it will eventually be mounted. If you bought an enclosure at the last swap meet that will work perfectly, you'll need to consider this up-front. This could influence your selection of components or your decision on the layout.

## Tip #2

Except for the tip I just mentioned, don't decide on a board size until you make an initial layout and get a rough idea of how much board area is required. Once you have a good handle on this, be sure to select either a standard PCB size or the size of a remnant you have on hand. After all, why cut a larger board down to size if you don't have to?

## Tip #3

Don't choose a double-sided board unless you're forced into it. Start by running traces on the solder side only. If you run into too many intersecting traces you can always switch to a two-sided board. It's often surprising how many traces will fit on only one side, especially if your layout program has an excellent "auto-route" algorithm. It's

also worth remembering that jumper wires can frequently get you out of a bind on a single-sided board (and even on some double-sided ones!).

## Tip #4

When possible, place mounting holes at half-inch increments from one another. Then, when you go to drill the mounting holes in the project case, you'll find that it's much easier to mark off half-inch distances than an odd number of 0.1-inch increments, the step size that most programs use.

## Tip #5

For dual-sided boards, be sure to write "Solder Side" and "Component Side" right on the layout. This will help you later on when you have to determine the orientation of the clear artwork against the board. See Figure 1.

## Tip #6

It's also a good idea to write the date and a version number right on the artwork—preferably on both sides of the board. This helps you keep track of multiple versions of the same project. I got hit by this once, and it took me nearly an hour to figure out that I had accidentally used an old version of my artwork on the bottom side of the board! See Figure 2.

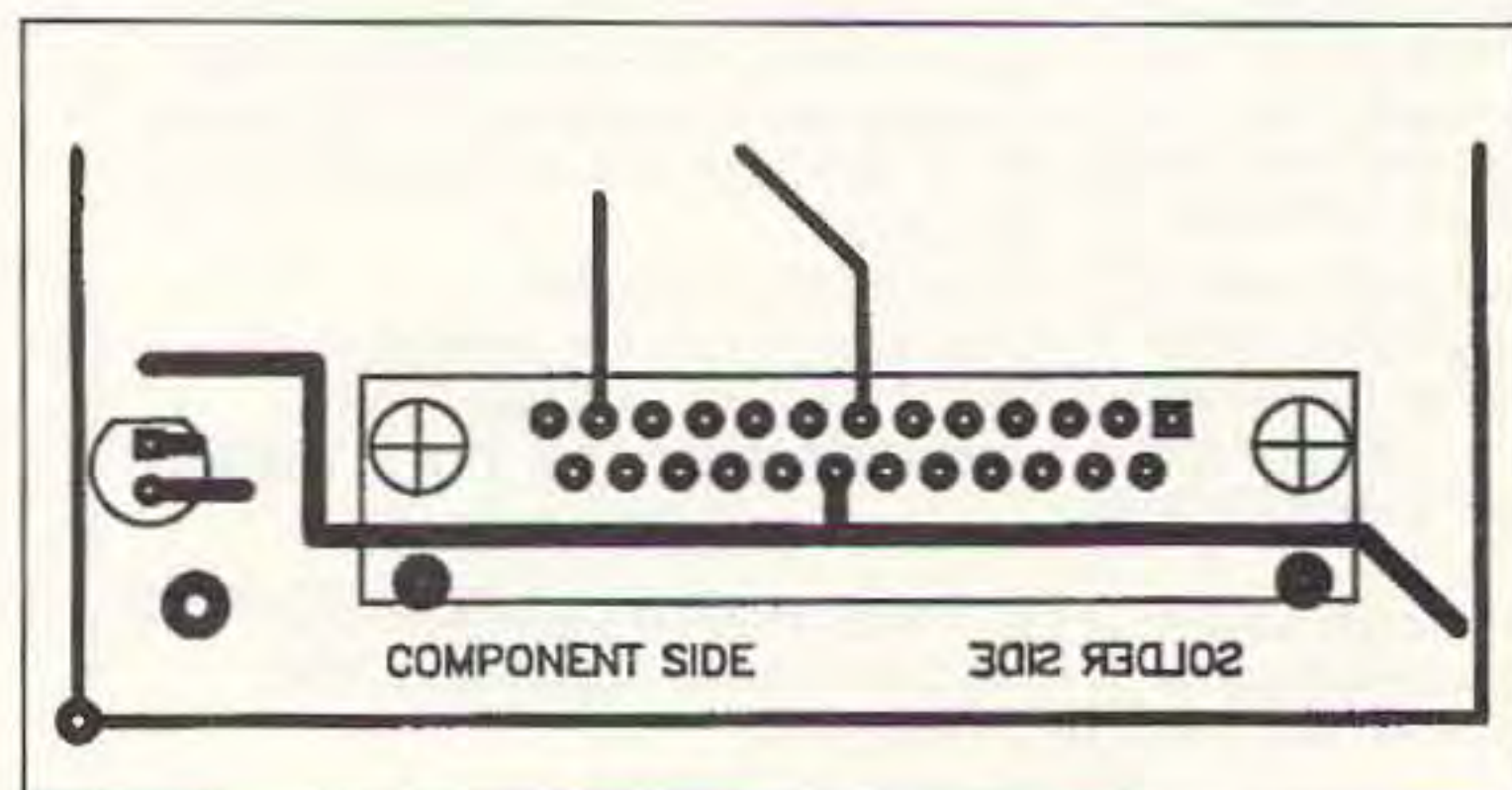


Figure 1. Tip #5.

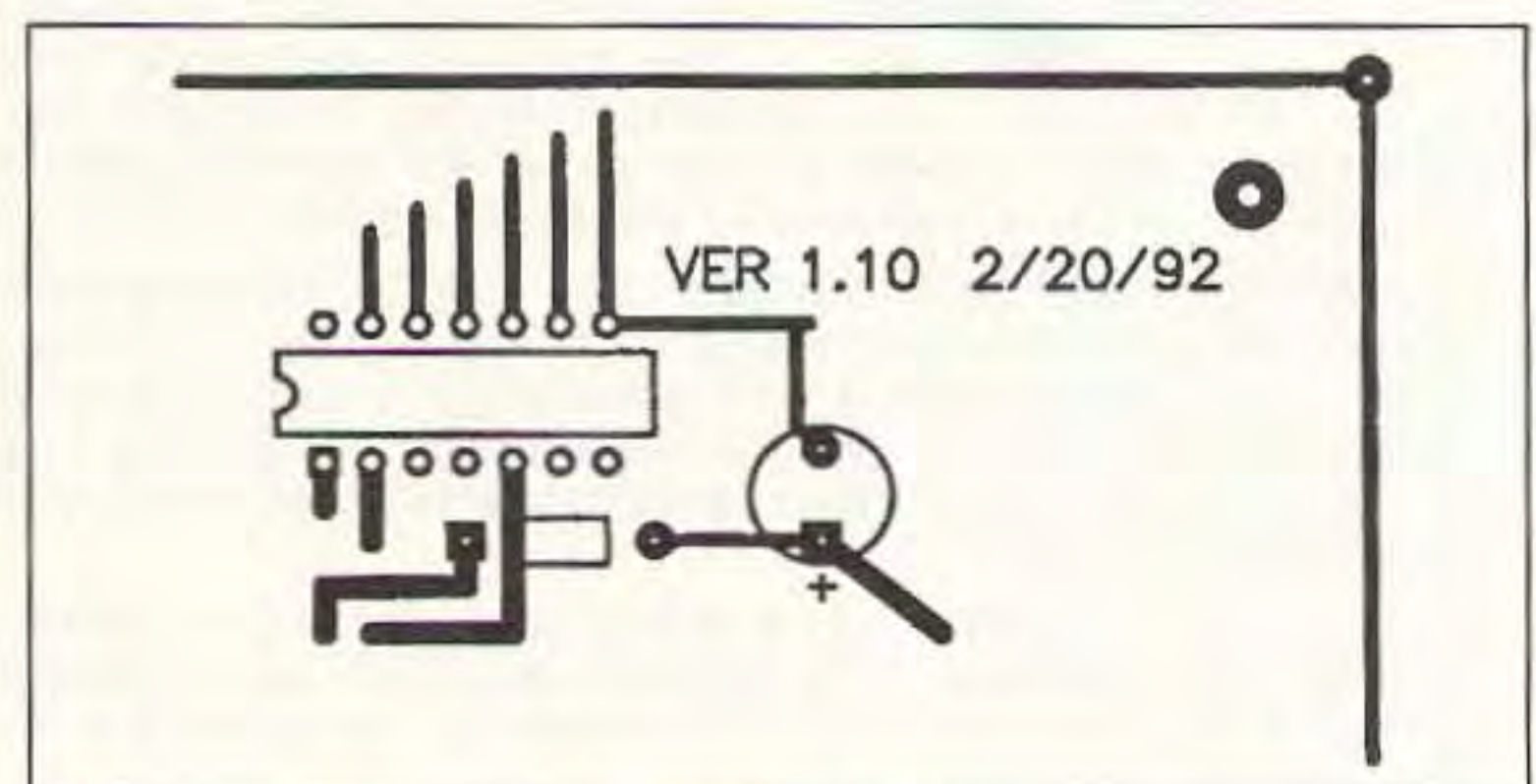


Figure 2. Tip #6.

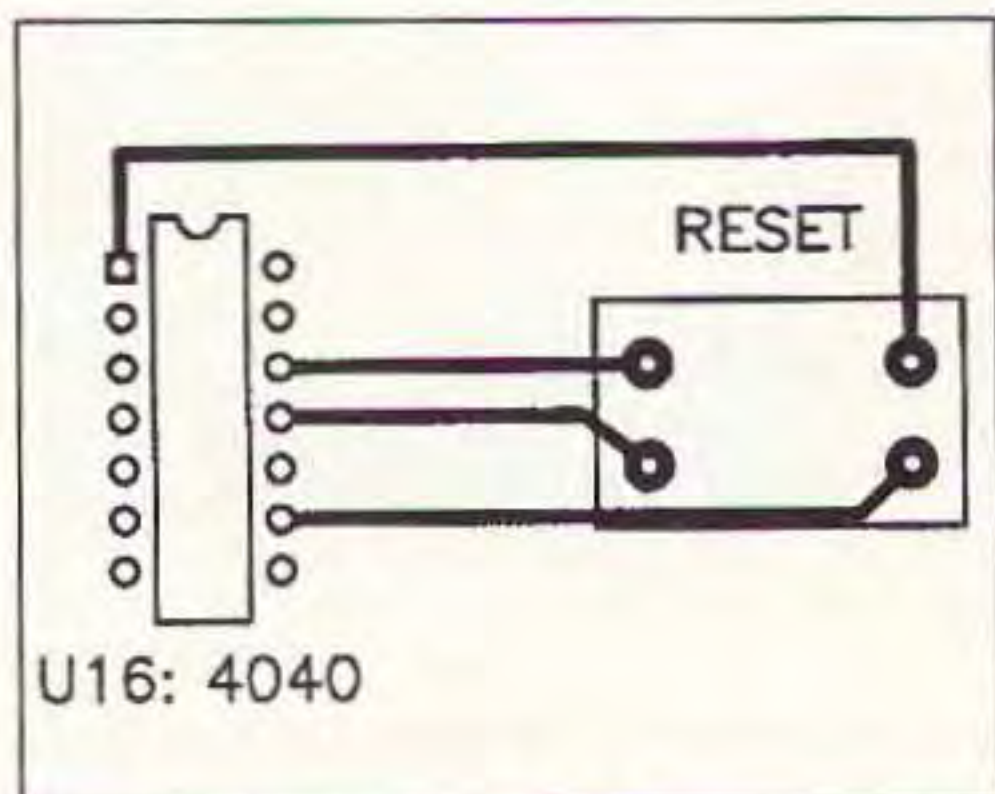


Figure 3. Tip #7.

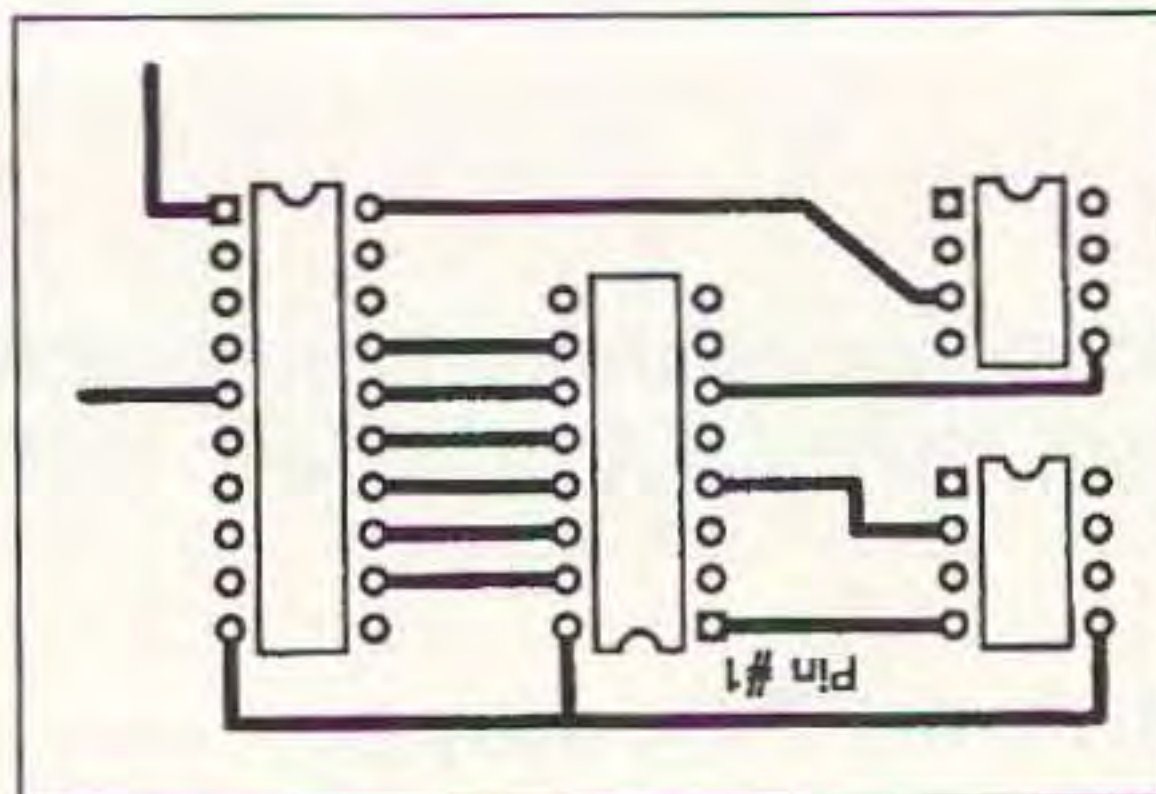


Figure 4. Tip #8.

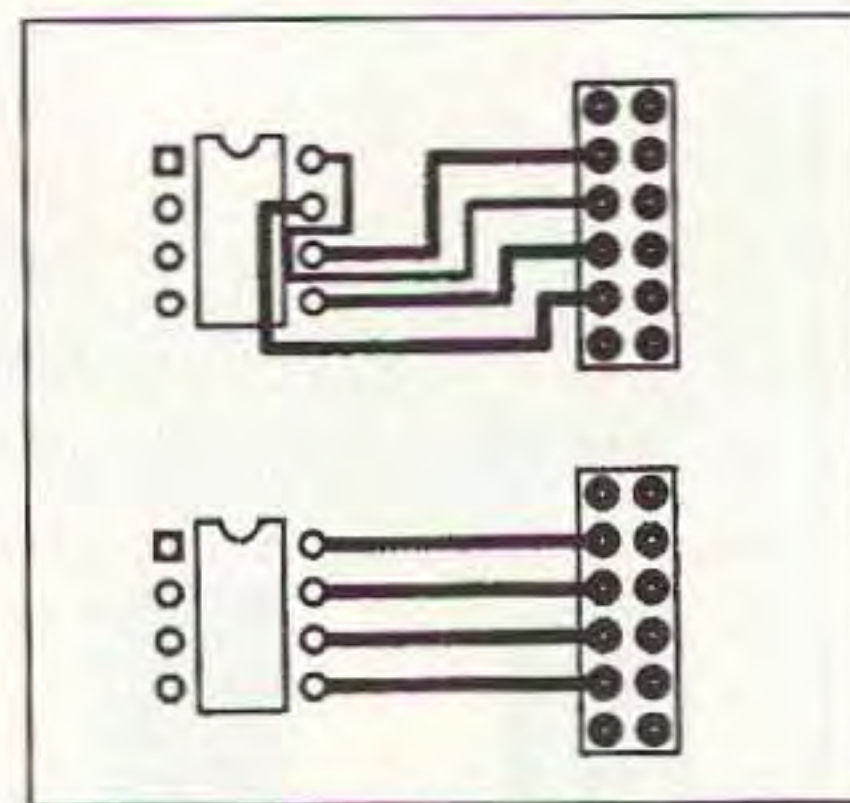


Figure 5. Tip #9.

### Tip #7

Most hams don't place a silk-screen layer on their boards. In its place, consider putting important information right on either the top or bottom copper layers. If you have the room, component names and/or values are often convenient to have right on the board. See Figure 3.

### Tip #8

Try to mount all IC packages in the same direction, unless this violates one of the other rules mentioned above. If you are forced to mount one IC "upside-down," be sure to clearly mark pin #1 on both sides of the board (Tip #7). See Figure 4.

### Tip #9

Change the schematic whenever it helps! It's amazing how many people seem to think a schematic is set in stone when they get to the layout phase of a project. If you can redefine connector pin numbers, or use one IC gate in place of another, it frequently simplifies the board layout. Just be sure you go back and update to print so you'll remember that you made the change later on.

Although I don't recommend it, some people tend to take this tip to an extreme—they actually lay out the board *before* they draw the schematic. See Figure 5.

### Tip #10

Now here's one that is really simple, but again, it's surprising how many people never seem to think of it. Remember that all pads do not have to be the same size, not even all the pads for a single component. If you need extra room (to lay a high current

trace, for example) make some of the pads smaller. This is especially true for unused pads, which you may even be able to eliminate completely. See Figure 6.

### Tip #11

Several times during the design, print out an exact-size copy of the layout, being sure to include the silk-screen layers even if you won't be using them. Place the actual components on this printout to make certain there's adequate space for everything. Pay particular attention to large capacitors, mounting holes for connectors, and any components whose templates you created yourself.

Also, be sure connector pin numbers line up correctly.

If you designed for a male DB25 connector, but the layout software put down a female DB25, your pin definitions are not going to be what you expect.

### Tip #12

Place extra pads on unused connector pins or IC gates. You may not have a need for them later, but if your initial design doesn't work, they will make it much easier to modify the board further down the line. You may even want to include pads or component space for any "upgrades" that you might want to wire in later. See Figure 7.

### Tip #13

If you don't use through-plated holes (as is the case with most hams), make certain that all the traces you run on the top layer can be soldered to the components on the top layer. For example, straight PCB-mounted connectors are frequently encased

## Computer Design Aids

### Commercial Packages

Tango  
Accel Technologies, Inc.  
6825 Flanders Drive  
San Diego CA 92121-2986  
(800) 488-0680

DC/CAD  
Design Computation  
Rt. 33 Sherman Square  
Farmingdale NJ 07727  
(903) 938-6661

OrCAD  
OrCAD Systems Corporation  
3175 N.W. Alcock Drive  
Hillsboro OR 97124-7135  
(503) 690-9881

Shareware  
(Most programs are available on electronics-oriented BBS systems across the country.)

PC Trace  
PCB  
PCB\_SRCS  
PCROUTE

in plastic in such a way that you can only solder them to the bottom side of the board—the top side of the pins are completely blocked off.

If this causes you a problem, there are two things you might try: Either mount them to a component a few millimeters above the board so you can get an iron in there, or use a via (through-hole) near the

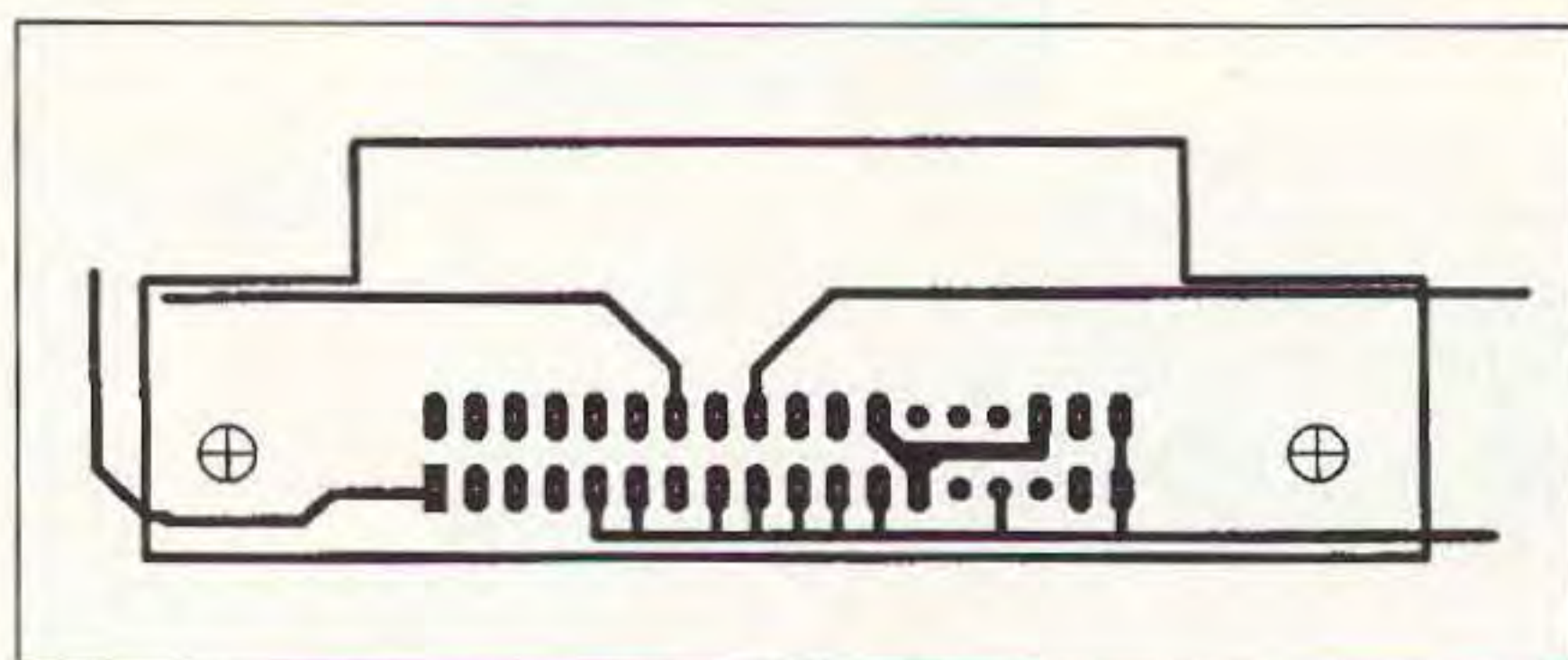


Figure 6. Tip #10.

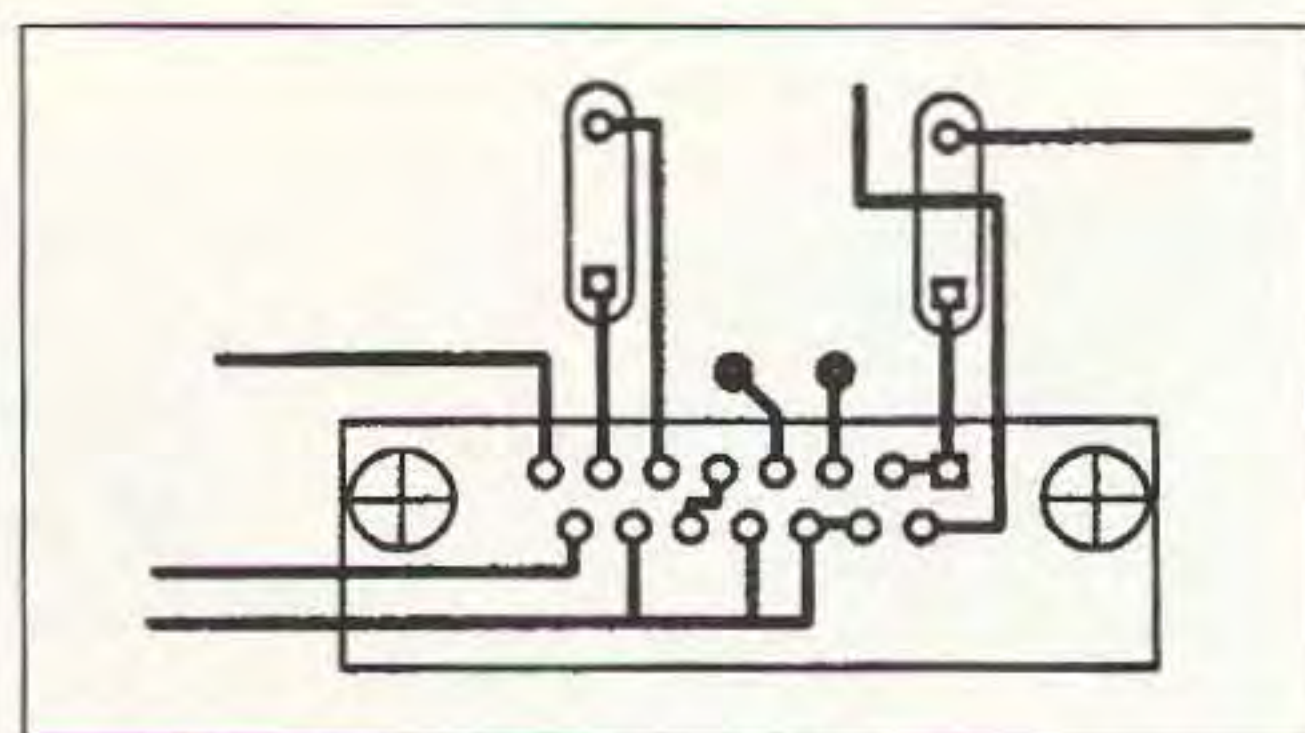
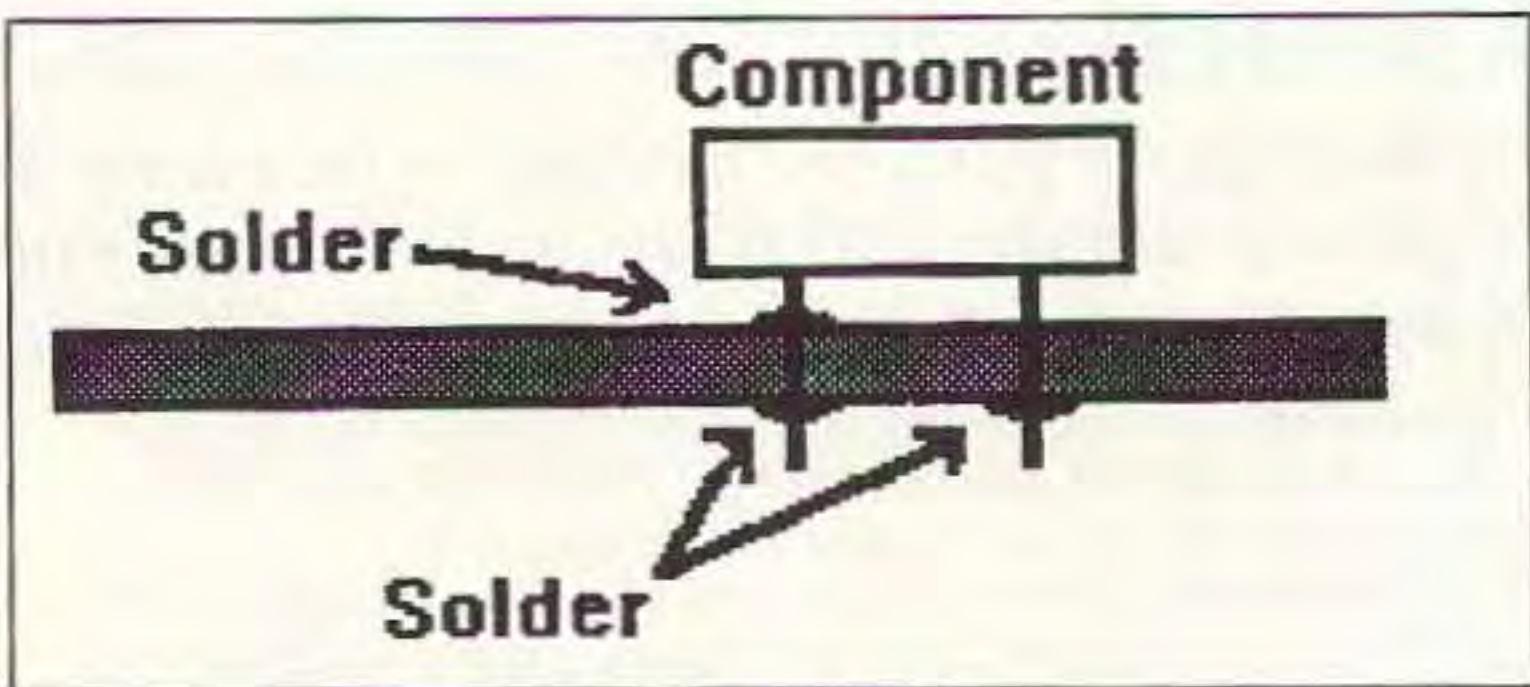


Figure 7. Tip #12.



*Figure 8. Tip #13.*

problem pad. See Figure 8.

#### **Tip #14**

Once again, for those who don't use through-plated holes, keep vias to a minimum. You'll have to solder through every via that you design in, which is a pain in the neck. This is another handicap you may find with most auto routing routines, which tend to use vias like they're going out of style.

#### **Tip #15**

Consider creating your own personalized checklist of things you want to examine before you're finished with a layout. Before I came up with my list I

frequently forgot to size my mounting holes correctly, or to run an automatic check to verify all my trace routings.

#### **Tip #16**

Don't throw away all the "good old" design rules that you used when you laid out PCBs by hand. For example, you still should keep the analog and digital sections separated, be sure to use grounding planes where necessary, and don't make the traces too small for the expected maximum current load. Most software just isn't smart enough yet to check for stuff like this (but the way things are moving, it's only a matter of time!).