PCB ROUTERS AND ROUTING METHODS

BY: LEE W. RITCHEY, SPEEDING EDGE, COPYRIGHT SPEEDING EDGE DECEMBER 1999

FOR PUBLICATION IN FEBRUARY ISSUE OF PC DESIGN MAGAZINE

INTRODUCTION

Routing of printed circuit boards has ranged from hand layout using tape on Mylar to 100% autorouting of all wires. The methods chosen have been driven by market place pressures and product complexity as well as the skill of the PCB designer. The need to complete routing rapidly with large numbers of nets and electrical constraints, as exist in super computer like products, has driven the development and use of specialized autorouters. At the other end of the complexity scale, PCB layout tools have been optimized to allow hand routing of virtually all wires. Some call this latter method “electronic taping.”

This diverse set of design problems has given rise to two distinct routing strategies and router types. These are maze routers and X-Y based routers. In both cases, the router may be shape based or gridded. When to employ each type of routing method has been the source of considerable confusion. When one router type is applied to the other routing problem, the results can be disappointing. The experience can cause the user to decide autorouters are no good and continue hand routing long after it becomes an uneconomical choice. As an example, using an X-Y router on a two routing layer design yields poor results. Similarly, using maze routing on a design with high pin count BGAs and many routing layers, results in many unrouted wires due to early blocking of routing space by wrong way routing. Understanding the router types and when to use each one is important to optimizing the PCB layout process.

This article will explore the two types of routers. Since most designers started their careers with maze routers and are familiar with how they operate, more time will be spent on the operation and advantages of X-Y routers. This should help the reader understand their benefits and where they fit in the ever more complex world of PCB layout.

TWO ROUTER TYPES

The two basic routing choices are maze routing and X-Y routing. Figure 1 uses a simplified “rats nest” to illustrate the basic difference between the two types of routing strategies. It can be seen from this diagram that X-Y routing involves at least two routing layers with wires travelling in only one direction on each layer. Maze routing allow the wiring of complete nets on a single layer, eliminating the need for layer changing vias. From the drawings in Figure 1 the reader can begin to see some advantages and disadvantages of each choice. Understanding the advantages and disadvantages is key to successful, on time completion of a design.

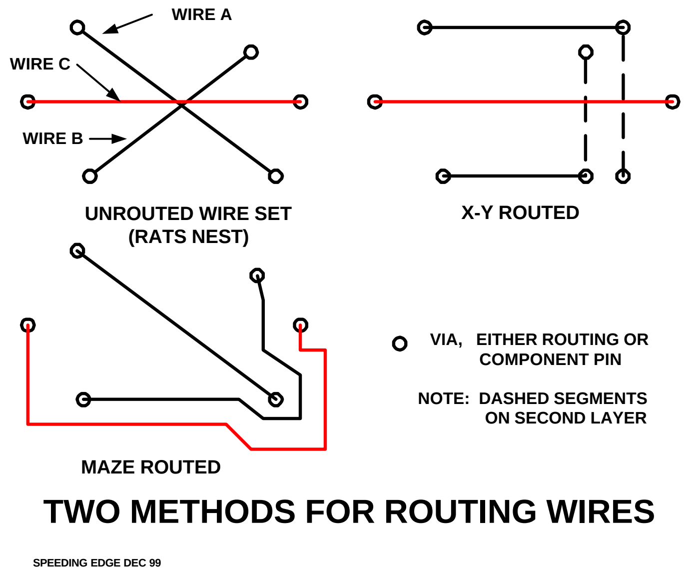


FIGURE 1: ROUTING METHODS

WHEN TO USE EACH ROUTER TYPE

As might be expected, the two routing methods in Figure 1 (maze and X-Y) are best used in certain situations.

Maze routing, the most commonly used method, works best when there are only two routing layers. These layers are nearly always the outer layers of a PCB, which contains the component mounting pads. These mounting pads interrupt the routing surface in such a way that making long, straight runs from one pin to another is difficult, if not impossible. This maze routing works especially well if the pin out of busses is done in such a way that all of the wires in the bus can be “sweep” routed side by side. This is common in the PC world, where the designers of the chip sets insure that the pin outs are optimized to allow this. It works poorly when ICs with large busses are pinned out such that the bus must be inverted from one end to another or when the bus must connect to more than one IC. It also works poorly when a design contains high pin count BGAs or PGAs.

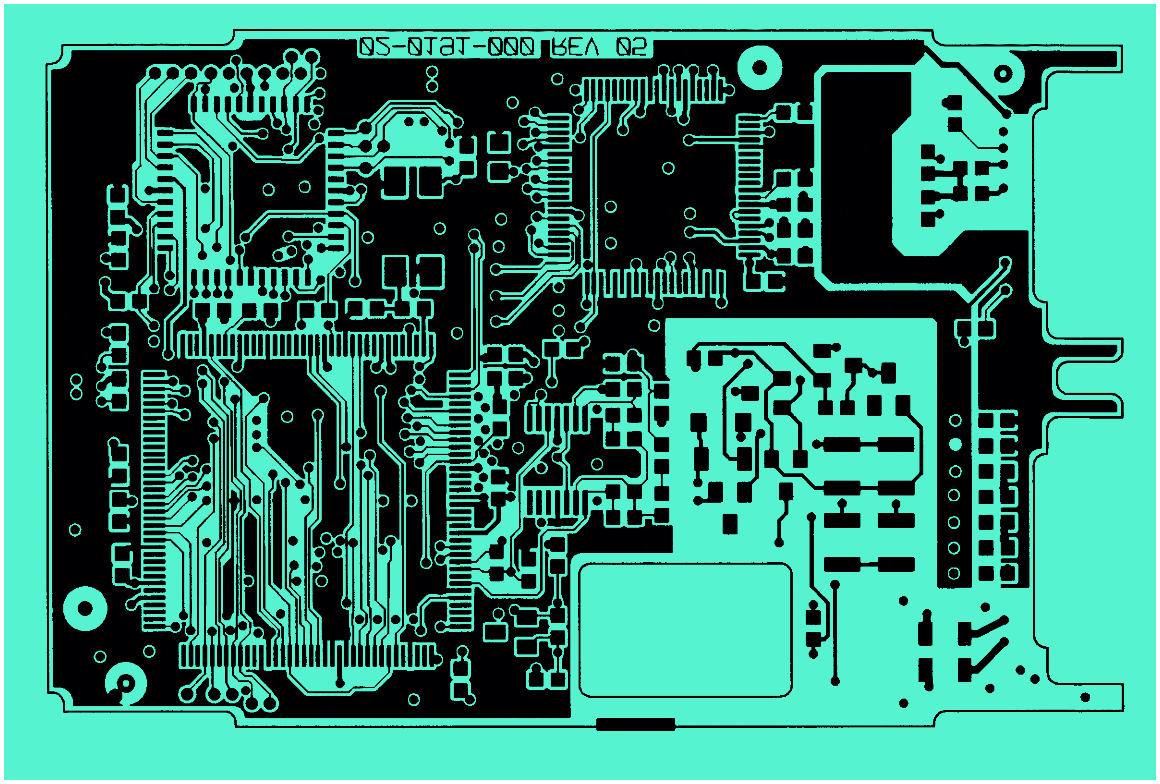


FIGURE 2, AN EXAMPLE OF MAZE ROUTING

Figure 2 shows an outer layer of a PCB that has employed maze routing. Routing is done at a wide variety of angles, including both X and Y in the same layer. This “wrong way” routing is beneficial as long as the wrong way portion of a route does not block the path of another wire as happened in Figure 1. This wrong way routing forces wires routed later to take round about paths. These round about paths are often much longer than they would be if routed more directly.

X-Y routing, the method of choice for high complexity, high performance designs with many high pin count parts, works best when a design requires more than two routing layers to contain all of the wires. As can be seen from Figure 1 routing a wire using the X-Y technique does not block the routing surface. As a result, later wires can be easily routed through each layer. This type of routing lends itself to automatic routing. It also lends itself to implementing large numbers of constraints on nets, such as length matching, layer to layer coupling control, and add length to do timing related tuning.

SOME DEFINITIONS

In order to understand the language of routing some definitions are in order.

Manhattan Length, Figure 3 illustrates this basic concept in routing**.**

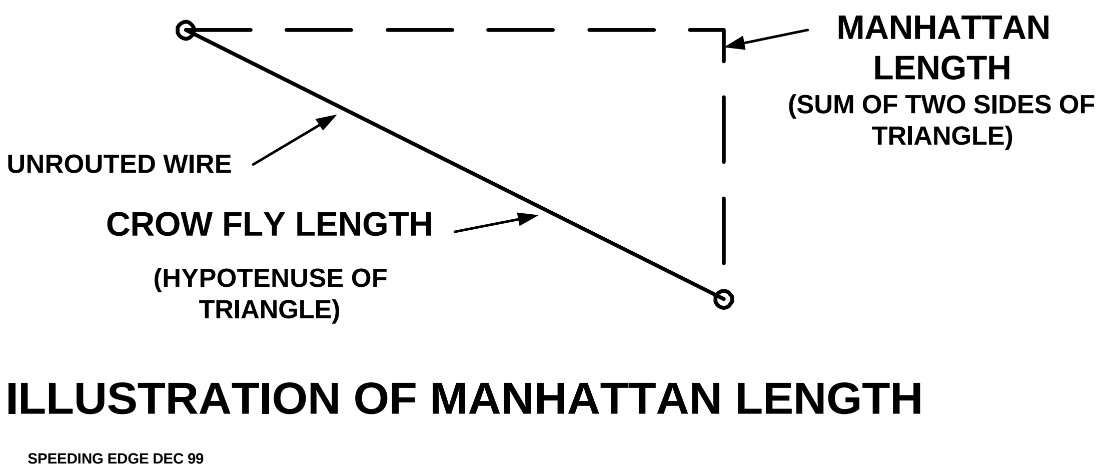


FIGURE 3, MANHATTAN LENGTH

Manhattan Length is the shortest path that a wire can have when it must be connected using only segments that are confined to either the X axis or Y axis. Calculating the Manhattan Length is quite simple. One need only subtract the X coordinates of the two end points from each other and the Y coordinates of the two end points from each other and sum these two dimensions.

Knowing this, it is easy to see how preroute analytical tools can estimate the length of nets, and, therefore, their time delay, prior to routing. With a well run autorouter, it is possible to have post route lengths that agree with preroute predictions to small fractions of a nanosecond. This is one of the more valuable features of X-Y routing.

Clearly, the “Crow Fly Length” is shorter than the Manhattan Length. One might be tempted to use the “Crow Fly Length” to keep the flight time between two points to a minimum. This works for the first few wires, but fails for subsequent wires for the reason shown in Figure 1; later wires are forced to be longer, just to make a connection.

Detour Routing- Detour routing is any routing of a net or wire that exceeds the Manhattan Length. In Figure 1, both wire 2 and wire 3 have exceeded Manhattan Lengths. In this case, this was forced on those wires by the wrong way routing of wire 1. If timing were dependent on maintaining the Manhattan length predicted at preroute analysis, this design would fail its timing specifications. When designs are high speed and timing budgets are worked out at the preroute stage, which is common in very high speed, high performance designs, allowing this detour routing may be fatal.

Net- A collection of wires that connects all of the points or pins in a single circuit.

Wire- The connection between any two adjacent pins in a net.

Segment- A portion of a wire when routed. In figure 1, wire C has only one segment while wires A and B have two segments. It is possible for a wire to be made up of several segments if a number of vias are needed to find open space in the signal layers. However, it is uncommon to see a wire with more than three segments.

Straight Wire- a wire that is pure horizontal or pure vertical. Wire C in figure 1 is a straight wire. These wires usually need to be routed first, because the number of possible solutions without detour routing is limited.

Rats Nest- A "crow flies" plot of all of the potential connections between the pins of all of the parts in a printed circuit board. It illustrates the demand for wire space that a particular component placement puts on the wiring surfaces of a proposed PCB stackup. This is a valuable plot, because it allows a designer to assess the distribution of wiring in a design. Based on these plots, placements can be adjusted to even out the wire demand and routing strategies can be devised to insure all wires fit into the minimum number of layers.

Routing Via or Turning Via- a via used to change layers or change directions when routing a wire.

METHODS FOR ADDING LENGTH OR ACHIEVING A KNOWN TIME DELAY

Often, it is necessary to add length into a net or a wire to achieve a predetermined time delay. There are two ways to do this. They are serpentine routing and "trombone" routing. These two methods are illustrated in Figure 4.

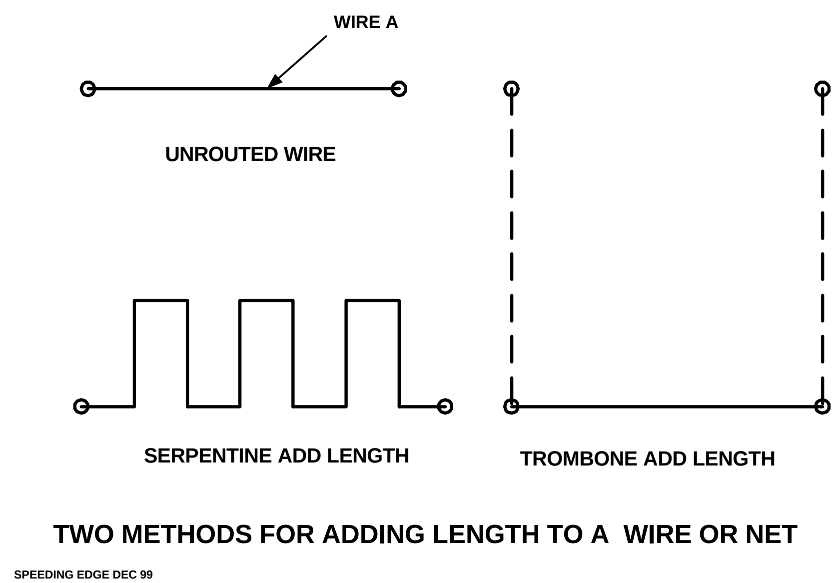


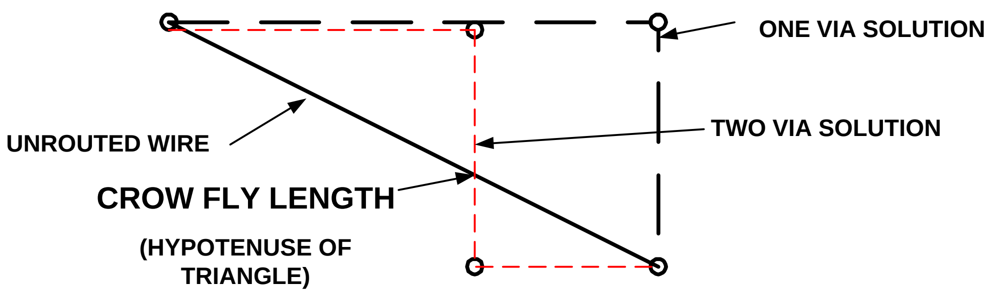
FIGURE 4, LENGTH TUNING METHODS

The serpentine method of tuning shown on the lower left is quite popular. It has its origin as an autorouting technique in the early 1980s at a company named Shared Resources. It is a handy way to get extra length into a trace or wire. However, it was abandoned early on because of the effect it had on the routing surfaces. It is easy to see that the serpentine structure blocks the routing surface in both the X and Y directions. As a result, the routing surface is cluttered for all later wires that might need to pass through this part of the board in either the X-axis or the Y-axis. What is not obvious is the fact that this structure blocks all potential via sites in its area. This means that the supply of layer changing vias is diminished in the parts of the PCB where this kind of tuning is used. This turns out to be a severe handicap in high layer count designs.

Trombone length tuning, illustrated in the lower right of Figure 4, was devised as a solution to the problems represented by serpentine tuning. It can be seen that the added length has been achieved by adding segments or by adding length to segments that were already used to route the wire. This kind of tuning does not block any of the routing surfaces. Further, the number of possible tuning solutions can be quite large by using segments in a variety of directions and in any of several routing layers. Trombone tuning lends itself well to automatic length matching of multiples of wires. All that is needed is to insure that there are enough available via sites to complete the routing. In order to insure space is available for the added segments and length, length tuning is done early in the routing process.

X-Y ROUTING AFFORDS MANY ROUTING SOLUTIONS FOR MOST WIRES

Figure 5 illustrates how X-Y routing can be used to find a routing solution for a wire in a crowded PCB. Consider the one via solution shown. There are two possible solutions in a two routing layer design, one of which is shown. The second solution is the other side of the rectangle formed by the unrouted wire end points. When two more routing layers are added, the number of possible combinations jumps up to eight. When six routing layers are used, the number of possible solutions goes even higher, and this is using only one routing via.



ONE AND TWO VIA ROUTING SOLUTIONS

SPEEDING EDGE DEC 99

Consider the case where two routing vias are used as is shown with the lightly dashed or red solution. By moving the via pair left or right, it is possible to find many solutions. If the two vias are located on a horizontal axis even more solutions are possible. Add into the picture that the three segments can exist on three different routing layers and is can easily be seen that there are an enormous number of possible solutions to this one wire. All that is required is that there be sufficient numbers of available via sites, that wrong way routing doesn't block paths and that the router be capable of performing this kind of searching.

Consider the case where the two ends of the unrouted wire are close to the same X coordinate or Y coordinate (the wire is mostly horizontal or vertical). The number of possible one and two vias solutions grows smaller until the case where an unrouted wire is all horizontal or all vertical, a "straight wire". In this case, there are no one or two via solutions without exceeding the Manhattan distance, detour routing). Clearly, straight wires must be given first priority when routing a design in order to insure maximum probability of success without detour routing.

CONDITIONS FOR SUCCESSFUL X-Y ROUTING

A number of elements are needed in order to insure successful X-Y routing. Among these are: more than two routing layers, surfaces that contain a sufficient quantity of routing via sites, a routing strategy that defines the order in which the router is to proceed through the wire or net list and a router that is capable of performing the searches involved in locating zero, one and two via solutions in a multilayer routing structure.

From the discussions above, it can easily be seen that two routing layers do not contain enough possible solutions to solve routing problems, mainly due to the fact that the surfaces are interrupted by component mounting pads.

Availability of routing via sites is a complicated problem to work. It takes quite a bit of experience to correctly estimate this number. From experience routing hundreds of PCBs with high pin count ICs at Shared Resources, we determined that there needed to be at least one available routing via site for each component pin in the design. These via sites need to be distributed in areas where it is likely that wires will need to make turns or change layers. As design contain more and more surface mount parts and parts on both sides of a PCB, the number of available via sites is reduced. This gives rise to a need for buried and blind vias. Buried vias require the use of sequential lamination, a very costly solution. Blind vias give rise to the need for additional process steps to build the PCB. Usually, it is more economical to start with a larger PCB in order to allow room for vias than to resort to either of these tactics.

There is a tendency to think of autorouters as programs that look at a design and determine how to proceed to a successful solution. Just mash the button and out comes a routed PCB. I like to think of autorouters much the way that Gene Amdahl once described our first big computer to a reporter who though that powerful computers of the kind we were making would one day eliminate human thinking. To this Gene answered “Maam, what you don’t understand is what we have made here is an extremely fast idiot! Humans have to decide what needs doing and tell it what to do.” Autorouters are like that. We need to decide the routing strategy and tell the router how to proceed. Failure to do so results in bad results. Like all computing, garbage in garbage out.

SUMMARY

PCB designers have at their disposal a number of well developed routing tools. Each of these tools was developed to solve a particular set of design problems. Understanding the types of tools and what they are good at is fundamental to a good outcome. What is well established from many years of successful designs is that autorouters when properly matched to the problem and when properly operated yield accurate results faster that ca never be achieved with hand routing. The key is knowing which to use, how and when.

References:

McCloud, Jonah, Speedy Gridless Router Designs Denser PCBs, **Electronics**, July 1987.

Ritchey. Lee W, A comparison of Routing Methods, **Printed Circuit Design**, November 1987.

Ritchey, Lee W. Complex PCB Design Methodology, Part 1, **Printed Circuit Design**, October 1990.

Ritchey, Lee W, Complex PCB Design Methodology, Part 2, **Printed Circuit Design**, November 1990.

Murphy, Daniel W, Characteristics of Four Large Multilayer PCBs, **Circuit Design**, May 1990.