

## GENERATOR ANALOG AND DIGITAL SIGNALS

A.S. Popov

Scientific Supervisor: PhD, Associate professor A.V. Obkhodtsky

Language Advisor: Senior teacher N.V. Demyanenko

Tomsk Polytechnic University, Russia, Tomsk, Lenin str., 30, 634050

E-mail: [asptomsktpu@gmail.com](mailto:asptomsktpu@gmail.com)

Annotation: in this article, we will focus on the creation of printed circuit boards for electronic devices. In fact, a whole science with a lot of nuances, calculations, etc. In this article, we will only consider the tracing process for example emulator digital and analog signals.

The process of creating roads of the conductor on board called tracing or wiring. These roads allow interaction between the various elements of a PCB. Connection tracking is usually the final stage of the design and the design of electronic equipment is to determine the lines connecting the equipotential contact elements and components that make up the projected unit.

Tracing task is one of the most challenging in the general problem of design automation CEA. This is due to several factors, in particular to the variety of ways of implementing structural and technological connections. From a mathematical point of view, tracing - the most complex task of choosing from a huge number of variants of optimal solutions.

The main task of routing is formulated as follows: according to the given scheme to lay the required conductors on a plane (circuit board, chip etc.) to implement the predetermined engineering compounds taking into account the predetermined constraints. The main limitations are the width of the conductors and the minimum distance there between.

There are several tracing methods. For simple devices manual modes are mostly used, for more complex - automatic or interactive ones. Now let's consider an automatic and manual ways as an example.

Figure 1 showsthe device - anemulatorofdigitalandanalogsignals. This is a screenshot of the program P-CAD- 2006 PCB. There has not been applied trace, only elements of the device are arranged and organized communication between their findings. The task is to trace, to start an automatic method.

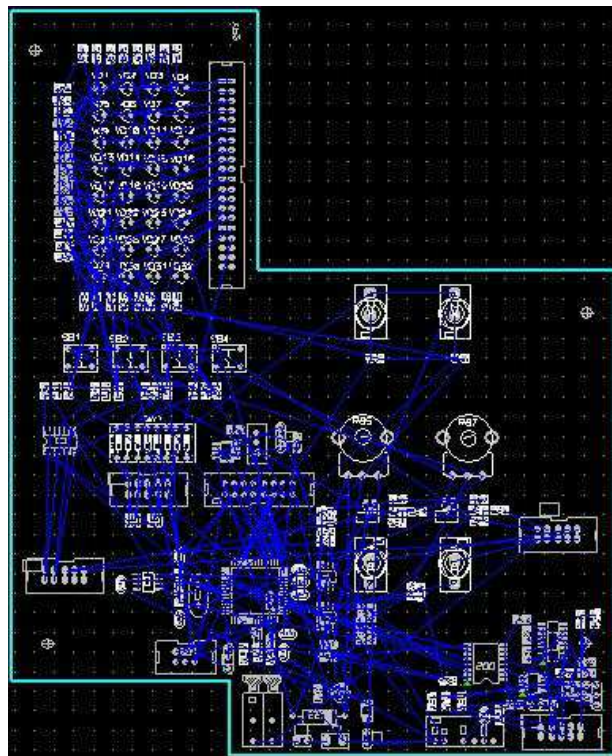


Figure 1 - emulator signals without trace

Figure 2 shows the result of automatic routing. It was performed using a standard automatic tracer embedded in P-CAD 2002. As a result, you can see many flaws. The most important thing - there should be no right angles to the road (Figure 3). When the conductor circuit board rotates through an angle of  $90^\circ$  can be reflected signal. This happens mainly because of the change of the width of current flow path. At the apex of the angle, width of the track is increased 1.414 times, which leads to a mismatch of the characteristics of the transmission line distributed capacitance and especially the self-inductance of the track.

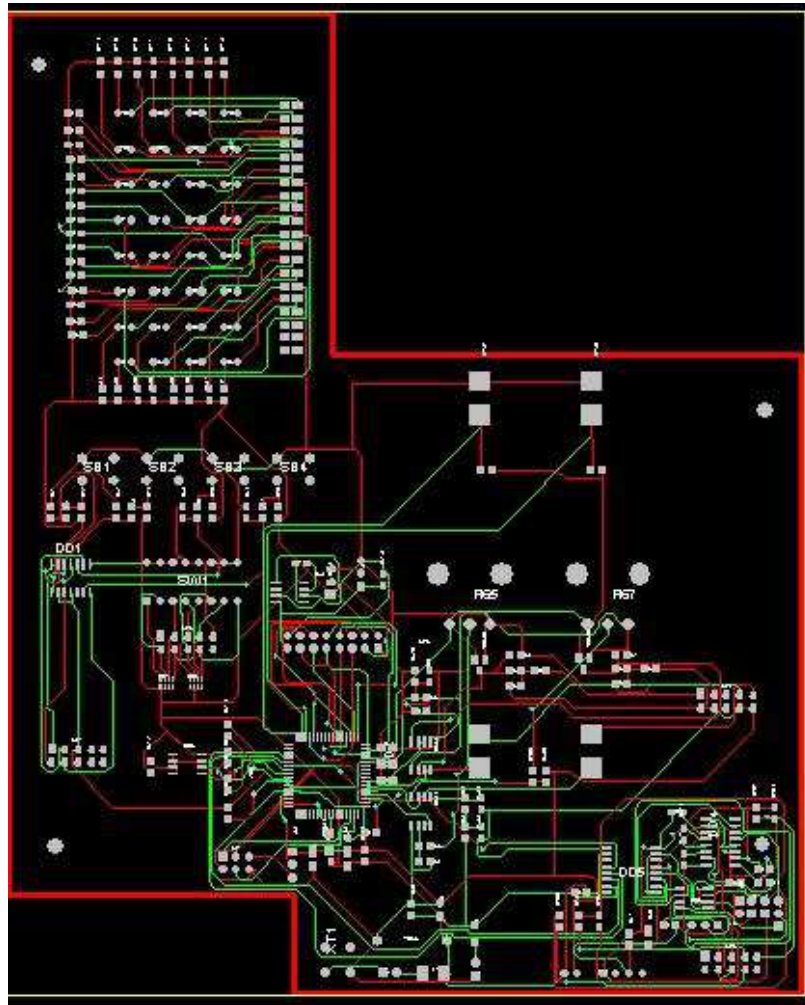


Figure 2 - automatic routing

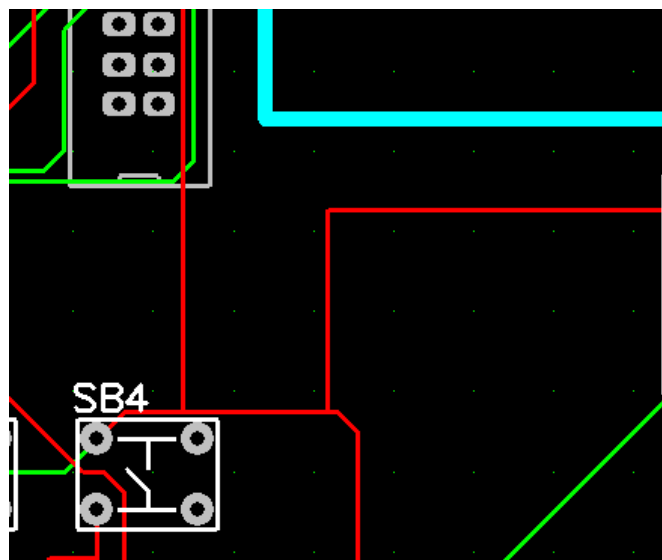


Figure 3 - shortcomingstrace

Moreover, extended paths to the terminals on the power are required. If this is avoided, then due to the action of a common bus of large current a track can be burned and the device becomes unusable. As a rule, a common bus is usually placed to the power. Figure 4 shows a variant of the trace, here - it's a large training ground, occupying the largest area.

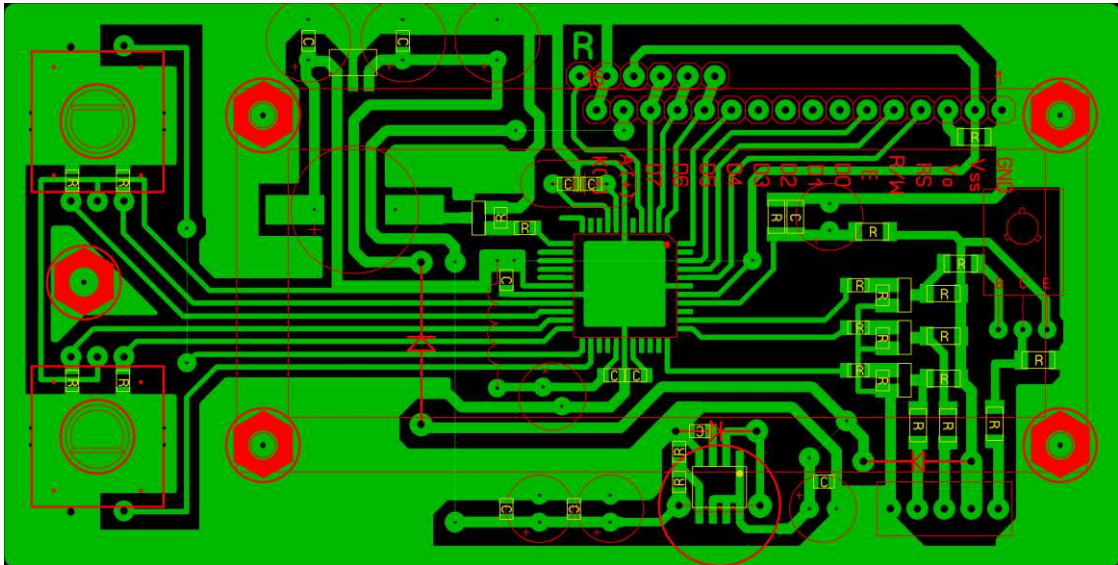


Figure 4 - common bus

It will be recalled that the estimated tire, which is typically a common wire may be disposed between the layers of the board. This arrangement is necessary when tracing schemes for devices operating at a high frequency to minimize the influence of the signals from different elements to each other.

If you use the manual method, it is possible to take into account all the shortcomings and do what is shown in Figure 5. The differences are more than noticeable.

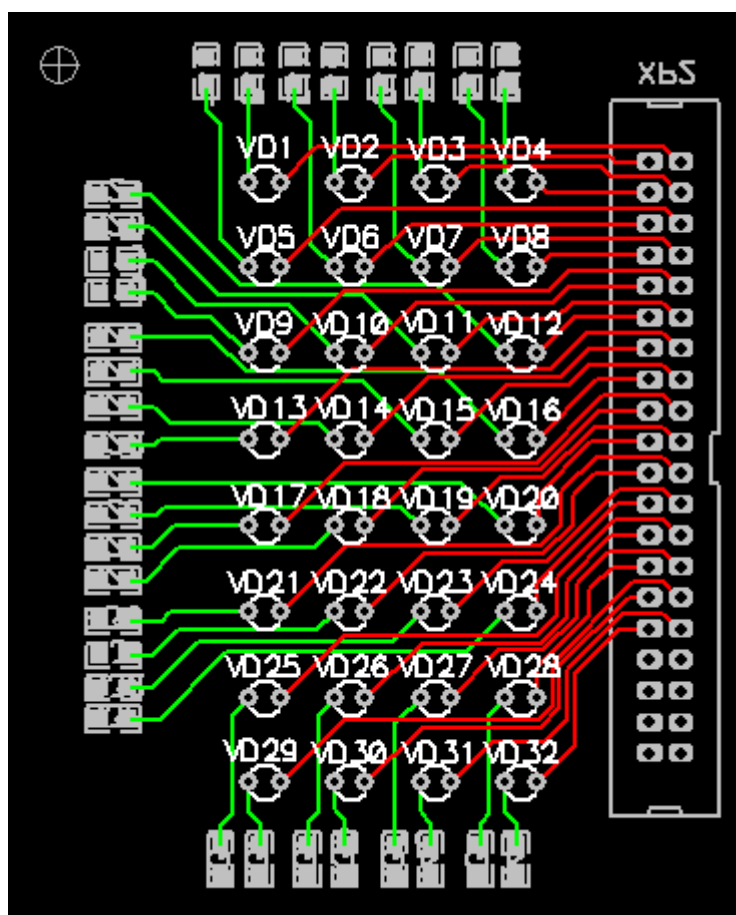


Figure 5 - manual tracing

Emulator signals - a relatively simple device, a printed circuit board which consists of only two layers, you can use manual routing, it does not take much time. However, if there is a development of more complex devices such as computer motherboard, then tracing the manual method is practically impossible. Then the most suitable is automatic routing.

References:

1. Bruce Carter. Machinery PCB layout.2005.
2. R. Linsker. An iterative-improvement penalty-function-driven wire routing system. 1984.
3. F. Rubin. An iterative technique for printed wire routing. 1974.
4. Tech PCB layout [Network resource]. – Access mode: <http://cxem.net/comp/comp40.php>