E. O. HOLLAND and Miss P. R. K. CHAPMAN

Printed wiring layout techniques

Printed wiring has been used in Marconi equipment for almost 20 years and in that time many innovations have been introduced both in materials and in manufacturing techniques. Though originally largely used for consumer products for ease of mass production, Marconi introduced it to achieve consistent and improved performance and increased reliability.

Early board designs, then as now, based on a 0·1in grid matrix, had to take account of varying sizes of components not designed for this form of construction. The introduction of the transistor and later integrated circuits brought about significant changes in the design of printed board layout including the use of double-sided boards. The density of components was considerably increased and with it the complication of wiring layout.

This has led to several new methods ensuring the greater accuracy of layout, including specially prepared 'pad masters' and different coloured tapes.

Another recently introduced method employs photo plotting machines controlled by punched paper tapes. By using a digitizer it is possible to plot the co-ordinates of tracks and components and to use the resultant punched paper tapes for the production of accurate photo masters.

Various computer programs have been developed to speed up the production of printed wiring boards. It is possible to work out the routeing of highly complex boards. By using visual display and light pen, additions and improvements can be made.

Introduction

Printed wiring has been used in Marconi equipment for almost 20 years, the company being one of the first to apply the technique to professional electronics production. Automatic printed board punching machines, designed and built in the Company, were in operation in 1958, and at about the same time one of the first printed board flow-soldering machines was acquired and put into service.

The first applications of printed wiring were to television studio equipment, and the very first printed board put into production was an accessory unit for the Mark III television camera[1].

There had been already some use of printed boards elsewhere, particularly in consumer products where there were obvious advantages in mass production. However, printed wiring was introduced into Marconi equipment to achieve consistent and improved performance, by accurately defining the position of each component and interconnecting conductor, so that each production circuit was electrically and mechanically similar to the original design. Another major objective was greater reliability. In addition, the two-dimensional nature of the printed board led to easier assembly of components and to machine soldering, all helping to improve quality.

Over the years the processes and materials employed have been continuously updated to suit the changing forms of electronic components and to meet increasingly high quality standards and more stringent environmental conditions. This article is concerned in the main with the effect of component evolution on printed board layout, the process by which the design engineer's circuit requirements are translated into the photomaster for board production.

PRINTED BOARD MANUFACTURE

The basic method of printed board manufacture originally adopted has been retained for the vast majority of electronic assemblies for communications equipment. Briefly, in this the raw material is an insulated board coated with copper on one or both sides, the desired pattern of interconnections being obtained by selective removal of copper by etching. Etching is controlled by a photo-sensitive protective layer, which is exposed to light via a photographic master pattern, the photomaster. After etching, boards receive a 'legend' by silk screen printing, the legend indicating the position and circuit identity of each component to aid assembly, inspection and test and subsequent maintenance. Boards are next drilled to receive the component end wires on assembly and are finally soldered.

Layout technique

HISTORY

From the beginning a 0·1in grid matrix was adopted as the dimensional basis for layout, although at first very few components were available whose wire terminations would directly fit the grid. The standard hole size for the component wires was then 0·052in set by typical wire diameters, and it was punched or drilled into the centre of a copper 'pad' of 0·15in diameter. The purpose of the pad, a locally enlarged area of the copper track or wiring, is to ensure an electrically and mechanically sound soldered joint where the component wire and the track meet. Early printed boards carried miniature 7- and 9-pin thermionic valves and the large passive components associated with them and their h.t supplies of about 250 volts.

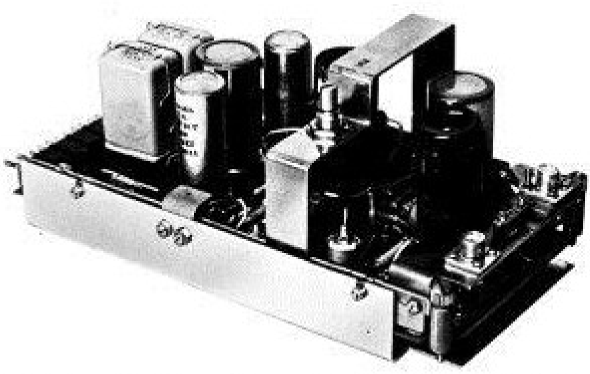

Figure 1. An early design using a printed wiring board which supported valves, relays and large capacitors

The first board layouts were designed at twice full size, a scale which has continued to be used for most purposes. Conductors were laid down in self-adhesive black tape and paper cut-outs on to a thin transparent plastic sheet placed over a master grid pattern of 0·1in squares. The paper cut-outs provided commonly required patterns, for example for pads and valveholders, etc. On the reverse or 'legend' side of the board layout, on a second transparent sheet, labels in a range of shapes and sizes, representing the various component outlines, were used to arrange the components in the optimum positions for interconnecting. On completion, the wiring-side and the legend-side layouts were separated for the photographic processes, yielding the photomaster for etching and the silk screen for legend printing.

The major problem in layout at first was the accommodation of the components in the board area available, a consequence of being limited to a two-dimensional surface on which to mount them. Interconnection was relatively simple, and it was sufficient to have a conductor pattern on one side of the board only. A fairly early layout is shown in figure 1.

ADVENT OF SOLID-STATE DEVICES

The introduction of the transistor brought about a significant change in the design of printed board layouts. Supply voltages dropped by a factor of at least 10, valve heater wiring was no longer required, valveholders disappeared and power dissipation fell dramatically.

The integrated circuit followed the transistor, and was often packaged in the same sized case, but it had up to 10 lead-out wires to connect instead of three.

The problems of printed board design had now changed, both active and passive components being very much smaller no longer presented the original difficulty of finding space for them on a given size of board. Now many more parts could be accommodated in the same area, and the problem shifted to the interconnections.

To match the smaller components, and in particular the narrower spacing between their connection wires, the width of the copper tracks and the space between them were reduced. Both these changes were permissible electrically owing to the lower levels of current flowing in the conductors and to the lower voltages applied.

In parallel with the reduction in component size, the copper-clad base material became available having copper on both sides, which allowed wiring tracks on both sides of the board.

A little later the plated-through hole process was evolved for these double-sided boards. This enabled a direct connection to be made between one side and the other at discrete points each defined by a pad with a drilled hole which was subsequently plated internally with copper. For some applications through connections have also been made by means of rivets or eyelets.

The reduction in track and spacing widths, as well as the need to align the two conductor patterns at the pads in double-sided boards, called for increased accuracy in layout and dimensional stability in the finished artwork. To meet this requirement a new material was adopted as a base for layout, being a highly stable translucent plastic film overprinted with a very accurate 0·1in grid, which, being coloured a pale blue, could be prevented from appearing in the final photomasters by normal photographic methods.

Integrated circuits developed at a rapid pace. The packaging soon included square or rectangular multipin forms, with the now familiar moulded dual-in-line in a range of sizes governed by the number of pins, from four upwards, containing an increasingly wide range of circuit functions.

As the most commonly used integrated circuit logic families use this standard package, printed board layouts for logic systems consist of uniform rows of these components. The interconnections then comprise a pattern of horizontal conductors on one side of the board and of vertical conductors on the other, with plated-through holes providing the connections between the two sides.

New layout problems

In the interests of performance and reliability, it is now often desirable for a complete logic system or subsystem to be constructed on a single printed board. Inter-board connectors are thereby avoided, as are the electrical interfacing provisions necessary for the extended signal paths, and the total number of integrated circuits required is kept to a minimum. This practice leads, however, to the need to accommodate 50 or more 14-pin units on a single board measuring perhaps 20 × 30mm (8in × 11·5in), and the resulting very high density of interconnections gives rise to new problems in layout.

The layout of such boards is an extremely difficult task, even for highly skilled layout staff, and calls for great tenacity. The intense concentration necessary leads to fatigue in a relatively short period of work, and over the many weeks taken to complete the layout there is a progressive loss of efficiency, and the probability of errors increases.

Layout comprises three main operations: deciding the best position for each i.c package, which can hardly be separated from the second, working out the best routeing for wiring. Finally, the artwork must be completed with the necessary accuracy. Aid for the human operator can be sought under each category.

To improve accuracy and in particular to ensure registration of the two sides of double-sided boards, photomasters have been laid out at four times full size. However, this is not very practical in most cases as the photomaster is too large for comfortable handling.

A method which ensures there is very little displacement between the pads on each side of a double-sided board is to employ a specially prepared 'pad master'. The circuit is designed in the usual way and when the positions of the pads are finalized, a twice full size artwork of the pads and certain critical tracks is made very accurately. Two copies of this artwork are then reproduced photographically on a very stable plastic film. The wiring pattern of each side of the board is then laid on them. Whilst this method has been used successfully for some years, it suffers from creeping of the tape and the possibility of small pieces being detached.

One other method of improving the accuracy of the artwork is to use red and blue translucent tape. This involves firstly the positioning of black pads on a piece of stable film. One side of the wiring is then put onto this using red tape. The film is then turned over and the second side is taped using blue tape and the reverse side of the same black pads. When the artwork is completed the two sides are separated by using colour filters and complementary films in the photographic process. This method ensures that the pads on each side of the photomaster are accurately aligned.

Photo-plotting

A new method of producing accurate photomasters is by use of the photo-plotting machine, controlled by punched paper tapes, and illustrated in figure 2. It consists of a table on which a photographic film or plate is held. Above this is a light source, which is moved on instruction from the tapes in the X and Y directions, or at 45°, in steps which are multiples of 0·001in. The light source is masked by a shutter, also controlled by the tapes, containing apertures of various sizes corresponding to the standard conductor track widths and pad diameters.

As the light source moves across the table the film or plate is exposed, producing a very accurate photomaster of one side of the printed board. It is a relatively speedy process, for example the photomasters for a complex board containing 64 integrated circuits have been completed in less than 2 hours. The need for photographic reduction is eliminated, and also the chance of damage by mishandling which is always present with conventional artwork. The photo-plotter master can readily be reproduced as necessary by re-running the tapes, which are not themselves susceptible to damage.

Tape preparation

The photo-plotter eliminates the need to produce artwork manually to the necessary accuracy for direct photographic production of the master. It is still necessary however first to complete the board layout design.

The problem is then to translate the positions of the tracks and pads on to a punched paper tape to operate the machine. It would be possible in principle to punch manually the co-ordinates of each point in the layout, and the dimensions of the tracks and pads on to a paper tape, enter this tape into a computer and run a program to convert to the photo-plotter tape code, but, as this could involve many thousands of instructions, it is impractical.

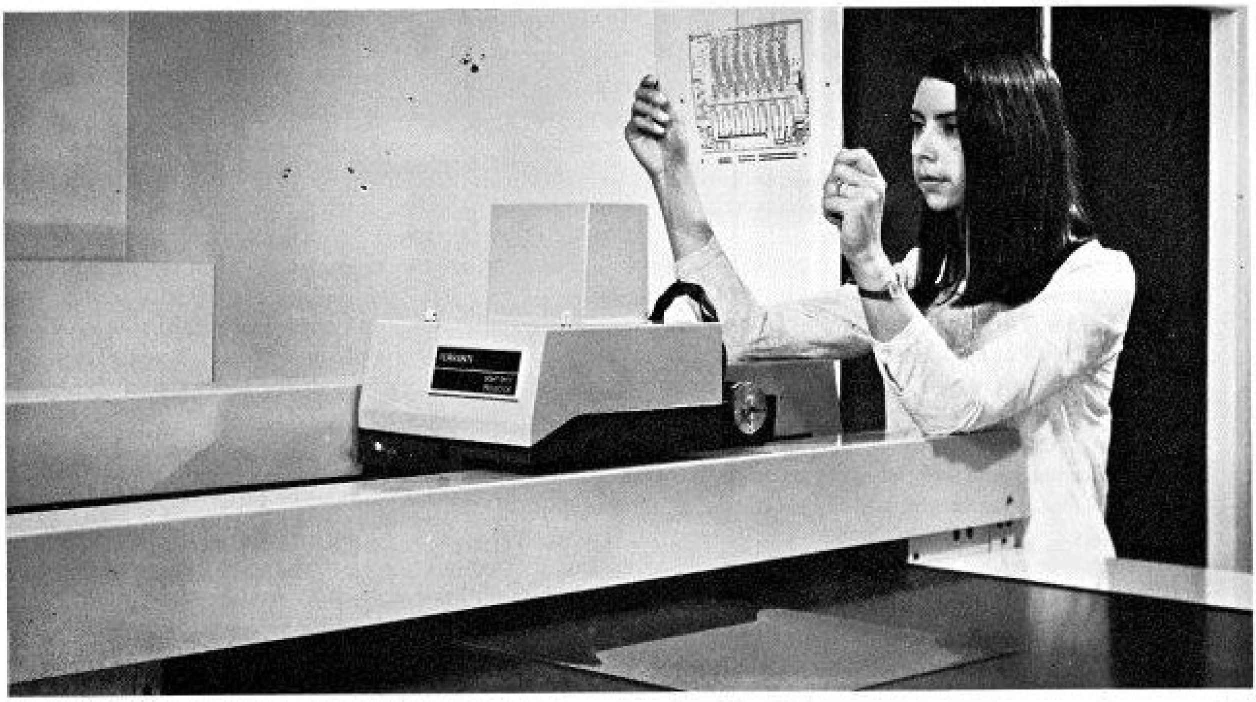

Figure 2. A master plotter for artwork generation

A digitizer can be used to speed up and simplify this process as illustrated in figure 3. It consists of a working surface which incorporates a standard grid matrix on which the photomaster to be digitized is aligned, a paper punch which automatically punches the dimensions on to tapes, a movable head and a keyboard on which additional information can be added manually.

The head, which is fitted with cross-wires for accurate alignment with the photomaster details, is moved over the layout surface point by point and the corresponding co-ordinates are entered on the tape. It is worthwhile to plan the track digitizing so that the photo-plotter will have the smallest movement from track to track as this will shorten the time taken on the photo-plotter and hence the cost. If this planning stage is carefully carried out, the time on the digitizer is also shortened.

Although the use of the digitizer and photo-plotter improves the standard of the artwork, they increase the length of time required for the completion of the photomaster and also the cost. Because of this, various computer programs have been developed both to position the components of the circuit and to connect them. From the outputs of these programs it is very easy to obtain the tapes required to operate the photo-plotter.

Computer aided layout

A suite of programs which has been used by the Broadcasting Studio Engineering Department was developed at the Marconi Research Laboratories[2]. It is intended for the design of logic boards, as already described. The first program in the suite is for 'placement', that is to say, it positions the i.c packages relative to each other and to the board's edge connector, if any. It also produces a density count of the minimum number of tracks passing between the packages in the horizontal and vertical directions. From this it can be calculated whether the circuit will fit into the board size tentatively allocated. A choice of two placements is given, and it is also possible to specify in advance some or all of the i.c package positions. The edge connector positions can also be pre-set. This particular program can also be used to decide placement for subsequent manual design of connections.

When the placement has been approved a following program then determines the routeing of the i.c connections. First, a short 'geometry' tape of the board is put into the computer. This specifies the co-ordinate points for the packages and edge connector, the desired track and track spacing dimensions, and the pad diameters.

Routeing is based primarily on an X-Y wiring system, i.e, following rectangular co-ordinates, although a limited number of 45° tracks can be put in. The X-direction tracks are all placed on one side of the board and, for any change of direction, a plated-through hole is put down and the necessary Y-direction track put on the other side of the board.

The routeing worked out by the computer can be transcribed into paper tape and plotted directly by the photo-plotter. It can first of all be inspected on a computer visual display unit or drawn to scale on a graph plotter as a permanent record. The program also produces a list of connections it has been unable to make. Experience shows that it will usually connect about 95 per cent of the circuit, and the connections it omits can be added by machine or hand digitizing.

Figure 4 shows a layout for a complex board consisting of 14-pin dual-in-line digital logic integrated circuits. In the illustration the photomasters for the two sides of the board are shown superimposed. The package placement and the conductor routeing have been designed by the computer, and its output has been automatically drawn on a photoplotter. The board size is 289mm × 208mm (11·4in × 8·2in).

Other suites of programs have been developed in the G.E.C. Companies. One is an interactive system in which the routeing is shown on a visual display unit, and improvements and additions can be made to the pattern by use of a light pen. Another more comprehensive program developed by the G.E.C. Hirst Research Laboratories can place and route both logic circuits of uniform packages and circuits including a variety of component types, referred to below.

Discrete component boards

The emphasis has so far been on logic circuit boards since these have presented the greatest difficulty in layout. Boards containing a variety of component types, usually called 'discrete' components in the present context, in fact represent generally the greater volume in production. They are less difficult to lay out because the components on average have many fewer connections per unit area. There would nevertheless be some advantages in automating the layout process. The computer programming is more difficult as the components are of different shapes and sizes and their connecting wires vary in number and position, so that much more data has to be handled.

Conclusion

With the growing sophistication of modern electronic equipment, the packaging and wiring up of components becomes increasingly difficult. It is only by using expensive and elaborate aids as described, provided by computer programs and peripheral hardware, that the design layout of printed boards can be executed economically and in a short space of time.

References

[1]E. Davies: The Application of Printed Wiring to Development and Small Batch Production with Particular Reference to Television Equipment; J. Brit. *I.R.E*, Vol. 20, p. 265, April, 1960.

[2]K. H. Hosking: The Use of a Very Fast Routeing Algorithm for Printed Board Design; *Marconi Review*, Vol. 34, No. 182, p. 207, 3rd Quarter, 1971.

Bibliography

Scarlett, J. A.: Printed Circuit Boards for Microelectronics. *The Marconi Series, Van Nostrand Reinhold*, 1970.

Figure 3. A digitizer in operation

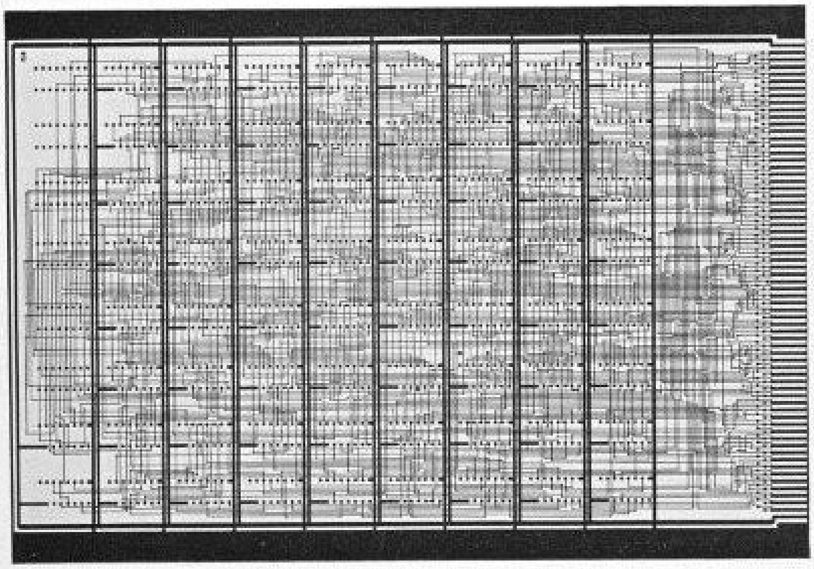

Figure 4. A complex I.C board, where package placement and routeing have been designed by computer, and the resultant output has been automatically drawn on an artwork plotter