

## DECLARATION OF PAMELA STANSBURY

I, Pamela Stansbury, declare:

1. I am an Administrative Supervisor of the Cornell University Library, and specifically the Original Cataloging Unit, located at Cornell University, Ithaca, New York 14853. I am knowledgeable regarding the policies and procedures of the Library as they relate to the receipt, cataloging, and tracking of books and I have personal knowledge of the facts set forth below. I am familiar with these procedures because I have been employed by the Library since 1982, where I began in Technical Services as a Serials Assistant, then a copy cataloger in the Cataloging Unit in 1984. I have held my current position since the mid-1990s.
2. The Cornell Library holds an original copy of the monograph, *A guide to printed circuit board design* / Charles Hamilton. London : Butterworths, 1984.
3. Exhibit A to this Declaration is true and accurate copy of the cover and front matter with library date stamp of *A guide to printed circuit board design*, from the Cornell Library collection. Exhibit A also includes an excerpt of pages 14 to 29 of that volume, showing Chapter 3 entitled "The design layout."
4. I have been asked for information about *A guide to printed circuit board design* – specifically when this item was first made publicly available by the Cornell Library. From my review of the Library's records and my knowledge of the Library's standard procedures, *A guide to printed circuit board design* was cataloged and publicly available at the Engineering Library, in its public stacks, shelved by its Library of Congress classification system number within one or two days of June 7, 1985. A date stamp of "JUN 7 1985" on the inside of the front cover of the item indicates that it arrived at the Engineering Library on June 7, 1985 and at the point would be made available to the public.
5. My opinion is based on the date stamp inside the item's front cover, and my knowledge of the Library's standard procedures relevant to date stamping, cataloging, and shelving items. I have not seen any indication that *A guide to printed circuit board design* would have been an exception to the Library's standard practice. The Library maintains a searchable and publicly accessible catalog of its publications. Therefore, as of June 7, 1985 for *A guide to printed circuit board design* any member of the public could have located this item by searching our catalog by author, title, or subject matter.

I declare that all statements made herein of my knowledge are true, and that all statements made on information and belief are believed to be true, and that these statements were made with the knowledge that willful false statements and the like so made are punishable by fine or imprisonment, or both, under Section 1001 of Title 18 of the United States Code.

 (signature)

Pamela J. Stansbury (name printed)

State of New York

County of Tompkins County

Sworn to before me this 31st day of January, 2020 .



Notary Public

ANDREA DENISE SMITH COLON  
NOTARY PUBLIC-STATE OF NEW YORK  
Tompkins County  
Registration #01SM6276256  
Commission Exp. 02/11/2021



# A Guide to Printed Circuit Board Design

Charles Hamilton

CORNELL UNIVERSITY LIBRARY



\*3 1924 004 223 537\*

Butterworths



CORNELL UNIVERSITY  
LIBRARIES  
ITHACA, N. Y. 14853



Engineering Library  
Carpenter Hall

K 7868.P7H21  
A guide to printed circuit board design



3 1924 004 223 537

All books are subject to re  
Engineering

DATE C

SEP 18 2000  
OCT 24 2002  
NOV 06 2005  
SEP - 1 2011

3 1924 004 223 537

PRINTED IN U.S.A.

**Charles Hamilton**

# **A Guide to Printed Circuit Board Design**

**Butterworths  
London.Boston.  
Durban.Singapore.Sydney.Toronto.Wellington**

All rights reserved. No part of this publication may be reproduced or transmitted in any form or by any means, including photocopying and recording, without the written permission of the copyright holder, applications for which should be addressed to the Publishers. Such written permission must also be obtained before any part of this publication is stored in a retrieval system of any nature.

This book is sold subject to the Standard Conditions of Sale of Net Books and may not be re-sold in the UK below the net price given by the Publishers in their current price list.

First published 1984

© Butterworth & Co. (Publishers) Ltd, 1984

**British Library Cataloguing in Publication Data**

Hamilton, Charles

A guide to printed circuit design.

1. Printed circuits

I. Title

621.381'74 TK7868.P7

ISBN 0-408-01398-2

**Library of Congress Cataloging in Publication Data**

Hamilton, Charles.

A guide to printed circuit design.

Bibliography: p.

Includes index.

1. Printed circuits. 2. Electronic circuit design.

I. Title

TK7868.P7H35 1984 621.381'74 83-18975

ISBN 0-408-01398-2

Phototypeset by  
PRG Graphics Ltd, Redhill, Surrey  
Printed and bound in England by  
Page Brothers Ltd, Norwich, Norfolk

## Preface

Of all the changes and advances the engineering industry in the twentieth century electronics must be one of the most significant point of view of the designer or draughtsman.

In the particular fields of electronic computers, etc. — the design of the printed circuit has assumed an ever-increasing importance serving these industries. Yet it is not the components used in electronic equipment and, of necessity, had to be mounted in forms of discrete wiring providing them. PCBs, where they were used, were

The actual invention of the printed circuit was by Paul Eisler, in February 1943, but there were attempts to try and eliminate complications these early attempts used a non-conductive material on which a conductive metal was sprayed.

The development of the 'subtractive' PCBs followed by the introduction of use of semiconductors and integrated circuits — and the resultant comp

Increasingly, too, whole sections of the industry are now turning to the use of PCBs in the place of what until fairly recently were mechanical

## Preface

Of all the changes and advances that have taken place in the engineering industry in the twentieth century, the impact of electronics must be one of the most significant — not least from the point of view of the designer or draughtsman employed in that industry.

In the particular fields of electronics — radio, television, computers, etc. — the design of the printed circuit board (PCB) has assumed an ever-increasing importance in the drawing-offices serving these industries. Yet it is not so many years ago that the components used in electronic equipment were bulky and heavy and, of necessity, had to be mounted on metal chassis, with cable forms of discrete wiring providing the interconnections between them. PCBs, where they were used, were extremely basic.

The actual invention of the printed circuit has been credited to Paul Eisler, in February 1943, but there had been numerous earlier attempts to try and eliminate complicated discrete wiring. Most of these early attempts used a non-conductive plastics base upon which a conductive metal was sprayed, coated or chemically deposited.

The development of the 'subtractive' process of manufacturing PCBs followed by the introduction of and rapid expansion in the use of semiconductors and integrated circuits inevitably led to the increased use — and the resultant complexity — of the PCB.

Increasingly, too, whole sections of the engineering industry are now turning to the use of PCBs in their products, products which until fairly recently were mechanical or electromechanical.



Examples of this continuing trend include domestic appliances, vehicles and vehicle components, telephone exchanges and communications equipment. New products, too, must not be forgotten — from personal computers and TV games to advanced military missile systems and even satellite and space technology.

It is therefore very much with these developments in mind that this book was written. It is intended primarily as a guide to young designers and draughtsmen(women) who wish to acquaint themselves with the basic design principles of PCBs. Also, more experienced mechanical designers may wish to widen their understanding to include PCB design, in view of the ever-widening applications mentioned.

Some circuit diagrams can look extremely complicated, and faced suddenly with one of these, a young designer, in particular, may feel less than adequate for the task. However, it should be noted that a basic knowledge of mechanical design practice will stand the would-be PCB designer in good stead — not only in understanding the mechanical parameters surrounding the PCB, but also in the design of the PCB itself. A large proportion of the design problems likely to be encountered will be of an essentially mechanical nature. A mechanical designer will therefore find his existing skills invaluable in the field of PCB design.

Other attributes which will help the would-be PCB designer are a methodical and careful approach, patience, and — probably most important of all — a real desire to see a PCB design through to a successful completion. These qualities, together with the observance of the basic principles and procedures outlined in this book will go a long way to solve most, if not all, PCB design problems.

However, it must be realised at the outset that this short guide cannot, and does not claim to, be an authority on the whole subject of PCB design, covering all aspects in depth. PCB design has many facets which can be, and sometimes are, significantly affected by continuing improvements in manufacture, new components and ever-changing assembly methods. Also, the application for the particular PCB is extremely important. A board designed to military specifications will differ significantly from one designed for the mass market consumer field.

Nevertheless, the aim is that the general content of this guide will

be a definite and practical aid to those engaged in the design and layout of high-quality PCBs for subsequent use in professional applications. It is hoped that professional designers will adhere to basic principles as far as is possible and follow step-by-step instructions.

be a definite and practical aid to those who wish to be engaged in the design and layout of high-quality commercial PCBs (Class 1-3) for subsequent use in professional equipment. The guide will adhere to basic principles as far as is possible, presented in easy-to-follow step-by-step instructions.

C.H.

## Acknowledgements

I am indebted to Pye Telecommunications Ltd, Cambridge, England, for granting me permission to have this book published. In fact, I could not have written the book without the background and the experience gained in designing PCBs for the company during the last 10 years. However, the opinions expressed in this book must not be construed as reflecting official company policy with regard to the design of PCBs. The contents of this book are a direct result of my own conclusions — based on my practical experience in designing PCBs.

I am also indebted to Pye Telecommunications Ltd for allowing the reproduction of company drawings used in this book.

The author gratefully acknowledges permission granted to include extracts from the following papers:

C.J. Thwaites (1972), 'Assuring reliability in soldered joints with particular reference to mass-soldering of printed circuit assemblies' The International Tin Research Council.

Henry G. Manfield (1982), 'Printed circuits — a general overview of the various types and methods of manufacture' *Electronic Production*.

The author also gratefully acknowledges illustrations supplied by the following drawing-office equipment manufacturers/suppliers:

Cherry (Drawing Office and Printing Equipment Ltd),  
House on the Creek,  
Ray Mead Road,  
Maidenhead,  
Berks

Swann-Morton Ltd,  
Owlerton Green  
Sheffield S6 2BJ

# Contents

	Preface	
	Acknowledgements	
<b>Chapter 1</b>	Understanding the circuit diagram	1
<b>Chapter 2</b>	Compiling a Component Information File	9
<b>Chapter 3</b>	The design layout	14
<b>Chapter 4</b>	Preparing the master artworks	30
<b>Chapter 5</b>	Generating CAD master patterns	53
<b>Chapter 6</b>	Preparing the production drawing and production photography	57
<b>Chapter 7</b>	Preparing the assembly drawings	63
<b>Chapter 8</b>	Two case histories	68
<b>Chapter 9</b>	Manufacturing and flow soldering the PCB	73
	Glossary	83
	Guide to manufacturers and distributors	87
	Index	89



### 3 The design layout

Before commencing with the design layout proper, consideration must be given to the material to be used for the PCB. The material selected must:

- Be strong enough to withstand mechanical shock.
- Possess heat resistance for the various manufacturing processes.
- Have electrical properties adequate for the electrical circuit(s).
- Be able to resist environmental hazards.
- Be available in large sheets which are easily cut and drilled.
- Have flame-resistant properties.
- Be reasonably priced.

From this list it will be seen that a compromise must be made to obtain optimum properties. A large number of different materials are available throughout the world, suitable for every application.

The most commonly used material is flame-retardant epoxide-impregnated glass-fibre cloth laminate with electrolytically deposited copper foil on one surface or, most usually, both surfaces. Phenolic paper laminates are also used for the many mass-consumer type of applications and, again, many grades are available. These materials are more easily punched or cropped than the epoxide/glass materials. They are also much cheaper. At the other end of the scale, materials are available with very low losses for microwave applications. Also, flexible materials, especially those made from copper clad polyimide, are used to make printed wiring harnesses and other circuits where twisting and bending properties are important.

All these materials, both rigid and chemicals. Ceramic and vitreous are usually used to make 'hybrid' circuit boards, as defined here.

Also worth mentioning at this stage are the problems which arise. These occur when a large number of components are to be made and assembled on a PCB area available becomes 'over-density' of individual PCBs and then joining them together, a multi-layer board resin used for the power supply, signal transfer between circuits can be minimised by the various layers, relative to one another.

However, as already stated, the designer must proceed to the design layout on the PCB. The draughtsman should already have gathered the component details and a full in-depth discussion with the engineer. The circuit must precede any work on the PCB (Chapter 1).

Leading on from this, the first of the mechanical details affecting the design are as are known. These are most usually:

(1) *The basic shape and size of the board* should be made to keep the board clear of notches and cutouts. However, when the internal corners should be rounded to facilitate routing of the board profile by the board manufacturer.

(2) *The PCB fixing hole position* connection, the total free movement of the board should be calculated and added to screw-head clearance to avoid any short circuits to adjacent components.

(3) *Inputs/outputs to and from the board* edge connectors, other plugs and sockets for discrete wiring terminations, etc.

(4) *Fixed component positions on the board* panel-operated components — position of components on the board.

All these materials, both rigid and flexible, are based on organic chemicals. Ceramic and vitreous enamelled substrates are most usually used to make 'hybrid' circuits, rather than printed circuit boards, as defined here.

Also worth mentioning at this stage are 'multi-layer' circuits. These occur when a large number of tracks and components have to be made and assembled on a PCB to the extent that the surface area available becomes 'over-dense'. By making a series of individual PCBs and then joining them, stackwise, one on top of the other, a multi-layer board results. The various layers can be used for the power supply, signal tracking, etc. Thus, 'cross-talk' between circuits can be minimised by careful layout and placement of the various layers, relative to one another.

However, as already stated, the base material for most PCBs is usually rigid copper-clad glass epoxy laminate. We will therefore proceed to the design layout on that basis – bearing in mind that the draughtsman should already have obtained the circuit diagram and gathered the component details. Also, and most importantly, a full in-depth discussion with the engineer/designer responsible for the circuit *must* precede any work done on the design layout (see Chapter 1).

Leading on from this, the first thing to do is to set down the mechanical details affecting the design of the PCB, or as many as are known. These are most usually as follows:

(1) *The basic shape and size of the PCB.* Where possible, efforts should be made to keep the board profile regular, avoiding any notches and cutouts. However, where these are deemed necessary, then the internal corners should be radiused. This will allow NC routing of the board profile by the board manufacturer, if required.

(2) *The PCB fixing hole positions and sizes maximum.* In this connection, the *total* free movement of the PCB fasteners *must* be calculated and added to screw-head or nut dimensions (maximum), to avoid any short circuits to adjacent conductors.

(3) *Inputs/outputs to and from the PCB.* This category may cover edge connectors, other plugs and sockets, headers and pin positions for discrete wiring terminations, etc.

(4) *Fixed component positions on the PCB.* For example, front-panel-operated components — potentiometers, push-button or

rotary switches, on-off controls, etc.

(5) *All metalwork/ancillaries affecting the PCB.* For example, screens, mounting frames, heat sinks, board guides, transipads, etc.

(6) *Any height restrictions on the PCB.* These must be established and noted at the outset of the layout, with particular regard to the positioning of large components.

In establishing the mechanical parameters as above, efforts should be made to design to 'on-grid' positions, thus making the mechanical accuracy of the initial layout easier to achieve. This will, in turn, lead to easier generation of accurate artwork later.

The mechanical details (1-5, above) are drawn, *viewed from the*



Figure 3.1 The Cherry adjustable light-table

*component side of the PCB*, in pencil drafting film (Draftex) which has precision grid. The grid and the Draftex are mounted on a light-table fitted with parallel motion (see Figure 3.1).

The mechanical details are drawn (first choice) or 2:1 — it being borne in mind that the scale of layout that can be accommodated and CAD techniques is of the order of 1:1.

Figure 3.1 illustrates the Cherry A parallel motion. The overall working area is 914mm; it has an illuminated top. Illumination is provided by eight fluorescent tubes. The stand will raise and tilt the illuminated top. A soft green anti-glare filter is available. Alternatively, colour-corrected tubes are available. The stand can also be supplied with a solid state dimming unit, powered by a mains supply.

If cost or available space is of prime importance, a flat light-box (Figure 3.2) may be used.

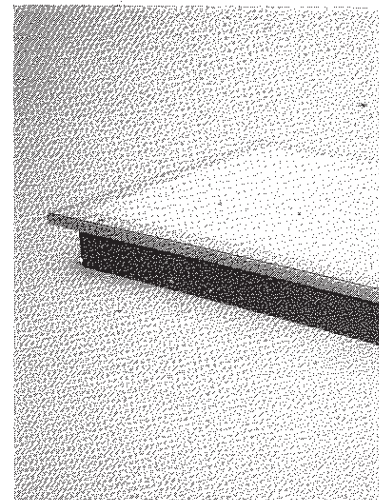


Figure 3.2 The Cherry flat light-box

*component side of the PCB*, in pencil, on 0.003in thick polyester drafting film (Draftex) which has been placed over a 0.100in precision grid. The grid and the Draftex sheet should preferably be mounted on a light-table fitted with a drafting machine/parallel motion (see Figure 3.1).

The mechanical details are drawn at the preferred scale of 4:1 (first choice) or 2:1 — it being borne in mind that the maximum size of layout that can be accommodated by industrial photographers and CAD techniques is of the order of 950mm × 700mm.

Figure 3.1 illustrates the Cherry AO 'Brilliant Desk', fitted with a parallel motion. The overall working area of this stand is 1219mm × 914mm; it has an illuminated area of 1016mm × 762mm. Illumination is provided by eight fluorescent tubes and the spring-loaded stand will raise and tilt the illuminated surface to the desired position. A soft green anti-glare filter is supplied as standard. Alternatively, colour-corrected tubes with a white filter are available. The stand can also be supplied with additional features, e.g. solid state dimming unit, powered cooling, precision glass grid, etc.

If cost or available space is of prime importance, the Cherry AO flat light-box (Figure 3.2) may be used. This unit can be placed on

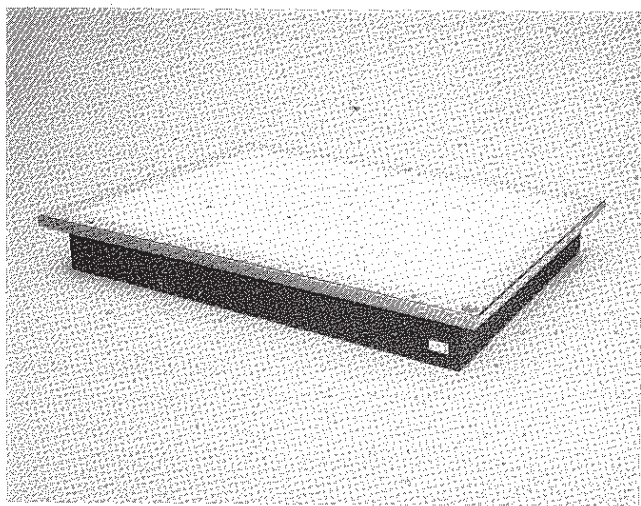


Figure 3.2 The Cherry flat light-box



top of existing plan chests or desks. A fixed-angle version is also available at no extra cost. The illuminated surface is identical with the one shown in Figure 3.1, and the same additional features can also be supplied as required.

With the mechanical details drawn, we can now move on to the generation of the components to be used in the design. It is in connection with this that the designer will find the 'overlay technique' a considerable aid, particularly if the design is at all complex. The overlay technique involves a first overlay sheet for the component outlines and a second overlay sheet to carry all of the interconnections between the component outlines, i.e. the 'tracking' on the PCB.

The provision of component outlines for the first overlay sheet can be most readily achieved by purchasing a system developed by Bishop Graphic Inc., called the New Bishop Electronic Circuit Layout Design System. This comprises a collection of individually die-cut, transparent layout patterns of the more commonly used electronic components, called Puppets (Trade Mark). These Puppets adhere positively to any glossy drafting film, but can be lifted easily with the aid of tweezers and rearranged at will, until a satisfactory layout configuration is achieved.

Alternatively, if the drawing-office concerned has flat-bed diazo printing facilities 'in-house', then it is entirely possible to construct a similar system in the following manner:

- (1) Take the component outline drawings already referred to in Chapter 2 (Figure 2.1) and make Ozakling copies of these on a flat-bed printer.
- (2) Peel the backing off the Ozakling copies and place on a sheet of Craylux.
- (3) Cut out the individual component outlines with scissors, peel the backing off the Craylux sheet, and the component outline is ready for use.

Whichever system is used, it will be found that the Puppets and/or component outlines will both adhere in the prescribed manner on 0.002in thick Melinex glossy drafting film. This Melinex sheet is the first overlay sheet.

While varying positions for the components are being tried, the interconnecting tracks are line-drawn on the second overlay sheet,

which is the top sheet in the overlay. 0.003in thick Draftex. The matt finish will effectively prevent the Puppets/copies from coming off the Melinex sheet underneath.

The solder side tracks can be drawn on the second overlay sheet in blue (or in any combination of colours on one side from the other). A clutch pen or Pentel coloured leads will be found suitable. Pentel coloured leads, with their fine codings, polarity symbols, etc., are 0.9mm diameter Pentel lead, so as to avoid tracking.

In the system described, rapid plan reorientation of the Puppets/components is possible — rather than the more conventional method of redrawing — does not lend itself to the particular requirements of the overlay system.

To recap the overlay system, refer to

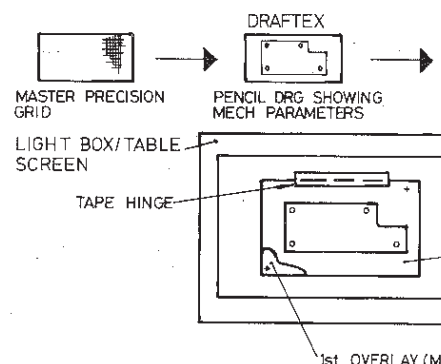


Figure 3.3 The overlay drawing system

Note that the second overlay sheet is drawn on the first overlay sheet so that it can be easily lifted for access to the component outlines underneath for initial plan reorientation.

As already stated, in using the overlay system, a light-box is recommended, as there is no need for the designer and the precision grid.

which is the top sheet in the overlay system. This top sheet is 0.003in thick Draftex. The matt finish on the second overlay sheet will effectively prevent the Puppets/component outlines from lifting off the Melinex sheet underneath.

The solder side tracks can be drawn in red and the component side tracks in blue (or in any combination of colours to distinguish one side from the other). A clutch pencil carrying 0.9mm diameter Pentel coloured leads will be found suitable for the purpose. Circuit codings, polarity symbols, etc., are added using ordinary black 0.9mm diameter Pentel lead, so as not to conflict with the coloured tracking.

In the system described, rapid placement and any subsequent reorientation of the Puppets/component outlines is easily achieved — rather than the more conventional drawing and erasing, which does not lend itself to the particular requirements of PCB design.

To recap the overlay system, refer to Figure 3.3.

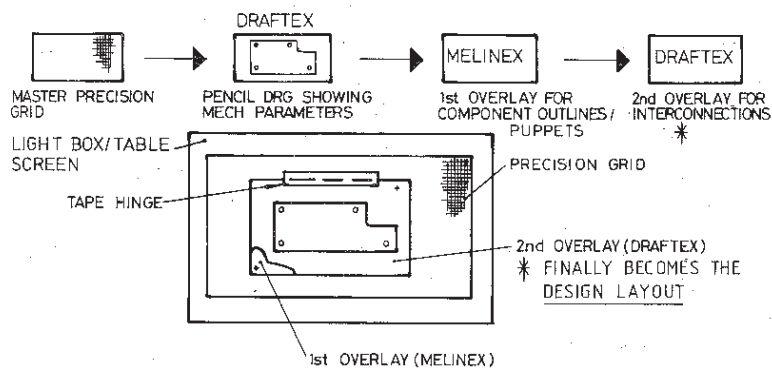


Figure 3.3 The overlay drawing system

Note that the second overlay sheet is hinged along the top only, so that it can be easily lifted for access to the Puppets/component outlines underneath for initial placement and any subsequent reorientation.

As already stated, in using the overlay system a light-table or light-box is recommended, as there are three sheets between the designer and the precision grid. Consequently, the Puppets/

component outlines may not be clearly visible under the second overlay sheet without the assistance of back-lighting.

However, the PCB designer quickly realises that it is not the fitting of the components into the available space which is the main consideration — the orientation of the components relative to one another is the really crucial factor. It is in determining the component configuration which satisfies both the circuit design (in general, short interconnections) *and* the required component density that most of the design time is spent. Therefore, methods which will enable the designer to do this rapidly and effectively, such as the method described, are extremely important.

As the layout is being developed — and probably at a very early stage — the designer will be able to decide an important and fundamental feature of the PCB — namely whether the board material will be single- or double-sided. Where the circuit is simple and straightforward, efforts should be made to contain the tracking to one side, i.e. single-sided material, even if link wires have to be fitted. The designer should compare the possible cost savings offered by cheaper single-sided board material, as against the cost of fitting links. A general rule is difficult to formulate, but for small boards containing less than 50 components, anything in excess of four or five links should probably be double-sided. Where link wires are used, they are best fitted from the component side with a common mounting pitch, to aid production assembly methods.

With more complex circuits, the designer will automatically use double-sided material. In the case of radiofrequency (RF) circuits, it may be necessary to introduce a ground plane (—ve) to achieve a shielding effect. If this is the case, the copper surface of the component side of the board will be largely given over to the ground plane (see Figure 3.4).

Link wires may also be used on double-sided boards, but these must be insulated (sleeved) from the ground plane/component side tracks.

With regard to holes, selection is largely dependent upon the particular company standards and production methods (see also Chapter 4). Most production departments today favour 'straight-through' flow soldering techniques for all but the very smallest quantities of boards incorporating conventional axial/radial leaded components. Therefore, single- and double-sided boards should

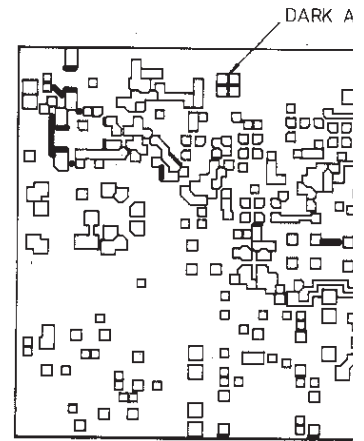


Figure 3.4 A typical RF ground plane. The side on a double-sided board

specify plated holes (tin/lead), as the soldering technique. Non-plated holes should be a minimum, as these will normally require the plating process on the PCB.

A useful technique, as the design progresses, is to use a colour (say) green all of the ground plane. The ground (—ve) connections are then identified by subsequent linking on the layout; this

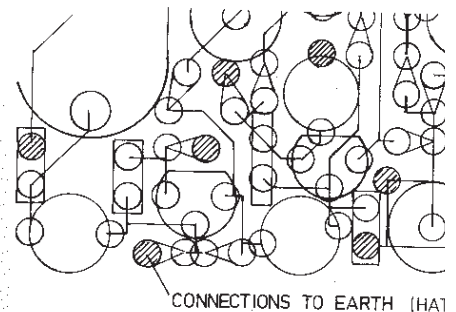


Figure 3.5 Identifying ground connections

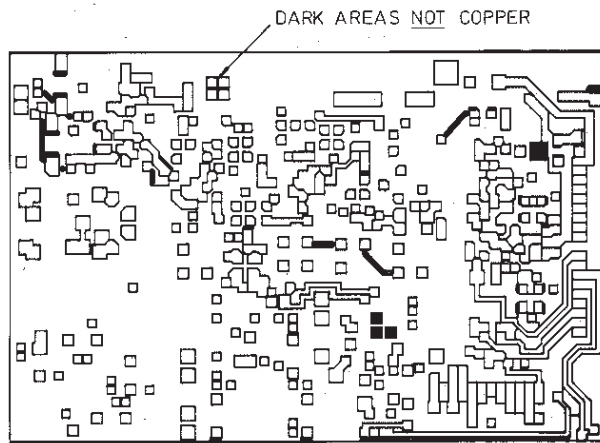


Figure 3.4 A typical RF ground plane. The copper surface of the component side on a double-sided board

specify plated holes (tin/lead), as this greatly enhances the flow soldering technique. Non-plated holes should be kept to a minimum, as these will normally require a secondary drilling-out after the plating process on the PCB.

A useful technique, as the design layout progresses, is to 'hatch' or colour (say) green all of the ground connections (see Figure 3.5). The ground (–ve) connections are then more easily identified for subsequent linking on the layout; this is shown as tracking on the

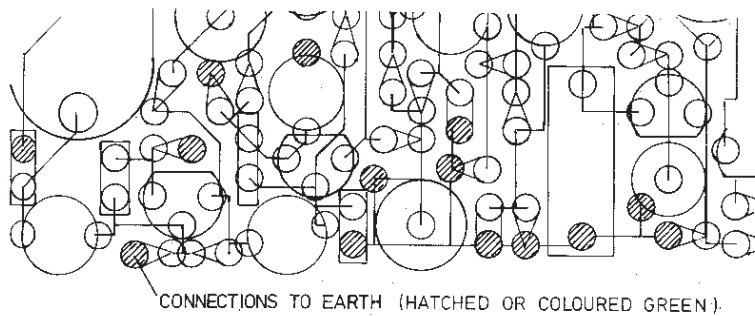


Figure 3.5 Identifying ground connections



solder side (where possible) and the remainder through plated holes to the ground plane on the component side of the board.

On boards of this type, component side tracks will probably be unavoidable where high-density tracking and packaging of the components are required. In this case, the component side tracks should avoid running through the critical RF areas of the board — which require, as far as possible, unbroken areas of ground plane (see Figure 3.4).

Company standards will probably specify the type and grade of board material to be used. As already stated, however, this is usually glass epoxy laminate for good-quality commercial PCBs used in professional equipment. Material thickness is normally 1.6mm (0.062in) with 1oz copper cladding (before plating) single- or double-sided; 0.8mm (0.031in) material thickness is also available for a minority of applications.

Additional aspects which must be considered and catered for at the design layout stage are listed below.

(1) *Access and servicing.* Easy access to the PCB (both sides) must be designed into the equipment, with as few fixings as possible, to aid quick removal. Ideally, 'plug-in' boards present the fewest problems from the removal point of view. Discrete wiring connections directly soldered to the board should be kept to the absolute minimum, as desoldering is time-consuming and damage can easily occur to adjacent components/tracking nearest to the soldered connection. Also, component densities must not be such that low-profile components are incapable of being extracted because of adjacent overhanging components. Puppets/component outlines can be laid out touching one another but *never* overlapping, regardless of height.

(2) *Thermal considerations.* The PCB designer should be aware of the heat-sensitive and/or heat-developing components, so that he can locate them to the best advantage. Usually, heat-developing components are raised above the board or fixed to adjacent metalwork to aid heat dispersal. 'Clip-on' heat sinks are also available for most of the standard transistor packages.

(3) *Tuning and adjustment of the variable components.* This can cover tuning coils, variable resistors/capacitors/trimmers, potentiometers, etc. It is important that access to the variable

components be as unrestricted as regard to vertically mounted components in a direction parallel to the surface, getting over the top of, or in between, therefore a prime consideration for components needing adjustment/tuning.

(4) *Location of high-impedance circuitry must* be located away from signal tracking. Such circuitry is excluded from these types of track.

(5) *Logical signal flow.* The designer should follow the circuit diagram with a view to considerably improve serviceability. Components related to a particular diagram are placed close together in a logical manner on the design layout. It is important to show the signal flow through on the design layout to analogue circuitry.

(6) *Screening of individual components/stages,* particularly at radio frequencies, may require screening to avoid interfering electrically with adjacent stages. Screening can vary from a simple shield to a completely enclosed metal box. In partitioning may also be necessary.

(7) *Maintaining company quality.* The individual company and the type of PCB designer should ensure that he has quality control procedures, organized in a concept of 'fitness for use' may be the ultimate in reliability may be the aim.

(8) *Environmental tests.* These tests assess the ability of a PCB assembly to operate under temperature, humidity, mechanical stress. Test procedures should be selected from IEC 68, IEC 489-1 (Part 1) and D international standards. Tests should be internationally accepted tests to ensure performance to be produced for in complete equipments, according to

components be as unrestricted as possible — particularly with regard to vertically mounted components that have to be adjusted in a direction parallel to the surface of the PCB, which means getting over the top of, or in between, components. Access is therefore a prime consideration in the placement of variable components needing adjustment/tuning.

(4) *Location of high-impedance circuitry.* High-impedance circuitry *must* be located away from both supply and high-level signal tracking. Such circuitry is extremely sensitive to 'pick-up' from these types of track.

(5) *Logical signal flow.* The design layout should attempt to follow the circuit diagram with regard to signal flow. It will considerably improve serviceability if adjacent stages on the circuit diagram are placed close together, and follow on in the same manner on the design layout. It is then possible to easily trace the signal flow through on the design layout. This applies particularly to analogue circuitry.

(6) *Screening of individual components/stages.* Some components/stages, particularly those operating at radio frequencies, may require screening to prevent them from interfering electrically with adjoining circuitry. The type of screening can vary from a simple vertical metal plate to a completely enclosed metal box. In the case of the latter, internal partitioning may also be necessary.

(7) *Maintaining company quality standards.* This will depend on the individual company and the type of product it produces. The PCB designer should ensure that he is familiar with the company's quality control procedures, organisation and agreements. The concept of 'fitness for use' may be the rule in one company, while the ultimate in reliability may be the only acceptable criterion in another.

(8) *Environmental tests.* These are necessary to establish the ability of a PCB assembly to operate within specified limits of temperature, humidity, mechanical stress and other conditions. Test procedures should be selected from the recommendations of IEC68, IEC489-1 (Part 1) and Defence Standard 07-5. These internationally accepted tests enable a recorded statement of performance to be produced for individual PCB assemblies or complete equipments, according to agreed levels of assessment,

and also covering conditions of service.

(9) *Provision for test points and other electrical tests.* In order to facilitate the main requirements of electrical testing — including automatic test equipment (ATE) — it is extremely important for the PCB designer to liaise effectively with the test equipment engineer(s) *at the initial design layout stage*. Facilities required on the PCB by test equipment engineering will include a minimum of two holes for test jig location. A number of test pads will also be required on the solder side of the PCB — these are usually incorporated in the conductor pattern. (See also Chapter 4.)

(10) *Ease of component assembly.* This will depend on the type of components used (discrete, dual-in-line packages, chips, etc.) and the assembly method employed (manual, auto-insert, auto-placement, etc.). Again, the PCB designer must liaise effectively with the production engineering department as the design layout progresses to ensure that the PCB will be compatible with the production method(s) used. Consultation with production engineering must *never* be left until the design layout is completed. This will only lead to inevitable changes which will be difficult to incorporate at a late stage in the development of the PCB. (See also Chapter 7.)

(11) *RF considerations.* With RF circuitry, extra care must be taken with the design layout. Tracks must be kept short, a good ground plane maintained, and decoupling components placed as close as possible to their respective stages.

(12) *VHF/UHF considerations.* With VHF/UHF designs, the tracks themselves may become part of the actual circuit. 'Stripline' (see Glossary) designs especially will specify a track length and width, which must be followed exactly. In this situation, the total involvement of the engineer/circuit designer is crucial to a successful design.

(13) *Calculating terminal areas (pads) and track widths.* This subject is covered fully in Chapter 4.

(14) *Awareness of minimum clearances and minimum track widths.* This subject is also covered fully in Chapter 4.

(15) *Provision for alternative components.* See Chapter 2.

The PCB designer should endeavour to incorporate *all* of the above features (1)–(15) affecting the design layout as it progresses.

Guidance may be necessary. The consultation and liaison with the engineer must be a good design practice. The PCB designer must develop a good working relationship from the start, through to completion, with the engineer at every stage — as well as the other departments. However, if the PCB designer finds the circuit's requirements, owing to the engineer's help must be sought to ensure that without detriment to the overall system, the PCB designer takes the trouble to develop a good relationship, as already stated, the engineer/circuit designer receptive solution will usually be achieved.

## Computer-type logic circuit

In the case of computer-type logic, particular attention has to be paid to decoupling.

## Ground distribution

Ideally, the PCB ground system should be on one side of the board and all signal tracks on the other. In reality, however, this is not always possible because of component and tracking requirements. The ground distribution should be carried out as far as possible on the ground distribution system should be everywhere, even if this means large ground tracks.

To further approach the ground plane, the horizontal and vertical power and ground tracks should be tied together using plate package (DIP) site — or at least as close together of horizontal and vertical tracks. Because long 'floating' distribution lines or noise distribution systems, thus all

Guidance may be necessary. Therefore, the fullest possible consultation and liaison with the engineer concerned is essential to good design practice. The PCB designer must therefore cultivate a good working relationship from the beginning of the design, through to completion, with the engineer being closely involved at every stage — as well as the other departments mentioned above. However, if the PCB designer finds that he cannot fully incorporate the circuit's requirements, owing to, say, lack of space, then the engineer's help must be sought to find an agreed compromise, without detriment to the overall specification of the PCB. If the PCB designer takes the trouble to cultivate a good working relationship, as already stated, then he will usually find the engineer/circuit designer receptive to change, and a compromise solution will usually be achieved.

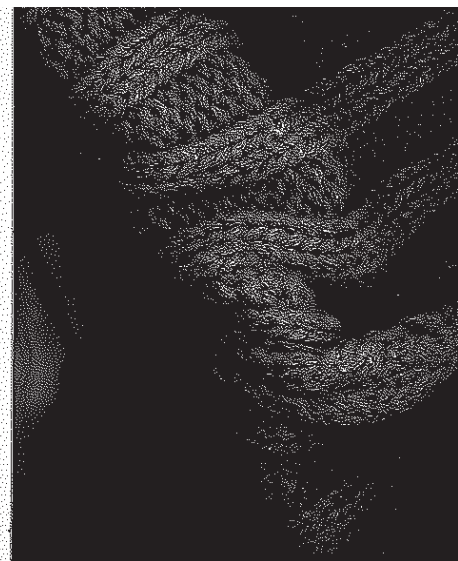
### Computer-type logic circuits

In the case of computer-type logic circuits (see Figure 1.4), particular attention has to be paid to ground distribution and decoupling.

#### Ground distribution

Ideally, the PCB ground system should consist of a ground plane on one side of the board and all signal and power distribution on the other. In reality, however, this can rarely be achieved — because of component and tracking densities — but the concept should be carried out as far as possible. This means in effect that the ground distribution system should be as wide as possible everywhere, even if this means large variations in the width(s) of the ground tracks.

To further approach the ground plane ideal, a grid of horizontal and vertical power and ground tracks on opposite sides of the board should be tied together using plated holes at each dual-in-line package (DIP) site — or at least as often as possible. The tying together of horizontal and vertical tracks is important electrically, because long 'floating' distribution lines can easily act as antennae or noise distribution systems, thus allowing noise to propagate and





exceed device thresholds. In addition to reducing ground noise, an effective ground grid can help to reduce 'cross-talk' between address and data lines.

Figure 3.6 shows an effective ground/power grid for a typical

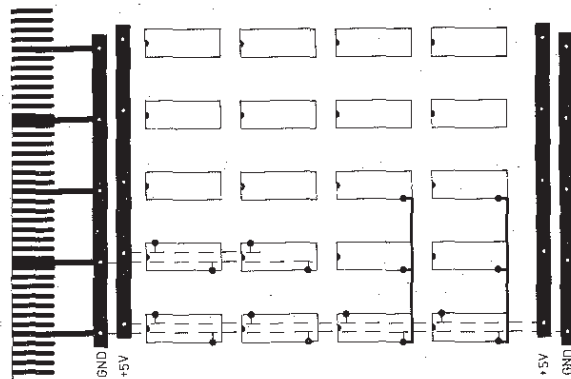


Figure 3.6 A ground/power grid for a computer 'plug-in' logic PCB

computer 'plug-in' logic PCB. Note that the power (5V) and ground rails run horizontally *inside* the rows of DIPs, and are interconnected via plated holes to the rails at the left- and right-hand edges of the PCB.

The ground pins of the DIPs should additionally be connected vertically — a track of the same thickness as used for the signal interconnections should be sufficient for this. Note also that the ground connections are brought out horizontally at the end of each row of DIPs to the edge connector (five positions) situated at the left-hand edge of the board. If these basic design rules are followed (as far as is possible within the constraints of the circuit), then a satisfactory ground/power grid system will result.

In any event, engineer's advice should be sought regarding the ground/power distribution network on logic PCBs.

## Problems

Engineer's advice may also be helpful in diagnosing capacitance problems on the logic are as follows:

**Signal lines.** Excessive capacitance on a signal line can result in false triggering and/or noise. It can also cause capacitance loading, which slows down pulse circuits and increases delay. High impedance MOS modules are being used, which are usually critical. To minimise signal distortion, signal lines should be avoided.

*Shielding of conductors.* Line can be shielded or isolate other conductors. Line can be shielded by ground conductors parallel to critical conductors. These ground lines then act as a shield.

**Power conductors.** Capacitance is needed to act as a filter for power conductors. They need high capacitance to ground. When voltage changes due to load variations, the capacitance can be provided by running the conductors one over the other (on double-sided boards) or as wide as possible.

**PROMs and ROMs** — (program can easily be used in much the same transistor-transistor logic (TTL) design, attention being paid to ground decoupling.

Having considered all of the foregoing, it must now be checked and is usually best achieved with the connections from a dyeline print colouring them in while he does so), connections called out and checked. With the roles reversed in this manner layout are usually immediately apparent.

## Problems

Engineer's advice may also be necessary to avoid or minimise capacitance problems on the logic PCB, typical examples of which are as follows:

*Signal lines.* Excessive capacitance between signal lines can cause a signal on one line to be injected on to another line, which can result in false triggering and/or noise. Excessive capacitance can also cause capacitance loading, which degrades wave shapes in pulse circuits and increases delay times. Furthermore, when high-impedance MOS modules are being used, this capacitive loading is usually critical. To minimise signal line capacitance, parallel lines should be avoided.

*Shielding of conductors.* Line capacitance can be used to shield or isolate other conductors. Lines can be isolated by running ground conductors parallel to critical or noise-generating lines. These ground lines then act as a shield.

*Power conductors.* Capacitance can also be designed into a PCB to act as a filter for power conductors. Usually, voltage supply lines need high capacitance to ground to reduce noise and minimise voltage changes due to load variations. In this case, maximum capacitance can be provided by running the supply lines parallel, or one over the other (on double-sided boards), and also by making the supply lines as wide as possible.

*PROMs and ROMs* — (programmable) read-only memories — can easily be used in much the same manner as other types of transistor – transistor logic (TTL) design elements, with the usual attention being paid to ground distribution networks and decoupling.

Having considered all of the foregoing and with the design layout completed, it must now be checked against the circuit diagram. This is usually best achieved with the draughtsman calling out the connections from a dyeline print of the circuit diagram (and colouring them in while he does so), with the engineer following the connections called out and checking these on the design layout. With the roles reversed in this manner, any errors on the design layout are usually immediately apparent. As errors/omissions are

found, these must be noted and a list made for subsequent correction by the PCB designer.

With the checking completed, the Puppet/component outlines can be drawn in pencil (black, 2H) on the second overlay sheet, so as not to lead to confusion with the coloured tracking. The mechanical details are also added in the same manner.

Thus, the second overlay sheet now becomes the completed design layout, as it now carries all of the design information on one single sheet — i.e. mechanical parameters, component outlines and

codings, as well as the coloured t both sides of the PCB. (See Figure

The Puppet/component outline filed on Melinex sheets in the ( subsequent re-use. Puppets m component outlines generated 'in shelf life (approximately 1 year), the Ozakling from the Craylux bac

The next step is the preparatio first prototype of the PCB.

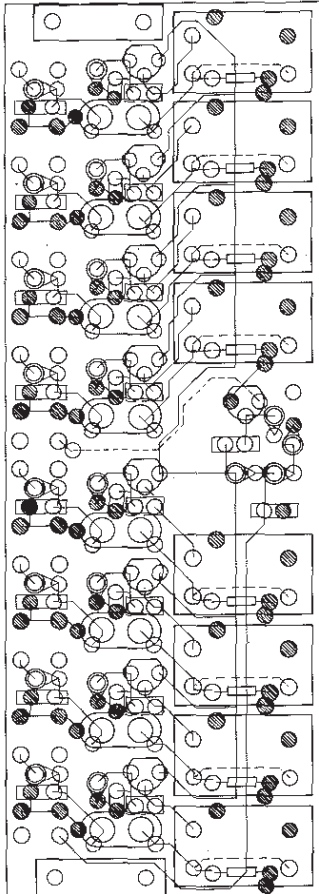
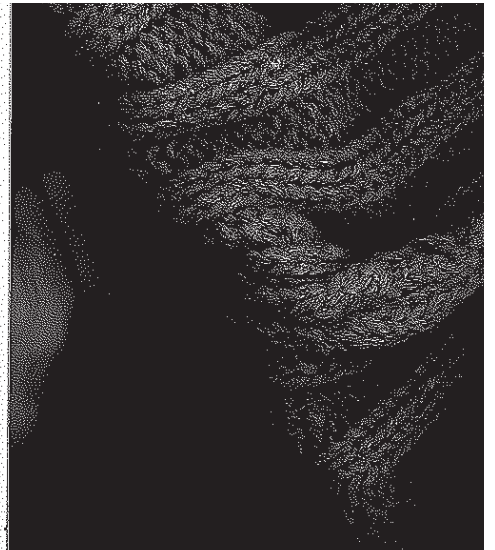


Figure 3.7 A typical PCB design layout

codings, as well as the coloured tracking for one or, most usually, both sides of the PCB. (See Figure 3.7.)

The Puppet/component outline layer can now be dismantled and filed on Melinex sheets in the Component Information File for subsequent re-use. Puppets may be stored indefinitely, but component outlines generated 'in-house' will only have a limited shelf life (approximately 1 year), owing to inevitable separation of the Ozakling from the Craylux backing.

The next step is the preparation of the master artworks for the first prototype of the PCB.

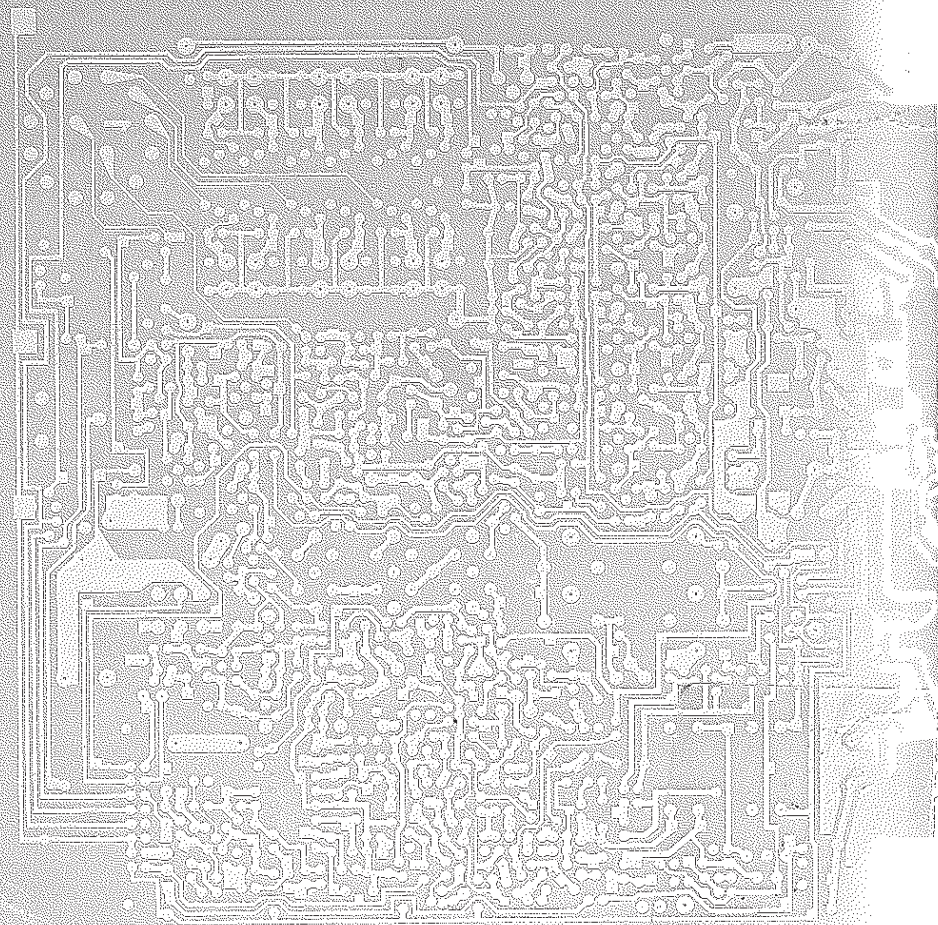




In this age of microelectronics, the design of the printed circuit board has become an increasingly important part of the drawing office that serves the electronics industry. Increasingly, too, many parts of the mechanical engineering industry are turning to the use of printed circuit boards in their products.

However, printed circuit boards can look, and sometimes are, extremely complicated; the high density of components on modern boards combined with the mass of tracking may deter the would-be designer. The aim of this book is to strip away the mystique surrounding PCB design. The logical step-by-step approach explains the fundamentals and shows that a design undertaken in a methodical and careful manner will solve most if not all of the problems likely to be encountered on any conventional board.

This book will therefore be an invaluable guide and practical aid to the young or inexperienced designer in the drawing office.



**Butterworths**

Borough Green, Sevenoaks, Kent TN11 8BH, England

ISBN 0409 03111