ROBO
GRAPHICS

P.C.B.
ARTWORK

1/.Draw board outline on 8x8 GRID.
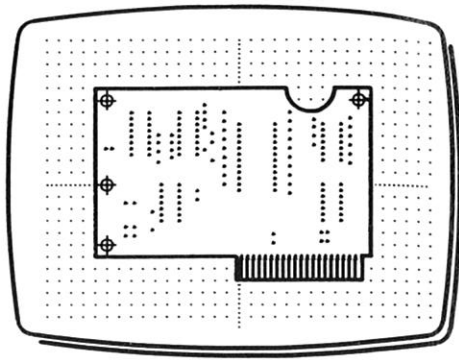
2/.COPY pad patterns from Library.
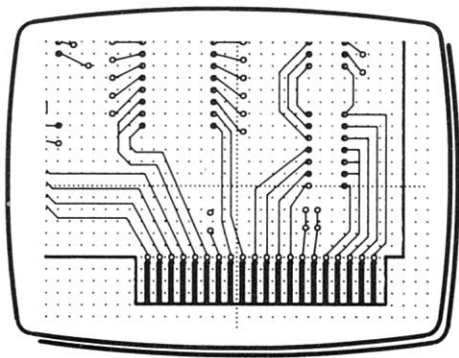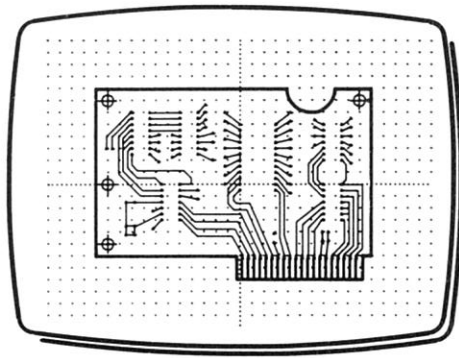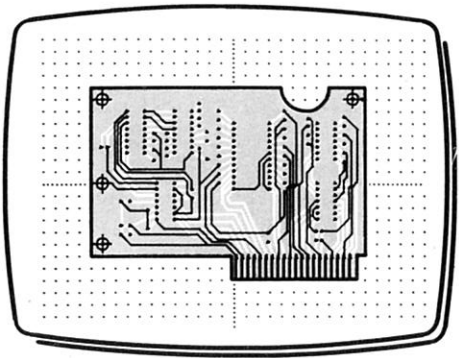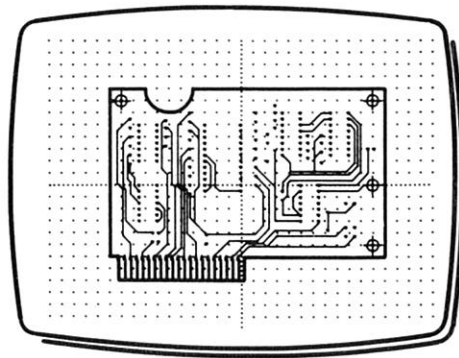
3/.ZOOM to make connections.

4/.FILE finished board.

5/.COPY back & draw component side.

6/.FLIP component side and FILE.

P.C.B.
ARTWORK

The pad patterns on this disk are designed to be
assembled using the 8x8 LOCKED GRID to produce
circuit board track layouts, drilling masks and
component overlays.

You can suppliment these patterns by drawing
your own commonly used shapes and filing them
onto a separate Library disk of your own.

The complete circuits can be filed on ARCHIVE
disks for permanent storage, recalled for
changes and plotted onto film or paper for
checking and use as artwork.

Using a suitable plotter with ROBOPLOT software
and technical pens you can to produce:

* Over-size artwork for photo-reduction.
* Same size positives for direct exposure
* Step and repeat of the same, or mixed circuits.
* Mirror images for double sided boards.
* Component layout for silk-screen.
* Multi-layer artwork.
* Local or global changes of pad shape.

USING ROBOGRAPHICS

This guide assumes that you are familiar with
the basic operation of the ROBOGRAPHICS system.
You should read the user manuals thoroughly
and learn the general drawing techniques first.

IMPORTANT

NEVER USE TWO LIBRARY DISCS WITH THE SAME
VOLUME NUMBER TO DEVELOP THE SAME CIRCUIT.
This also applies to backup copies you may
make of your own or the PCB Library disc.
If you do at the least you will confuse the
system with unexpected results, and at worst
you may lose parts of your circuit.
YOU HAVE BEEN WARNED!

USING THE DISK

SIZE OF BOARD

The only practical limit to the size of circuit that can be produced on the system is the time taken to replay the screen after ZOOM and PAN.

SAVING TIME

Various techniques can be used to save time:-
- Hit the Space bar to curtail screen replay.
- Employ horizontal layering.
- Use SKELETON pad shapes.
- Step and repeat bussing etc.
- FILE common modules for re-use.

It is worthwhile to experiment and a logical procedure is essential. A suggested procedure and an example are included as a guide.

The use of an ACCELERATOR CARD in the APPLE II is strongly recommended to speed up processing by nearly 4 times.

PAD SHAPES

Whilst assembling the cicuit and for check plotting use the SKELETON shapes to save time.
When producing same size artwork the IDEAL pads are used by the update procedure to achieve the required shapes. You may draw your own IDEAL pads as required, determining the best shape for your needs by test plotting.
The IDEAL pads on this disk are designed to be used with a .5mm wet-ink pen for same size plots.

COPY SCALE

The pad patterns appear at different sizes in the Library for clarity. When you COPY them onto the workpage with an 8x8 grid, size can be controlled in steps to correspond to the level of ZOOM used. The relative COPY scale factors for BASE PAGE are indicated on the Index contents illustrations. For other PAGE VIEWS multiply this COPY factor by the ZOOM factor indicated in the Table on page 8.

CONTENTS   INDEX A

| PAD TARGET.	I.C. TARGET.	SPARE.	SPARE.
PAD SKELETON.	I.C. SKELETON.	IDEAL PAD.	IDEAL I.C.
0.1" SPACE.	0.2" SPACE.	0.3" SPACE.	0.4" SPACE.
TO5.	TO5 CASE.	TO92.	TO92 CASE.
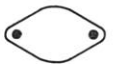
COPY SCALES FOR 8 POINT GRID = 0.1"

PAD TARGET, SKELETON & IDEAL = 6.25 (12.5 AT x2 ZOOM)
IC TARGET, SKELETON & IDEAL = 12.5
BOTTOM 8 PATTERNS   = 25.0

USE SPARE BOXES TO FILE FINISHED CIRCUIT FOR PAD CHANGE.

CONTENTS — INDEX B

4 PIN ROUND CAN.	6 PIN ROUND CAN.	8 PIN ROUND CAN.	10 PIN ROUND CAN.
TO3 CONNECTION	TO3 CASE.	MINIATURE PR15/H POT CONNECTION	MINIATURE PR15/H POT OUTLINE.
TO66 CONNECTION	TO66 CASE.	SUB MIN PR10/H POT CONNECTION	SUB MIN PR10/H POT OUTLINE.
EDGE CONNECTION	10/WAY 0.1"EDGE CONNECTION	20/WAY 0.1"EDGE CONNECTION	25/WAY APPLE BUS.

COPY SCALES FOR 8 POINT GRID = 0.1"

ROUND CANS & POTENTIOMETERS = 50.0
TRANSISTORS & EDGE CONNECTORS =100.0

(Note 20 way connector extends beyond the box)

CONTENTS — INDEX C

8 PIN DIL.	8 PIN DIL CASE.	14 PIN DIL.	14 PIN DIL CASE.
16 PIN DIL.	16 PIN DIL CASE.	18 PIN DIL.	18 PIN DIL CASE.
20 PIN DIL.	20 PIN DIL CASE.	22 PIN DIL.	22 PIN DIL CASE.
24 PIN DIL.	24 PIN DIL CASE.	40 PIN DIL.	40 PIN DIL CASE.

COPY SCALE FOR 8 POINT GRID = 0.1"

ALL DIL I.C.s AND CASES = 100.00

(Note 40 pin DIL extends beyond the box).

BOARD SIZE

This PCB system uses the 'convenience' 8x8 grid for assembly and connection of your circuit. Other GRIDS may be used but the 8x8 LOCKED GRID should always be restored before ZOOM, PAN, COPY, MOVE and FILE.

The actual size of the grid can be determined at plot time under the PLOT TO SIZE option. For example one 8-point division at BASE PAGE can be plotted as 0.1" (G = 2.54mm) giving same size artwork.

The ROBOGRAPHICS BASE PAGE gives you 32x24 8-point divisions, so by making each division = 0.1" you have an overall area of 3.2"x2.4" (81x60mm).

If you call each BASE PAGE division = 0.2", then by ZOOMing x2 your screen view will still be 3.2"x2.4", each 8-point division still representing 0.1", but the BASE PAGE will represent twice that size, i.e. 6.4"x4.8" (162x121mm).

Similarly you can ZOOM to x4, x8 and beyond, so allowing a BASE PAGE big enough to accommodate your board while always working at an enlarged screen view where each grid division represents 0.1".

The Table below gives the ZOOM FACTOR required and the 'G =' plot value to produce same size artwork for different board sizes. The exact value for G = is the millimetre equivalent to 0.1" (or multiple). G can be entered with up to 8 signicant figures to fine tune the final artwork size.

BOARD SIZE in inches upto	3.2x2.4	6.4x4.8	12.8x9.6
ZOOM FACTOR from BASE PAGE	x1	x2	x4
G = GRID SIZE (for SAME SIZE)	2.54	5.08	10.16

Larger board sizes are of course possible.
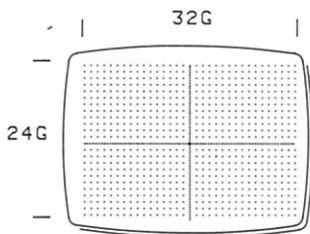
CONNECTIONS

Detailed notes on track thicknesses and layering are given later in this guide but here are a few basic procedures to follow.

Make preliminary connections with a single line, in sections if necessary. Once the design is firmed up you can go over the tracks adding continuity and thickness.

ALWAYS make connections on the GRID. If the 8x8 GRID does not give a convenient connection point you have two options. Either ZOOM to a suitable magnification or use the variable GRID set to 4x4. Before switching the 4x4 GRID on ensure that the origin cursor has been planted on an 8x8 LOCK POINT (e.g. screen centre). Also always revert to the 8x8 GRID before using COPY or ZOOM .

The SKELETON pad circle is designed to allow direct LOCK POINT connections at right angles. When making connections at 45 degrees simply draw to the nearest LOCK POINT. Although the line may not actually touch the circle when viewed in ZOOM this does not matter because it will when you plot it, due to the pen thickness, and when the IDEAL pads are substituted for the SKELETON the connection will be assured.

A common problem is to draw a long track from one side of the board to another. This can be acheived by ZOOMing and drawing the line in sections, moving across the board with PAN. Another, often faster way, is to PLANT A POINT IN ZOOM (see manual) at the start point, then PAGE and ZOOM on the end to PLANT ANOTHER POINT. Return to PAGE and use FIND to locate the start point. Switch the variable GRID on (set to 4x4) and draw the track as near as you can to the end point. Now FIND the end point and draw back to the end of the incomplete track.

MAKE CONNECTIONS ON GRID. GAPS ON DIAGONALS WILL FILL WHEN PLOTTED.

EXAMPLE FINISHED PAD SET
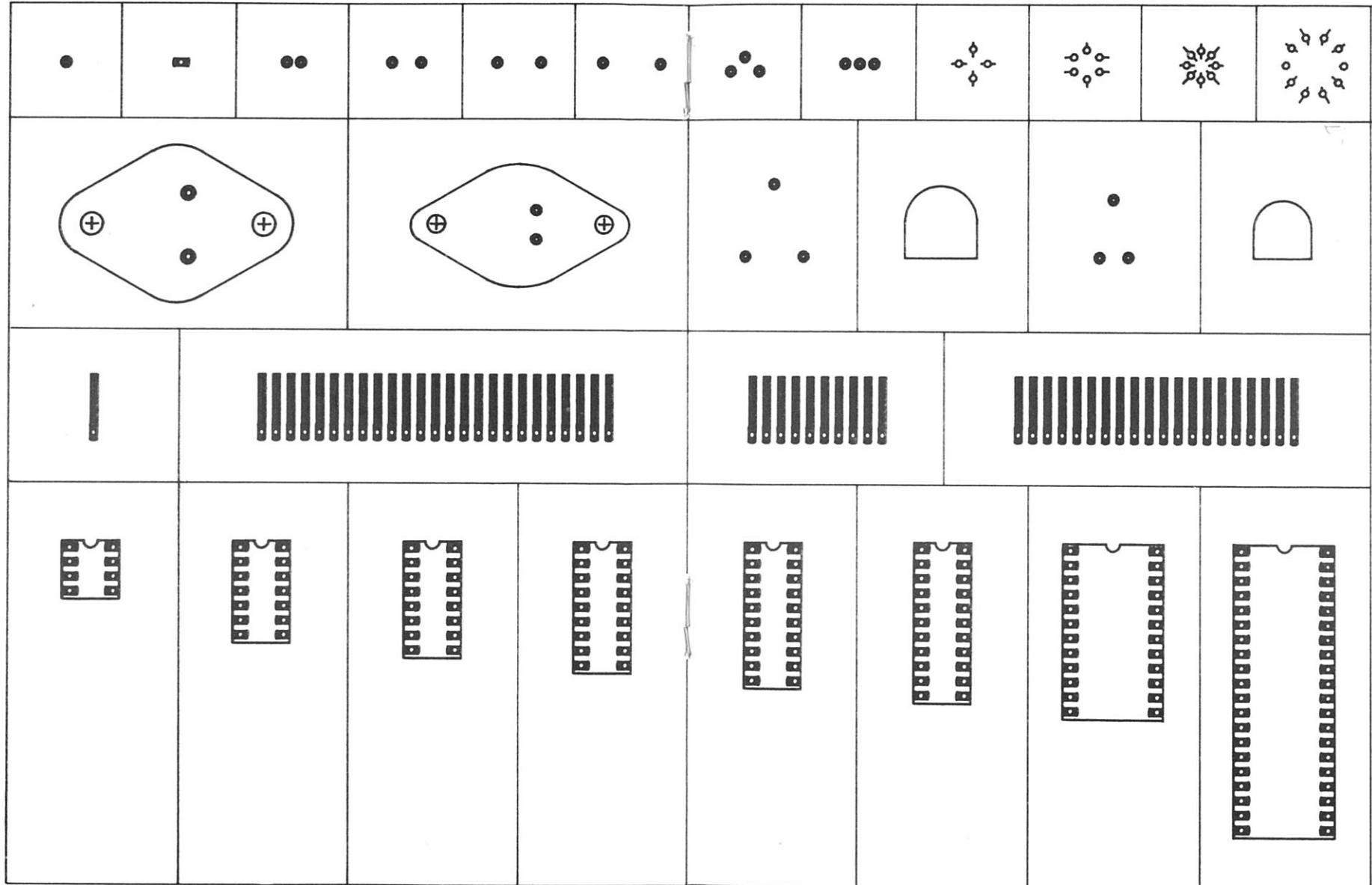
This is a same-size plot with the IDEAL shapes filed in the TARGET boxes.

CUSTOM ELEMENTS

CUSTOM PAD SHAPES
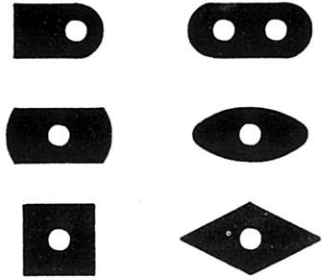
Any shape of pad can be drawn on the system,
saved on one of your own Library disks, and used
as a 'special' in combination with the shapes on
the PCB LIBRARY disk.
You can also replace the IDEAL shapes on the PCB
disk for those of your own design which, when
FILEd in the TARGET boxes, will then appear
throughout the circuit.
When drawing your own pads first COPY the IDEAL
pad supplied (full-size, screen centre) and use
it as a guide to draw over.
Use multiple lines, or circles, to infill the
shape. Remember that the line you draw is in
fact the centre line of the pen stroke, so
you must make allowance for the pen thickness
in order to achieve a neat, fully filled, shape.

SPECIAL PADS
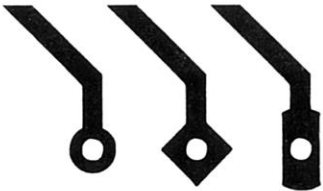
Individual pads can be changed using EXCHANGE.
You can do this at any stage, while the circuit
is in skeleton form or after an update to the
IDEAL shapes. This is particularly useful when
the IDEAL pads, although fine in most places,
are found to be to big or the wrong shape for
a particular position on the circuit.

REPEATED MODULES
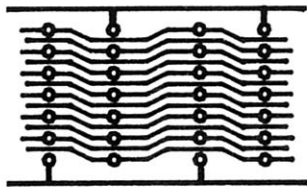
If your design has any repeated elements which
occure frequently, such as the IC bussing shown
here, it is worth taking the time to construct
the element and FILE it on your own Library.
It can then be COPIED and repeated as required
saving a lot of time and effort as well as
ensuring precise repeatability.

# TRACK THICKNESSES

TRACK THICKNESS

There are three ways of producing tracks to
specific widths.

DRAW TRACKS AS SINGLE LINES.

The thickness will be determined by the pen used.
Different track widths can be produced by using
different colours on screen to select different
pens at plot time. For same size artwork track
widths of .1mm through to 1mm are possible
depending on pen availability.

DRAW CLOSE SPACED PARALLEL LINES.

Particularly effective on power and earth buses.
Calculate the spacing required to ensure that the
plotted lines merge to a single line and the
number of lines to produce the required width.

DRAW WITH NIBS 0.

Build up areas of solid copper for edge connectors,
infills between pads, heat sinks, busses etc. with
adjacent areas of NIBS 0. With a .5mm pen this will
plot as solid black irrespective of ZOOM FACTOR
and G = value so it is a very easy technique to
use.

NOTE:PLOT WITH THE PEN SIZES THAT YOU DESIGNED FOR!
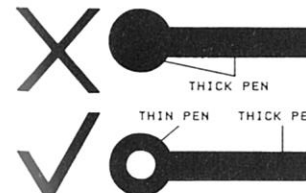
Although you can use a different pen thickness to
plot with than you originally intended this will
also alter the size of the pads if they are drawn
in the same 'colour'. To avoid this the pads
should either be assembled on a seperate layer or
in a different colour to the tracks so they can
be plotted with the correct pen irrespective of
any changes in track thickness that you may wish
to make.

THICK PEN

THIN PEN   THICK PEN

EXAMPLE PROCEDURE

1/.Format a separate LIBRARY disk to 16,16,4 and an ARCHIVE disk for each circuit board and always have pre-formatted blank Library, Archive and Buffer disks available.
IMPORTANT
NEVER USE TWO LIBRARY DISCS WITH THE SAME VOLUME NUMBER TO DEVELOP THE SAME CIRCUIT.

2/.Use the 8x8 LOCKED GRID to maintain size ratios and to facilitate connections.

3/.Determine board dimensions and therefore the 'size' (in 0.1") of a GRID division at BASE PAGE from Table 1. Draw the outline of the board, component side up, including any mounting holes and registration marks. FILE this outline onto your LIBRARY assigned to this project and WIPE.

4/.COPY the outline back onto 8x8 GRID (Full size, centre screen).

5/.Working on the TRACK side of the board first as if viewed from above. COPY and roughly position I.C. pad patterns from LIBRARY ensuring they are at the correct size. For larger boards you may have to ZOOM first. Use ZOOM and MOVE to accurately position patterns if necessary. ERASE the board outline.

6/.FILE layer of pad patterns, WIPE, COPY back onto 8x8 GRID (Full size, centre screen). Draw tracking and any additional pads for 'track-throughs' etc. and then ERASE the COPYied pads before FILEing. This process is only necessary for larger boards and is known as HORIZONTAL layering. This replenishes the memory and as you only file new data at each stage it saves time on screen replay and facilitates future changes.

WARNING

USE THE TARGET BOX PADS TO COPY ONTO YOUR BOARD IF YOU WANT THEM TO UPDATE. OTHERWISE USE THE SKELETON PADS DIRECTLY, THESE WILL NOT UPDATE WHEN YOU CHANGE THE CONTENTS OF THE TARGET BOX.

7/.Once all the layers for a side are finished and FILEd, WIPE. Assemble the complete side ensuring that it is composed entirely of COPYied layers and FILE to the library disk.

8/.If the board is finished go to stage 12. If the board is double sided go to stage 9.

9/.WIPE the screen, COPY the assembled trackside (Full size, centre screen) with COLOUR OVERRIDE for easy differentiation.

10/.Assemble the top side layer of pads and connections as before using the coloured underside pattern as a guide. When complete ERASE the underside COPY and FILE the top side. You can assemble other layers in the same way.

11/.Component overlays can be generated by UTILS LOADing the track side pad layer and EXCHanging the relevant cases for the I.C. pad patterns, FILE into a new Library box. It is possible to produce a drilling guide in a similar way.

12/.Layers on the underside of the board need to be 'FLIPED' over. This is achieved by COPYing each layer (8x8 GRID, full size, centre screen) with 'X FLIP' engaged and FILEing to a new box in the LIBRARY.

13/.PLOT using 'PLOT TO SIZE' and input the 'G =' value for your chosen page size from Table 1. This will give you proofs at same size or by using PLOT LOCKED as large as can be accommodated by the plotter to facilitate checking, if changes are necessary the relevant layer can be UTILS LOADed from library, ammended and FILEd over the original. See Fig. A.

CHANGING THE PAD SHAPES.

FILE CIRCUIT IN SPARE BOXES

When the design has been finalised the SKELETON pads need to be replaced by the IDEAL pads using the GLOBAL UPDATE technique.

1/.UTILS LOAD the layer containing the pads to be updated from LIBRARY.

2/.FILE into one of the spare boxes provided on INDEX A of the PCB disk 228.

LOAD AND FILE IDEAL PADS IN TARGET BOXES.

3/.UTILS LOAD (or DRAW your own) required pad shape on a new workpage.

4/.FILE pad shape into TARGET pad box on INDEX A. Note that two target boxes are provided, one for general pads and one for the I.C. pads which are a different shape. Repeat the process for both pad shapes if required. The screen display is not necessary and can be stopped at any time after it has started with the keyboard space-bar.

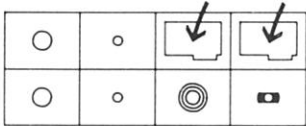LOAD CIRCUIT OUT TO PLOT OR FILE ON ARCHIVE DISK.

5/.LOAD the now updated layer from the spare box again using the space bar to save time. You can let enough of the screen replay to check the update has occured but it is entirely automatic and can be used as many times as required.

6/.FILE onto a fresh ARCHIVE DISK. (If you have changed the pads on different layers to different shapes you must FILE each layer on a different ARCHIVE DISK, otherwise the last shape used will overwrite the others.)
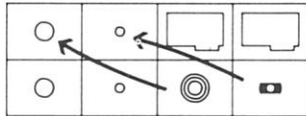
7/.PLOT TO SIZE from the ARCHIVE DISK as required to produce the various pieces of artwork.

LOAD AND FILE SKELETON PADS INTO TARGET BOXES TO RESTORE.

8/.Restore the PCB disk to its original condition by LOADing SKELETON shapes and FILEing into TARGET box ready for the next time you want to use it. Make sure you label them correctly to avoid confusion.
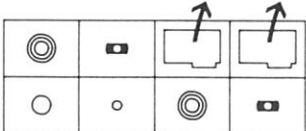
SPECIAL PAD ARRANGEMENTS.
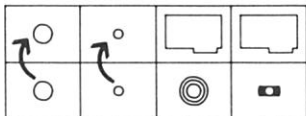
Prepare 'specials' in advance if possible to save time, otherwise follow the procedure below.

If you require an I.C. pattern to contain different shaped pads within itself, as is necessary when tracking between pads, proceed as follows:

UTILS LOAD the relevant standard pattern from the LIBRARY DISK.

ERASE the pads required to be different.

DRAW or COPY from a different source pads as required.

FILE the ammended pattern on your own assigned LIBRARY DISK.

UTILS LOAD the layer containing the pattern to be changed.

EXCHange the old pattern for the new one.

FILE over the original layer on LIBRARY, which will update the complete side.

The new pads now incorporated into the pattern will not be affected by any global changes made by FILEing shapes in the target boxes.

If you have already updated the shapes by FILEing onto PCB disk 228 it will be necessary to repeat the process through to FILEing over the original on ARCHIVE.
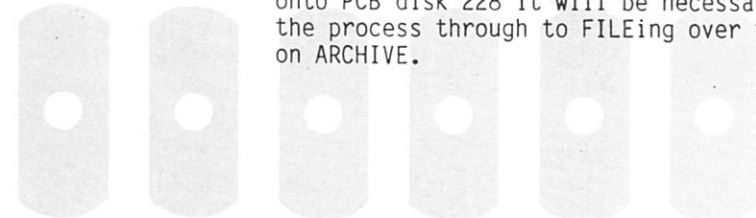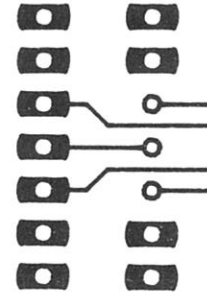
PLOTTING FOR ARTWORK

You can plot your circuit at any stage once it has been FILEd onto a LIBRARY or ARCHIVE disk. With the ROBOPLOT software in the system, first LOAD the circuit, then assign pens using CHANGE COLOUR if required, select PLOT TO SIZE and enter the relevant value for G, finally position the cursor on the paper map and press the RED button to plot.

PLOTTERS

The quality and accuracy of the output depends largely on the plotter used. If you are prepared to photo-reduce the output to produce your final artwork master then very acceptable results can be obtained from even the cheaper plotters.
To produce same size artwork directly the only plotters we recommend are the CALCLOMP 81, HP7470, HP7475 and HP7580/7585. Certain WATANABE plotters have also been found suitable providing they use technical pens for different line thicknesses.

PENS

FOR GOOD RESULTS YOU MUST LOOK AFTER YOUR PENS. Cap all pens when not in use and clean technical pens regularly.

Technical pens are essential for solid, clean-edged lines of consistant width.
Ceramic technical pens can be used but often require overplotting to acheive a suitable density of ink.
Fresh fibre tipped pens can be used and are best for producing over sized artwork for photo-reduction.
Even so it a good idea to overplot the design twice in the same position to ensure a dense black with no holes.

PAPER

The highest contrast comes from plotting on to white cartridge paper. Tracing paper is suitable for producing a positive for direct exposure providing the ink is dense enough. Plastic film and cell require special inks. Some experimentation will show you the best pen/paper combination for your plotter and particular purpose.
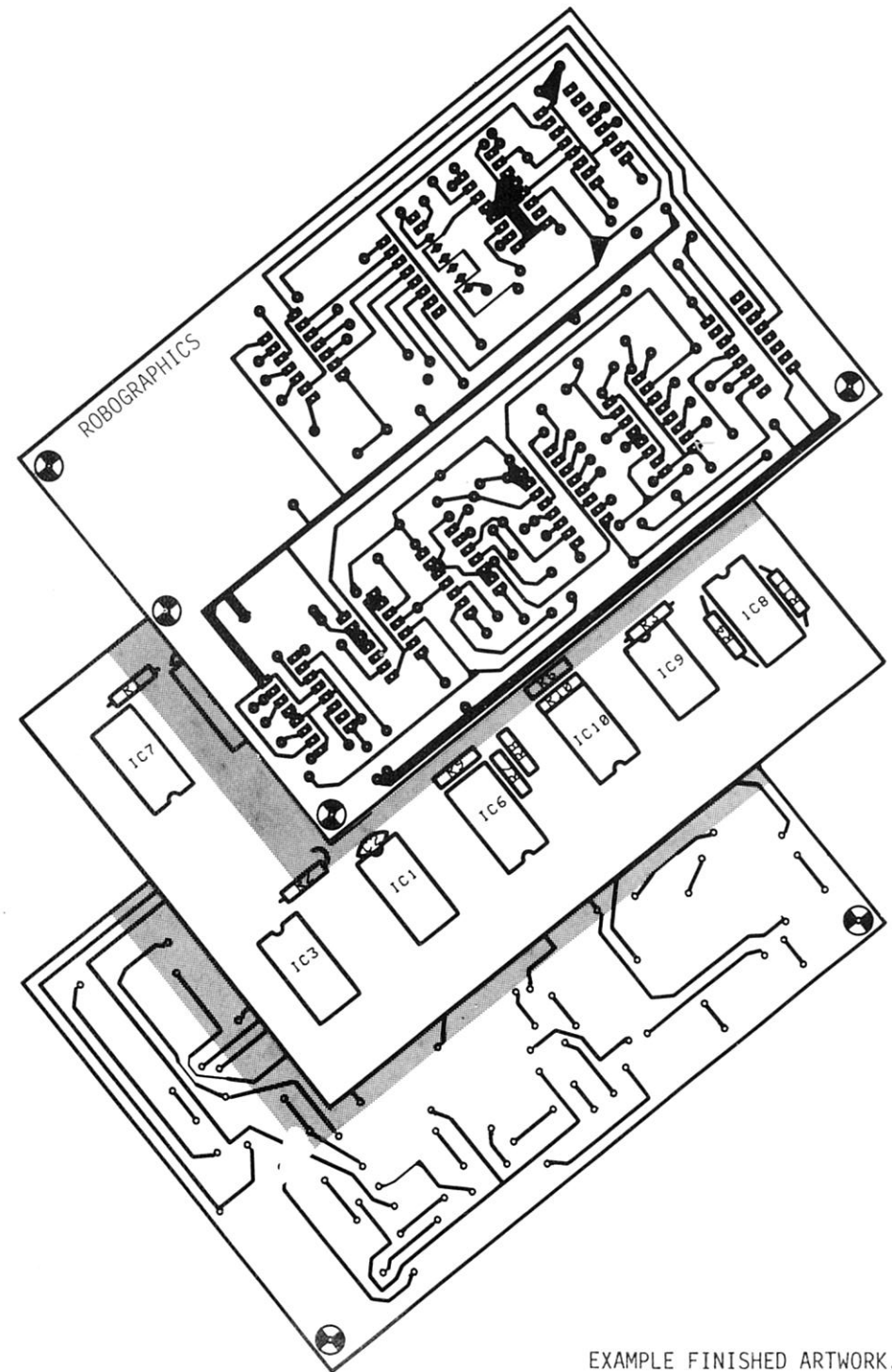
EXAMPLE FINISHED ARTWORK.
(Plotted same size, reduced to fit page).
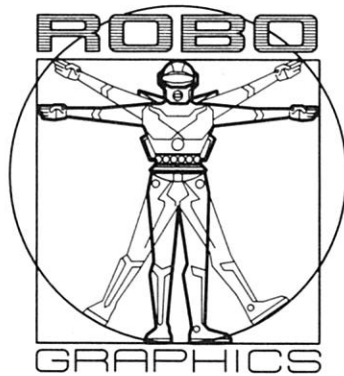
ROBO
GRAPHICS

ROBOCOM LTD., CIL BUILDING, GOODWIN STREET, LONDON N4 3HQ

TELEPHONE 01-263 3388 or 01-263 8585