PCB - Memotech CP/M Printed Circuit Board Drawing Tool
==

This is a program for producing printed circuit board layouts for printing, photo-resist and etching. It requires a Memotech with CP/M, 80 column board and mono monitor (or MEMU in Mono CP/M mode, thanks Andy). The screen attributes used are such that the layout does not show on a colour monitor. I may at some time fix that.

The program is essentially a monochrome paint program, you paint where you want copper, and leave the remainder clear. It has no understanding of the circuit being produced or the required layout, that is entirely up to you.

The program can create layouts at either 60ppi (pixels per inch) or 120ppi. 60ppi was chosen partly because it matches printer capabilities, and partly because it is the minimum resolution required to be able to have IC pads at 0.1 inch spacing, with a track between adjacent pads. The board can have multiple layers. The board size and number of layers is limitted by available memory.

To start the program, type:

PCB filename

where filename is the name of the file for the PCB layout. If the file exists it is loaded. Otherwise you are asked whether you want to create it. When creating a new file you are asked to supply:

* A title for the board.
* The length and width of the board (in multiples of 1/10 inch).
* The number of layers for the board.

Note that these cannot be changed once the file has been created.

Once the layout has been created, you are presented with a view of one of the board layers, with a flashing pixel cursor. The cursor can be moved using the arrow keys. On reaching the edge of the screen the board is scrolled to bring some more into view. If the cursor reaches the edge of the board then it wraps around to the opposite edge. The bottom line of the screen shows the current cursor position (in inches) the current layer, and the editing mode.

The following single letter commands can be used:

M - Enter Move Mode

In move mode the cursor can be moved without affecting the board layout.

D - Enter Draw Mode

In draw mode the curor leaves a trail of copper behind as it moves.

E - Enter Erase Mode

In erase mode the cursor erases any copper it passes over.

N - Next layer

Move on to the next layer of the board. If on the last layer then go back
to the first. Note that Drew or Erase mode (if set) is retained, affecting
the current pixel on the new layer.

S - Save Design

Save the current design from memory onto disk then continue editing. It
is probably a good idea to do this periodically.

X - Save Design and Exit

Writes the current design to file and then exits to CP/M.

Q - Quit

Exits the program without saving the design.

P - Print

Print the design. This command presents a menu of four different print
options.

D - Draft: Print a draft on a DMX80 (or compatible) printer. The draft
is printed at 72ppi, so is not to scale, but is suitable for checking
the layout.

Q - Quality: Attempt to print a correctly scaled high density print on
a DMX80 printer (or compatible). Experience suggests that these prints
are not really adequate for photo-etching.

H - HP Draft: Print a draft on a HP printer using PCL5 (tested on an
HP690C). The layout is scaled up to 300ppi or 600ppi, smoothing diagonal
lines or corners in the process.

T - HP Transparancy: As per HP draft but selecting a high ink mode.
I have had best results using this mode on normal paper and then applying
a UV transparent spray, rather than printing on transparent film.

Function keys - Draw Pads

Places one or more pads:

F1 - Small IC pads with chip axis across board (asks for IC dimensions).
F2 - Small IC pads with chip axis along board (asks for IC dimensions).
F3 - Medium IC pads with chip axis across board (asks for IC dimensions).
F4 - Medium IC pads with chip axis along board (asks for IC dimensions).
F5 - Single large pad at cursor.
F6 - Single very large pad at cursor.
F7 - Single medium pad on all layers.

F8 - Single large pad on all layers.

F - Fill

Fill or erase a region. If the cursor is on a blank pixel when this command
is issued then the surrounding area will be filled up to the surrounding
copper. If the cursor is on a copper pixel when the command is issued then
the surrounding copper is erased. Move mode should be selected before issuing
this command or the result will probably not be as expected. The command will
not always completely fill (or erase) an arbitrary complex area, but it is
good enough that a few fills will normally get the job done.

B - Block Move or Copy

Position the cursor at the top left corner of the block to move or copy, then
press B to start the process. Move the cursor to the bottom right corner of
the block and press M to move or C to copy. Move to the top left corner of the
destination and press either R to completely replace the existing contents of
the destination, or A to add the copper from the source to the destination.

A - All Layers Move or Copy

Works as per the B command, but moves or coppies all layers rather than just
the current one.

Known Bugs
----
1. When inserting IC pads (F1 to F4), the first prompt occasionally has a
spurious control character in the response. Use backspace key to remove this
before entering value.

2. As remarked above,the fill command will not completely fill complex shapes.
It scans up and down from the cursor position to find the top and bottom
boundaries, then left and right on each row to fill. For best results, start
with the cursor in the tallest part of the region to fill.
→

There are a couple of example PCBs included on the disk

e.g. Navcomp1.pcb is a minimal Z80 microcontroller, with Z80, three
ROM/SRAM chips at 0x0000, 0x4000 and 0x8000, oscillator and memory
decode.