How I built my own
...and details to allow you do the same.

...on-line since 1997 !

by Grant Searle

Last update: 10th March 2013

Please note that you are NOT allowed to reproduce any of this page elsewhere on the Web without my permission.
MOST IMAGES ON THIS PAGE ARE MY OWN WORK

Update 7th March 2013 - After many years, a page overhaul is underway.
Update 9th March 2013 - Track layouts now available to download, 2732 EPROM PCB mod plus other page changes. See below.
Update 10th March 2013 - Heatsink details added.

CONTENTS

INTRODUCTION

The Jupiter Ace has a relatively straightforward design which makes debugging of any hardware faults straightforward. If consists of the following modules:
The timing for ALL parts of the circuit
The CPU/ROM/RAM area
The display area

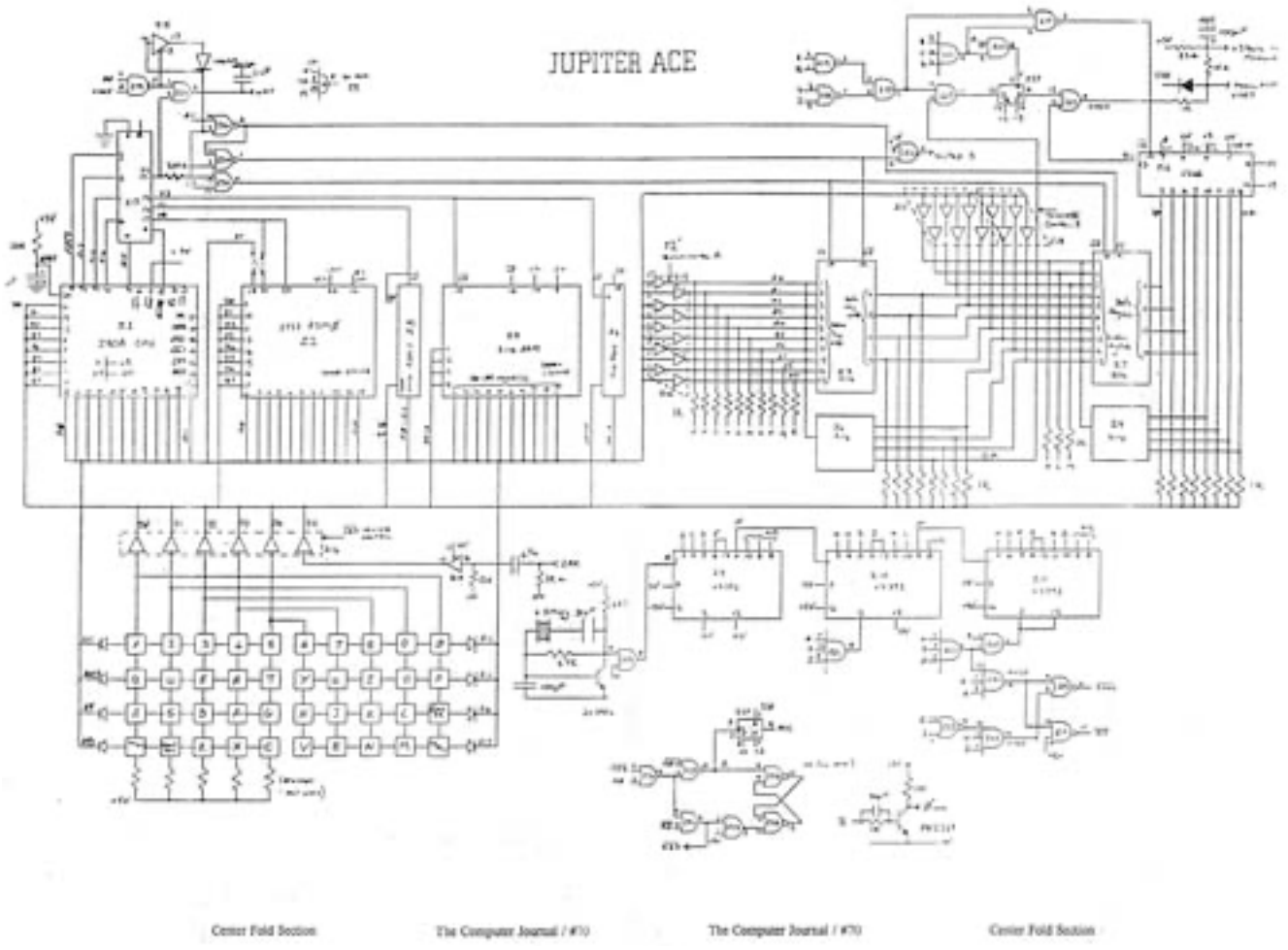
I must point out this is **not** a job for the absolute beginner. You will need to blow your own EPROM image and it will probably be a distinct advantage to have access to an oscilloscope in case it doesn't work. I can assure you the circuit supplied here DOES work without any modifications.

I have, however made the following modifications for my own version:

1. I used a 6264 SRAM to make the program memory 5K instead of 1K. The other 3K is not wasted - it is also available within the memory map.
2. The original Jupiter Ace used two 4K ROMS. I have used a single 8K ROM.
3. The Ace uses two 2114s for the display RAM and two 2114s for the character RAM. I actually had a large amount of 8K x 8 RAMs in my spares box so, although wasteful, works out cheaper and construction is slightly easier. The 6116 2K x 8 RAM chips are also suitable alternatives. Tie any unused address lines to one of the supply rails.

CIRCUIT DIAGRAM

This is a scan from "The Computer Journal", number 70, and appears to be the same as in "Ace User" number 3.
Special thanks to Peter Liebert-Adelt for supplying me with photocopies of this and other information many years ago.



Click here to download the ACE schematics

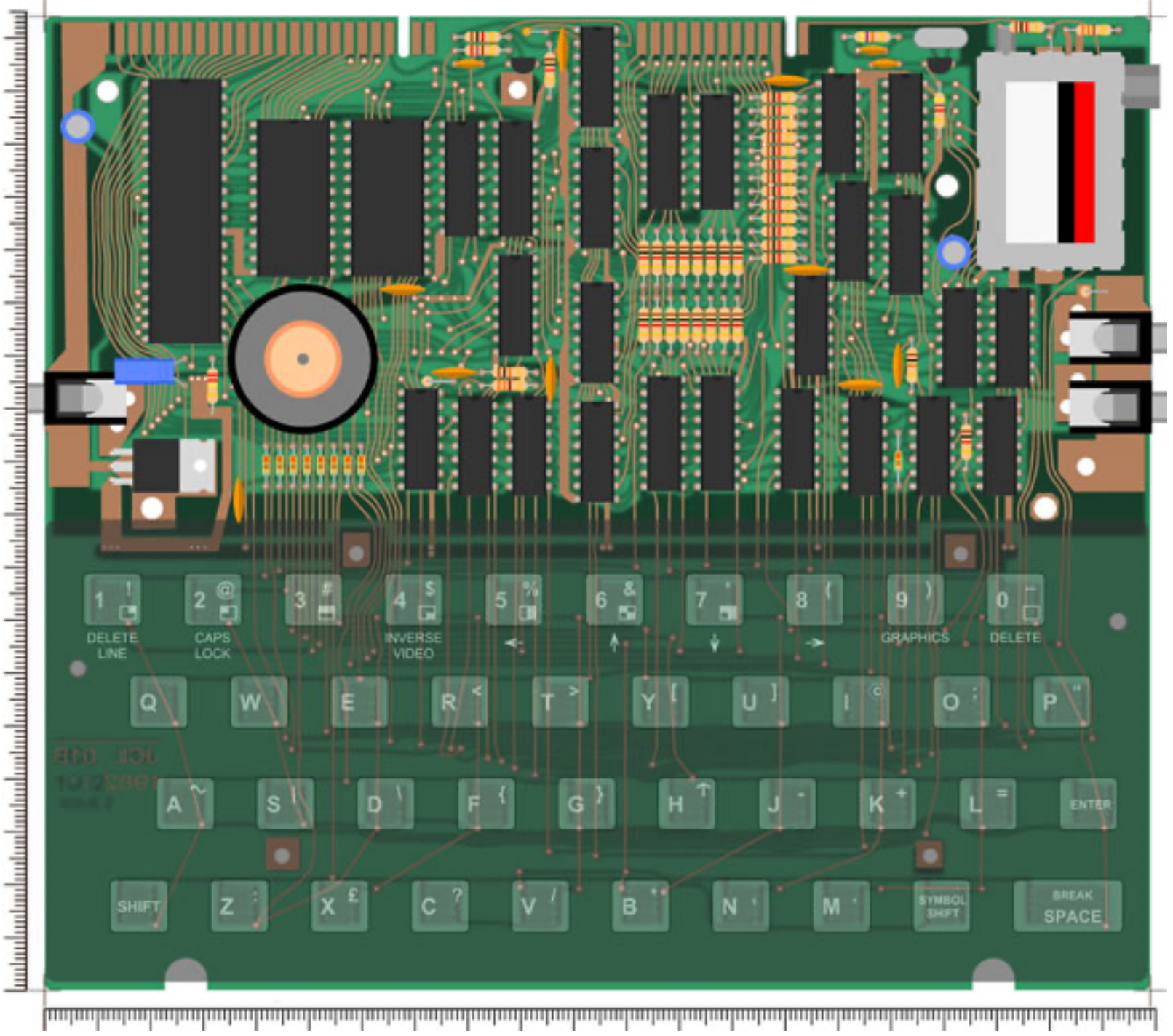
There are areas of the schematic that are dificult to read and some areas that are missing some detail, so I recommend you look at Bodo Wenzel's redrawn schematics using Orcad here
I will annotate the schematic with missing/corrected information when I get time.

CONSTRUCTION ON AN ETCHED PCB

Several construction techniques can be used. You could use the accurate track layouts that I provide on this page or you could construct the design on stripboard.

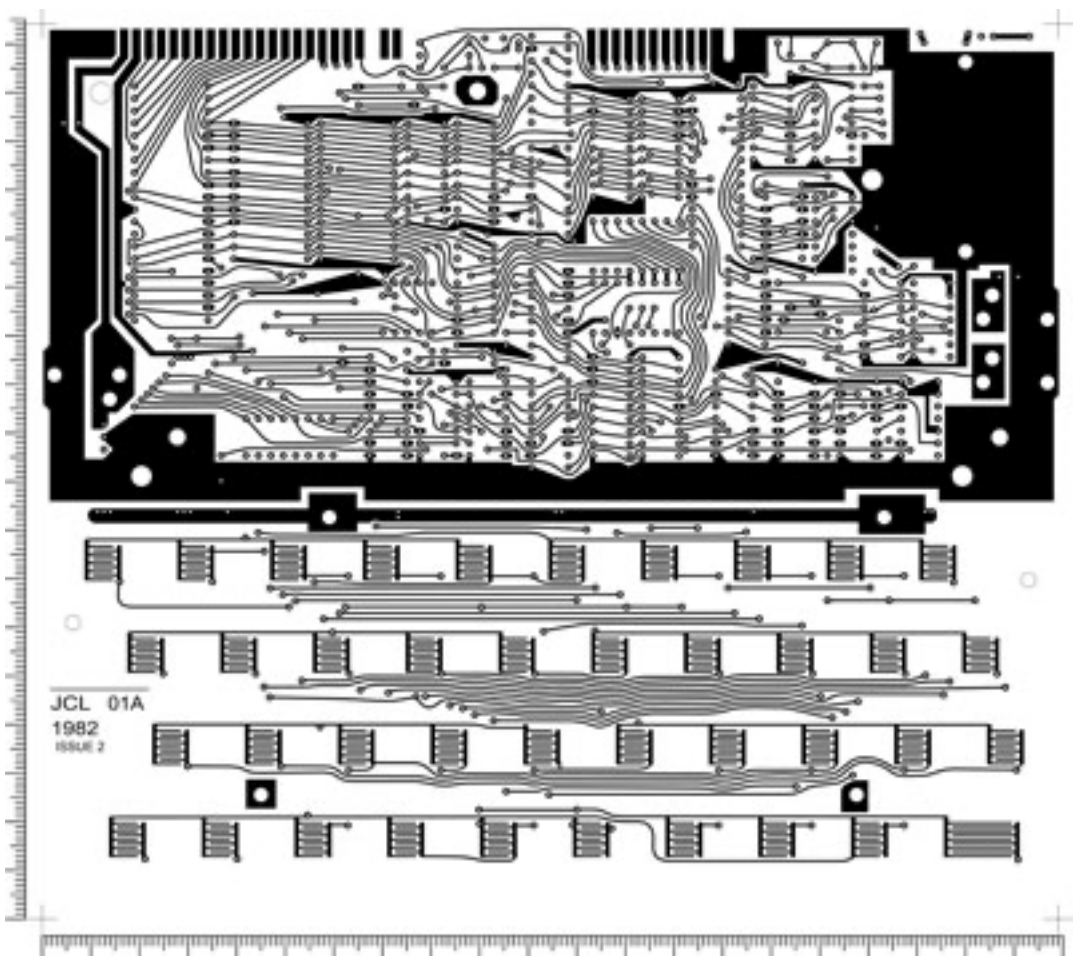
Click here or on the image below to download the PDF.
Note: If that link fails, you can also get it from my SkyDrive here.

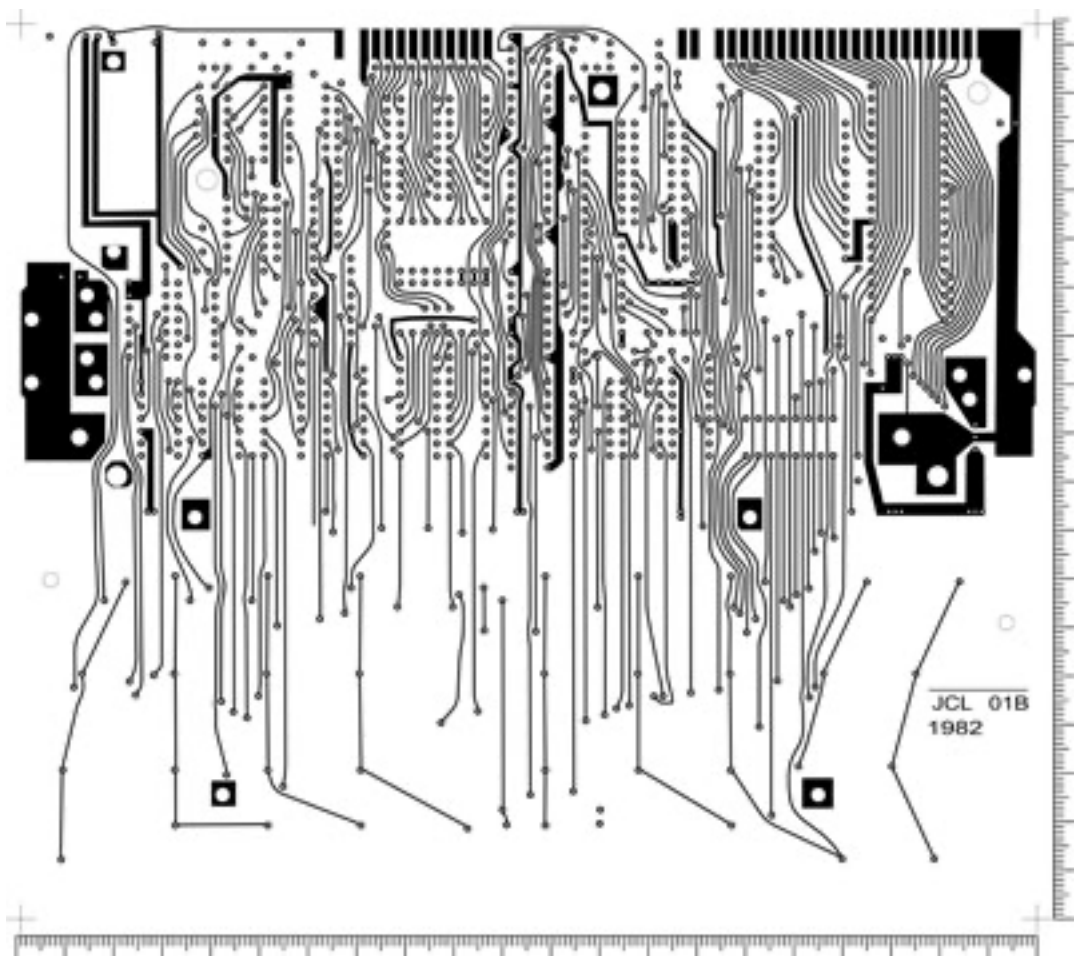
This PDF contains the composite image shown below, keyboard overlay, front and back PCB tracks and front and back REVERSED PCB tracks (for acetate photo-printing).

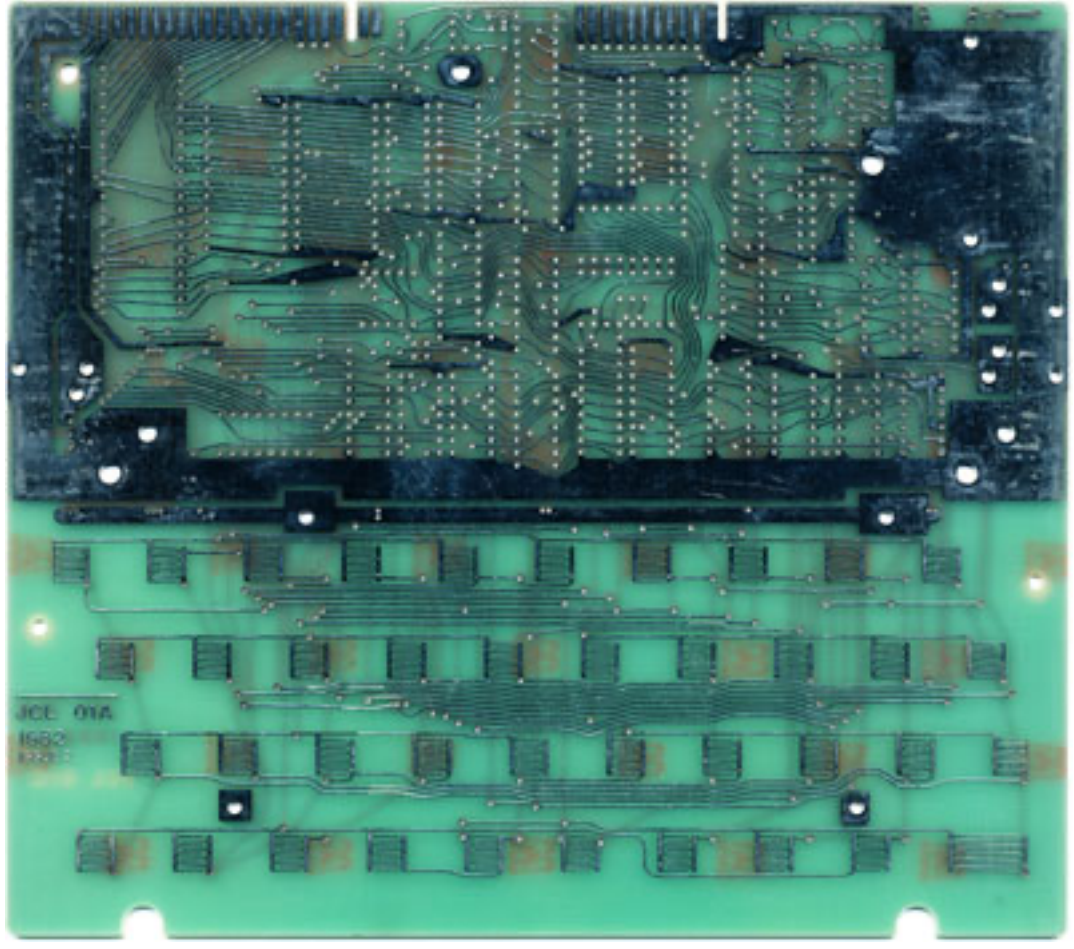


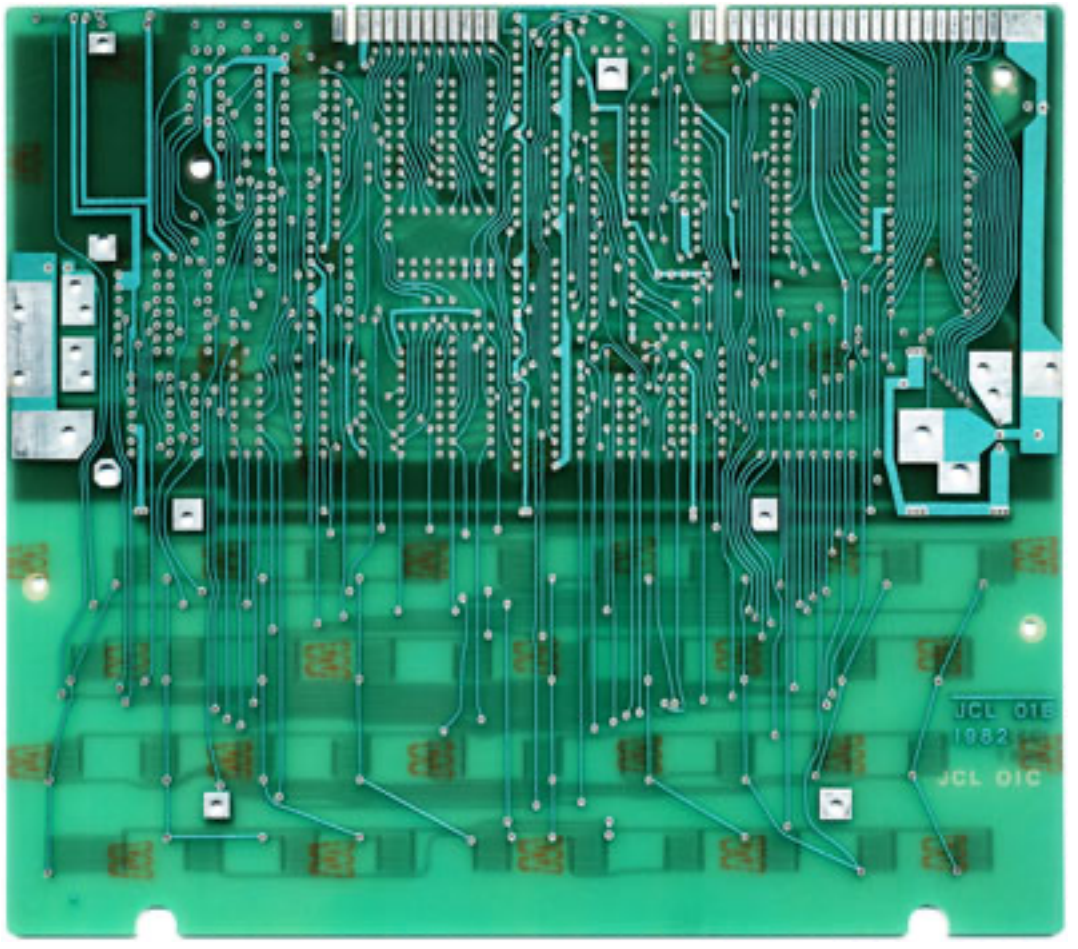
The techniques for constructing the PCB are identical to those on my ZX80 page. Please go here for details.
The tracks will fit on to A4 acetate sheets so can be handled by any laser printer.

I have spent a long time making the layouts as accurate as possible (within a fraction of a mm).
I was provided with very high resolution scans of an issue 2 PCB and after ensuring accurate sizing and overlaying (using Photoshop to stretch and skew as needed) of the two sides I was then able to overlay my tracks on top of the pictures.
Some of the "anomalies" with the original layout have remained, for faithful accuracy to the original. These include some sharp corners on a couple of the tracks and also some of the pads are different shape (the "DIL" package pad instead of round).
There is also one track on the reverse of the original PCB that doesn't go anywhere (only connected at one end). For accuracy, I have included it in my re-drawn version.
Any remaining anomalies such as varying track width or wobbly lines due to the original hand-drafting have been corrected in my version.



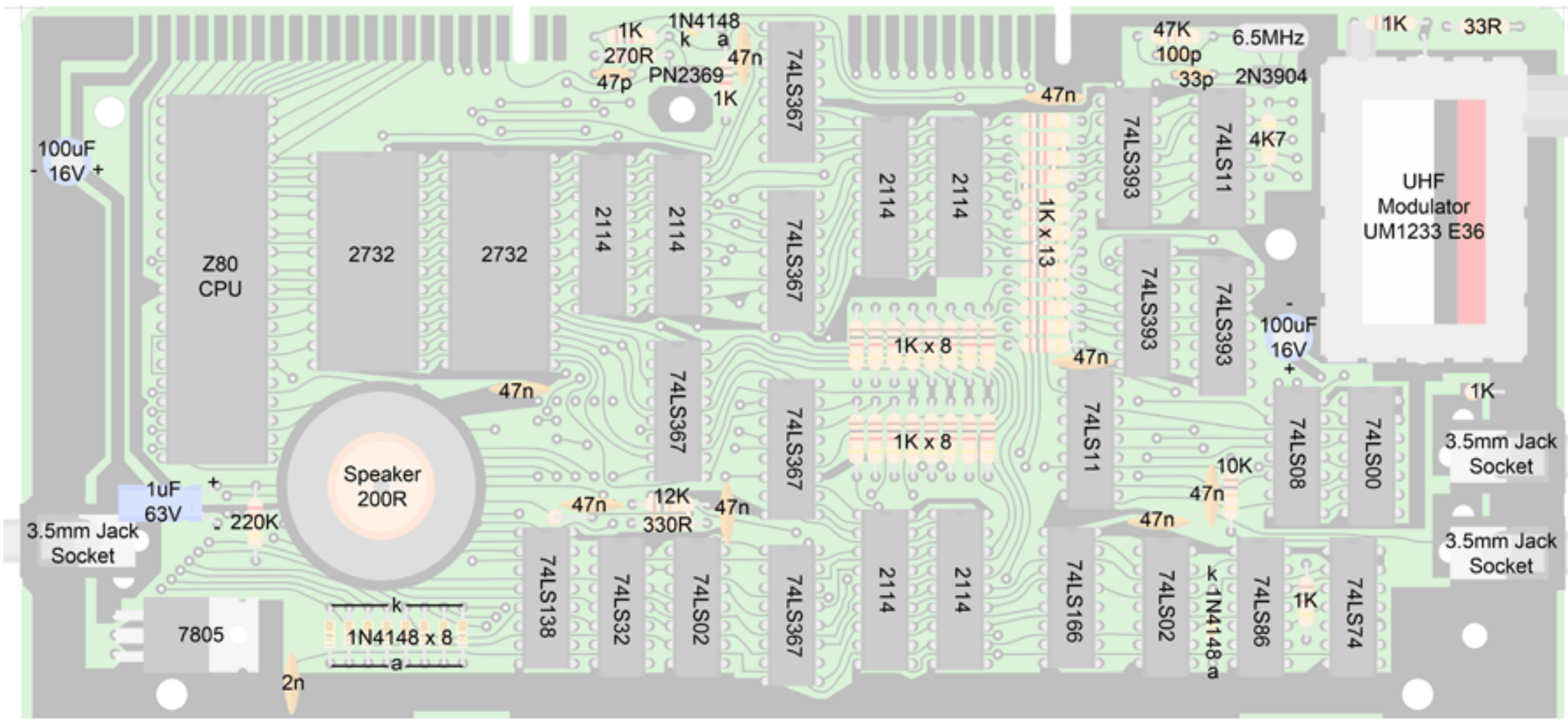






COMPONENT LAYOUT

All components are mounted on the top-half of the board.
Position and values of all components to be fitted is shown here.



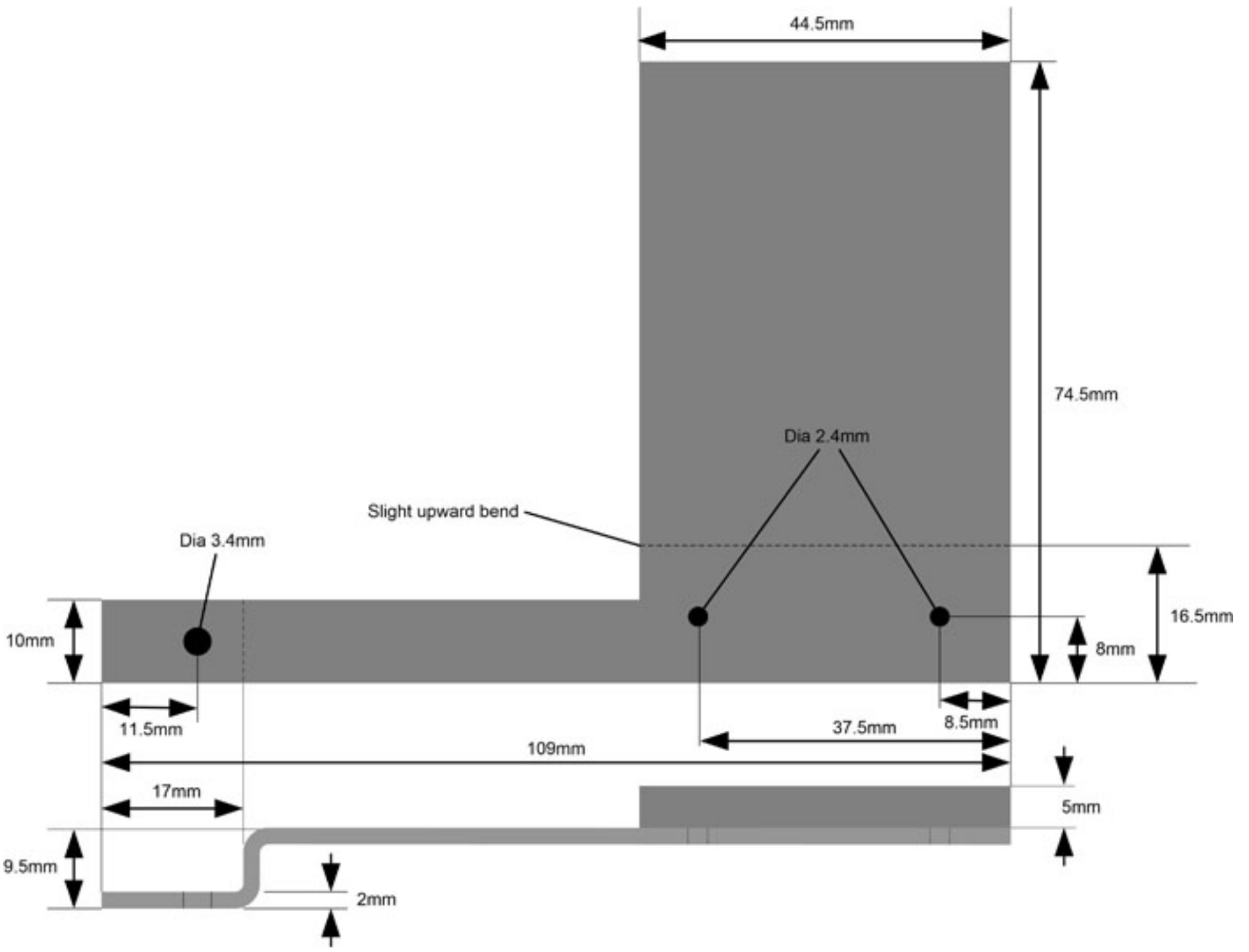
All chips are shown with pin 1 top-left (notch at the top).
Other components are mounted as shown. Where the orientation of the component is important, the relevant connections are shown.

HEATSINK

The regulator must be mounted on a heatsink.

The heatsink details from an original Jupiter Ace is shown here. It is made from 2mm aluminium sheet and is placed between the regulator and the PCB. No insulating kit is required.

Details of this heatsink are shown here:

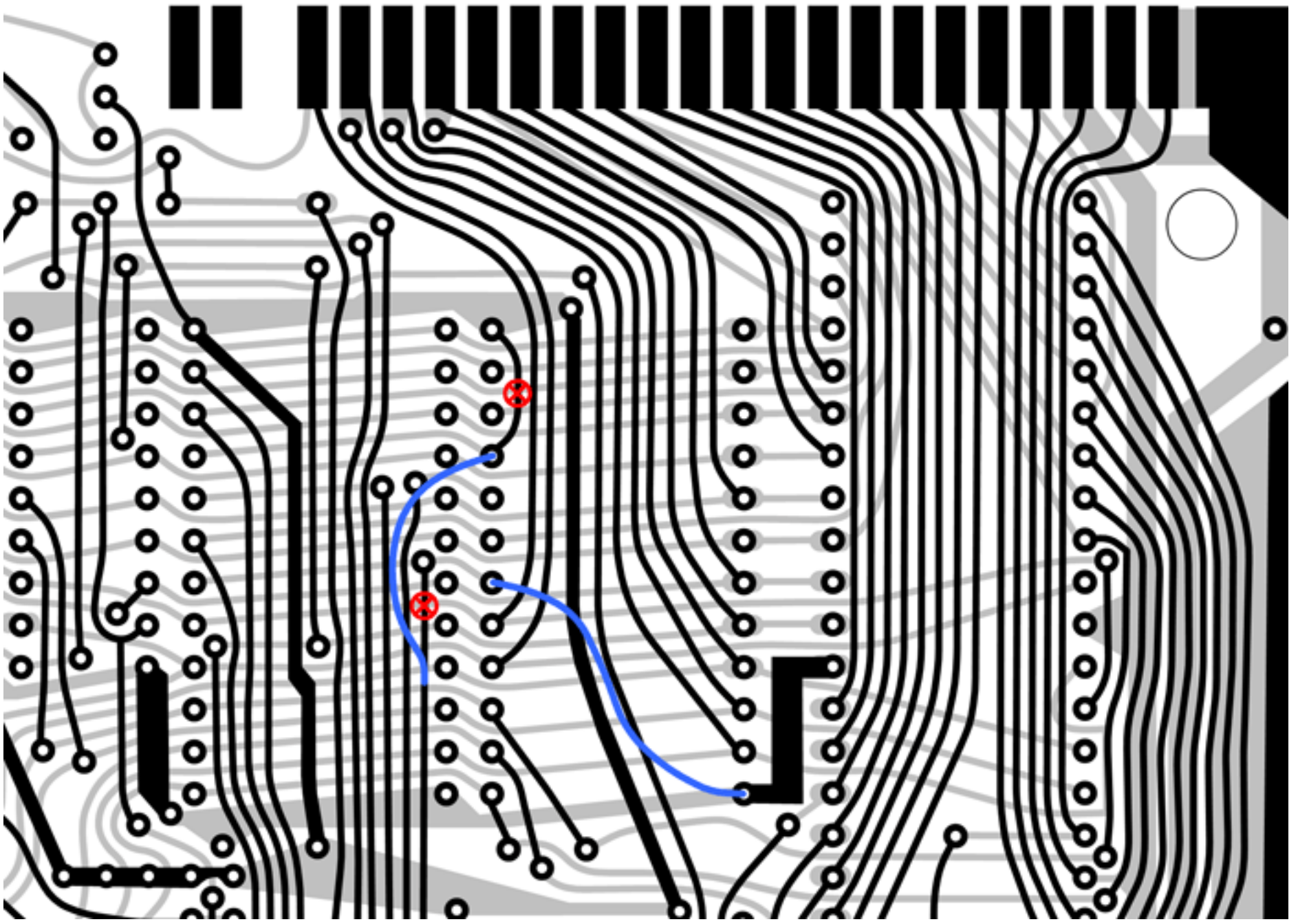


The two 2.4mm holes serve no purpose whatsoever, so can be omitted.

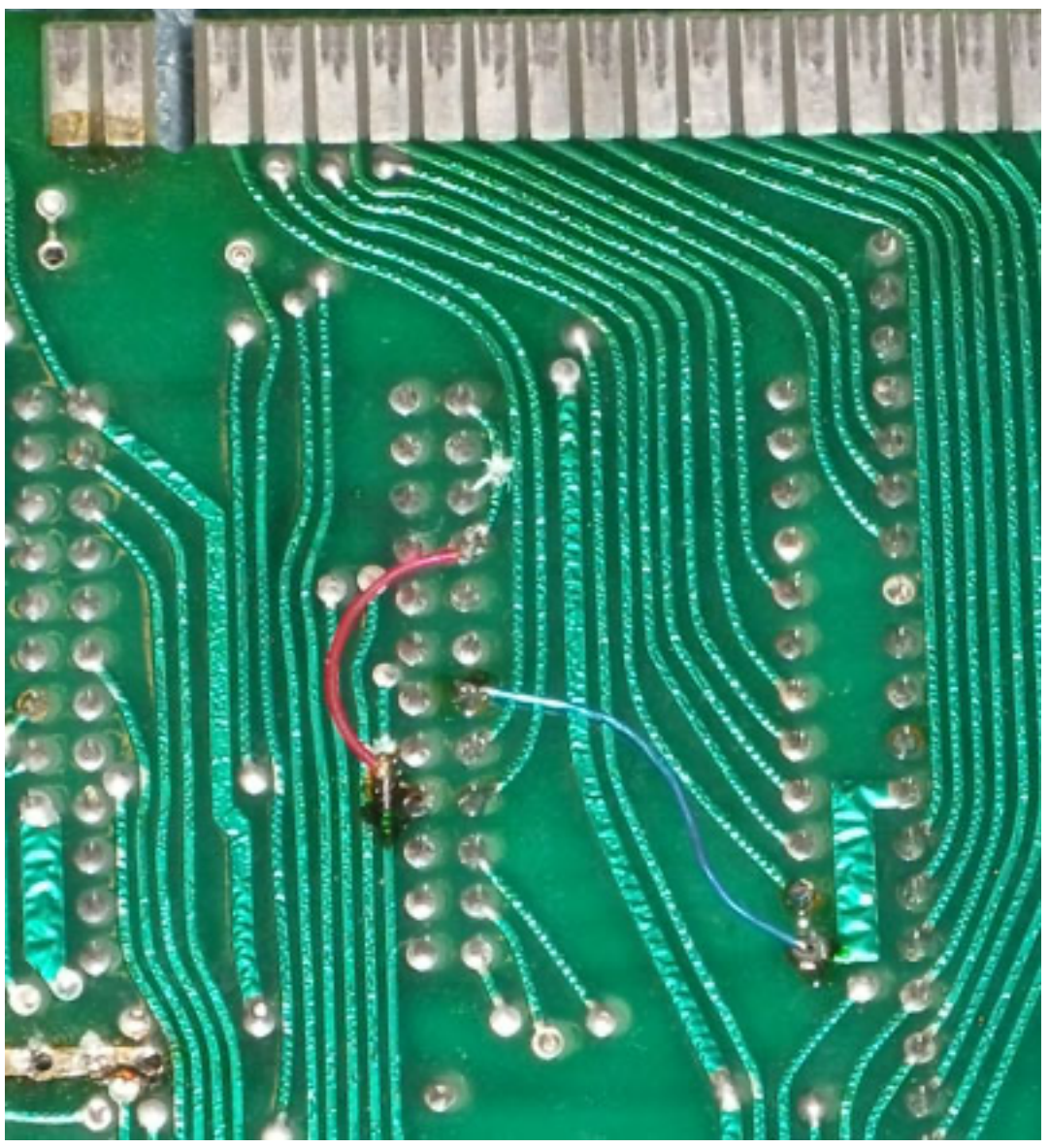
PCB MODIFICATION FOR 2732 EPROMS

The Jupiter Ace layout needs a small modification if using 2732 EPROMs instead of the manufacturer- supplied ROMs.
This is due to a slightly different pinout on pins 21 and 18.

To make the modification, please refer to the diagram below.
This is looking at the underside of the PCB. Track cuts that are needed are shown by the red circles. Connections then need to be made as shown in blue.

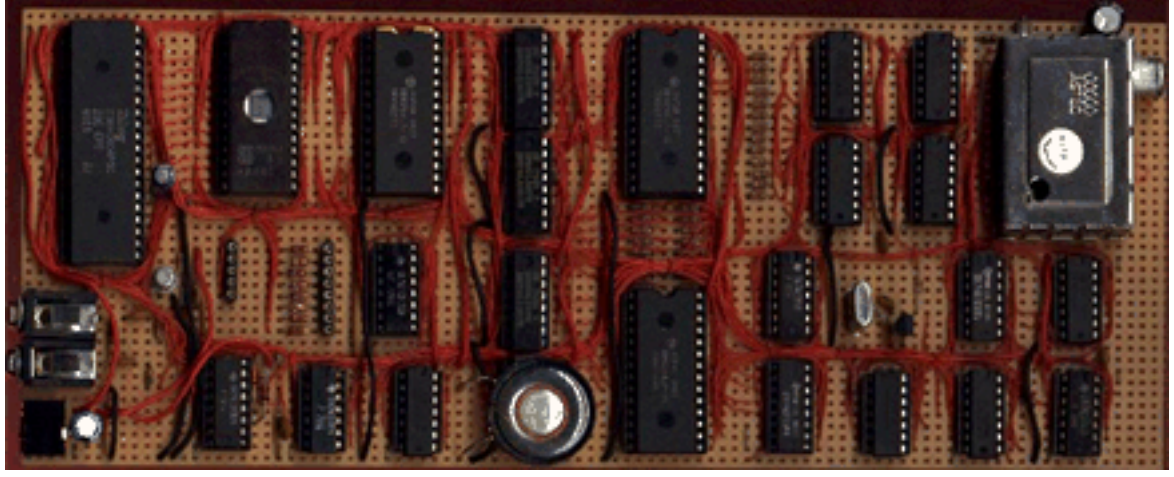


Here is the modification made to an actual Jupiter Ace...



CONSTRUCTION ON STRIPBOARD

As an alternative to the etched PCB, the complete computer could be built on two pieces of perforated stripboard - one to hold the keyboard and one to hold the rest of the computer. All digital connections are made using wire-wrap wire as this is a lot thinner than conventional hook-up wire so the data and address bundles can be tied to form a neat layout. The positioning of the ICs in relation to each other matches the positioning in the original Ace.


Click picture to view larger image


Click picture to view larger image

I recommend printing the circuit out on an A3 sheets if possible. The scan is not perfect but hopefully you should be able to work out any areas which aren't very clear.

Begin construction by positioning the IC sockets (see my pictures below for guidance). Next I recommend wiring the data and address buses for the top half of the circuit. As each wire is soldered it, mark it off on the circuit diagram. This helps avoid any errors resulting from missing connections. Wire in the logic ICs, resistors, capacitors and diodes. As virtually all pins on every IC are to be connected, any errors made will become obvious as more connections are made. For initial testing, there is no need to wire the keyboard into the circuit.

Once all components are soldered and without any ICs inserted into the sockets, connect a continuity tester between the power supply pins. If a short exists then check the underside of the board for any missing track breaks or flakes of copper between tracks.

The main board can be tested before the keyboard is connected if desired. If all is well then wire in the keyboard if not already done.

POWERING UP

Connect a power supply (current regulated to 500mA if possible).

Check the +5V and 0V connections on each IC socket. Turn off the supply and insert the ICs.

Turn on the power supply and the television or monitor. If the circuit is working then expect a power consumption of around 280mA. A working circuit should show an small white square in the bottom left of the screen.

PARTS LIST

No Component type
Z0 Z80 (cpu)
Z1,Z2 2732 (4K x 8 eprom) I actually used 2764s in my design as I had some in my spares box
Z3-Z4 2114 (1K x 4 sram) If not using the PCB layout then I recommend using one 6264 to allow 5K RAM
Z5-Z6 2114 (1K x 4 sram) I recommend using one cheaper 6116 (2k x 8) if not using the PCB layout
Z7-Z8 2114 (1K x 4 sram) I recommend using one cheaper 6116 (2k x 8) if not using the PCB layout
Z9-Z11 74LS393
Z14-17 74LS367
Z19 74LS00
Z20 74LS08
Z21,22 74LS11
Z23 74LS86
Z24,25 74LS02
Z26 74LS32
Z27 74LS74
Z28 74LS166
Z29 74LS138
U27 7805 (+5V reg)
X1 6.5 MHz xtal
Q1 2N2369, PN2369 etc
Q2 2N3904
D1-11 1N4148
C1 39p ceramic
C2 100p ceramic
C3 47n
C4 47p ceramic
C5 -

C6 47n
C7 1u electrolytic
C8 2n2
C9 100u electrolytic
C10 100n
C11,12 1u tantalum
C13-21 100n
R1 4k7
R2 47k
R3 1k5
R4 1k5
R5 12k
R6 1k
R7 1k8
R8 270R
R9 220R
R10 10k
R11-23 1k
R24 10k
R25 220k
R26 330

RN1,2 1k x 8
L1 RF choke
LS1 200R (approx) speaker My design used a speaker salvaged from an old Spectrum board!
S1-40 SPST switch (keyboard)
JP1 3.5mm socket (audio out)
JP2 3.5mm socket (power in)
JP4 3.5mm socket (audio in)

RF Modulator

ROM IMAGE

The ROM image is 8K long (2764 EPROM) or two 4K images (eg. 2732 EPROM)
Click here to download the binary image.
(In Internet Explorer, click using the right-hand mousebutton then select save location from the menu.)

KEYBOARD CAPS / OVERLAY

If constructing the PCB version, the PDF contains the keyboard layout. The original has a rubber keyboard, similar to the ZX Spectrum.
However, you could construct a membrane keyboard (similar to the ZX80 or ZX81) as the PCB is suited for this type of keyboard.
This would be constructed the same way as I did my ZX80 keyboard. Please see here for details.

PICTURES OF MY COMPLETED STRIPBOARD VERSION

Front of PCB
Back of PCB
Keyboard

ACKNOWLEDGEMENTS

Peter Liebert-Adelt - I couldn't have made it without the extensive information sent to me - thanks!

SOME OF MY OTHER PAGES

Build your own ZX80 - my page showing you how to build this old micro
  |__ ZX80 to ZX81 conversion - build the NMI generator needed to convert the ZX80 circuit into a ZX81
  |__ ZX80 software - Type in a Space Invaders game into the ZX80

Build your own Jupiter Ace - my page showing you how to build this old micro
Build your own UK101 - my page showing you how to build a greatly simplified version of this old micro

Pong - Pictures of my build of the Atari classic arcade game
My Machines - My collection of classic 80's micros

I hope this page has been useful.

Grant.

To contact me, my current eMail address can be found here. Please note that this address may change to avoid spam.