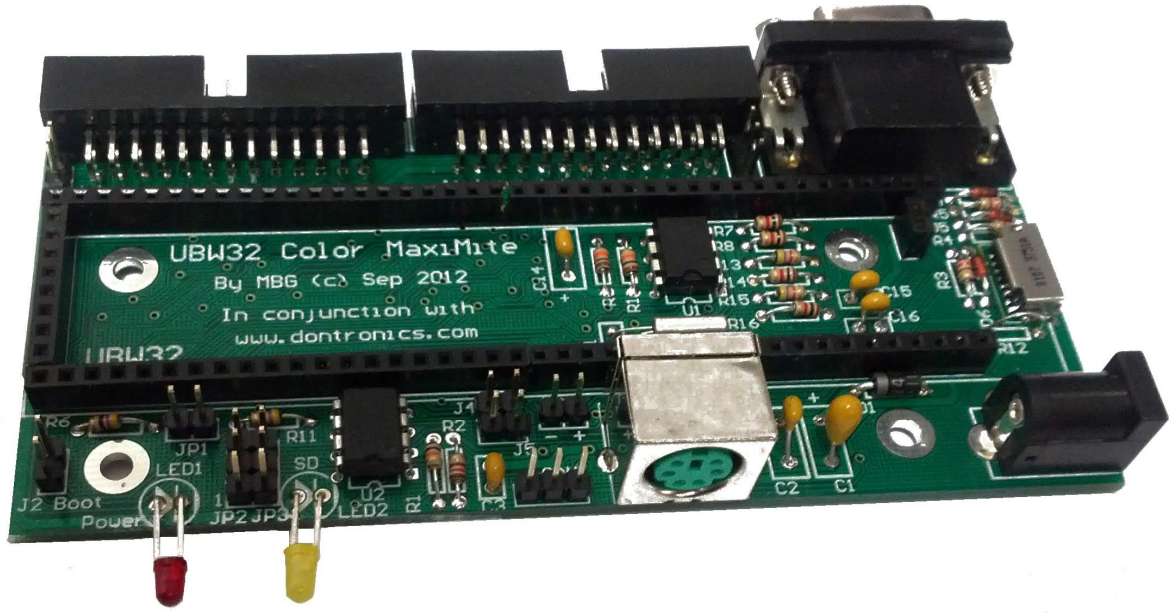
# UBW32 - MCC

(Maximite Colour Computer for the UBW32)



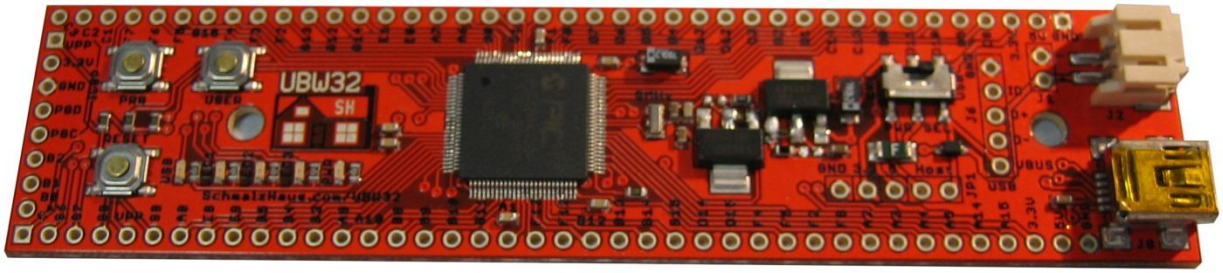

## Preamble

When Geoff Graham first introduced his Maximite Colour Computer (MCC) to the world he announced that it was based on the same hardware as the UBW32(MX795) Pic32 Computer, in fact I believe that all his prototyping for his design was done using the UBW32. See http://geoffg.net/maximite.html

He also released to the public schematics and information on how to convert a UBW32 into an MCC. This project is almost entirely based on that design and many thanks and kudos must go to Geoff for another great Aussie design idea.

When I started out I was going to do a full MCC which includes the Arduino header footprint until I realised that this would be a reasonably large sized PCB and my thought process was that people don't buy a UBW32 with the intention of storing it in a shoe-box. They want to mount it in a match box instead. i.e. They want it to be as small as possible. So I laid out all of the parts and cramped them together with minimal spacing and measured the board size. I then looked for the smallest case that was also cheap that would fit. I came up with a neat little case from Takachi Electric which is available from RS Components (SW-120B P/N 373-2441), then pushed the components even more to make it fit.

# UBW32




The UBW32 is a 4.4" x 1" (110mm x 25mm) PCB based on the Pic32 (MX795L) processor, designed by Brian Schmaltz and available for under $40Aus from Dontronics.com and Sparkfun.com. Full details can be found at http://www.schmalzhaus.com/UBW32/

The UBW32 natively comes with its own pre-installed operating system and bootloader and first has to be loaded with Geoff's MMBASIC. There are 2 ways to run the UBW32 as a MCC.

**Method 1.**

Using the native UBW bootloader and HIDBootloader.EXE, available from http://www.schmalzhaus.com/UBW32/ flash the latest MMBASIC.Hex file into the UBW32.

This has 2 minor disadvantages:

Pin B5, or maximite Pin(1), of the UBW32 will not be available for use as the native bootloader configures this as a VBusOn signal and cannot be over ridden.

Also to flash new firmware into the UBW32 would require pressing the boot button on the UBW32 itself (which would be inside the case).

**Method 2.**

If access is available to a PicKit3 programmer then the Maximite firmware and bootloader can be programmed into the UBW32 which effectively will turn the UBW into a fully functional MCC. A version of firmware with the bootloader included is available from Geoff in his MCC construction pack.

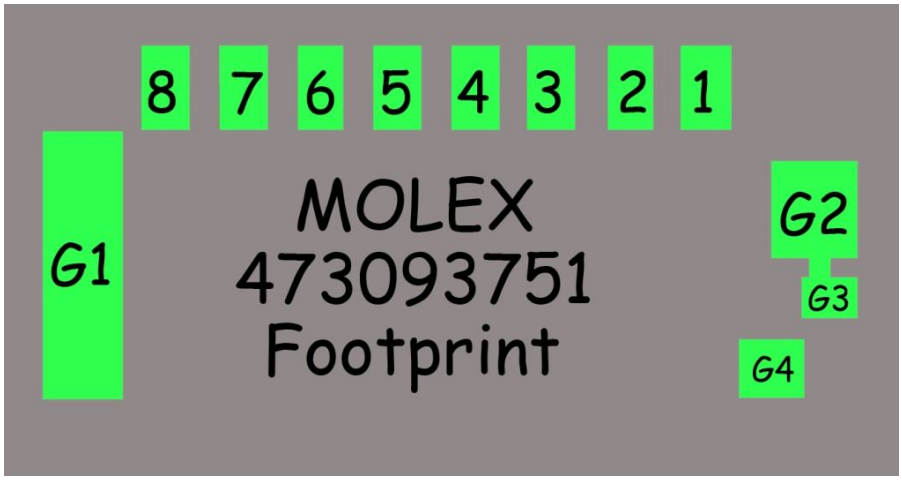
After programming the MCC bootloader into the UBW32 the native (onboard) Boot switch will no longer be active. A switch across J2 on the UBW32-MCC PCB will become the new boot button. An advantage of this method is that Pin B5/Maximite Pin(1) will become available for use by MMBasic.

## Construction

Construction of the UBW32-MCC is very easy with the one exception of the Surface mount micoSD connector. This will require a certain amount of skill as well as a clean, sharp-pointed iron and good lighting to perform. It should be soldered first as access will be near impossible if the other components are soldered first. A useful tool to aid soldering this part is some liquid flux. I purchased mine from ebay for about $2 including postage, just paint this on the areas to solder and the solder will flow easily.

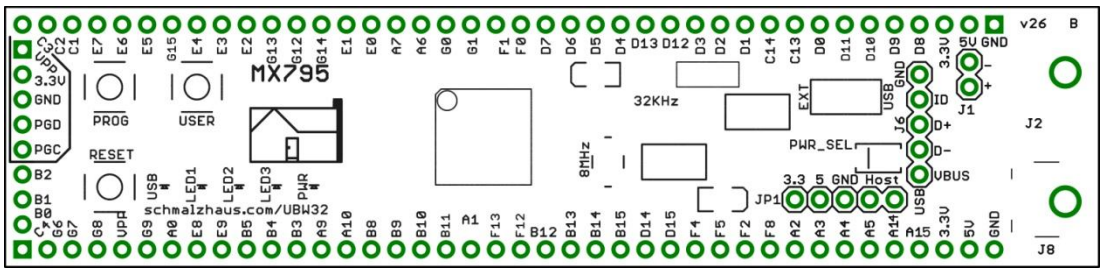
**MicroSD.**

The microSD connector is the only difficult part to solder, referring to the following layout of the footprint you will see the pads for the switch (for card inserted) are G3 and G4, as G3 is very difficult to solder, due to the metalwork of the uSD case, I recommend soldering the Pin for G3 to the casing (which is GND) before positioning the connector for soldering to the PCB.




Next, manually align the uSD connector so that it fits perfectly to the outline on the PCB, (when viewed from above) then whilst holding it in place, carefully solder the G1 pad and check the alignment of the 8 electrical pins and the other pins. Reheat the joint and re-align if not satisfied. Solder the G2 pad on the other side and recheck before soldering the 8 data pins.

Lastly touch the G4 pad with the point of the iron and hit with a bit of solder. The connector should be solid and all pads should be making a good connection and not shorting with neighboring pins, a good magnifying glass is a must here. You can test the function of the switch (G3 and G4) by using a meter to measure between GND and the pads for D5 and D6 of the UBW32 connector (see layout below). With a microSD inserted you should see GND appear at D5 and D6.



**Other Components.**

After you are 100% happy with the uSD connector solder all of the resistors, diodes and capacitors making sure that there is no possibility of solder shorts to nearby pads or the shielding `hash', If satisfied move onto the larger connectors and jumper pins leaving the female headers for the UBW32 board till last.

**Note!** R2 (120R) is only needed if you wish to have termination on the CAN bus. Fit this as you require.

**UBW32 Socket** (Marked in **RED** on the Overlay), 2x40pin, 1x8pin & 1x2pin L/P Female sockets.

If you intend to use the case I suggest the female headers **must** be low profile to accommodate the overall height and allow the lid to close.

I chose headers that I purchased from http://semtronics.com.au that are manufactured by Oupiin.com (Oupiin offer free samples and have many re-sellers)

Suitable options for these are:

Oupiin
http://www.oupiin.com/sampleDetail.asp?id=567

3M from Mouser (20 pin strips)
http://au.mouser.com/ProductDetail/3M/929870-01-20-RA/?qs=%2fha2pyFadujRJI8DsGJUZVv%252b%2fMngCvsbITP7xV71iKbeOLSZm5jmeQ%3d%3d

Or a standard (non Machine pin) IC socket cut to suit similar to this one.
http://au.element14.com/fci/dilb40p-223tlf/dip-socket-40-pos-through-hole/dp/1924566

If you find the low profile female sockets hard to obtain, standard 8.5mm female sockets will work but the white (J2) header will have to be removed from the UBW32 to fit the suggested case.

All of the above information is assuming that you used standard 0.025" square pins soldered onto the underside of the UBW32 along both sides (40pins each) the Lefthand side (8pins) and for J1 (2pins).

The battery backup for the RTC can be a small coin battery with short flying leads cheaply available from ebay (mine was under $3 inc postage) similar to this one. (the connector may need to be replaced)

# Parts List for UBW-CMM

| | |
|---|---|
| R1 | 10k |
| R2* | 120R* |
| R3 | 120R |
| R4 | 120R |
| R5 | 120R |
| R6 | 120R |
| R7 | 1k |
| R8 | 1k |
| R9 | 10k |
| R10 | 10k |
| R11 | 120R |
| R12 | 2R2 |
| R13 | 4k7 |
| R14 | 1k |
| R15 | 4k7 |
| R16 | 1k |

| | |
|---|---|
| C1 | 10uF 16V Tant. |
| C2 | 100nf |
| C3 | 100nf |
| C4-C13 | Not Used |
| C14 | 100nf |
| C15 | 47nf |
| C16 | 47nf |

| |
|---|
| Case/Enclosure SW-120B |
| RS Components 373-2441 |

| | |
|---|---|
| D1 | 1N4004 |
| D2-D3 | Not Used |
| D4 | 1N4148 |
| D5 | 1N4148 |
| D6 | 1N4148 |

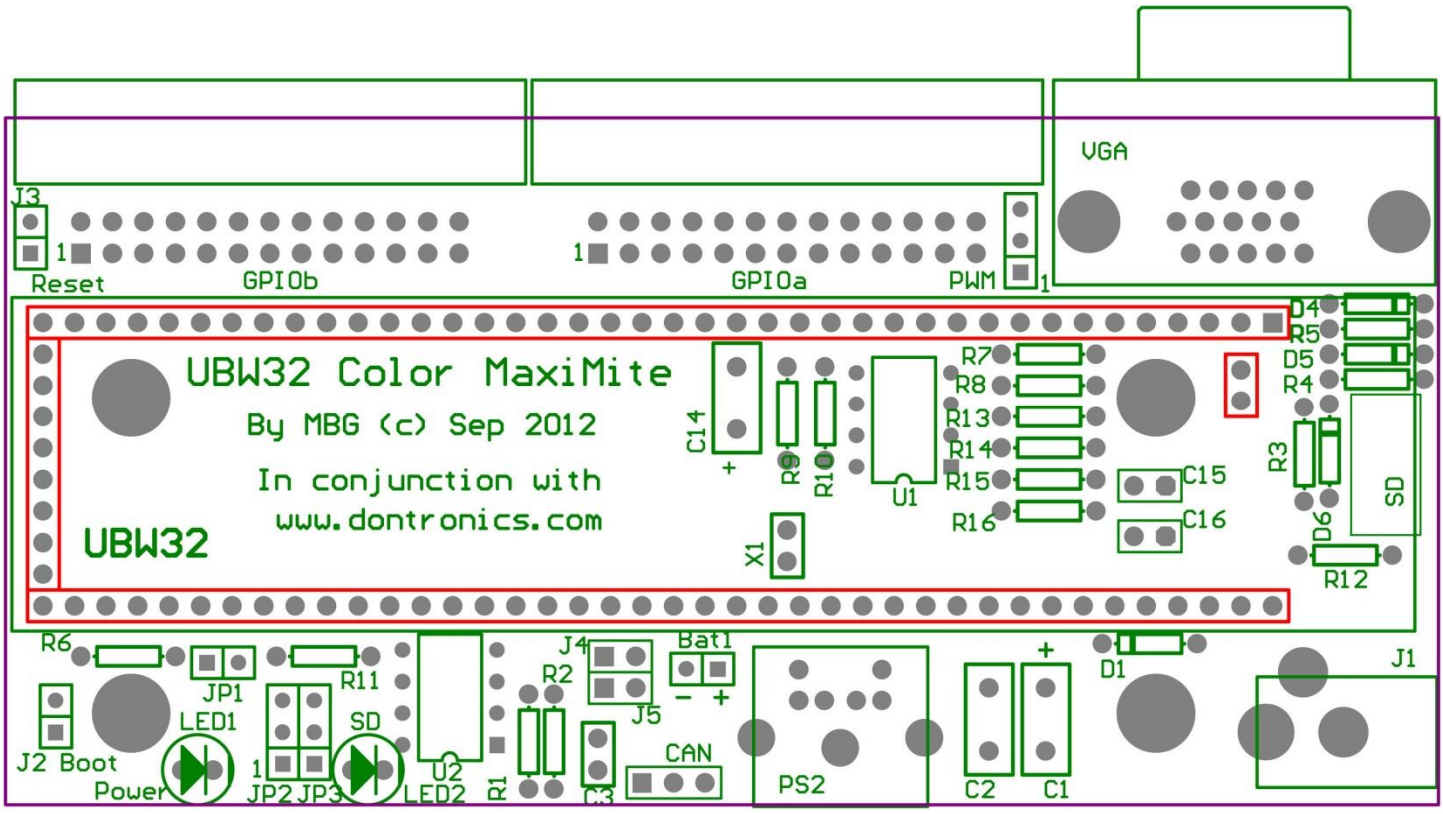
| | |
|---|---|
| X1 | 32kHz Xtal |

| | |
|---|---|
| U1 | DS1307 |
| U2 | MCP2551 |

| | |
|---|---|
| LED1 | 3mm Red LED |
| LED2 | 3mm Green LED |

| | |
|---|---|
| GPIOa | 26way male R/A Header |
| GPIOb | 26way male R/A Header |
| PWM | 3pin Male 0.1" Header |
| CAN | 3pin Male 0.1" Header |
| JP2 | 3pin Male 0.1" Header |
| JP3 | 3pin Male 0.1" Header |
| JP1 | 2pin Male 0.1" Header |
| J1 | 2.1mm Power Jack (PCB) |
| J2 | 2pin Male 0.1" Header |
| J3 | 2pin Male 0.1" Header |
| J4 | 2pin Male 0.1" Header |
| J5 | 2pin Male 0.1" Header |
| Bat1* | 2pin Male 0.1" Header* |
| VGA | 15pin Female HD R/A 'D' (Element14/Farnell P/N 1056018) |
| SD | micro-SD, MOLEX- 473093751 (Element14/Farnell P/N 1355805) |
| PS2 | 6pin Mini-Din Female (Element14/Farnell P/N 5749265-1) |
| UBW32* | Female Header strip-Low Profile for UBW32 (2x40 row, 1x8 row 1x2 row)* |

* = See Text



**Jumpers/Headers**

J1 - External Power input (7.5v – 15v DC), not needed if only powered from USB

J2 - External Boot button (Only if UBW32 is flashed with MCC bootloader.)

J3 - External RESET button, if required

J4 - Link if CAN required (F0 for CAN-Rx) (default Left open)

J5 - Link if CAN required (F1 for CAN-Tx) (default Left open)

JP1 - Links B5 to Pin(1), (only install if UBW is flashed with MCC bootloader.)

JP2/JP3 - Link Pins 1 & 2 (default) if SPI required on GPIOb or 2 & 3 if $I^2C$ required instead.

Bat1 - Connect the external coin batter to this header (check polarity carefully).

PWM - Sound/PWM output Pin1 (Gnd), Pin2 (Right or PWM#2), Pin3 (Left or PWM#1)

JPX - Configuration header for GPIOb,

CAN - Can Output, Pin1 – GND, Pin2 – CAN-H, Pin3 – CAN-L

**JPX Header**

When designing this board I explored various options for the pinout of the 2nd header. I had two in mind,

Firstly was a neat high-order layout in the same vein as the standard GPIOa header but with pin(21)-Pin(40) instead of Pin(1)-Pin(20).

Secondly I didn't wish to lose functionality of the Arduino if possible, so I have added a configuration header to the bottom layer that would allow the standard MCC Arduino footprint to appear if a Dontronics Donduino Cross was plugged into GPIOb see the following link for details:

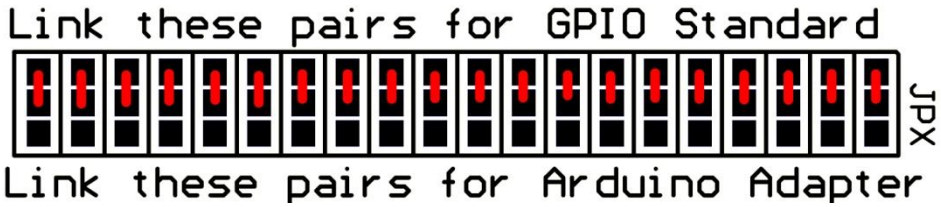
http://www.dontronics-shop.com/donduino-cross-letter.html

The JPX header is designed so that the pads can be solder linked easily, simply short all 20 pins with the selection you wish, either for standard layout or Arduino layout.

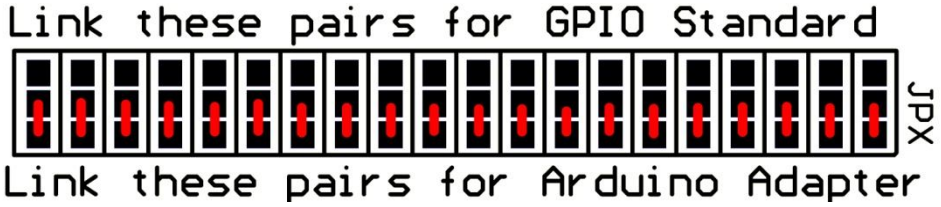
NOTE! That the reset and Vin signals do not transfer through this header as I wanted to maintain the standard voltage and GND signals for compatibility between the two GPIO headers.

# JPX Header

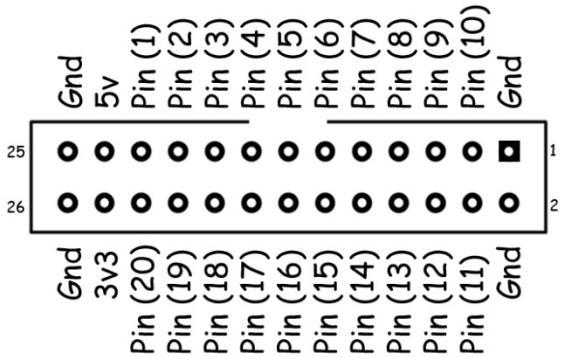
Optioned For Standard Layout




Optioned For Arduino Layout




## GPIOa Connector



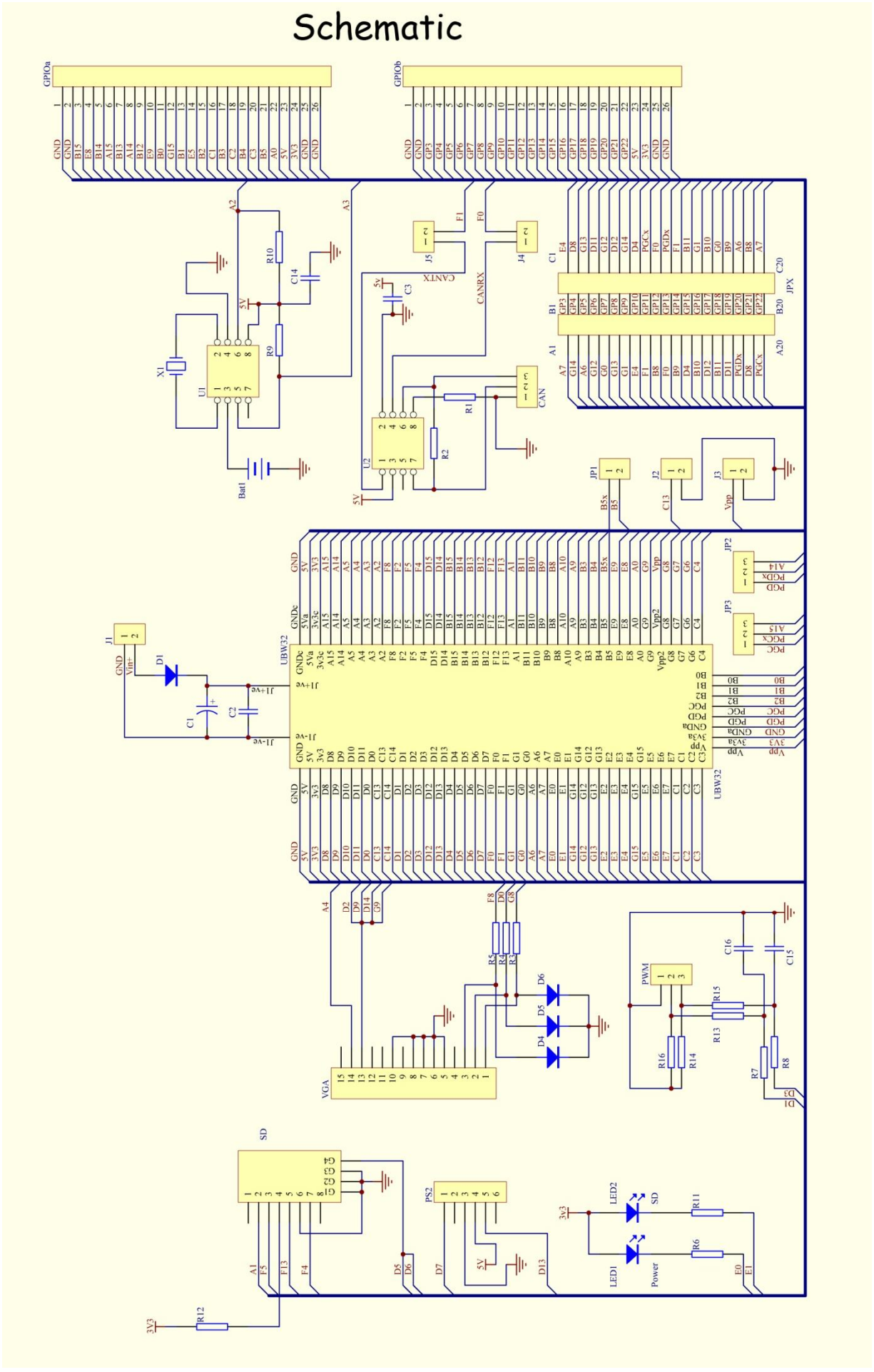

26 Pin Maximite GPIO Header
*(looking into Rear of Male Connector)*

## GPIOb Connector


JPX Optioned as `Standard'


JPX Optioned as `Arduino'

# Schematic


GPIOa
GPIOb
UBW32
JPX
CAN
CANTX
CANRX
VGA
SD
PS2
PWM
LED1
LED2
Power
Bat1
X1
U1
U2
J1
J2
J3
J4
J5
JP1
JP2
JP3
D1
D2
D3
D4
D5
D6
C1
C2
C3
C13
C14
C15
C16
C20
R1
R2
R3
R4
R5
R6
R7
R8
R9
R10
R11
R12
R13
R14
R15
R16

## Notes, Errata, Brickbats etc.

Firstly,

My humble apologies for not including a reference to **Geoff Graham** ( http://geoffg.net/ ) on the Overlay of the UBW32-MCC for without his tireless efforts this board would never have been.

There is a track missing on the UBW32-MCC PCB, associated with the auto-detect for Composite Video, I was under the impression that Composite was not supported anymore on the CMM. I was wrong! ... The fix is easy though... If you wish to use pin 9 as a composite/VGA select pin on the UBW32-MCC PCB then you will need to run a jumper from VGA pin 9 to UBW-C14 (about a 25mm or 1" link).

The connector for the coin battery is at .1" spacing instead of 2mm spacing so the wires from the battery will have to be either soldered directly or onto a .1" female header. As the Battery is fully insulated I installed the battery under the UBW32 and extended the wires around the uSD end of the UBW32 to the Bat1 header. In retrospect I wish I had put the Bat1 header under the UBW32.

The holes for the main Power socket are too large (Thank god for solder)

A couple of capacitors have a 0.200" footprint when they should be 0.100" (no big deal)

The PWM/Sound header and the CAN header are a bit cramped for space but still workable.