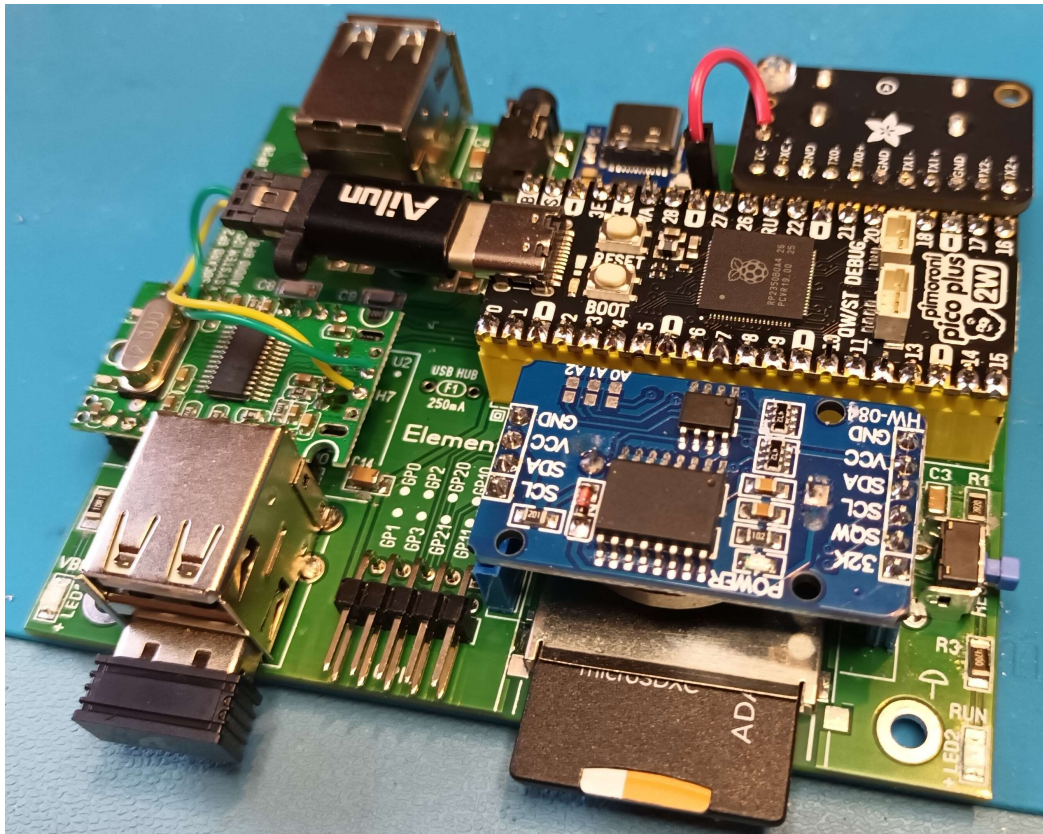


Elemento



Elemento isn't so much as a computer system to use as a practical construction project that can be used once you've taken it as far as you want to go. I've designed it specifically so that you can build it then upgrade or modify it yourself as I want to encourage people to get "hands on" with electronics. There are comprehensive assembly instructions and notes on tools and soldering so, even if you've hardly soldered anything before, you stand a good chance of getting this working. It's designed from the start to run MMBasic on any 40-pin Raspberry Pi Pico or pin-compatible clone, so there should be no compatibility issues. You can buy a new Pico module or use up one that was in the back of a drawer (yes, I find those sometimes too!).

In it's minimal form this is a relatively basic board compared to some. It includes:

- PWM stereo audio line out (not great but cheap on GPIO pins!).
- Standard or micro SD card (your choice).
- Connectors for a RTC with backup battery.
- Four USB-A sockets (powered but no data, sockets for a hub module are included).
- Connector for a display module (VGA or HDMI).
- Eight uncommitted digital GPIO pins.
- Qwiic connector for System I2C (optional)

It is very easy to include the above options, and others, on-board.

Elemento

Changes etc.

Revisions

PCB version 1.0 wasn't released as the prototype PCB is unworkable. It was used to verify mechanical fitting of the components though. This was the version originally used in forum photos.

PCB version 1.1 – an attempted bug fix - abandoned during design.

PCB version 1.2 – working prototype

- Has errors that link the top and bottom sides of the board. This doesn't cause a problem (and is probably an advantage in some respects), but for this particular design I would have preferred the position of the link to be in a more predictable place! This was unforeseen and is a function of how JLCPCB handle simple holes. All holes are drilled prior to plating, even board fixings and the locating holes for the audio jack and SD card. I should have put a pad on at least one side to isolate them. Another top-bottom link is the top hole of the side-mounting Reset switch (not its support bracket). This is a thermal pad with links on both sides of the board.
- More serious, TXD and RXD to the CP2102 module are reversed. This can be patched beneath the board. This is important if a USB system is being built.
- G and K on the VGA module are reversed, which messes up the colours. If you swap the 390R and 820R resistors the module will work ok but the labels are wrong.
- HDMI pin marking is wrong. Corrected to OPTION HDMI PINS 1,3,6,4

PCB Version 1.3 – all Version 1.2 errors noted above should now be corrected.

Currently the photos shown in this manual are of the prototype Version 1.2 board, which does not include the new items listed below.

- ☑ A normally closed GND link has been added beneath the board so that the two sides can be isolated from each other for experimental purposes.
- ☑ R6 and R7 have been added, If an amplifier is being used these will make no difference but they will reduce the maximum volume if headphones are being used.
- ☑ Normally closed links for GP20 and GP21 added.
- ☑ A Qwiic connector has been added for System I2C. R8 & R9 System I2C pull up resistors also added.
- ☑ K3A option added.

Still to do

The LCD option is a possible further development. It will allow a serial-connected LCD display panel to be used and will effectively replace the internal SD card socket as the display panel will also require SPI2. The SD CS signal is duplicated on the display module connector so that a SD card on the display panel can be used if present. The LCD display module is work in progress.

Elemento

Elemento isn't designed for particularly high performance and the audio quality is fairly poor (although there is very little background noise and the output is protected and DC isolated so you can use headphones), but the cost and complexity are low enough for just about anyone to have a go at building it. At a rough estimate I'd rate the build difficulty to be slightly more advanced than building something on strip or pad board. That's mostly because the resistors and capacitors are all surface mounted. However, they are still fairly easy to work with after a little practice and I have included lots of notes on doing so at the end of this manual. Please don't let it put you off. :)

Unlike the "test rig" type boards Elemento can have an enclosure. It is made from ABS plastic, 100mm x 80mm x 29mm and the top simply clips onto the bottom plate. It has a decent finish and is very low cost.

The user's GPIO pins are uncommitted and are presented as four pairs, arranged to give you SPI and I2C ports, but use them as you will. Unfortunately at this time it hasn't been possible to make the ADC inputs available.

1	GND		
2	3V3		
3	GP10	I2C2 SDA	
4	GP11	I2C2 SCL	
5	GP20	PWM2A	
6	GP21	PWM2B	
7	GP2	SPI CLK	
8	GP3	SPI TX	
9	GP0	SPI RX	COM1 TX
10	GP1	SPI CS	COM1 RX

Unfortunately the use of GP20 and GP21 is restricted to PWM2 or conventional digital I/O as their other functions, I2C and COM2, are both reserved for internal use.

The following pins and signals are reserved for internal use:

GP4 & GP5 (I2C SDA / I2C SCL) are used for System I2C
GP6 & GP 7 (PWM3A/B) are used for PWM audio
GP8 & GP9 (COM2 TX / COM2 RX) are used for the serial console
GP12 to GP19 are used for the display
The SD card uses:
GP22 – SD CS
GP26 - SPI2 CLK
GP27 – SPI2 TX
GP28 – SPI2 RX

Additionally, an optional Qwiic connector can be fitted. This allows 3V3 I2C modules to be connected to Elemento without using the user's GPIO port. It is connected to the System I2C bus, which is normally only used for the RTC.

Elemento

The display module connector can be used as additional GPIO if no display module is installed. Note that GP22 is also used as the front SD card select signal.

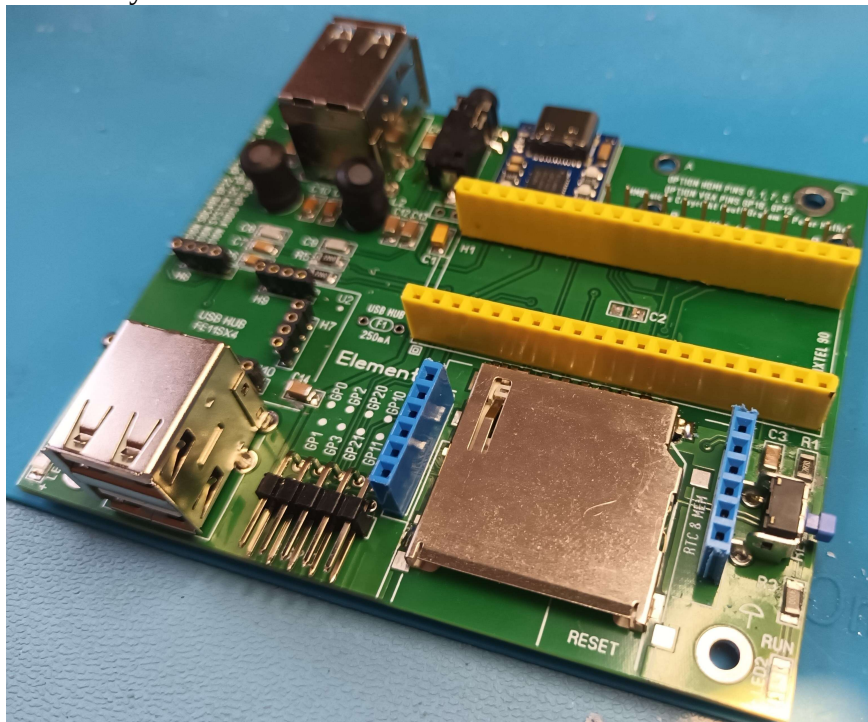
Pins listed from left to right:

Signal	Mode	HDMI	VGA	LCD (proposed)
5V		5V (wired)	No pin	(2) 5V
GP22		No pin	No pin	(14) SD card CS
GP12	HSTX0	TXC-	Blue	(13) SPI2 RX
GP13	HSTX1	TXC+	Green	(12) LCD CS
GND		GND	GND	(1) GND
GP14	HSTX2	TX0-	Dk Green	(11) SPI2 CLK
GP15	HSTX3	TX0+	Red	(9) SPI2 TX
GND		GND	GND	(1) GND
GP19	HSTX7	TX1-	No connection	(5) Touch IRQ
GP18	HSTX6	TX1+	No connection	(3) Touch CS
GND		GND	GND	(1) GND
GP17	HSTX5	TX2-	Vertical sync	(8) LCD RS
GP16	HSTX4	TX2+	Horizontal sync	(10) LCD DC

The numbers in brackets are the provisional pin numbers on a polarised 2x7 connector for the LCD.

The design is built in “Stages”, of which most people will require more than one. Here is an overview, the Assembly section contains full details.

A typical Stage 1 assembly:



Elemento

Stage 1

This is the minimal PCB. It includes all the connectors for the other stages to be added, even when it has been screwed down onto the base of the enclosure in most cases. You will also select your chosen SD card socket and hence the RESET button position. The USB sockets are included and powered but not active.

The Qwiic connector (K7) is optional as it isn't used by the rest of the system.

This stage is actually of some use on its own. If you plug in, say, an original PicoMite and power it via its USB socket you will be able to configure and use it via the MMBasic USB console using Tera Term or similar as expected, just as you would one of the commercial "test rig" boards. This includes access to audio, SD card and all the available GPIO pins. As the Pico is being powered via its USB connector the VBUS LED will light. The RUN LED will also light as the PicoMite will be providing a regulated 3V3 supply. The USB sockets can be used for normal 5V supplies up to a combined maximum of 200mA continuous. If you add Stage 4 (and provide power / access the console via the USB-C connector) you can configure it to use a USB keyboard and mouse.

Stage 2

Fitting Stage 1 into the enclosure. Specific details are provided for installing the display modules. You need to decide which display you are going to use at this stage and fit the required spacers to the PCB before screwing it into the case. (For my prototype module I marked through the PCB holes onto the base and drilled three clearance holes for the spacer fixing screws. I can now change the display module without removing the PCB. :))

Stage 3

Installing a display module. The module will have to be pre-assembled then plugged in and secured by its fixing screw(s).

Stage 4

Adding the 4-port USB hub. This plugs into H7 to H11. When a PicoMite is running a USB version of the firmware a short lead is used to connect the hub to the USB port of the PicoMite. The USB port is used in Host mode so cannot be used for access to the console or for powering the board. These functions are now carried out via the USB-C socket on the rear. The console is accessed via the serial port at 115200 baud 8N1.

Stage 5

The RTC module is optional and some may consider it a luxury, but once set it can time and date files written to an SD card or USB stick. It plugs into sockets H4 and H5.

Elemento

I have made no reference to the MCU that you use, either in the text or in the Bill of Materials. It's entirely up to you, providing that your module is pin-compatible with a genuine Raspberry Pi Pico. You can find the standard diagram for the PicoMite in the MMBasic manual. There are no connections to ADC_VREF or 3V3EN and the TP pads are also ignored. Pin 40 is only connected to the VBUS LED for indication and any 5V supply out of the USB connector is ignored, the hub being powered independently.

The standard Raspberry Pi Pico works, and the cheap “black version C” clone (otherwise known as YD-RP2040) works fine for USB and VGA but keeps the VBUS light on while powered up. The much more capable (and much more expensive!) Pimoroni Pico Plus 2 or Pico Plus 2 W modules are also supported. You get PSRAM included on them and, in the latter case, wi-fi and Bluetooth connectivity (the Qwiic connector on these is connected to System I2C and 3V3 – they don't include a Qwiic lead or male pin headers with the module). Hopefully this will give new life to those older modules that you aren't using but don't want to get rid of!

The only thing that really divides the capabilities of the various devices is that those based on the RP2040 can't have HDMI capability as MMBasic requires the HSTX module, which isn't present on this chip.

You will need to install the appropriate version of MMBasic for your hardware, of course. These are detailed in the MMBasic manual.

If you've built a VGA Elemento with a RP2040 you can still upgrade the board to HDMI later using a RP2350 module and an Adafruit DVI breakout board.

You can build a single Elemento, but you have to have 5 PCBs anyway as that's the minimum quantity, so why stop at one? :) Most of the small components (e.g, resistors, capacitors & LEDs) can be found as cheap packs on AliExpress (although for some reason I had to get 2n7 capacitors and a couple of resistor values separately). This makes it very cheap to at least partially equip all 5 boards should you feel like it.

The most expensive specialised component is probably the Adafruit DVI breakout module. It's not expensive (less than £2), but shipping charges may make it more sensible to order a few at the same time, even if you don't need them immediately.

The only active components apart from the Pico are the RTC and the USB hub. This can be a cheap build!

Elemento

Bill of Materials - Stage 1

Name Value Comment

1off PCB – custom design – see note at end of Bill of Materials

C1 22uF 1206 capacitor
C2 100n 1206 capacitor
C3 10uF 1206 capacitor
C4 10uF 1206 capacitor
C5 10uF 1206 capacitor
C6 33n 1206 capacitor
C7 33n 1206 capacitor
C8 2n7 1206 capacitor
C9 2n7 1206 capacitor
C10 68n 1206 capacitor
C11 68n 1206 capacitor
C12 10uF 1206 capacitor
C13 10uF 1206 capacitor
C14 10uF 1206 capacitor

All the capacitors are standard 1206 size MLCC

F1 0.2A RXEF020 Self resetting fuse 6mm diameter x 5mm pitch (Littelfuse)

H1 2off 1x20 0.1" female socket header for Pico
H2 1x13 0.1" male pin header (this will need to be cut down from a 15-pin or larger)
H3 - reference no longer used -
H4 6-way 0.1" female socket header
H5 6-way 0.1" female socket header
H6 2x5 0.1" right-angle male pin header
H7 1x4 2mm female socket header
H8 1x4 2mm female socket header
H9 1x4 2mm female socket header
H10 1x4 2mm female socket header
H11 1x4 2mm female socket header

I use the gold-plated turned-pin "snap off" type in 2mm pitch for H7 to H11.

K1, K2, K3 and K3A are alternatives. Fit only one

K1 X5F-40T Push-pull SD card connector 11-pin, long body – AliExpress - [Link](#)
K2 DM1AA-SF-PEJ(72) Push-push SD card connector
K3 micro SD card breakout
K3A TFP09-2-12B SMD push-push micro SD card socket

K3A is not recommended with the enclosure due to small projection of the card.

K4 double USB-A socket
K5 double USB-A socket
K6 SMD 3.5mm jack RS 705-1490
K7 JST SH1.0 vertical 4-pin Qwiic connector (optional)

Elemento

L1 4m7H 6mm vertical inductor (8mm high, 5mm pitch)
L2 4m7H 6mm vertical inductor

LED1 1206 LED (your choice, I suggest red)
LED2 1206 LED (your choice, I suggest green)

R1 2R2 1206 resistor
R2 1K8 1206 resistor (using red LED)
R3 470R 1206 resistor (using green LED)
R4 220R 1206 resistor
R5 220R 1206 resistor
R6 220R 1206 resistor
R7 220R 1206 resistor
R8 10K 1206 resistor
R9 10K 1206 resistor

S1 B3F-315n Horizontal 6x6 tactile switch

U1 USB-C USB-C to TTL converter CP2102

You will also require a USB-A to USB-C lead to power the board via U1. Additionally you may need a second lead or an adapter to power the Pico module directly while installing or updating MMBasic. Note that both of these must include the data signals, power only leads cannot be used.

You cannot usually power U1 from a USB-C supply as it does not include the necessary components to activate the output.

Bill of Materials – Stage 2

The enclosure is intended to be what looks like a ventilated active antenna or air temperature sensor case to me. It is a simple 2-piece moulded click-together design I found on AliExpress. Those of you with 3D printers could probably produce something similar (or even much nicer!), but it was only £1.47 in white or black and the outside finish is good. I used nylon spacers and screws for the display modules.

Name	Value	Comment
------	-------	---------

1off	Enclosure - Link to AliExpress	
------	--	--

4off	#4 x 6mm fixing screw for PCB (max thread dia 2.9mm, head height 3mm)	
------	---	--

Elemento

Bill of Materials – Stage 3

For HDMI display:

1off	Adafruit DVI breakout module
1off	11mm female/female M2.5 tapped spacer fitted into hole “A”
2off	M2.5 x 6 screw
11-way	0.1in female socket header fitted from the socket side of the module
1-way	female socket header on short flying lead soldered to 5V on the module

For VGA display:

1off	PCB – custom design
1off	VGA connector, compact design - AliExpress link
2off	270R 1206 resistor
1off	390R 1206 resistor
1off	820R 1206 resistor
11-way	0.1in long-pin female socket header fitted from socket side of the PCB
2off	15mm female/female M2.5 tapped spacer fitted into holes “B”
4off	M2.5 x 6 screw

For LCD display [anel with SPI interface:

1off	PCB – custom design
1off	right-angle 2x7 male boxed header
13-way	0.1in long-pin female socket header fitted from socket side of the PCB
2off	15mm female/female M2.5 tapped spacer fitted into holes “B”
4off	M2.5 x 6 screw

Bill of Materials – Stage 4

Name	Value	Comment
------	-------	---------

U2	FE1.1SX4	USB 4-port hub module - AliExpress link
5off	strip of 4	2mm pitch male header pins
I use the gold-plated turned-pin “snap off” type in 2mm pitch		

Suggested

small size micro USB to USB-C adapter

micro USB male plug

small size (I used 30 AWG) silicone insulated wire if available

Bill of Materials – Stage 5

RTC module

CR2032 battery

strip of 6 0.1” male header pins (see Stage 4 Assembly section)

strip of 4 0.1” male header pins

Elemento

Ordering the PCBs

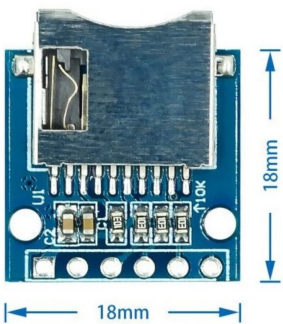
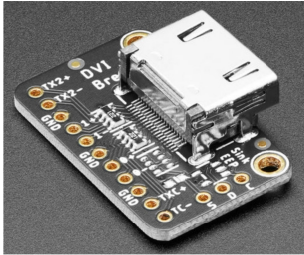
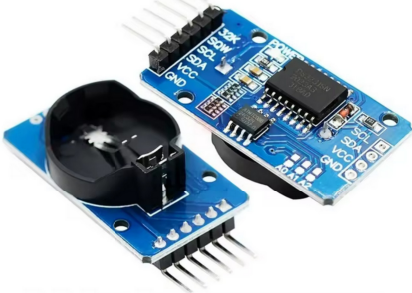
I ordered the PCBs online from JLCPCB. Their web site is quite easy to use. I recommend using the default thickness of 1.6mm for the main board as using anything less may allow the points of the fixing screws to push through the bottom of the case very slightly. The display module PCB thickness is immaterial – I used 1.2mm for these.

Tools

If you haven't got them already then tools will be the biggest cost. You will almost certainly need a multimeter, preferably one with a diode test function that will test your LEDs (even many cheap ones will do that now). You will also need suitable soldering tools – your dad's old Solon 25W poker isn't much use here! See the Soldering section for a lot more information.

Any Pico or clone in a breadboard will provide convenient 5V and 3V3 supplies from your PC's USB socket, useful for checking your LEDs operation via their series resistors. This is handy if you have decided to go for alternative colours. The VBUS LED and resistor require 5V and the RUN LED and resistor 3V3.

Elemento

 K3	 U1	 K6
 DVI Breakout	 VGA Socket	 U2 USB Hub
 H6 (typical)	 Enclosure	 RTC Module

Elemento

Assembly

I recommend that you follow the suggested assembly sequence if you are a beginner. I have attempted to arrange things so that you will always have reasonable access to the area that you are working on. The general rule is to fit the lowest components first then work upwards.

Stage 1

Fit the two LEDs. There is usually a T symbol on the bottom, the long leg of which points towards the cathode (marked + on the PCB¹). Test them before and after soldering to make sure that they still work. At this stage you can take them off fairly easily and try again if they didn't survive. If you have used colours other than red or green you may wish to use different values for the series resistors. remember that LED1 has a resistor from a 5V supply and LED2 has a resistor from a 3V3 supply with less current capability though. The suggested values supply approximately 2mA to each LED.

Fit the remainder of the resistors and the capacitors.

Fit your choice of SD card socket or the micro SD card module. The latter should be fitted flat to the PCB and the connections "staked through" with pieces of bare wire then trimmed.

Fit the reset button. If you are using K1 or K2 then this must be in the side position. If you are using K3 or K3A and wish to use the front position for Reset then close the LK1 solder blob link beneath the board.

Fit K6.

Fit U1, mounted flat to the board and staked down as described for K3.

Fit K7 (if required)

Fit H7, H8, H9, H10 & H11. Note that these are 2mm pitch, not 2.54mm (0.1"). If you have a USB hub ready then I suggest plugging 4-way strips together first, support the board on blue-tack, drop the connector pairs into the holes, align the hub onto them and solder at least one pin on each onto the hub. You can then turn the entire board and hub over and solder the sockets on the underside of the board. You can flip it back and complete soldering the hub if you like – you can always unplug it if it's in the way. This technique ensures that all the connector pins are perfectly aligned.

From this point it's relatively easy to complete the remainder of Stage 1 assembly. I suggest that you mount the rest of the pin headers and sockets next, perhaps holding them down with bits of blue-tack while you turn the board over to solder them. K4 and K5 are the highest components so fit them last.

1 Some expect + on a LED to appear at the end made positive to light it, which is the opposite! The reason for the + being at the "pointed" end of a diode is historical. "Cathode" refers to the cathode of a valve (tube for those of you in the US) rectifier, where the positive supply to the circuit appears, so it is always marked with a "+" symbol. An LED is powered by passing current from anode to cathode. In this case we are powering the anode via a resistor and grounding the cathode so you see the end with a + symbol connected to GND.

Elemento

Stage 2

Once the enclosure cutouts are completed assemble the base and PCB into the top by offering the rear components through the holes first then “hinging” the base up into position until it clicks.

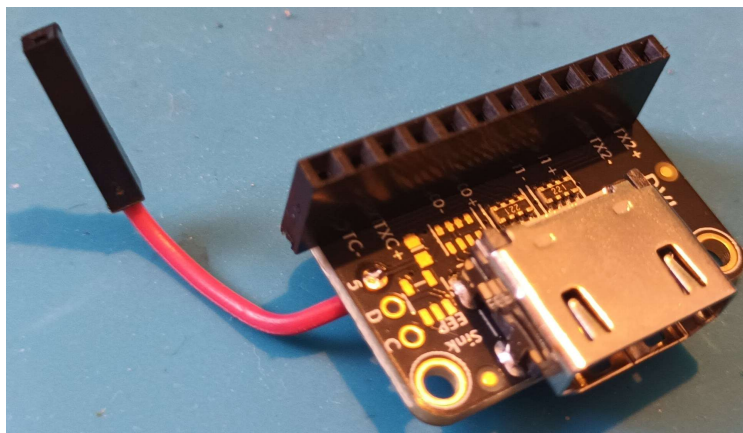
Stage 3

When installing any of the display modules do not solder the female socket header to the module PCB without following these instructions. Fit the module mounting pillars onto the main PCB. Plug the female header down onto the male pins, aligning it to the right hand side. Offer the display module down onto the pins of the female header and fasten the module down onto its mounting pillar(s). Solder the connections onto the module. Remove the fixing screw(s) and unplug the module.

For HDMI display

Fit one 11mm long M2.5 spacer into the PCB hole marked “A”, inserting the screw from the bottom of the board.

Use a 11-way 0.1” pitch female socket header for the Adafruit DVI breakout module. As 11-way is non-standard you can use 12-way if you pull put the contact to the left of TC-. Note that the module aligns to the right of H2 when plugging it in. The flying lead and connector provide 5V to the module.



For VGA display

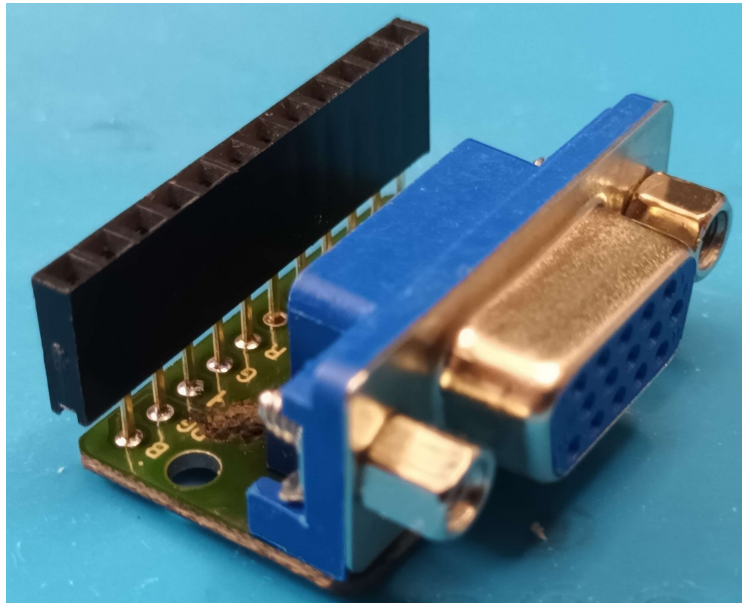
Fit two 15mm long M2,5 spacers into the PCB holes marked “B”, inserting the screws from the bottom of the board.

Working on the little VGA PCB, fit the four resistors where shown. Fit the VGA connector from the opposite side of the board to the resistors.

Use a 11-way 0.1” pitch female socket header for the VGA module. As 11-way is non-standard you can use 12-way if you pull put the contact to the far left. Note that the module aligns to the right of H2 when plugging it in.

Elemento

You should end up with something similar to this:



If you are going to use the enclosure then you should even up the heights of the moulded in standoffs first!. They are not very accurate. I put a stack of three M5 plane washers over each in turn and shaved a tiny amount off each with a sharp knife (although sandpaper is fine). The PCB should then be fitted to the baseplate, The little locating peg on the baseplate should be under the USB hub position.

The display module may now be plugged in and retained using its fixing screw(s).

If using the HDMI module then fit a short link from 5V on the Adafruit module to the 5V pin at the left hand end of header H2.

Stage 4

Fitting the 4-port USB Hub.

You will need to fit it with 2mm pitch male pin headers to line up with the female headers on the PCB. There is no common arrangement for connecting to the USB system across the 40-pin Pico-type modules. I suggest getting some micro USB male plugs, a micro USB to USB-C adapter and some 30 AWG silicone insulated wire from AliExpress or similar. This wire is excellent for soldering, with no insulation “run back” (it’s used a lot on drones). Make up a lead from the micro USB plug to the USB hub. You can now use it directly for standard Pico modules or, using the adapter, for clones that use USB-C. Just twist the wires, don’t worry about screening. The distance is short. Don’t connect the supply pins, only D+ and D- are needed here. If you are fussy about wire colours then D+ is green and D- is white, but apparently the electrons aren’t bothered. :)

Elemento

Stage 5

There are two ways to fit the RTC module. Which you use will depend on whether you intend to use the suggested enclosure. The interior height is limited so the RTC is fitted inverted, with the battery holder on the bottom side. There is just enough clearance beneath it for this. If there is no height restriction the RTC can be fitted with the battery holder on top, which is much easier.

To fit in “battery on top” position

Straighten the pins on the module. Solder a 4-way strip of male header pins into the other holes, on the same side as the 6-pin connector. Plug the 6-pin connector into the **left-hand side** socket (H5). The 4-pin connector will go into the lower 4 sockets of H4.

To fit in “battery underneath” position

Fitting the RTC module is awkward. I have only ever seen them supplied with pre-soldered 90 degree header pins at the 6-way end. This means removing the connector. The best way I’ve found to do this so far (I haven’t tried hot air yet) is to use flush-cutting snips to cut the pins down to the insulator then use a fine bladed screwdriver to pull the insulator off. Each pin can then be unsoldered and removed individually with tweezers. I then clear the holes by supporting the board and heating one side of the hole while using a solder sucker on the other. This usually works reasonably well. The male pin headers can then be fitted from the battery holder side. Plug the 6-pin connector into the **right-hand side** socket (H4), the 4-pin connector will go into the upper 4 sockets of H5.

Note that only H5 carries any signals or power. H4 is used purely as mechanical support.

Elemento

Modifications

There are a few of changes that the user can make to customize their build.

Solder blob link, LK1 can be broken to disconnect the incoming 5V supply and route it via J1, which is not normally fitted. This allows the user to fit a power switch, if required, which plugs into J1.

There are two alternative positions for reset button access. Both are intended for poking the button with a small object, not for a button cap. In fact, if you fit a cap to the button in either position you will lock the case on! The front position can only be used if K3 is used as it is covered up by K1 or K2. As a safety measure, in case the front position is shorted down by K1 or K2, it has to be enabled by solder blob link LK2.

If it is considered that 200mA is insufficient for the USB ports then simply short out F1. However, for this type of fuse the rating is the maximum that can be carried without entering the trip state. The trip state is entered at 400mA, at which point the current is limited as the resistance increases (not disconnected). The suggested one has a maximum trip time of 2.2s. Be aware – these can reach 85°C in the tripped condition. Even with F1 shorted out there is still a fixed limit of 500mA (which is the standard maximum current for a single USB2 output) for the entire system, set by a fuse on U1.

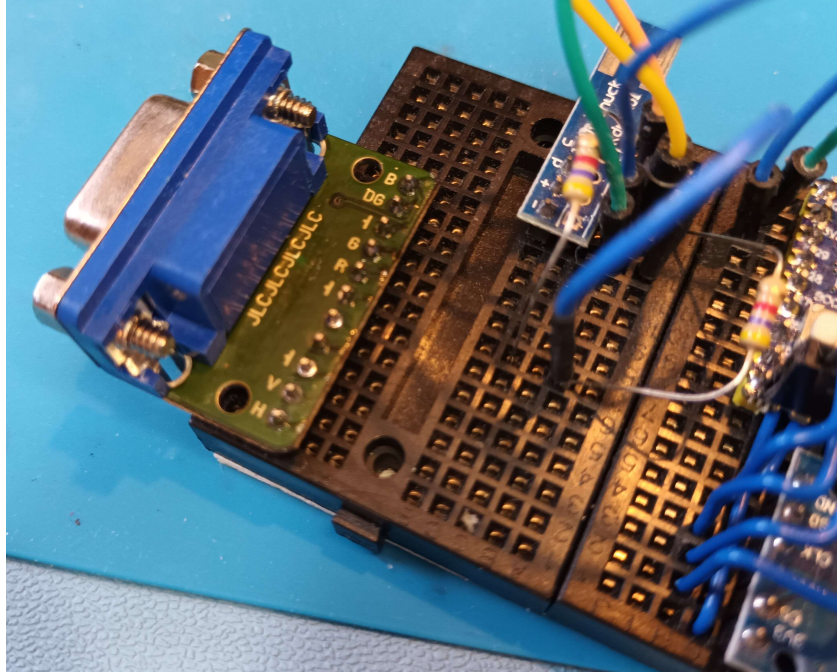
The audio output is intended to drive the volume control of a following low gain amplifier stage but it is quite capable of driving medium-high impedance headphones directly. The sound may be rather lacking in base in this case. This can be improved by increasing the value of C12 and C13 to 100uF. If the maximum volume is too loud then R6 and R7 can be increased in value. This is better than using the PLAY VOLUME command in MMBasic as the sound quality is better maintained.

The top layer of the PCB is connected to GND. The bottom layer is linked to it via a link marked GND underneath the board. Thus the bottom layer can be isolated for experimental purposes.

As mentioned previously, GP20 and GP21 on the user's GPIO connector are of limited use. However, some MCU modules bring out additional pins. For example, the YD-RP2040 makes GP29 (ADC3) and GP23 (which would normally be used to control the on-board LED via a link) available. GP20 and GP21 have links underneath the board which allow them to be isolated and pads which allow them to be wire-linked. Thus up to two such specialised pins can be made available to the user if required.

Elemento

It is possible to invert the VGA display module to use it as a VGA interface onto a breadboard:



Elemento

Configuration

Configuration of Elemento depends a lot on which items are fitted. I have split up the necessary OPTION commands here for future reference. Some are also written on the PCB. I suggest that you start by copying down the list for your particular build.

Check the MMBasic manual for the OPTION settings, particularly for the LCD panel. Some OPTIONS can have additional settings and many vary between MMBasic version numbers!

For a HDMI display	OPTION HDMI PINS 1,3,6,4
For a VGA display	OPTION VGA PINS GP16, GP12
If display is either of the above	OPTION SDCARD GP22, GP26, GP27, GP28
If display is NOT one of the above or there is no display	OPTION SYSTEM SPI GP26, GP27, GP28 OPTION SDCARD GP22
OR	
If the display is a LCD panel with a SPI interface (See MMBasic manual)	OPTION LCDPANEL <paneltype>, <orientation>, GP16, GP17, GP13 OPTION TOUCH GP18, GP19
In all cases	OPTION SYSTEM I2C GP4, GP5 OPTION AUDIO GP6, GP7

GP26, GP27 and GP28 are SPI2. If you wish to use the SD card socket on a LCD display panel instead of the one on the main board then use GP22 as its select pin. Note that you cannot have SD cards in both sockets as they will both activate at the same time.

Example 1 – RP2040 based without a display

```
OPTION SYSTEM SPI GP26, GP27, GP28
OPTION SDCARD GP22
OPTION SYSTEM I2C GP4, GP5
OPTION AUDIO GP6, GP7
```

Example 2 – RP2040 based with VGA output

```
OPTION VGA PINS GP16, GP12
OPTION SDCARD GP22, GP26, GP27, GP28
OPTION SYSTEM I2C GP4, GP5
OPTION AUDIO GP6, GP7
```

Example 3 – RP2350 based with HDMI output

```
OPTION HDMI PINS 1,3,6,4
OPTION SDCARD GP22, GP26, GP27, GP28
OPTION SYSTEM I2C GP4, GP5
OPTION AUDIO GP6, GP7
```

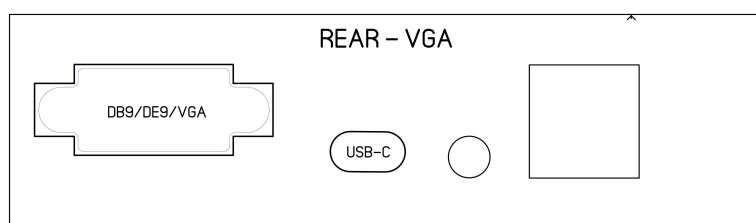
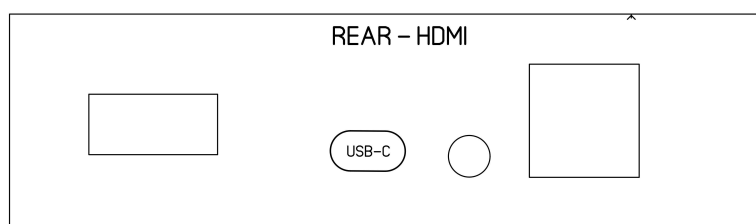
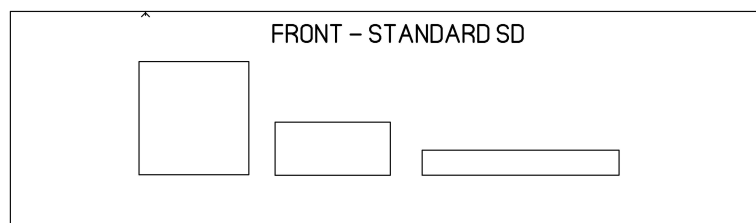
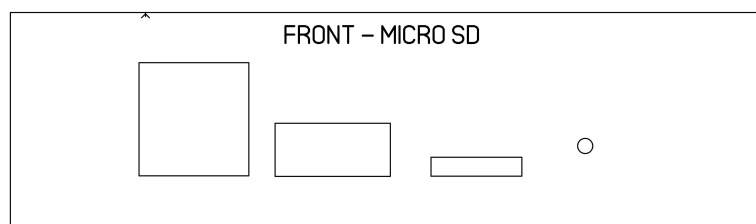
Elemento

Enclosure

These are full size templates for marking out the cutting positions on the enclosure. Make sure this page is printed out full size. Each template should be 100mm wide x 29mm high. The size should be correct if the page is printed out 1:1 on A4 paper.

Stand the template up on a flat surface, against the face to be cut, with the bottom corners aligned. The little mark at the top aligns with the notch across the front of the enclosure. Mark through with a sharp object such as a scribe then join the dots after removing the template. The material is quite soft and is easily drilled and filed. Initially I drilled holes and used a pair of snips to nibble bits out.

The front micro SD template is not suitable for K3A as it stands. The slot position is too high.



ENCLOSE CUTTING TEMPLATES
ALIGN BOTTOM CORNERS AND TOP MARK WITH NOTCH

Elemento

Soldering

A lot of this will be preaching to the converted. Sorry, but please bear with me. :)

The secret to all successful soldering is cleanliness. For the solder to make a connection it has to make a very close molecular bond with a surface. If there is anything, such as tarnishing or surface film, in the way then that bond isn't possible. For this reason it's important not to handle the conductive pads of a PCB before soldering to it as oils can be transferred onto the pads. It's good practice to clean the surface of the pcb with isopropyl alcohol immediately before you start work on it. They are usually pretty clean from the manufacturer so keep unused boards in their packing until needed.

During the soldering process a flux is used. When heated this becomes "activated" and "softens" the surfaces of the component and pad, allowing the solder to flow sufficiently to make that close bond that's needed.

The bond finally happens once the heat source is removed and as the surfaces are cooling. This is why it's important that nothing moves during this time – the joint will fail and the solder may, in some cases, crystallise rather than flow, indicating a "dry joint".

All the above applies to soldering in general.

Surface Mount Devices

The main thing with SMD is don't let it scare you. Beat it into submission. :)

You need the right tools for this job.

You need good magnification and a **very** good light. I got a cheap (about £17 from AE) Chinese illuminated magnifier, a bit like an Anglepoise light, a while ago and it's been worth every penny. If you can't see what you are doing then nothing will work.

I have a small mono microscope that I sometimes use for inspection. I can solder under it but it's not that easy and I can't afford a binocular microscope. The lens is fine in most cases. A small 30x21 mm jewellers lens is usually fine for inspection.

You also need tweezers. Not the "el cheapo" variety, ones that give you precision control. My favourites are "Mechanic Aax-17 Chip soldering tweezers", which are kinked. Less than £4 at AE.

Use a temperature controlled iron (preferably a T12 type) with a pointy bit. I've found that my favourite for SMD and small joint soldering is the curved pointy one (T12-J02). You can solder in odd positions with that or "drag" solder over a row of pins. There is also a longer version, T12-JL02. I spent around £30-35 on a T12 setup including a second "handle" with a soft silicone lead and some extra bits. Seriously, it's revolutionised how I work. I don't think I could go back to a conventional iron now, this is so much easier. It even switches itself off if I leave it for too long (and comes back up to temperature in a few seconds when I want it again). It takes seconds to change the

Elemento

temperature and not much longer to change the bit for a different one (even while hot if you grab it using a silicone mat).

Get proper gel flux and solder paste in syringes. You use tiny amounts so it lasts for ages. Keep it in the fridge when you aren't using it and allow the solder paste (at least) to warm up before you use it otherwise you'll have a heck of a job trying to get it out of the syringe. Yes, you can solder SMD components with wire solder but you really need three hands to do that. Mix solder paste and flux as a sloppy mix on a bit of plastic or something to make it stick better - it makes everything so much easier. Plop some on the (cleaned) pads with a toothpick, hold the component down onto it with the tweezers, a couple of taps with the iron and it's done. Don't be afraid to solder fairly hot (I tend to use something around 300°C). You'll do less damage to components soldering them hot and quickly than by using a lower temperature for a few seconds more, especially LEDs.

If you mix a little too much of the flux and solder paste don't waste it, the mixture will keep for several days even at room temperature.

If you think you need to then you can get a couple of the SMD practice kits from your favourite Chinese bits seller and complete them before tackling anything costly. However they might put you off as they go down to very small sizes. *The Elemento board is *by far* easier to solder than any of those kits!* Practice, practice, practice. It doesn't really matter if you kill a few cheap components and one of the PCBs. You will curse and swear as tiny components ping off somewhere or as they refuse to be soldered where you want them. You will also accidentally push them out of position while soldering. It's all part of the learning process. :) You will get the hang of it eventually.

I've started laying out more boards using size 1206 components. They are (for me) a good compromise. I can go down to 0805 reasonably well, but 1206 is pretty easy. I can still melt 0805 LEDs too easily, :(Yes, you will kill some LEDs at first. Yes, that bag of 600 that you got from AE for £3.50 will still last you for years. :)

For a bit more adventure in SMD soldering, you can get very cheap PTC heating plates which have a fixed temperature of 260°C. They are great to use as soldering hotplates, if a little crude as they heat very fast and take a long time to cool. They are very useful for stripping components off boards, even removing LEDs from aluminium PCB. If you decide to try one then remember that 260C is very hot. I recommend using metal spacers between the heating plate and strips of wood then more spacers from other holes in the wood to the mounting surface. You need to break the conduction path. There are quite good quality variable temperature hotplates for less than £30 if you prefer something a bit nicer.