

ASSEMBLY INSTRUCTIONS FOR THE REVISION G BOARD PBUFF SHORT FORM KIT.

Items supplied in kit:-

One bare double sided, plated through printed circuit board.
One 2764 or larger EPROM programmed with PBUFF Ver 2.2, Ver 3.2, (or Ver 4.0 or later when available.)
These assembly instructions.

Ver 2.2 is an 8K to 64K buffer using 1 to 8 by 4164 Drams.

Ver 3.2 is a 256K buffer using 8 by 41256 Drams.

For both these above versions, DON'T install E15 (555), the board will not work. You don't need to install E14 (6264), C7, C8, R7, & R8

Ver 4.0 and later, are 256K buffers that make use of only 8 by 41256 Drams.

The following components MUST be installed.

E14 (6264), E15 (555), C7, C8, R7, & R8

The kit builder should have a reasonable tool kit which will include a Multi-Meter. A logic probe may be needed to help you with any hardware problems that you could run into. The logic probe I use is a 3800A. It is available at most electronic outlets for around \$30.00 Don't rush out and buy a logic probe because I said you may need one, you probably won't, but if you do run into real strife then it may be the only way out.

The following items are needed to complete the project:-

I have broken this into Section A, Section B, and Section C.

Section A items are needed only if you don't have an existing power supply capable of delivering +5 volts at around 300mA

I have used Dick Smith Catalogue numbers on some components only for reference purposes. You may find other outlets more convenient.

SECTION A. (External power components.)

Plug-pack 9 volts AC or DC at 400-600mA. The current required to run the full 8 memory chips is just under 300mA, but I would recommend a larger supply than this as a safety margin. PBUFF will accept AC or DC input. Experienced builders may prefer to use a suitable AC transformer.

(DSE Cat M-2155 9.5VAC multi-tap @ 1A)

IF YOU WISH TO ADD A SERIAL BOARD LATER ON, THEN A 9 VOLT AC SUPPLY MUST BE USED SO THAT -12 VOLTS CAN BE GENERATED FOR IT'S RS232 CIRCUITS.

Diodes D1-D4 1N4002 (4)

Capacitor C3 10uF Tantalum @16 Volts.

Capacitor C4 2500uF Electrolytic @16 Volts. (PCB type mount.)

7805 +5 Volt regulator in TO-220 case.

A "U" shape type heat sink to suit regulator.

Section B. (Main board components.)

Capacitors BC1-BC11 .01uF Ceramic (11) (bypass caps)

Capacitor C1 220pF Ceramic

Capacitor C2 22uF Tantalum @16 Volts.

Capacitor C5 150pF Ceramic

Capacitor C6 680pF Ceramic

Capacitor C7 ~~.1uF Ceramic~~ .01uF Ceramic

Capacitor C8 .01uF Ceramic

Resistor R1 100K

Resistor R2 270 OHMS

Resistor R3 10K

Resistor R4 ~~100 OHMS~~ 330 OHMS

Resistor R5 1K

Resistor R6 4K7

Resistor R7 100K

Resistor R8 1K

Resistor Network SIP package, 7 x 4K7 resistors with one common pin. (8 pin device)

280A CPU

8255 Programmable Peripheral Interface.

74LS00 Quad 2-input nand gate.

74LS04 Hex inverter.

555 Timer I.C.

Crystal 3.58MHZ in a small case. (DSE Cat K-6031)

Transistor Q1 BC548 Any NPN Small Signal Amp will do. (BC108, BC148,

BC208, or DSE CAT Z-1308 DS548)

VERSION 2.2 ONLY.....

4164(s) 64K dynamic rams. One required for 8K, two for 16K, four for 32K, and eight for 64K. Any type are suitable, as the software provides for both 7 bit and 8 bit address refresh. This includes Texas Instruments TMS 4164's.

VERSION 3.2 AND LATER.....

41256(s) 256K dynamic rams. Eight required.

Section C. (Misc. components.)

Chip sockets:- My usual rule is to fit sockets to all 40 pin devices, all ROM, and all RAM. This only leaves the two 74 series chips and the 555 timer without sockets, so my advice is to fit sockets for all I.C.'s. If you experience problems debugging the hardware of this project then sockets will be a life saver.

Sockets required:-

40 pin (2)

28 pin (2)

16 pin (8) 64K (4) 23K (2) 16K (1) 8K

14 pin (2)

8 pin (1)

If you are going to expand your memory at a later stage, then it would be advisable to fit the full eight RAM sockets now.

Version 3.2 and later..... The full 8 sockets must be fitted. 256K (8)

Two push button (make) momentary switches for the RESET and TEST buttons.
SPST switch for the COPY switch. (Optional hardware pause SPST switch.)
One red LED 5mm, and LED mounting bezel. (DSE Cat Z-4030, and H-1910)
A suitable case. I have designed the board mountings to fit the Dick Smith Cat H-2505. This is a superb plastic instrument case at a low cost.

The Centronics input and output wiring can be done in a variety of ways. I have designed a small board to assist you in making these connections to the J2 header point. This board is the BPIO board. (Back Panel input/output) See my BPIO ad for detailed information on this board.

The equivalent BPIO circuit can be built using DB25 input and output or Centronics input and output, or any other way you wish, just make sure that the circuit basics are followed.

If you are going to connect a computer or printer switch, or serial board later on, then I strongly recommend that the header principle, and flat ribbon cables be used for input/output cabling to J2.

ASSEMBLY:-

Can't think of anything else that we need, so if you have all of the above parts let's get to it. With your multi-meter check that ground and +5 volts are not shorted together. As you progress along, do the same test for +5V and ground shorts at regular intervals, as these two signals go to every chip, and at least one signal to almost every other component, they are the most likely to short due to soldering faults. They are also the hardest fault to find and isolate when all your sockets are installed.

Have a look at the Printer Buffer Overlay. This is drawn from the component side of the board. Now look at your printed circuit board. My name is on the component side of the board, the other side is the solder side. The words "MADE IN AUSTRALIA" can be seen on the solder side of the board.

Examine the board on both sides for defects before assembly. Any fault on the solder side can be easily fixed, but once you have your sockets in, the component side of the board is impossible to fix without removing these sockets, so pay particular attention to the component side artwork before assembly. Check any suspicious sections with a meter.

Install the I.C. sockets into the board. Take note of the orientation of pin 1 of the I.C.'s, as I have not arranged the direction of the chips in a logical fashion. This has been done to minimize the size of the board.

The RAM software drivers require the most significant bytes to be installed first, that is, an 8K buffer needs only the data bit 7 RAM chip installed. This is shown as E8 on the overlay. Capacitor BC8 is the bypass cap. for E8

Best I give you a table at this stage for the various memory sizes.

RAM chip configuration:-

4164(s)	Capacitor(s)
8K install E8	and BC8 (Data bit 7)
16K install E8, E7	and BC8, BC7 (Data bits 7 and 6)
32K install E8, E7, E6, E5	and BC8, BC7, BC6, BC5 (Data bits 7, 6, 5 and 4)
64K install all RAM chips	and all bypass capacitors.
256K (Version 3.2 and later)	install all RAM chips and all bypass capacitors.

Once you have determined which sockets are to be installed and have them in position, they can be soldered in.

Install the resistor network into the board as shown on the overlay, that is with pin one towards the dynamic rams, and solder in.

Install all the resistors as shown on the overlay into their mounting holes.

They can now be soldered in and the legs cut off flush with the solder joint.

Install the BC capacitors. (Refer to RAM chip configuration.) Install C1, C5, C6, C7, and C8 capacitors. Solder these caps in and trim off the legs as before.

HEY !!!! Instead of me saying solder in component "X" and trim off the legs, you just add "AND TRIM OFF THE LEGS", every time I mention "SOLDER IN", OK?

Capacitors C2, C3, and C4 are polarized and must be soldered in the correct way around. These caps. have a plus sign (+) on one end of them, or in the case of C4, a minus sign (-) on one end. Solder them into the board as per the overlay.

Install diodes D1-D4. You will notice a bar up one end of the diodes. Make sure that these are also mounted the correct way as shown in the overlay. Solder in these components.

Solder in the 7805 regulator. The words "7805" must be facing towards the diodes. The heat sink can be fitted to the 7805.

Position transistor Q1 into the board. If you use a DS548, the "CBE" connections are marked on the case and line up with the overlay as far as the flat side is concerned. The BC548 and the others, although not marked, should mount into the board the same way. If you end up with some obscure brand that you are not sure of the pinout, try reversing the Q1 connections if the clock fails to run.

Solder in the crystal, don't push it fully into the board, but leave enough length on the legs so that it will not short out the tracks underneath.

FRONT PANEL CONTROLS.....

J3 (Control) has been set out in a 16 pin dual row header. If you wish to, a 16 pin header, crimp connector, and approx. 9 inches of 16 wire flat ribbon cable may be used for front panel wiring. Don't cut off the spare wires.
This wiring can also be done without the headers etc. as follows:-

RESET button..... Connect two insulated wires (approx. 9 inches long) from the reset button to the two points as shown on the overlay.

COPY switch... The same wiring method as the reset button, a pair of wires connected from the holes marked COPY to the SPST switch.

TEST BUTTON..... Same method as RESET button.

DATA LED..... The LED has one leg longer than the other. The longest leg is the anode, and the short leg is the cathode. Connect an insulated wire from the anode to the hole marked LED+, and from the cathode to the hole marked LED-. (See overlay.) The cathode leg can also be found by the "FLAT" edge on the LED itself.

OPTIONAL HARDWARE PAUSE.... Same as above again, but connected to the two "pause pads". A sharp 45 degree angle track juts out from the component side of J3 near the hardware pause connections. This

track needs to be cut for the hardware pause feature. This can be done with a sharp pair of side-cutters or a STANLEY knife.

All floating wiring goes into the component side of the board the same as any component.

J1 HEADER..... FUTURE EXPANSION, NOT USED FOR Ver 2.2 or 3.2 EPROM'S.

A 40 pin header can be soldered into this position to extend the Z80 bus pinout. Pin 1 of J1 is also pin 1 of the Z80, pin 2 is pin 2 etc.

J2 HEADER.....

A 26 pin male header can be soldered into the J2 position. Using a 26 pin IDC type crimp header and flat ribbon cable, connections can be made to other I/O boards or connectors.

J4 SERIAL BOARD POWER OUT.....

Used only for the SERIAL board. The AC connection is used to generate -12V on the SERIAL board. A strip of 4 male pins must be soldered into these pads if a serial board is to be used.

The ROM high/low selection is done at the point marked "H L" on the overlay. The default low condition is selected. If high needs to be selected, cut the track on the component side of the board between the low and centre pad, and connect the centre pad to the high pad. This can be done with 3 male posts and a test link if you wish.

This strap is used to select upper or lower half of the EPROM. A 2764, or larger can be supplied with 2 versions of software. EG:- Lower bank 256K Version, upper bank 64K version. This EPROM would be labeled as follows :- PBUFF V3.2/V2.2 With low selected, (normal) Ver 3.2 is selected. Changing the strap to high will select Ver 2.2 if you have a V3.2/V2.2 EPROM, then the kit can be tested in the V2.2 mode with one only 4164 or 41256 as the 41256 chips look like 4164's when V2.2 is selected.

IF YOU HAVE ONLY A ONE VERSION EPROM, THE ABOVE ROM SELECTION INFORMATION CAN BE IGNORED AS THE LOW SELECTION IS THE NORMAL (DEFAULT) BOARD SETTING.

The pads marked GND & +5 are used for test connections. Two posts can be soldered into these pads.

Don't install the chips yet. Connect up your power-pack, transformer, or +5 volt supply. The power pack connection points are shown on the overlay. As the circuit has a diode bridge at the input, it doesn't matter which way around you connect the positive and negative terminals. If you are using an existing +5 volt supply then connect the power up to the +5volts and ground "POSTS".

With your multi-meter check that ground and +5 volts are not shorted together.

Power up the unit and check that the +5 volts is there before proceeding. Check the polarity of the +5 volts.

All OK? Power down and install chips E12 (8255) and E13 (74LS04).

DON'T---DON'T---DON'T---PUT ANY CHIPS IN THE WRONG WAY AROUND. YOU WILL MOST CERTAINLY DESTROY THEM!!!!!!!!!!!!

CHECK THE OVERLAY FOR PIN 1 ORIENTATION. THE NOTCH UP ONE END OF THE CHIP, AND THE SMALL BLACK DOT SHOWS PIN 1 LOCATION. YOU WILL FIND SIMILAR MARKINGS ON EACH CHIP.

Power up again and check the +5 volts. If the LED is not already on, press the reset button. The LED should light up brightly.

NO LED?..... Then you have one of the following faulty:-

E12, E13, RESET button, reset circuit, LED, LED circuit, or more likely you have the LED wired in back to front.

LED OK?.....

Power down, and install E9 (74LS00), E10 (Z80A), and E11 (EPROM), and power up.

NOTE ***** VERSION 4.0 AND LATER.....

Additional kit builder diagnostic information and operating instructions differ from earlier Versions as the static ram and 555 interrupt tests have been added.

Version 2.2 and 3.2 read on:-

The LED should be flashing at approx. 1 cycle per second. If the LED is flashing, it indicates that the Z80A is running and the software is doing it's job, but a RAM fault exists. With no RAM installed this is to be expected. If the LED doesn't flash then move on to the "IT DOESN'T WORK DEPARTMENT".

Happy and no smoke? Good it's time to insert one RAM chip. Power down and insert a RAM chip into socket E8 (Data Bit 7) watching out for pin 1 (look at the overlay.)

COPY ON/OFF..... The copy switch is in the COPY ON position when the switch contacts are made, and in the COPY OFF position when the switch contacts are open. Switch to COPY ON, power up and check your volts again. At this stage, the LED should be flashing at approx. 3 cycles per second, which indicates that RAM is present and can be written to and read from.

Switch to COPY OFF. The LED should go out. Press the reset button. The data LED should light up brightly. Release the reset button and the LED should go out. This indicates that the software is working.

All looking OK? Power down, install additional RAM chips to suit your memory requirements, (Version 3.2 all eight 41256's must be installed.) and power up again. Repeat the above simple test procedures using the COPY and RESET switches. A simple RAM test is performed before the LED will flash. This test may not show up a shorted track, or a faulty RAM in some cases. If you have a suspect RAM, it can be further tested on it's own in the DATA bit 7 position by setting COPY ON, power up, and the LED should flash at approx. 3 CPS. If at any time the LED flashes at approx. 1 CPS, then a RAM fault has been detected.

LOOPBACK TEST ...

What still happy? Power down, connect the output port pins 1 (STROBE), 10 (ACK), and 11 (BUSY) to the input port pins 1, 10, and 11. This can be done by simply plugging the input and output of your IBM printer cable into the BPIO board. If your input and output can't readily be connected together, then 3 jumper leads can be used to connect pin 1 to pin 1 etc. When this has been done, set COPY OFF, power up, and press TEST. The TEST function outputs a sign on message to the printer. In this case, it outputs a string of characters to it's own input port that keeps cycling through the buffer until RESET is pressed. The TEST function will light up the DATA LED to about half brilliance.

Still happy? Power down, connect a printer, set COPY OFF and power up and press TEST. If all is well you should get the sign on message complete with memory size information. When the test facility is used, a ROM check is also done. If any byte in the ROM is incorrect then an error will be reported on the printout.

Power down, have a quick read through the OPERATOR INSTRUCTIONS, connect your computer to the input plug. Power up, and try printing a file. If either works OK, or doesn't work at all. If it works then you can slap it in a box, and forget it. Bob's your Uncle. If it doesn't work then go on to the "IT DOESN'T WORK DEPARTMENT". Hey, that's now.

BUT FIRST, TIME FOR A CAUTION !!!!!!! WHEN DRILLING THOSE HOLES IN THE FRONT OF YOUR CASE, MAKE SURE THAT THE FRONT PANEL HARDWARE DOESN'T INTERFERE WITH THE COMPONENTS ON YOUR PBUFF BOARD.

"IT DOESN'T WORK DEPARTMENT."

YES, with the full RAM installed, the 7805 will get warm enough to burn your finger, but no other component should.

I don't intend to hand feed you a step by step "POWER UP, POWER DOWN, POWER UP" type format. Any time you plug or unplug any connector or chip then all power must be turned off. Disconnect the input and output plugs.

Let me outline the five major faults found in kits:-

- (1) Bad socket solder joints. (open circuit)
- (2) Faulty socket connections. (open circuit)
- (3) Solder shorts between tracks. (short circuit)
- (4) Point not soldered. (open circuit)
- (5) Artwork or tracks faulty. (open or short.)

Faults 1 and 2 can usually be found with a logic probe. Any pin on any chip should be in one of three states. High, low, or bobbing. Well almost every pin. If no LED's light up, this is known as a high impedance state. Check over all chips for ground, +5 volts and other signals. Make sure that there are no high impedance states anywhere.

Exceptions to the rules:-

The input and output ports of the 8255 could show anything random as this may not get programmed correctly on power up.

The data bus may also show high impedance states under certain conditions. Pressing the reset button will bring some life into the data bus.

If a high impedance state is found where it shouldn't belong, then check with your meter for a connection between the solder pad under the board and the pin of the chip for zero ohms. If you find no connection then pull out the chip and check between the solder pad and the socket connector itself. Still no connection? Try soldering this point again. If this doesn't work then you could be up for a complete socket replacement. A nasty job for a newcomer.

Sometimes, bending the legs of the chips at a slightly different angle, or just pulling and re-inserting a chip can remove socket problems.

Faults 3 and 4 are ones that you have created. A close visual inspection and perhaps a check around with an ohm meter will usually overcome this fault.

A magnifying glass may help you.

Fault 5 is the result of my board manufacturer. Should never happen, but it does from time to time. I have threatened him with instant death if it happens again. Can be overcome using faults 3 and 4 detecting methods.

CLOCK SIGNAL CHECKS.....

With a logic probe check E10/6. This should be bobbing up and down, as this is the clock signal. No clock? Remove E10 and check E13/4 for activity. Still no clock? Check E13/3. If no clock is found here, try removing E13 and test the socket contact of E13/3. No clock? It has to be one of the following:- Crystal, C1, C6, R1, R2, or Q1. Is Q1 connected the right way around?

If you are getting the clock to pin 6 of the CPU socket without the Z80A in, and no clock with the Z80A in, then it may be the Z80A itself.

If you have a good range of spare chips then substitution may bring results.

CLOCK WORKING, BUT STILL NO RESULTS?

Remove E7 and check that the LED flashes at approx. 1 CPS. The software checks the RAM and if none is present the LED is flashed at the slow rate. If this checks out OK, then it should mean that the system is working up to the point short of a RAM check. Install E7, set COPY ON and check for the faster flash rate. If this checks out then it appears that the system is working.

OK, where do we go from here? Perhaps you have incorrectly wired up the output plug. For instance, a high on the output port busy signal will stop any printout.

When the input and output plugs are connected together, COPY OFF, and TEST pressed, pins 1, 10, and 11 should be bobbing.

Press the reset and check the MREQ line, Z80A pin 19. If it is bobbing then the CPU is attempting to read the ROM.

Press the reset and check the IORQ line, Z80A pin 20. If this is active then the CPU is attempting to access the 8255 ports.

Check all ROM and RAM signals. All pins except ground and +5 volts should be active.

The CPU... Activity on most pins. Reset can be checked using the reset button. Pin 26 should be low when the reset button is pressed and high when the reset button is released.

E13/8 should show the inverse levels to the CPU/26. (See the reset circuit diagram.) The 8255/35 should be the same levels as E13/8.

As mentioned before, the DATA LED should light up when the RESET button is pressed, and go out when released. This is an indication that the reset circuitry is working correctly.

WAIT (24) should be high. BUSREQ (25) high. INT (16) high. A13 (3) bobbing after reset means that the CPU is trying to send information to the printer. NMI (17) high, goes low when TEST pressed.

QUINELLA TEST...

What would happen if you had a short between Address 7 and Data bit 3, or A13 and ground, or etc., etc., I don't know, but I dare say the beast won't work at all. How do you find these shorts?? Do what I call a quinella test.

Using your meter, check for zero ohms between pin 1 and pin 2 of the Z80A. Then check pin 1 and 3, 1 and 4, right up to pin 1 and 40. Step on to pin 2 and 3, 2 and 4 etc., up to pin 2 and 40. Get the idea? You end up checking pin 39 and 40. Zero ohms will be found at some points, as these points are tied together. Check the circuit to verify these shorts. If a short is found where it shouldn't be, you have found your problem, well at least you know which tracks to look at. This quinella test can be done on all chips if no fault is found at the Z80A. Try the 8255 next.

If your printout is almost correct, but you get things like upper case "A" changing to "B" and "1" changing "2" You have data bit zero set all the time. "A" is 41Hex, and "B" is 42Hex, so data bit zero is never clear. You will need an ASCII to Hex lookup table to sort this one out. If you don't understand the Hex jumbo, ASK someone who does. Once you have determined which data bit(s) is faulty, try swapping the chip in this ram position with another to see if the fault conditions change. This will prove the ram chip. In most cases it will not be the ram chip, but your soldering or a faulty socket at the "BAD" location.

I could write a hundred pages and still not cover your problem, but with the above information and a little common sense, you should be able to overcome all but the really obscure fault.