Dragon 32 Rev 2 Build Instructions

Introduction

This is a rough guide on how to assemble the Dragon 32 Rev 2 reproduction board. It assumes you have collected together all the parts you will need using the bill of materials in the project repository.

Preparation

Clean the mainboard to remove contaminants, soapy water first, dry and then a wipe down with isopropy alcohol.

Make sure you have a good supply of solder, a temperature controlled soldering iron. If you are using lead based solder set the temperature to between 360C and 370C. The board can tolerate higher for lead-free solder.

A ready supply of solder flux can be really helpful if you find yourself having to remove a component from the board.

You will need short tipped, angled wire cutters for trimming the component leads after soldering. Don't cut the leads right down to the board or deep into the solder.

It is also a good idea to have a multimeter on hand to check component values and test for any shorts after soldering.

The project repository on GitHub includes an interactive Bill Of Materials that allows you to cross-reference components against position on the board (and mark off all the parts that you've fitted). To use this download the ibom.html file to your computer and open it from the local file system (this should open up a web browser).

If you can't or don't want to use the iBOM it is a good idea to print off the Dragon32__Assembly PDF on to large form paper and mark the positions off from there as they are fitted to help avoid getting lost.

Build Order

Work with the components with the lowest profile first - resistors, then diodes, then capacitors, transistors, etc.

Leaving the chip sockets until last helps as they tend to get in the way when fitting other items.

PAL/NTSC configuration

If you are setting the board up to use NTSC you need to start by modifying JP15/16 and JP17/18. By default these are pre-bridged for PAL use. You need to cut the trace joining pads 1 and 2 on both jumpers. Pad 1 is marked with a triangle.

Once the trace is cut, use a blob of solder to join pads 2 and 3.

To use NTSC you need to provide an adapter to the VDG that includes an MC1372 NTSC modulator chip. This replaces most of the discrete logic on the board.

The output of the adapter board is wired to PL8. Ground on the left, composite NTSC video on the right.

The rest of this build assumes you are assembling for a PAL machine.

Resistors

Fit resistors first making sure the solder is forming on both sides of the board, not just the bottom.

Regardless of board configuration all of the resistors should be fitted.

Save the two potentiometer resistors RV1 and RV2 for later.

Diodes

Fit all 9 diodes in much the same way as the resistors but making sure the polarity marking matches the board.

Inductors

If you want to have the option of two button joysticks (non-standard but used in some CoCo games) then you'll need to fit all 4 inductors (L1-L4).

If you only want single button joysticks you only need to fit L1 and L2.

Soldering is the same as for resistors.

Capacitors

If you are using 4164 DRAM chips you need to miss some components out. C12 and C50-C57 must not be fitted - failing to do this will result in corrupted memory read and write.

Other than this fit all capacitors making sure that the tantalum and electrolytic capacitors are fitted the right way around.

Tantalum capacitors have a + symbol next to the positive side. Electrolytic capacitors have an arrow pointing to the negative side.

The board is marked to show a + next to the positive side and a filled semicircle on the negative side.

It is worth saving two of the long legs after trimming the legs back. You can use these for securing the crystals.

Save the trim pot capacitor C7 for later.

Transistors

TR1 is an NPN transistor driving the coil of the cassette remote relay. It uses a wide TO-92 footprint.

TR2 and TR3 are NPN transistors in the video amplification circuit.

TR4 is a PNP transistor acting as the final amplifier in the video circuit.

TR2-TR4 all use a narrow TO-92 footprint.

It may require some leg bending to form your transistors to the right spacing. This is best done with a pair of tweezers to reshape the legs as needed. Trying to do this by hand is likely to result in snapped legs.

Make sure the transistors are fitted the right way around according to the outline printed on the board.

Jumper Posts

The board has 7 configuration jumpers. All of these are 3 posts. Soldering them can be tricky when trying to get them aligned vertically. A simple approach is to only tack a single pin and then adjust the angle until happy.

If you know the configuration you are going for and do not wish to change it later you can safely use a piece of wire to bridge the desired contacts.

This is fine for jumpers JP1/2 through to JP9/10 but it is recommended to fit the three post headers for JP11/12 and JP13/14.

Connectors

The keyboard connector can be vertical or a horizontal as desired. Original boards used a vertical orientation but later boards and the D64 moved to horizontal to avoid clashing with the keyboard itself. I recommend using the horizontal option as it leaves more space for connectors.

The joystick and cassette port connectors have an overlapping design, if you want to get them nicely lined up start with the 5-pin cassette socket and tack it in place on a single pin.

You can then do the same with the two joystick ports (6-pin or 270deg 5-pin) and align them to the edge of the board. This can be a tricky task so don't rush it.

The power board connector needs to be fitted with the vertical guide towards the bottom of the board (assuming you don't have a plain pin header).

The cartridge port connector can be very entertaining to fit. The pins like to bend in transit so you'll likely find yours do not line up on the first attempt. It is very easy to bend the pins on the cheaper connectors resulting in the pin not fitting properly. Double check that all pins have fitted through the mounting holes on the board before soldering.

Variable Components

C7 should fit easily but some varistors have legs that can split into multiple layers. Make sure that all of the leg makes it through the mounting holes.

RV1 needs to be fitted at an angle so that the adjustment side is facing more upwards. The long, flexible legs of the part make this relatively easy to achieve. If you fit it flush and then trim the legs before realising you can't easily adjust it, you will need to obtain a replacement or work with it as-is.

RV2 has a top adjustment and can be fitted flush to the board but the legs of the component need to be checked. Some versions of the part have the legs in-line while the board expects the middle leg to be offset. As with RV1 the legs are flexible and be easily formed to the right configuration.

The orientation of RV1 and RV2 is not critical, it just changes which way you turn the adjustment.

Crystals

Assuming you have the larger housed crystals you will need to bend the legs so the crystal lies flush with the board. This is not essential but it is recommended. Two additional solder points are provided for each crystal to allow a wire to be formed over the body and prevent the crystal being moved. A convenient source for the wire to do this is the long legged capacitors you hopefully saved earlier.

Sockets

When fitting the sockets make sure the orientation is correct and the notch of the socket matches the outline on the board.

Most of the chips are aligned with the notch to the top of the board but RAM, ROM and SAM chips have the notch towards the bottom of the board.

As with the header pins it can be a little tricky to get the sockets to fit flush and aligned. Tacking one pin at each end of the socket to hold it in place and then heating one pin while adjusting the position gives a good result.

Be very careful to avoid bridging the pins on the socket while soldering. It is worth checking with a multimeter to double check as the pins can bridge on the top of the board, hidden underneath the socket.

If you are using original ROMs for U17 and U18 it is worth fitting regular double wipe leaf sockets. If you are using EPROMS in adapters you can

either fit the adapters straight to the board (definitely recommended for square pins) or fit turned pin sockets (only for round pins on the adapter).

Before Fitting Chips

As a precaution, before fitting the chips to the board it is worth making sure none of the voltage pins of the power board connector are shorted to ground (that would be bad).

The power connector has the voltages as +12v, +5v, -5v, Gnd going from right to left. You don't need to connect to the power supply to test, you just need to check continuity between the power pins and the different power pins on the various sockets.

If you fit your configuration jumpers next to the power connector (JP1/2 to JP9/10) you can make sure the right voltages are available on the board by checking for continuity from the respective power pins.

The +5v pin is connected to almost every single chip on the board as Vcc. On the 74LS logic chips this is the upper right pin. The lower left pin is connected to ground.

The RAM chips take a little more attention. For 4164 chips there are only two power pins, +5v and ground. +5V should be the top right pin. Gnd should be the lower left pin.

The LM1889 modulator IC9 needs different voltages, -5V to pin 5 (5th pin down on the left side) and +5V to pin 14 (5th pin down on the right side).

The amplifier at IC25 needs +12V on the top right pin.

The amplifier at IC10 needs -5V on the lower left pin and +5V on pin 7 (2nd pin down on the right).

You should also have +5V on the left pin of R67. You should have -5V on the left pin of R69. These are the top and bottom resistors just to the left of the ram bank.

Bonus Jumpers

JP11/12 is used to disable the inverter driving the strobe pin of the printer port. While this is "on" (11) the strobe pin will not work for input. If you

want to use the printer port for bit banging a serial port you will need to disable the inverter by setting the jumper to the 12 position.

JP13/14 defines whether the board is working in "half-good" 4164 mode or full 64k mode. If you only want your board to work as a D32 then set this to 13, if you want to be able to access the full 64k memory mode (which will break your ROM access) then set the jumper to 14.

In effect JP13/14 is the part of the memory upgrade mod that requires bodging and cutting traces between chips.

Fitting Chips

Make sure you have a grounding strap on your wrist (or ankle) and it is actually connected to a proven ground point (radiators are a good source). The LM1889 and DRAM chips are very susceptible to static discharge! Leave these until last.

Fit all your chips paying very careful attention to orientation, notches must match with the board. Fitting chips the wrong way round will likely destroy them when the power is switched on.

ROM Chips

If you have original ROMs you fit these directly to sockets. EPROMs require an adapter board which probably has a socket on top so the adapter can be directly soldered to the board, unless you want to keep changing them. If you are going to chop and change between original and EPROMs I recommend fitting turned pin sockets and creating a sandwich of regular leaf pin and turned pin sockets for the ROMs to fit in. This will avoid damaging your valuable ROMs and maintain good contact to the board.

First Power Up

Before turning the power on for the first time it is worth removing the LM1889 modulator, it provides the colour signal but the video will work in monochrome without it.

When powering on for the first time always be prepared to turn the power off in a hurry. If it doesn't show a picture on your monitor it does not mean

it isn't working, you may need to adjust RV1 to bring the output into an acceptable voltage range

If anything smokes or looks to be getting hot, turn the power off immediately. You most likely have a short (you did check for shorts?) or a polarised capacitor in backwards.

Assuming nothing goes pop you should be able to adjust the video output to get a stable picture. If all goes well, turn the power off and reinsert the LM1889 modulator.

Try the video again and you should be welcomed by the regular green screen. You can adjust the video timing by turning the head on C7. This is best done with a plastic screwdriver, not a metal one. When set correctly you should be able to just about freeze horizontal dot crawl but won't be able to remove horizontal artifacts.

Troubleshooting

No Colour

In most cases this will be a failed LM1889. Test the clock signal on PIN 17 of the modulator. If you can see a good 4.3MHz signal you should be seeing colour. Check RV1 is set appropriately and that the colour signal is arriving at R7 (lower leg, not the top one) and again at R10 (left leg, not the right).

If the signal is getting that far then you likely have a bad TR2 (or the wrong component, or a short between pins on TR2 - easily done).

If the signal is not getting to the test points, check for continuity.

If you are only getting clear colour on alternating lines the problem will sit with the amplifier at IC10. This mixes the signal for alternating lines. The chip is pretty robust but I've seen failures during testing that left me scratching my head...

Garbage Picture

There are many, many ways that this can happen. The most common cause is a stuck data line, either due to bad RAM/ROM or a short between data lines or address lines.

Test your RAM - either in a known good device or with a dedicated RAM tester (beware that some testers can show a failure for good chips due to differences between TTL and CMOS). The ROMs on the Dragon are generally robust but the same chip type is a common source of failure on other computers (cough Commodore). It is worth checking the legs are making good contact with no sign of corrosion (blackening).

No Response At All

If the relay doesn't click on powering up (assuming you fitted one) and there is no picture at all, the chances are the CPU is not cycling. Either it hasn't come out of reset or is being locked up by another line.

A bad PIA can cause the CPU to run very, very slowly by generating interrupt requests at high frequency. Try removing the PIAs and try running without them.

Check the reset line is going high (about 5V) and the same again for halt, mrdy, extal, xtal, bs and ba pins. If any of these are low it can stop the CPU operating.

Check the Q and E pins are oscillating at (about) 0.9MHz, no clock means no CPU. You can also check the SAM is receiving the full 14MHz clock from the crystal.

If all is well with those pins, test the address and data lines with an oscilloscope or logic probe. You should see activity with the lines moving from high to low.

If the CPU is doing nothing it may be faulty, either test the CPU in a known good machine or try a different CPU.

If the CPU is proven good you can turn your attention to the SAM, this governs the CPU activity and memory addressing. I've yet to see one fail but if it does you have no hope of getting the computer to work until it is replaced. More likely the SAM is not getting the input signals it needs to operate.

Check the clock input and the Q/E output of the SAM. If the clock is present and correct, check the SAM is generating RAM addressing signals on the Z lines and RAS/CAS lines.

The SAM also decides what device is being addressed it does this though a multiplexer at IC33. Check the operation of this chip carefully. Also make sure you have a jumper clip on JP13/14 as a missing signal here will disable the multiplexer!