Digital Signal Processor . . . Two-way Active Crossover . . . Eight-channel Parametric Equaliser . . .

IT'S ALL OF THESE...
and more!

We introduced our new, very versatile hifi stereo digital signal processor (DSP) last month. As we said then, it is a monster project, built with seven modules. Based around a powerful 32-bit PIC processor and high-quality analog-to-digital (ADC) and digital-to-analog (DAC) converters, it can be used as a two-way active crossover and/or a multi-band parametric equaliser – and much more! In this second instalment, we finish describing the circuit and present the parts list and board assembly instructions.

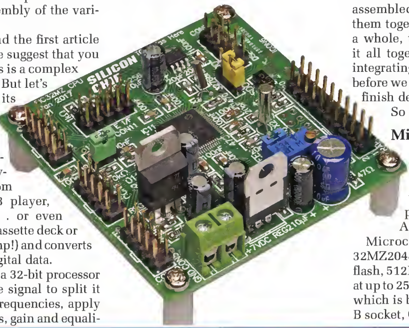
We rather left you hanging at the end of the article last month, because we didn't have room to describe all the circuitry in this advanced device. We'll rectify that shortly, covering the CPU board and some extra bits and pieces before we get into the assembly of the various modules.

If you haven't read the first article in the May issue, we suggest that you do so now, since this is a complex and capable design. But let's just briefly revisit its capabilities before continuing the circuit description.

This device accepts a stereo line-level audio signal (from a disc player, MP3 player, smartphone etc . . . or even *[cough splutter!]* a cassette deck or turntable with preamp!) and converts it to high-quality digital data.

It then sends it to a 32-bit processor which processes the signal to split it into high and low frequencies, apply any necessary delays, gain and equalisation before feeding the results to two hifi stereo DAC boards.

These convert the digital signals back into two pairs of stereo signals which can then be fed onto individual power amplifiers for the woofers and tweeters.



It's controlled using a graphical LCD, rotary encoder and two pushbuttons and the configuration is stored in an EEPROM chip, so you don't have to set it up each time.

For flexibility, It's built using seven distinct modules. Once you've assembled these, you can connect them together and test the system as a whole, then start work on putting it all together in a proper case and integrating it with a hifi system. But before we get to that stage, we need to finish describing how it works.

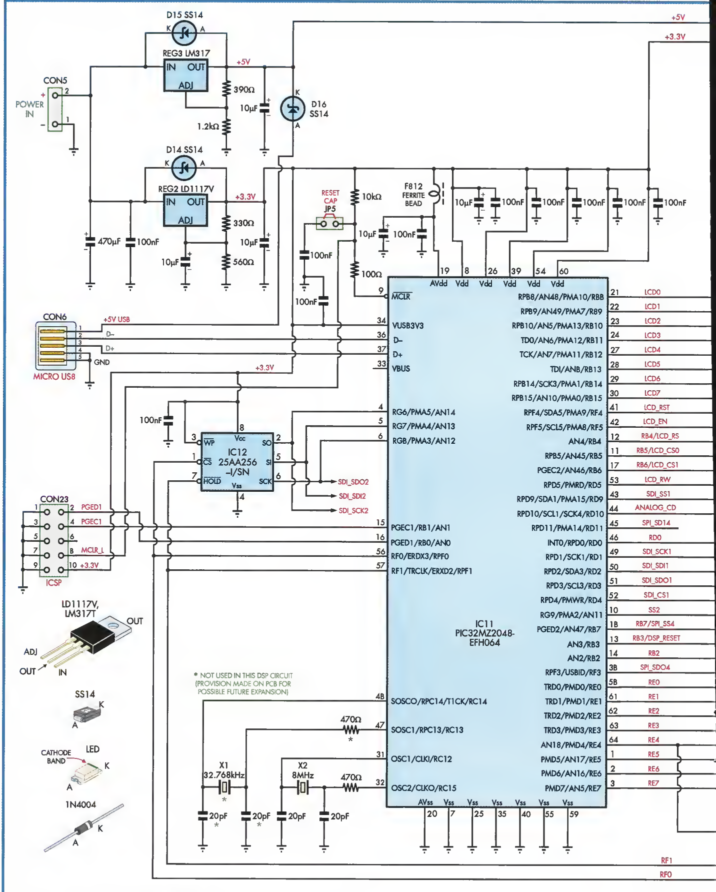
So let's get back to it.

Microcontroller board

The circuit of the microcontroller board is shown in Fig.7. This is designed so that it can be used in other projects (just as you can the ADC and DAC boards).

Microcontroller IC11 is a PIC32MZ2048 32-bit processor with 2MB flash, 512KB RAM and which can run at up to 252MHz. It has a USB interface which is brought out to a micro type-B socket, CON6, although we haven't

Part II – Design by Phil Prosser . . . Words by Nicholas Vinen



SC ©2019 **DSP CROSSOVER** CPU BOARD CIRCUIT

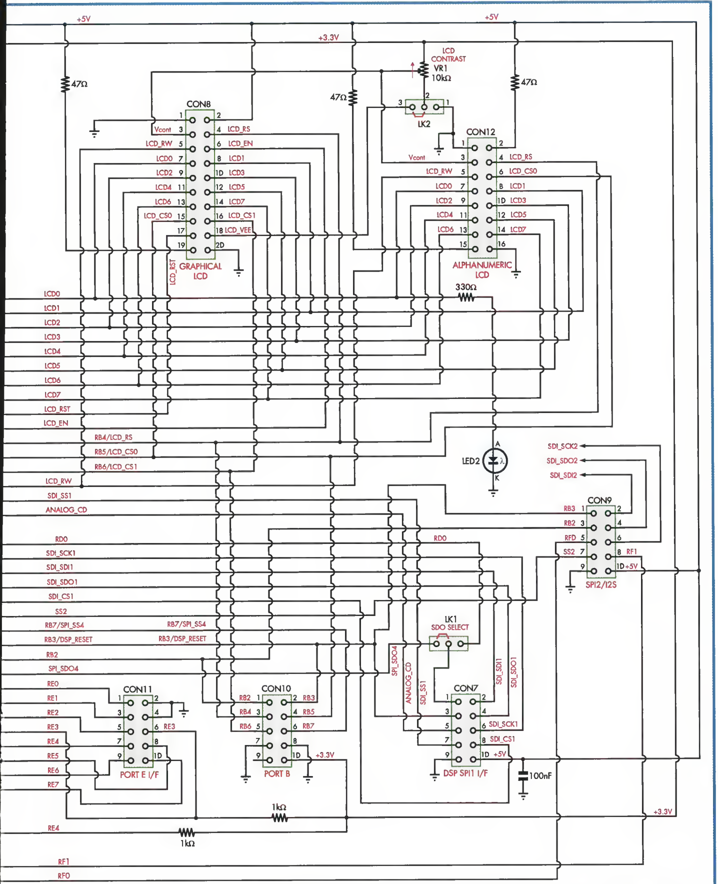
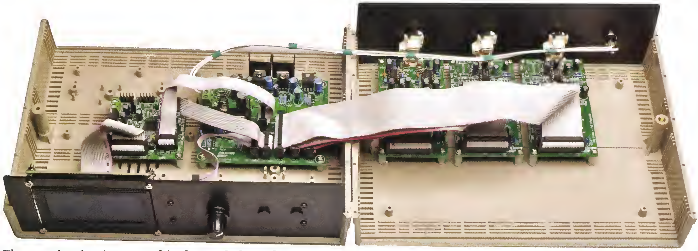


Fig.7: the CPU board is based around 252MHz/330MIPS 32-bit processor IC11, which performs all of the I/O and DSP tasks internally. Besides connectors to go to the other components, the board carries serial EEPROM IC12, two crystals and a power supply for the PIC. The graphical LCD is connected via CON8



The completed unit mounted in the two halves of an instrument case. An alternative would be a 2U rack-mounting case.

used it in this project – it's there 'just in case' for other projects.

The PIC is also fitted with an 8MHz crystal for its main clock signal (X2). Provision is made on the PCB (and shown in the circuit) for a 32.768kHz crystal for possible future expansion but they are not used in this project and can be left out.

There is also provision for an on-board serial flash (IC12) which is connected via one of the hardware SPI ports.

Two of the other audio-capable SPI ports are wired up to CON7, which connects to CON17 on the power supply/signal routing board (described last month), and therefore ultimately to the ADC and DAC boards.

LK1 allows two different pins to be used for SDO4 (serial data output #4); this function can be internally reconfigured in IC11, and since some functions are shared, there may be times where you want to use the alternative pin.

CON11 on this board connects to CON18 on the power supply/routing board and feeds the master clock (MCLK) through to the ADC and DACs, from output pin RE5 of IC11. As mentioned earlier, the other I/O pins connect to the front panel control board. Its circuit is shown in Fig.8. It carries two pushbutton switches and a rotary encoder, which are used to scroll through menus and make selections.

The user interface is displayed on a graphical LCD, which is wired up to CON8 on the micro board, via a ribbon cable. This provides a reasonably standard 8-bit parallel LCD drive interface. The eight LCD data lines (DB0-DB7) are driven from a contiguous set of digital outputs of IC11 (RB8-RB15). This allows a byte of data to be transferred to the display with just a few lines of code and minimal delay.

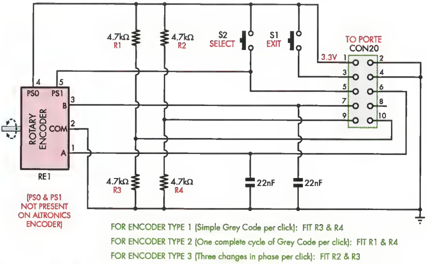
The other LCD control lines are driven by digital outputs RB4, RB5, RB6, RD5, RF4 and RF5 and the screen is powered from the 5V rail, with the backlight brightness set with a 47Ω resistor. LCD contrast is adjusted using trimpot VR1, which connects to CON8 via LK2.

LK2 is provided so that VR1 can also be used to set the contrast on an alphanumeric LCD, which can be fitted in place of the graphical one and controlled by same pins (via CON12). But again, we are not using that in this project. As we said above, this board is intended to be generic, so it has a few options we are not using.

CON23 is a somewhat unusual in-circuit serial programming (ICSP) header. It has a similar pinout to a PICkit 3/4 but not directly compatible; it's designed to work over a longer cable. Since each signal line has at least one ground wire between it, signal integrity should be better.

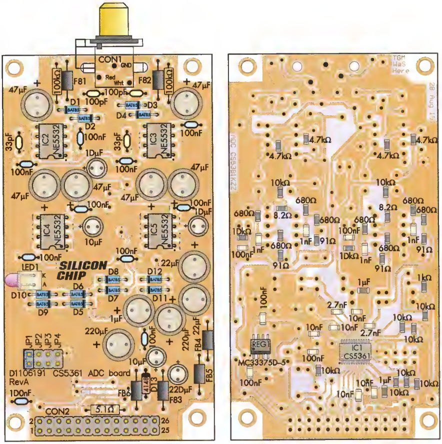
Jumper leads could be used to make a quick connection to a PICkit to program the microcontroller the first time. Or you could attach a 10-pin IDC connector to the end of a ribbon cable and then solder the appropriate wires at the other end of the cable to a 5-way SIL header as a more permanent programming adaptor for development use.

There are two regulators on the board; REG3 derives a 5V supply from 7V+ DC applied to CON5, which is used to power the LCD screen and is



SC ©2019 **DSP ACTIVE CROSSOVER** FRONT PANEL CIRCUIT

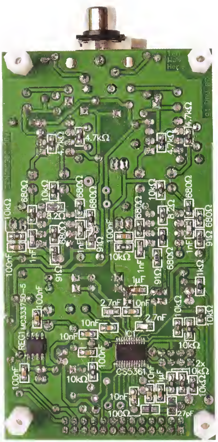
Fig.8: the front panel circuit is elementary. Two momentary pushbuttons and a quadrature (incremental) rotary encoder to CON20, which is wired back to the signal routing board and then onto the PIC32. Different combinations of resistors R1-R4 are fitted so that the CPU knows what sort of signals to expect from the rotary encoder. The two capacitors help to debounce the encoder's digital outputs.



TOP OF BOARD

UNDERSIDE OF BOARD

Fig.9: the ADC board has components on both sides; SMDs on the bottom and through-hole components on the top. Be careful with the polarity of the ICs, REG1, D1-D13 and the electrolytic capacitors. Note that diodes D1-D12 do not all face in the same direction...



... and here's the underside photo to assist you with construction (the top side was shown last month). The use of IC sockets is optional but highly recommended – just in case, just in case!

also fed to CON7 and CON9. REG2 is used to produce a +3.3V rail from the same source (CON5), to power microcontroller IC11 itself.

However, note that in this project, we're not feeding power in via CON5. Instead, the 5V supply comes from the main power supply board over the ribbon cable to CON7. It then powers the LCD screen and flows through schottky diode D15 to the input of REG2, which then powers REG2 and thus the 3.3V rail for the micro.

We're also not using the USB interface or USB connector CON6 in this project, nor are we using the extra microcontroller I/O pins which are broken out to headers CON9 or CON10. CON9 could potentially be used to connect another ADC and/or DAC board in other applications where more channels may be necessary (eg, a three-way crossover).

LED2 is connected from LCD data line LCD0 to ground, with a 330Ω current limiting resistor, so it will flash when the LCD screen is being updated.

Front panel board

The front panel circuit, Fig.8, was mentioned above. In addition to the two pushbuttons and rotary encoder, there are four 4.7kΩ resistors shown, but only two of these are actually fitted.

These resistors indicate to the CPU board what type of rotary encoder has been fitted and therefore how to interpret the data from it.

R3 and R4 are fitted when a standard gray code or 'quadrature' rotary encoder, which is a standard encoding method but not used by either of the encoders we tested.

R1 and R4 are fitted when an encoder is used which produces the same quadrature signals but it goes through one complete (four-pulse) cycle for each step that the encoder is rotated (ie, 11 -> 10 -> 00 -> 01 -> 11 clockwise or 11 -> 01 -> 00 -> 10 -> 11 anti-clockwise). This is the code that the Altronics S3350 rotary encoder produces.

R2 and R3 are fitted for an encoder which produces three state changes per click (11 -> 10 -> 00 -> 11 clockwise or 11 -> 01 -> 00 -> 11 anti-clockwise). This is the code that the Jaycar SR1230 rotary encoder produces. If this encoder is used, pushbutton S1 does not need to be fitted as the encoder has an internal pushbutton, activated by pressing in the knob, which is connected in parallel with S1.

The two 22nF capacitors help to debounce the signals from the rotary encoder, to ensure that it works reliably.

Debouncing is also performed in software, but it helps to have the hardware to reduce glitches at the digital inputs.

The PCB has two different mounting locations for the two possible rotary encoders, because the Jaycar SR1230 is a vertical type while Altronics S3350 is right-angle mounting.

Therefore, if using the Altronics encoder, you would either need to chassis-mount the pushbuttons and wire them back to the board, or surface-mount the encoder on the board so that it is vertical (more on that later).

Construction

Start by assembling the PCBs. We'll do that in the same order that we presented the circuit, starting with the ADC board. This is built on a PCB coded 01106191, measuring 55.5 x 102mm. The overlay diagrams for this board are shown in Fig.9.

It has parts on both sides - SMDs on the bottom and through-hole on the top, so both sides are shown in Fig.9.

It's best to fit all the SMD parts to the underside first, starting with IC1. This is the only fine-pitch part on the board. It comes in a 24-pin TSSOP package. First, identify the pin 1 dot printed on its top surface and orientate the part so that dot is towards the nearby DIL header as shown. Then put a little solder on one of the corner pads and heat that solder while sliding the chip into position.

Use a magnifier to check that all the pins on both sides are correctly lined up with their pads. If not, reheat the solder on that one pin and gently nudge the IC ever so slightly in the right direction. Repeat until it is properly lined up, then tack down the pin in the opposite corner.

Next, spread a thin smear of flux paste over all the pins, then load your soldering iron tip with a little solder and run it along the pins on one side. Stop and add more solder if you are running out and repeat until there is enough solder on all pins. Don't worry if some are bridged; we'll clean that up later. Repeat for the other side.

Now add more flux paste to any areas where you suspect there may be bridges and apply some solder wick. Wait for the flux to smoke and the solder to reflow into the wick before sliding it away from the IC. Repeat for any suspected bridges, then clean that area of the board using flux residue remover, isopropyl alcohol or methylated spirits and inspect it under magnification.

Again using a magnifier, make sure there is solder from each pin to the pad below and that none are bridged. Add a little flux and then a dab of solder to any pins which do not appear to be soldered properly. Use the procedure described above to remove any bridges. Clean and re-inspect until you are happy that all the solder joints are good.

Now move on to REG1, which has much bigger and more widely spaced pins. Use a similar procedure to solder it in place, again ensuring that its pin 1 dot is orientated correctly, ie, on the side facing the DIL header.

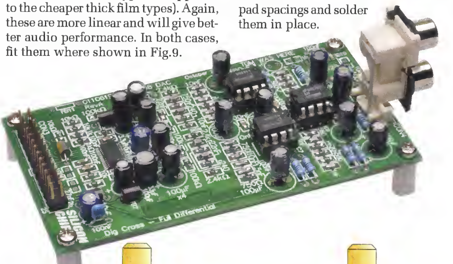
Now move onto the SMD resistors and capacitors. You can use a similar procedure – load one pad with a little solder, slide the part in place while heating that solder, check its orientation, then wait for the first joint to solidify and solder the opposite side of the part to its pad. Add a dab of flux paste to the first pad and touch it with your soldering iron to reflow that joint and ensure it is nice and smooth.

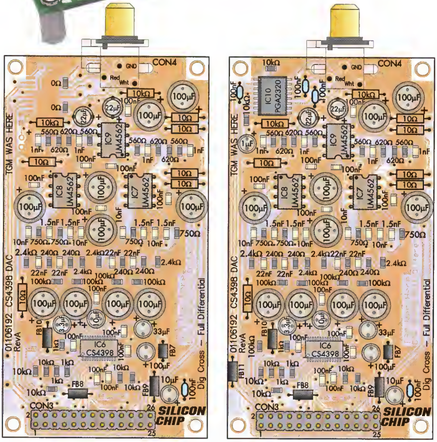
Note that some capacitors are specified as C0G/NP0 types. These are important to obtain good audio quality as they are far more linear than X5R, X7R or Y5V dielectrics. Similarly, some resistors are thin film types (as opposed to the cheaper thick film types). Again, these are more linear and will give better audio performance. In both cases, fit them where shown in Fig.9.

Through-hole components

Now flip the board over and start fitting the axial through-hole components, starting with the three resistors, then the 13 diodes. Be careful that the diode cathode stripes face as shown in Fig.9, noting that many of them face in different directions, and make sure D13 is the larger type.

Follow with the ferrite beads; if yours are just loose beads, feed diode lead off-cuts through them and then bend them to suit the pad spacings and solder them in place.





Figs.10a (left) and 10b (right): unlike the ADC board, this DAC board has a mixture of through-hole and SMD components on the top side, and no components on the bottom side. The version at the left is what's required for this project; the version at right has optional volume control IC10 fitted.

Next, solder the IC sockets in place and make sure they are orientated as shown. You could solder the ICs directly to the board, which would give better long-term reliability, but that would make it harder to swap the chips over in future if you needed to do that.

Now fit the ceramic capacitors. The 100nF multi-layer types are shown in blue in Fig.9 while the others are shown in yellow. Follow with the electrolytic capacitors, ensuring that in each case, the longer lead goes through the pad marked with a "+" symbol. You may need to bend the leads in some cases to match the hole spacings on the PCB.

Next mount the headers for CON2 and JP1-JP4. You can snap these from a longer dual-row pin header strip. Make sure they have been pushed down fully before soldering the pins.

We soldered the clipping LED (LED1) directly to the board but you could fit a 2-pin header instead, and run leads to a front panel clip indicator LED. Either way, the longer anode lead should be connected to the pad marked "A" on the PCB.

The last part soldered to the board is CON1, the dual vertical RCA socket. We found that we had to use a 2.5mm drill bit, turned by hand, to slightly elongate the holes for the plastic posts before it would fit into the board. This has the advantage (compared to specifying larger holes on the PCB), of ensuring a very tight fit which provides good mechanical anchoring for the sockets.

Once you've pushed the sockets into their mounting holes (be careful not to break the plastic!), solder the three pins. You can then plug op amps IC2-IC5 into their sockets, and shorting blocks JP1-JP4 into position, and this board is complete.

Moving on to the DAC board

Two identical stereo DAC boards are required to provide the four audio outputs in this project. You can assemble them one at a time or in parallel. The overlay diagram for this PCB is shown in Fig.10(a).

It's another double-sided board, coded 01106192 and measuring 55 x 101mm.

This time, there are no components on the bottom side, but there is a mixture of SMD and through-hole components on the top. The version on the right, Fig.10(b), shows IC10 and its associated components fitted. But those are not required for this project, so build the version at left.

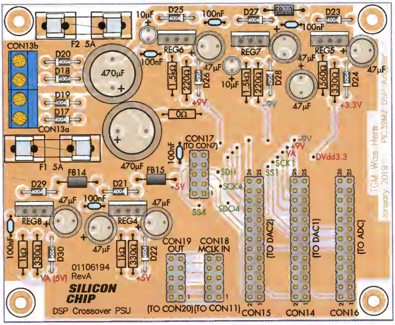
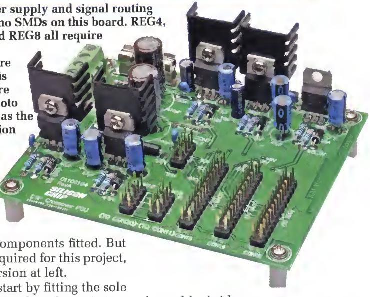


Fig.11: the power supply and signal routing PCB. There are no SMDs on this board. REG4, REG6, REG7 and REG8 all require flag heatsinks. Although they are not shown in this diagram, they are shown in the photo at right. REG4 has the highest dissipation so fit a larger heatsink to it, if possible. Also note the various test points.



Once again, start by fitting the sole fine-pitch IC to the board. IC6 is in a 28-pin TSSOP package. Use the same procedure as described above, for IC1 on the ADC board.

Then solder all the SMD resistors and capacitors, again using the same procedure as before.

Note that all the SMD capacitors with values below 100nF should be C0G types and many of the resistors are thin film types, again for linearity, to provide low distortion.

The two 0Ω resistors are soldered across pads 9 & 11 and 14 & 16 of IC10's footprint, so that the audio bypasses this chip and goes straight to the output.

Be careful to avoid shorting these pins to pins 10 and 15 in between, as those connect to ground, so you won't get any output on that channel if there is a solder bridge.

You can now fit the through-hole axial components, ie, the remaining resistors and the ferrite beads, followed by the IC sockets for IC7-IC9. Be careful with the orientation of these sockets as they don't all face in the same direction.

Next, mount the single through-hole ceramic capacitor, followed by the electrolytics, again taking care to ensure that the longer leads go to the pads marked "+". Then fit DIL header CON3, followed by dual RCA socket CON4. Again, you will probably have to slightly enlarge the bigger PCB mounting holes to get the socket to fit into the board.

Plug the op amps into the sockets, making sure each pin 1 dot lines up with the notch in the socket (check

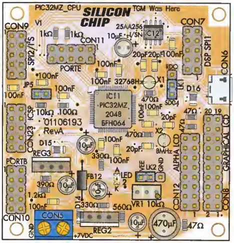


Fig.12: the CPU board uses mostly SMD parts, but there are also some through-hole parts and connectors, all on the top side. Note the orientation of IC12, IC13 and MELF diodes D14-D16. The jumpers for LK1, LK2 and JP5 are shown in their normal operating positions for this project.

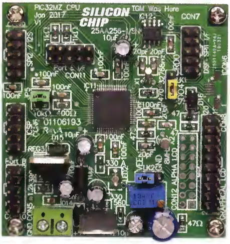


Fig.10 if you're unsure) and the DAC boards are finished.

You can then move onto the power supply and signal routing board.

Power supply board assembly

There are no SMDs on this board. It's built on a double-sided PCB coded 01106194 which measures 103.5 x 84mm.

Overlay diagram Fig.11 shows where the components go.

Start by fitting the resistors as shown, then the diodes, which are all 1N4004 types. But they face in different directions, so check carefully to make sure the cathode stripes are orientated as shown in Fig.11.

You can then mount the ferrite beads, as before, using component lead off-cuts if they do not have their own leads. You can also use a component lead off-cut instead of the 0Ω resistor.

Then fit the pin headers, ensuring that each one is pushed down fully before soldering. As mentioned earlier, these can be snapped from longer dual-row headers, as long as they are snappable types. Follow with the ceramic capacitors, then the electrolytic capacitors. In each case, the longer lead goes into the pad marked with a "+" sign.

Now solder the four fuse clips in place, with the fuses clipped into each pair to ensure that the retaining tabs are on the outside and that they line up properly.

Ideally, use a blown fuse while soldering and then replace it with the specified fuse once the clips have cooled down. You will a need quite hot iron to get the solder to flow well, and use a generous amount.

Next, dovetail the two 2-way terminal blocks together (if you don't have a 4-way block) and solder it with the wire entry holes facing the edge of the board.

Before fitting the regulators, consider how you are going to mount the heatsinks. We used 6021-type flag heatsinks but mounted them upside-down to avoid fouling components around the regulators, because we had pushed the TO-220 packages all the way down before soldering them.

We think that this will also reduce temperatures on the board, because it keeps the fins away from the board, and allows cooling air to more easily circulate.

But if you want to fit flag heatsinks 'right-way-up', you could do so by fitting them to the regulators first before pushing them down, then lifting them slightly before soldering the leads.

Note that REG4, which supplies 5V to the CPU board and for the LCD, has quite high dissipation.

If you can fit a bigger heatsink than specified to this regulator, that would be even better. But the 6021-type should be adequate. REG5 does not need a heatsink as its dissipation is quite low.

Having sorted out the heatsinking, fit the five regulators. REG7 is the LM337 negative type; the other four are all LM317s, so don't get them mixed up.

Once the regulators and heatsinks are installed, the power supply board is finished and you can move onto the last major board, which hosts the main CPU.

CPU board assembly

This board is smaller and has mostly SMD components. It's built on a double-sided PCB coded 01106193 which measures 60.5 x 62.5mm. Fig.12 shows where the components go.

Start with the CPU, IC11, which is in a 64-pin quad flat pack. Its pin pitch is slightly larger than the TSSOPs but it has pins on all four sides. Use the same basic technique, but make sure that the pins on all four sides are properly lined up on their pads before soldering more than one pin. Follow with IC12, an 8-pin SOIC package device, which is a much simpler affair.

Then move onto the SMD capacitors and resistors, followed by LED2. SMD LEDs typically have a green dot or marking to indicate the cathode, and this is on the opposite side from the anode, which goes to the pad marked "A" on the PCB. But it's best to check the LED with a DMM set to diode test mode before soldering it. If it lights up, the red probe is on the anode.

Next, fit SMD diodes D14-D16. These are schottky diodes in a MELF cylindrical package. We used "SMA" (DO-214AC) package diodes on our prototype, but they barely fit on the provided pads and are much trickier to solder. The MELF diodes will be much easier. Like through-hole diodes, they have a stripe at the cathode end and this must be orientated as shown in Fig.12.

Now you can solder ferrite bead FB12 in place, followed by pin headers CON7-CON11 and CON23. There is no need to fit a header for CON12. You can also now fit the pin headers for LK1, LK2 and JP5, followed by optional screw terminal block CON5, with its wire entry holes towards the nearest edge of the board.

Next, mount crystals X1 and X2, taking care to avoid putting too much stress on the leads as they are relatively thin. Gently bend them to fit the pad spacings.

If using a large (HC-49 style) crystal for X2, fit an insulating washer underneath it so that its metal can won't short on any of the components below, since the leads may not be stiff enough to hold it firmly in place without resting on them.

You can then install trimpot VR1, with its adjustment screw positioned as shown, followed by the electrolytic capacitors, with their longer leads to the pads marked "+".

Solder REG2 & REG3 in place, with the metal tabs orientated as shown. Don't get them mixed up as they are different types - REG3 is a standard LM317 adjustable regulator while REG2 is a special low-dropout type. Neither requires a heatsink.

Finally, insert the jumper shunts for LK1, LK2 and JP5 as shown in Fig.12.

Front panel & LCD assembly

This board has just a few components and is fitted just behind the unit's front panel, next to the LCD, allowing the rotary encoder shaft and pushbuttons to poke through holes drilled in that panel. It's built on a double-sided PCB measuring 107.5 x 32.5mm. The PCB overlay diagram is shown in Fig.13.

Start by fitting the resistors. Four are shown in Fig.13, but only two are fitted, as shown on the circuit diagram, Fig.8. For the Altronics S3350 rotary encoder, fit R1 and R4. For the Jaycar SP0721 encoder, fit R2 and R3.

Follow with the two 22nF capacitors, which should either be fitted to the underside of the board, as shown in Fig.13, or laid over on the top side of the board, so they will clear the front panel. Then solder the 10-pin DIL header in place, on the underside of the board.

That just leaves the rotary encoder and pushbutton(s). As explained earlier, if you're using the Jaycar rotary encoder (or an equivalent), it has an integral pushbutton, so you don't need to fit S2. You can still fit S2 if you want; it will merely provide an alternative way to use the SELECT function.

Also keep in mind that if you use the Jaycar encoder, this board is then mounted directly to the front panel of the unit.

But if you fit the Altronics encoder in the usual manner, ie, with its shaft parallel to the PCB, you would need to mount it differently, and that would probably require S1 and S2 to be mounted directly on the front panel and wired back to this board (two wires required for each).

To avoid that, you could bend RE2's three pins down and mount it vertically on the board, like RE1. You would need to solder stiff wire to its two mounting lugs, bend these over under the board and attach them to the mounting holes using a generous amount of solder, to provide sufficient mechanical strength.

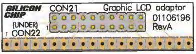


Fig.14: the LCD adaptor is dead simple and just connects pins 1-16 of DIL header CON21, mounted on the top side, to pins 1-16 of SIL header CON22, on the other side of the board. You could use a header socket for CON22, but it will be more reliable if you solder it to the LCD pin header.

Once RE1/RE2 and S1/S2 are in place, this board is finished.

Building the LCD adaptor

The LCD has a 20-pin SIL header, but it is connected to the CPU board via a 10x2 pin DIL header and DIL IDC connectors.

So we have designed a small adaptor board to make this a 'plug and play' affair. It's coded 01106196, measures 51 x 13mm and shown in Fig.14. The only parts on this board are the SIL and DIL headers.

Most suitable LCD screens have a 20-pin header with pin 1 (Vss/GND) at right (looking at the LCD screen with the connector at the bottom) and pin 20 (K-) at left. If your screen has a different pinout then you will need to come up with a different connecting arrangement.

Start by soldering a 20-pin SIL header to the LCD, on the back of the board (ie, the opposite side to the LCD screen), with the longer pins projecting out the back. Then solder the DIL pin header to the top side of the adaptor board, as shown in Fig.14.

You can then place this adaptor board over the pin header sticking out the back of the LCD, making sure that its pin 1 at left lines up with pin 1 on the LCD. Solder all 20 pins.

Making up the cables

You will need seven interconnecting cables to complete the unit, and they're also handy to have for testing, so let's make them up now. These are shown in Fig.15.

There are three 10-way cables, one 40cm long and two 15cm long; one 20-way cable, 30cm long; and three 26-way cables, 20cm, 30cm and 35cm long. Cut each section of ribbon cable to length, leaving around 5cm extra

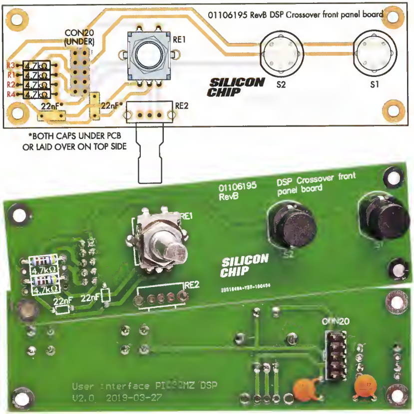


Fig.13: the front panel PCB. Note that only one of RE1 (Jaycar SR1230) or RE2 (Altronics S3350) is fitted and in the case where RE1 is used, pushbutton S2 is redundant and may be left off. Also, if RE1 is fitted, fit resistors R2 and R3; if RE2 is fitted, fit resistors R1 and R4.

PARTS LISTS

Stereo ADC input board

1 double-sided PCB coded 01106191, 55.5 x 102mm
1 dual vertical RCA socket (CON1)
1 13x2 pin header (CON2)
4 8-pin DIL IC sockets (for IC2-IC5)
1 4x2 pin header (JP1-JP4)
4 jumper shunts (JP1-JP4)
6 ferrite beads (FB1-FB6)

Semiconductors
1 CS5361-KZZ or CS5381-KZZ high-performance stereo ADC, TSSOP-24 (IC1)
4 NE5532 dual low-noise op amps, DIP-8 (IC2-IC5)
1 MC33375D-5.0R2G SMD low-dropout linear regulator, SOIC-8 (REG1)
1 5mm red LED (LED1)
12 BAT85 schottky diodes (D1-D12)
1 1N4148 small signal diode (D13)

Through-hole capacitors
3 220µF 10V electrolytic
6 47µF 25V electrolytic
2 22µF 50V electrolytic
4 10µF 50V electrolytic
1 1µF 50V electrolytic
10 100nF 50V multi-layer ceramic
2 100pF C0G/NP0 ceramic
2 33pF C0G/NP0 ceramic

SMD capacitors (all 2012/0805 X7R unless otherwise stated)
2 1µF 6.3V
5 100nF 50V
5 10nF 50V
2 2.7nF 50V C0G/NP0 5%
4 1nF 50V C0G/NP0 5%

Resistors (all SMD 2012/0805 1% unless otherwise stated)
2 100kΩ through-hole 1/4W 1% metal film
11 10kΩ
4 4.7kΩ thin film*
1 1kΩ
8 680Ω or 681Ω thin film*
4 91Ω thin film*
2 8.2Ω
1 5.1Ω through-hole 1/2W 1% or 5%
* *eg, Yageo RT0805FRE07 or RT0805FRE13 series*

Stereo DAC output board (per board, two required)

1 double-sided PCB coded 01106192, 55 x 101mm
1 13x2 pin header (CON3)
1 dual vertical RCA socket (CON4)
3 8-pin DIL IC sockets (for IC7-IC9)
4 ferrite beads (FB7-FB10)

Semiconductors
1 CS4398-CZZ high-performance stereo DAC, TSSOP-28 (IC6)
3 LM4562 dual ultra-low-distortion op amps, DIP-8 (IC7-IC9)
1 PGA2320IDW stereo volume control chip, SOIC-16 (IC10; optional - see text)

Through-hole capacitors
11 100µF 16V electrolytic
1 33µF 25V electrolytic
2 22µF 50V electrolytic
2 10µF 50V electrolytic
1 3.3µF 50V electrolytic
1 100nF 50V multi-layer ceramic

SMD capacitors (all 2012/0805 50V ceramic)
12 100nF X7R
4 22nF C0G/NP0 5%
4 10nF C0G/NP0 5%
4 1.5nF C0G/NP0 5%
4 1nF C0G/NP0 5%

Resistors (all SMD 2012/0805 1% unless otherwise stated)
2 10kΩ through-hole 1/4W 1% metal film
5 100kΩ
5 10kΩ
4 2.4kΩ or 2.43kΩ thin film*
3 1kΩ
4 750Ω thin film*
4 620Ω thin film*
4 560Ω thin film*
4 240Ω thin film*
6 10Ω through-hole 1/4W 1% metal film
2 0Ω
* *eg, Yageo RT0805FRE07 or RT0805FRE13 series*

Extra parts needed if IC10 is fitted
1 ferrite bead (FB11)
1 1µF 50V electrolytic capacitor
3 100nF 50V multi-layer ceramic through-hole capacitors
1 100kΩ SMD 2012/0805 1% resistor
2 10kΩ SMD 2012/0805 1% resistors

CPU board

1 double-sided PCB coded 01106193, 60.5 x 62.5mm
1 2-way mini terminal block, 5.08mm spacing (CON5; optional)
5 5x2 pin headers (CON7,CON9-CON11,CON23)
1 10x2 pin header (CON8)
2 3-pin headers (LK1,LK2)
1 2-pin header (JP5)
3 shorting blocks (LK1,LK2,JP5)
1 ferrite bead (FB12)
1 32768Hz watch crystal (X1)
1 miniature 8MHz crystal (X2) OR
1 standard 8MHz crystal with insulating washer (X2)
1 10kΩ vertical trimpot (VR1)

Semiconductors
1 PIC32MZ2048EFH064-250I/PT 32-bit microcontroller programmed with 0110619A.HEX, TQFP-64 (IC11)
1 25AA256-I/SN 32KB I^{2}C EEPROM, SOIC-8 (IC12)
1 LD1117V adjustable 800mA low-dropout regulator, TO-220 (REG2)
1 LM317T adjustable 1A regulator, TO-220 (REG3)
1 blue SMD LED, SMA or SMB (LED2)
3 LL5819 SMD 1A 40V schottky diodes, MELF (MLB) (D14-D16)

Capacitors
1 470µF 10V electrolytic
5 10µF 50V electrolytic
11 100nF SMD 2012/0805 50V X7R
4 20pF SMD 2012/0805 50V C0G/NP0

Resistors (all SMD 2012/0805 1%)

1 10kΩ	1 1.2kΩ	2 1kΩ
2 470Ω	1 560Ω	1 390Ω
2 330Ω	1 100Ω	3 47Ω

Front panel interface

1 double-sided PCB coded 01106195, 107.5 x 32.5mm
1 5x2 pin header (CON20)
2 4.7kΩ 1/4W through-hole resistors
2 22nF through-hole ceramic capacitors
2 PCB-mount snap-action momentary pushbuttons (S1,S2)* [Jaycar SP0721, Altronics S1096]
1 3-pin rotary encoder (RE1/RE2) [eg, Altronics S3350 or Jaycar SR1230 with integrated pushbutton]
1 knob (to suit RE1/RE2)
* only one required if using Jaycar SR1230 encoder

Power supply/routing board

1 double-sided PCB coded 01106194, 103.5 x 84mm
4 M205 fuse clips (F1,F2)
2 5A M205 fast-blow fuses (F1,F2)
3 ferrite beads (FB13-FB15)
2 2-way terminal blocks, 5.08mm pitch (CON13)
3 13x2 pin headers (CON14-CON16)
3 5x2 pin headers (CON17-CON19)
4 6021 type mini-U TO-220 heatsinks (for REG4 & REG6-REG8) [Jaycar HH8504, Altronics H0635]

Semiconductors
4 LM317T adjustable 1A regulators, TO-220 (REG4-REG6,REG8)
1 LM337T adjustable -1A regulator, TO-220 (REG7)
14 1N4004 400V 1A diodes (D17-D30)

Capacitors
2 470µF 16V electrolytic
7 47uF 25V electrolytic
2 10uF 50V electrolytic
6 100nF 50V through-hole multi-layer ceramic

Resistors (all 1/4W 1% metal film)
2 1.5kΩ   2 1kΩ   1 560Ω
3 330Ω   2 220Ω

LCD assembly

1 128 x 64 pixel graphical LCD with 16-pin connector
1 double-sided PCB, coded 01106196, 51 x 13mm
1 13x2 pin header
1 16-pin header

Chassis parts, connecting cables etc

1 2U rackmount case or similar
1 M205 'extra safe' fuseholder
1 1A slow-blow M205 fuse
1 5A 250VAC DPST or DPDT switch
28 9mm long M3 tapped spacers
56 M3 x 5mm black panhead machine screws
3 No.2 x 6mm self-tapping screws
1 1m length of 26-way ribbon cable#
1 30cm length of 20-way ribbon cable#
1 1m length of 10-way ribbon cable#
6 26-pin IDC line plugs
2 20-pin IDC line plugs
6 10-pin IDC line plugs
1 1m length 10mm diameter heatshrink tubing
10 small cable ties
4 instrument feet with mounting screws
or 1.3m length 26-way(+) ribbon cable

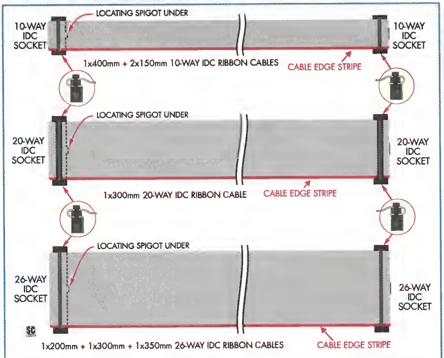


Fig.15: here's how to make up the seven ribbon cables required to connect the various boards together. Three ten-way cables are required in two different lengths, plus one 20-way cable and three 26-way cables, each a different length.

in each case for crimping to the connectors.

You can strip these cables out of ribbon cables with more wires, by making a small cut between two wires and then separating the sections by pulling them apart.

It's best to use a dedicated IDC crimping tool for this job, such as Altronics T1540. You can use a vice, but you have to be careful to avoid crushing and breaking the plastic IDC connectors.

Each connector has three parts: the bottom part, which has the metal blades that cut into the ribbon cable; the middle part, which clamps the cable down onto these; and a locking bar at the top that holds it all together once it has been crimped.

Note how, as shown in Fig.15, the cable passes between the locking bar and upper part before folding over on the outside edge and then being crimped underneath.

So with this in mind, slightly separate the three pieces without actually taking them apart, and feed the ribbon cable through as shown. Ensure there is enough "meat" for the metal blades to cut into, then place it into your crimping tool or vice without allowing the cable to fall out. Clamp the three pieces together, gently at first, then more firmly.

The trick is to crimp it hard enough to ensure that the blades cut fully through the insulation and make good contact with the copper wires, without pressing so hard that you break the plastic.

If using a vice, it's best to wedge a piece of cardboard between each end of the connector and the vice, to provide some cushioning.

Once you've crimped a connector at one end of the cable, do the one at the other end, making sure that when you're finished, the locating spigots will both be facing in the same direction – see Fig.15. Then repeat this procedure for all the other cables that are required.

Next month

The final article in this series will cover testing all of these assembled boards, programming the microcontroller and putting it all together in its case.

We'll also have some performance measurements and instructions for using the finished unit. SC