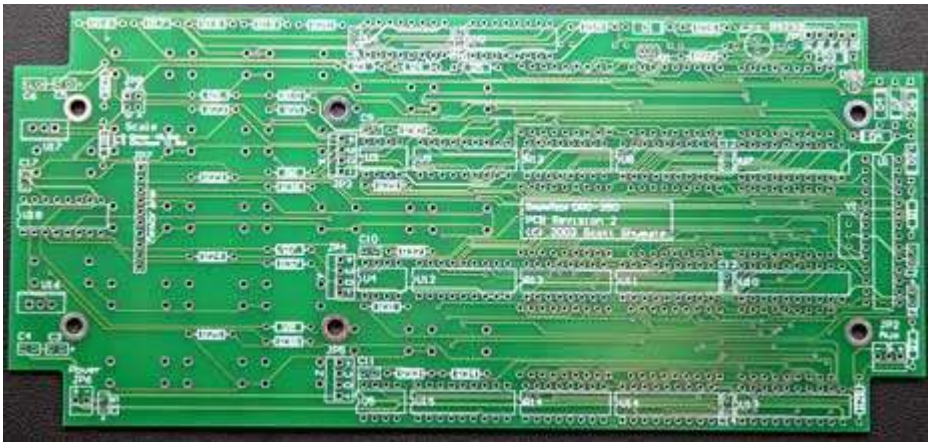
Construction Guide – European Version

PCB

This section describes how to build up the DRO-350 printed circuit board (PCB). The bare PCB is available for purchase on the order page.



Bare DRO-350 PCB

Static Protection

Several of the ICs in the DRO-350 are static sensitive. The 74 series chips used in the DRO-350 are the HC variety, which stands for high-speed CMOS. HC devices have static protection on their pins but are still more static sensitive than the old TTL devices which were very hard to destroy. If possible, you should build the PCB on a static safe workspace that has a static mat. If this is not available, you should at least work on a non-carpeted area and wear a static strap at all times. Avoid handling the ICs any more than necessary and keep them in their static safe packaging until they are ready to install.

Soldering

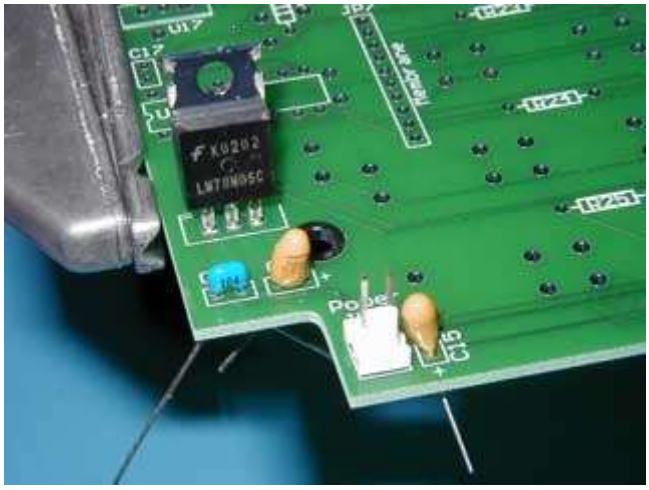
A few quick words on soldering. Soldering is about applying the right amount of heat and solder to lock a mechanical connection in place. What this means is you don't want solder to be the electrical conductor. Instead you want a mechanical connection to be the point of electrical conduction.

A good solder joint has a nice, even flow and is shiny and smooth. If a solder joint is grainy or dull appearing, it is not a good joint and may crack or break in the future. Choose a solder iron that is appropriate for the task at hand, too little power and you will either be forced to heat the joint longer than is safe or you will not have enough heat to allow the solder to flow well, too much power and you risk heat stressing the component. For DIP soldering PCBs, a 25-30W iron is probably about right. A temperature controlled iron is even better but I suspect not many people attempting this project will have access to one.

When soldering on the PCB, put the tip of the solder iron on the pad and the pin at the same time. Apply the solder roughly at the point where the pin, pad, and solder iron tip meet. Quickly feed the solder until a small puddle extends to the edge of the pad and immediately stop. You don't want the tip of the iron on the pin and pad for more than 2-3 seconds. If you don't have much experience soldering, I would strongly suggest practicing a bit before starting on the PCB. Don't get discouraged, it just takes a bit of practice.

Construction Steps

Step 1. Build the 5 Volt Power Supply



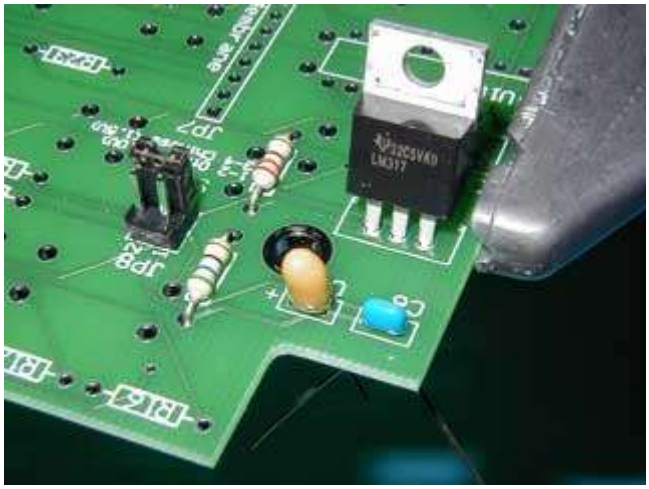
The 5 volt power supply consists of device U16 and capacitors C3, C4, and C15. An optional MTA-100 connector can also be used for the power entry connector JP6. Pay special attention paid to the polarity of C15 and C3. These are tantalum capacitors and have a marking on the POSITIVE lead. Make sure to insert the marked lead to the side marked with a + on the PCB. Also notice that the pad this lead is inserted into is square instead of round.

First, insert the capacitors C3, C4, and C15 into their respective positions and bend their leads apart at about a 45 degree angle. Next insert U16 and solder one pin from the component side to hold it in place. Flip the PCB over to the display side and solder all of the unsoldered leads. Use diagonal cutters to clip all of the leads.

Install the optional 2 pin MTA-100 jumper in JP6. To solder the jumper in place, put a small pool of solder on one of the pads. While holding the jumper in one hand, heat the pool of solder with the iron and quickly insert the jumper. Finish soldering after flipping the board to the display side.

Test the 5V supply by probing with a voltmeter. Temporarily connect the 9V external power supply to JP6. Make sure the correct polarity is observed as shown on the PCB silk screen. The Aux connector JP2 is a convenient place to probe between the + and - pads. The voltmeter should indicate 5.0V ±0.2V.

Step 2. Build the 1.5 Volt Scale Power Supply



The 1.5 volt power supply is what powers the Chinese scales and consists of device U17, capacitors

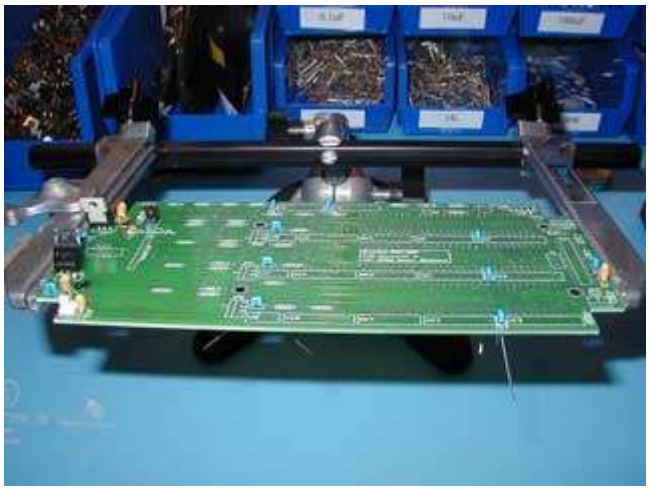
C5 and C6, and resistors R11 and R35. Again, pay special attention paid to the polarity of C5. Observe the same capacitor polarity markings as described in step 1.

First, insert the resistors R11 and R35 into their respective positions and solder them in place from the component side of the PCB. Next, insert the capacitors C5 and C6 and bend the leads apart. Insert U17 and solder one pin from the component side to hold it in place. Flip the PCB over to the display side and solder all of the unsoldered leads. Use diagonal cutters to clip all of the leads.

With the 9V external power supply connected, test the 1.5V supply by probing with a voltmeter between pad 3 on JP8 and the - pad on JP3. The voltmeter should register 1.5V ±0.06V.

If you plan to use Chinese scales, then either solder a 100 mil, four pin header in JP8 or install a jumper wire between pads 3 and 4 on JP8. To solder the header in place, put a small pool of solder on one of the pads. While holding the header in one hand, heat the pool of solder with the iron and quickly insert the header. Finish soldering after flipping the board to the display side.

Step 3. Install the Decoupling Capacitors



Before installing any ICs, it is easiest to install the decoupling capacitors first. All the decoupling capacitors are 0.1uF ceramics and do NOT have polarity so you can solder them in any orientation. The decoupling capacitors have designators C2, C8, C9, C10, C11, C12, C13, C14, C16, and C17.

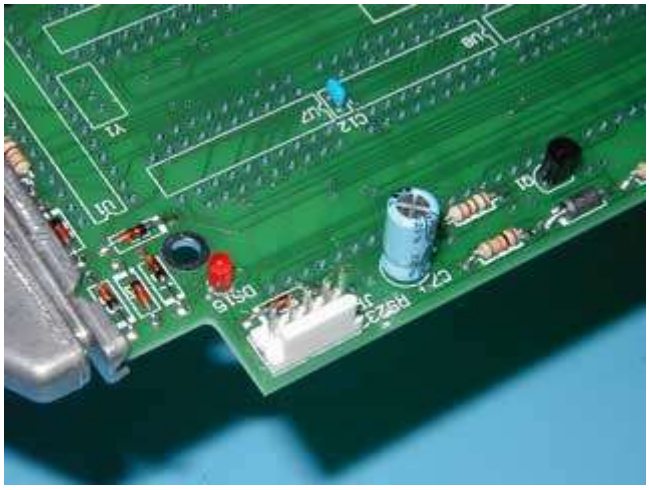
There is a 10uF tantalum capacitor next to the PIC with designator C1. This capacitor provides filtering for the PIC since it is several inches away from the 5V power supply. This capacitor is polarized so solder it in place as described previously for the tantalum capacitors in the power supplies.

Insert all of the indicated capacitors and spread their leads apart as before. Solder them in place from the display side. Use diagonal cutters to clip all of the leads.

Step 4. Install the Resistors

Next, install all of the resistors. It is easiest to use a triangular lead forming tool but not required. The resistor lead are spaced at 400 mil. Run through the BOM installing each resistor value at its correct designator. It is easiest to solder the resistors from the component side. After soldering, you probably want to run back through the BOM double checking the correct resistor values.

Step 5. Build the PIC Programmer



Build the serial programming circuit around the PIC and also install the ceramic resonator. Begin by installing diodes D1-D7 with the correct orientation of the anode and cathode. The cathode is indicated on the PCB by a white bar on the silkscreen and a square pad instead of a round pad. The cathode-side of the diode itself is usually marked with a bar or solid colored area. The diodes are mounted on 400 mil centers like the resistors so a lead forming tool would be handy here too. Be very, very careful that the correct diode part number from the BOM is installed in the correct designator on the PCB. Bad things could happen if not. Solder the diodes from the component side of the PCB.

Insert the transistor Q1 with the correct orientation as noted on the PCB. The transistor will have a flat side that is installed in the same position as the flat side on the silkscreen. Bend the outer leads of the transistor outward to hold it in place but do not solder it yet.

Insert the T1-size (3mm) LED with designator DS15. The important thing to note about the LED is that the cathode is the shorter lead and MUST be installed in the square pad on the PCB. The round pad is for the anode, which is the longer lead. After inserting the LED, bend the leads apart to hold it in place but do not solder it yet. If installing the optional light pipe indicated on the BOM, then the height of the LED should be as low as possible. Otherwise, the height is not at all critical.

Install the electrolytic capacitor C7 next. Note that electrolytic capacitors are polarized like tantalum capacitors. Pay careful attention to the markings on the electrolytic because the negative lead is usually marked on an electrolytic where as the positive lead is usually marked on a tantalum. Install the electrolytic capacitor with the negative lead in the pin opposite the positive pad marked on the PCB. Again, bend the leads apart to keep it in place but do not solder.

Now, flip the board to the display side and solder the leads for Q1, DS15, and C7.

Install the optional 5 pin MTA-100 jumper in JP1. To solder the jumper in place, put a small pool of solder on one of the pads. While holding the jumper in one hand, heat the pool of solder with the iron and quickly insert the jumper. Finish soldering after flipping the board to the display side.

Lastly, install the 20MHz ceramic resonator Y1. The resonator does not have an observed polarity so it can be installed in any orientation. Carefully solder one of the leads of the resonator to the PCB to hold it in place and then finish soldering the other leads from the display side.

Installing IC1 at this stage allows you to program the PIC, or check out the programming of the PIC if it was supplied programmed. It is useful to complete this at this stage if you intend to use the PIC programmer, since any problems can more easily be addressed. If you do not plan to use the programmer, since the PIC has been supplied programmed, you can continue to the next phase of building.

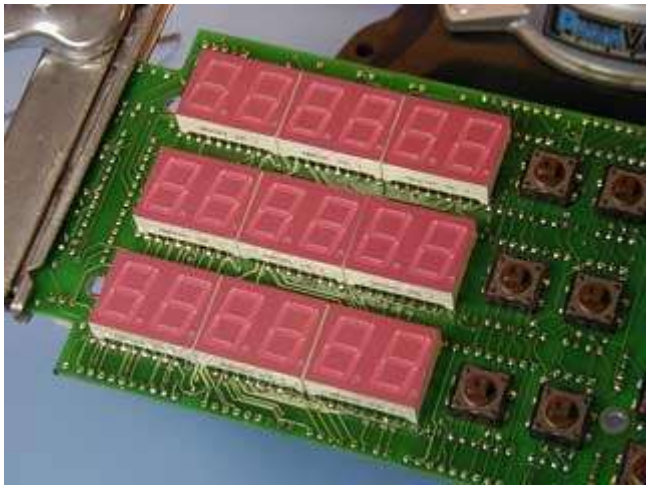
Step 6. Install the ICs, Resistor Packs and Seven Segment Displays

At this point there are two options for continuing the build. If the PIC has already been programmed, then it is recommended that steps 6 and 7 are combined in three stages, building and testing each row of the display as you go. Start with the IC's and displays for the top, row, and power the unit up, to check for the dro350 banner line. Clearing any problems found now will allow you to proceed to the second row, knowing that the initial batch of soldering has been completed successfully.

Install the ICs next but be careful not to force them into the PCB. If the pins do not line up, gently bend them by pressing the side of the IC on a hard surface with a static safe bag between the pins and the surface. If completing the build in stages, first fit U2, U7,8 & 9 and resistor pack R12, followed by displays DS1,2 and 3 (the top row of displays).

To solder the ICs and resistor packs, carefully solder one pin on each from the component side. This serves to hold it in place so you can solder the rest of the pins from the display side. After soldering one pin on each, flip the board and solder the rest of the pins. Be careful not to start soldering with the one pin you soldered before otherwise it might fall out.

Install the seven segment displays on the same side of the PCB as the tact switches. That is, the display side of the PCB. Be careful not to solder any of the seven segment displays upside down or it will be a pain to remove it. Double check that the decimal point on each display is in the correct position before soldering. Solder the leads from the component side of the PCB. If any of the IC leads interfere with the displays fully seating, gently bend them out of the way with sharp-nosed pliers or tweezers.



At this point, the first line of the display can be tested, and should display dro350. If not, then fault finding at this stage is easier than with a fully populated board. Contact the user group for help.

If the first row is working correctly, proceed to the second row, with U10 to 12, R13 and DS4 to 6, the middle row. This will add r4 or which ever version of software is loaded. If the first row becomes corrupted, you know that you have a solder splash or short on the second row of IC's which can be sorted at this stage.

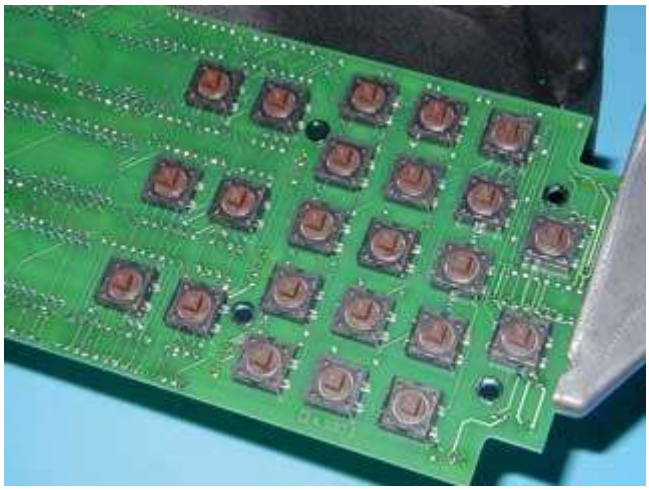
The third row can be completed with U13-15, R14 and DS7 to 9, and will display either mill or lathe (mill was added at version 4 and the display will be blank for previous versions of mill code).

Step 7. Install the Scale interface

Finally insert the ICs with designators U3 to 6 and U18. This completes the scale interface, but we need switches fitted before we can test the scales themselves.

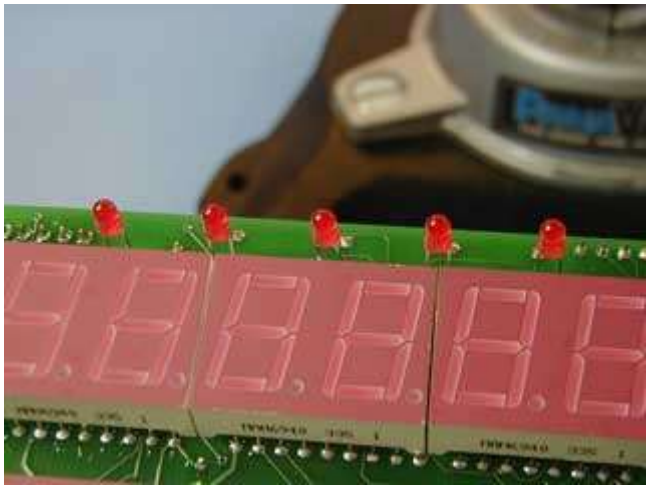
Install the optional 3 pin MTA-100 jumper in JP2 and the optional 4 pin MTA-100 jumpers in JP3-JP5. To solder a jumper in place, put a small pool of solder on one of the pads. While holding the jumper in one hand, heat the pool of solder with the iron and quickly insert the jumper.

Step 8. Install the Tact Switches



Install the seven tact switches on the display side of the PCB from the components. There is nothing special to installing the tact switches other than to make sure they are fully seated in the PCB. Flip the PCB over and solder all of the tact switch leads from the component side of the PCB.

Step 9. Install the Indicator LEDs

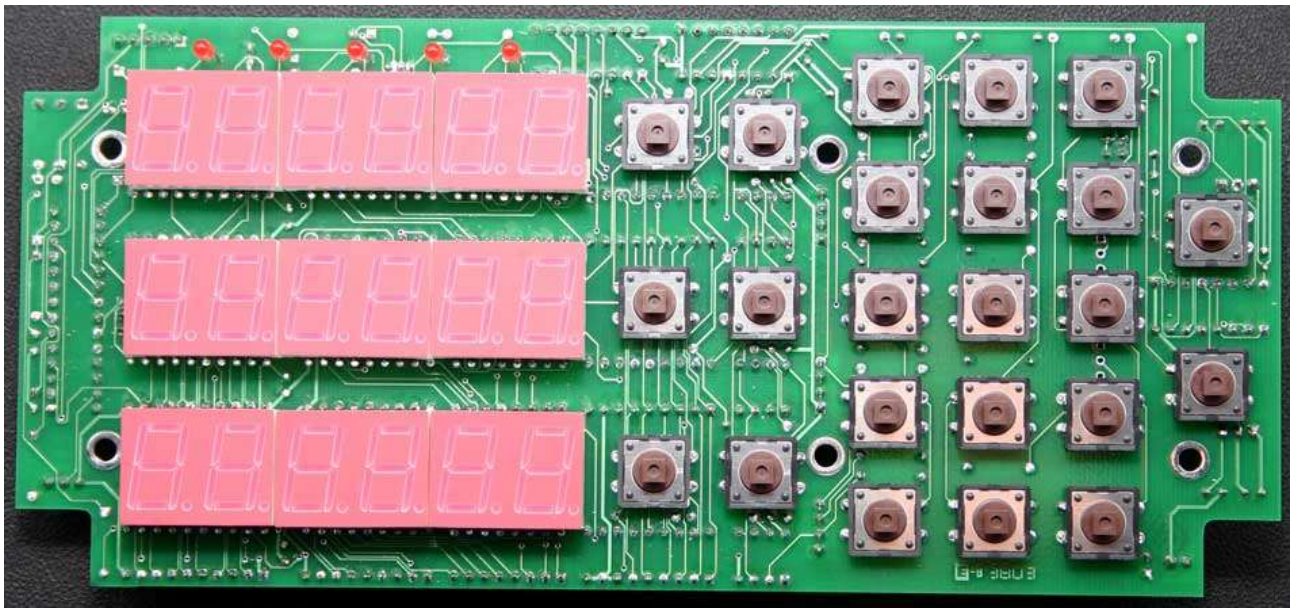


There are five T1-size (3mm) LEDs on the display side of the PCB. The LEDs have designators DS10-DS14 on the schematic but note that there is no silkscreen on the display side of the PCB. The pads are easy to locate since they are just above the seven segment displays and are the only pads without something soldered in them.

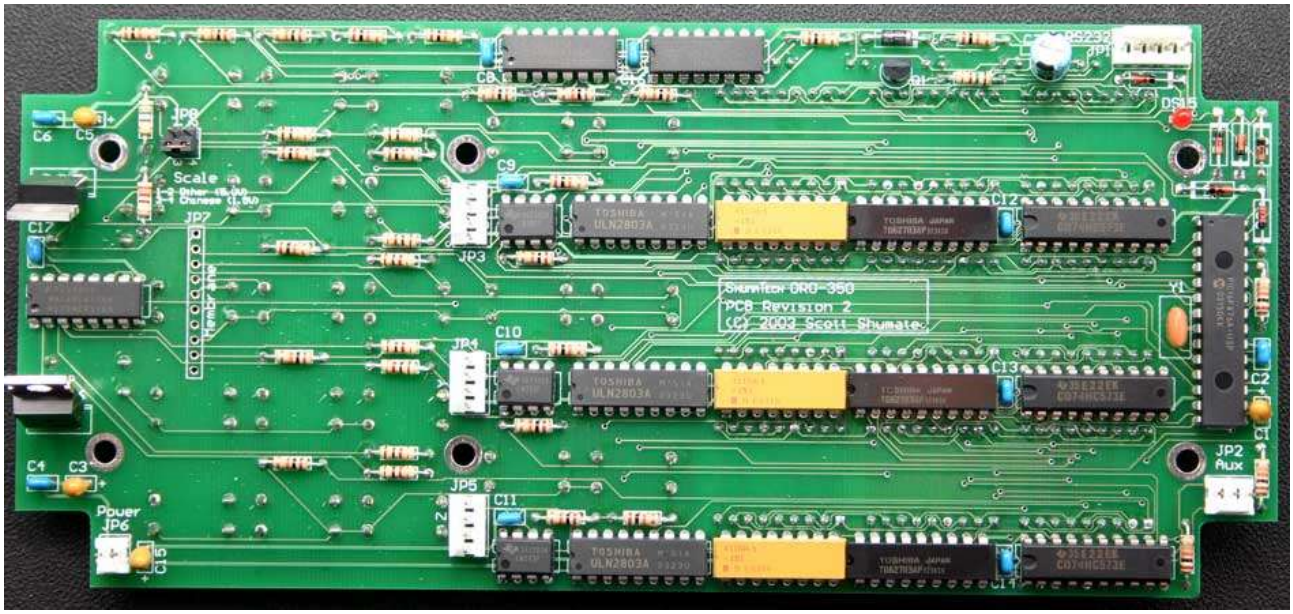
The important thing to note about the LEDs is that the cathode is the shorter lead and MUST be installed in the square pad on the PCB. The round pad is for the anode, which is the longer lead.

The height of these LEDs is somewhat critical. You don't want to install them higher than the face of the enclosure since they would prevent the overlay from laying flat. You also don't want to install them so low that they are hard to see when the overlay is installed. The best height I have found is to have them stick out 1/8" (125 mil) past the face of the seven segment display. This way, they poke up into the holes you drill into the enclosure but are recessed 25 mils or so from the face of the enclosure.

Completed Board



Display Side of a Completed DRO-350 PCB



Component Side of a Completed DRO-350 PCB

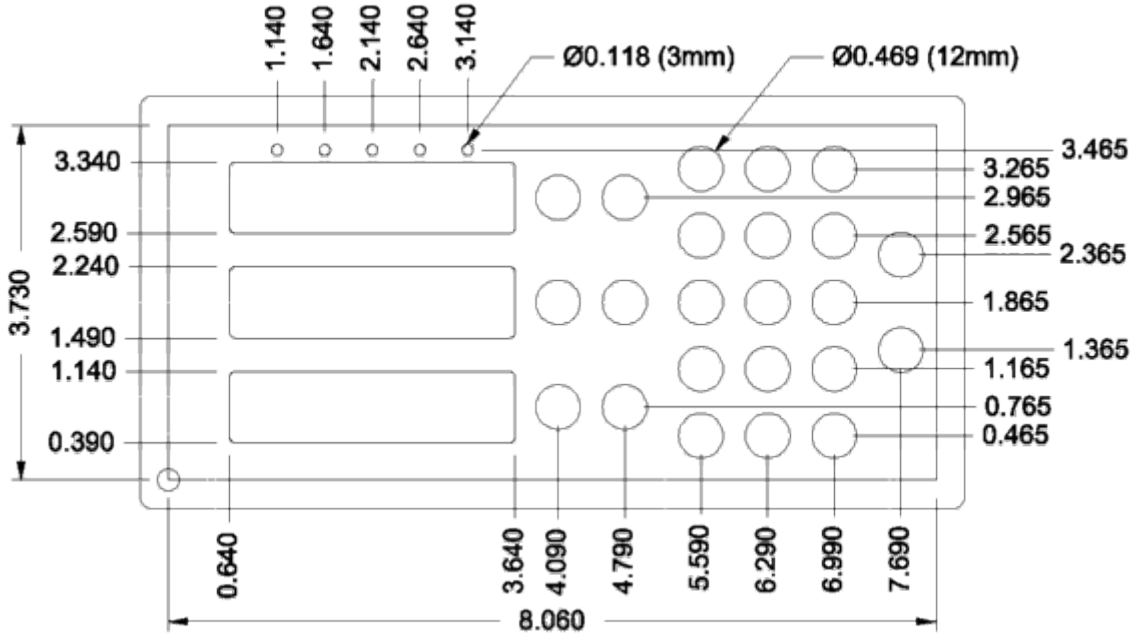
Enclosure

Step 1. Machine the Front Panel

The enclosure is machined using standard milling and drilling operations. Note that when cutting plastic, you should use a fairly high RPM and use a slow feed while drilling or use a wood block or something behind the front panel. The enclosure is made of ABS plastic so use as fast an RPM as possible without melting the plastic.

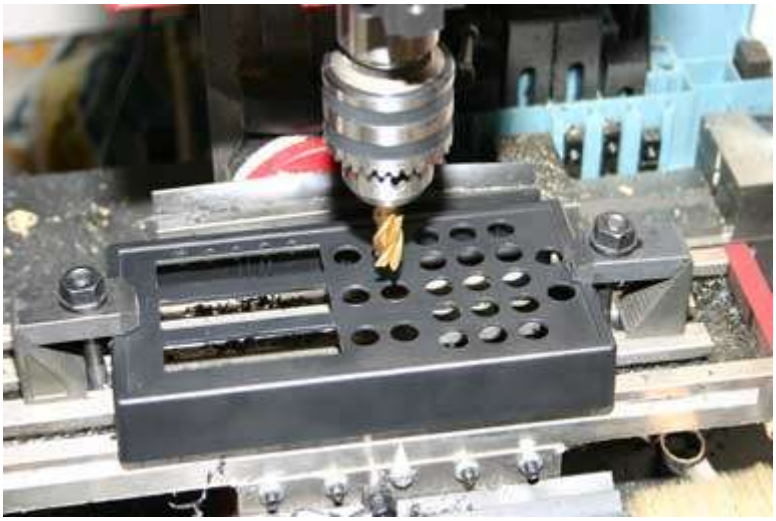
None of the drill holes in the drawing are critical. You can do fine using the closet nominal drill size that is greater than the called-out diameter. For example, the 0.469 (12mm) diameter called out for the tact switch buttons can be drilled with a 1/2" diameter drill bit. The overlay will cover up any overage or mistakes so don't worry about precision here.

For the display cut-outs, use an 1/8" or so end mill, either two or four flute. The following figure is a dimensioned drawing of the front of the enclosure.



DRO-350 Front Panel Drawing

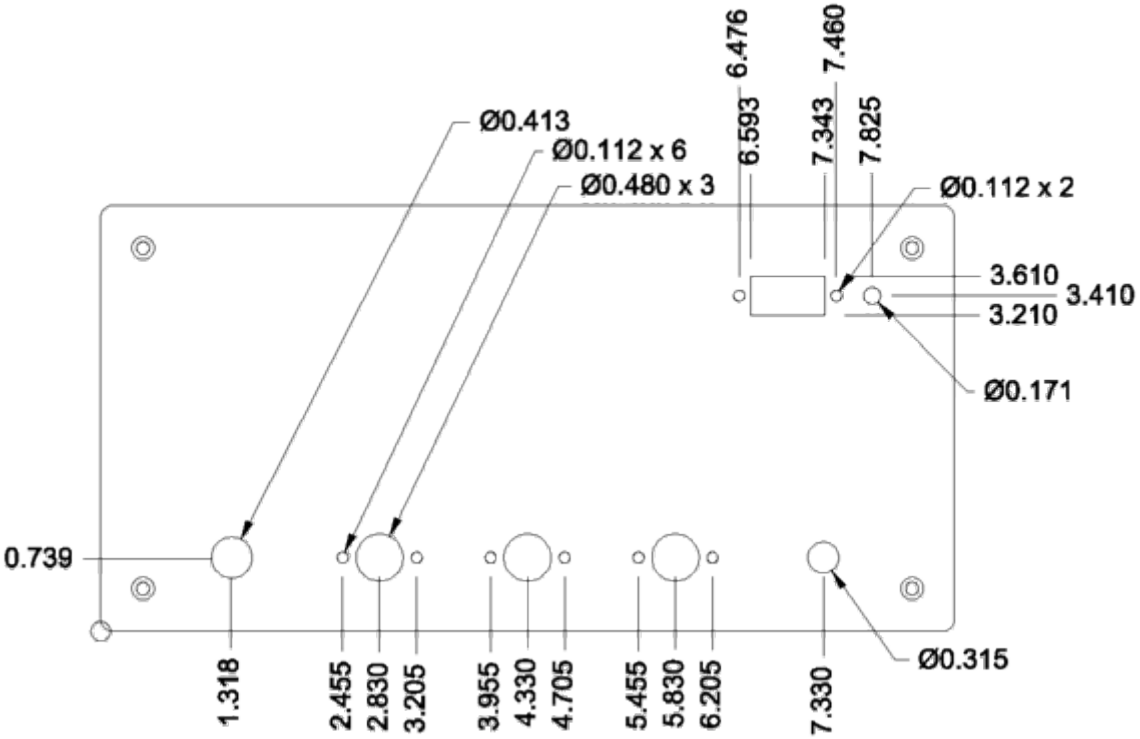
When you finish the milling and drilling operations, you should end up with something that looks like the following picture.



DRO-350 Front Panel After Milling and Drilling Operations

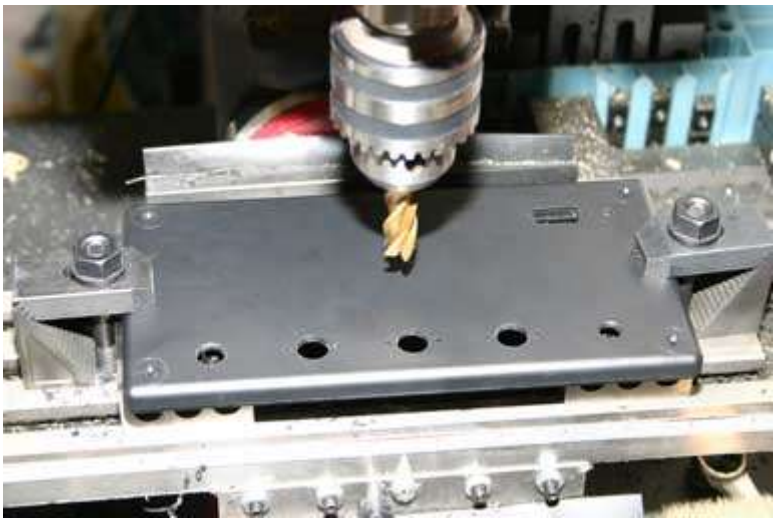
Step 2. Machine the Rear Panel

The layout of the rear panel is not at all critical except for the position of the 0.171" diameter hole for the programming LED. This can be fitted with a plastic light pipe, to bring the illumination to the back panel. You can place the power, scale, and aux connectors wherever you like. Just be careful to stay away from the stand offs that are on the inside. The following figure shows the layout I used if you are looking for ideas.



DRO-350 Rear Panel Drawing

(Power and Aux connectors have different hole profile – need to change drawing)



DRO-350 Rear Panel After Milling and Drilling Operations

Step 3. Install the Stand offs

The stand offs are mounted to the back panel of the enclosure and attach to the component side of the PCB with the seven segment displays and tact switches facing out. The six plastic posts in the back panel must be tapped M3 to hold the stand offs. You should use a small hand tap with a gentle touch. Be careful to stop as soon as the front of the tap hits the bottom of the post hole otherwise you will strip the threads. If you do, don't worry since you can just epoxy the stand offs in place just as well. Another option is to drill through the boss, and tap right through, then back fill the outside of the case with a little spot of super glue.

Tapping the Plastic Posts M3

The hex stand off are made of steel and have an M3 male thread on one end and an M3 female thread on the other end. The male thread is slightly too long for the plastic posts but, by fitting an M3 nut to the stud, the length can be reduced, and an 18mm stand off with a 2mm nut gives the right spacing, and the larger surface area of the nut sits neatly against the plastic boss.

Hex Stand off with Nut Space (Replace with UK picture when available)

Install the six hex stand offs in the tapped posts. I would recommend tightening them firmly by hand. If you use a tool, be careful not to strip the threads.

Hex Stand off Installed in the Back Panel (Replace with UK picture when available)

Step 4. Install the Connectors

The three scale connectors are 4 pin mini-DIN type that mount to the back panel with six M2.5 x 12mm machine screws. The connectors listed in the bill of materials have a hole on either side that must be tapped M2.5 before mounting with the screws. This is most easily done with a small hand tap.

Tap the Mini-Din Scale Connectors with an M2.5

After tapping, mount the three scale connectors to the back panel with the M2.5 x 12mm screws.

Scale Connector Mounted to Back Panel with M2.5 Screws

The auxiliary connector is a 3.5mm stereo jack. Remove the nut included with the connector and mount it on the back panel.

Auxiliary Connector Mounted to the Back Panel with the Included Nut

(Replace with UK picture when available)

The DC power connector is a standard 2.1mm type. The connector listed in the bill of materials is mounted to the inside of the case via a pair of M2.5 bolts with nuts and lock washers.

DC Power Connector Pushed Through the Hole in the Back Panel

(Replace with UK picture when available)

Overlay

The overlays are silk-screened onto the back of transparent 10mil Lexan film. They are screened with three colours: black, white, and a near-cyan colour. The windows for the seven segment LED displays and the indicator LEDs above them are water clear. There two types of overlays, one with X, Y, and Z designations for mills and one with X, Z1, and Z2 designations for lathes. Both types of overlays are available for purchase on the order page.

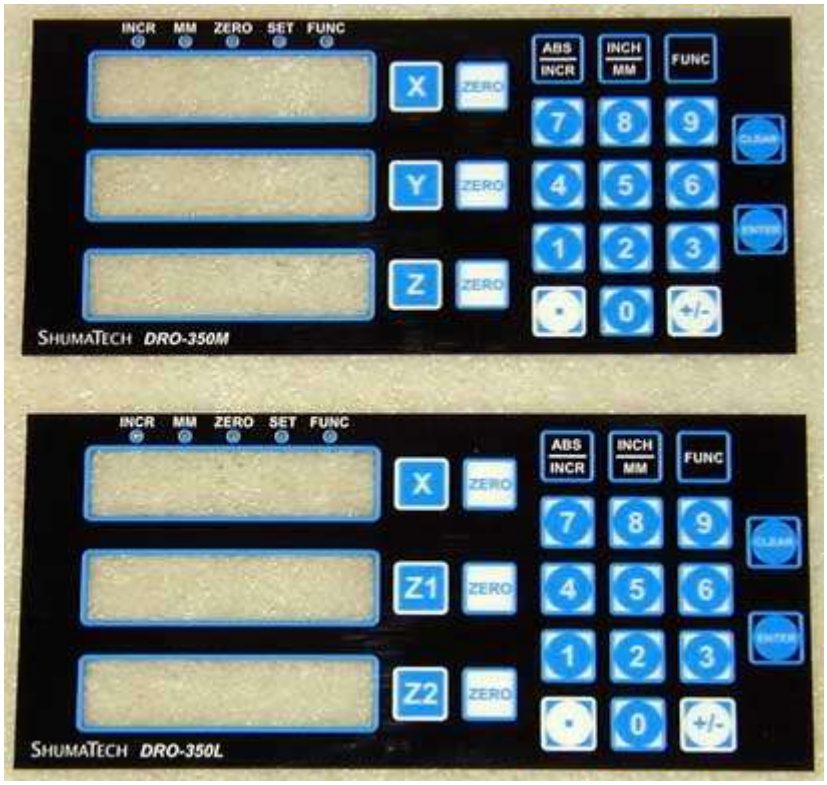


Photo of the DRO-350M and DRO-350L Overlays

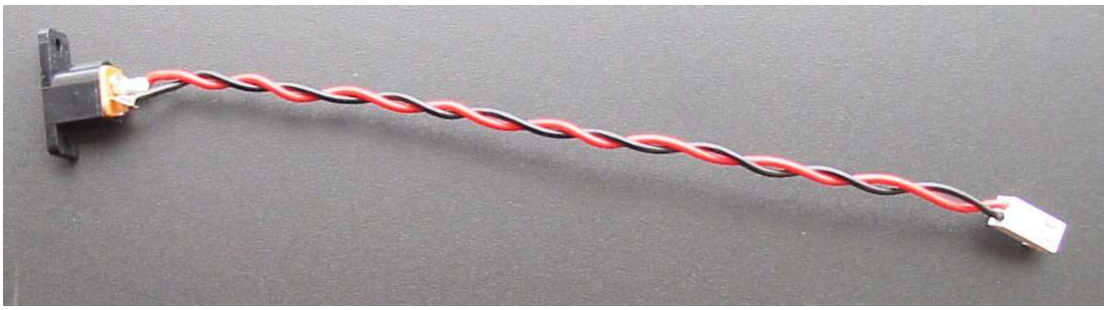
The overlays are best installed by masking the front of the completed enclosure with masking tape and spraying on a layer of adhesive such as Super77 onto the front of the enclosure.. The overlay can then be pressed into place onto the front of the enclosure. IMPORTANT: Do not forget to remove the plastic protective film on the front of the overlay. Otherwise, the transparent windows will be slightly opaque instead of crystal clear.

Photo of the Partially Removed Protective Film on the Overlay

An alternative method of fixing is to use double sided tape, and cut out all the holes. A little bit of a fiddle, but provides a strong fix without the problems of adhesive spreading.

Power Cable

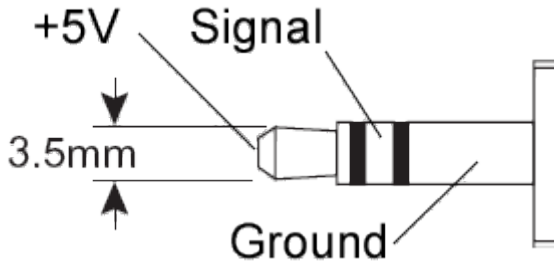
The power connector is a 2.1mm power jack socket with positive on the pin. This can be changed to a larger one if required, but the 2.1mm is one of the standard sizes available on mains power packs.



This is connected to the case by 2 off M2.5 bolts with nuts and lock washers.

Auxiliary Cable

The standard being used for the auxiliary connector is a 3.5mm (1/8") stereo plug. The tip is 5V, the middle ring is the auxiliary signal, and the inner ring is ground as shown in the following picture. The tip is 5V to ensure that the auxiliary device is not energized until the plug is fully inserted.



Auxiliary Cable Plug Signals

Auxiliary Cable 3.5mm (1/8") Stereo Connector

Programming Cable

The following table shows the pin-outs from the female DB-9 connector to the PCB designator for the DRO-350 programming interface.

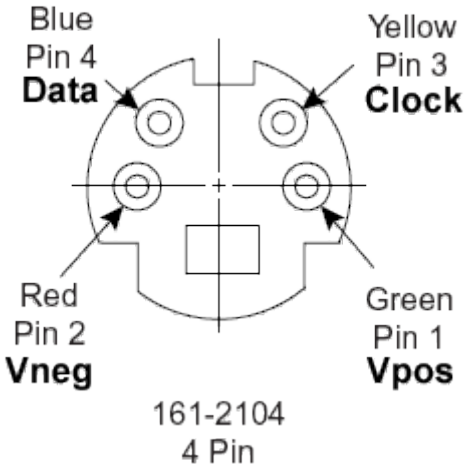
RS-232 Signal	PCB Designator	Female DB-9 Pin
CTS	C	8
DTR	D	4
Tx	T	3
Ground	G	5
RTS	R	7

Programming Cable DB-9 Serial Connectors

Scale Cables

If you can handle basic soldering, it is easy to build cables for the Chinese scales yourself. In some respects, I would almost recommend it over buying the commercial cables. The big problem with the commercial cables is that the end that plugs into the scale is poorly designed. It can be difficult to get a good connection without a lot of fidgeting with the connector. Also, the connector sometimes has a tendency to short across the pads which will either prevent the DRO-350 from reading the scale or even cause the DRO-350 to shut down power to the scales if the positive supply is shorted.

The pin-out for the DRO-350 female mini-DIN jack is shown below. The plug is male and engages into the jack. Just imagine sticking the plug into the screen and you will get the pin-out right. Solder the cable to the four cups/posts and reassemble the plug.

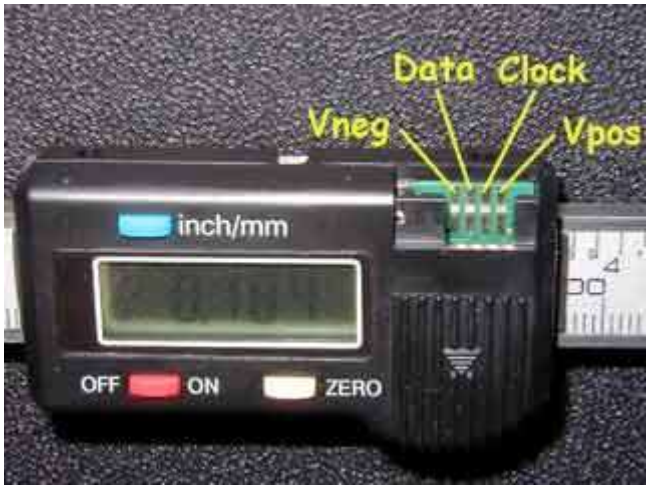


DRO-350 Four Pin Mini-DIN Female Jack – Front View

Scale Cable MTA Receptacle

The Chinese scale side of the cable is just a little more difficult. I know of no off-the-shelf plugs that work with the jack on these scales. The jack on the scale is basically just four pads on a PCB. Even if the plugs were available, I would still recommend you do the following to get a solid, reliable connection.

What I recommend is to solder the four wires from the cable directly onto the pads on the scale. Start by tinning the four pads with solder. You don't want a big puddle of solder but you do want enough to make a solid connection to the wire on the cables. Next, strip the end of each wire and attach it to each pad by holding the wire against the pad with one hand and touching the pad with the solder iron with the other hand. Look at the diagram of the scale jack below to figure out which wire to solder to which pad. The most important thing here is to hold the wire and scale as steady as possible so that you get a good, strong solder joint.



Chinese Scale Jack

To cover the jack, you can cut a small notch in the little plastic cover for the jack and route the wires through it. I will post pictures of everything I have described as soon as time allows.