Decoder constructional notes. (Decod10 and Decod11)

Although I would not recommend making decoders for complete electronic novices, it is relatively straightforward if you have a steady hand and some sort of magnification. (The ‘Optivisor’ is highly recommended)

Making the PCB

Both Decod10 and Decod11 have been designed specifically for DIY construction. Decod10 is physically larger and uses larger components so is easier to make. Both versions can be made on single sided PCB. The long version has all the components on one side. There is the option to halve the length by effectively folding the circuit in two and linking the edge tracks round the end. Alternatively the decoder can be cut in two and fitted in separate places in the loco. If you opt for the short version, the artwork on the resources site has a mask for a double sided PCB. As there are no holes or ‘vias’, the alignment of the two sides is not critical.

The website artwork is a .PFD file which can be printed directly onto clear film to create a photomask. It must be printed at 100%. Make sure the ‘print to fit’ option is off. A good quality Laser printer is recommended with a resolution of at least 600 dpi.I use a HP Laserjet 6MP in Postscript mode at 600dpi. Try it on paper first to avoid wasting film. Check the films for obvious blemishes using a light box and a magnifying eyepiece (watchmakers eyeglass). Pinholes can be touched up but it requires a very steady hand. The image on the .PDF files has the top layer already mirrored so the ink will be in contact with the photoresist. If you are making a double sided version, cut one of the layers to leave about 1 cm along the side of the images (don’t separate the individual decoders) and about 0.5 cm at the ends. Leave the other layer uncut. Line up the films with the ‘ink sides’ inwards. Tape the two sheets together along the 1 cm sides with thin ‘Sellotape’ keeping the tape clear of the image. You now have an open ended pocket. Cut a strip of double sided photoresist PCB so it slides into the pocket and clears the ends of the cut sheet. Tape the protruding ends to the larger sheet. The board cannot move and the layers are held in alignment. Expose both sides and develop and etch as usual. I strongly recommend using 0.8mm thick (or less) PCB. (available in UK from MEGA Electronics). Hint. If using a bubble etch tank with a plastic ‘basket’, drill a hole in the PCB first and tie a piece of string through it. Hang the string out of the tank. I lost several boards through the mesh of the basket till I did this. It also makes it easier to fish the board out.

Clean the resist off after etching with acetone. For soldering, I have used a very fine pointed bit in a temperature controlled iron (Weller). Early decoders were made using fine solder but I have found solder paste much easier to use especially for Decod11 with its very small pad spacing. It is also much better for any surface mount components. The paste can be bought in small (25g) syringes and will last a long time (keep in a fridge). Squeeze out a small blob onto a smooth surface and use a cocktail stick to transfer small quantities of paste to the pads. Make sure the paste does not run between pads and do not use too much paste.

Fix the surface mount components as follows. Carefully paste the pads. Hold the component in the correct place with tweezers and touch the iron between the contact and the pad making sure the paste flows. For transistors and ICs , paste all the pads and place the device so the legs line up. Press the soldering iron bit down on a leg. This will solder that leg. Then check alignment and adjust if necessary. Hold the device down and then solder the rest of the legs. Finally, clean off flux residue with acetone and inspect carefully with a good magnifier (watchmakers eyeglass). Check for unmelted solder paste between legs and under components. If desired, coat the whole decoder with a suitable varnish. Silicone conformal coating is best.
For both Decod10 and Decod11, put the bridge rectifier diodes and C1 on first and the PIC processor last. I program the PIC before soldering it on but it can be programmed (or reprogrammed) 'in situ' if necessary. There are pads on the PCB to attach wires for programming. Five wires are needed. This saves paying for an adapter for the SOIC or SSOP chips as these can be expensive.

Suggested test sequence.

Assemble the decoder without the PIC. Test for shorts across D5 (the 12v supply) and across C1 (the 5v supply). If OK, connect a DC supply of about 12v (NOT a DCC signal) to the 'track' inputs. The polarity does not matter. Check the voltage across C1. It should be about 5.5v. This ensures the voltage regulator is OK and saves blowing up a PIC. Now connect a motor across the motor outputs. It should run as normal. Reverse the DC input and check the motor reverses. Now install the PIC and program it. (Remove the DC supply first!). Connect to a DCC system and check both the motor drive and functions.

The program has default CVs set. The default address is 3. The functions are set so F0 toggles output F0, F1 for output F1 etc except for output F5 which should flash with an F0 command. Consult the program listing for the CVs available and their operation.

Options.

Both PCBs have an option for the PIC resonator or crystal. If you use a resonator with built-in capacitors, cut the small tracks from the capacitor pads to the zero volts line. This will reduce the risk of the resonator ends shorting to ground. If you use a resonator with external capacitors, cut the track from the resonator centre pad to leave this isolated.

It is possible to use the power half (right hand side) as a separate booster. This needs its own track input for power and the two direction lines, the PWM drive and ground from the decoder. If you separate the power stage from the decoder altogether then one power stage needs to provide the input signals for the decoder. In this case it is essential that the ground line at the bottom of the decoder section is connected directly to the ground end of C1 at the decoder section itself.

The long versions of both decoders have a lower 'profile' than most commercial decoders as they have no connector and the components are on one side only. I have been able to fit these into gaps between the motor and body where

conventional decoders were too thick. If you do not need the function drivers, this section of the board can be completely cut off adjacent to the PIC so reducing the length. (long version only)

With suitable choice of rectifier diodes (eg RB051L-40 3 amp Schottky) and MOSFETs with a lower ON resistance, Decod10 can be rated upto 2.5 amps. Decod11 has an upper limit of 1.5 amps and realistically 1 amp for continuous rating.

The function drivers on Decod10 have a maximum current of 500mA each but the total will depend on the choice of the bridge rectifiers and the motor current.

The function drivers on Decod11 are limited to 100mA each.

Sizes (using 0.8mm thick PCB)

Decod10 (long)	2.7 in x 0.68 in x 0.14 in.	68.6mm x 17.27mm x 3.56mm
Decod10 (folded)	1.35 in x 0.68 in x 0.227in	34.3mm x 17.27mm x 5.77mm
Decod11 (long)	2.0 in x 0.48 in x 0.115 in.	50.8mm x 12.2mm x 2.92mm
Decod11 (folded)	1.0 in x 0.48 in x 0.184 in.	25.4mm x 12.2mm x 4.68mm

Although these are more in board area than some commercial decoders, the actual volume is less than most due to their thinness. They are more ‘loco’ shaped. Decod10 is a suitable width to fit most narrow hood US diesels in HO and the long version will replace the usual lighting board.

Mike Bolton 21/4/00