Mr. Kaypro

By Charles B. Stafford

Regular Feature

Kaypro Support

Advent Decoder Part 2

Personality decoder board Part 2, for All DSDD CP/M Kaypros except K-4x, and Robie

WHEREIN we shall complete the construction project started two issues ago, and install it, thereby completing yet one more step in the transmogrification of Darth Vader's lunch box.

For those among you who have just joined us, (and those who, like me, only have a 64k memory) we started this project two issues back, by etching a printed circuit board using copier toner for photo-resist, and collecting the rest of the necessary parts. We also discussed a second option of using a commercially available prototype circuit board. The schematic diagram from which we are working was also published. We did not however get into the theory of operation. We shall remedy that oversight now.

THEORY of OPERATION

The circuitry on the Personality Decoder board is divided into three distinct functions; The Personality part dealing with telling the BIOS how many and what variety of drives are installed, the actual Decoder part that separates the "drive select" signals and routes them to the proper drive, and a timing circuit that allows a "fast step" option for those drives that can handle it.

PERSONALITY

The 8 position dip switch, 4.7k ohm resister pack, and the 74LS151, IC2, comprise the Personality section. When one of the switch sections is closed it pulls the corresponding pin of IC2 low. Various combinations of pins "pulled low" result in a discrete number being transmitted back to the BIOS via the "read" line from pin 6. When a drive is selected for reading or writing, the TurboBios interprets the number returned on the ready line to determine whether or not the drive is installed, and what type it is.

DRIVE SELECT

The decoding is handled by IC3, the 7445. It takes the inputs from the drive A and drive B lines, combined with "motor on" signal and transmits a signal on pins 5,6,7,or 9 which are routed to the four select lines on floppy drive cable via connector J2. The TurboBios designates four drive select signals on two lines by using the "both on" and "both off" states as well as "A only on" and "B only on" and IC3 figures out what it really means.

NOTE: If this project is intended for a K-10 You should not install the DRV B select line between J 1 pin 12 and IC3. The connection and pull-up resistor between IC2 and IC3 should be installed however. The reason is that the BDOS for the K-10 uses the DRV B select line is used to access the hard-drive.

After installation the floppy drives will the first and third logical drives and the dip switch should reflect this.

A FASTSEEK option is included for those floppy disk drives that are capable of stepping faster than 6ms per track. Use of the FAST SEEK option (FSO) can lead to significant speed improvements and noise reduction when used with some drives. For those who are curious, it's the series capacitor, reducing the time constant that does the trick. THIS OPTION ONLY WORKS WITH THE TURBOROM. It is NOT compatible with any other ROM including the original Kaypro ROM, NOR is it compatible with any software which directly accesses the floppy disk controller, such as Uniform, Media Master, or Fastback. To use the FSO and these programs on the same computer, you must use a single pole double throw toggle switch as described in note 6, Section E.6. page E-7, of the TurboRom Manual.

SIDE SELECT

The schematic also shows a "side select" line for K-II/83 only. This was included in the design for those who had not yet done the II to 4 upgrade yet, or who were doing it in conjunction with the installation of the Personality Decoder board. The K-II/83 mother-board did not include a trace to pin 32 of the floppy connector, and this is a relatively painless way to do it. It can be included or not at your discretion.

ON TO CONSTRUCTION: LAYOUT

For those of you who etched the custom circuit board, layout is easy, just put the proper component in the holes as labeled on the circuit board, and rejoin us a few paragraphs later.

This stage can make assembly very easy or more intricate, depending on your own proclivities. There are several choices, ranging from "let's just get this thing together as painlessly as possible" to "WOW, how did you do that?" Observe Figure 1. This is a representation of the "top" (non-solder) side of the Syntax PC-462905 Uni-Board mentioned in the first half of this project.

Each group of three holes in each row is

in one solder pad, so three leads can be connected by putting one in each hole and soldering it on the back side. This will make life a great deal easier (as I learned from an HCW [HIGHLY CERTIFIED WIZARD]). There are also two "busses", each consisting of half the perimeter holes and lines of holes between the three hole pads, also making life a little easier. The holes on the board outside the boundaries of the drawing were omitted, because we don't need them. Now, the choices, the bus on the top and right sides will be the "ground" bus, and the bus on the left and bottom sides will be the "power" bus. For ease of construction, the "output" header, J2, will be placed so that one row of pins is in the ground bus, and the other row is in the first (left-most) column of "three"s. This is convenient because, checking the schematic diagram, all the even-numbered pins must be grounded.

J1 and the "dip" switch will be placed using the same consideration. The other sockets are 4-holes wide, so if we place them over the center line of the three hole columns we lose one hole each pad, but, if we place them so as to straddle the ground and power busses, we won't use up any more holes than necessary, making wiring easier. Figure 2 shows the result as seen from the top. J1 is the final area where choices are important. Perhaps the most elegant solution is to put J1 on the bottom side of the circuit board, so that the finished board will just plug onto the motherboard, and the existing cable will plug onto J2. The difficulty encountered here is "side-soldering" a connector not intended for side-soldering. It can be done, by pulling the pins far enough out of the connector shell so they can be soldered, but so that the plastic shell will still maintain the alignment of the pins, and by using a block under the board to keep the pins from falling all the way into the holes until they are soldered, side-soldering the pins, and then pushing the plastic shell back down on the pins.

A second option is to just put J1 on top of the board, solder it on the bottom and plug the finished product onto the mother board upside down. The disadvantage here is that one must remove the personality-decoder board completely to reconfigure it. The third option and perhaps most reasonable compromise would be to put another header at the J1 location and use a cable to connect it to the mother-board. The caveat here is to make sure that J1 and J2 are clearly labeled, to eliminate confusion when doing the final installation.

Once you've decided on the layout, put the components in place, invert the board (you might want to use a folded towel to put it on, to compensate for component thickness) and solder the pins at the diagonal corners of each component. Soldering only the diagonal pins now will allow for inspection of the result and adjustment if necessary (adjusting one pin is a lot easier than de-soldering all 14 or 16. When you install the header at J2 (also at J1 if that's your choice), if you use a right angle header as I did, just let the pins barely protrude from the bottom of the circuit board, so that there will be enough room to put the plug on when you're finished.

With the position of the dip switch shifted to take advantage of the ground bus, one column of holes is covered up. Installation of the strip resistor on the bottom of the circuit board using these holes (with the "dot" end toward the bottom [power bus end] of the board) will free up the last holes in the pads for wiring, so that there will only be one lead in each hole.

So endeth the layout portion of the project.

WIRING

Before we really get too involved, you might want to make at least two copies of the schematic diagram, and two of Figure 3. We'll use one of each during the wiring phase, and another during the checking phase. Some colored pencils (not all red, but at least three different colors) and a highlighter will also come in handy.

What might at first have seemed to be a real chore, has been greatly simplified by the choices we made during layout. For instance, all the ground connections to J1, and J2, have already been made, as have the connections from the resistors to the dip switch.

So here we go! First things first, the power and ground connections to the three ICs. Following the table on the schematic and remembering that IC pins are numbered counter-clockwise from the notch when viewed from the TOP, make, install and solder jumpers from the power and ground busses to the appropriate pins of the IC sockets. All wiring can be done on the top of the circuit board, but a more elegant solution here is to use bare wire jumpers on the bottom of the board from the appropriate pin to the bus, which is right next door, so to speak. While we are in the neighborhood, a similar jumper from the powerbus to the resistor would be appropriate. See Figure 3.

SOLDERING

Perhaps a short refresher on soldering is appropriate here. (All you HCWs can take a short nap now) For the rest of us, as a general rule, the smaller the soldering iron the better, and Radio Shack varieties work as well as the higher priced spread. Thermostatically controlled, variable temperature irons are very nice for commercial applications, but not necessary for us AWs (Apprentice Wizards). My favorite weapon is a 15 watt refugee from Radio Shack, which has a very fine tip, and is light enough to be very handy. It will probably not last more than 5 years, but I'll be ready for a change by then anyway, and it was very inexpensive.

Eutectic solder (63/37) works the best, but 60/40 is an acceptable substitute. Rosin flux is the only way to go and a small can will last almost forever. Heat up the iron, tin (or re-tin) the tip (dip the tip in the flux, cover it with solder and wipe it off with an old piece of damp towel) and we're ready to press-on.

Hold the tip against the lead or pin protruding from the solder pad, count three and apply the solder. Not much, just enough to fill the hole, remove the iron and proceed to the next one.

The numbering of the pins of J1 depends on whether you used a header, or a socket. Headers are numbered from left to right with odd pins on the top and even on the bottom. Sockets on the other hand are numbered from right to left, again with the odd pins on the top, and the even pins on the bottom.

Starting with pin 10 of J1, run a wire to pin 13 of IC3. Run another wire from the remaining hole at pin 13 of IC3 to pin 11 of IC2. Run a 4.7k ohm resistor from the remaining hole at pin 11 of IC2 to the power bus. See Figure 4. Grab your highlighter, and one of the duplicate schematics and highlight the circuit you just wired. It should be labeled DRV A on the schematic. Now start at pin 12 of J1 and using the same procedure run the DRV B select line, and all the rest of the circuits as shown on the schematic, highlighting each circuit as you put it in.

A SMALL HINT

Before you start running wires, you might want to use one of your duplicate copies of Figure 3 and your colored pencils to decide how to route the wires, and where to put the resistors. The way we ran the DRV A line is not the way that's shown on the schematic, but electrically they're the same. We could have put the resistors from the extra hole at pins 10 & 12 at J1 to the nearest power buss (See Figure 4) and it would still be electrically the same. The reason we can get away with it is that we're working at relatively low frequencies and with relatively short conductor paths.

TO CONTINUE

When you get to the SIDE SEL, READY, VCC, and FAST SEEK lines, Radio Shack sells a set of "Micro-clip test leads" which are very inexpensive and which do the job ideally. They come as a pair, and are about 12 inches long. Cut them in half and you'll have the four leads you need, except that two will be black, and two will be red. Pieces of masking tape or spots of plastic model paint on the clips will handle the labeling.

When the schematic diagram is completely highlighted, you are finished with the wiring phase and it's time for INSPECTION.

INSPECTION

This is our last line of defense against catastrophes. Actually, it isn't all that bad, We're only working with 5 volts, and even inverted polarity won't smoke anything. That isn't always true, but with these components, we're lucky.

Using the second duplicate schematic as a reference, check each conductor's endpoints on the circuit board and as you find them correct, highlight the conductor on the schematic diagram. This may seem redundant, but it's the only way to be sure.

TESTING

The easiest and scariest part, will it, or won't it? Nothing left to do but take the bull by the horns. Install the board as you planned either by plugging it onto the motherboard, or using an auxiliary cable (don't forget to insulate the bottom) and plug the 34 conductor ribbon drive cable onto it. Set the switches appropriately, and now it's time to deal with the clip leads.

The READY line

Find and carefully remove the floppy controller. It will be a 40 pin flat pack IC, designated as U-82 on '83 models, U-44 on '84 models, and U-74 on K-10 '83 models. Carefully bend out pin 32 to about a 45 degree angle and reinsert the floppy controller into its socket matching the notch in the IC with the notch in the socket. Connect the READY microclip to pin 32, the one you just bent out.

VCC

The Personality/Decoder board must be supplied with +5 volt DC power. This is done by connecting the VCC microclip to pin 14 of U-86 on '83 models, pin 14 of U-72 on '84 models, or pin 16 of U-32 on K-10 '83 models.

If you have installed 96tpi drives, you can install the FAST SEEK option. It is a two step process, first connect the FAST SEEK microclip to pin 22 of the floppy controller by removing the IC, bending out the pin (22) and reinserting the IC into its socket (see "The READY Line" above), and second, use TURBOCFG to specify the appropriate step rate. See your TurboRom manual Chapter 4 for particulars.

SIDE SELECT

If this installation is part of a K-II to K-IV upgrade project, AND the K-II has not been modified to use double-sided drives, cut the conductor identified as J3 on the schematic and connect the SIDE SELECT microclip to pin 13 of U-72.

If Your Computer is already using double-sided drives, remove this microclip lead to prevent inadvertent shorts.

NOW

Power up the computer, the power LED will light, the disk drive should run, and the Drive A LED should light.

WHAT TO DO IF IT DOESN'T

First turn off the power and disconnect the power cord, then do a physical inspection, making sure that the right ICs are in the proper sockets, and none of the pins got folded under. Second, reinspect the wiring, using the schematic, paying close attention to any solder that might have sneaked over onto a forbidden solder pad creating an unplanned short-circuit. Third, unplug the ribbon cable to the drives and check that the select line for FD1 is high (5.0 V) initially but goes low (0) when the computer tries to boot. If it does, but the computer still won't boot, recheck the dip switch to make sure that sw2 is set for the drive you have as DRV A. The next suspect is the ribbon cable itself.

As a last resort you can reach me weekends and evenings at (916)483-0312, or CompuServe 73664,2470, or Internet 73664.2470@cis. GOOD LUCK AND HAPPY SOLDERING :)

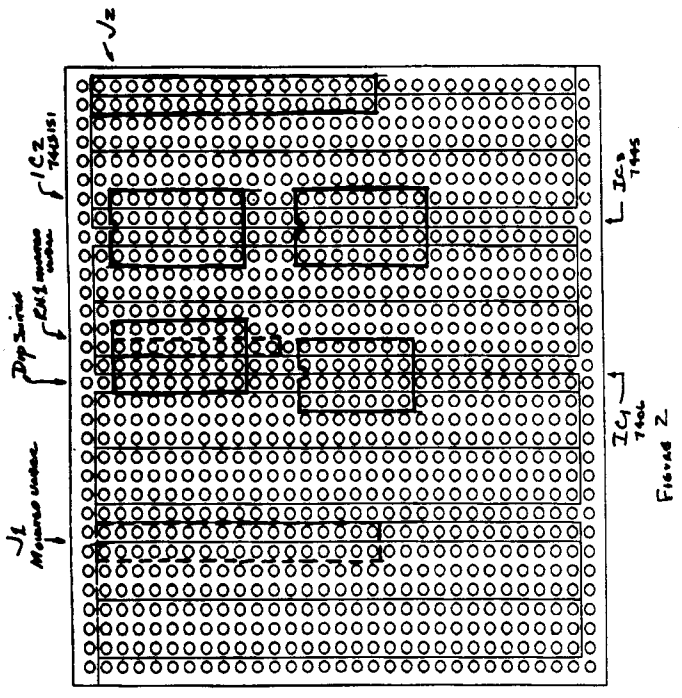

Figure 2

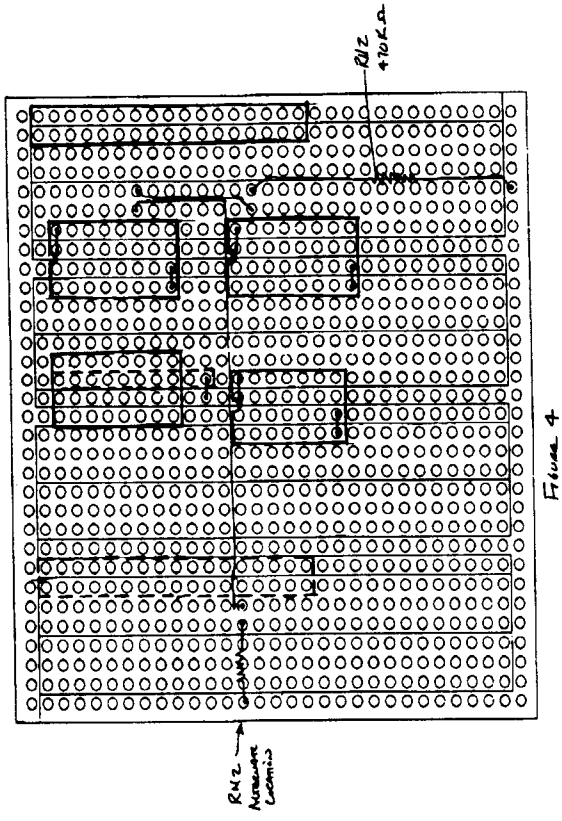

Figure 4

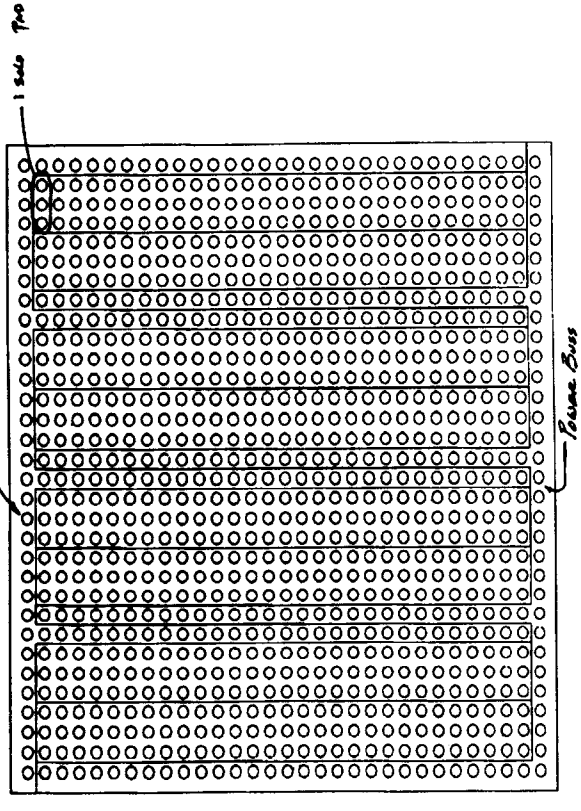

Figure 1

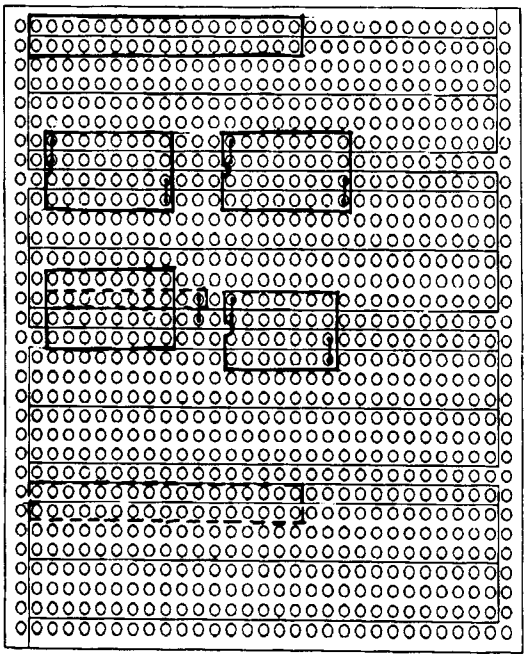

Figure 3