TCJ Center Fold

Mr. Kaypro

By Charles Stafford

Special Feature

Intermediate

Advent Decoder Board

WHEREIN We Undertake The Construction Of An Implant

In The Beginning, there were ANALOG computers, and only HIGHLY QUALIFIED WIZARDS (HCWs) were allowed in the same room with them. These ANALOG computers conversed in varying voltages and currents and all was well. Since they were HARD-WIRED and constructed of potentiometers, coils, meters, and relays, they were RELIABLE if not inscrutable.

Then Came The Binary and Boolean Algebras, and VACUUM TUBES and later SEMI-CONDUCTORS, and MICRO-PROCESSORS, and DIGITAL computers. The HCWs eschewed their ANALOG computers and defected en masse to the new gods of speed. These DIGITAL computers conversed in 0s and 1s and their communications were voluminous, and required the efforts of KEYPUNCH OPERATORS (a lesser form of HCW) and PROGRAMMERS (a special breed unto themselves), and these communications were contained in many, many boxes of cards which were carefully warehoused.

The Quest for Speed continued, and MAGNETIC TAPE was re-discovered, as was ROTARY MOTION and the DISK DRIVE was born and mutated into several varients. And the HCWs were ecstatic, because only they could see the BITS and BYTES on the diskette.

Today the common sizes are 5.25" double-sided double-density (DSDD) (360k), 5.25" high-density (HD) (1.2mb), 3.5" DSDD (720k), and 3.5" HD (1.44mb), with 3.5" very-high-density (VHD) (2.88mb) on the horizon. (I made up the VHD designation). Most KayPro s were delivered with the 5.25 DSDD variety, the exceptions being the K-2s, which had 5.25 SSDD 180k drives, and the Robies and K-4Xs which had 5.25 2.6mb DriveTec drives. The DSDD drives in the KayPros had a capacity of 390k, 10% more than their IBM brethern, but more is better, right? Sometime during the drive evolution, the QUAD density drive was developed, with 96 tracks per inch instead of 48, resulting in 160 tracks total and a capacity of 790kb.

Some of the more enterprising HCWs transplanted these DSQD drives into K-4s and thus were born K-8s. Microcornucopia and Avent Products formalized the modifications and they became very popular since they were within most budgets and HARDDRIVES were extremely expensive.

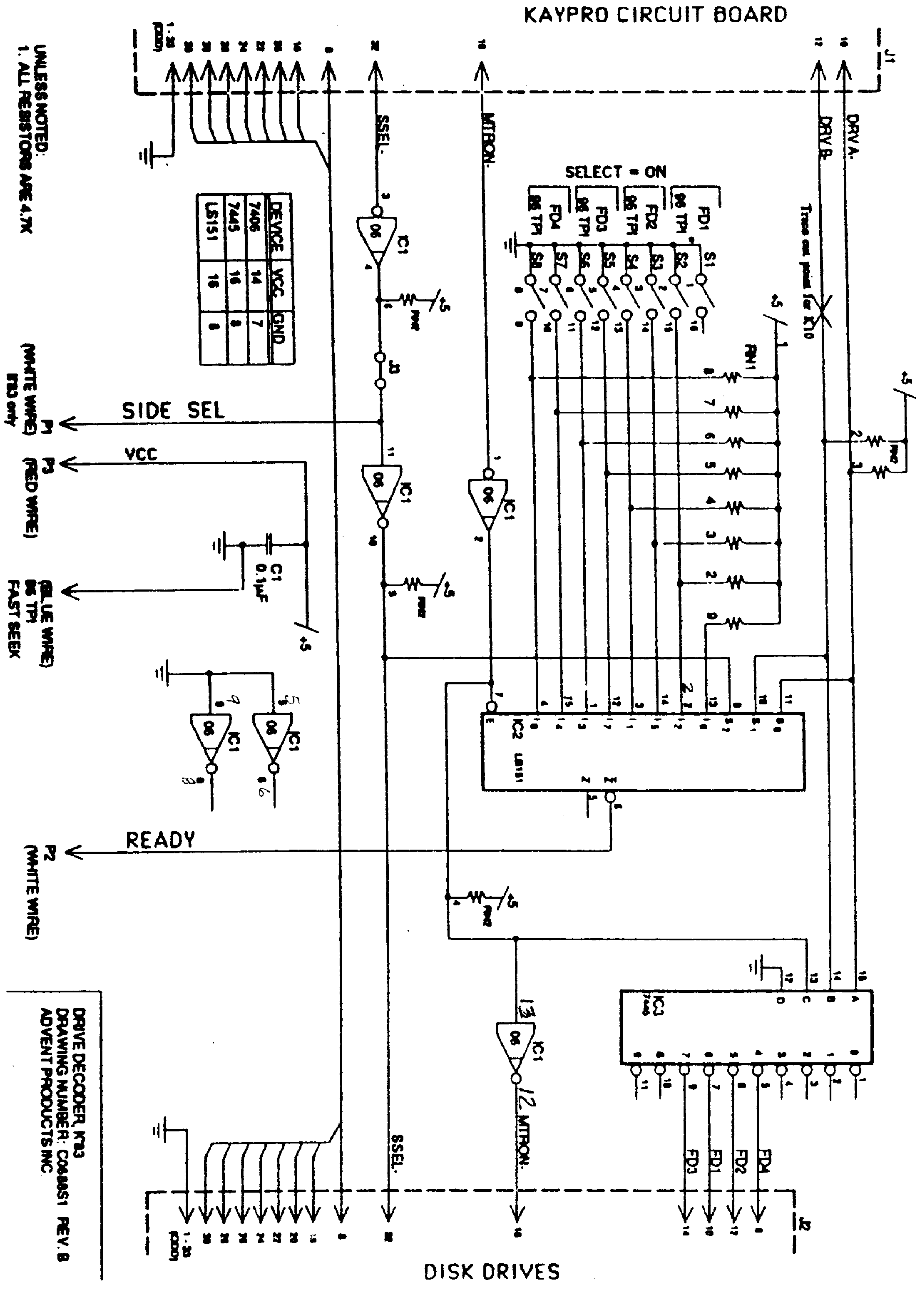
The two conversions used different designs based on their respective monitor roms. The MicroCornucopia (MicroC) monitor rom uses a utility program to configure the drive bios and thus only requires a multiplex decoder on the drive select lines to select the proper drive. The KayPro mother-boards were designed with only two drives in mind, so only drive select A and drive select were implemented. By multiplexing both lines we can come up with four choices, and one of four drives. The drawback to using the MicroC rom is that when you install a harddrive, the Micro-C rom won't boot off the harddisk.

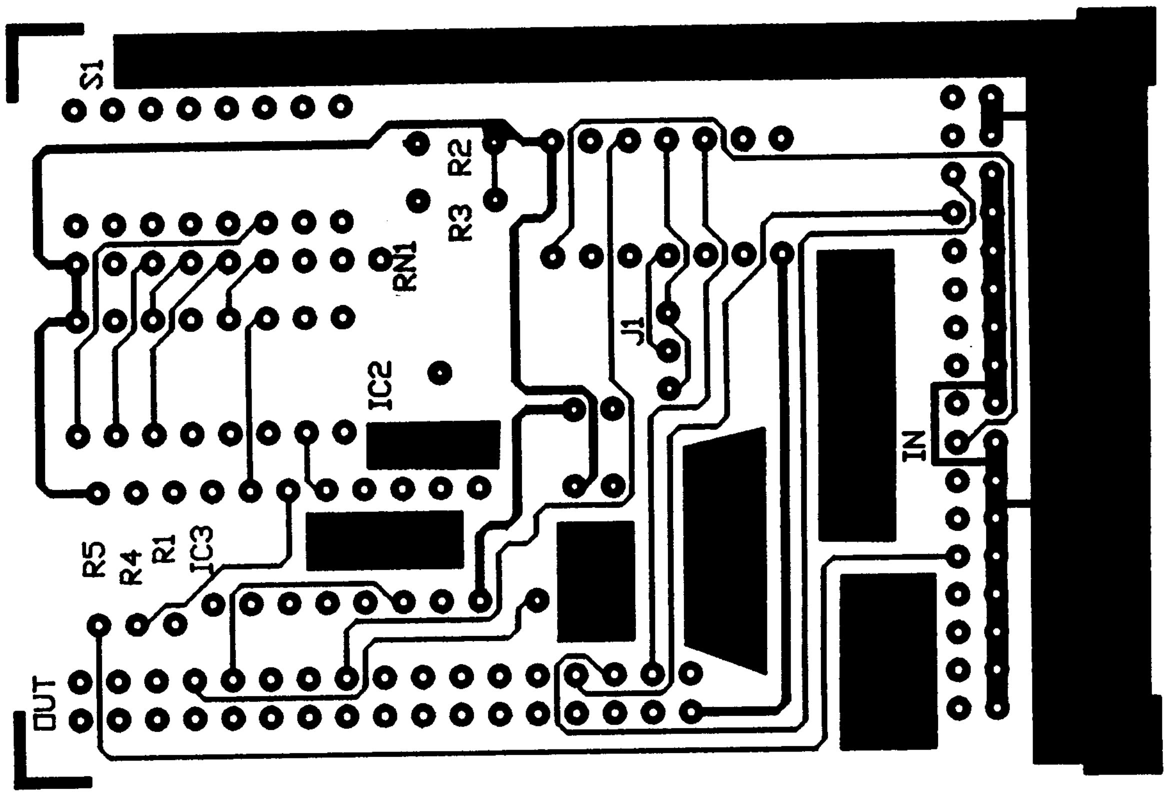
The Advent TurboRom on the other hand will boot off a harddrive, but dynamically configures the drive bios based in an inquiry to the ''Personality-Decoder Board''. The advantage is that reconfiguring drives only requires only rearranging the hardware and setting the switches on the PDB. The disadvantage is that the PDB is more complex. Unfortunately Advent is out of the CP/M business and the stock (theirs and mine) of PDBs is completely gone. Fortunately, we can build one fairly easily.

We will attack this project on two parallel tracks. One, with a custom printed circuit board that you can make, if you feel up to it, and, the other, with an off the shelf prototype board. The difference is where you put the labor, into point-to-point wiring, or into the manufacture of a printed circuit, which will be considerably smaller.

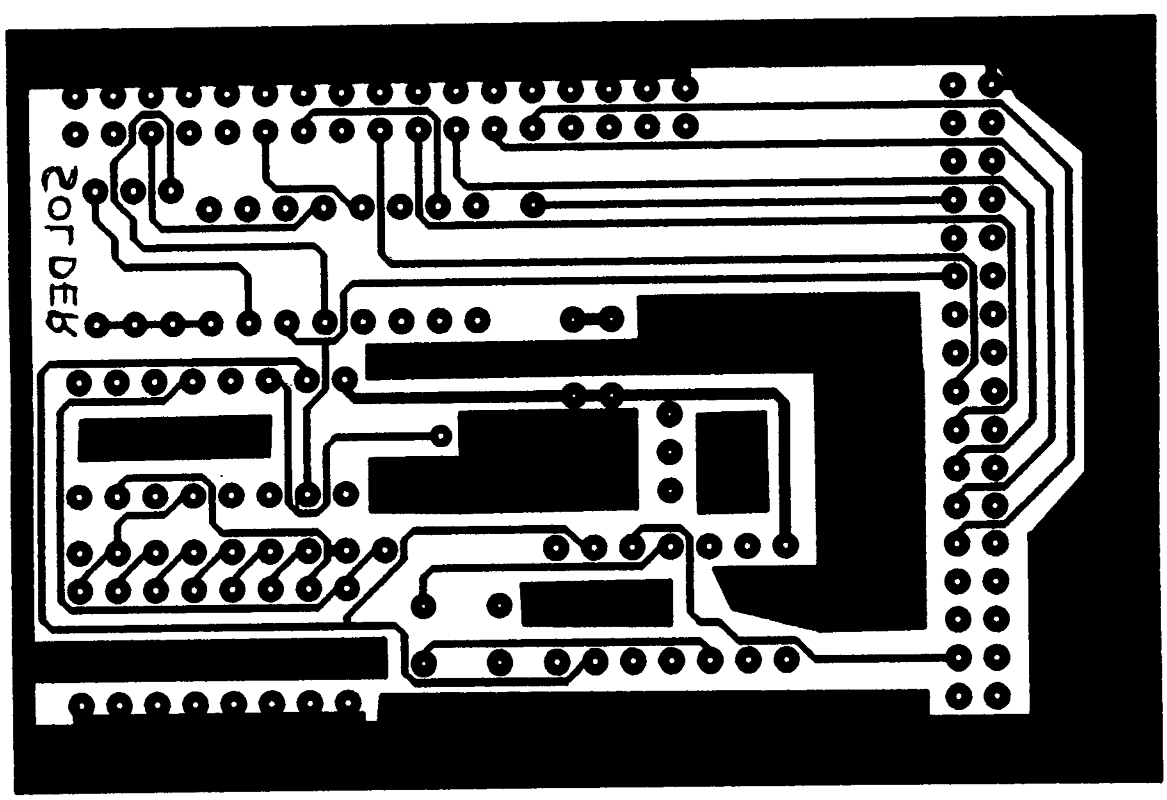
This will also be a two article project. We will start with making the custom circuit board, and do the assembly in the next issue.

Some time ago I mentioned that I knew nothing about making circuit boards and Ed Fanta called me and volunteered to make some for the PDB. The process that seems the best for the novice is the copier method. Here are Ed's notes on the entire process and the ''plots'' as well. NOTE These plots are twice as big as you will want for the finished product. Reduce them to half size on the copier you are using and they'll be just right. Experiment on plain paper before making the mask material to make sure the size is proper. You can check the size by measuring the distance across eleven pins of an IC. The spaces


KAYPRO CIRCUIT BOARD
J1
DRV A.
DRV B.
SSEL.
MTRON.
SELECT = ON
FD1
FD2
FD3
FD4
96 TPI
S1
S2
S3
S4
S5
S6
S7
S8
Trace cut point for K10
RN1
RN2
IC1
IC2
LS151
IC3
7445
C1
0.1µF
DEVICE
VCC
GND
7406
7445
LS151
SIDE SEL
VCC
READY
P1
P3
P2
(WHITE WIRE)
FD3 only
(RED WIRE)
(BLUE WIRE)
96 TPI
FAST SEEK
UNLESS NOTED:
1. ALL RESISTORS ARE 4.7K
DRIVE DECODER, K'83
DRAWING NUMBER: C0688S1 REV. B
ADVENT PRODUCTS INC.
J2
DISK DRIVES



TOP or component side



BOTTOM or solder side

should be 0.1 inch each, so the distance across ten spaces (eleven pins) should be one inch. The complete directions for use are on the package of material for the masks. Both the mask kits and etchant are available from most electronic hobby stores.

Ed's notes:
Hello Chuck
Things are finally winding down a bit, the light at the end of the tunnel is getting closer. I am getting time to write this and Eric found the time to plot out the etching masks. The following is a few ideas about the PCB process.

Etch Tank

Find a tall, narrow tank. A good start is look in the housewares department. No, no, not the kitchen cabinets, go to a department store and look over the plastic wares. The one I found most suitible was a beverage container for dispensing softdrinks, it is about 10 inches high, 8 inches wide, and about 3 inches deep with a snap on lid. A tray for holding the board to be etched might be made from a smaller such container or, as I have used, a cut down antifreeze jug. The tray should have holes cut in the side and 2 parallel rows of holes cut in the bottom. The holes in the bottom are for our next stop, the aquarium department. The small aeriator pumps work well to keep the etchant stirred up. The holes in the bottom have a section of aeriator tubing laced through them, melt holes in the tubing with a piece of wire heated over a candle or some such, then heat the end of the tube so the plastic just starts to melt and squeeze the end shut with a pair of pliers. Always keep the pump above the liquid level or remove the tray when not in use. Now look a little further down the aisle from the pumps and pick up an aquarium heater, a small one is just fine. Make sure it will fit over the lip of the etch tank and will clamp securely.

Drilling circuit boards

Drilling is the most tedious part of the process, patience and a good eye are required. If you get a good etch the centers of the pads will be etched and will help center the drill bit, if not use a sharp awl to mark the centers. If your registration between sides of the board is not perfect or the bond of the copper to the fiberglass is weak it is likely you will peel the traces off the off side of the board. One way to get around these problems is to drill the board half way through, flip the board over and finish drilling. Twice the fun! A good drill press with a spindle stop comes in very handy, see if you can borrow the neighbors. Carbide drill bits are used in the industry as they maintain a good sharp edge. For hobbiest use go to a hardware store or a machine shop supply and buy a couple of no. 63 or 64 size high speed bits, the edge will not last as long but they will bend long before they break.

Putting on the etch resist

Etch resist can take several forms. Photosensitive, manually applied rub-ons, iron-on, and silk screen. Manually applied is OK for small, simple jobs, but is tedious for anything more than a couple of IC's. Photosensitive has been the way to go in the past as it will give good resolution, with the tradeoffs of hard to register on a double sided board, and getting the proper exposure. Silkscreen is the way to go when making a bunch of boards but has a higher initial cost for the equipment and still takes some work to get a double side board registered. Iron-on is probably the best for the average hobbiest as it has the lowest cost and takes no special equipment. There are several types of iron-on film available, many are advertised in magazines. Another idea is to use transparency film made for use in a photocopier. It works almost as well and is available at most office supply stores. The big problem to using this method is finding the right copier, many (particularly Xerox) fuse the toner onto the film at too high a temperature and it will not transfer to the board properly. Use of the film is simplicity, simply copy your artwork onto the film, lay it on your board, and run over it with a laundry iron. Be very careful with the film after copying as the toner will flake off, do not bend or fold the film. Registration of the two sides can be a problem unless you have X-ray vision, an idea for that is to add a couple of extra pads in opposite corners of the board and drill them after ironing on the first side, then simply sight through the film to line up the holes. A small prick punch or a fine awl should be used to mark the holes for the drill.

Etching the board

Good etchant, good agitation, and the proper temperature make for a well etched board. A properly set up tank will etch a board in 5 to 8 minutes when the etchant is fresh, if the etch time rises above 12 to 15 minutes it is time to replace the etchant. Try to keep the temperature around 110 degrees.

Soldering the components

Most hobbiest boards will not have the luxury of having a solder mask screened on so we fake it! If you have a good steady hand and a fine brush you can use paints or laquers to make a fine line between pads to keep the solder from jumping across and making a solder bridge. If you have an unsteady hand and slop some on the pads simply use a small tool to scrape it off the pad. Tinning is a good way to make it easier to make the solder take hold. There are several tinning solutions available but another way is to apply a very thin coat of paste flux, a little solder and then chase it around the board with a soldering iron.

Be careful not to let the layer get too thick, you want just enough to coat the pads. When soldering a double sided board without plated through holes there is the problem of soldering the component side. One way is to cut short pieces of fine wire and place them in the hole with the socket lead. Another way is to forgo the sockets and solder the IC leads directly to the traces, just make sure you have good components. A third way

Continued on page 33

4. General access: First access, initialization

```
procedure HD_Init(Cyls,Heads,Secs:integer);
begin
port[Dig_Out]:=6;
delay(10);          (* Drive Software Reset *)
port[Dig_Out]:=2;
Wait_Ready;
port[IDE_SecCnt]:=Secs;
port[IDE_CylLow]:=lo(Cyls);
port[IDE_CylHigh]:=hi(Cyls);
port[IDE_SDH]:=pred(Heads)+$A0;
IDE_Command(Cmd_Initialize);
end;
```

5. Data access: Single sector read

```
function HD_ReadSector(Cyl,Head,Sec:integer; var
Buf:BufType):boolean;
begin
Wait_Ready;
port[IDE_SecCnt]:=1;
port[IDE_SecNum]:=Sec;
port[IDE_CylLow]:=lo(Cyl);
port[IDE_CylHigh]:=hi(Cyl);
port[IDE_SDH]:=$A0+Head;
IDE_Command(Cmd_ReadSector);
HD_ReadSector:=Read_SecBuf(Buf);
end;
```

7. Data access: Single sector write

```
function HD_WriteSector(Cyl,Head,Sec:integer; var
Buf:BufType);
begin
Wait_Ready;
port[IDE_SecCnt]:=1;
port[IDE_SecNum]:=Sec;
port[IDE_CylLow]:=lo(Cyl);
port[IDE_CylHigh]:=hi(Cyl);
port[IDE_SDH]:=$A0+Head;
IDE_Command(Cmd_WriteSector);
HD_WriteSector:=Write_SecBuf(Buf);
end;
```

Now we have reached the end of the "behind IDE" article series. In another column I will describe my revised IDE interface board for the 8-bit ECB bus in somewhat more detail than in TCJ #56. This will include a TTL equivalent of the GAL contents, for those who are inexperienced in reading a Boolean equation design, or who want to build it up using discrete logic.

For a list of abbreviations, see parts II and III of this article.

Mr. Kaypro. Continued from page28

is to use wire wrap sockets and leave them a little ways above the board so you can reach under with the solder using the iron on the end of the lead under the board and letting the heat travel up the lead. A special thank you to Eric Craig of C&C Machine for doing the plotting and letting me do some of the layout work. Thanks Eric!

Chuck
I hope this looks good enough to be used in one of your articles, another set of plots will be coming soon as I was not happy with the set you received earlier. Ed.

Rest assured Ed, and the new plots will be published in the next issue. The list of parts that will be needed to complete the project is presented last.

The circuit board can either be one you make using the plots printed, or one of the prototype boards listed above. Both 218905 and 207906 are 3/4 hole per pad double sided, with through plated holes, while 462905 is single sided. 462905 will work just fine, but will require more thinking, when it comes to connector layout.

For those of You who elect to manufacture your own printed circuit board, good luck. For those of you who elect not to, hang loose and we'll finish construction and installation in the next issue.

P.S. I owe some of you replies to your letters, keep the faith, and I'll get to it "real soon now". CBS

Foundation	1ea . custom printed circuit board OR SYNTAX # 218905 or 207906 or 462905
ICs	1ea. 7406 1ea. 7445 1ea. 74ls151
Sockets	1ea. 14 pin dip 2 ea. 16 pin dip
Resistors	1ea. 8 section sipp 4.7k per section 5ea. 4.7k .0125 watt
Miscellaneous	1ea. 8 position dip switch 1ea. 34 pin IDH pcb connector 1ea. 34 conductor double in-line header 1ea. 0.01 microfarad disk ceramic capacitor