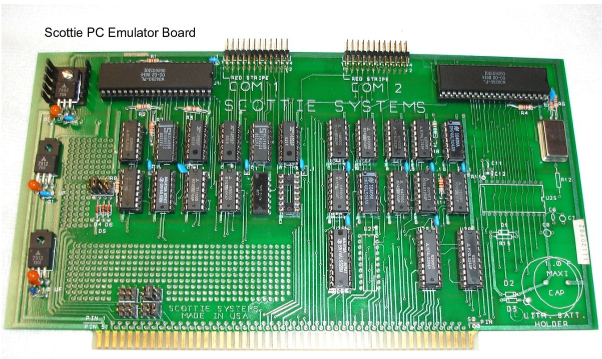
Z-100 LifeLine
Since 1989

April 2022

#WEB

This article was published in issues #74/75, April & June 2001
~~~~~~



Scottie Board PC-Emulator For the Z-100

By Steven W. Vagts
Editor, "*Z-100 LifeLine*"

SCOTTIE BOARD
PC-emulator for the Z-100

HISTORY

Note: A complex, thorough review of the IBM-compatibility issue and the Scottie Board was published by Timothy Weil in Sextant #34, Late Spring 1988, entitled **"IBM Emulation With the Scottie Board"**. For those without access to this issue, I summarized the pertinent parts here.

Patrick Swayne, Heath/Zenith Users' Group Software Engineer introduced ZPC, a software solution to the IBM-compatibility problem in September 1985. This program had continued to evolve over the years to take advantage of numerous hardware and software upgrades, including the new version 4 monitor ROM and the heavily modified version 4.05 of MS-DOS.

Nevertheless, this software approach to IBM PC-emulation had certain limitations that could not be overcome. This required that the application programs also be "patched" to make them work.

In the April 1986 issue of REMark, Patrick Swayne introduced a "simple circuit" design to reduce the number of programs requiring patches. The **ZPC Hardware Support (ZHS)** board was a S-100 board designed to run IBM software with minimal user intervention.

The new board reduced program patches considerably, but required users to build the board from scratch.

The Scottie Board, designed by Bob Sicotte of Scottie Systems, took the ZHS design a bit further. There were options for a real-time clock and for IBM-compatible serial ports (COM1 and COM2).

The combination of the Scottie Board and ZPC yielded an IBM-compatible system that rivaled the other two popular Z-100 hardware emulator alternatives, **EasyPC** from UCI Corp. and **Gemini** from Gemini Technology Corp., at about a third of the cost.

The Scottie Board required 768 kilobytes of random-access memory (RAM) and version 2 or higher of the Z-100's native MS-DOS. This was a major factor, as the other two emulator boards required Zenith's PC-DOS be loaded in a separate partition on the computer's hard drive.

There are four main areas of incompatibility between the Z-100 and IBM PC that are overcome by the Scottie Board hardware and ZPC software combination:

1. Mapping of Input/Output Addresses:

The I/O addressing problem represented the difference between 8 and 16-bit computer architectures. While the PC could conceptually access up to 65,536 port addresses, the Z-100 used a maximum of 256.

If a program tried to write to one of those 16-bit addresses, the Z-100 would simply ignore the upper 8 bits. The Scottie/ZHS board provided hardware to intercept or lock out certain addresses and provided the necessary video and keyboard ports for video memory access and interrupt control.

2. Location and Nature of Routines in the Basic Input/Output System (BIOS):

The IBM and Z-100 BIOS routines are stored in different locations. The PC had its BIOS routines stored in read-only memory (ROM), whereas the Z-100 loaded its BIOS into RAM from disk on bootup.
~~~~~~

The PC's ROM BIOS included graphics routines that were not present in the Z-100's BIOS.

There was a difference in the set of interrupt signals used by each machine's keyboard, peripheral devices, etc. to gain the attention of the CPU. Calling a certain interrupt on the PC could cause a completely different response from the Z-100.

To fix all the above, ZPC monitored calls to the ROM and contained translation tables to route the calls appropriately.

Finally, when attempting to write to video memory, PC programs would test it first for an appropriate value before writing. With the Scottie and ZPC combination, the PC video commands would use the video-control port on the Scottie Board.

3. Location of Video RAM:

The PC's video RAM was lower in the address space than Z-100 video RAM. ZPC reserved an area of memory to emulate the PC's video RAM.

4. Different I/O Chips Are Used:

The Z-100 used 2661-2 serial I/O chips while the PC used 8250 chips. The Scottie Board provided optional PC-compatible COM1 and COM2 serial ports.

OPERATION

Operation of the Scottie Board was virtually transparent. Using the SETZPC utility, ZPC's program parameters could be customized to suit a particular Z-100 environment. ZPC had undergone numerous revisions over the years and a full description was outside the purpose of this article, but a few of the more important and pertinent options are:

* **Video mode** (0-7): You could choose color or monochrome, adjust column width, and choose graphics-board emulation (CGA). This setting could be adjusted to accommodate different application programs.

* **Font Style** (IBM-PC, Z-100, or user supplied): ZPC allowed a flexibly defined character style, including the use of the Z-100's ALTCHAR.SYS.

* **Normal Monochrome Color** (green or yellow)

* **NumLock Setting** (on or off): This set the default status of the Num Lock key in PC mode.

At the conclusion of SETZPC, you had the ability to either save this configuration or activate it for the current session only. Should you switch between Z-100 and PC modes, ZPC would default to the last parameters saved.

To accommodate the requirements of different application programs, the video mode could be altered every time the emulation mode was entered - on the command line that invoked emulation. You just had to include the appropriate number of one of the eight mode settings.

Going into PC mode was as simple as typing PC (or PC n, where n is the video mode number); you just typed Z-100 to exit.

The keyboard layout for an IBM PC differed radically from the Z-100 layout. ZPC documentation supplied a diagram which mapped IBM keys to Z-100 keys, but this mapping varied between revisions. Please refer to the readme files that accompanied each ZPC release.

THE ZHS FUNCTIONS SECTION OF THE SCOTTIE BOARD

Overview: I/O reads from address range 3D8H-3DBH generate an alternate 1 and 8 (bits 0 and 3) on the S-100 data bus. This simulates a status read from the IBM-PC's CRT Controller Status Port (Video Retrace Port Test of ZHSTEST.COM).

I/O Writes to this address range generate VI6 (Vectored Interrupt 6), which is serviced by Pat Swayne's ZPC Software in normal Z-100 fashion (Video Control Port of ZHSTEST.COM).

Hardware: Refer to the Scottie Board Schematics, Figures 3 and 4. U20, enabled only for I/O activity, decodes address range 3D8H-3DBH. When this range is addressed, a low asserted signal is output at U20 pin 8, and is inverted by U21 to enable U10 pins 2 and 5.

Reads: pDBIN (Processor Data Bus Input) is generated at S-100 bus pin 78, satisfying U10 pin 1. U10 outputs its low asserted pin 3, enabling both U23 Tri-State gates to place D0 and D3 onto the Scottie Board Internal Bi-Directional Data Bus (see Figure 4).

In addition, U10's pin 3 enables U24 pin 2. U24's output pin 6 enables Tri-State Buffer U28 pins 1 and 19, placing the Scottie Board Internal Bi-Directional Data Bus onto the Computer's S-100 Bus.

Writes: pWR (Processor Write asserted low) from S-100 bus pin 77, inverted by U13, satisfies U10 pin 4 and U10's pin 6 is asserted low. U10 pin 6 force sets D Flip-Flop U16, asserting a high at U17 pin 5. Inverted by U17, its output pin 6 generates VI6 (Vectored Interrupt 6) on the S-100 Bus pin 10 and the ZPC Software handles it in Z-100 fashion.

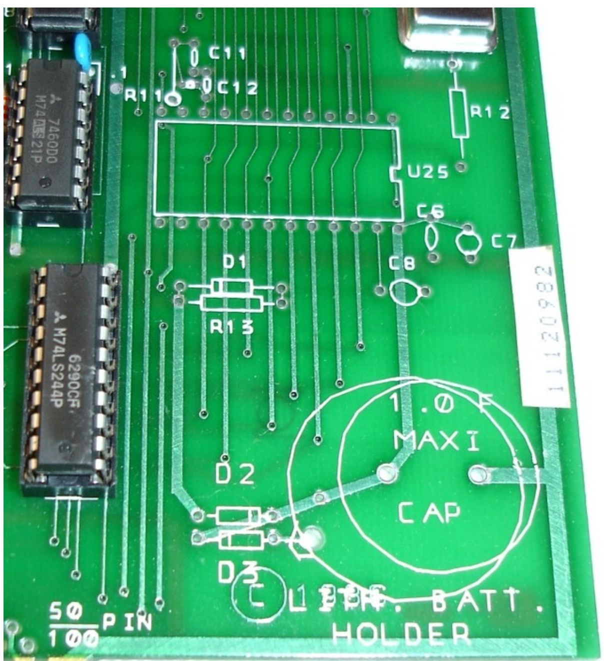
When the signal INT is asserted on pin 73 of the S-100 bus by the 8259A Interrupt Controller, D Flip-Flop U16 is clocked to the reset state, removing VI6.

Hardware: U26 decodes I/O addresses 60H and 61H and asserts its pin 19, enabling U14 pin 9. pDBIN (S-100 buss pin 78) satisfies the AND function at U14 pin 10, force setting D Flip-Flop U16 (second half). U16 pin 9 is asserted high, inverted by U17, and VI7 is generated on the S-100 buss, pin 11.

Again, when the master interrupt controller generates INT on the S-100 bus, the D Flip-Flop U16 is clocked to the reset state, removing the VI7 request.

THE SCOTTIE CLOCK

Overview: The Scottie Clock (if installed) is usable for real time scientific applications. It counts month, date, hour, minutes, seconds, 1/10 seconds, 1/100 seconds, and 1/1000 seconds (milliseconds).



Scottie PC Emulator Board
Real Time Clock area

When the computer is powered down, the clock runs at very low power drain from either a lithium battery or a 1 farad capacitor. When operated at approximately 2.65 volts from the lithium battery, clock current drain is about 160 pica amperes. With higher input voltages, the drain is increased but is still a very small nominal value.

Capable of interrupting as fast as 1 millisecond, or as late as one year minus one millisecond, it can be used for scheduler operation or other long term timed event operation as long as the event does not occur during machine power-down.

Programming: Programming is quite simple. The clock IC's RAM locations are loaded with BCD values which will constitute a match with the clock IC's hardware registers, at the time the scheduled interrupt is desired. If a specific hardware register match is not needed, the hardware registers RAM counterpart can be loaded with its most significant bit set, which constitutes a match in all cases and may be used to contain the year. (For more information on programming, see the National Semiconductor Data Sheet for the MM58167A).

The Clock has 32 port addresses assigned from 02C0H thru 02CFH. See the Scottie Clock Support Program assembly source code files for specific Hardware Register and RAM address assignments.

As delivered, one of the RAM locations is used to store the year, and the clock does not generate interrupts. It can easily be tied into one of five available Vectored Interrupt lines readily accessible on the Scottie Board (inverter buffered).

The Clock IC's programming is protected from spurious activity during the computer's power up cycle by a timing circuit.

Hardware: Refer to Figure 5. When the Clock IC's (U25) PDN pin 23 is low, the Clock IC refuses all access regardless of the state of the control pins 1, 2, and 3. Power up RC timing to this pin consists of a 1 uf dipped tantalum capacitor and 100K resistor. U25 PDN pin 23 is held low for approximately 75 milliseconds during power up. At power down, the 1 uf capacitor is rapidly discharged to the waning +5 volt power regulator thru a 1N4148 diode.

I/O addresses in the range 2C0H-2CFH are decoded by U27. The least significant address bits A4 thru A0 are presented directly to the Clock IC U25. U27 output pin 19 is asserted low, enabling Clock CS pin 1, and tri-state RDY line driver U23 pin 13. The Clock IC internally addresses the appropriate hardware register or RAM location. Read/Write activity with the Clock IC is controlled, with CS asserted, by the WR and RD pins. These controls are timed by S-100 bus signals pWR (pin 77) for writes and pDBIN (pin 78) for reads, thru U11.

When the Clock IC, U25, is accessed during a ‘roll-over operation’ (when updating its registers), U25 pin 4 is pulled low by the Clock IC stating that it is not ready for the access. This is buffered onto the S-100 bus RDY line 72 by the now enabled U23 tri-state driver, and holds off access completion until U25 is ready.

Two gates of NAND gate U22 control the Scottie Board's internal bi-directional data bus interface to the S-100 bus when used by the Scottie Clock. U22 pin 3 enables U24 pin 12 (see Scottie Board Schematic page 2), which in turn enables U28 pins 1 and 19, for writes to the clock from the S-100 bus DO lines.

Similarly, but in the opposite direction, U22 pin 6 enables U24 pin 5, which in turn enables U29 pins 1 and 19, for reads from the clock to the S-100 bus DI lines.

Clock IC, U25, power is derived from one of two sources:

- For operation when the computer has power on, via a 1N4148 diode, D2, from the +5 volt regulated power. Being higher than the 3 volt lithium battery, a 1N4148 diode, D3, in series with the lithium battery is blocked, and no current flows from the battery, nor does the +5 volt regulated power reach the battery.

- When the computer is powered down, diode D2, in series with the +5 volt path, is reversed biased, blocking flow into the shutdown +5 volt regulator and battery power flows to the clock voltage decreased by one diode drop (thru D3).

When the 1 farad capacitor option is installed in place of the lithium battery, diode D3, in-line from the battery, is replaced by a shorting wire, and a 1N4001 diode is used in place of the 1N4148 diode, D2, in series with the +5 volt bus.

This allows startup for a completely discharged 1 farad capacitor, which initially presents a dead short load to the +5 volt regulator. Initially, it takes approximately 5 seconds to charge the new 1 farad capacitor to a value which draws a current value within the current handling capability of the +5 volt regulator.

For normal operation, the 1 farad capacitor never drops lower than the +5 volt regulator can handle (unless you discharge it on the bench), and can operate the Clock IC for months with the computer power off.

WAIT STATE GENERATOR

Overview: A single wait state may be inserted when operating at 8 Mhz (or even at 5 Mhz), and is primarily intended for use with the COM ports. The 8250A used in the COM ports, although the fastest available, have a guaranteed access time of 165 nanoseconds. Many will operate at 7.5 Mhz and some at 8 Mhz. Bottom line, go for reliable operation without the shorting blocks. If problems arise, then insert a wait state.

Wait state paths have also been provided for VI6 and VI7. The diode OR gate is expandable to any number by the addition of 1N4148 computer diodes for your circuits.

The three diodes which are normally present are, from left to right (with the board viewed top edge up):

1) VI7 (keyboard port) wait state
2) VI6 (video control port) wait state
3) COM1 and COM2 ports wait state

Hardware: Refer to Figure 3. Hardware consisting of U3, part of U14, U15, and part of U17 generates the wait states. Wait states may be selected for VI6, VI7, and COM1 or COM2 activity. This is owner controlled by installing shorting blocks which complete the path for diode anode hookup. When any cathode is pulled low, U14 pin 13 is enabled, generating one wait state.

At the trailing edge of the high half phase of T1, from the S-100 buss, pin 24, U17 pin 10 goes low clocking U3 pin 14. Internally, U3 counts to 1. When the second clock pulse is received, U3 internally counts to 2, asserting pin 9.

As the high half phase of clock cycle T3 goes low, the inverted clock from U17 pin 10 goes high, satisfying U15 pin 5 and U15 output pin 6 asserts high. U17 inverts the signal which pulls down the XRDY line at pin 3 of the S-100 bus (XRDY is an OR function with RDY bus pin 72).

With the XRDY line low during the low half phase of T3, a wait state of one clock cycle is injected. T4 begins on the next clock rising edge. When the CPU begins the next T1 of an instruction fetch, pSYNC is generated at S-100 bus pin 76 and counter U3 is cleared.

SCOTTIE BOARD COM OPTIONS

Overview: COM1 and COM2 are virtual clones of the IBM-PC Asynchronous Interface Adaptor and Aux Asynchronous Interface Adaptor. These are implemented with 8250A USARTs. A Master Crystal Oscillator provides a 1.8432 Mhz clock pulse to pin 16 of each COM port. The TxBAUDCLK pin 15 of each COM port is hooked to their own RxBAUDCLK pin 9. All I/O signals of the USART are at TTL levels.

The COM1 base address is at 2F8H. COM2 is located at 3F8H. Address bits A2 thru A0 are presented directly to the USARTs.

The guaranteed access time of the INS 8250A is 165 nanoseconds. All of those used on the Scottie Board to date will run at 8 MHz.
If a problem arises, just inject a wait state. Test, and if it works without the wait state invoked, then leave the shorting block off.
If you have any glitches, then install the shorting block.

Hardware: Refer to the respective COM port schematic, Figures 6 (COM1) and 7 (COM2). The U9 address decodes the range 2F8H thru 3F8H and asserts a low from its pin 8 when I/O is done to that address range. U9 pin 8 enables U11 pin 5 and both USARTs U1 and U2. The inverted signal from U21 pin 12 enables U10 pins 10 and 12, and U15 pin 12.

Address bit 8 from the S-100 buss pin 84 is used to select the COM1 USART (pin 12), and inverted to select the COM2 USART (if A8 asserted, then COM1; if A8 not asserted, then COM2).

Once fully selected by asserted signals CS2 on pin 14 and CS0 on pin 12, one or the other USART will respond (dependent on address bit 8) to either DOSTR for writes to the USART, or DISTR for reads from the USART. DISTR is timed with S-100 bus signal pDBIN, and DOSTR is timed with S-100 bus signal pWR.

Both COM1 and COM2 have RS-232 1489 receivers, and 1488 transmitters for the 8 lines shown on the schematic. These signals are available near the top edge of the card as a Gold Flash 2 x 12 x .1" header. A small ribbon cable is recommended to run from the header to the rear panel of the computer to facilitate standard serial cable hookup.

Both COM port cables, as supplied, were DTE, although DCE could be fabricated and may be found with some boards.

S-100 BUS TIMING

General and Simplified Overview: (Refer to ANSI /IEEE Standard 696 for detailed Bus specifications)

Writes:
- Address & data lines asserted by bus master.
- Address & data de-skew delay, lines become stable.
- Bus status lines assert (directing traffic).
- Write accomplished by pWR at its trailing edge (Processor Write asserted low).

Reads:
- Address lines asserted by bus master.
- Address de-skew delay, lines become stable.
- Bus status lines assert (directing traffic).
- Read accomplished by pDBIN, at its trailing edge (Processor Data Bus Input).

H/Z-100 MODIFICATIONS OVERVIEW

Often referred to as the **ZPC V2 modification**, the modifications to the Z-100 motherboard and the video board are made necessary by the H/Z-100 I/O addressing scheme, which uses only the lower address byte for I/O (256 I/O port addresses).

Since no check is made for bits set in the upper byte, it can erroneously decode 16 bit I/O addresses which coincidently have an H/Z-100 port address in the low order byte (i.e., non-H/Z-100 software).

Attempts to use software intended for a machine which uses a 16 bit I/O address range (65,536 I/O port addresses) will result in lock-up or run-away (e.g., IBM-PC software write to I/O address 3DAH will write to H/Z-100 I/O address DAH, unless inhibited by modification).

Three things MUST be accomplished to run the maximum number of IBM-PC programs without patches:

1. Lock-out certain 16 bit address access to H/Z-100 ports (Accomplished by motherboard and video board modifications, which are discussed next).

2. Provide hardware to intercept 16 bit I/O accesses (Accomplished by the Scottie Board).

3. Process new interrupts in Z-100 hardware fashion (Accomplished by ZPC Software).

~~~~~~

Update on ZPC V2 Mod: Mike Zinkow wrote the following regarding the ZPC V2 Modification in response to a comment in Larry Davis' letter in LifeLine issue #39 and several phone calls. It seems that several readers did not understand the need for the ZPC V2 modification. So...

One of the reasons the H/Z-100 crashes when we attempt to run PC software is that the IBM-PC is I/O mapped for 64K, (the only parameter that out-classes the Z-100). That is, it uses 16-bit addressing to control the I/O functions.

The Z-100 is mapped for 256 byte I/O addressing, and only decodes the lower 8 address bits for Z-100 native mode I/O activity (the Scottie Board decodes both 16-bit & 8-bit addresses).

When we attempt to run PC software addressing the I/O ports, the PC software corrupts programming of the video port(s), and/or keyboard port(s). The memory mapping port is also corrupted when using PC communications programs.

This erroneous address decoding is taken care of with a modification to the Z-100, which locks out the PC software attempts to write to or read from non-existent I/O ports. This is accomplished by using the ZPC V2 modification harness assemblies.

The ZPC V2 Video Board modification traps the I/O port calls that corrupts the CRT-C's registers with or without a Scottie board and prevents trashing the video.

The ZPC V2 Motherboard modification traps the I/O port calls that corrupts the Z-100's Memory Mapping port with or without a Scottie board and prevents the Z-100 from going into never-never land.

Unfortunately, if you have a Gemini board installed, this modification cannot be done using a piggyback socket for the I/O Decoder ROM U179 because the Gemini board's mounting position is such that there is insufficient room to install a piggyback socket between the Gemini board and the motherboard, necessitating direct soldering to the U179 PAL's pin #11.

Now the Z-100 is fixed so that it can no longer be corrupted by PC software with or without a Scottie Board.

If you have a Scottie Board installed in the S-100 bus card cage, the Scottie Board fields or intercepts the PC programs I/O port addresses sent to non-existant PC hardware. The necessary flag signals and interrupts are generated by the Scottie Board for the PC Video Retrace port, the PC Video Control port, and the PC Keyboard port so that the ZPC software can translate the commands into those which can be handled by the H/Z-100 hardware and eliminate the need to patch PC programs to run properly on the Z100. The Scottie Board Comm option is a true PC clone serial port.
~~~~~~

Therefore, unless you NEVER run anything but Z-100 native software, you should install the ZPC V2 modification!

~~~~~~

Once the modifications are made, the PC's 16-bit I/O addressing would not be so foreign to the Z-100. The Scottie Board generates the necessary interrupts and other signals so that ZPC will know when to translate PC functions into H/Z-100 functions.

Note: It will also be necessary to still patch some programs to run, even with ZPC version 2 (or greater) software and the Scottie Board because of Illegal Screen RAM writes, but the number of patches necessary is VASTLY reduced.

The ZPC V2 modifications should be installed in any machine on which the user intends to run 'PC' software. These modifications are described next.

Before proceeding to dismantle your computer, please complete the following:

- Read all documentation with ZPC. Install version 2 (or later) of ZPC and ensure it operates properly.

Note: To make best use of the advances in ZROM and Z-DOS, ZPC has become very version sensitive. The version of ZPC distributed with each version of the ZROM and Z-DOS must have that software already installed to operate properly.

- Ensure the computer is working properly. The Scottie Board installation will NOT fix existing problems with the computer.

MOTHERBOARD MODIFICATIONS

The motherboard modifications lock out the critical addresses which would cause 'hang up' when, by coincidence, the low byte of the 16 bit I/O address happens to write to the H/Z-100's Memory Mapping Port.

The motherboard modification DOES NOT REQUIRE motherboard removal, nor soldering. The modification is completely reversible.

The modification came in a kit with 4 IC sockets prewired using #30 wires. For those individuals wishing to install the Scottie Board, but not having the harness available, a harness can easily be fabricated following the Mod Harness Schematic, Figure 8, and estimating the lengths of wires needed by comparing the socket locations once you have access to the motherboard.

Notes:

- Power must be turned off before making modifications to or installing boards!
- Use extreme care when removing Integrated Circuits (ICs) from their sockets. An IC removal tool is best, though often a miniature screw driver may be safely used. (A damaged IC socket is nearly impossible to repair because of the multilayered circuit boards being used.)
- Be careful not to mix up the ICs.

[ ] Gain access to your motherboard using the disassembly procedures provided on the "*Z-100 LifeLine*" Website (Remove top cover, disk drive /monitor yoke, middle cover, keyboard, and video board).

[ ] Assemble a socket/wiring harness using new low profile IC sockets and #30 wire as shown on the Mod Harness Schematic, Figure 8. The lengths of wire needed can be measured off the motherboard.

[ ] Locate and remove IC U169. Insert the new socket U169 into the existing socket, then reinstall IC U169 into the new socket.

[ ] Locate and remove IC U171. Insert the new socket U171 into the existing socket, then reinstall IC U171 into the new socket.

[ ] Locate and remove IC U146. Insert the new socket U146 into the existing socket, then reinstall IC U146 into the new socket.

[ ] Locate and remove IC U179. Insert the new socket U179 into the existing socket, then reinstall IC U179 into the new socket.

[ ] Dress down the new wires so that they lie between the other ICs on the motherboard. Do NOT attempt to stretch the wires.

The motherboard modification is complete.

VIDEO BOARD MODIFICATIONS

The video board modifications lock out IBM-PC 16 bit I/O addressing from corrupting programming of the Z-100's video controller.

The video board modification DOES NOT REQUIRE SOLDERING, but it DOES REQUIRE REMOVAL, if only to provide access to the motherboard. The modification is completely reversible.

The modification came in a kit with 2 IC sockets prewired using #30 wires. For those individuals wishing to install the Scottie Board, but not having the harness available, a harness can easily be fabricated following the Mod Harness Schematic, Figure 8, and estimating the lengths of wires needed by comparing the socket locations once you have access to the video board.

[ ] Assemble a socket/wiring harness using new low profile IC sockets and #30 wire as shown on the Mod Harness Schematic, Figure 8. The lengths of wire needed can be measured off the video board.

[ ] Locate and remove IC U369. Insert the new socket U369 into the existing socket, then reinstall IC U369 into the new socket.

[ ] Locate and remove IC U357. Insert the new socket U357 into the existing socket, then reinstall IC U357 into the new socket.
~~~~~~

[ ] Dress down the new wires so that they lie between the other ICs on the video board. Do NOT attempt to stretch the wires.

The video board modification is complete.

BATTERY INSTALLATION INSTRUCTIONS

Make sure there are no fingerprints, oil or salt residue from your skin on the top and bottom battery surfaces.

To improve the electrical connection, a new Lithium Battery may need some new deep scratches as you see on the battery originally supplied.

When the battery is handled, please grasp by the edges only. Raise the battery holder tang only the MINIMUM necessary to insert the Lithium battery.

Align the battery so that a scratch on the battery slides under, and is clamped by the battery holder tang. This ensures new metal to metal contact, without fingerprints, between the battery and battery holder contact surfaces.

If your Scottie Clock fades, or looses time, then in all probability the battery and holder are not in reliable contact with each other. To fix this, use a jeweler's screw driver, or perhaps an awl or sewing needle to rotate the battery a few degrees in the holder, causing one of the scratches on the battery and the tang to 'scrape' a new metal to metal contact.

SCOTTIE BOARD INSTALLATION

The Scottie Board is installed into any of the spare S-100 card slots in the card cage at the rear of the computer. Hold the Scottie Board with the ICs facing the front of the computer and the gold edge fingers toward the bottom. Start the PC board into the card guides and ease it down into the connector. When you feel contact with the connector, make sure that it is centered, then push down until it snaps in.

If you have one or both COM cable options, install each 'inside cable', securing the DB25 connector to the rear panel of the computer in place of a plastic knock-out.

Reassemble the computer by reversing the disassembly procedures, except for the top cover. This will be replaced after the testing is complete. Power up the computer as normal.

Copy CLOCK.COM and SETCLK.COM from the software disk to your system boot disk. An example of an AUTOEXEC.BAT file is provided in the next section, TESTING AND OPERATION.

Once the AUTOEXEC.BAT file has been prepared, reset your computer and boot up. The system time and date should have been set by the Scottie Clock and the new command line prompt should show the system date and time, and path to your root directory.

To set your Scottie Clock, just run the normal MS-DOS TIME and DATE commands as stated in the MS-DOS manual, then type:

SETCLK{ret}

You will get a message:

"System Clock Set from Scottie Clock."
and a <Beep>.

TESTING AND OPERATION

Your Scottie Board COM 1 and COM 2 Vectored Interrupt SHORTING BLOCK selection was factory set to look like this:

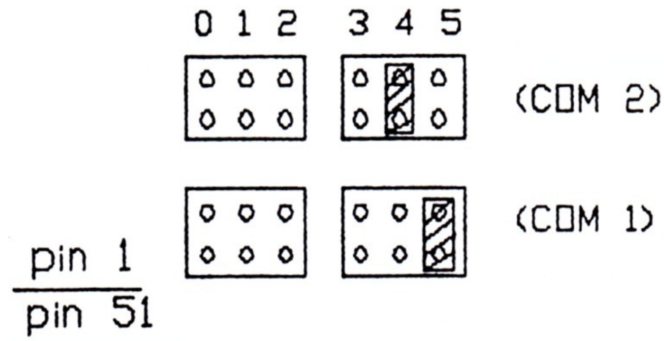


Figure 1.
Scottie Board COM Shorting Blocks

After installing the mods and the Scottie Board, run ZHSTEST. This program is provided on the clock support disk or with the later versions of ZPC. ZHSTEST will check out both the Modification's ability to lock out 16 bit I/O addressing from corrupting the H/Z-100 programming and the Scottie Board's ability to generate vector interrupts VI6 and VI7.

In your AUTOEXEC.BAT file, install the INT14 driver. If you have COM1 only, then install as **INT14 1{ret}**. If you have COM1 and COM2, AND YOU DO NOT WISH TO DIRECT OUTPUT TO AUX (using a native port) instead of COM2, then install as **INT14 2{ret}**.

These changes can be modified WITHOUT rebooting. Your AUTOEXEC.BAT file should look something like this, but with YOUR disk complement and personal preferences.

To view your AUTOEXEC.BAT file, type at the system prompt:

TYPE AUTOEXEC.BAT{ret}

AUTOEXEC.BAT should be similar to:

```
ECHO=OFF
PATH=E:\;E:\DOS;E:\ZPC
ZPC
ANSISYS
INT14 2
CLS
PROMPT $d $t$_$p$g
CLOCK
```

Note: Users of version 3.0 and later of the Z-100's Monitor ROM need not include the INT14 Driver and ANSISYS statements in the batch file provided here. Both had been incorporated when ZPC was rewritten to take full advantage of the new ZROM.

Note: An AUTOEXEC.BAT file may be prepared with any word processor or editor program which will allow you to generate or edit an ASCII text file. You can also use the MS-DOS COPY CON command:

COPY CON AUTOEXEC.BAT{ret}

Note: Placing all the ZPC files in a ZPC directory and including C:\ZPC in AUTOEXEC.BAT's PATH statement keeps all the ZPC routines together and they can be run from anywhere in the directory hierarchy.

Reboot the computer.

The motherboard modification will inhibit erroneous writes to the Memory Mapping hardware in the Z-100, the Scottie Board 8250A COM Ports will answer up, and the INT14 service routines will handle the interrupts just as in the IBM-PC. Most Software, such as PROCOMM or IBM-PC Kermit, which use COM1 or COM2 will not know the difference.

The Scottie Clock simply uses IBM-PC Clock addressing criteria, for which the Z-100 modifications lock out conflicts.

ZHSTEST will also determine the speed of your interrupt controllers (the 8259As). Located immediately to the left of U14, there is a 4 x 2 x .1" header with three 1N4148 diodes below it. This circuitry was added to allow the user to inject one wait state for the VI6 and VI7 interrupts generated by the board. This will allow your machine to run at its maximum rate.

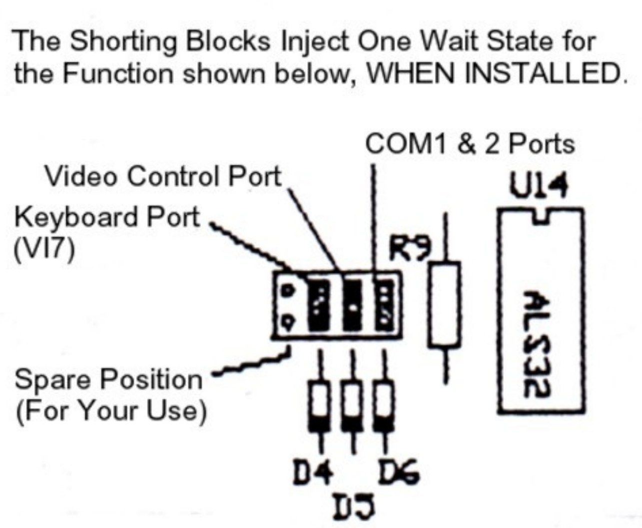


Figure 2.
Wait State Shorting Blocks

Start out with the three shorting blocks **removed** from the header. Run ZHSTEST. If it finds any of the ports slow, it reports one or more of the following messages:

"The video control port does not work."
"The video retrace port does not work."
"The keyboard is slow."

If any of these messages are given, check the installation of the board, reboot, and run the test again. If the message(s) still appear, try installing the appropriate shorting block(s) and run the program again. If it still reports any of the blocks slow, you will need to install a W2 shorting block on your motherboard or upgrade the 8259A controllers to 8259A-02, and replace some of the support ICs with faster chips.

The W2 header, marked J106 on the motherboard, is near U233. (It can be found 3" aft of the right front corner.) Located under the keyboard, this will involve disassembly of the computer again.

If the J106 header is missing, then you have the OLD motherboard and will need to install the wait state modification per an article in REMark, July 1985, page 27.

Once the test is satisfactory, the following messages should appear:

"The video control is ok."
"The video retrace is ok."
"The keyboard is ok."

This completes the testing and initial operation. Reassemble the computer in the reverse order of disassembly.

As with the other PC-emulator systems, not all IBM-PC programs will run. There are a few helpful tips, however:

- After running an IBM-PC program, and before running the next, always RETURN TO THE Z-100 MODE. This will clear any settings left by the last PC program. Your version of ZPC may contain Z100.COM or simply Z.COM to return to the native mode.

- The easiest means of running any PC program requiring ZPC is to use a batch file. For example, PRO.BAT located in the root directory, would launch PROCOMM:

```
CD E:\PC\PROCOM
PC          (or ZPC, if it is not
PROCOMM      already running)
Z100
CD E:\
```

Thus to run a PC program, you need type only one small command, in this case PROCOM, to run the program under ZPC! And, best of all, PROCOM can be run from anywhere.

SCOTTIE BOARD BREAKOUT SWITCH

Mike Zinkow of Parts is Parts developed a Z-100 hardware Breakout Switch circuit that can be added to the breadboard area of the Scottie Board. The Breakout Switch permits "breaking out" or pausing any program being run on the Z-100 so that an analysis of the state of the flags and registers can be made to show what the program is doing (or not doing). For example:

Suppose we had a new software routine that kept producing an odd result. Pressing a switch on the back panel causes the program to pause immediately and the Z-100 to return to the hand prompt.

Once at the hand prompt, the version 3.x and 4.x monitor ROMs provide a wide range of functions similar to DOS's DEBUG program to analyze the state of the flags and registers set by our new program.

Once we are ready to resume, pressing {**G**} for 'Go On' at the hand prompt returns us to our program at the exact point where we departed to resume operation.

This was a very critical element in aiding the development of the Z-100's version 4 Monitor ROM and MS-DOS changes. And recently I had to install the breakout switch to ease trouble-shooting of the ailing CDR-317 hard drive controller card, described in issue #74.

The installation is relatively easy, but does require experience in soldering components to a circuit board. The following instructions should provide sufficient detail to ease the pain.

PARTS LIST

1		14-pin low profile IC socket for U19 - may not be needed. (Already present on some boards)
1	U19	74ALS05 IC
1	R14	10 Ohm resistor (Brn-Blk-Blk)
1	R15	1000 Ohm resistor (Brn-Blk-Red) or (Brn-Blk-Blk-Brn)
1	R16	470 Ohm resistor (Yel-Vio-Blk-Blk)
1	R17	6810 Ohm resistor (Blu-Gry-Brn-Brn)
3	R18-20	10000 (10k) resistor (Brn-Blk-Org) or (Brn-Blk-Blk-Red)
1	C25	100uF, 25VDC Radial lead electrolytic Capacitor
1		2x1x.1" 2-pin right-angle header
1		.100" 2-terminal housing
2		Crimp terminals for housing/header
1	J18	3.5mm Mono chassis mount enclosed jack with mounting hardware (optional)
1		6 foot 2-conductor shielded cable w/1 molded 3.5mm mono plug (optional)
1	S5	Red single pole (NO) push button switch with mounting hardware
1		.025" square straight pin (to be used to ream the mounting holes for the 2x1x.1" header
1		Length of small heat shrink tubing
1		Length 30awg Kynar Red wire or similar
1		Length 30awg Kynar Black wire or similar
1		Length 30awg Kynar Blue wire or similar
1		Length of 2-conductor ribbon cable

INSTALLATION

Notes:
- Please read through the entire procedure before proceeding. If you have any questions, please email me BEFORE proceeding.
- Though this installation requires simple soldering procedures, there is the potential of damaging your Scottie Board. Please proceed with caution.
- Turn off and unplug the Z-100 from power.

[ ] Open your Z-100 and remove the Scottie Board.

[ ] Study the installation diagrams for proper parts layout and row and column designations.

Note: This assumes that no breadboard projects have already been built in the breadboard areas. If an area is already used, adjust your installation accordingly.

Notes:
- Rows and Columns will be referenced as **Rowx,Colx**, where x is the row and column numbers (i.e., Row12,Col14 is row 12, column 14 as shown in the Non-Maskable Interrupt (NMI) Figures.
- (**NS**) means do not solder the connection.
- (**S**) means to solder the connection.
- (**Sx**) means x connections (wires) to solder.
- Mount all parts on the component side of the board, like those already installed.

1. [ ] The holes in the breadboard areas were not designed to accept the oversized leads of some headers. Therefore, it may be necessary to slightly ream out the 2 holes for the 2 terminal right-angle header. Using a straight pin and a pair of pliers, ream out the 2 holes at the bottom right corner (Row 25,Col39 & Row25,Col40) of the breadboard area below U19, as shown on the installation diagram, NMI Figure 1, Inset A.

2. [ ] U19. Install the 14-pin IC socket observing correct orientation as shown on the circuit board. Solder ONLY pins 7 and 14 at this time. This socket is already installed on some boards.

3. [ ] Locate the Plated Thru Hole (PTH) in the +5v buss between the prototype area and U3 pin 5. With a sharp instrument, such as the pin or a dental pick, scrape away the solder resist around the PTH on both sides of the board.

4. [ ] R15. Hold a 1000 Ohm (brn-blk-red) resistor with long-nose pliers close to the resistor body and bend the leads straight down to fit the hole spacing on the circuit board. Install one leg of R15 into the hole at Row4, Col7 (NS) and the other lead into Row8,Col7 (NS) in the breadboard area adjacent to U3. See the NMI Figures.

5. [ ] R18. In a similar manner, bend the leads of a 10k Ohm (brn-blk-org) resistor and install in Row4,Col8 (NS) & Row8,Col8 (NS).

Resistors R19 and R20, below, are not used for the breakout function and can be omitted. They are installed for use by other features.

6. [ ] R19. In a similar manner, bend the leads of a 10k Ohm (brn-blk-org) resistor and install in Row4,Col6 (NS) & Row8,Col6 (S).

7. [ ] R20. In a similar manner, bend the leads of a 10k Ohm (brn-blk-org) resistor and install in Row4,Col5 (NS) & Row8,Col5 (S).

8. [ ] Prepare a piece of 30awg Red wire 1-1/2" long and strip 3/16" from one end and 1" from the other end. Bend the 3/16" end and insert it into the PTH near U3 pin 5 and solder the connection.

9. [ ] Loop the 1" end around R18 (S3) at Row4,Col8 as close to the board as possible. Solder the connection and cut off the excess RESISTOR lead length.

10. [ ] Continuing with the 1" end, loop the end around R15 (S3) at Row4,Col7 as close to the board as possible. Solder the connection and cut off the excess RESISTOR lead.

11. [ ] Still continuing with the 1" end, loop the end around R19 (S3) at Row4,Col6 as close to the board as possible. Solder the connection and cut off both excess RESISTOR lead lengths of R19, leaving the unused lead about 1/8" long for future use.

12. [ ] Still continuing with the 1" end, loop the end around R20 (S2) at Row4,Col5 as close to the board as possible. Cut off both excess lead lengths of R20, leaving the unused lead about 1/8" long for future use.

13. [ ] 2x1 Right-angle Header. Install the right-angle header (NS) into the 2 reamed out holes at Row 25,Col39 and Row25,Col40. Do not solder the connections yet.

14. [ ] R14. Form the leads of a 10 Ohm resistor as described in step #4. Insert R14, as shown on the installation diagram, NMI Figure 1, into the holes in Row26,Col35 (NS) and Row26,Col39 (NS).

15. [ ] Refer to NMI Figure 2. On the back side of the board, bend the lead of R14 (Row26,Col39) (S) toward the right-angle header tightly to the board and hook the lead around the right-angle header (Row25,Col26) (S2). Solder the lead to the right-angle header and trim off the excess lead.

16. [ ] C25. Observing polarity, as indicated on NMI Figure 1, insert the positive lead of a 100uF, 25v electrolytic capacitor into Row24, Col36 (NS) and the negative lead into Row24, Col37 (NS).

17. [ ] Prepare a piece of 30awg black wire 3/4" long. Strip 1/8" of insulation from the ends of the wire and form a hook into each stripped end. Refer to NMI Figure 2 and hook one end around the unsoldered right-angle header terminal (Row25,Col40) (S). Carefully solder the connection.

18. [ ] Hook the free end of the black wire around the negative lead of C25 (Row24,Col37) (NS).

19. [ ] Locate the Plated Thru Hole in the Ground buss below U19 pin 7, and as described in step #3, remove the solder resist on both sides of the board.

20. [ ] Prepare a piece of 30awg black wire 1-1/2" long. Strip 3/16" of insulation from each end of the wire. Form a hook into one of the ends and hook the end around the negative lead of C25 (Row24,Col37) (S3). Carefully solder and trim off the excess capacitor lead.

21. [ ] Insert the free end of the black wire into the hole in the Ground buss below U19 and carefully solder the connection.

22. [ ] R16. Form the leads of a 470 Ohm (yel-vio-blk-blk) resistor as described in step #4. Insert R16, as shown in NMI Figure 1, into the holes in Row21,Col35 (NS) and Row25,Col35 (NS).

23. [ ] R17. Form the leads of a 6810 Ohm (blu-gry-brn-brn) resistor as described in step #4. Insert R17, as shown in NMI Figure 1, into the holes in Row20,Col34 (NS) and Row20,Col38 (NS).

24. [ ] Prepare a piece of 30awg blue wire 6-1/2" long. Strip 1/2" of insulation from one end and 3/16" of insulation from the other end of the wire. Form a hook into the 3/16" stripped end. Refer to Figure 2 and hook the end around R15 (Row8,Col7) (S2). Carefully solder and trim off the excess RESISTOR lead.

25. [ ] Loop the free end of the blue wire, with 1/2" stripped, around R14 (Row26,Col35) (S3), R16 (Row25,Col35) (S3), and hook the end of the wire around the positive lead of C25 (Row24,Col36) (S2). Carefully solder the connections. Trim off all the excess leads.

26. [ ] Prepare a piece of 30awg blue wire 6-1/2" long. Strip 1/8" of insulation from one end and 3/8" of insulation from the other end of the wire. Form a hook into the 1/8" stripped end. Locate U13 pin 3 and hook the end around U13 pin #3 (S2). Carefully solder the connection.

27. [ ] Loop the free end of the blue wire, with 3/8" stripped, around R16 (Row21,Col35) (S3), then hook the end of the wire around R17 (Row20,Col34) (S2). Carefully solder the connections and trim off all excess leads.

28. [ ] Prepare a piece of 30awg blue wire 2-3/4" long. Strip 3/16" of insulation from each end. Turn the board component side up. Refer to the installation diagram. Locate S-100 terminal #12. Insert a scrap piece of wire through the hole. Turn the board over and insert one end of the prepared wire into the identified hole (S), discarding the scrap wire. Carefully solder the connection.

29. [ ] Form a hook in the free end of the blue wire and loop it around R17 (Row20, Col38) (NS).

30. [ ] Prepare a piece of 30awg blue wire 3-1/2" long. Strip 1/8" of insulation from one end and 3/16" of insulation from the other end. Form a hook into the 1/8" stripped end and loop it around U19 pin #2 (S3) and carefully solder the connection.

31. [ ] Form a hook in the free end of the blue wire and hook it around R18 (Row8,Col8) as close to the board as possible. Solder the connection and cut off excess lead lengths.

32. [ ] Prepare a piece of 30awg blue wire 4" long. Strip 1/8" of insulation from each end and form a loop in each end. Locate U13 pin 4 and hook one end of the wire around U13 pin #4 (S2). Carefully solder the connection.

33. [ ] Locate U19 pin 1 and hook the free end of the blue wire around U19 pin 1 (S2). Carefully solder the connection.

34. [ ] If not already done, solder the remaining U19 socket pins. (Pins #3, 4, 5, 6, 8, 9, 10, 11, 12, and 13.)

CAUTION: Please follow standard IC handling precautions when installing the IC in the next step.

35. [ ] U19. Install the 74ALS05 integrated circuit chip into IC socket U19, observing the pin 1 orientation.

36. [ ] Check the soldering for poor solder joints, unsoldered connections, solder bridges, and untrimmed leads.

37. [ ] To secure the wires to the PC board, either tape them to the board, or use a few dabs of silicon caulk, hot glue, or specially designed PC board glue.

38. [ ] Remove any flux residue with either an aerosol flux remover, following the manufacturer's instructions, or with rubbing alcohol and a small stiff brush, such as an old toothbrush.

INTERCONNECTING CABLE

Note: The Breakout Switch can be located anywhere in the computer chassis. Alternatively, it can be placed at the end of a cable plugged into a jack at the rear of the computer. This would allow you to mount an exterior switch at any convenient location.

WARNING: Do NOT drill the back panel while it is attached to the computer. The metal shavings would wreak havoc!

The following describes placing the switch in one of the plastic blank connector covers. This permits easy removal and, more importantly, precludes having to remove the computer's back panel.

39. [ ] Refer to Figure A on page 5. Prepare a 2-conductor ribbon cable approximately 15" long. (Longer length is ok.) Separate the cable 1/2" on one end (End A) and 1-1/2" on the other end. Strip 1/8" from each conductor on the 1/2" end (End A) and 1/4" from each conductor on the other end. Twist the strands together and tin all ends with solder.

40. [ ] Refer to Figure B on page 5. Mechanically fasten and carefully solder the 2 connector pins to the ends of the 1/2" separated End A of the cable. (Be careful not to allow solder to get under the locking tab.)

41. [ ] Refer again to Figure B and with the terminal housing's slots up, insert the connector pins into the 2 terminal housing. Push the pins in until you hear a faint click, locking the connector pin into the housing. Gently tug at the wires to be sure the connector locked into the housing. Put the cable assembly aside for now.

BACK PANEL

42. [ ] Remove one of the plastic blank connector covers from the back panel. One close to the top, right side may be more accessible.

43. [ ] Drill a hole (about 1/4"?) in the blank to fit either the external jack or the breakout switch. Mount the switch or jack in the hole.

44. [ ] Form a hook in each of the tinned ends of the free end of the 2 conductor ribbon cable assembly and hook each end around each switch terminal (either conductor to any terminal). Solder the connections.

45. [ ] Slipping the connector end of the switch cable through the open connector hole in the back panel, remount the plastic blank (with the switch installed) in the back panel.

46. [ ] Holding the Scottie Board just above the card cage, install the Breakout Switch connector on the new header of the Scottie Board.

47. [ ] Lower the Scottie Board, with the component side facing forward, into the card cage and press into its S-100 bus connector.

48. [ ] Reconnect any COM1/COM2 cables, if they are installed.

49. [ ] Make a final, careful inspection of the computer and replace the cover.

50. [ ] If you are using an external breakout switch, assemble the external cable with the switch at one end and a cable plug at the other.

TESTING

51. [ ] Turn the Z-100 on and after the boot completes, press the Breakout Switch. A register display should appear.

52. [ ] Pressing the {HELP} key should give a list of MTR-100 ROM commands.

53. [ ] If you have a MTR-100 ROM version 3.x or 4.x, press the {**G**} key. '*Go On*' should appear. Press the {**RETURN**} key. The computer should return to the MS-DOS prompt, ready to accept commands.

You may press the Breakout Switch and return to the MTR-100 ROM hand prompt at anytime. For example, to turn off the key click - press the Breakout Switch, the {**k**} key, the {**G**} key, and finally the {**RETURN**} key to finish.

SCOTTIE BOARD CLOCK

Using the schematic provided last time, I also installed the clock option on my board. There is one omission on the board's silk screen and hole layout, however, that is a "gotcha".

The schematic shows a crystal, X1, but the board does not include a location to mount it! This makes it easy to miss.

When mounting R11 and C12, leave the components about 1/8th inch off the board and hook/solder the crystal's leads to the appropriate leads of R11 and C12. Hot glue the crystal to the board after testing.

For convenience, here is a list of the parts needed to install the clock option:

1	X1	32.768 Khz Micropower Crystal
1	U25	MM58167A Clock chip
1	U25	24-pin, low profile IC socket
1	U27	74ALS520 integrated circuit
1	D1	1N4148 general purpose diode
1	D2	1N4148 general purpose diode
1	D3	1N4148 general purpose diode -optional
1	R11	200k Ohm resistor (Red-Blk-Yel)
1	R12	20k Ohm resistor (Red-Blk-Org)
3	R13	100k Ohm resistor (Brn-Blk-Yel)
1	C6	10uF, 25Vdc capacitor -optional
1	C7	0.1uF mono capacitor
1	C8	1.0uF capacitor
1	C11	20pF capacitor
1	C12	20pF capacitor
1		3 Volt Lithium BR2325 or Equiv Battery
	or	1.0 Farad, 5.5V capacitor

Believe it or not (I thought it was a misprint), a 1 Farad capacitor does exist. I do not know how easy they are to locate, but I found one in a junk 8088 laptop (a spare for my Z-181).

I also found that even a 2200uF capacitor will hold a charge for a couple of days, if you can not find the 1.0 Farad job.

An excellent, reasonably priced, source of obsolete integrated circuits is Unicorn Electronics, Inc., 1142 State Route 18, Aliquippa, PA 15001, (724) 495-1230 or (800) 824-3432. They may have the crystal and capacitor, too.

Final notes:

- Both modifications worked flawlessly and are well thought out - fine additions to the Scottie Board. The clock freed up one of my Smartwatch clocks for another computer.

- The Scottie Board software is included with the MS-DOS version 4 software sent with the version 4 MTR-ROM. If necessary, I can send a copy of the necessary software on a 5-1/4" floppy.

- There are also procedures for installing a Scottie Alarm Clock, Heartbeat, Standby Interrupt, and Decoded Port Address 20h/21h modifications. I thought that I had also heard of work for a bi-directional parallel port some time ago.

If you have any of these, or any other adaptations to the Scottie Board that you would like to share, I'd be happy to publish those modifications also.

Now, if you are interested in speeding up your Z-100, with a Scottie Board installed, the following are articles from Mike Zinkow that I published in issue #41.

SCOTTIE Board 8MHz & Faster Operation Modification

from Mike Zinkow, Parts Is Parts, 7-20-95

Parts List: The following resistors may be either 1/8 Watt or 1/4 Watt size:

R21 - 10KΩ (Brn-Blk-Org or Brn-Blk-Blk-Red)
R25 - 2200Ω (Red-Red-Red or Red-Red-Blk-Brn)

Installation Instructions: PLEASE read all the instructions before proceeding with this modification. Also, please read the sheet about Safety and good work practices that came with the SCOTTIE Board.

Unplug the Z-100 from its power source. Gain access to the Z-100's card cage and remove the SCOTTIE Board.

(NS) means do not solder the connection. (S) or (Sx) means solder the connection, x indicates the number of connections at the point to be soldered.

The Scottie Board Modification:

[ ] 1. Locate the lower Plated Thru Hole (PTH) in the vertical +5V bus trace between the upper left proto area and U3. With a sharp instrument, such as a dental pick, scrape away the solder resist around the PTH on both sides of the board.

[ ] 2. Locate the PTH in the trace between the 2 upper left proto areas below the silkscreen printing of resistor id `R9'. Scrape away the solder resist as described in step #1.

[ ] 3. Locate the PTH in the end of the trace from U15 pin 5, located between U14 pin 14 and U15 pin 1. Scrape away the solder resist as described in step #1.

[ ] 4. Locate the PTH at the end of the vertical +5V trace from U14 pin 14. Scrape away the solder resist as described in step #1.

[ ] 5. Install R21, the 10KΩ (Brn-Blk-Org) resistor, vertically into the PTH in the trace between the upper left proto areas and the lower hole in the vertical +5V bus trace. Hold the resistor with long-nose pliers close to the resistor body and bend the lead straight down over the resistor body to fit the hole spacing on the circuit board, so that both leads extend from one end (like a radial leaded component). Insert the unbent lead of R21 into the hole in the horizontal trace below the `R9' designation (S) and the bent lead into the hole in the +5V bus trace (S) and keeping the resistor body vertical and close to the board, solder the connections and trim off excess lead lengths.

[ ] 6. In a like manner, install R25, the 2200Ω (Red- Red-Red) resistor, with the unbent lead into the PTH in the trace from U15 pin 5 (S). Insert the bent lead of R25 into the hole in the +5V trace from U14 pin 14 (S), and keeping the resistor body vertical and close to the board, solder the connections and trim off any excess. Please check for unsoldered/cold soldered connections and shorted connections. Check that all IC pins are properly inserted, and that they are neither bent in nor bent out.

[ ] 7. Remove any flux residue with either an aerosol flux remover, following the manufacturer's instructions, or with rubbing alcohol and a small stiff brush. An old tooth brush or an acid brush with its bristles cut down to 1/4" works fine.

[ ] 8. If you have previously installed the Parts Is Parts NMI modification, or the SCOTTIE Clock Standby Interrupt modification, connect the 4 terminal ribbon cable connector to the Scottie board with the identified conductor to the right.

[ ] 9. Install the Scottie board into the card cage with the component side facing forward.

[ ] 10. If you have either/or the COM1/COM2/PPI options on the Scottie board, connect the COM1 cable to the upper left hand connector with the identified conductor to the left and in a similar manner connect the COM2 cable to the Scottie board upper right hand connector. Connect the PPI cable to the 26-pin header near the S-100 Bus fingers with the identified conductor to left.

[ ] 11. Reassemble your Z-100.

Operational Note: This modification increases the speed of the Open-Collector Inverter by reducing the depth of saturation, allowing faster switching speed but with an increase in device power consumption.

SCOTTIE Board 10MHz & Faster Operation Modification

from Mike Zinkow, Parts-Is-Parts, 7-20-95

Parts List: In addition to the 8MEGMOD modification kit (The article described earlier), the following parts are required. The resistors may be either 1/8 Watt or 1/4 Watt size:

R23 - 2200Ω (Red-Red-Red or Red-Red-Blk-Brn)
R24 - 2200Ω (Red-Red-Red or Red-Red-Blk-Brn)
U16 - 74F74 IC
U17 - 74S05 IC
U28 - 74AHCT244 IC
U29 - 74AHCT244 IC
1-1/2" #30AWG Kynar wire

Installation Instructions: Please read all the instructions before proceeding with this modification. Also, please read the sheet about Safety and good work practices that came with the SCOTTIE Board.

Unplug the Z-100 from its power source. Gain access to the Z-100's card cage and remove the SCOTTIE Board.

(NS) means do not solder the connection. (S) or (Sx) means solder the connection, x indicates the number of connections at the point to be soldered.

Note: The 8MEGMOD modification must also be installed for this modification to function correctly!

The Scottie Board Modification:

[ ] 1. Locate the Plated Thru Hole (PTH) in the trace coming from U17 pin 6 as shown on the component placement diagram. With a sharp instrument, such as a dental pick, scrape away the solder resist around the PTH on both sides of the board.

[ ] 2. Locate the 3 vertical traces below U17. Selecting the center trace, scrape away the solder resist from the PTH as described in step #1.

[ ] 3. Locate the PTH in the vertical +5V trace from U16 pin 14. Scrape away the solder resist from the PTH as described in step #1.

[ ] 4. Install R23, the 2200Ω (Red-Red-Red) resistor, vertically into the PTH in the trace from U17 pin 6 (S). Insert either lead of R23 into the hole and keeping the resistor body vertical and close to the board, solder the connection and trim off excess lead length.

[ ] 5. In a like manner, install R24, the 2200Ω (Red- Red-Red) resistor, into the PTH in the middle trace below U17. Insert either lead of R23 into the hole and, keeping the resistor body vertical and close to the board, solder the connection and trim off any excess.

[ ] 6. Strip 3/16" of insulation from each end of the 1-1/2" piece of 30AWG wire. Insert one end into the hole in the +5V trace above U16 pin 14 (S) and solder the connection.

[ ] 7. Bend the free ends of resistors R23 and R24 toward each other, form hooks in each lead, and hook the leads together. Form a hook in the free end of the wire from step #6 and hook it through the hooked R23/R24 leads (S3). Solder the connection and trim off excess lead lengths.

[ ] 8. Remove any flux residue with either an aerosol flux remover, following the manufacturer's instructions, or with rubbing alcohol and a small stiff brush. An old tooth brush or an acid brush with its bristles cut down to 1/4" works fine.

Note: Please read the IC safe handling instruction sheet that came with the SCOTTIE Board before proceeding.

[ ] 9. Remove U12 (74LS74) and place it into an anti- static foam pad or other anti-static protective storage device.

[ ] 10. Remove U16 (74ALS74) and insert it into U12's empty socket, observing pin #1 orientation. Insert IC # 74F74 into U16's empty socket, observing pin #1 orientation.

[ ] 11. In like manner, remove U17 (74ALS05) and place it in the anti-static storage device. Insert IC # 74S05 into U17's empty socket, observing pin #1 orientation.

[ ] 12. In like manner, remove U28 (74LS244) and place it into the .i4 anti-static storage device. Insert IC # 74AHCT244 into U28's empty socket, observing pin #1 orientation.

[ ] 13. In like manner, remove U29 (74LS244) and place it into the anti-static storage device. Insert IC # 74AHCT244 into U29's empty socket, observing pin #1 orientation.

Please check for unsoldered, cold-soldered, and shorted connections. Check that all IC pins are properly inserted, and that they are neither bent in nor bent out.

[ ] 14. If you have previously installed the Parts Is Parts NMI modification or the SCOTTIE Clock Interrupt Modification, connect the 4-terminal connector to the 4-terminal right-angle header with the identified conductor to the RIGHT.

[ ] 15. Install the Scottie board into the card cage with the component side facing forward.

[ ] 16. If you have either/or the COM1/COM2/PPI options on the Scottie board, connect the COM1 cable to the upper left hand connector with the identified conductor to the left and in a similar manner connect the COM2 cable to the Scottie board upper right hand connector. Connect the PPI cable to the 26-pin header near the S-100 Bus fingers with the identified conductor to left.

[ ] 17. Reassemble your Z-100.

Operational Note: The resistors in this modification increase the speed of the Open-Collector Inverters by reducing the depth of saturation, allowing a faster switching speed, but with an increase in device power consumption. The IC changes are required to reduce the propagation delay, and provide faster data access set-up times.

Note: The 8MEGMOD modification must also be installed for this modification to function correctly!

CLOSING:

I do not have the assembly procedures and parts list for the initial Scottie Board. However, if you wish to construct a Scottie Board from scratch, I have put together a parts list from the schematics by Figure Number, that may help (See Page 20). **You will also need a S-100 Prototype Board.**

I hope this information is helpful...

If you have any questions or comments, please email me at:
z100lifeline@swvagts.com

Cheers,

Steven W. Vagts

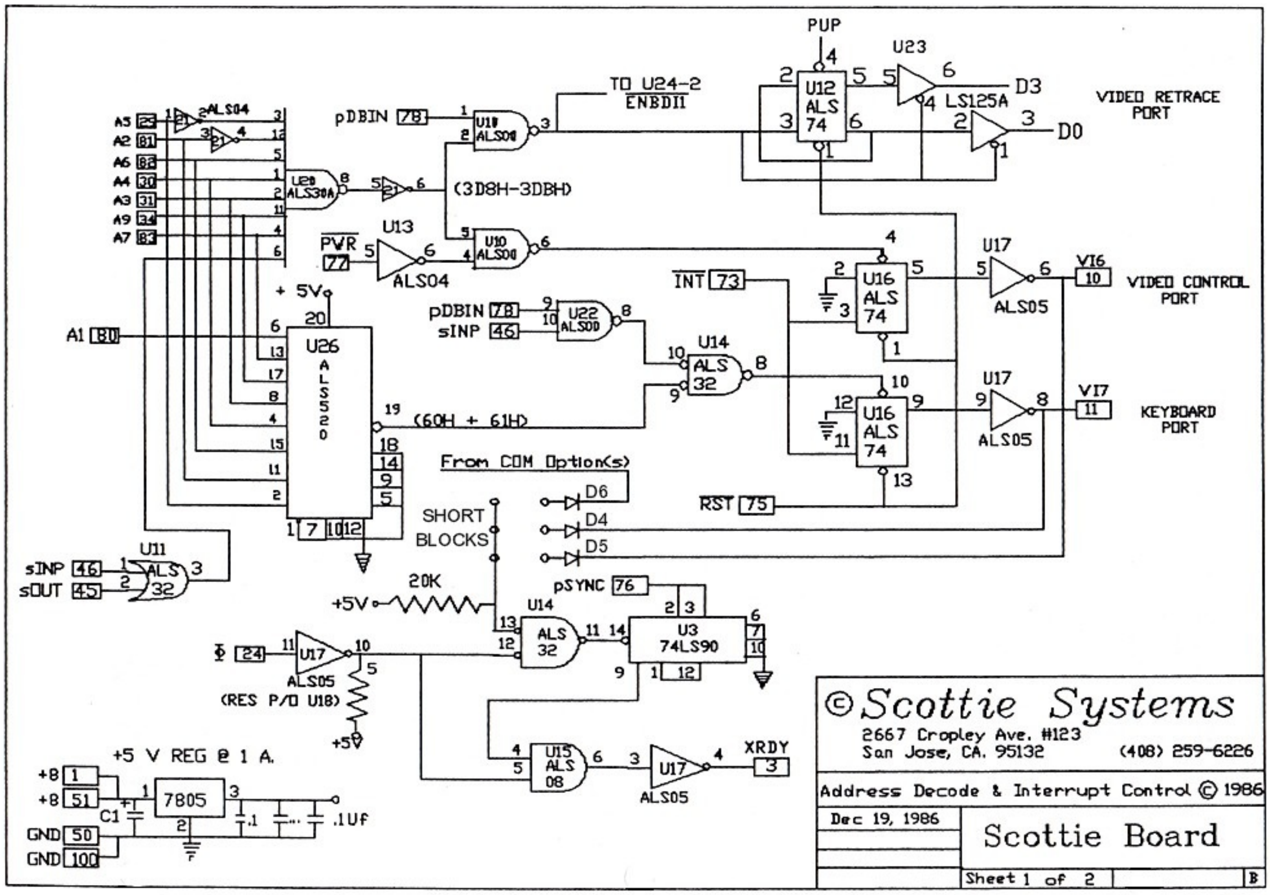


Figure 3 - Scottie Board Address Decode & Interrupt Control

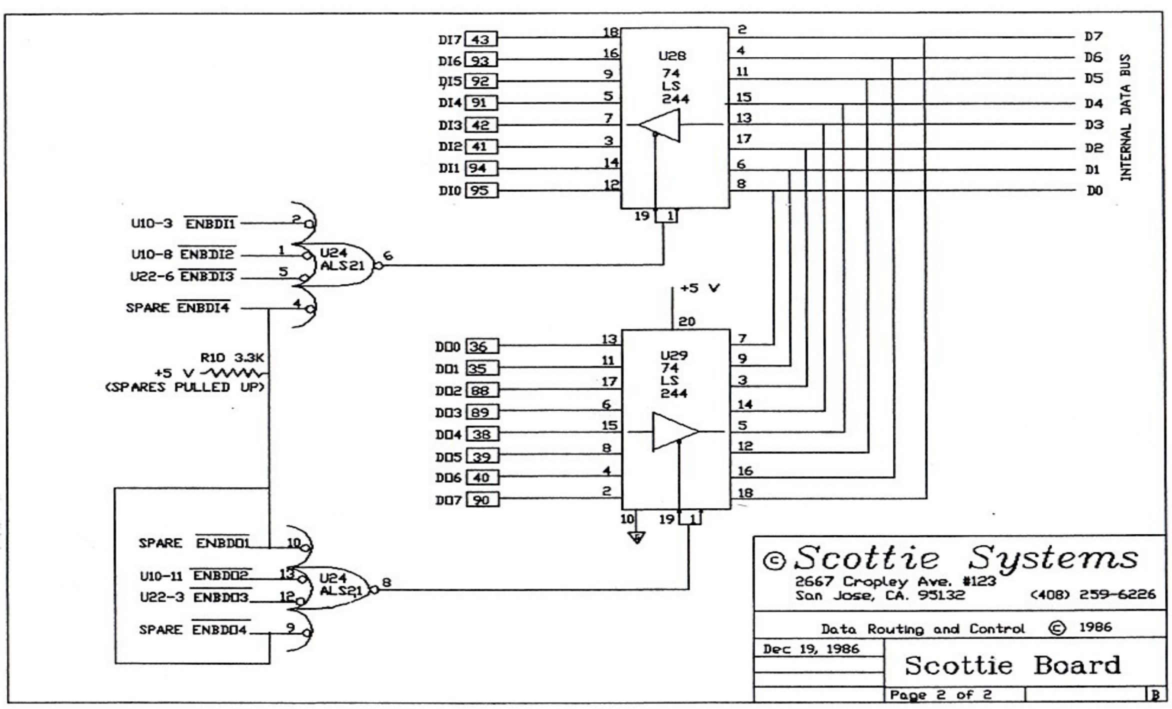


Figure 4 - Scottie Board Data Routing and Control

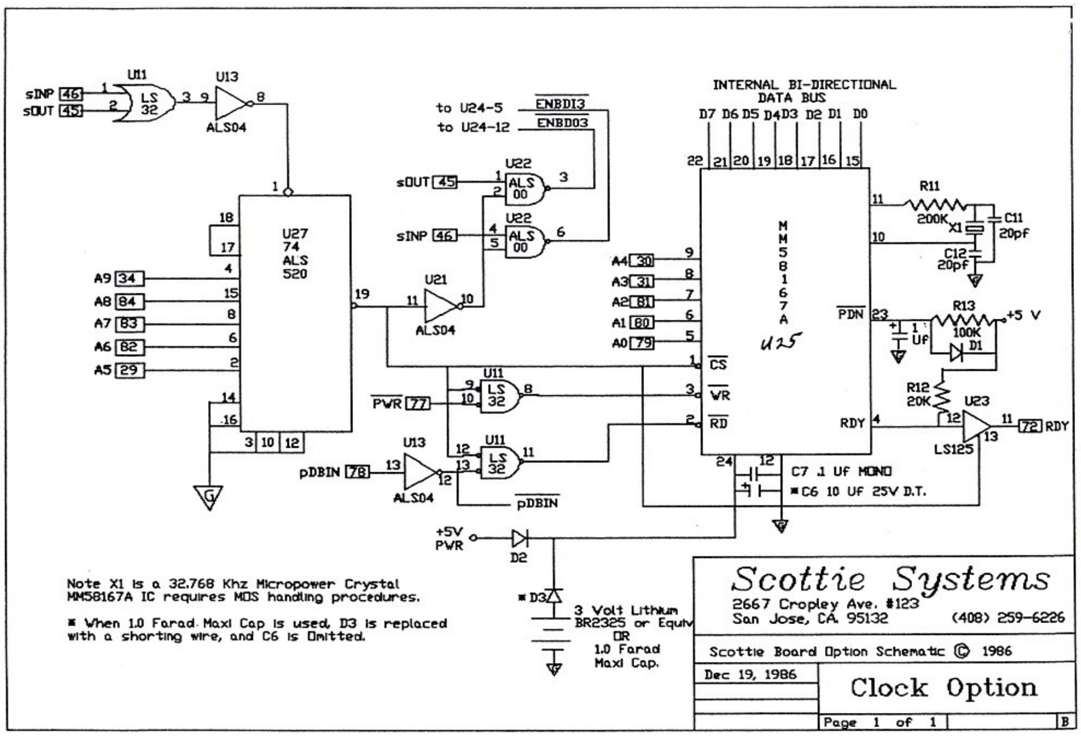


Figure 5 - Scottie Board Clock Option

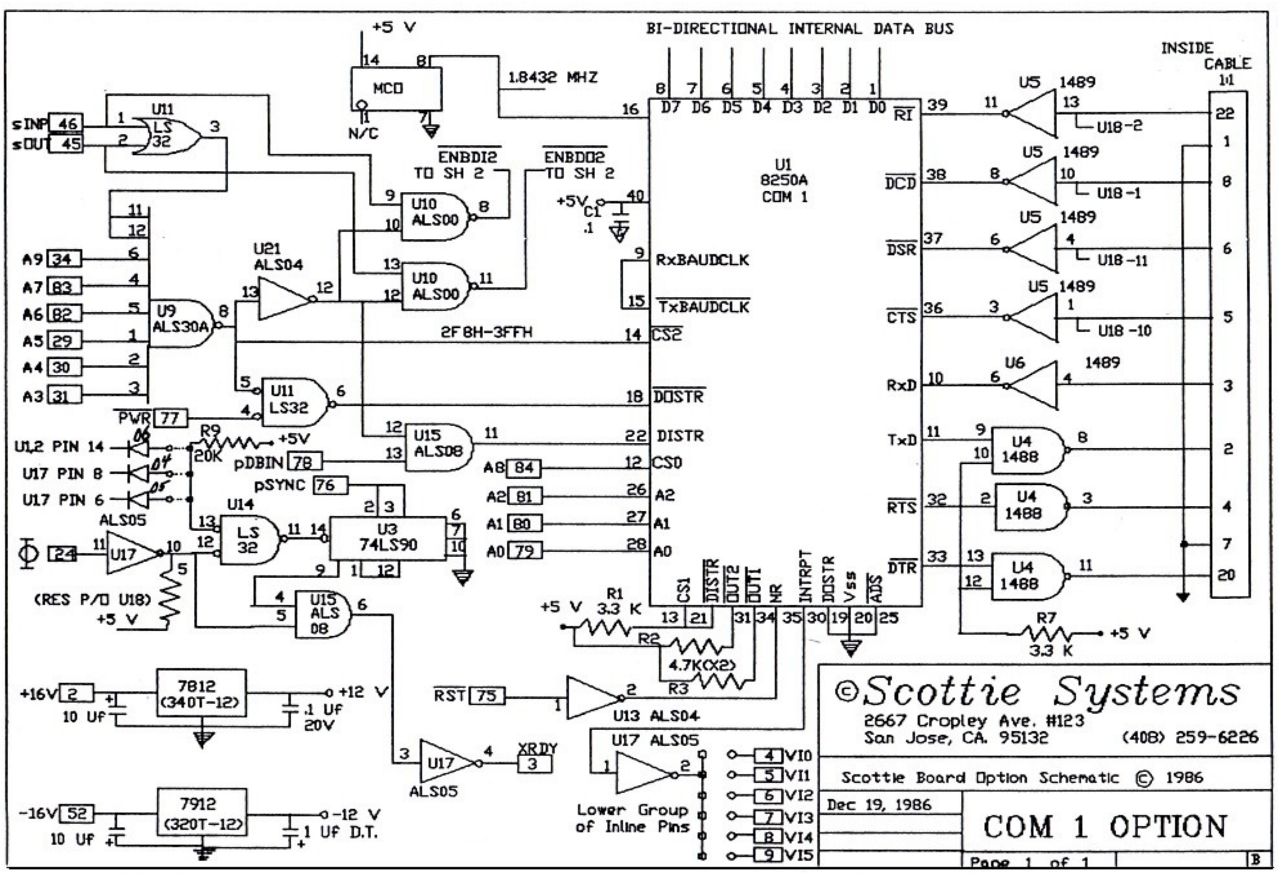


Figure 6 - Scottie Board COM 1 Option

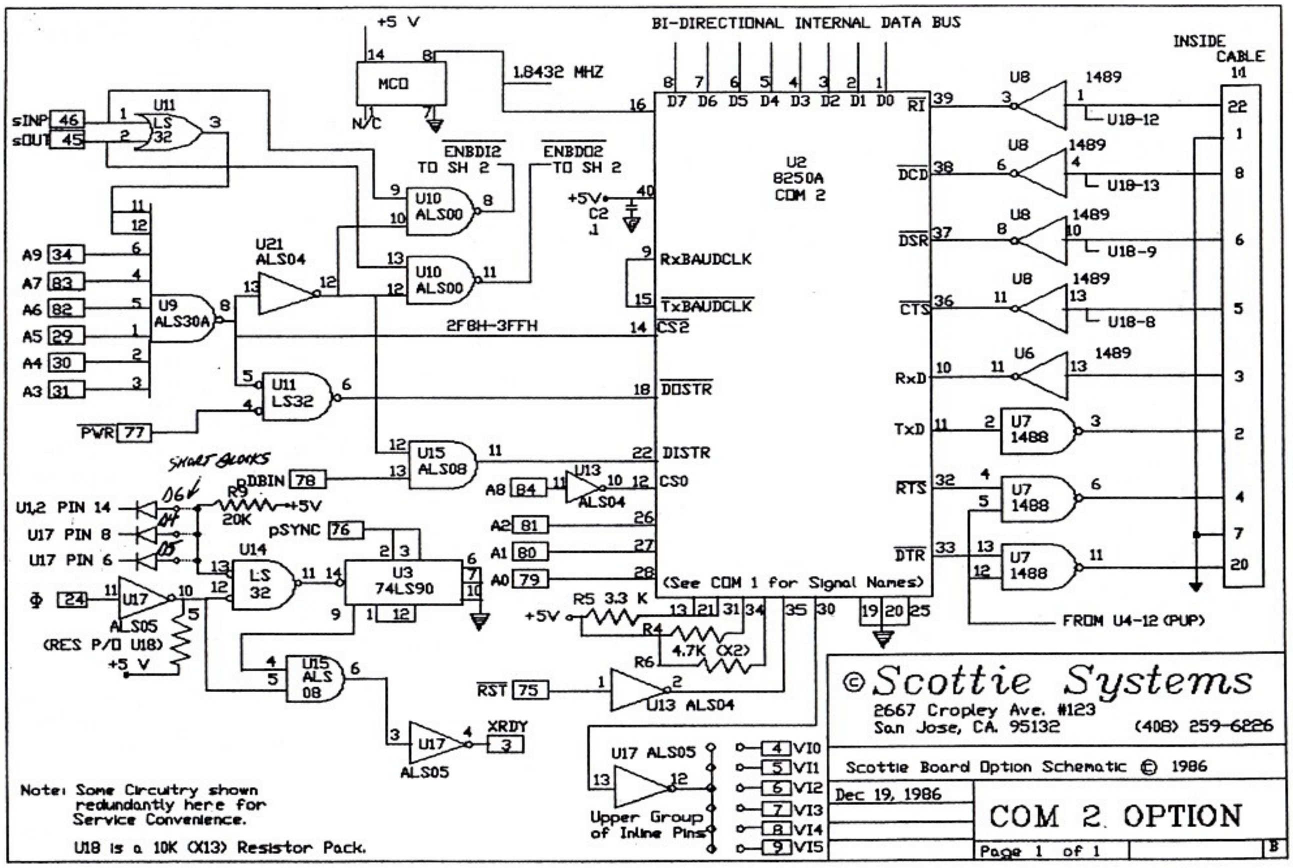


Figure 7 - Scottie Board COM 2 Option

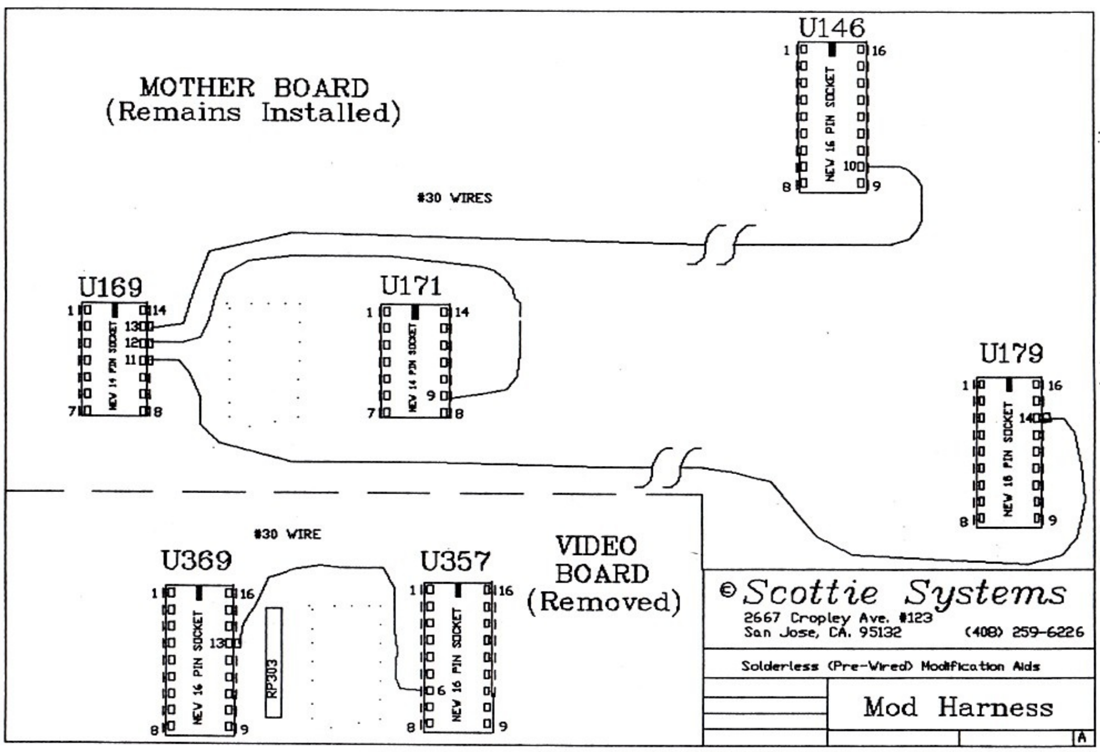


Figure 8 - Scottie Systems Motherboard & Video Board Modification Harnesses

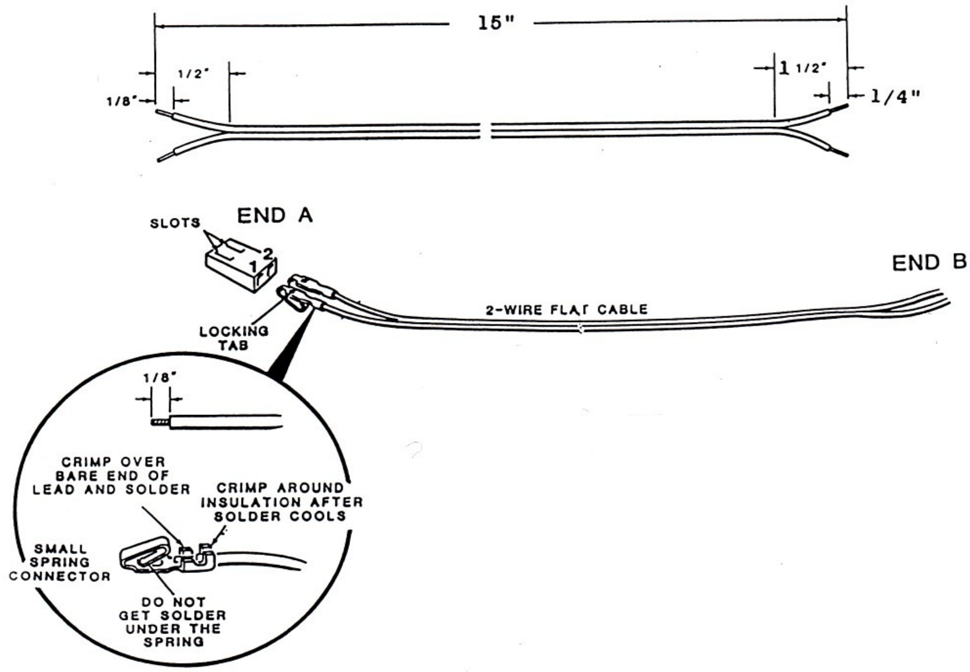


Figure 9 - Scottie Board Breakout Switch Harness

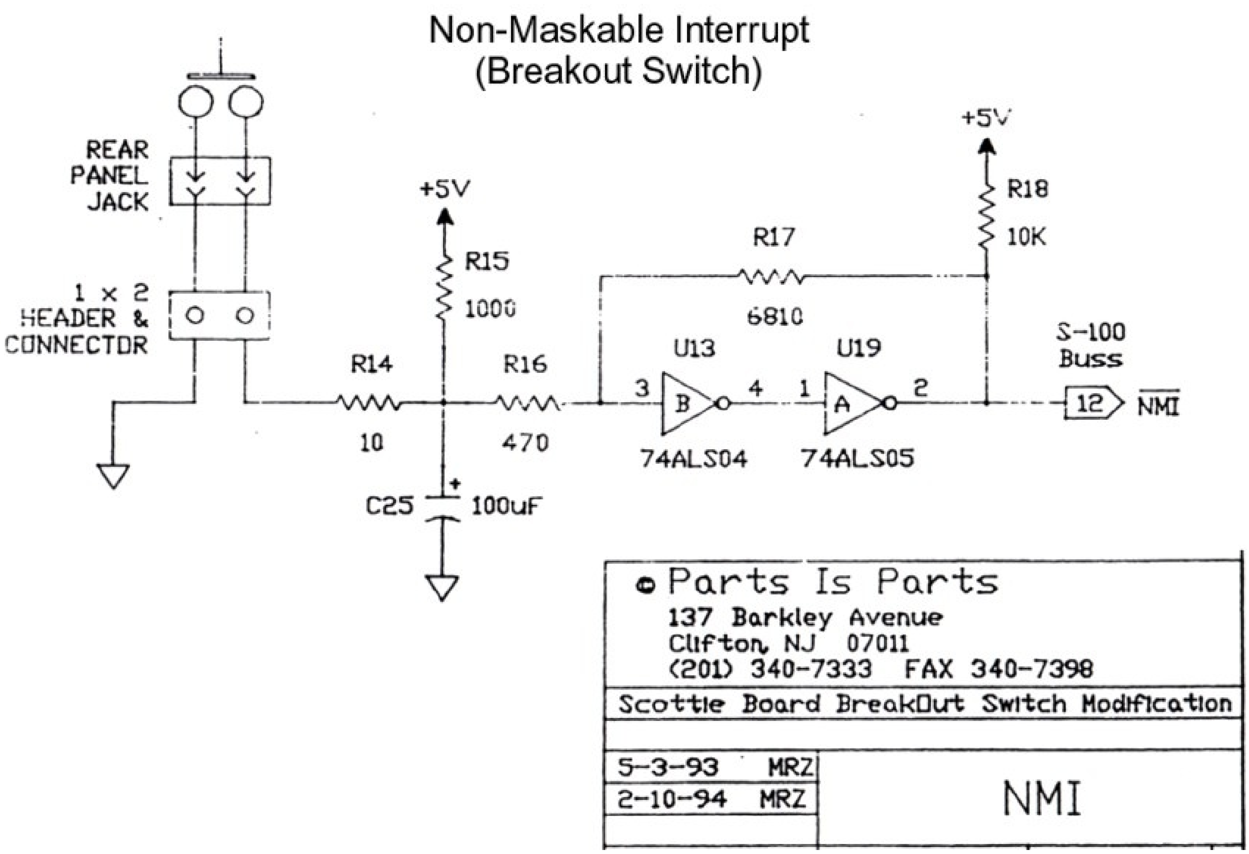


Figure 10 - Scottie Board Breakout Switch Circuit

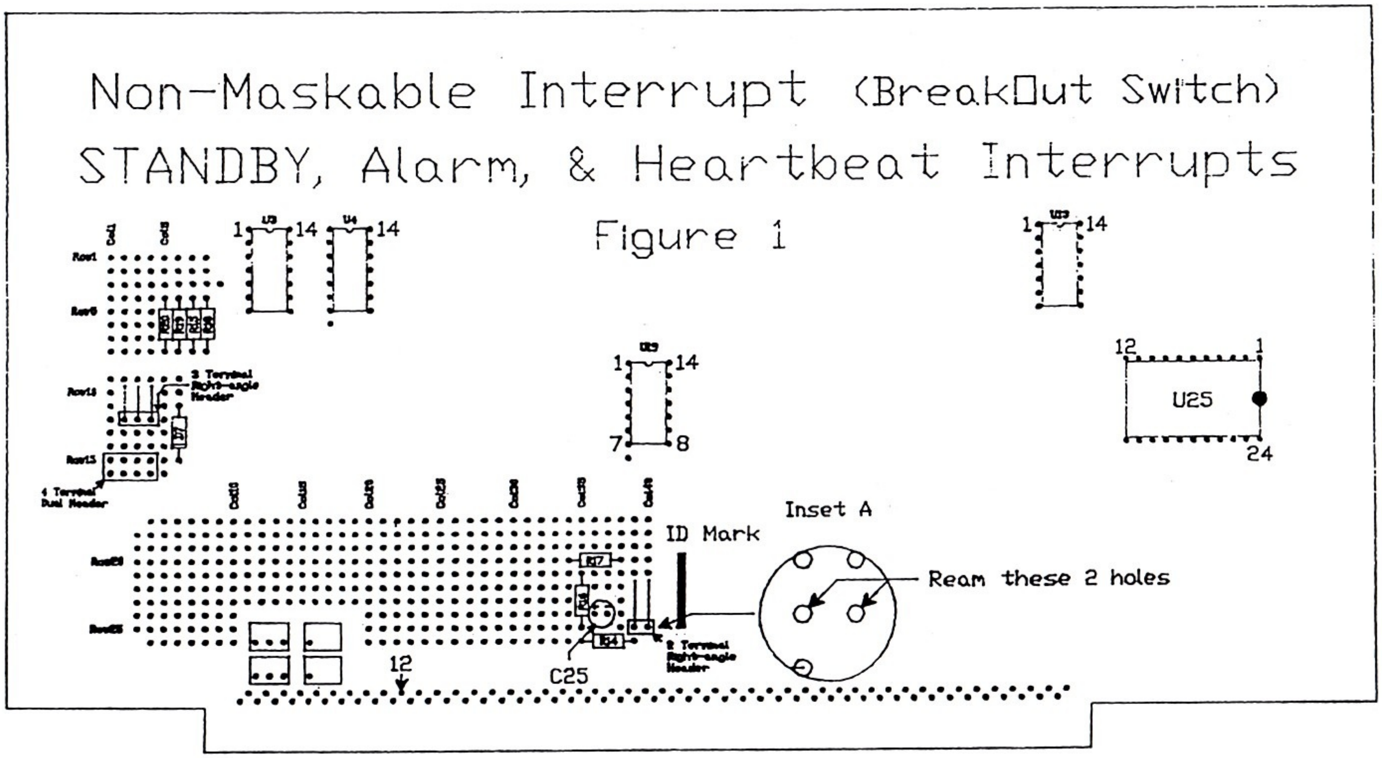


Figure 11 - Scottie Board Breakout Switch Board Layout 1

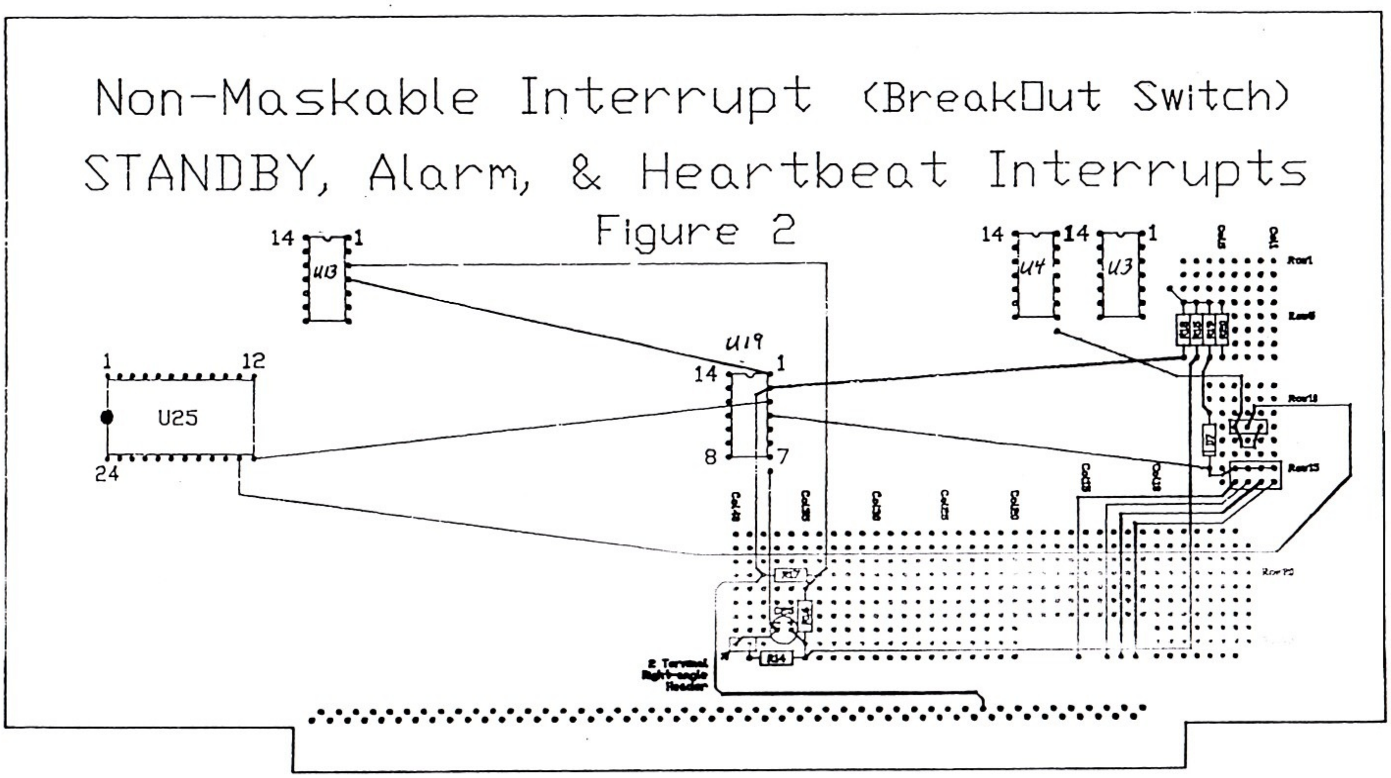


Figure 12 - Scottie Board Breakout Switch Board Layout 2

Scottie Board Parts List:

I do not have the assembly procedures and parts list for the initial Scottie Board. If you wish to construct a Scottie Board from scratch, I have put together a parts list from the schematics by Figure Number, that may help. **You will need a S-100 Prototype Board.**

Figure 3: Address Decode & Interrupt Control

1	U3	74LS90	14-pin Decade Counter
1	U10	Marked 74ALS04 (2 sections)	14-pin 6sec, HEX Inverter
		But diagram shows possibly 74LS00	14-pin 4sec, 2-Input NAND Gate
1	U11	74LS32 (1 section, +2 for clock)	14-pin Quad 2-Input OR Gate
1	U12	74ALS74 (1 section)	14-pin Dual D Flip-Flop
1	U13	74ALS04 (1 section, +2 for COM, +1 for BO)	14-pin, 6sec, HEX inverter
1	U14	Marked 74LS32 (1 section, +1 for Clock)	14-pin Quad 2-Input OR Gate
		But diagram shows a NAND Gate with inverted inputs - **These are equivalent**!	
1	U15	74ALS08 (1 section)	14-pin Quad 2-Input AND Gate
1	U16	74ALS74 (2 sections)	14-pin Dual D Flip-Flop
1	U17	74ALS05 (4 sections)	14-pin 6Sec, Hex Inverter, Open Collector
1	U20	74ALS30A	14-pin 8-Input NAND Gate
1	U21	74ALS04 (3 sections, +1 for Clock)	14-pin 6sec, HEX Inverter
1	U22	74ALS00 (1 sections, + 2 for Clock)	14-pin Quad 2-Input NAND Gate
1	U23	74LS125A (2 sections, + 1 for Clock)	14-pin Quad Buffer/Line Driver w/3-state outputs
1	U26	74ALS520	20-pin 8 bit comparator
3	D4,D5,D6	1N4148 Diodes to set Wait State (See text)	
1	REG	7805, +5v Reg @1Amp	
1	C1	Plus ?? Electrolytic Cap (Possibly another 10uF?)	
3	C?	Plus two 0.1uF Caps, and one unidentified, possibly 1.0uF?	
1	R?	20K ohms (Red-Blk-Org) to Short Blocks)	
1		4x2 Header	

Figure 4: Data Routing & Control

1	U24	74ALS21 (2 sections)	14-pin Dual 4-Input Positive AND Gate
1	U28	74LS244	20-pin Octal Buffer/Line Driver w/3-state outputs
1	U29	74LS244	20-pin Octal Buffer/Line Driver w/3-state outputs
1	R10	3.3K ohms	

Figure 6: COM1 Option;   Figure 7: COM2 Option

1	U1	8250A (COM1)	40-pin 8-bit CPU with ROM/RAM
1	U2	8250A (COM2)	40-pin 8-bit CPU with ROM/RAM
1	U3	74LS90	14-pin Decade Counter
1	U4	1488 (COM1) (3 sections)	14-pin DTL Quad Line Driver
1	U5	1489 (COM1) (5 sections)	14-pin DTL Quad Line Receiver
1	U7	1488 (COM1) (3 sections)	14-pin DTL Quad Line Driver
1	U8	1489 (COM1) (5 sections)	14-pin DTL Quad Line Receiver
1	U9	74ALS30A	14-pin 8-Input NAND Gate
1	U10	74ALS00 (2 sections)	14-pin Quad 2-Input NAND Gate
1	U11	74LS32 (2 sections)	14-pin Quad 2-Input OR Gate
		Note:	The two different diagrams are equivalent.
1	U13	74ALS04 (2 sections)	14-pin 6sec, HEX Inverter
1	U14	74LS32 (1 section)	14-pin Quad 2-Input OR Gate
1	U15	74ALS08 (2 sections)	14-pin Quad 2-Input AND Gate
1	U17	74ALS05 (4 sections) (1 each COM1 & COM2, 2 shared)	14-pin 6Sec, Hex Inverter, Open Collector
1	U21	74ALS04 (1 section)	14-pin 6sec, HEX Inverter
1	U18	Resistor Pack (size??) (sec5)	
1	R1	3.3K ohms (COM1)	
1	R2	4.7K ohms (COM1)	
1	R3	4.7K ohms (COM1)	
1	R4	4.7K ohms (COM2)	
1	R5	3.3K ohms (COM2)	
1	R6	4.7K ohms (COM2)	
1	R7	3.3K ohms	
1	R9	20K ohms	
1	C1	0.1uF (COM1)	
1	C2	0.1uF (COM2)	
1	REG	7812 (340T-12), +12V regulator	
1	C?	Plus 10uF Electrolytic Cap	
1	C?	Plus 0.1uF, 20v cap	
1	REG	7912 (320T-12), -12V regulator	
1	C?	Plus 10uF Electrolytic Cap	
1	C?	Plus 0.1uF, 20v cap	
1	MCO	1.8432 MHz Crystal Oscillator	4-pin, Unicorn Electronics has for $2.79, 4/22
3	D4,D5,D6	1N4148 Diodes to set Wait State (See text)	
1		4x2 header	
1		6x2 header	