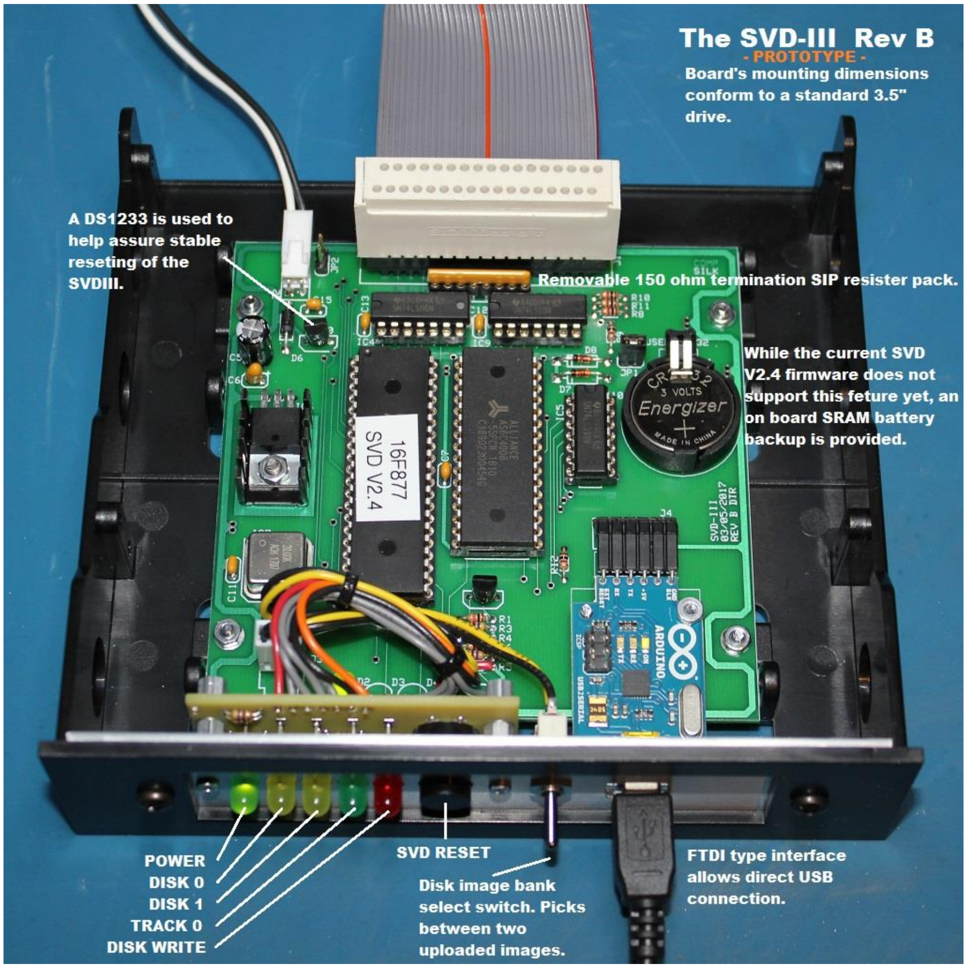


Semi-Virtual Disk III

Revision B

1 - Introduction

The original idea behind the Semi-Virtual Disk (SVD) was invented by Eric Rothfus back in 2003. As Eric stated on his website:

"The SVD was built for fun, and was built with parts that were easy to come by and easy to use. Total parts cost is around $60. Schematics, board layout, and all software can be found here, so feel free to build one yourself. What I recommend, though, is that you just buy one here. I don't intend to make any money off of the SVD; whatever profit beyond the cost of parts will be donated to worthy vintage computer sites and concerns."

The SVD is basically a device that allows you to upload disk images to onboard SRAM. These disk images are used to simulate a floppy disk to your retro computer. Disk images are simply copies of a disk stored in files on your modern day computer.

Eric continued to improve and support the SDV project for several years as time permitted from all his other responsibilities. Around 2008 all improvement activity seemed to stop. The last webpage update was on October 2014.

The updated hardware design presented here is my contribution to the project.

All the features of the original SVD-II work in the new SVD-III. This is no surprise since only the hardware was updates. No changes to the firmware were made and hardware testing was done using the last SVD release V2.4.

In addition to optimizing the hardware design, several enhancements were added.

I my original SVD-II board an annoying power up reset bug found. Sometimes the microprocessor would not reset correctly on power up. You were required to press the reset button to force a clean reset. In addition, the lower 8-bit of the SRAM address line counter had no provisions to assure it would reset in a zero state. A DS1233 was added to assures that both the microprocessor and 8-bit address line counter would be in a solid reset condition after the power supplies became stable.

When drive termination was required, an external 150 ohm resistor pack was placed on adapter board between the SVD-II and the interface cable. A 150 ohm termination SIP resistor pack was added to the main board. This allows the termination resistor pack to be removal or addition as needed right on the SVD-III. No external hardware needed.

A connector used to supports a FTDI module is used in place of a RS232 interface. Through the FTDI module; you simply plug the SVD-III in to a standard USB port to the supporting computer system, select the correct COM port on the PC software and start uploading. The use of the FTDI means that you can set the baud rate to the maximum that the SVD software can handle without issue. An Arduino FTDI module (since I have link 50 of them still in their unopened original anti-static bags) is used. The board should work with others versions though.

The SRAM has been consolidated in to a signal chip. In addition, the new SRAM chip doubles the disk image memory space. The current version of SVD firmware does not support this larger memory size. It is hope that a future updates might take advantage of this increase capacity. In the meantime a disk image bank select switch option was added. This bank select switch allows two image banks to be supported.

Testing shows that you can place the SVD-III to use Bank 0 and upload disk images to that location. You then switch to Bank 1 and upload another set of images there. You can then switch between the two Banks. For example you may uploaded HDOS in to Bank 0 and CP/M in to Bank 1. Then you would switch between the two Banks and perform a cold boot on the computer to enter the new operating system environment.

One limitation found using this bank switching is if you upload two disk images into Bank 0 and then upload only a signal disk image in to Bank 1. When you switch back to Bank 0 that should have two disk images but it turns out you lose the second image. Looks like the firmware setups an image counter so if you upload only one image, the SVD firmware will not look for a second image.

A battery backup to maintain the SRAM after a power down was also added. Again the current firmware does not use the SRAM after a reset or power cycle until you upload a new disk image. It is hoped that someone will fix the firmware to take advantage of the battery backed up of SRAM since this would be a really nice feature to have.

2 - PC board Parts list

Label	Description	Digikey PN	MFG	MFG PN	Cost each
B1	Battery CR2032 3V Lithium	P189-ND	Panasonic	CR2032	$0.29
B1	Battery holder type CR2032	36-1066-ND	Keystone	1066	$1.79
C1	Capacitor, ceramic, 0.001uF, 50V, 20%	399-9770-ND	Kemet	C320C102M5R5TA	$0.29
C2	Not used	N/A	N/A	N/A	$0.00
C3	Not used	N/A	N/A	N/A	$0.00
C4	Not used	N/A	N/A	N/A	$0.00
C5	Capacitor, aluminum 47uF@25V 20%	493-15369-1-ND	Nichicon	UKL1E470KEDANATD	$0.49
C6	Capacitor tantalum 2.2uF 35V 10%	399-3558-ND	Kemet	T350C225K035AT	$1.16
C7	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C8	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C9	Not used	N/A	N/A	N/A	$0.00
C10	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C11	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C12	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C13	Capacitor, ceramic, 0.1uF, 50V, 20%	399-4151-ND	Kemet	C315C104M5U5TA	$0.14
C14	Not Used	N/A	N/A	N/A	$0.00
C15	0.001uF	399-9770-ND	Kemet	C320C102M5R5TA	$0.29
D1	LED, Green diffuse 5mm round	754-1265-ND	Kingbright	WP7113LGD	$0.28
D2	LED, yellow diffuse 5mm round	754-1268-ND	Kingbright	WP7113LYD	$0.28
D3	LED, yellow diffuse 5mm round	754-1268-ND	Kingbright	WP7113LYD	$0.28
D4	LED, Green diffuse 5mm round	754-1265-ND	Kingbright	WP7113LGD	$0.28
D5	LED, red diffuse 5mm round	754-1266-ND	Kingbright	WP7113LID	$0.28
D6	Diode, 1N4001 50V@1A	1N4001GOS-ND	On Semiconductor	1N4001G	$0.19
D7	Diode Schottky SD103C	SD103CVSCT-ND	Vishay	SD103C-TR	$0.28

D8	Diode Schottky SD103C	SD103CVSCT-ND	Vishay	SD103C-TR	$0.28
IC1	IC, SRAM 512K x 8 55nS	1450-1027-ND	Alliance	AS6C4008-55PCN	$5.03
IC1	Socket, 32-pin dip	AE10030-ND	Assmann	AR 32 HZL/01-TT	$2.87
IC2	IC, DS1233	DS1233-10+-ND	Maxim	DS1233-10+	$1.56
IC3	Not used	N/A	N/A	N/A	$0.00
IC4	IC, 74LS06	296-15069-5-ND	TI	SN74LS06N	$0.87
IC4	Socket, 14-pin dip	AE10012-ND	Assmann	AR 14 HZL-TT	$0.79
IC5	IC, 74LS393	296-1665-5-ND	TI	SN74LS393N	$1.19
IC5	Socket, 14-pin dip	AE10012-ND	Assmann	AR 14 HZL-TT	$0.79
IC6	IC, 7805 5V@1.5A TO-220	MC7805CT-BPMS-ND	Micro Commercial	MC7805CT-BP	$0.48
IC6	TO-220 heatsink	HS107-ND	Aavid	577202B00000G	$0.35
IC6	Machine Screw, M3 x 8mm	335-1149-ND	APM	RM3X8mm 2701	$0.43
IC6	Hex Nut, M3	H762-ND	B&F	MHNZ 003	$0.04
IC7	OSC, XO 20.000MHz TTL	535-9173-5-ND	Abracon	ACH-20.000MHZ-EK	$1.68
IC8	IC, 16F877 MCU 8BIT 14KB FLASH	PIC16F877-20/P-ND	Microchip	PIC16F877-20/P	$5.93
IC8	Socket, 40-pin dip	AE10018-ND	Assmann	AR 40 HZL-TT	$2.10
IC9	IC, 74LS00	296-1626-5-ND	TI	SN74LS00N	$1.14
IC9	Socket, 14-pin dip	AE10012-ND	Assmann	AR 14 HZL-TT	$0.79
IC10	IC, DS1233	DS1233-10+-ND	Maxim	DS1233-10+	$1.56
J1	Header, 34-pin, 0.1 pitch	A33168-ND	Amp	5103308-7	$2.44
J2	Header, 2-pin, 0.1 pitch, right angle	A1926-ND	Amp	640457-2	$0.14
J3	Header, 3-pin, 0.1 pitch, right angle	A19480-ND	Amp	640457-3	$0.17
J4	Socket, 6-pin 0.1 pitch	SAM1089-50-ND		SLW-150-01-G-S	$0.84
JP1	Header, 2-pin 0.1 pitch	3M9447-ND	3M	961102-6404-AR	$0.16
JP1	2-pin Jumper	952-2165-ND	Harwin	M7567-46	$0.23
JP2	Header, 2-pin 0.1 pitch	3M9447-ND	3M	961102-6404-AR	$0.16
JP2	2-pin Jumper	952-2165-ND	Harwin	M7567-46	$0.23
Q1	Transistor, 2N3906 PNP 40V TO-92	2N3906FS-ND	Fairchild	2N3906BU	$0.19
R1	Resistor, 10K ohm, 1/8 watt, CF, axial	CF18JT10K0CT-ND	Stackpole	CF18JT10K0	$0.05
R2	Not Used	N/A	N/A	N/A	$0.00
R3	Resistor, 470 ohm, 1/8 watt, CF, axial	CF18JT470RCT-ND	Stackpole	CF18JT470R	$0.05
R4	Resistor, 470 ohm, 1/8 watt, CF, axial	CF18JT470RCT-ND	Stackpole	CF18JT470R	$0.05

R5	Resistor, 470 ohm, 1/8 watt, CF, axial	CF18JT470RCT-ND	Stackpole	CF18JT470R	$0.05
R6	Resistor, 470 ohm, 1/8 watt, CF, axial	CF18JT470RCT-ND	Stackpole	CF18JT470R	$0.05
R7	Resistor, 470 ohm, 1/8 watt, CF, axial	CF18JT470RCT-ND	Stackpole	CF18JT470R	$0.05
R8	Resistor 1K ohm	CF18JT1K00CT-ND	Stackpole	CF18JT1K00	$0.05
R9	Resistor 1K ohm	CF18JT1K00CT-ND	Stackpole	CF18JT1K00	$0.05
R10	Resistor 1K ohm	CF18JT1K00CT-ND	Stackpole	CF18JT1K00	$0.05
R11	Resistor 1K ohm	CF18JT1K00CT-ND	Stackpole	CF18JT1K00	$0.05
R12	Resistor 1K ohm	CF18JT1K00CT-ND	Stackpole	CF18JT1K00	$0.05
RN1	Resistor network SIP 150 ohm	4308R-101-151LF-ND	Bourns	4308R-101-151LF	$0.84
RN1	8-Pin SIP socket	A460-ND	Aries	40-0518-10	$0.85
FTDI	Arduion FTDI module		Arduino		$14.75
FTDI	Connector Right Angle 6-Pin	3M9471-ND	3M	961106-5604-AR	$0.45
FTDI (2)	Screw, M2.5 x 12mm Nylon	36-29334-ND	Keystone	29334	$0.21
FTDI (6)	Nut, Hex M2.5x0.45 Nylon	36-4687-ND	Keystone	4687	$0.14
PCB	SVD-III Rev B PC Board	N/A	DTR	SVDIII-Rev B	TBD

3 – Assembly Notes

Assembly

Follow the instructions carefully and read the entire step before you perform the operation. Solder a part or group of parts only when you are instructed to do so.

Each circuit part in an electronic kit has its own component number (R2, C4, stc.). Use these numbers when you want to identify that same part in the various sections of the Manual. These numbers, which are especially useful if a part has to be replaced, appear:

- In the Parts List
- At the beginning of each step where a component is installed
- In the schematic

SAFETY WARNING: Safety glasses are recommended. Avoid eye injury when you cut off excess lead lengths. Hold the leads so they cannot fly toward your eyes.

Soldering

Soldering is one of the most important operations you will perform while assembling your kit. A good solder connection will form an electrical connection between two parts, such as a component lead and a circuit board foil. A bad solder connection could prevent an otherwise well-assembled kit from operating properly.

It is easy to make a good solder connection if you follow a few simple rules:

1. Use the right type of soldering iron. If available a temperature controlled soldering iron is recommended. Otherwise us a 25 to 40-watt pencil soldering iron. In either case use a 1/8" or 3/16" chisel or pyramid tip for best results.
2. Keep the soldering iron tip clean. Wipe it often on a wet sponge or cloth; then apply solder to the tip to give the entire tip a wet look. This process is called tinning, and it will protect the tip and enable you to make good connections. When solder tends to "ball" or does not stick to the tip, the tip needs to be cleaned and retinned.
3. A good solder connection is made when you heat the component lead and the foil on the circuit board at the same time. This will allow the solder to flow evenly onto the lead and foil. The solder will then make a good electrical connection between the lead and the foil.

Board Assembly

NOTICE- There was an error in the foil layout on the revision B boards. On J4, pins 4 (TXO) and 5 (RXI) where reversed. Two cut and jumps need to be made on the board to connect this wiring error. Some of the kits shipped have had the 6-pin connector at J4 already installed and the cut-and-jumps already done.

Refer to the silk screen on the printed circuit board or the Board Layout in Appendix B for parts locations on the board.

Install the elevent 1/8 watt resistors at the following locations.

( ) R1: 10K ohm, 1/8-watt, 5% (brown-black-orange)

( ) R3: 470 ohm, 1/8-watt, 5% (yellow-violet-brown)

( ) R4: 470 ohm, 1/8-watt, 5% (yellow-violet-brown)

( ) R5: 470 ohm, 1/8-watt, 5% (yellow-violet-brown)

( ) Solder the leads to the foil and cut off the excess lead lengths.

( ) R6: 470 ohm, 1/8-watt, 5% (yellow-violet-brown)

( ) R7: 470 ohm, 1/8-watt, 5% (yellow-violet-brown)

( ) R8: 1K ohm, 1/8-watt, 5% (brown-black-red)

( ) R9: 1K ohm, 1/8-watt, 5% (brown-black-red)

( ) Solder the leads to the foil and cut off the excess lead lengths.

( ) R10: 1K ohm, 1/8-watt, 5% (brown-black-red)

( ) R11: 1K ohm, 1/8-watt, 5% (brown-black-red)

( ) R12: 1K ohm, 1/8-watt, 5% (brown-black-red)

( ) Solder the leads to the foil and cut off the excess lead lengths.

NOTE: Resistor R2 is not used on this board.

NOTE: The next three diodes are polarized and need to be installed with the banded end matching the outline on the PC board or on the board layout. The lead on the banded in should be inserted into the square hole on the PC board.

( ) D6: Black diode labeled 1N4001

( ) D7: Orange colored glass schottky diode

( ) D8: Orange colored glass schottky diode

( ) Solder the leads to the foil and cut off the excess lead lengths.

NOTE: 14-pin, 32-pin and 40-pin IC (integrated circuit) sockets are used in this kit. Make sure all pins are straight. Carefully insert the socket pins in to the circuit board holes. Make sure that the index notch on the IC socket is on the same end as pin one (a square pad indicates pin 1) of that IC location. All sockets are placed on the compunet side and solder on the solder side of the board. Start by soldering only a single pin to the board. Verify that the socket is aligned and tight against the component side of the board. If not carefully re-heat the pin and reposition the IC socket as needed. Once the IC socket is correctly placed, solder the remaining pins.

40-pin IC socket at the following location.

( ) IC8

32-pin IC socket at the following location.

( ) IC1

14-pin IC sockets at the following locations.

( ) IC4

( ) IC5

( ) IC9

( ) RN1: Use the same process as the IC sockets above to install the 8-pin serial in-line socket.

CAUTION: The 20MHz oscillator module that you will be installing next step is a CMOS device that can be damaged by static electricity. Use the following sequence when you install this oscillator integrated circuits.

1. Pick up the conductive foam block with the oscillator mounted on it.
2. Hold the oscillator in one hand and pull the conductive foam pad from the pins and set the foam pad aside.
3. Pick up the circuit board while you hold the oscillator.
4. Carefully insert the oscillator in indicated holes. Make sure to install the oscillator in the correct orientation on the board. Carefully bend the leads just enough to hold the oscillator in place before you set the circuit board down in to your work surface for soldering.
5. Verify that the oscillator is in the correct orientation and solder the four oscillator leads to the board.

The oscillator is now protected by circuit board's sockets foil.

( ) IC7: 20MHz oscillator

Install the ten capacitors at the following locations.

( ) C1: 0.001uF ceramic.

NOTE: The next two capacitors are polarized and have positive and negative leads. Make sure that the positive lead is inserted into the hold marked + (the square hole) on the board.

( ) C5: 47uF polarized aluminum electrolytic capacitor.

( ) C6: 2.2uF polarized tantalum capacitor.

( ) C7: 0.1uF ceramic.

( ) C8: 0.1uF ceramic.

( ) Solder the leads to the foil and cut off the excess lead lengths.

( ) C10: 0.1uF ceramic.

( ) C11: 0.1uF ceramic.

( ) C12: 0.1uF ceramic.

( ) C13: 0.1uF ceramic.

( ) C15: 0.001uF ceramic.

( ) Solder the leads to the foil and cut off the excess lead lengths.

NOTE: Capacitors C2, C3, C4,C9 and C14 are not used on this board.

NOTE: If you plan to use the remote LED display board option skip the five step for D1 through D5.

NOTE: Installing the LEDs, position the flat side as shown to match the outline on the board. Note that the short LED lead is in the square hole on the board. Hold the LEDs tight to the board and solder the leads to the foil and cut off the excess lead lengths.

( ) D1: Green LED.

( ) D2: Yellow LED.

( ) D3: Yellow LED.

( ) D4: Green LED.

( ) D5: Red LED.

NOTE: The next two part installed are the DS1233. These integrated circuits are packaged in a TO-92 style case and look like transistors. Make sure that the parts used at IC2 and IC10 are not a transistor and are labeled DS1233.

CAUTION: If these parts are installed backwards, they will be destroyed on power up. When installing the DS1233, make sure to align the 3-pin package so the flat side of the part matches the outline on the board's silk screen or the board layout in the appendix.

( ) IC2: Align the DS1233 IC to the silk screen on the board or board layout. Insert each of the 3-pin in to their respective holes and carefully push the part to about 0.1 inches above the board. Bend the leads slightly on the back to hold the part in place..

( ) IC10: Repeat the process above with the second DS1233.

( ) Q1: Align the 2N3906 PNP transistor to the silk screen on the board or board layout. Insert each of the 3-pin in to their respective holes and carefully push the part to about 0.1 inches above the board. Bend the leads slightly on the back to hold the part in place.

( ) Solder the leads to the foil and cut off the excess lead lengths.

( ) J1: 34-pin header. Make sure to install the 34-pin header so that pin 1 is in the hole with the square pad on the board. Solder only one pin then verify that the header is align and tight to the board and pin 1 is in the correct hole. Then solder the remaining pins.

( ) J2: 2-pin right angle header. Match the silk screen or board layout and solder.

( ) J3: 3-pin right angle header. Match the silk screen or board layout and solder.

( ) J4: 6-pin straight header. Solder only one pin then verify that the header is align and tight to the board. Then solder the remaining pins.

( ) JP1: 2-pin jumper header. Match the silk screen or board layout and solder.

( ) JP2: 2-pin jumper header. Match the silk screen or board layout and solder.

( ) B1: CR2032 Battery holder. Match the silk screen or board layout and solder.

( ) IC6: Locate the following four parts used to assemble the 7805 voltage regulator to the board:

- Machine Screw, M3 x 8mm
- Hex Nut, M3
- TO-220 heatsink
- IC, 7805 5V@1.5A TO-220

( ) The three leads on the 7805 need to be pre-bent before installation. Notice that the leads on the 7805 are wider where they exit the black body of the 7805 and then narrow down. Where the transition from wide to narrow occurs, bend the three leads down towards the metal heat sink side of the 7805 to an angle of 90 degrees. Temporarily set the 7805 aside until required.

( ) From solder side of the board, insert the M3 x 8mm machine screw through the mounting hole at IC6. On the component side of the board, Place the TO-220 heatsink's mounting hole over the protruding M3 machine screw. Make sure to align the heatsink so that it fits best on the large foil area for IC6.

( ) Install the 7805 at IC6 so that the three pre-formed leads pass through the correct holes and the mounting hole on the metal heatsink tab passes through the protruding M3 machine screw. Do not solder.

( ) Secure the heatsink and 7805 regulator to the board using the M3 machine screw with the M3 hex nut. Torque the nut on to the screw just enough to secure the parts. Do not over tighten.

( ) Verify the correct placement and alignment of the 7805 assembly and, solder the leads to the foil and cut off the excess lead lengths.

4 – Testing

Before we install the ICs, FTDI module or battery; we will perform a few basic power up operational tests.

( ) Carefully check any for solder bridges between pins. If a solder bridge has occurred, hold the circuit board solder side down and hold the soldering iron tip between the two points that are bridged. The solder will flow down the soldering iron tip.

( ) Verify that all parts are in their correct locations.

( ) Set an ohm meter to read the lowest resistance setting and measure across the two pins on J2. If the resistance reading in close to zero (a short) then some form of electrical short exists. Carefully inspect the board assembly for solder bridges or parts installed backwards or at incorrect locations. Correct and issues found and re perform this test.

( ) We will be using a bench or other 8-24 volt DC power supply. Make sure the power supply is off. Connect the positive lead from the power supply to pin 1 on J2 (the pin with the square pad) and the negative lead to pin 2 on J2.

( ) Set a voltmeter to read a positive 5 volts DC range. Connect the positive lead of the voltmeter to pin 14 on IC5 and the negative lead to pin 7 on IC5.

( ) Turn on the power supply and note the reading on the voltmeter. You should be reading a positive DC voltage between 4.9 and 5.1 volts. The green power LED should be on. Turn off the power supply.

( ) If you plan on using the battery backup option, install the CR2032 battery at location B1 and insert the two pin jumper on to JP1.

( ) Set a voltmeter to read a positive 5 volts DC range. Measure the voltage reading between the positive pin 32 on IC1 and the negative pin 16 on IC1. You should have a reading between 2 and 3 volts DC.

( ) While leaving the voltmeter connected to the same pins in the previous test, turn on the power support. The Green power LED D1 should light and the voltage read should now be between 4 and 5 volts DC.

( ) Turn off the power supply, disconnect the power connect at J2 and remove the CR2032 battery at B1.

5 – Final assembly

NOTE: In the following steps, install ICs (integrated circuits) in the designated sockets. Be careful to match the pin 1 end of each integrated circuits to the index mark on the socket.

Before you apply downward pressure to an integrated circuit, make sure each integrated circuits pin is centered in its proper socket hole. Handle integrated circuits with care, as their pins bend very easily.

CAUTION: The integrated circuits that you will install are CMOS or MOS devices that can be damaged by static electricity. Use the following sequence when you install the integrated circuits.

1. Pick up the conductive foam block with the desired integrated circuits mounted on it.
2. Hold the IC in one hand and pull the conductive foam pad from the pins.
3. Pick up the circuit board while you hold the integrated circuits.
4. Carefully insert the IC in its socket before you set the circuit board down in to your work surface.

The IC is now protected by circuit board's sockets and foil.

( ) IC1: AS6C4008 – 512K x 8 SRAM

( ) IC4: 74LS06 – Hex open collector inverters

( ) IC5: 74LS393 – Binary counter

( ) IC8: PIC16F877 – 8-bit MCU

( ) IC9: 74LS00 – Quad NAND gates

( ) J4: Locate the following parts used to assemble the FTDI module to the board:

- Quantity 2 - Screw, M2.5 x 12mm Nylon
- Quantity 6 - Nut, Hex M2.5x0.45 Nylon
- Header Right Angle 6-Pin
- Arduino FTDI module

( ) From the solder side of the board, insert a M2.5 x 12mm Nylon screw through the right hand mounting hole within the FTDI module area of the board just below J4. From the component side, use two M2.5 x 0.45mm Nylon nuts to secure the M2.5 x 12mm screw. The two Nylon nuts should create a board-to-board spacer between the SVD-III and soon to be installed FTDI boards.

( ) Using a similar method as in the previous step, insert a M2.5 x 12mm Nylon screw through the left hand mounting hole within the FTDI module area of the board just below J4. From the component side, use two M2.5 x 0.45mm Nylon nuts to secure the M2.5 x 12mm screw.

WARNING: In the next steps you will be using a 6-pin right-angle header to connect the FTDI module to the 6-pin connector at J4. If you are planning on powering your SVD-III through the USB power on your PC, it is not recommended to connect the two pin power connect at J2 at another power source. This could provide a path to back feed supply voltage from the SVD-III through the FTDI module back in to your PC's USB port.

( ) Optional modification of the 6-pin right-angle header. If you plan on using the USB connection, through the FTDI module to power your SVD-III, no modifications are required. If you plan on using the on-board voltage regulator to power your SVD-III, you should remove pin 3 on the 6-pin right angle header before using it to connect the FTDI module to the SDV-III. The removed pin isolates the USB voltage supply from the on board regulator supply.

( ) Plug the right angle 6-pin header in to the 6-pin connector of the FTDI module. Using the two mounting holes of the FTDI module, align the holes to pass over the two M2.5 x 12mm screws. Press the FTDI module down so that the open end of the six pin header plugs in to the 6-pin connector at J4.

( ) Verify the correct position and alignment of the FTDI module over the two screws and that the 6-pin header is engaged in to both the connector at J4 and the connector on the FTDI module. Now use the remaining two M2.5x45 Nylon nuts to secure the FTDI module to the SVD-III board assembly.

This completes the assembly of the SVD-III board.

6 - Circuit description

When reviewing the following circuit description, please refer to the schmatic located in Appendix A.

The heart of the SVD-III is the Microchip PIC16F877 MCU labeled IC8 on the board. The firmware programmed within the PIC16F877 performs two key operations. First it creates the RS232 serial port used to upload or download disk images to/from SVD-III's SRAM.

Next the it listens to the floppy control signals from your old computer and generates the appropriate "floppy-looking" signals back. The in essence convinces your old computer that it is talking to a real floppy drive unit.

Disk images are stored in IC1, a 512K x 8 bit static RAM. The 512K bytes RAM is split into two 256K Banks. Bank selection is handled by changing the high/low state of the A18 address line on IC1. IC2 a DS1233 and the 0.001uF capacitor C1 is used to create the switch debounce used by the incoming control signal on pin 2 of connector J3. When pin 2 on J3 is pulled low, address line A18 on the SRAM is also low selecting Bank 0. When pin 2 on J3 is opened, the DS1233 will continue to hold the input to A18 active low state for approximately 350ms to assure there is no switch bounce.

The lower 8-bits of the SRAM are driven by IC5, a 74LS393 setup as an 8-bit counter. The PIC simply pulses the memory increment line to increment the lower 8-bits of the SRAM address. The upper address bits are directly separated by the PIC16F877.

The SRAM maybe protected with a battery backup. Two SD103C schottky diodes are used to combine the board's +5 volt DC supply source D7 and a +3V DC battery source supplied by a CR2032 at B1. The positive supply from the battery passes through shorting jumper JP1 allowing the battery supply to be disabled when not needed.

Power-on and RESET button debouncing is supplied by IC10, the second DS1233 on the board. This DS1233 monitors the power supply voltage and will pull and hold the PIC18F877's MCLR (pin 1) low until a safe operational voltage is restored. Like the other DS1233 it also acts as a debounce circuit for RESET switch input on pin 1 of J3. C15, a 0.001uF capacitor, helps the DS1233 setup proper de-bounce timing when the RESET signal is released. Pulling pin 1 of J3 to ground will place the SVD-III in to a RESET mode.

The output of IC10 is also passed to the input of on the first open collector inverter in IC4, a 74LS06. Since the 74LS06's outputs are all open collector, the 1K ohm resistor at R12 is needed to pull the output high when the inverter input is low. The resulting RESET signal is connected to 74LS393 8-bit counter at IC5's clear(CLR) input. This will clear IC5's counter to a know zero state after a power cycle or RESET operation.

On board five volt power regulation is provided by is simple analog voltage regulator circuit made up of a 1N4004 diode D6, a 47uF electrolytic capacitor C5, a 7805 voltage regulator IC6 and a tantalum capacitor C6. Power is presented via connector J2. The positive side of J2 pin 1, is passed through the diode D1. This 1N4001 helps prevent accidental reversals on the power supply input voltage. In normal operation the 1N4001 is forward biased allowing current to flow through. If the voltage becomes reversed, the diode will be reverse biased blocking current flowing in to the regulator circuit. Capacitor C5 provided some DC input voltage ripple stabilization. The 7805 at U5 is used to regulate input voltages between 8 and 24 volts DC down to a regulated 5 volts DC. The 7805 is mounted to an appropriate heat sink so when input voltages are at greater than 12 volts DC the 7805 will run within the safe operational temperatures. U6 is used to stabilize the final regulated output voltage of the U5.

Six 0.1uF capacitors (C7, C8, C10, C11, C12 and C13) are used to help decuple IC generated power line noise on the 5 volt supply.

The floppy drive interface is through the 34-pin connector J1.

Drive selection is via pins 10, 12 and 14 for drive selects 0, 1 and 2. The MOTOR select Signal is inverted using one inverter channel in IC4 a 74LS06. Since the 74LS06 uses open collector outputs, the 1K resistor at R9 is used to pull the output high when the input to the inverter is low. The inverted MOTOR enable along with the three drive section signals are decoded by the 74LS00 NAND gate at IC9.

The outputs from the 74LS00 drive decoding are passed to both the PIC16F877 and two monitoring LEDs. LED D2 along with 470 ohm resistor R4 monitor the Drive 1 results while LED D3 and 470 ohm resistor R6 monitor the Drive 0 results.

The resulting drive select signaling passed to the PIC16F877 along with the monitoring LED status are shown in the next table.

MOTOR Enable	Drive Selected	Drive 0 - pin 17	Drive 1 - pin 18	Drive 0 LED	Drive 1 LED
Low	0	Low	High	On	Off
Low	1	High	Low	Off	On
Low	2	Low	Low	On	On
High	Don't care	High	High	Off	Off

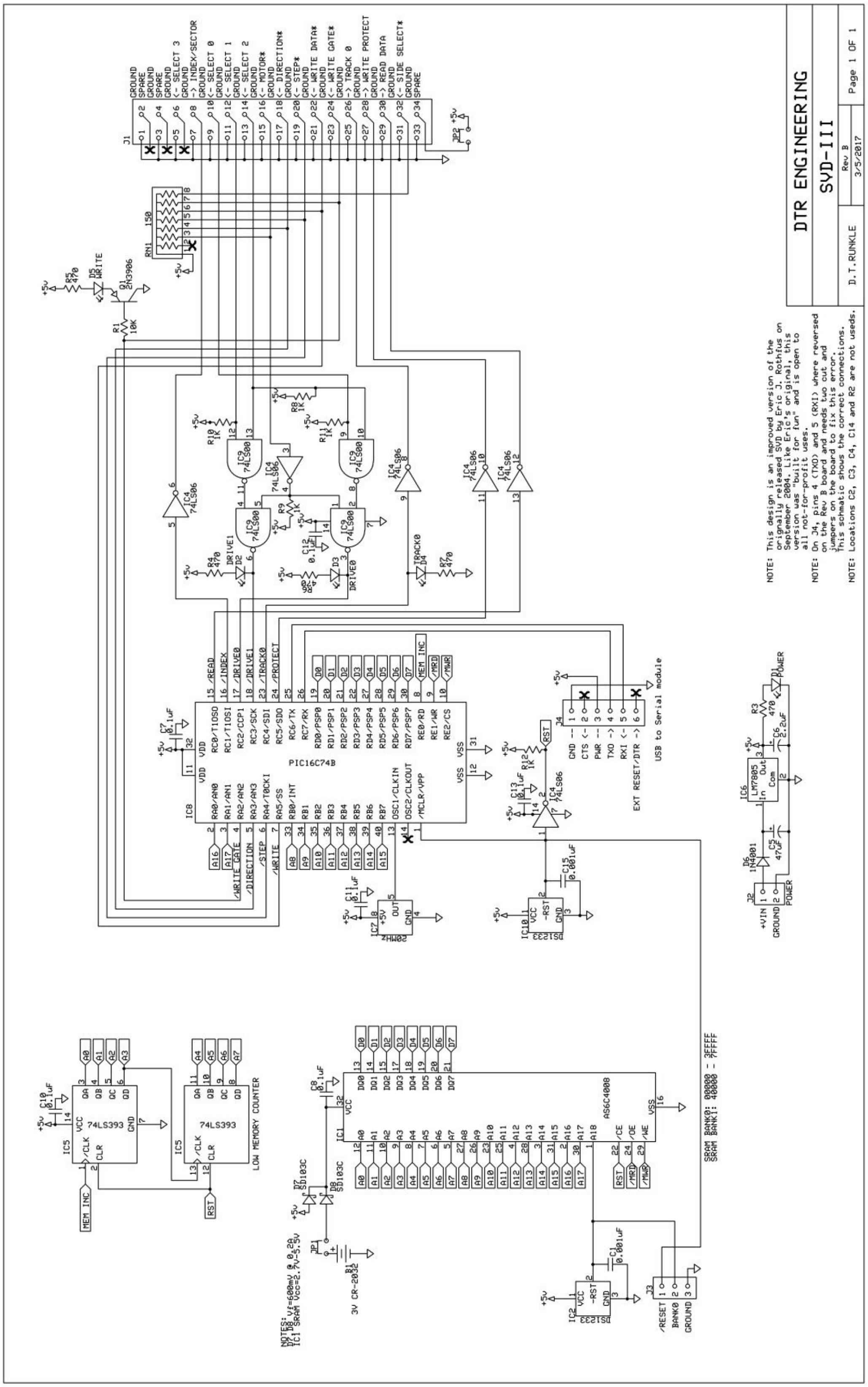
The four outputs INDEX/SECTOR, TRACK 0, WRITE PROTECT and READ DATA are sent from the SVD-III to the computer. Each of these outputs are passed through the four remaining open collector inverters found in the 74LS06 at IC4. The four output Signals are normally in the low or false state placing their respected inverters outputs in a high impedance or open state.

RN1 is a 150 ohm by seven SIP resistor package used as a terminator for the last drive in the floppy drive chain. RN1 is use to pull-up the input Signals MOTOR ENABLE, DIRECTION, STEP, WRITE DATA, WRITE GATE and SIDE SELECT. RN1 is setup with a SIP socket on the PC board allowing the terminator to be installed or removed as needed.

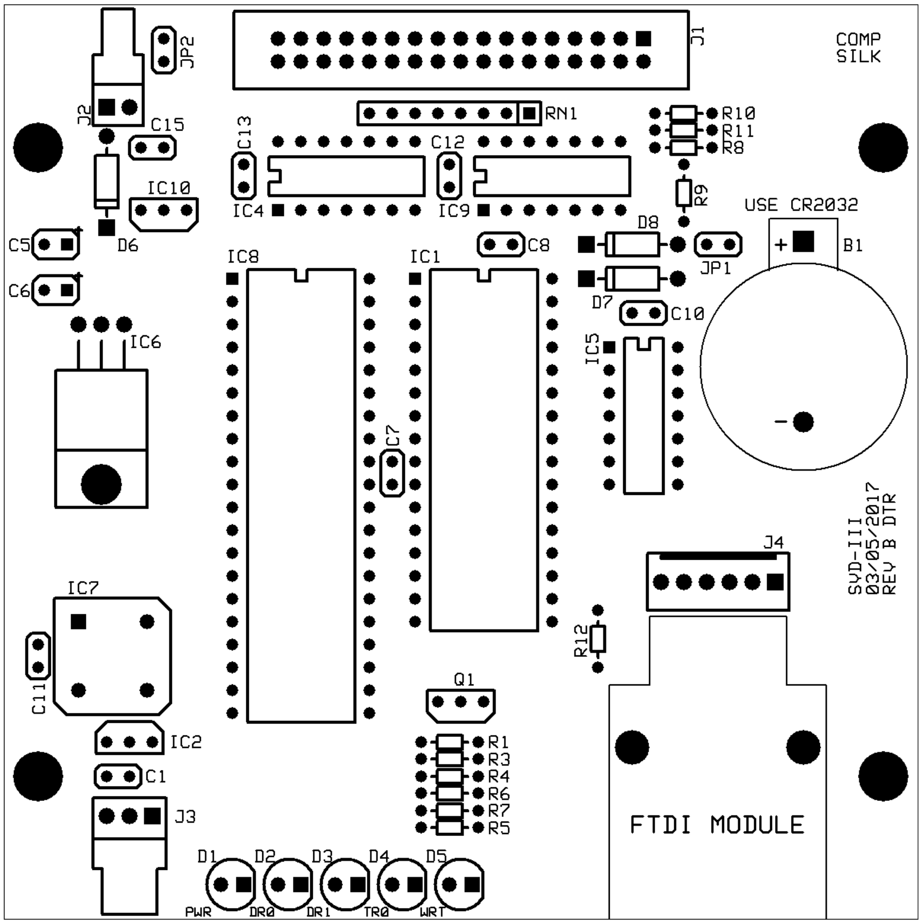
LED D4 and 270 ohm resistor R7 are setup to monitor the status of the TRACK 0 output signal.

10K ohm resistor R1, 2N3906 PNP transistor, LED D5 and 470 ohm resistor are used to provide monitoring of the WRITE GATE input signal. When the WRITE GATE is low, current flows from the base of PNP transistor Q1 turning the transistor on. This allows current to flow through the 470 ohm current limiting resistor R5 in to LED D5 lighting the LED.

Appendix A – The schematic



Appendix B – Board layout



Appendix C – Pinouts

J1 – 34-pin Disk Drive Header

Pin	Description
1	Ground
2	Not connected
3	Ground
4	Not connected
5	Ground
6	Not connected
7	Ground
8	Index / Sector
9	Ground
10	Select 0
11	Ground
12	Select 1
13	Ground
14	Select 2
15	Ground
16	Moter
17	Ground
18	Direction
19	Ground
20	Step
21	Ground
22	Write Data
23	Ground
24	Write Gate
25	Ground
26	Track 0
27	Ground
28	Write Protect
29	Ground
30	Read Data
31	Ground
32	Side Select
33	Ground
34	+5VDC when JP2 is installed

J2 – 2-Pin Power Connector

Pin	Description
1	+8 through 24 volts DC in
2	Ground

J3 – 3-Pin Mode Connector

Pin	Description
1	RESET
2	SRAM Bank Select
3	Ground

J4 – 6-Pin FTDI Connector

Pin	Description
1	Ground
2	Not connected
3	+5VDC power
4	TXO
5	RXI
6	Not connected

Appendix D – Costs

As mentioned in the introduction, the SVD-III is a built for fun. I have no plans to even try and make any money off this project. So all boards and parts pricing are designed to only reimburse my costs. Within the United State I'll be using United State Postal Service Priority Mail Small Flat Rate Boxes at $7.15 to ship boards and if requested parts. For shipments outside the U.S., I'll need to determine the shipping costs and any customs costs.

Since I will not be ordering and stocking boards, PC Board costs are determined by the quantity of boards that need to be ordered to cover any requests for boards. The more people that wish boards, the cheaper to boards will cost per person. The following table will give you a general idea of the current per board costs as of March 2017. Remember that boards are ordered in step of two. So if 14 boards are ordered the cost will be between the 10 and 20 amounts listed in the table.

Board count	Production & Shipping Costs	Setup Fee	Material Costs	Shipping	Discount	Cost per board
4 (Prototype)	$180.05	166.00	N/A	$14.04	$0.00	$45.01
10	$321.60	$250.00	$54.40	$17.20	$0.00	$32.16
20	$381.00	$255.00	$108.00	$17.20	$0.00	$19.05
30	$442.80	$260.00	$163.20	$19.60	$0.00	$14.76
40	$504.95	$265.00	$217.60	$22.35	$0.00	$12.62
50	$507.88	$270.00	$272.00	$25.50	-$59.62	$10.16

PC Board Costs

The current cost for all the parts (less PC board) ordered from Digikey is $57.25 as of March 31, 2017.

Optional - The cost for the remote LED PC board is $4.00. This board allows you to place the LEDS and reset button at a location separate from the main SVD-III board. Seven 24 AWG stranded wires are used to interconect the remote LED PC board to the main SVD-III PC board.