

EE 354
Assembly of the AT89C51CC03 board

August 1, 2017

The AT89C51CC03 board comes as a kit which you must put together. The kit has the following parts:

No.	ID	Description
1		1.5" x 3.25" printed circuit board labeled "AT89C51CC3" dated 07/01/11 . For this kit, the surface mounted parts have been installed for you. There are 4 capacitors on the bottom of the board and three on the top. The surface mounted USB communications chip has also been mounted.
2	C1	22pf surface mount bottom
3	C2	22pf surface mount bottom
4	C3	10uf surface mount top
5	C4	100uf surface mount top
6	C5	.1uf surface mount bottom
7	C6	0.1uf surface mount bottom
8	CN1	19-pin header for port 0 and 2 and for D/A converter plus power and ground
9	CN2	16-pin header for port 1 and 3 plus power
10	D1	Red Power
11	J1	USB connector
12	Q1	IRLML6402
13	R1	1K resistor
14	R2	10K resistor
15	R3	10K resistor
16	R4	1K resistor
17	R5	4.7K resistor
18	R6	0.51K resistor
19	R7	2K resistor
20	R8	3K resistor
21	R9	1K resistor
22	SW1	PSEN
23	SW2	Reset
24	U1	52-pin PLCC socket for 89C51CC03
25	U2	16-pin socket for D/A Converter - DAC7524
26	U3	USB Communications chip -FT232RL surface mount
27	U4	8-pin socket for op amp - LM358
28	XTal	28.2076 MHz crystal (short style)

All of the surface mounted parts have been soldered on for you. Figure 1 below shows the two sides of the board with the surface mounted parts in place. You will be placing parts on the top side and soldering the pins of those parts on the bottom side.

Verify that your board has all 6 capacitors in place and that C4 (the polarized capacitor) is on the board as is shown in Figure 1 (with the white stripe to the left).

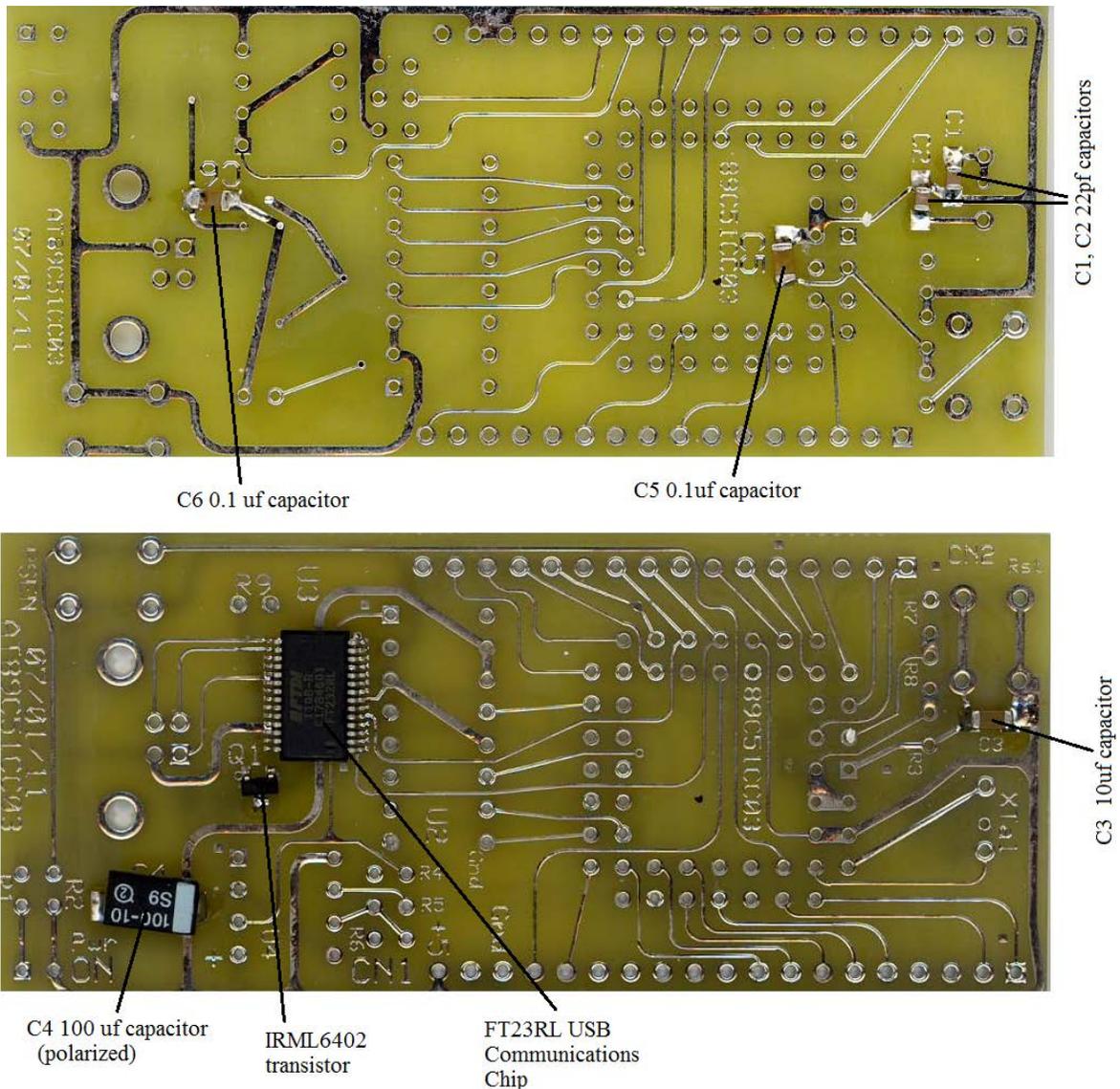


Figure 1 The bottom (above) and top (below) of the AT89C51CC03 USB board with surface mounted parts in place. Components should be placed on the top side. Pins on the bottom should be soldered in place.

To assemble:

1. The parts will all be placed on the top side of the board but many will be soldered on the bottom side. In general we solder the shortest parts in first because they cause the least disruption when we turn it over to solder on the back.
2. Put the IC sockets for U2 and U4 in place on the board. Most IC sockets have a small cutout on one side and it is accepted practice to place this cutout on the side nearest pin 1. Pin 1 on the printed circuit board is always designated by a square solder pad (other pins have a round

solder pad.) Put the sockets in place and turn the board over and solder the pins. You will have 16 pins to solder for U2 and 8 pins to solder for U4.

When you solder the pins make sure that the soldering iron tip touches and heats both the pin AND the pad around the hole. Touch a small amount of solder to the pin and it should flow to the pad. If it doesn't flow, it may not be properly connected.

3. Begin by putting the D1 (Power on LED) in place on the top side near the label that says "Pwr ON". The cathode on this diode is marked flat side and the short lead. This cathode goes in the square pad. Turn the board over and solder on the back. Trim the leads.
4. Put connectors CN1 and CN2 in place. Note that these go in with the short side up and the black plastic header on the bottom side of the board. See the picture of the completed board in Figure 2. CN1 is a 16-pin connector and CN2 is a 19-pin connector. These may come as one long strand of pins. If so count off as many you need and use a needle nose pliers or some other tool to break off what you need. These connectors are *placed on the bottom and soldered on the top*.
5. Put the switches Sw1 and Sw2 in place on the top side. These are small "tactile" push button switches that handle reset and programming (via the PSEN pin). The switches snap into the holes and you need not bend the pins to get them to fit. You have to try hard to get these in sideways. See Figure 2 if you have doubts. Turn the board over and solder on the back.
6. Put the resistors R1 to R9 on the board. To save space these resistors are mounted upright instead of laying down. See the figure of the completed board. Resistors are not polarized but it is important that you get the right values in the right place. The positions of all of the resistors are labeled but you may need a magnifying glass in some cases because the print is very small. See the Figure 2 photo.
7. Put the 52-pin socket in place on the board. It's easy to get this in backwards and if you do, it will have to be removed which is tedious. Note that the socket has one corner shaved off at a 45 degree angle. Looking at the top of the board as it is in Figure 1, this shaved off corner goes to the top right. Turn the board over and solder all 52-pins in place.
8. Put the crystal in place on the top of the board and solder it on the back. The crystal location is labeled "xtal" and is on the right side of the board in Figure 1. In Figure 1 note that there are three holes for the crystal and you are only going to use two of them (this board was made to accommodate crystals of differing sizes.) There are two larger holes with one on top and one below it with a smaller hole in between. The smaller hole between the two larger holes accommodates a tubular style crystal. This year we have a conventional crystal that will fit into the two larger holes. Put one wire of the crystal in the upper hole and the second in the lower larger hole. Solder these on the back. Try to keep the crystal leads short but the crystal body *should not be touching the pads*. You could stick a very small piece of Scotch tape the underside of the crystal to keep it from touching the board and shorting something out.

9. Finally, solder in the USB connector. This is the silver jack with four pins and two mounting plastic legs. Place a piece of Scotch tape on the bottom side of this jack to keep it from shorting out other lines on the board. Get the pins lined up correctly and push to get the legs to go into the holes. After you get the legs in turn it over and solder in the four pins on the back.

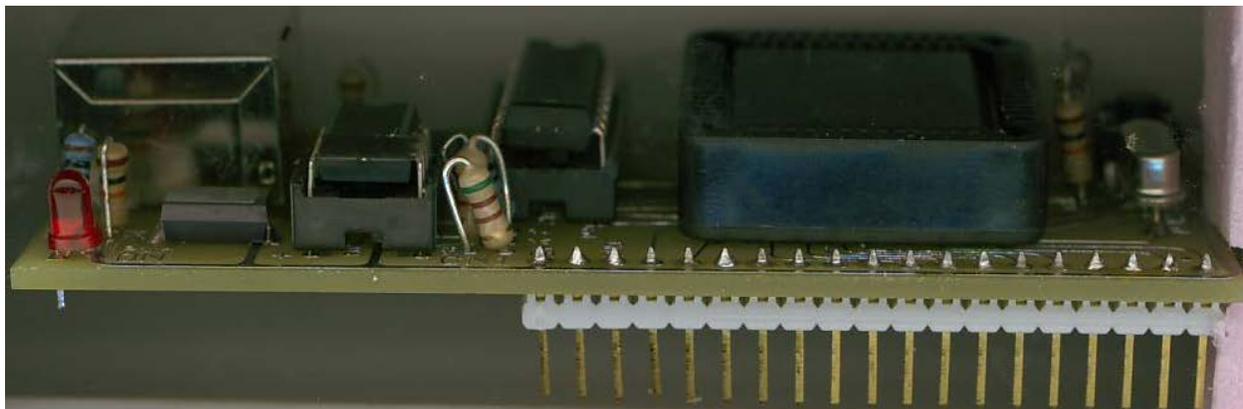
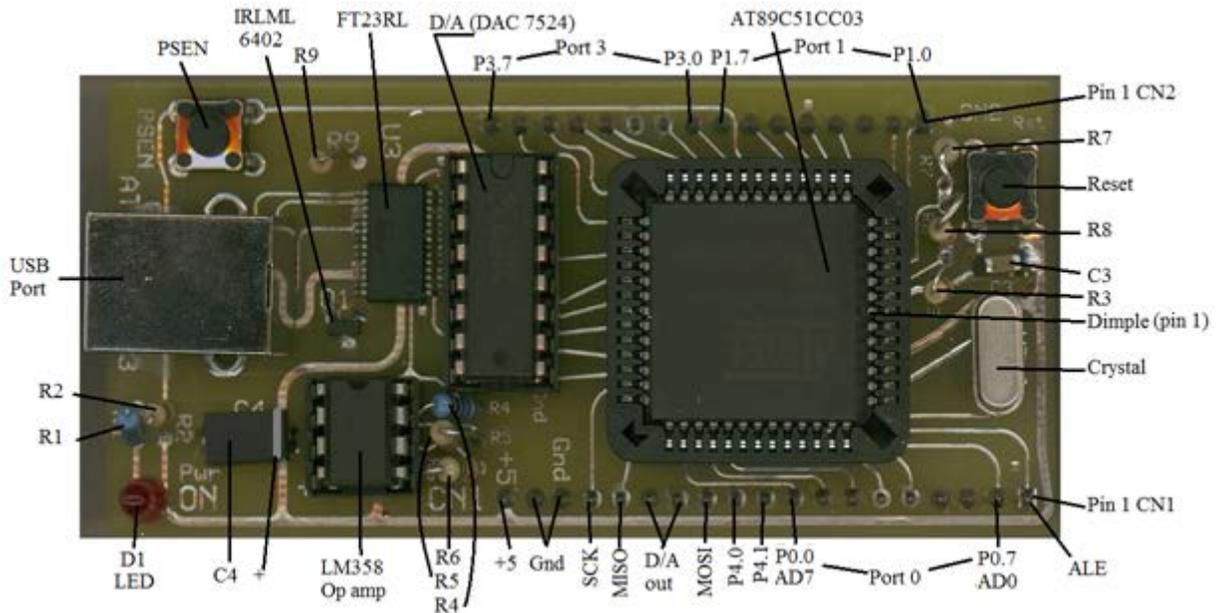


Figure 2

The top and side views of the finished AT89C51CC03 board.

To Test the board:

You will need a breadboard (see Figure 3), a downloaded zip test file called BdTestCC03.zip from the web site at <http://csserver.evansville.edu/~blandfor/EE354/index.htm> plus access to a computer with a serial port running Flip. (You can download Flip from the Atmel web site at http://www.atmel.com/dyn/products/tools.asp?family_id=604

1. Your board is made to plug into a standard plug board as shown in Figure 3. Note that you want to put the board in place such that it straddles the center of the plug board. Do not put

the CPU in the socket just yet. Connect the USB cable to the board. There is no need to connect the board to power and ground since it gets this from the USB connection.

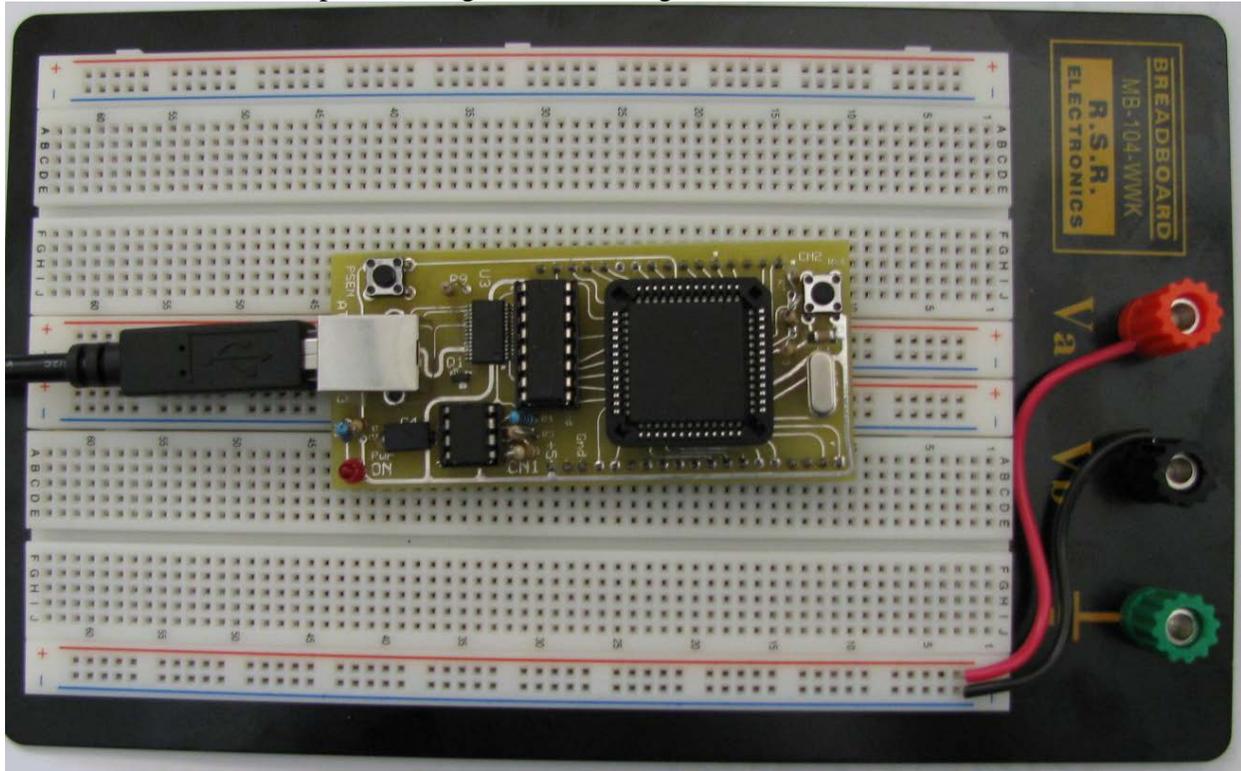


Figure 3

A breadboard with AT89C51CC03 board in place.

2. Plug the USB cable into your computer. Your computer may not be able to find the driver for this USB device. You can download it from <http://www.ftdichip.com/Drivers/VCP.htm>. Select and download the driver for your system and install it. After the driver is installed, again plug your board into the USB connector and connect to your computer. The red LED Pwr On light should come on.
3. If the red LED is not on then you have an error. Most likely you have soldered the LED in backwards. The other possibility is that there is a solder blob somewhere shorting out power and ground. Fix this problem or if you can't, see Jeff Cron or your instructor.
4. If step 2 goes well, unplug the USB connector and insert the CPU chip being careful to get pin 1 in the right place. Pin 1 is marked by a dimple on a beveled edge on the top of the chip. Pin 1 should go to the right (near R3) in Figure 2. Turn the power on and use your finger to see if the CPU or any of the other ICs are running hot. Warm is OK but hot is not good. If something is running hot turn it off immediately and see Jeff Cron or your instructor. Otherwise, you are ready to program the chip.
5. Download the project called BdTestCC03.zip from the web site at <http://csserver.evansville.edu/~blandfor/EE354/index.htm>
This program counts up forever on Port 0, 1, 2, and 3 and toggles pin P4.0. Pin 4.0 triggers a D/A conversion on the data in Port 2. You will see a ramp function coming from the D to A output pin and Ports 0, 1, and 3 will be counting. Unzip the file into its own folder. Inside the unzipped folder you will find a file named BdTestCC03.hex. If the file is not there, you can open the project in Keil μ Vision 3. Build the program to produce a hex file.

6. Connect a USB cable between your board and the computer. The red Pwr On light should come on. Run the Flip program on your PC. Push and hold down the PSEN button. While holding down the PSEN switch, push and release the reset button. Finally, release the PSEN button. This process of having the system reset while the PSEN button is pushed will cause the chip to run the boot loader (which is built in).

Note: If you do not have Flip on your computer you can download it from:

http://www.atmel.com/dyn/products/tools_card.asp?tool_id=3886&category_id=154&family_id=690&subfamily_id=1561

7. In Flip, use the device menu to select the AT89C51CC03 device. Use the file menu to find and load the hex file from step 5. Click on *Settings*→*Communications*→*RS232*. On the RS-232 setup menu you should find a new COM port. This will be a *virtual com* port that allows you to send serial data over a USB connection. In general it is the highest number COM port available. (On my machine it's COM5.) Select the virtual COM port and select the baud rate at 19,200. Click on *Connect*. If you get a time out error, try again resetting while the PSEN button is pushed and click on *Connect* once more. If this fails, you could have a bad cable or a board error. Try another cable and if you can't resolve this error see your instructor or Jeff Cron.
8. After you have successfully connected to the chip, verify that your Flip main screen looks like that shown in Figure 4. In particular be sure that the BLJB bit is *not checked*. This bit causes the system to always run the boot loader so if it is checked you can upload a program but you can't run it. Click on *Run* to program the chip.
9. Push reset after the chip has been programmed. The test program will run and you will be able to see a ramp function on pins 13 and 14 of CN1. You will also see the bits on the other ports toggle at various frequencies.

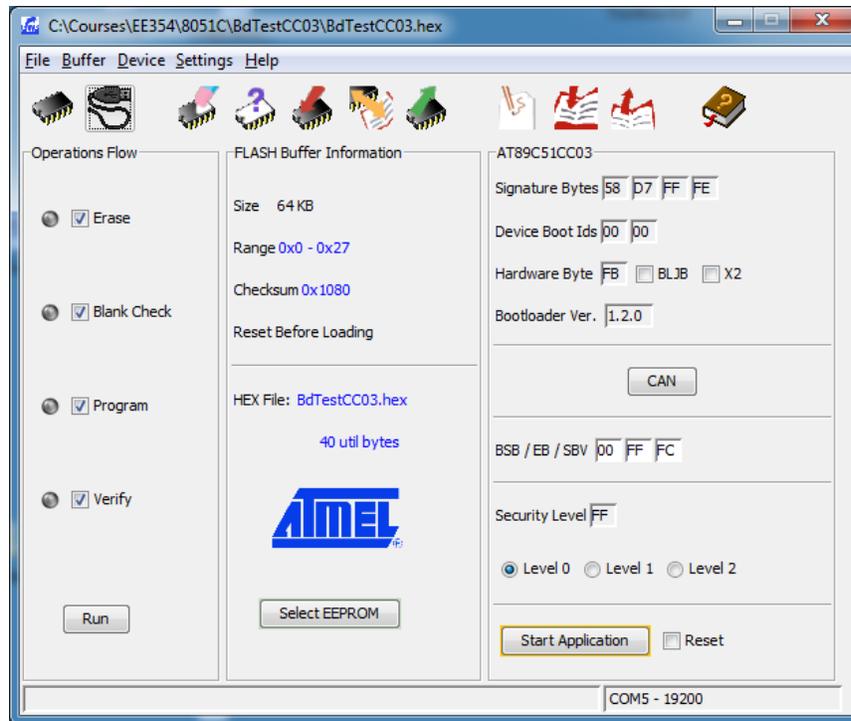
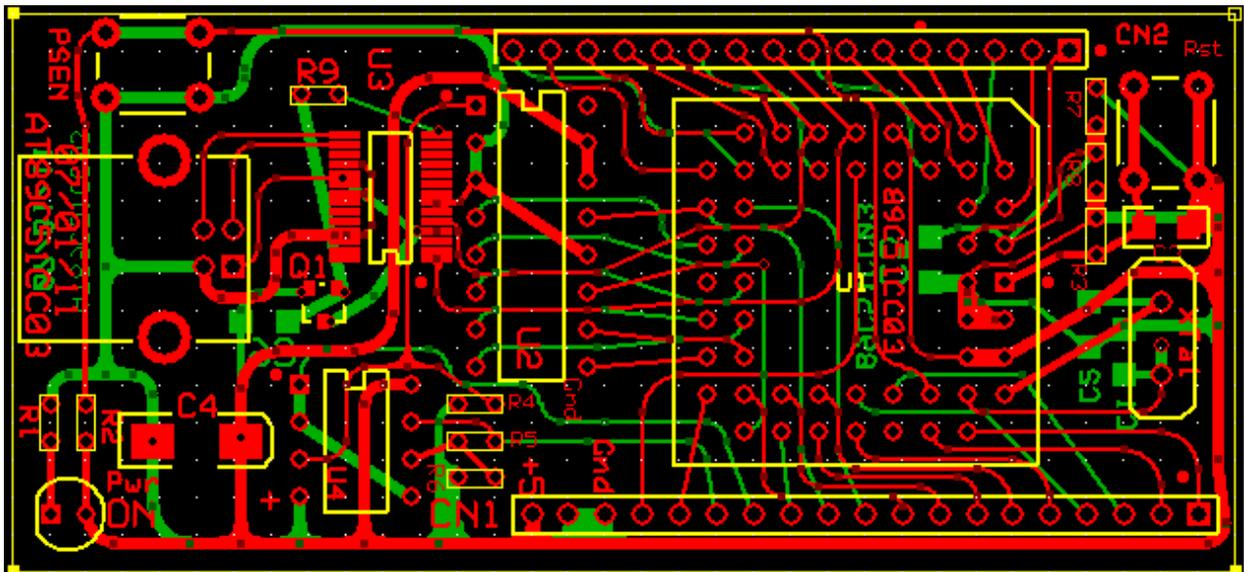


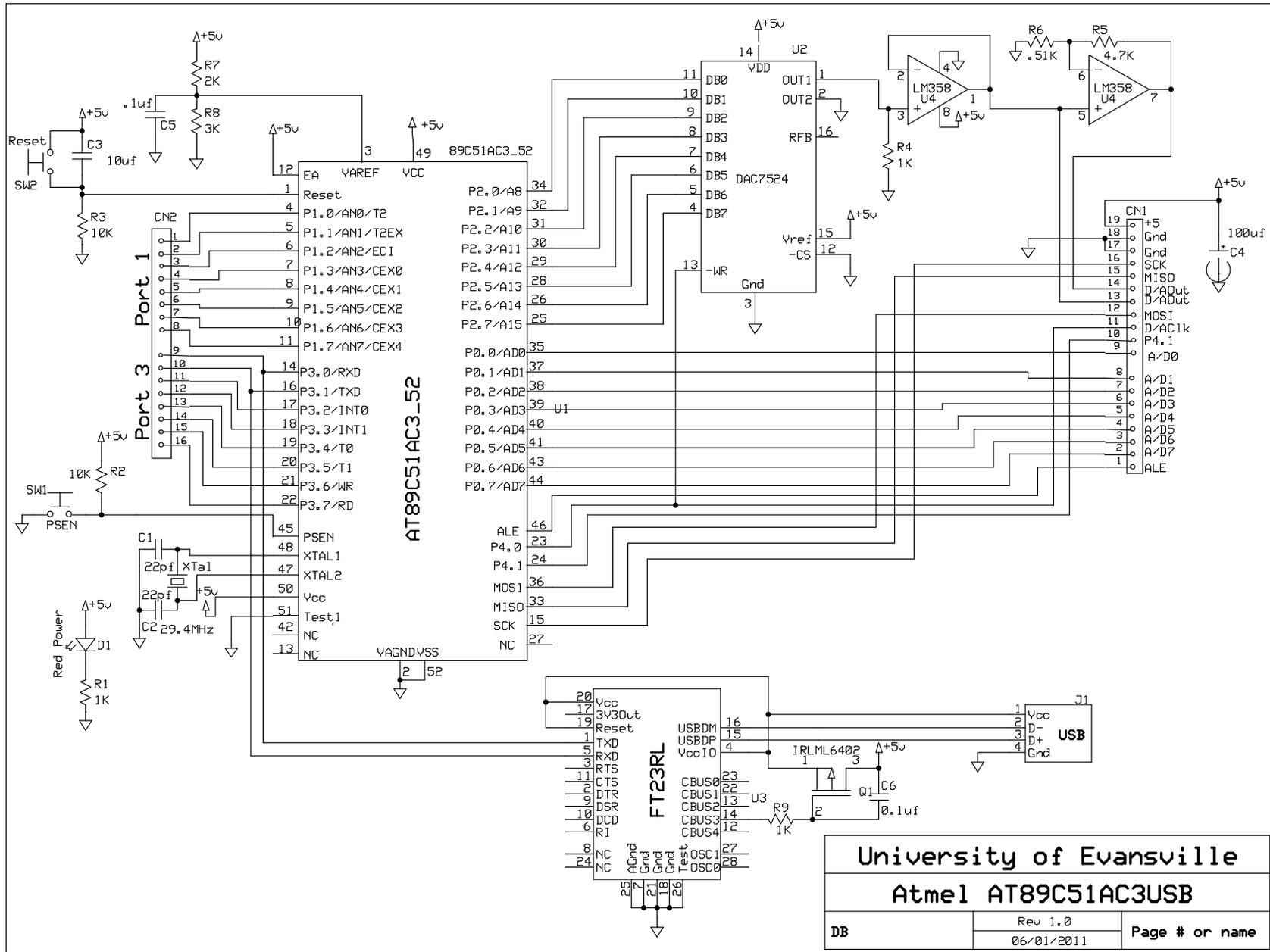
Figure 4

The Flip set up prior to programming.



AT89C51CC03 Board Layout.

Red is the top layer and green is the bottom layer.



AT89C51ACC03 Schematic

Revised September 9, 2012

