Building a USB Joystick Controller
Using the PIC18F2455 MCU

Project   : DIY USB controller for games on PIC18F2455

Publisher   : Willy

English Translation   : Dr. Andy Amer

MAIN FEATURES   :
- 6 axis 10 bit (1024 steps)
- 32 buttons
- Hat witch with precision (0 to 315°)

SPECIAL FEATURES :
- **Incorporated Bootloader**
- Easily extended with with small extra auxiliary cards.

Note:
This tutorial is only intended as a guide to construct the card without going into the complicated USB connection issues, HID devices, or programming of micro-controllers.
Only a basic understanding of electronics, soldering components and the construction of printed circuit boards (PCB) are needed.

DISCLAIMER

THIS DOCUMENTATION IS PROVIDED FOR THE PURPOSE TO BE USEFUL AID FOR IMPLEMENTING THE ABOVE MENTIONED CARD FOR YOUR OWN APPLICATION. IT IS MEANT TO BE USED WHILE COMPLYING WITH ALL SAFETY AND SENSITIVE ELECTRONICS HANDLING MEASURES. WE CAN NOT BE HELD RESPONSIBLE FOR ANY DAMAGE OR INJURY OF ANY KIND WHICH MAY OCCUR OR COULD OCCUR EITHER DURING BUILDING, TESTING, USE OR STORAGE. USE THIS MATERIAL AT YOUR OWN RISK AND JUDGEMENT.

if you have any problems or inquiries during construction process, testing, normal running operations and future modifications and/or expansion, you can post to the appropriate thread at XtremeRacers forum. Here is the link :
 http://www.xtremeracers.info/forums/viewtopic.php?f=45&t=5696

The project relies heavily on changing the sample included in the Mouse V2.4 Framework board from Microchip PICDEM FS USB development.

My special thanks to _**Rawin Rojvanit**_ without their contribution this project would not have been possible.

The MCHPFSUSB can be downloaded from:
MCHPFSUSB Framework v2.4
http://www.microchip.com/stellent/idcplg?IdcService=SS_GET_PAGE&nodeId=2651&param=en53444
94

Schematic Design
I worked from Figure A-2 found in the documentation "PICDEM FS USB User
51526a.pdf Guide "included in the Framework.

The basic changes are:
I've changed the mode for entering the bootloader in order to facilitate the availability of pin 25
of the PIC (RB4 – AN11). Also the switch S2 now connects to the pin 6 (RA4).

FIGURE A-2: **BOARD SCHEMATIC, PART 1 (MICROCONTROLLER, VOLTAGE REGULATION AND ASSOCIATED PARTS)**

Component list:

1	PIC18F2455 Microcontroller 1 (PIC18F2550) (IC1)
1	28 Pin integrated Soket, failing one 20 and one 8
1	Crystal 20 MHz (Y1)
2	Switch for PCB (S1, S2)
1	LED (D2)
2	Resistors 10Kohm (R1, R2)
2	Resistors 470ohm (R2, R4) can be 1Kohm
1	Resistors 1Kohm (R5)
1	Resistors 1Mohm (R6)
1	Electrolytic Capacitor 470nF, 16V (C6)
1	Electrolytic Capacitor 10uF, 16V (C5)
2	Ceramic Capacitors 22pF (C2, C3)
2	Ceramic Capacitors 0.1uF (104) (C1, C4)
2	Diodes 1N5817, 1N4001, 1N4148 (D1, D39) 1N4148 are convenient for the small size
1	Cable USB to connect to the PC
1	Strip of 40 pins
1	Strip of 40 pins female connectors
2	Strip of 20 cover for connectors. (jumpers)

PCB Construction

The PCB is designed for a single layer to be simple and we can accommodate all the components on the same layer, hence no need for double sided design. Also space is not critical.

Moreover, being a DIY version, it is convenient to use a basic technique to make it easy to construct. Anyway, I have not started yet to construct a double-sided PCB.
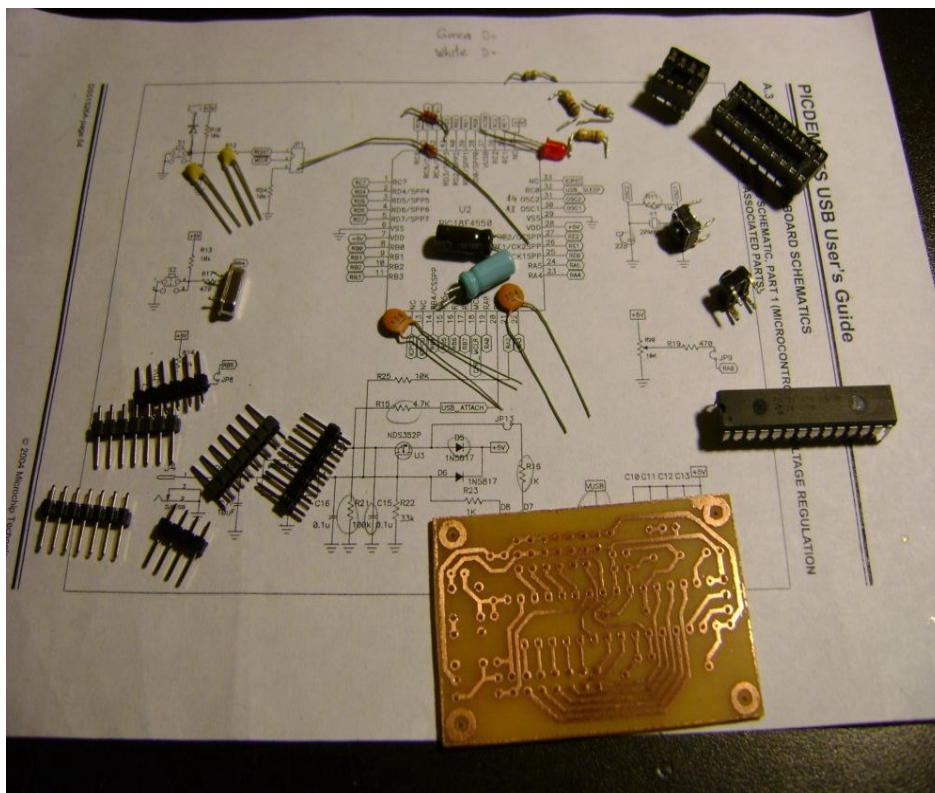
The method used for construction is to use a household iron to transfer the circuit from the printed paper to a copper plated PCB.

You will find a dedicated forum for supporting the contraction of the PCB at XtremeRacers, at the following link :

http://www.xtremeracers.info/forums/viewtopic.phpf=45&t=5110&sid=6a98edec78b3f1885a09d88f54eb7a36
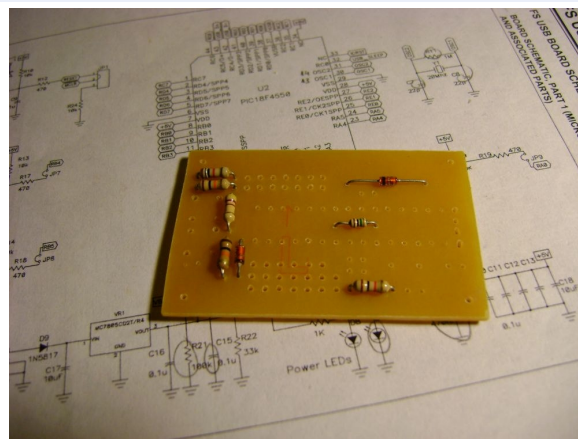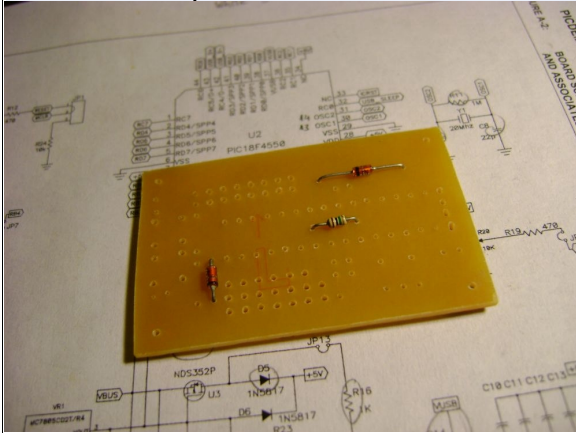
Here the PCB design with dimensions in millimeters so that you can scale them.
Download Link CAD file for printing "plate 18F2455 LT97.dwg"
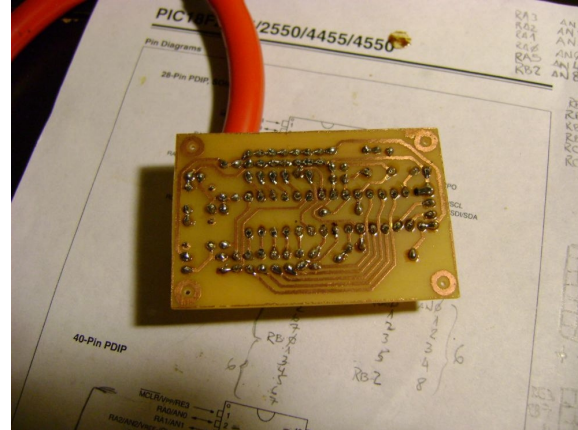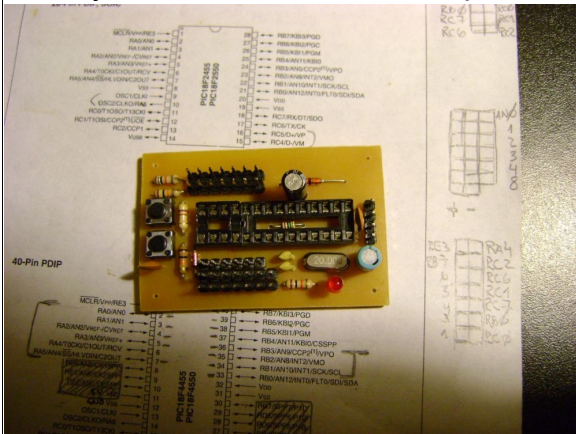http://uploading.com/files/O04HQ4KE/Placa 18F2455 LT97.dwg.html

PCB Assembly

The first components soldiered are the smallest size, which are the diodes and resistors.

Once you have soldered all components, It should look more or less like this:

Before you insert the PIC and connect the PCB to your PC we need to check and ensure that what we did is 100% correct.

CAUTION:
If the you reverse powered the PIC MCU, or the voltage is outside the specified values, the Chip will burn, and it will be useless to use.
Another point to consider is the power from the USB port of your PC. The USB ports are capable to deliver a currents of up to 100mA, so a reverse polarity connection probably consumes more current, so you would short-circuit the PCB.
Under these conditions it is more likely, the USB port will not communicate, and hopefully you do not burn anything else.

We now proceed to test the Board

With a proven 5V DC power supply, we'll check the connections to the corresponding power pins for the PIC18F2455. We'll use a multimeter to carry out the test.
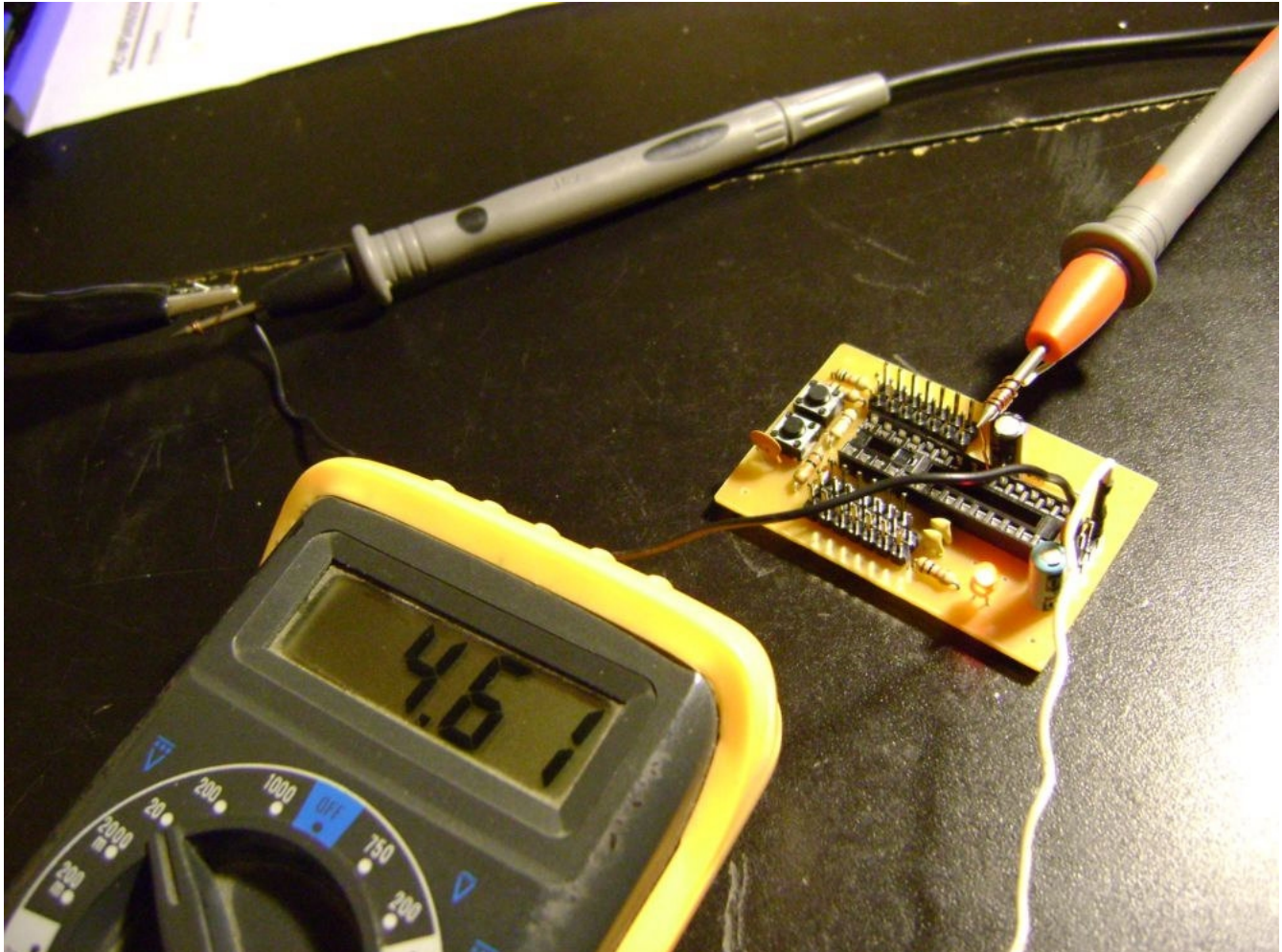When we connect the power, the LED should light indicating that power is present.

The pins in questions are :
Pin 8 & 19 – Vss
Pin 20 – Vdd

With the negative terminal of the multimeter (Black) connected to a negative Power point on the card, we start testing the voltage with the red terminal at the above Pins. This is before we insert the PIC18F2455 or connect the PCB to the PC.

Then we can check the continuity between the different pin socket for the PIC and pins for the analog connections and the rows and columns for the keypad.

Also we can test whether the S1 and S2 buttons are working well. With the multimeter positive terminal placed on pin 1 of the socket, it should indicate about 5V, and when you press the S1 the voltage should decreases to 0V. The same goes for pin 6 and S2.

Getting PIC18F2455 Bootloader
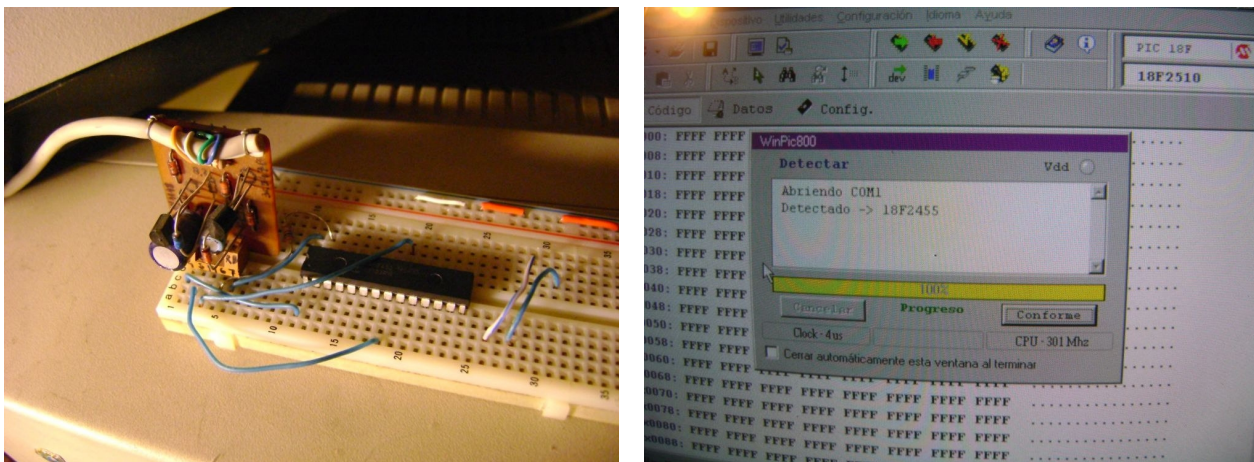Locate and download "MCHPUSB_Bootloader.HEX" from the following link :

http://uploading.com/files/YHIMECEX/MCHPUSB_Bootloader.hex.html

With any PIC programmer you must load the bootloader to the PIC18F2455 MCU.
I use a basic JDM programmer, which is very easy to build.

Another important tool is the software that will use to program the PIC. I use the WinPic800,
which is very useful. It is a complete package and regularly updated with a support services
for almost all the PIC micros.

The program will run on any low end and old PC (Pentium II) with a serial port and Win98.
To use the program in a current PC with Vista or XP, I have to do some odd jobs to load some
additional drivers, but this is beyond the scope of this tutorial.

First we should ensure that WinPic800 and the programmer are working OK, this is achieved
by running the tool and pressing Ctrl + D to detect the PIC18F2455 MCU.

Then we'll proceed with copying the HEX bootloader to the CIP.

**We have to check the USB cable wiring connections, so do not be tempted to plug the
board to the PC just yet.**

If you use a female USB connector mounting on the board instead of the four pins (J1), it is
convenient to use a cable with male connectors at both ends.

For those who opted for the strip of pins, you can buy a cable with male connectors, cut one
end to expose the wires to identify the cable's wires.

GND   : Black
D+   : Green
D-   : White
VBUS : Red

We set up the connector with the according to the PCB pin arrangements then connect it to
the board.

First time connection.

When you connect the PCB for the first time it will not be recognized by the PC, as the Bootloader should recognizes the card when the S2 switch is pressed.
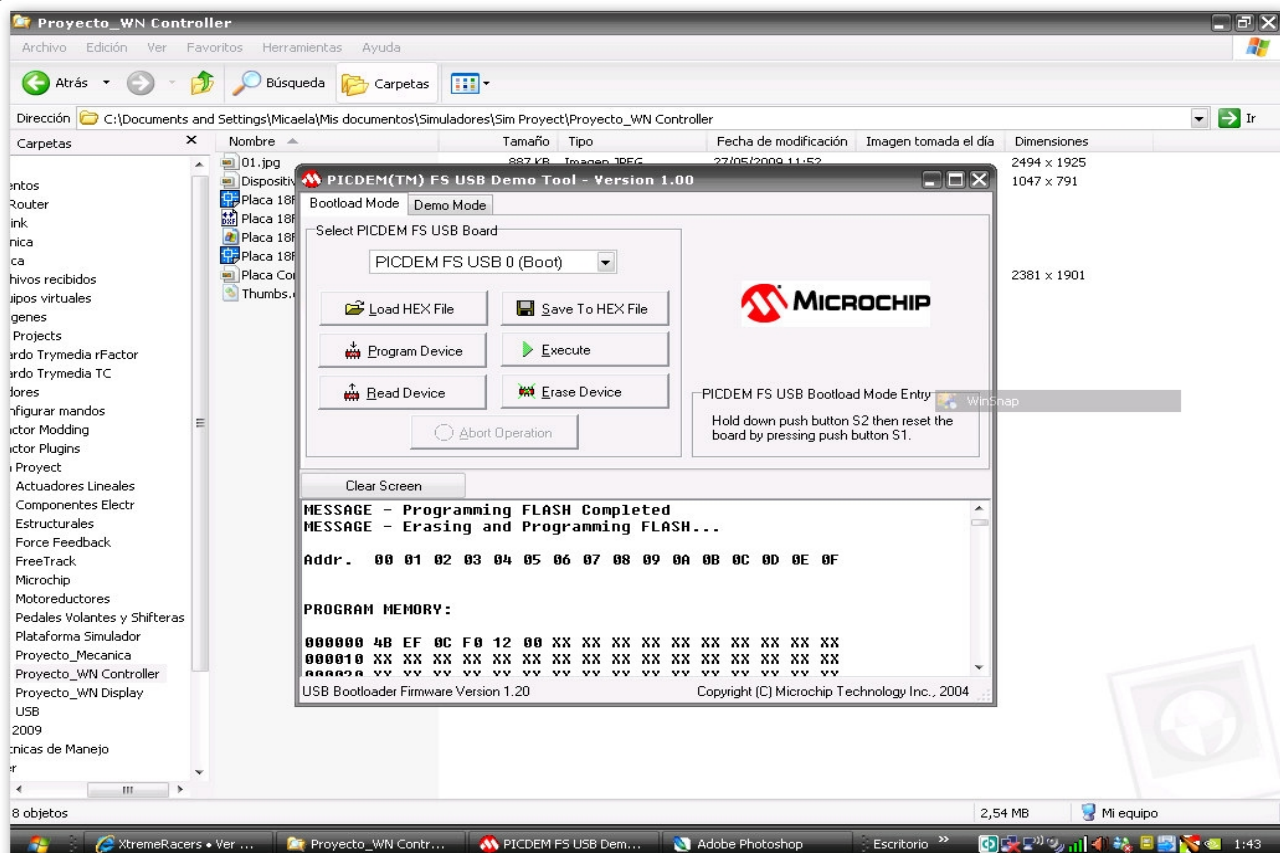
To continue we must first install the MCHPFSUSB Framework on the root drive of the connected PC.

It is highly recommended to read the "User Guide" of the PICDEM located in the documentation section of MCHPFSUSB software using PICDEM FS USB Demo Tool. The guide describes the installation of the drivers in details. You must read it in order to avoid any confusion.

Holding down S2 switch press the S1 switch (reset) once while keeping S2 pressed.

If everything went well, you should hear a beep sound from the your PC speakers as it detects the newly attached device.

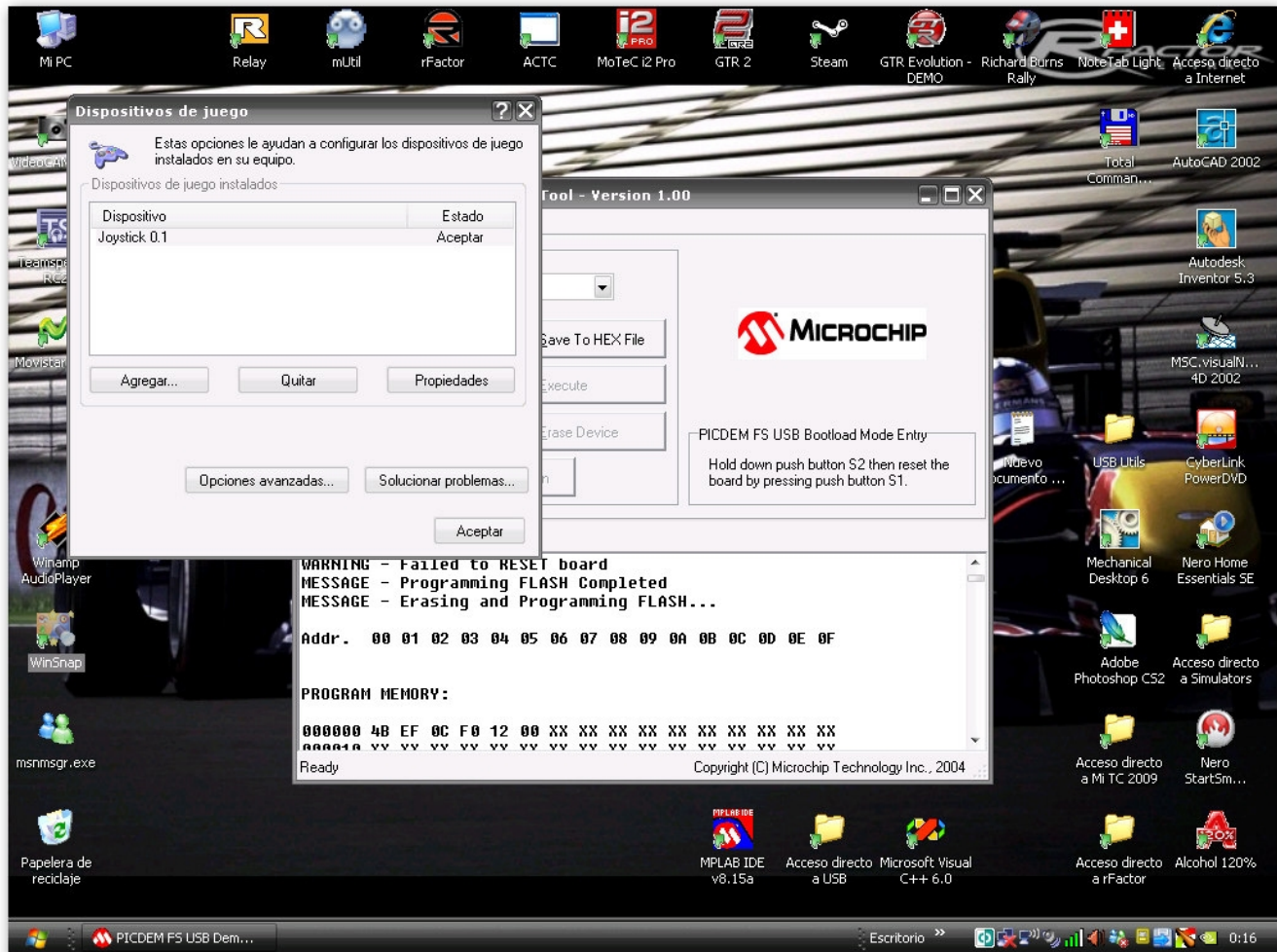Now run the PICDEM demo tool to start programming the PIC MCU as per the following picture.

Then load the. HEX our Joystick 0.1 by clicking "Load HEX File" .

The download for "MCHPUSB_Joystick 01.HEX" could be found here :
http://uploading.com/files/GRL238C2/MCHPUSB_Joystick 01.hex.html

Now click on "Program Device" and wait until the process ends, then press "Execute". This performs a soft reset to the MCU, just as if we pressed the S2 switch, the application will for the joystick 0.1 will start to run.

Go to control panel and click on the Game Controllers icon. In properties tab you should see our Joystick 0.1 working perfectly.

If the potentiometers are not connected or the button's pin array, it is likely to have erroneous readings picked up by the noisy behavior. This is because the micro legs should not be left with no connections.
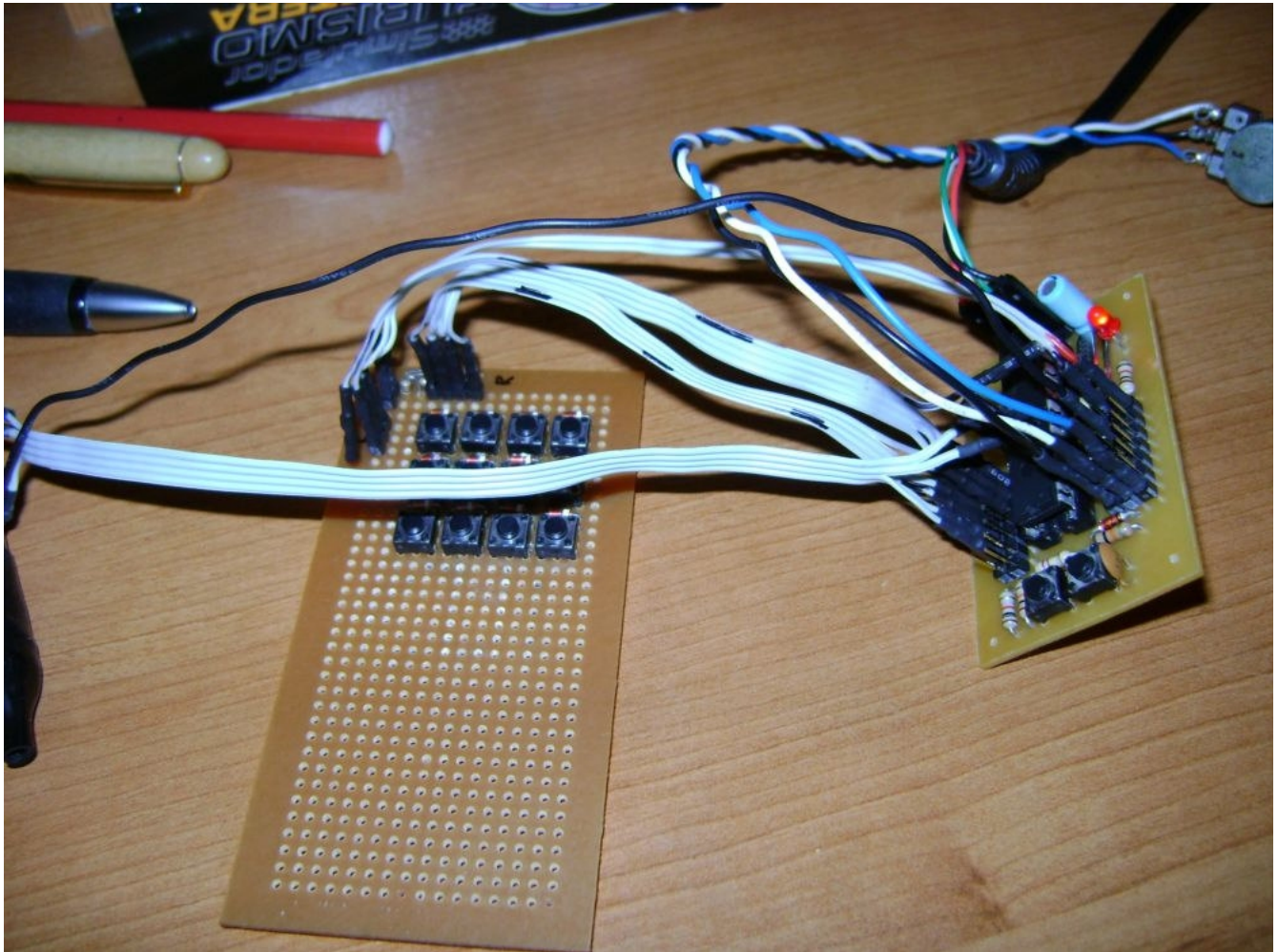
If you are not using all analog or the buttons array, for example the three X, Y, Z axes and the 9 buttons, you would be getting a 0V value at the Rx, Ry, Rz, RC7, RB0 and RC0.

Once everything is working properly you can replace the diode D39 which was used for circuit protection by bridging it.

That is all for now. Last revised (02/06/2009).
If you have any queries you can find me at :
http://www.xtremeracers.info/forums/viewtopic.php?f=45&t=5696

Translated By : Dr. Andy Amer
2nd March 2010