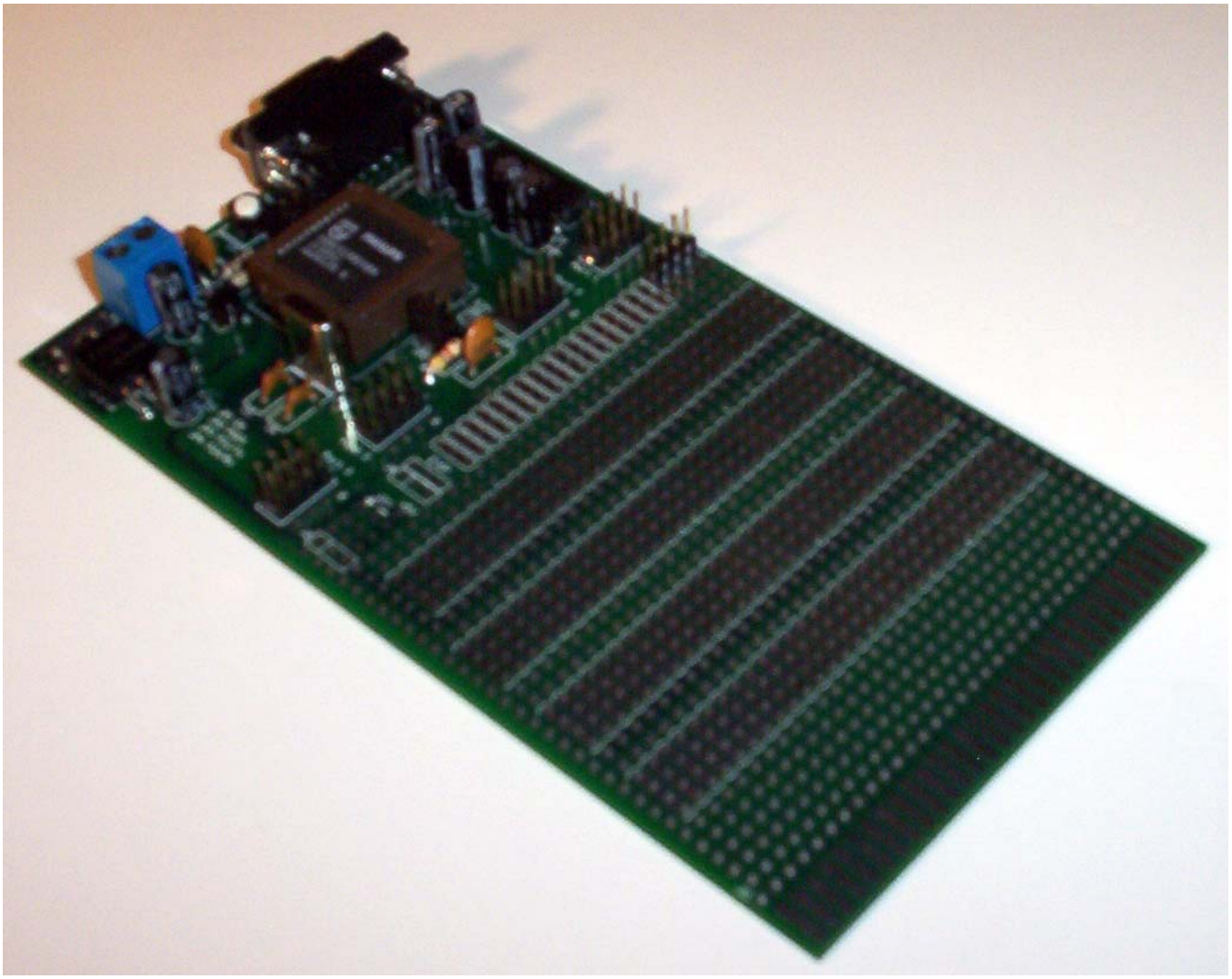
89C51PB Project Board



Avayan Electronics

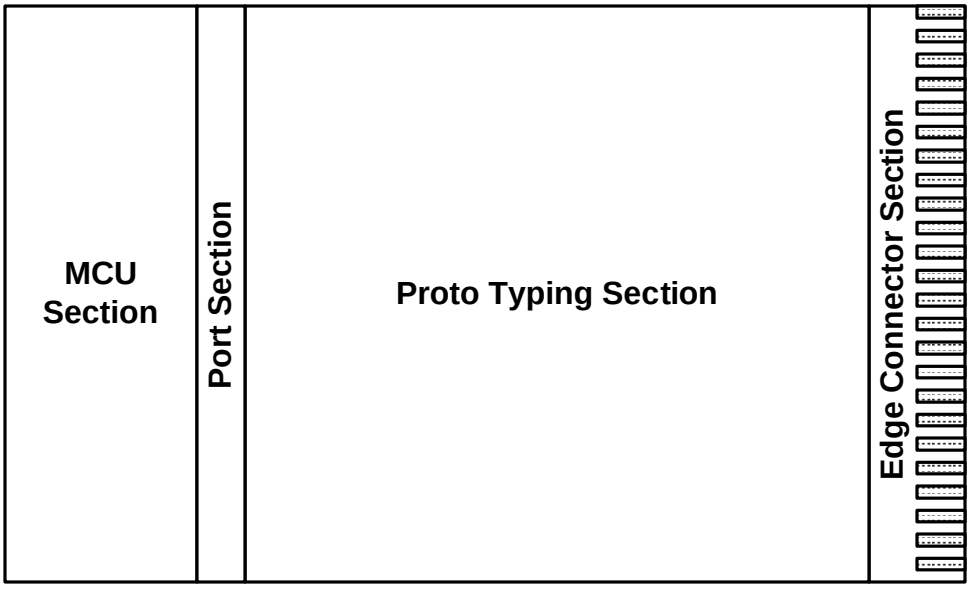
89C51FX, 89C51RX 8 bit Microcontroller Project Board

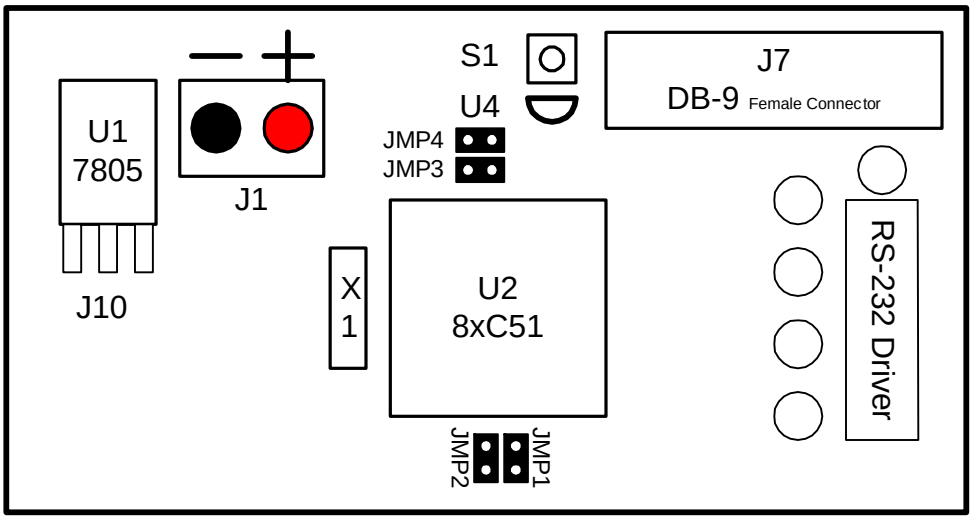
Finally, the project board our 8051 enthusiast have been waiting for. It will allow hobbyist and professionals to build their application fast whether it is soldered or wire wrapped. Compatible with most of the PLCC 44 pin devices revolving around the 8xC51 architecture. Some examples are 80C31, 80C32, 8xC51, 8xC51Fx, 8xC51Rx, 8xC51Rx+, and many others from companies such as Philips, Atmel, Dallas Semiconductor, etc.

The board comes ready to use with all the necessary support components such as an 16 MHz crystal working as external oscillator, a voltage monitor for Brown Out Detection, RS- 232 driver and female DB9 connector for serial communications and in system programmability (ISP), 5V voltage regulator good up to 1 Amp, reset button, header male connectors for all available ports (PORT0, PORT1, PORT2 and PORT3), and many jumpers to configure important signals such as EA and PSEN.

A huge prototyping area with bus connections will make concepts come to reality in a snap . To add flexibility to future designs, the bottom edge of the 89C51PB is a male edge connector perfect for adding these prototype boards to a back plane such, as the Project Board Mother Board (PBMB).

The 89C51PB board is divided into four main sections: Microcontroller Unit Section, Port Section, Prototyping section and the Edge Connector Section.





MCU Section:

This portion of the board contains the necessary circuitry to pr operly connect one PLCC-44 8xC51 microcontroller. U2 is a socket accepting a 44 pin device such as the P89C51FA, P89C51RC+, P89C51RB2H, P89C51RB2 or T89C51RD2 as examples from a wider range of available part s.

Power Connector: A two circuit high current terminal block is used to power the board with a source ranging from 6 to 24V. Pin 1 is V+ while pin 2 is V- (ground return). An LM7805 voltage regulator regulates the input voltage into a good VCC = 5V source that powers the devices for up to 1 Amp rated current. Because this board is diode protected, connecting the power backwards will not result in board damage.

Reset Manager: Some AVR microcontrollers do not have brown out detectors. To allow proper functioning, the 89C51PB includes an MC34064 voltage monitor. U4 and Q1 will make sure the reset line remains low unless enough power is being administered to the board. Also, th e S1 button will pull up the reset line generati ng a reset.

ISP Programming: For those microntrollers with ISP Flash (89C51's usually), In System programming is accomplished by connecting the microcontroller directly to the Serial Communication Port (JMP3 and JMP4 must be closed) and tying low the PSEN pin (JMP2) must be closed). Programs such as Flash Magic, readily available on the web can be used to read and program the microcontroller FLASH. It is just so easy!

Oscillator: A crystal is provided to work as system oscillator for each microcontroller. The 89C51PB usually ships with an 16 MHz crystal but other frequency resonators can be inst alled as well.

Microcontroller Section: (Cont.)

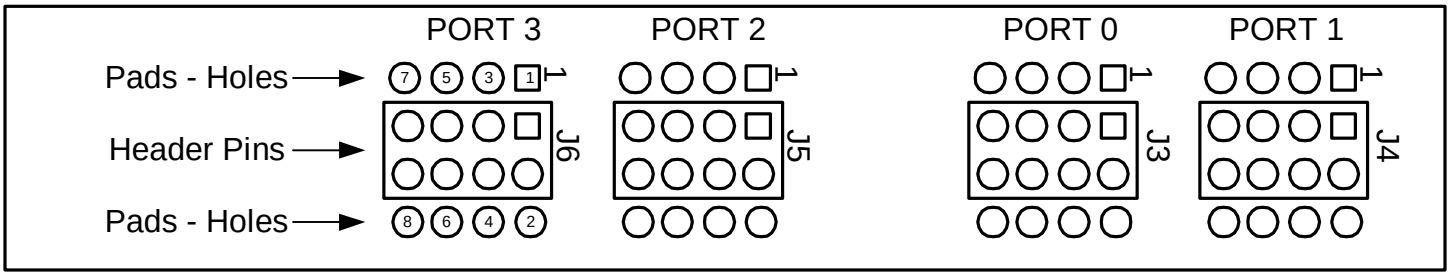
Serial Communication Port: Some projects require serial communication from the RS232 port found on conventional PC's. A female DB9 connected to the RS-232 driver allows this feature to be available. If JMP3 and JMP4 are closed, the serial communication port is connected to the microcontroller. If the application needs the Rx and Tx signals to be administered from t he edge connector, JMP3 and JMP4 should be opened.

Jumper1: This jumper either selects pin EA high or low. If jumper is open, the signal is leve l high. If closed, the signal is tied to level low.

Jumper2: This jumper selects PSEN low or floating. If jumper is open, the PSEN pin is floating and can be monitored as an output. If closed, the PSEN signal is tied low and ISP is enabled o n chips with the feature.

Jumper3: Selects Tx signal connectivity. If open, the Tx signal output from the RS-232 drive r is just connected to the J10 expandability connector. If closed, the Tx signal output from the RS-23 2 driver is connected to the J10 expandability connector and the Tx pin in the microc ontroller.

Jumper4: Selects Rx signal connectivity. If open, the Rx output signal from the RS-232 drive r is just connected to the J10 expandability connector. If closed, the Rx signal output from the RS-23 2 driver is connected to the J10 expandability connector and the Rx pin in the microc ontroller.



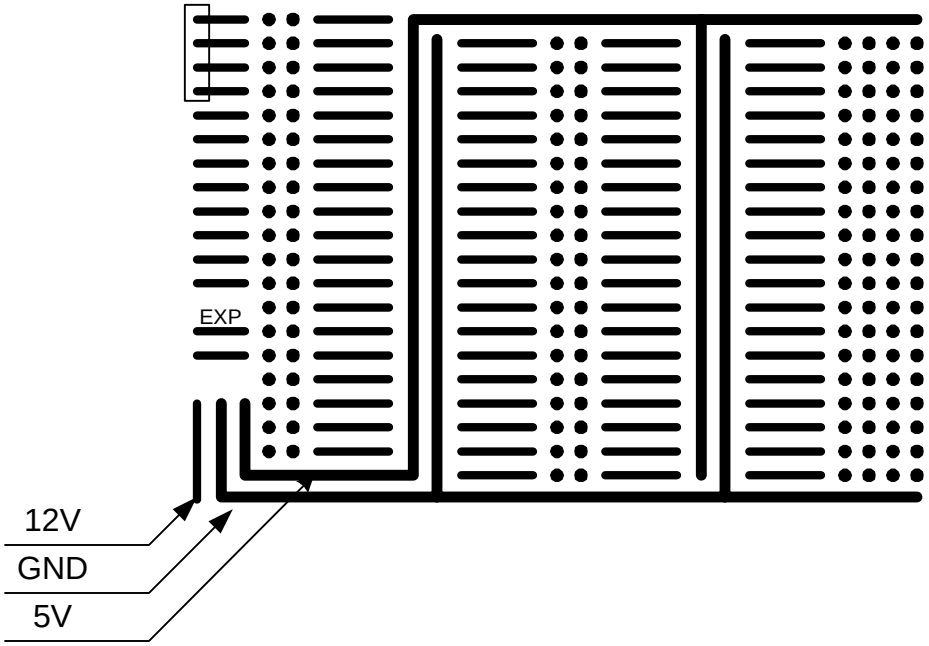
Port Section:

To ease the peripheral connection to the microcontrollers, all ports have been extended to a header pin slot as well as to a pad sector. The header connector allow for other connectors to be plugged directly into the board and also helps to do wire wrapping style of prototyping. The pads will allow to solder a connection from the port into another area such as prototyping or edge connector sections.

J6 corresponds to PORT 3, J5 to PORT 2, J3 to PORT 0 and J4 to Port 1. Each port has pin #1 as a square pad. Per example, pin 1 on PORT 0 would correspond to P0.0. Pin order has been sh own on J6. All other connectors observe the same pin order. (The board silk screen shows pin s 1 and 2 as a reference)

Prototyping Section:

Some space is allocated to add the extra circuitry that will make up the target system, in the for m of a pad grid. There are close to a 1000 pads carefully arranged in the mentioned area. A power bus runs through the prototype grid allowing easy access to VCC and GND lines. The gri d was designed to allow components such as through hole DIP IC's, resistors, capacitors, transisto rs, etc.



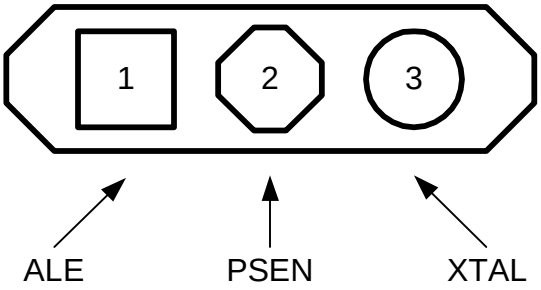
Prototype Diagram: Not exact amount of pads shown

The prototyping area has been designed with power distribution in mind. One small segment contains the voltage in (same voltage being connected at J1). Ground and VCC follows . The idea with these copper strips is to allow the user add extra 7805 voltage regulators i n case more amperage is needed. Also it adds the capability to use input voltage to power up prototypes circuitry.

Expansion Sub-Section:

Marked by EXP, an extra set of pads will allow the user to interface directly to the ALE, PS EN and XTAL lines. The idea is to allow projects to deal with these very important signals as when dealing with external memory. ALE is the rectangular pad (pad 1), PSEN is the octagonal pad (pad 2) , and XTAL is the circular pad (p ad 3).

Expansion Section: (Cont.)



Connector J10: In order to allow full use of the RS-232 two channel driver, there is a four pin connector on the top of the prototype area (J10). These four pins are as follows:

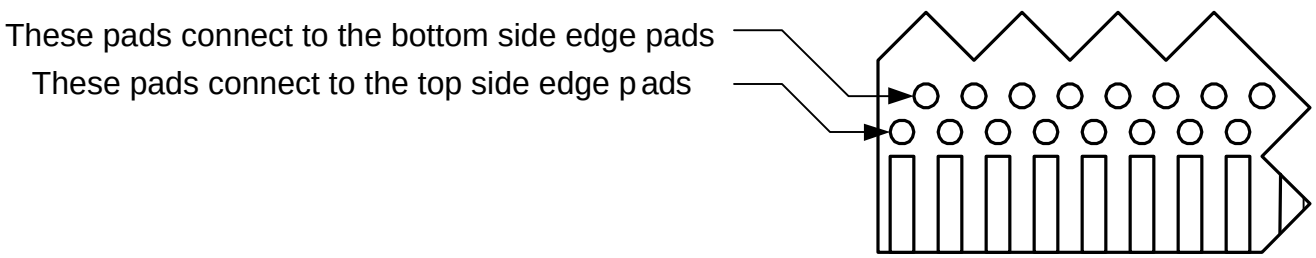
J10:1 PC Tx / MCU Rx (Data sent from the computer into the MCU)
J10:2 PC Rx / MCU Tx (Data sent from the MCU to the PC)
J10:3 PCDTR / MCU Sync (another RS-232 channel; Data from PC DTR to application)
J10:4 PCCSR / MCU ACK (Another RS-232 channel, Data from application to PC CSR)

NOTE: The names MCU Sync and MCU Ack are for reference only. They were taken from the Super Stepper architecture and by no means represent the only use for this two signals.

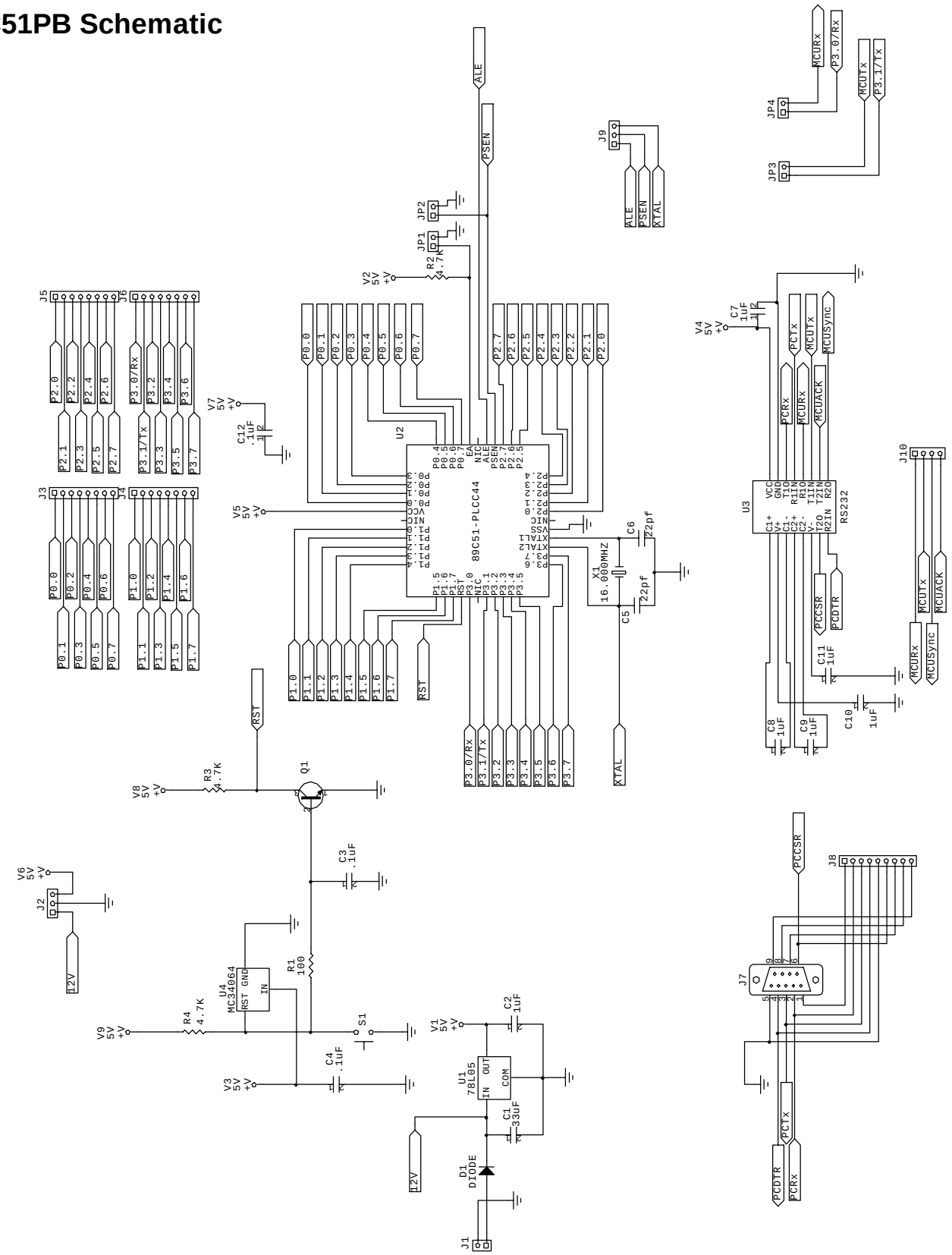
Edge Connector Section:

Expandability is not a problem with the 89C51PB thanks to the edge connector on the bottom edge of the board. Up to 62 different signals can be brought into or from the board by means of standard 62 pin, 100 mil from pin to pin edge connector. Avayan Electronics' MBPB-3 is a good example of a 3 edge connector mother board that will supply power, communication and control signals between three edge connector based boards.

Each tab on the edge connector, is connected to a pad. The user must connect input and output signals from this set of pads into any other pad located at the prototype section or the port section.



89C51PB Schematic



89C51PB Bill Of Materials (BOM):

Item	Count	Label-Value	PACKAGE	Designation
1	1	33uF	RB.1/.2	C1
2	6	1uF	RB.1/.2	C2,C7,C8,C9,C10,C11
3	3	.1uF	RAD0.2	C3,C4,C12
4	2	22pf	RAD0.2	C5,C6
5	1	DIODE	DIODE0.4	D1
6	1	CONN	PWR2	J1
7	4	CONN	IDC8	J3,J4,J5,J6
8	1	CONN	DB9/F	J7
9	1	CONN	SIP9	J8
10	1	CONN	SIP3	J9
11	1	CONN	SIP4	J10
12	4	CONN	SIP2	JP1,JP2,JP3,JP4
13	1	2N3904	TO-92B	Q1
14	1	100	AXIAL0.4	R1
15	3	4.7K	AXIAL0.4	R2,R3,R4
16	1	Switch	LTS_Pan_2C	S1
17	1	78L05	TO-220	U1
18	1	Socket	PLCC44/T950	U2
19	1	MAX232	DIP16	U3
20	1	MC34064	TO-92C	U4
21	1	16.000MHZ	RAD0.2	X1

89C51PB Kit Assembly Instructions:

Thank you for purchasing an Avayan Electronics' Evaluation/Project Board. We hope the tools satisfy you into making easy developing ideas. Please follow the next instructions when assembling your kit:

What you will need:
1. Soldering iron with fine point. 30W recommended.
2. Solder tin
3. (Optional) Board Vise

1. Make sure your kit contains all the components as specified by the BOM sheet.

2. Place and solder all thru hole components.

SOLDERING HINT: Place all thru hole components first and then from the board bottom, solder one of the component pins. Once every component has at least a soldered pin you can turn the board into the other side making it easier to solder the remaining pins (extremely helpful when soldering more than two pin parts). If it is hard to position the board in such a way that both bottom and top are accessible at the same time, solder each component as you go through the following steps.

a. Position the 33 uf cap into C1. Please observe polarity. The plus sign on the silk screen should match the plus sign on the capacitor.
b. Position the 1 uf cap into C2 observing propper polarity.
c. Position .1 uf ceramic capacitors on C3, C4, and C12.
d. Position the two 22 uf ceramic capacitors on C5 and C6.
e. Position the two circuit Terminal Block (blue plastic connector) into J1.
f. If you desire to have access to the 12V, 5V and Ground plane through test stakes, place the three circuit header connector on J2. This connector space is actually designed to add an optional 7805 voltage regulator in case the 5V plane needs more than 1A current.
g. Position the eight circuit dual row header connector on J3 (PORT0), J4 (PORT1), J5 (PORT2), and J6 (PORT3).
h. Position the DB9 female connector on J7. Place the 9 circuit header connector (SIP 9) on J8 if desired to have access to the RS-232 signals.
i. Position the four circuit header connector on J10.
j. Position the four two circuit header connectors labeled JMP1, JMP2, JMP3 and JMP4
k. Position the 2N3904 Transistor on Q1.
l. Position the 100 ohm resistor on R1. This resistor is color coded Brown-Black- Brown-Gold.
m. Place the three 4.7K resistors on R2, R3 and R4. These resistors are color coded Yellow-Purple-Red-Gold.

89C51PB Kit Assembly Instructions (Cont.):

n. Position the switch on S1. You may need to trim the leads before inserting the pin into the holes.

o. Position the LM7805 voltage regulator into U1. Bend the leads so that the regulator back lies at the heat sink square pad. At the end of the assembly it would be a good idea to solder the back of the regulator on this pad to improve heat dissipation. Have patience as soldering such a large pad takes time.

p. Position the 44 pin PLCC socket on U2.

q. Position the Reset Manager MC34064 on U4. Notice that the oval shape must correspond to that on the silk screen or else the part will be reversed polarized.

r. Position the 16.00 MHz (or any other desired for your particular project) on X1.

s. Position the 1n4003 diode on D1.

t. Position the RS-232 driver chip on U3. Please observe correct polarity.

u. Position the five 1 uf capacitors on C7, C8, C9, C10 and C11.