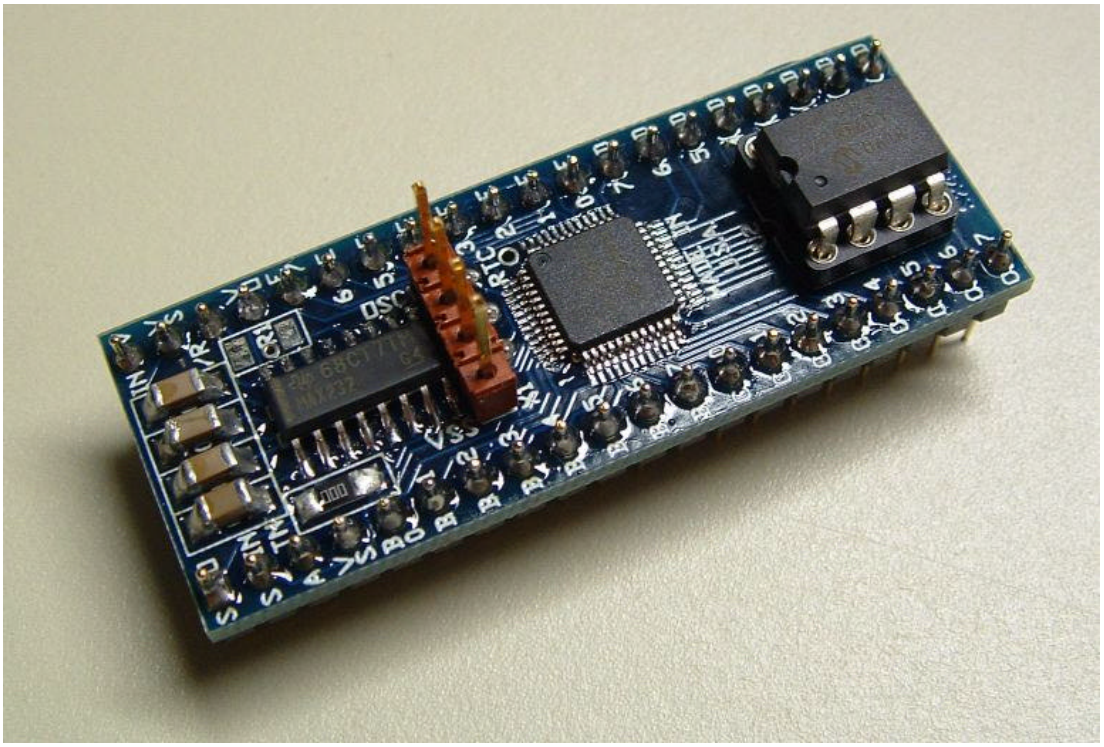
SX48
OEM Module

Surface mount/through hole kit



www.robotworkshop.com
"Our robots will blow you away"™

Manual Revision 1.0

NOTE: This kit provides a convenient method for using the SX48BD on a breadboard. It allows you to build a fully functional module using the SX48BD microcontroller. The finished module has a pinout that is compatible with the BS2p40 Stamp and in most instances can be used as an upgrade where more performance is required. This new module is programmed in either Assembly (SASM) or SX/B. We are not affiliated with Parallax but offer this kit that is complementary to their offering. The BASIC Stamp™ and the BS2p40 and Interpreter chip are trademarks of Parallax.

Table of contents

INTRODUCTION

The SX48 is an extremely useful little microcontroller that is robust and easy to use in a variety of projects. It has on-chip EE/Flash program memory and in system programming with on-chip debug. For those projects that need extra I/O the SX48 OEM module is an excellent choice. It offers 32 I/O pins, and on board RS-232 serial port, and a socket for an optional EEPROM. Although Parallax does offer a nice SX48 proto board (which is great for a lot of projects) there was still a need for a smaller version. This module helps fill that need and is perfect for those that want to use the SX48 on a breadboard or as an upgrade for the BS2p40 BASIC Stamp module.

Some people have tended to shy away from using surface mount chips like the SX48DB/TQ. Since there haven't been too many kits available using surface mount parts there hasn't been the inspiration to start using them. This kit fills that void for those few who would like to try building a surface mount kit and would like learn more about these great microcontroller chips. This kit has been designed with a mix of surface mount and through-hole components. The surface mount parts were selected so that they can be soldered with traditional hobbyist soldering equipment. No special equipment is required. Well, other than a magnifying glass and a good set of tweezers…

This module when finished should work in a regular 40-pin socket and has a pin out that is compatible with the BS2p40 module. There are however, a few things to note:

Disadvantages:

- Programming this module takes a little more effort than those familiar with the regular BASIC Stamp modules. It also requires an SX-Key/SX-Blitz device for programming instead of just a regular serial connection.
- This module is only available in kit form. It has to be built before using it. Although it doesn't require or assume previous surface mount experience anyone attempting this does need good soldering skills and good vision to successfully assemble it.

Advantages:

- Upgrade path for users of the BS2p40 module. This SX48 OEM module is programmed in SX/B (BASIC compiler) or directly in assembly for high performance. Access to interrupts enables more control and advanced programs.
- This module is only available in kit form and has to be built before using it. You get to learn some new surface mount soldering techniques!
- This module uses a MAX232 for the serial interface. It provides more of a true RS-232 standard signal and will theoretically be compatible with more serial devices than those that use discrete components to interface the signals.
- Your program is much more secure since it is stored directly inside the SX48BD microcontroller and cannot be read like an external EEPROM chip.
- The DIP form factor for the EEPROM enables it to be swapped out. You can download different data into different EEPROM chips. Just change data by swapping EEPROM chips instead of reprogramming. This could lead to experimentation with bank switching, etc.

NOTE: This kit requires excellent soldering skills to successfully complete. If you are unsure about building this kit it is suggested that you find someone that can assist with assembly, or purchase an assembled SX48 Proto board from Parallax.

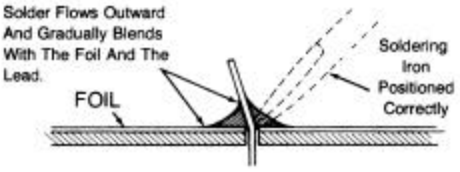
Overall this can be a rewarding kit to build and I hope that those attempting to build it will enjoy the process and perhaps learn a few new techniques. Once you start getting used to using the surface mount parts they are not that hard to work with.

CIRCUIT ASSEMBLY

This kit should be built using standard construction methods. The following items are required to build the kit: diagonal cutter, needle nose pliers, tweezers, soldering iron (pencil type) with very fine tip, a good quality solder (60/40 Rosin core, RoHS, etc.), solder wick, liquid solder flux (optional), a magnifying glass, and some patience. Follow the instructions carefully and read the entire step before performing each operation. Do not rush. To successfully assemble this kit you must have good soldering skills. A good solder connection will form the electrical connection between two parts, such as a component lead and a circuit board foil. Care also needs to be taken to ensure that there are no solder bridges causing shorts. A bad solder connection could prevent an otherwise well-assembled kit from operating properly.

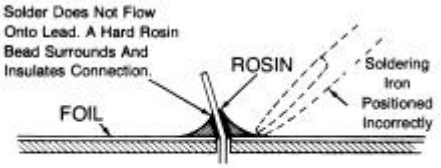
During assembly make sure you keep the soldering iron tip clean. Wipe it often on a wet sponge or cloth; then apply solder to the tip to give the entire tip a wet look. This process is called tinning, and it will protect the tip and enable you to make good connections. When the solder tends to “ball” or does not stick to the tip, the tip needs to be cleaned and retinned.

A GOOD SOLDER CONNECTION

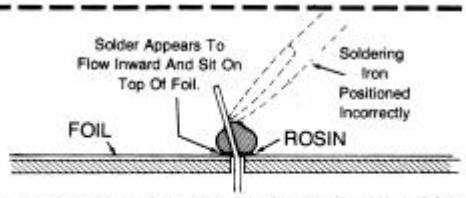


When you heat the lead and the circuit board foil at the same time, the solder will flow evenly onto the lead and the foil. The solder will make a good electrical connection between the lead and the foil.

POOR SOLDER CONNECTIONS



When the lead is not heated sufficiently, the solder will not flow onto the lead as shown above. To correct, reheat the connection and, if necessary, apply a small amount of additional solder to obtain a good connection.



When the foil is not heated sufficiently the solder will blob on the circuit board as shown above. To correct, reheat the connection and, if necessary, apply a small amount of additional solder to obtain a good connection.

The top side of the board says “SX48 OEM” on the right edge. During construction, components will be mounted on both sides of the board. The components soldered to the top of the board will be installed first. After installing each component at the specified location, solder it in place before proceeding to the next. When installing the DIP socket solder each of the two opposing corners first. Then gently press on the center of the socket while warming the solder on each of those corner pins with the soldering iron to make sure the socket is properly seated. Finally solder the rest of the leads on the socket. This method gives the board a much cleaner look than if the leads on the sockets have been folded over to hold it in place before soldering.

NOTE: The assembly instructions have been sequenced so that the installation of each component will not block the next one to be installed.

One of the most important aspects of assembly is the proper alignment of the components. Some people prefer to use a small dot of glue to hold each component in place while soldering. If this works out well for you, by all means use that method. Otherwise the following techniques give excellent results!

Before installing a surface mount component apply a small amount of solder to one of the pads on the PCB where the part is going to be installed. Then while warming the solder on that pad use the tweezers to set the part in place. You can do adjustments while the solder is still molten. If it takes too long, let the solder cool to ensure the part is not damaged by excessive heat. Once cool the joint can be warmed up again and the alignment can be adjusted until it is perfect. Before soldering any other joints on the part use the magnifying glass to verify that the alignment is ok.

Components installed on the top of the PCB:

- ❑ Install the Parallax SX48BD/TQ CPU *near the center of the PCB* at location **U1**. Pin one of the CPU should be orientated toward the upper left of the board. There is a small mark on the silkscreen, which shows the location. The alignment of this part is critical. First, tin the iron and tack the CPU in place. Do not worry about solder bridges at the moment. Once it is aligned solder one of the pins on the opposite corner. Double check the alignment with the magnifying glass. If all is well proceed to solder the rest of the pins. Since the pitch is so close on these pins just solder each side without worrying about solder bridges and let it cool before doing the next. After the chip cools go back over each side with solder wick to remove the excess solder. This will remove most of the solder and leave just enough to make a proper connection. Repeat the process if solder bridges still exist. Be sure to let the chip cool between each side so the microcontroller will not be damaged by excessive heat. The results can be fantastic if you take your time.

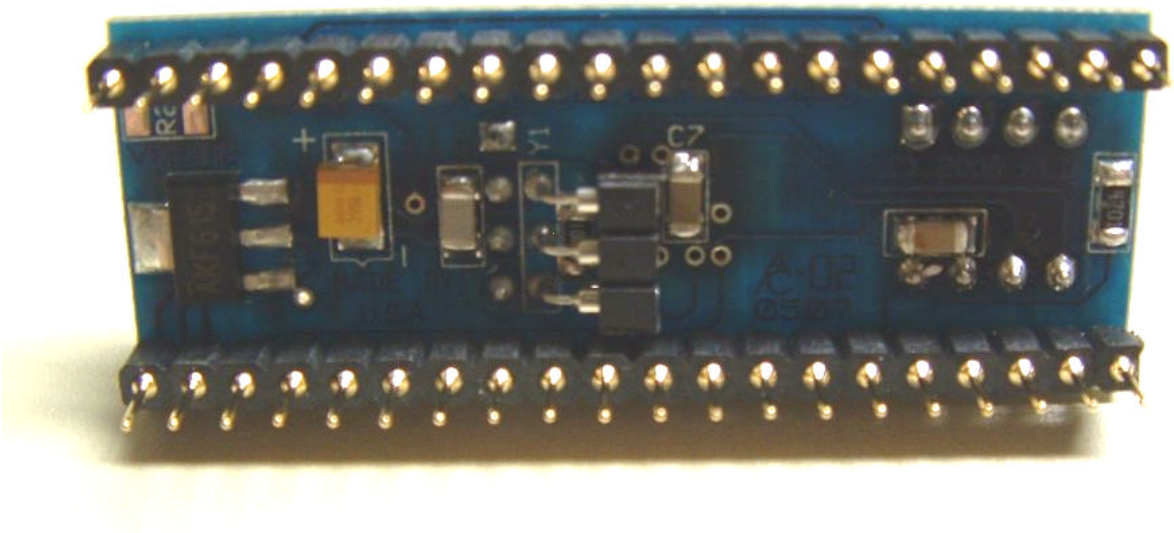
- ❑ The RTCC pin is brought out to a small via (labeled RTC) on the top of the board. Normally you do not need to use this pin then you should create a small solder bridge to ground pin next to it. These would be pins 47 and 48. This would be the only intentional solder bridge on the module.

- ❑ Install the MAX232D *near the left side of the board* at location **U3**. Pin one of MAX232D should be orientated toward the lower left of the board. The alignment of this part is critical. If the soldering iron is tinned it can warm one lead on the part to tack it in place. Once it is aligned the rest of the pins can be soldered.

- ❑ Install the four 1.0µf (105) 1206 surface mount caps *near the left edge of the board* at locations **C1**, **C2**, **C3**, and **C4**. These can be installed in either direction since polarity does not matter.

- ❑ Install the zero Ω (000) 1206 jumper *just below the MAX232D, near the left of the board* at location **D1**. This is the default build. In an alternate version a small DL4148 diode can be used with the band toward the left going to pin 1 of the module.

- ❑ Install the 8-pin DIP socket at location **U2**. The notch should point toward the left of the board.



Components installed on the bottom side of the PCB:

- ❑ Install the 5V (or 3.3v) ST-223 Voltage regulator at location **U4**. The large single tab goes toward the left near the edge of the board.
- ❑ Install the two .1µf (unmarked) 1206 surface mount caps at locations **C7** and **C8**. One is mounted directly under the CPU. The other is mounted *next to pins 7 and 8 of the 8-pin DIP*. These can be installed in either direction since polarity does not matter.
- ❑ Install the 5.1pf (unmarked) 1206 surface mount cap *just to the left of pads for the resonator* at location **C6**. This can be installed in either direction since polarity does not matter.
- ❑ Install the 10K Ω (103 or 1002) 1206 surface mount resistor *just to the right of the pads for the resonator* at location **R3**.
- ❑ Install a 4.7K Ω (472 or 4701) 1206 surface mount resistor *near the edge of the board* at location **R5**. This is the pull up resistor for the optional EEPROM in U2.
- ❑ Install a 4.7K Ω (472 or 4701) 1206 surface mount resistor *near the middle of the board* at location **R4**. This is the pull up resistor used to reset the microcontroller. If R1 is installed this should remain empty.
- ❑ Install the 15µf (156) surface mount capacitor (plastic rectangular box) *just to the right of the ST-223 regulator* at location **C5**. Observe the polarity as you do this! The stripe must be installed facing up toward the + sign. This is a tantalum capacitor and the proper polarity is critical.
- ❑ Install 3-pin socket for the resonator at location **Y1**. This is a right angle machined pin socket. The sockets should point away from the ST-223 regulator so that the resonator sits underneath the CPU. This works well since there is extra clearance provided by the 20-pin SIP headers on the sides.
- ❑ Install the two 20-pin (1x20) from the bottom side of the board. Insert the larger diameter pins into the holes of the PCB. Make sure the connector is flush with the board. Proper alignment is critical since the module will not seat properly if these are not installed correctly.

NOTE: The 4-pin header is installed on the top side of the PCB. It is installed last otherwise it would be difficult to solder some of the other components. When installing the headers, insert the shorter leads into the PCB. The longer lead is where the SX-Key (or SX-Blitz) will plug into.

- ❑ Install the 4-pin SIP header (or right angle header) at location **X1**. If installing the right angle version, the pins should go over the MAX232/MAX3232 chip at U3.

- ❑ *OPTIONAL:* De-flux and clean the board. This step is not required but when properly cleaned the board's appearance is better and it is easier to spot cold solder joints and solder bridges. Depending upon the type of flux used, this can be done economically using common rubbing alcohol and an old toothbrush.

- ❑ *OPTIONAL:* Install an EEPROM chip (like the 24LC128) in the 8-pin DIP socket at **U2**. The notch should point toward the left. One of these is included in the deluxe kit.

This concludes the assembly procedures for the SX48 OEM module. Congratulations and welcome to surface mount technology! Before applying power verify the correct location and orientation of all parts. Also check that there are no solder bridges or poor solder joints. Some of the traces are very close together and it is easy to accidentally create a solder bridge across a trace or two.

INSTALLATION AND OPERATION

This module conforms to the standard 40-pin socket as defined by the BS2p40 for ease of use on breadboards and any place that a BS2p40 module can be used. Since it uses a Parallax SX-Key or SX-Blitz for programming the serial port normally used for programming the BS2p40 is now available for use.

SPECIFICATIONS

Main CPU:	Parallax SX48BD/TQ 48-pin TQFP
Speed:	Up to 50MIPS with 50Mhz resonator or (32kHz to 4 Mhz via internal oscillator)
EE/Flash:	4096 x 12
RAM:	262 bytes
Timers:	Two 16-bit timers with 8-bit prescalers, 8-bit Watchdog timer, and 8-bit RTCC
I/O:	4 configurable 8-bit ports (32 bits total), plus 4 additional bits on Port A reserved for the MAX232 Serial driver (1 Serial In, 1 Serial Out) and onboard EEPROM
Power in:	Up to 12V unregulated
Power:	5V LDO regulator .8A (or 3.3v LDO regulator)

Serial Interface: More of a true RS-232 Interface via MAX232/MAX3232 chip.

IN CASE OF DIFFICULTY

PROBLEM	POSSIBLE CAUSE
Module is not seen by the SX-Key programming software	- Check for solder bridges and cold solder joints. - Check orientation of tantalum capacitor. - Remove resonator from socket if installed.
Output to a pin also changes output on neighboring pin.	- Check for solder bridges.

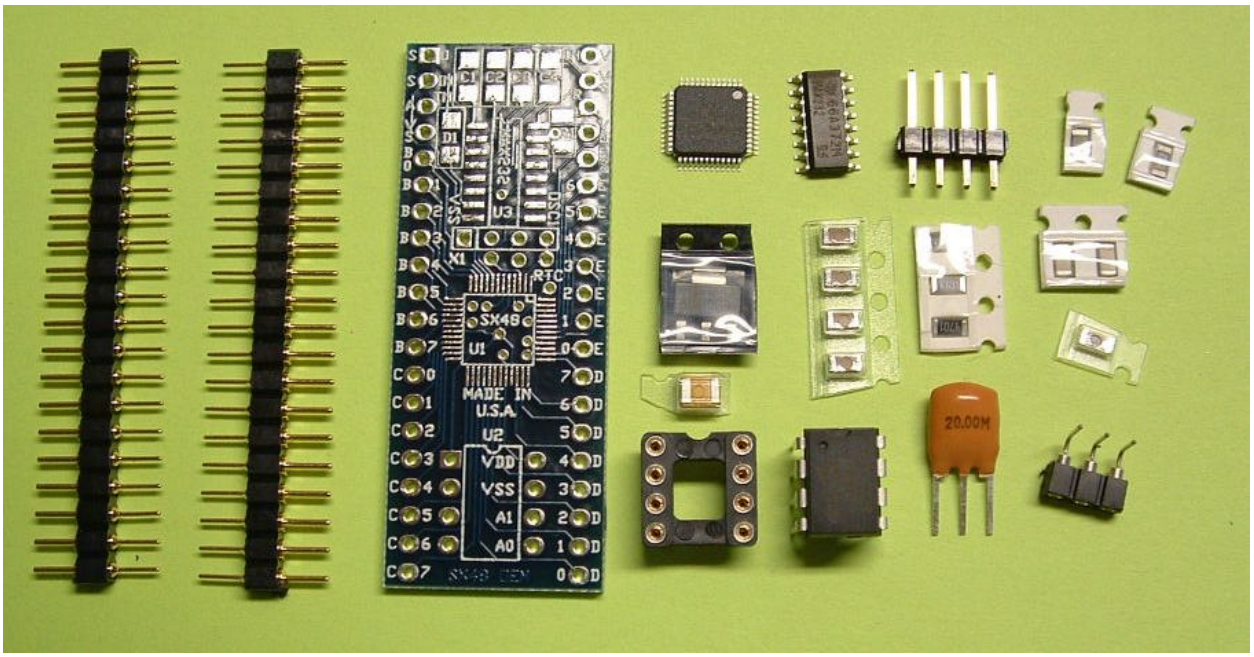
Finished SX48 OEM module pinout

The following table shows the pinout for the finished SX48 OEM module. It lists the pin name as well as the description. A slash before the Mnemonic name means that the line is active low.

Pin #	Name	Definition
1	SOUT	Serial data (from the SX48 through MAX232) - RA.3 (TX)
2	SIN	Serial data (into the SX48 through MAX232) - RA.2 (RX)
3	ATN	Attention – May be used to reset module
4	VSS	Ground
5	B0	Port B
6	B1	
7	B2	
8	B3	
9	B4	
10	B5	
11	B6	
12	B7	
13	C0	Port C
14	C1	
15	C2	
16	C3	
17	C4	
18	C5	
19	C6	
20	C7	
21	D0	Port D
22	D1	
23	D2	
24	D3	
25	D4	
26	D5	
27	D6	
28	D7	
29	E0	Port E
30	E1	
31	E2	
32	E3	
33	E4	
34	E5	
35	E6	
36	E7	
37	VDD	+5v out from regulator on module
38	/RES	Module Reset
39	VSS	Ground
40	VIN	Unregulated power in (up to +12 VDC)

RA.0 goes to pin 5 on EEPROM socket at U2 (SDA)
RA.1 goes to pin 6 on EEPROM socket at U2 (SCL)
RTCC goes to via marked RTC near pin 48 of the Microcontroller

REPLACEMENT PARTS LIST



Top of module

Quantity	Description	Location
1	SX48BD/TQ 48-pin TQFP microcontroller Chip	U1
1	MAX232D (surface mount) (Change to MAX3232 for 3.3v)	U3
1	EEPROM (8-pin DIP format) (Optional EEPROM like 24LC128)	U2
1	8-pin DIP socket	U2
4	1.0µf surface mount capacitors – 1206 (Change to .1µf for 3.3v)	C1, C2, C3, & C4
1	Zero Ω Jumper (000) -- 1206	D1
1	4-pin SIP header (or right angle header) for SX-Key / SX-Blitz	X1
1	Custom PCB p/n SX48 OEM	

Bottom of module

Quantity	Description	Location
1	5v LDO voltage regulator ST-223 package (Optional 3.3v)	U4
2	.1µf surface mount capacitors – 1206	C7, C8
1	5.1pf surface mount capacitor – 1206	C6
1	15µf surface mount tantalum capacitor – 16v 3528-21 (EIA)	C5
3	4.7K Ω surface mount resistors – 1206 (R2 Optional – See instructions)	R2, R4, and R5
1	10K Ω surface mount resistor – 1206	R3
1	3-pin right angle machine-pin SIP socket	Y1
1	Optional resonator (4.00 MHz, 20.00 MHz, etc.)	Y1
2	20-pin SIP machined pin M/M headers .100 inch	Edges of module

Optional parts (not included)

Quantity	Description	Location
1	DL4148 surface mount diode .15A 75v MINIMELF	D1
1	4.7K Ω surface mount resistor – 1206 (Optional for ATN reset)	R1 *instead of R4*

ALTERNATE CONFIGURATIONS

Serial Loopback to act like BASIC Stamp module:

- ❑ Install a small DL4148 signal diode (not included) at location D1 on the top side of the board instead of the zero Ω jumper. The band should go toward the left going to pin 1 of the module.

- ❑ Install a 4.7K Ω (472 or 4701) 1206 surface mount *resistor above the ST-223 regulator near the left side of the board* at location **R2**. This is the resistor that provides the feedback.

Serial ATN line used to RESET module:

- ❑ Install a 4.7K Ω (472 or 4701) 1206 surface mount resistor on the pads *just above the MAX232D* chip at location **R1**. When R1 is installed the 4.7K Ω pullup resistor at R4 should remain empty.

Extra TTL Serial port instead of EEPROM at U2:

- ❑ Instead of installing the 8-pin DIP socket at U2 a small 4-pin SIP header can be used in pins 5 though 8. There is a legend on the silkscreen showing A0, A1, VSS, and VDD. You can easily use these as generic I/O pins, two more TTL serial out ports to other devices, or even have it go to a separate MAX232 chip for another true serial port. There is a lot of flexibility.

3.3v version instead of the standard 5v build:

- ❑ There are three sets of parts that need to be exchanged in order to make this module operate as a 3.3v version. The voltage regulator (**U4**) needs to be swapped out with a 3.3v version. The MAX232D (**U3**) should be exchanged for a MAX3232, and the four 1.0µf caps (**C1**, **C2**, **C3**, and **C4**) on the top of the board need to be exchanged with .1µf caps.

REVISION HISTORY

Version 1.0 – Original Prototype. Required slight change to cut a trace on the output of the MAX232 going to the /RESET line and add a 4.7K resistor inline.

Version 1.1 – Has correction above applied to the PCB layout. No silkscreen or 4-pin programming header. (None of these were actually produced in physical form)

Version 1.2 – Current version. Has 4-pin programming header for SX-Key/SX-Blitz. Board also has silkscreen legends on both sides to aid in kit assembly.

REFERENCES

The Parallax Website and support forums:

http://www.parallax.com/

http://forums.parallax.com/forums/

SPECIAL THANKS

A note of thanks to the folks over at Parallax for handling the SX series chips and their support during the design of the module.

I'd also like to thank those people on the Parallax SX support forums for their support of the SX series chips and their feedback.