Chapter 12

XGS Pico Edition with PCB Construction – Bonus[2] Section !!!

This chapter covers the assembly and construction of the **XGS Pico Edition PCB** (printed circuit board) **Version**. If you're reading this chapter then you either bought a basic **XGS Pico Edition** and **Pico PCB Add-on Kit** or you are just interested in how the PCB version is constructed. In either case, to construct the project in this chapter you will need the following items:

§ Basic XGS Pico Edition Kit (either purchased or assembled from the parts list).

§ A XGS Pico Edition PCB Add-On Kit including PCB.

§ A soldering iron, solder, and wire cutters.

§ A Male to Male RCA A/V cable to connect your XGS Pico PCB to the TV.

§ **(Optional)** A 9V DC 300+ mA power adapter with 2.1mm jack with tip (+) and ring (-).

§ **(Optional)** A Parallax SX-Key programmer and serial cable.

Also, if you did purchase a basic XGS Pico Edition Kit on solderless breadboard, it is highly advised that you build that project as outlined in Chapter 11.16, so you understand all the parts, software, and programming of the system. If you only bought the XGS Pico Edition PCB Add-On then you will of course need to acquire all the parts contained in the basic XGS Pico Edition (which is outlined in Chapter 11.16) since the PCB version uses many of those parts.

TIP	As usual, it's a good idea to read thru the entire chapter before starting. If you wish you can skim the design sections since they are mostly review from Chapter 11, but I suggest that you definitely read the construction section *before* building the unit and soldering it together. Also, there is a video of the construction of the XGS Pico by Joshua Hintze for reference on the CD. The construction doesn't follow the same assembly order as we do in the written version here, but the video is good for reference. You can find it on the root of the CD: **xgs_pico_pcb_assembly_01.wmv**

This chapter is going to primarily cover the assembly and construction of your **XGS Pico Edition on the printed circuit board**, but leaves all technical descriptions out since they were covered in detail in Chapter 11, specifically **Chapter 11.16**. Of course, you might want to skip directly to building your Pico Edition directly on the PCB and not build the solderless breadboard version. This is fine, but remember once you solder it together, you can't really un-solder it! Thus, it's nice to have built it on the solderless breadboard for the experience before you try soldering.

Figure 12.1(a) – The XGS Pico Edition PCB Version
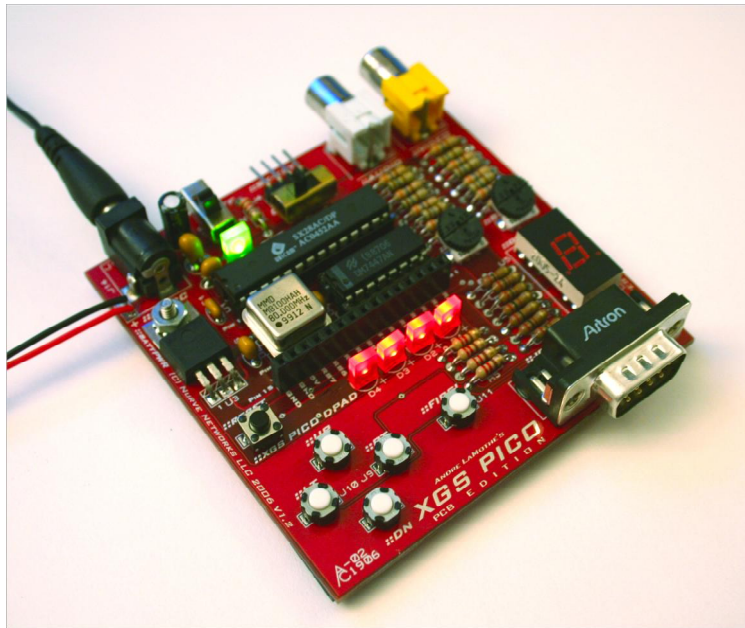
Figure 12.1(b) – The XGS Pico Edition PCB Version with annotation.
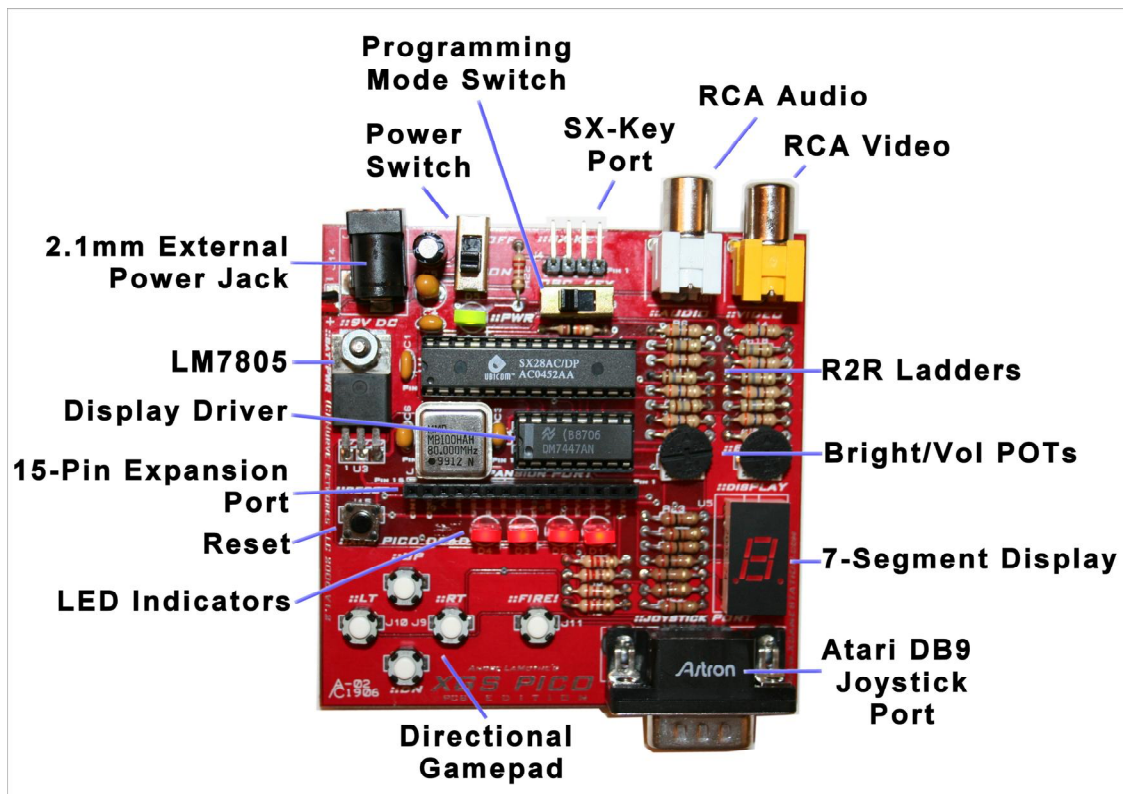
12.1 XGS Pico Edition PCB Version

To begin with Figure 12.1(a) shows the XGS Pico Edition PCB version completely assembled while Figure 12.1(b) shows all the sub-systems marked and annotated. As you can see there are some additions to the PCB version of the kit that give the Pico PCB Edition some added functionality. The new additions are:

§ A **switch** controlled interface that allows programming of the XGS Pico Edition with an SX-KEY rather than a physical jumper wire used in the solderless version.

§ A **7-segment display** driven by a 74LS47 display driver interfaced to the (4) LEDs that are already used out outputs.

§ An **on/off switch** (which always comes in handy).

§ RCA style **audio** and **video** output jacks.

§ Built in **D-Sub 9 male** Atari 2600 joystick compatible connector.

§ Built in directional **gamepad**, so no joystick is needed of one is not available.

§ A 15-pin single row **expansion header** that exports various (12) I/Os, power, and ground.

All these additions have been made *without* any significant changes to the basic XGS Pico Edition design itself. So, the designs of the base Pico Edition from Chapter 11.16 and the PCB version are nearly identical other than the added components. So all I/Os, etc. are in the same place and all the programs still run exactly the same on the PCB version. The PCB version is just better. Plus it has a built power switch, programming port, expansion interface, 7-segment display, and directional gamepad! Thus, the XGS Pico Edition PCB version is really a self contained embedded system that is totally portable and can of course generate video, audio, and control external systems as well as monitor (5) push button inputs via the game pad interface.

TIP	In the descriptions that follow, the actual schematic design files for all the systems are referenced. These files are all **Labcenter Proteus** .DSN files which can be loaded and viewed with Proteus software. Specifically, the "Isis" schematic capture program. To install the software simple run the installer located on the CD at: **CDROOT:\XGSME_HW_CD\Tools\Proteus**. Additionally, there are .JPGs' of all the designs as well, so you can view them on the computer and or print them.

Figure 12.2(a) – The Design of the XGS Pico Edition PCB Version.
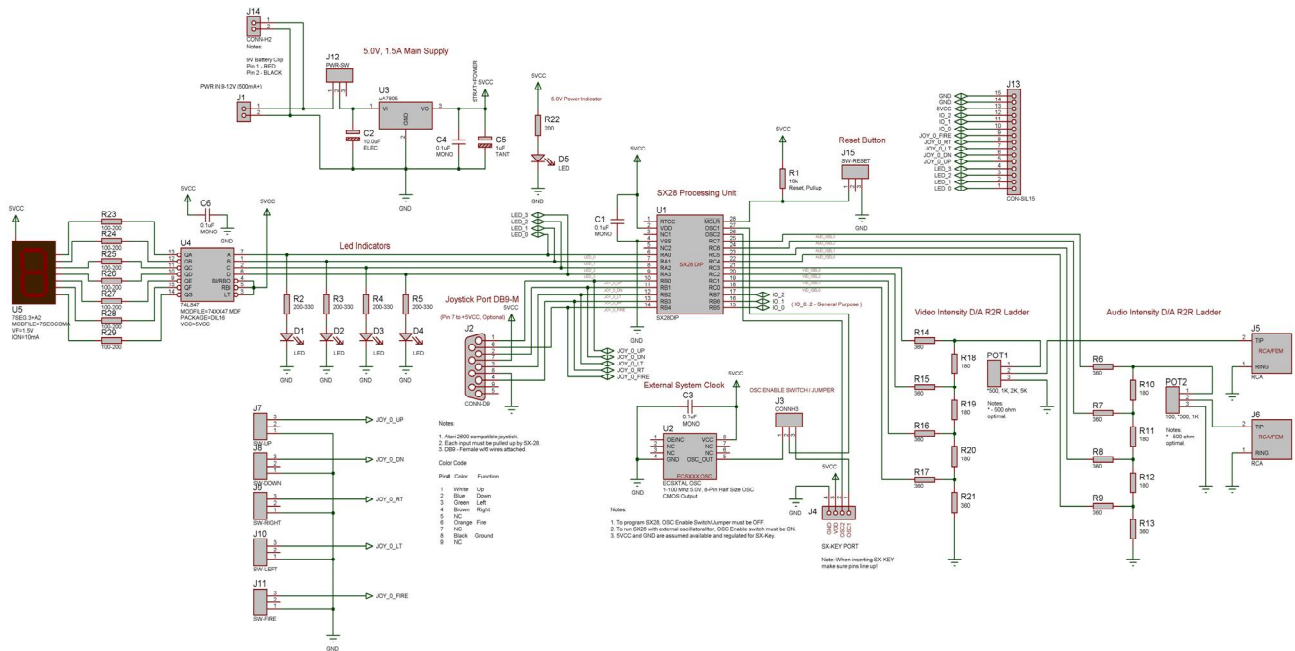
Figure 12.2(b) – The Design of the XGS Pico Edition PCB Version with Color Coding.
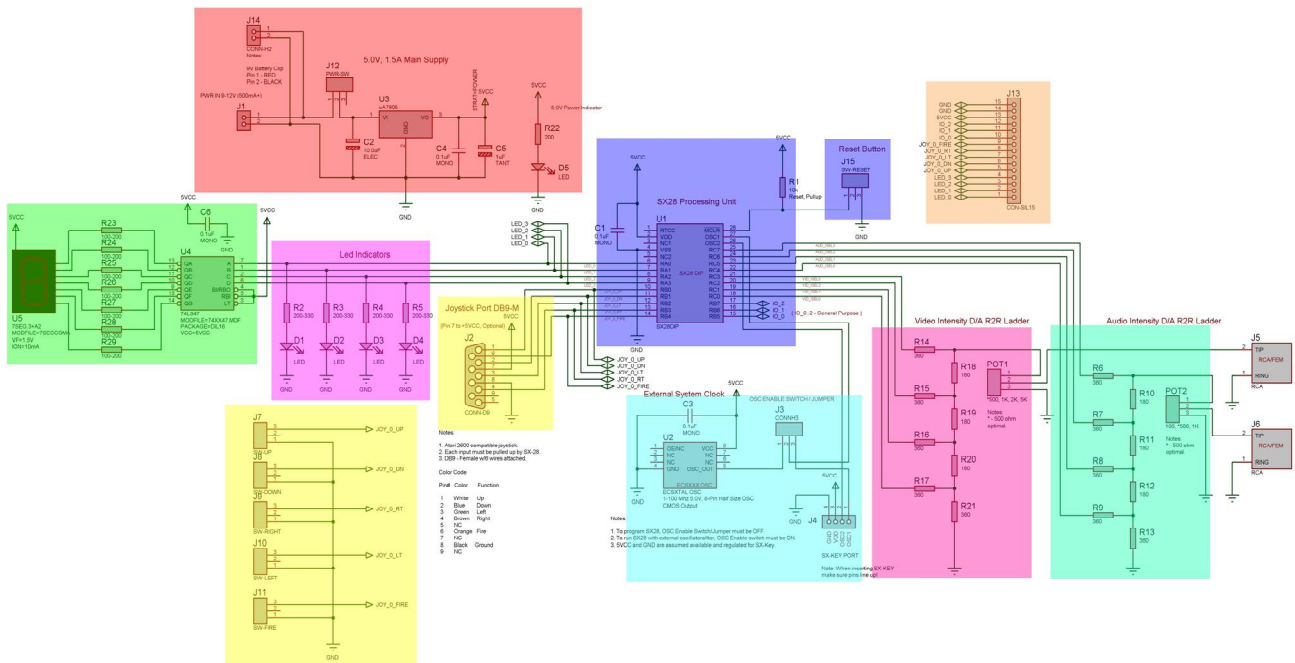
12.2 XGS Pico Edition PCB Version Designs

In this section we are going to briefly cover each of the sub-systems in the XGS Pico Edition PCB version **before** we solder anything together. All of the core systems are exactly the same as discussed in Chapter 11.16, so the material here is mostly review other than discussing the new enhancements. Also, we are always referring to the **XGS Pico PCB** version unless otherwise stated.

Let's start with an overview of the entire new design, this is shown in Figure 12.2(a). Figure 12.2(b) shows all the sub-systems color coded and separate. The Proteus design files and graphic for the complete XGS Pico Edition PCB Version are located on the CD here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_PCB_01_2R.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_PCB_01_2R.JPG

> **TIP**
>
> Later when you actually build your PCB you might want to print these files out, so you can have a high resolution copy in front of you. When printing, I suggest **landscape** mode in the highest resolution with color on.
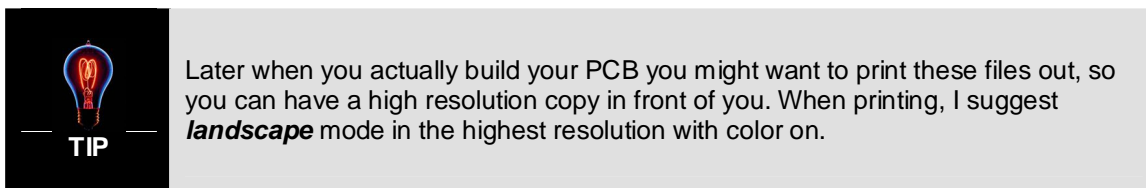
Figure 12.3 – The XGS Pico Edition PCB Version SX28 Main Processing Unit and Reset Circuit.
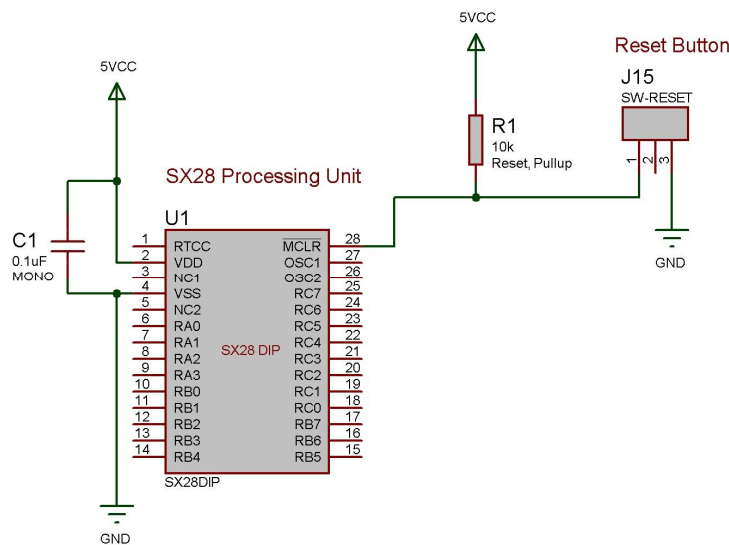
12.2.1 Processing and Reset Circuit

The same SX28 processor is of course used in the PCB version. Similarly the reset circuit is the same as the basic Pico version. Figure 12.3 shows the SX28 and reset alone. The Proteus design file and graphic for these systems are located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_sx28_reset_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_sx28_reset_PCB_01_2.JPG

Figure 12.4 – The XGS Pico Edition PCB Version Power Supply.
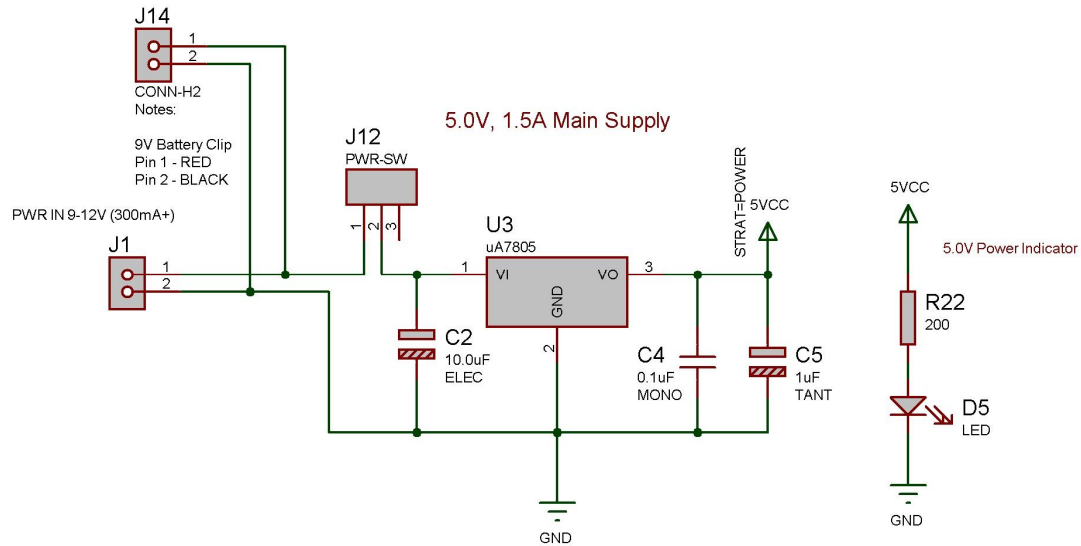
12.2.2 Power Supply

The 5V power supply for the PCB version has been slightly modified from the basic edition. Referring to Figure 12.4, the design has been enhanced to accept a 2.1mm female power jack for an external 9V 300+ mA power adapter at **J1** as well as a single pole double throw (SPDT) power switch at **J12**. Other than that, the design is the same. The Proteus design file and graphic for these systems are located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_power_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_power_PCB_01_2.JPG

Figure 12.5 – The XGS Pico Edition PCB Version Clock and Programming Port.
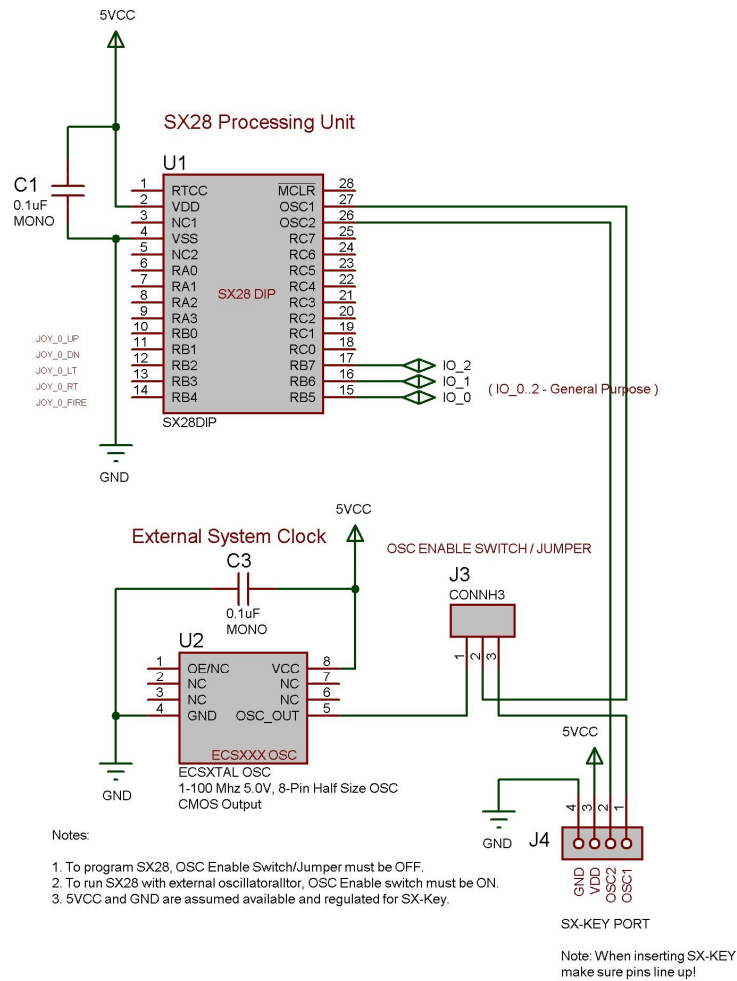
Figure 12.6 – A Close up of the Program/Run Mode Switch.
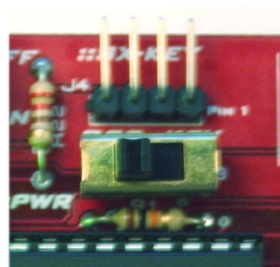
12.2.3 System Clock and Programming Port

The clocking and programming port on the Pico PCB version are a little more useful now. Referring to Figure 12.5, **J3** is no longer a simple jumper, but a single pole double throw switch (SPDT). This allows the Pico PCB to either be programmed via the SX-Key programming port at **J4**, or placed in *run* mode via the switch. The switch at **J3** basically **"gates"** the **OSC1** signal between either the on-board oscillator or the SX-Key programmer. Thus with a flip of the switch you can program the Pico Edition or place it in run mode and it will use the high speed oscillator. The PCB is labeled with the terms **"OSC"** and **"KEY"** above the switch and below **J4** to signify these two modes of operation. Figure 12.6 shows a close up of this. The Proteus design file and graphic for these systems are located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_clock_prog_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_clock_prog_PCB_01_2.JPG

Figure 12.7 – The XGS Pico PCB Version's I/O Sub-Systems.
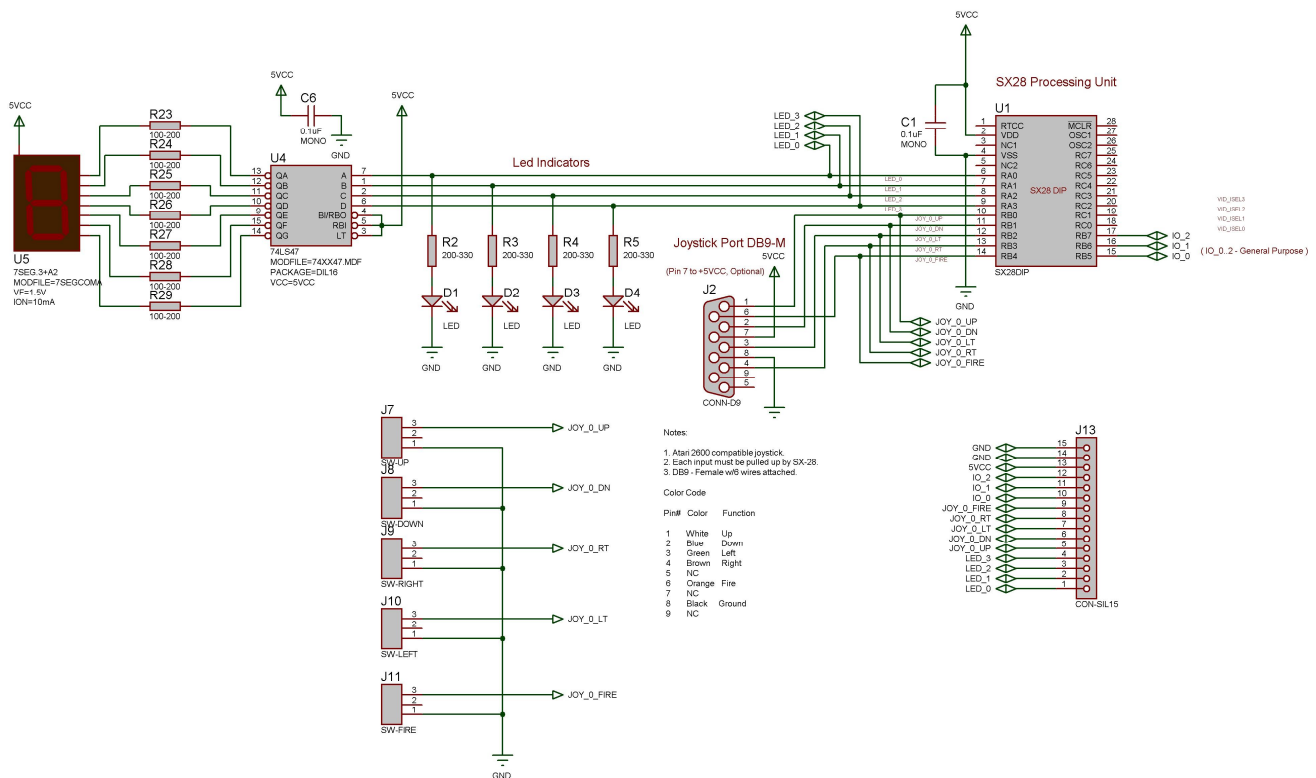
12.2.4 I/O

The PCB version's I/O is where most of the additions to the design have been made. Figure 12.7 illustrates the I/O systems alone. The new additions to the PCB version are the:

§   Built in Gamepad

§   7-Segment Display

§   15-Pin Expansion Interface

The design file and image of these systems can be found here for reference:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_io_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_io_PCB_01_2.JPG

The other I/O systems remain the same, but are slightly altered due to the mounting of components on a PCB. For example, the DSub-9 Atari 2600 connector is built into the PCB rather than cabled as it was with the basic solderless breadboard version of the XGS Pico Edition. In any event, let's cover each of the I/O sub-systems.

Figure 12.8 - The XGS Pico PCB Edition's LED Outputs.
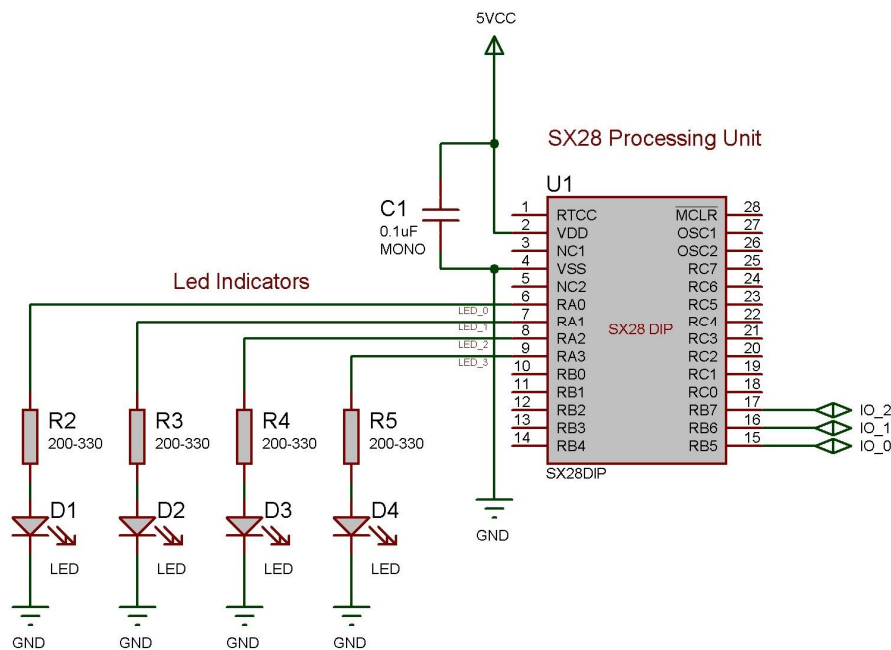
12.2.4.1 LED Indicators

The LED output indicators are the exact same in the in the Pico PCB version except that they also drive the inputs to the **74LS47** display driver (discussed in the next section). Figure 12.8 shows the design to the LED outputs, nothing more than I/Os, current limiting resistors and the LEDs themselves. The Proteus design file and graphic for these systems are located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_leds_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_leds_PCB_01_2.JPG

Figure 12.9 – The XGS Pico PCB Edition's New 7-Segment Display Hardware.
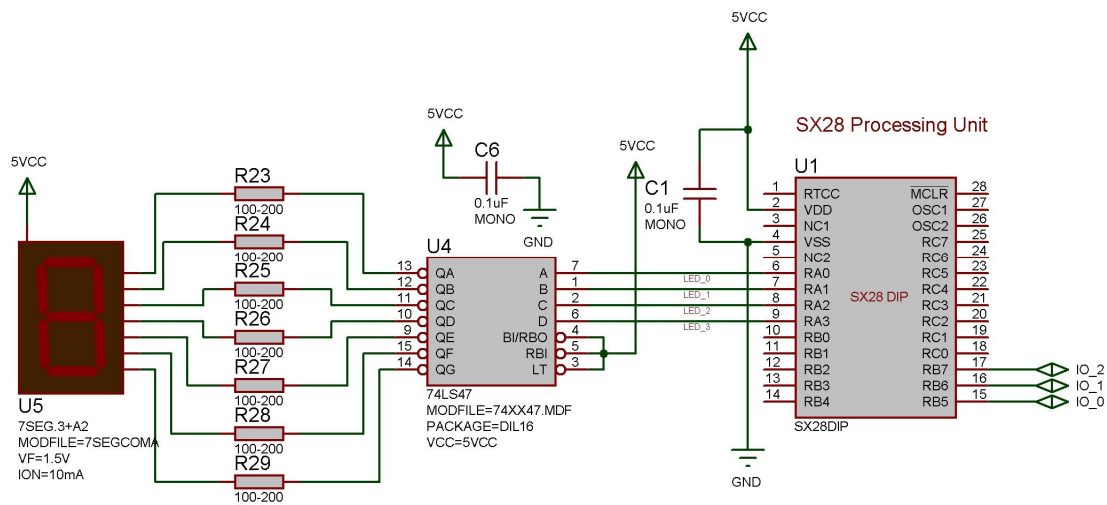
12.2.4.2 7-Segment Display

The solderless breadboard design for the Pico Edition simply didn't have enough room on the solderless breadboard to add another chip and a 7-segment display. But, with the PCB version there is room now. Figure 12.9 shows the design for the 7-segment display system. Basically, the (4) LED outputs from **RA0-RA3** on the SX28 drive the inputs to the **74LS47** common anode display driver at **U4**. This in turn drives a **LT312** style 7-segment display at **U5**. Additionally, current limiting resistors of 200-220 (**R23-29**) have been put in line with each display driver output and there is a de-coupling capacitor added across the **74LS47** at **C6**. These are the only new parts. Thus, any 4-bit code that you output from low to high across **RA0-RA3** will be displayed on the 7-segment display. Of course, the codes from 10-15 are only symbols since the **74LS47** does *not* output hex digits for these values. Figure 12.10 shows the symbols output from the **74LS47** for various inputs from 0-15.

> **NOTE**
>
> The **74LS47** is a *"common anode"* display driver. This means to activate any segment (light it) you must drive the segment logic LOW. Thus, all the segments have a common leg that must be tied to the positive supply or logic HIGH.

Figure 12.10 – An Excerpt from the Data Sheet for the 74LS47 7-Segment Common Anode Display Driver.

description

The '46A, '47A, and 'LS47 feature active-low outputs designed for driving common-anode LEDs or incandescent indicators directly. The '48, 'LS48, and 'LS49 feature active-high outputs for driving lamp buffers or common-cathode LEDs. All of the circuits except 'LS49 have full ripple-blanking input/output controls and a lamp test input. The 'LS49 circuit incorporates a direct blanking input. Segment identification and resultant displays are shown below. Display patterns for BCD input counts above 9 are unique symbols to authenticate input conditions.

The '46A, '47A, '48, 'LS47, and 'LS48 circuits incorporate automatic leading and/or trailing-edge zero-blanking control (RBI and RBO). Lamp test (LT) of these types may be performed at any time when the BI/RBO node is at a high level. All types (including the '49 and 'LS49) contain an overriding blanking input (BI), which can be used to control the lamp intensity by pulsing or to inhibit the outputs. Inputs and outputs are entirely compatible for use with TTL logic outputs.
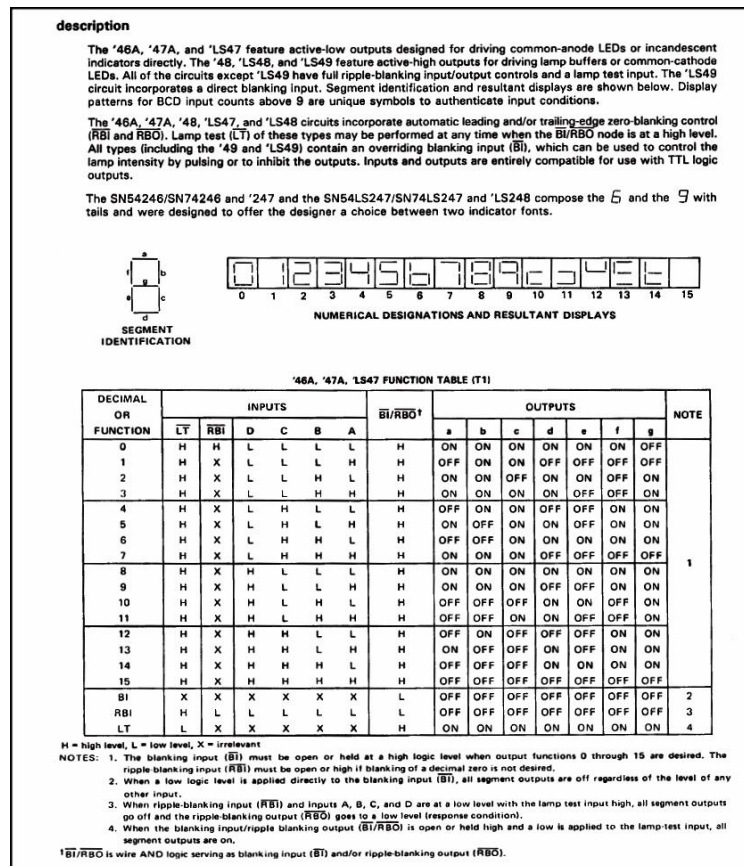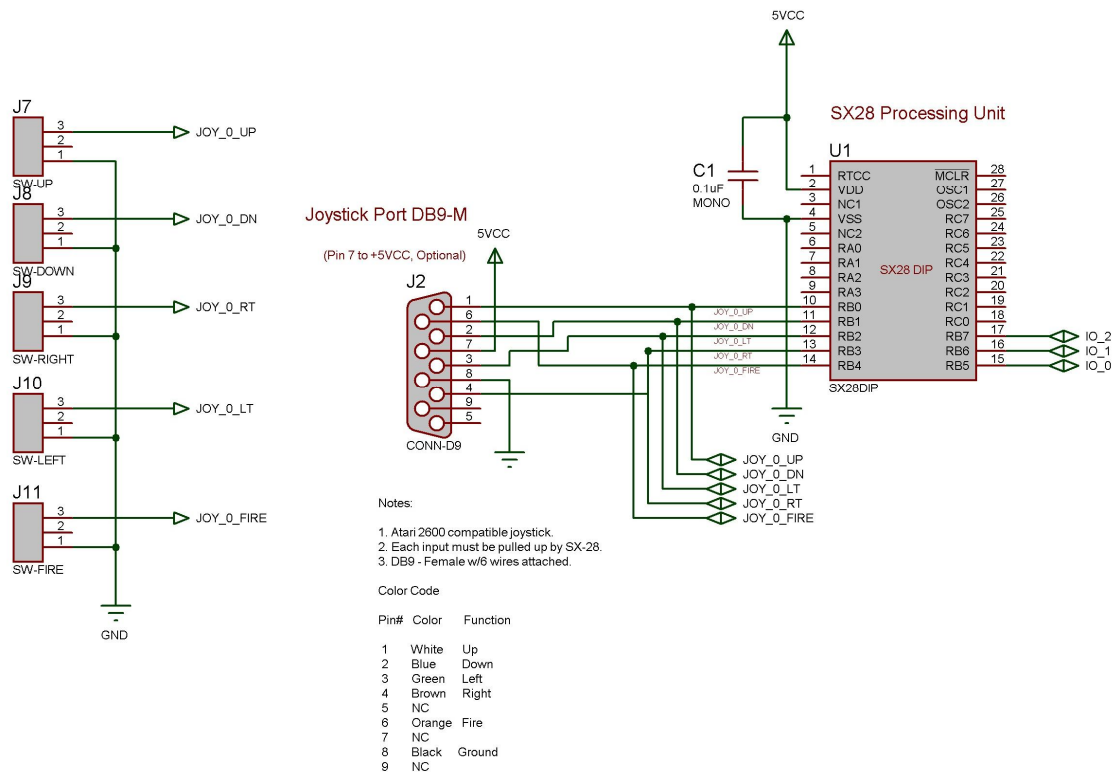
The SN54246/SN74246 and '247 and the SN54LS247/SN74LS247 and 'LS248 compose the 6 and the 9 with tails and were designed to offer the designer a choice between two indicator fonts.

SEGMENT IDENTIFICATION

NUMERICAL DESIGNATIONS AND RESULTANT DISPLAYS
0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15

'46A, '47A, 'LS47 FUNCTION TABLE (T1)

DECIMAL OR FUNCTION	INPUTS						BI/RBO†	OUTPUTS							NOTE
	LT	RBI	D	C	B	A		a	b	c	d	e	f	g	
0	H	H	L	L	L	L	H	ON	ON	ON	ON	ON	ON	OFF	
1	H	X	L	L	L	H	H	OFF	ON	ON	OFF	OFF	OFF	OFF	
2	H	X	L	L	H	L	H	ON	ON	OFF	ON	ON	OFF	ON	
3	H	X	L	L	H	H	H	ON	ON	ON	ON	OFF	OFF	ON	
4	H	X	L	H	L	L	H	OFF	ON	ON	OFF	OFF	ON	ON	
5	H	X	L	H	L	H	H	ON	OFF	ON	ON	OFF	ON	ON	
6	H	X	L	H	H	L	H	OFF	OFF	ON	ON	ON	ON	ON	
7	H	X	L	H	H	H	H	ON	ON	ON	OFF	OFF	OFF	OFF	1
8	H	X	H	L	L	L	H	ON	ON	ON	ON	ON	ON	ON	
9	H	X	H	L	L	H	H	ON	ON	ON	OFF	ON	ON	ON	
10	H	X	H	L	H	L	H	OFF	OFF	OFF	ON	ON	OFF	ON	
11	H	X	H	L	H	H	H	OFF	OFF	ON	ON	OFF	OFF	ON	
12	H	X	H	H	L	L	H	OFF	ON	OFF	OFF	OFF	ON	ON	
13	H	X	H	H	L	H	H	ON	OFF	OFF	ON	OFF	ON	ON	
14	H	X	H	H	H	L	H	OFF	OFF	OFF	ON	ON	ON	ON	
15	H	X	H	H	H	H	H	OFF	OFF	OFF	OFF	OFF	OFF	OFF	
BI	X	X	X	X	X	X	L	OFF	OFF	OFF	OFF	OFF	OFF	OFF	2
RBI	H	L	L	L	L	L	L	OFF	OFF	OFF	OFF	OFF	OFF	OFF	3
LT	L	X	X	X	X	X	H	ON	ON	ON	ON	ON	ON	ON	4

H = high level, L = low level, X = irrelevant

NOTES: 1. The blanking input (BI) must be open or held at a high logic level when output functions 0 through 15 are desired. The ripple-blanking input (RBI) must be open or high if blanking of a decimal zero is not desired.
2. When a low logic level is applied directly to the blanking input (BI), all segment outputs are off regardless of the level of any other input.
3. When ripple-blanking input (RBI) and inputs A, B, C, and D are at a low level with the lamp test input high, all segment outputs go off and the ripple-blanking output (RBO) goes to a low level (response condition).
4. When the blanking input/ripple blanking output (BI/RBO) is open or held high and a low is applied to the lamp-test input, all segment outputs are on.

†BI/RBO is wire AND logic serving as blanking input (BI) and/or ripple-blanking output (RBO).

The data sheet for the Texas Instruments implementation of the 74LS47 is located on the CD here:

CDROOT:\XGSME_HW_CD\Datasheets\sn74ls47.pdf

The Proteus design file and graphic for the 7-segment display system is located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_7seg_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_7seg_PCB_01_2.JPG

Figure 12.11 – XGS Pico PCB Joyport and Gamepad Design.

12.2.4.3 Joystick Port and Gamepad

The solderless breadboard version of the XGS Pico interfaces to an **Atari 2600** controller via a cabled male DB9 connector. However, with the XGS Pico PCB version there was the opportunity to add a DB9 connector to the PCB board itself. Additionally, a discrete momentary switch based *"direction pad"* or *"Dpad"* was added to the design in some free board space. This makes the system really cool since you don't need any other input device, you can just use the Dpad on the board to control games and programs. Figure 12.11 shows the design for the both the joystick interface DB9 connector along with the new Dpad. Of course, the Dpad is nothing more than 5 momentary switches which are paralleled to the Atari DB9 interface. Table 12.1 shows the mapping.
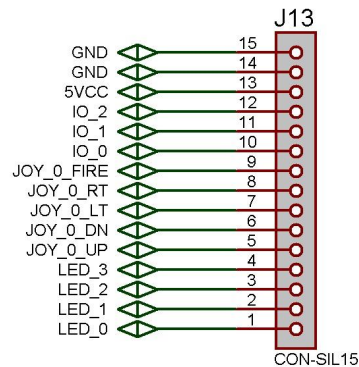
Table 12.1 – Dpad Mapping.

Direction	Connector/Switch
Up	J7
Down	J8
Right	J9
Left	J10
Fire	J11

The Proteus design file and graphic for the 7-segment display system is located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_gamepad_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_gamepad_PCB_01_2.JPG

Figure 12.12 – The XGS Pico PCB 15-Pin Expansion Header Interface.

12.2.4.5 15-Pin Expansion Interface

The solderless breadboard version of the Pico Edition more or less let's you insert a wire anywhere you like in the breadboard, but on a PCB version a decision had to be made on how to export out any signals. How many, and what kind of interface. A reasonable, compromise was made with available board space to use a 15-pin, single row female header at **J13**. Figure, 12.12 shows the design for it (which isn't much). The most important this is the I/O mapping which is shown in Table 12.2.

Table 12.2 – XGS Pico PCB Expansion Port Interface Mapping.

Header Pin	SX28 Signal / Name
1	RA_0 / LED_0
2	RA_1 / LED_1
3	RA_2 / LED_2
4	RA_3 / LED_3
5	RB_0 / JOY_0_UP
6	RB_1 / JOY_0_DN
7	RB_2 / JOY_0_LT
8	RB_3 / JOY_0_RT
9	RB_4 / JOY_0_FIRE
10	RB_5 / IO_0
11	RB_6 / IO_1
12	RB_7 / IO_2
13	+5V
14	GND
15	GND

Note: *Pin 1 is on the right of the header when the PCB is facing toward the front.*

As you can see from Table 12.2, the SX28 ports RA and RB are completely exported while RC is not. The Proteus design file and graphic for the 7-segment display system is located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_expansion_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_expansion_PCB_01_2.JPG

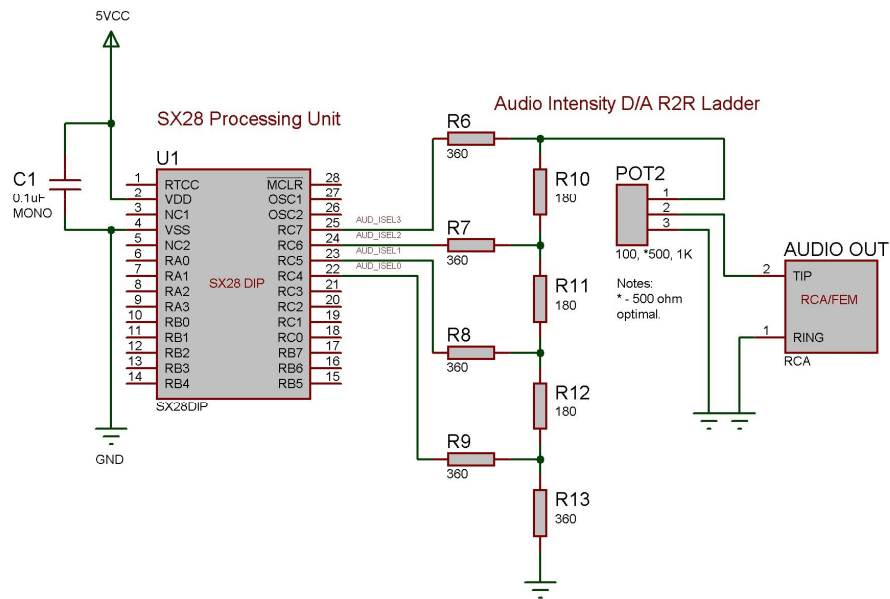*Figure 12.13 – The Pico PCB Edition's Graphics Hardware.*
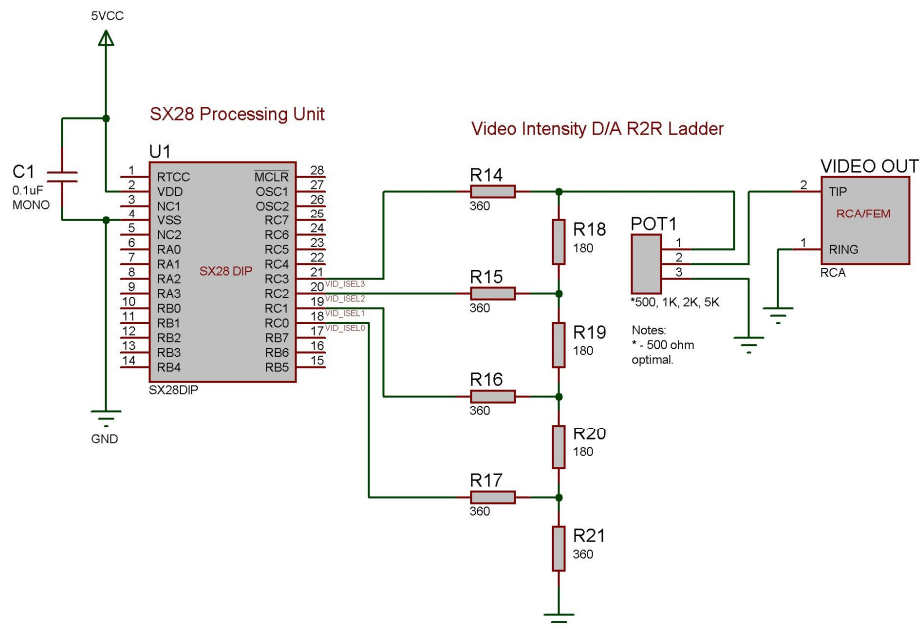
12.2.5 Graphics Hardware

The graphics hardware on the PCB version is identical to the solderless breadboard version; a simple 4-bit DAC is used to generate video. Figure 12.13 shows the design for reference. The Proteus design file and graphic for the graphics hardware is located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_video_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_video_PCB_01_2.JPG

Figure 12.14 – The Pico PCB Edition's Sound Hardware.

12.2.6 Sound Hardware

Similarly to the graphics hardware, the PCB version sound hardware is identical to the solderless breadboard version; a simple 4-bit DAC is used to generate audio via PCM algorithms. Figure 12.14 shows the design for reference. The Proteus design file and graphic for the sound hardware is located here:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_audio_PCB_01_2.DSN
CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_kit_audio_PCB_01_2.JPG

12.3 Assembly / Construction Preparation

Since there are a lot of parts that make up the XGS Pico PCB Edition, the first step is to collect everything you need to build the unit and get it organized and in front of you. There are two steps here; *step one* is to collect the parts from your XGS Pico Edition solderless breadboard edition that will be needed to build the PCB version. *Step two* is to organize the parts from the Pico PCB Add-On Kit. Of course, you might not have the XGS Pico Edition solderless version and have decided to simply use your own parts. This is no problem, just make sure you have everything listed in the next sections to begin construction.

NOTE

If you purchased a XGS Pico Edition plus PCB Add-On then you should everything you need, in a couple anti-static pouches.

Figure 12.15 – XGS Pico Edition Parts needed from Basic Kit.
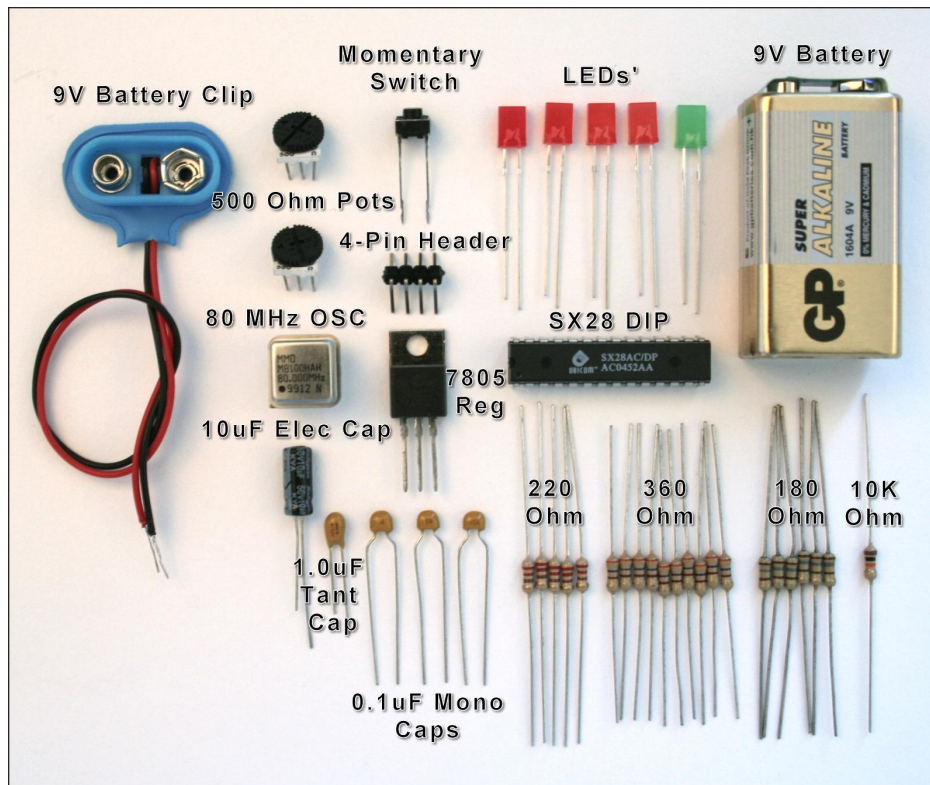
12.3.1 Collecting Parts from the Basic XGS Pico Edition

This section assumes that you have previously purchased or put together your own XGS Pico Edition solderless breadboard version kit as outlined in Chapter 11.16. To build the PCB version you will use a subset of these parts. Figure 12.15 shows all the parts collected from the XGS Pico Edition solderless version together for that you will need. Note that your parts might have slightly different styles and colors. And your resistance values might be 200 Ohm instead of 220, or 365 Ohm instead of 360 Ohm and so forth. Review the figures and then make sure you have collected everything listed in Table 12.3 below.

Table 12.3 – XGS Pico Edition Parts List

Quantity	Reference Designator	Description
ICs / Active		
(1)	U1	Ubicom SX28 Microcontroller DIP 28
(1)	U2	80.000 MHz 8-Pin DIP, Half Size Oscillator
(1)	U3	LM7805 Voltage Regulator, TO220 Package.
Optronics		
(4)	D1-D4	Red LEDs (Light Emitting Diode).
(1)	D5	Green LED (Light Emitting Diode).

Capacitors

(1)	C2	10uF Electrolytic Radial, 0.1" Lead Spacing (polarized).
(1)	C5	1uF Tantalum Dipped, 0.1" Lead Spacing (polarized).
(3)	C1, 3, 4	0.1uF Monolithic, 0.1" Lead Spacing (non-polarized).

Resistors / Pots

(2)	POT1, 2	500 Ohm Potentiometers.
(5)	R2, 3, 4, 5, 22	200-220 Ohm Axial 0.25W 5% Resistors.
(10)	R6-9, 13, 14-17, 21	360 Ohm Axial, 0.25W 5% Resistors.
(6)	R10-12, 18-20	180 Ohm Axial, 0.25W 5% Resistors.
(1)	R1	10K Ohm Axial, 0.25W 5% Resistors.

Other / Mechanical / Cable / Wiring / Power

(1)	J4	4-Pin Right Angle Header for SX28 Programming.
(1)	N/A	9V Battery.
(1)	J14	9V Battery Clip/Strap W/4" Leads.
(1)	SW1/J15	SPST Momentary Micro Switch 2-Lead.

NOTE	Your kit might have slightly different resistor values. For example, instead of 200 Ohm you might have 220 Ohm. Instead of 360 Ohm, you might have 365 Ohm (more precise). The best thing to do is sort your parts and know the values before you begin.

Now, that you have all the parts separated from the XGS Pico Edition solderless kit, next let's get the parts from the XGS Pico PCB Add-On kit.

Figure 12.16 – The XGS Pico Edition PCB Add-On Kit Parts.
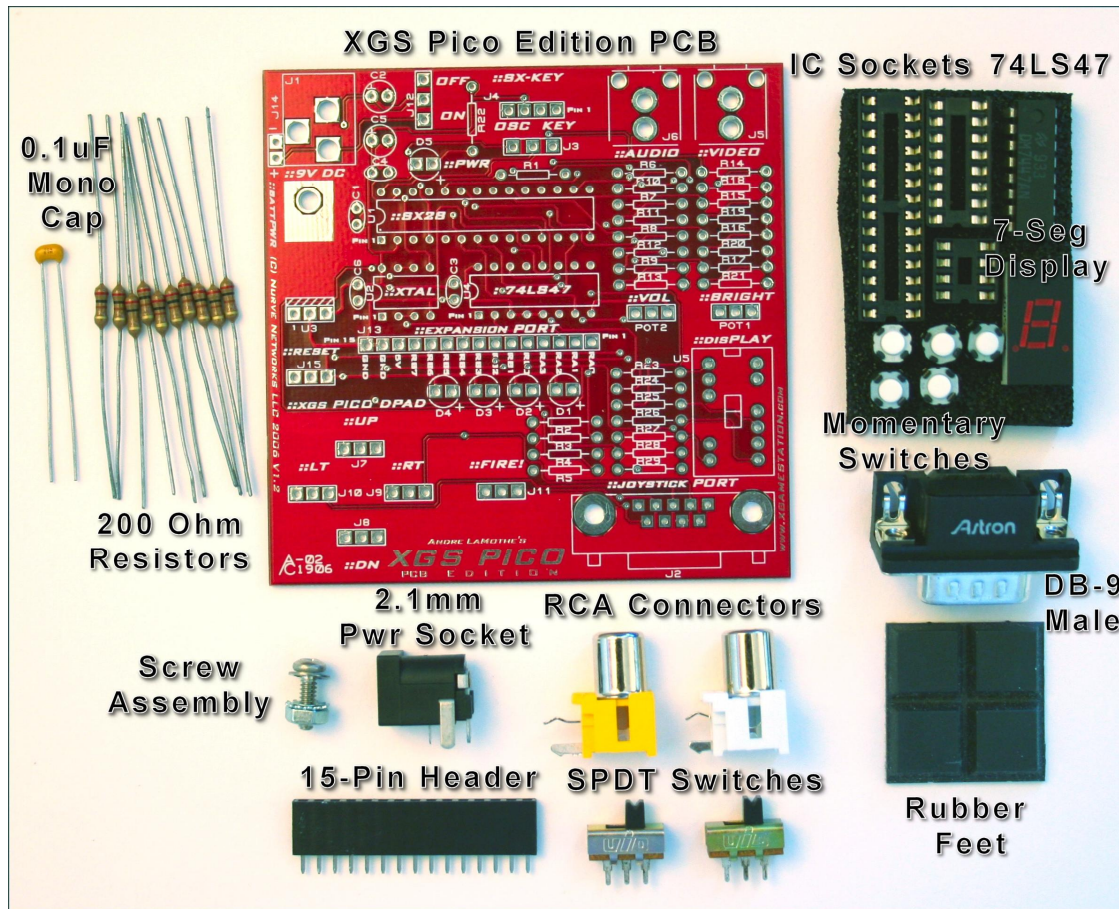
12.3.2 Unpacking Parts from the XGS Pico Edition PCB Add-On

To build the XGS Pico Edition PCB, you should have in your possession the **Pico PCB Add-on Kit**. This kit contains the PCB along with a number of extra parts. Figure 12.16 shows all the parts laid out nicely. Starting from left to right, top to bottom you see the passive cap and resistors, the PCB, some foam with IC sockets along with the momentary switches for the Dpad and finally the 7-segment display. Moving down, are the *"mechanicals"* such as the screw assembly for the voltage regulator, power jack, RCA connectors, and rubber feet. Finally, on the very bottom of the photo is the 15-pin expansion interface header along with the two switches needed on the Pico PCB version (power and the SX-Key switch).

Go ahead and make sure you have all the parts shown in the Figure 12.16. Of course, once again you might have a capacitor that is a different color or resistors that are slightly different values and or mechanicals that have slightly different designs or colorings. Table 12.4 below formally lists the exact parts you should have and their final *reference designators* in the design.

Table 12.4 – XGS Pico PCB Add-On Parts List

Quantity	Reference Designator	Description
ICs / Active		
(1)	U4	74LS47 7-Segment Display Driver.
(1)	U4 Socket	16-Pin Dip Socket 0.3" Width for 74LS47.
(1)	U1 Socket	28-Pin Dip Socket 0.3" Width for SX28.
(1)	U2 Socket	8-Pin Dip Socket 0.3" Width for Oscillator.
Optronics		
(1)	U5	LT312 7-Segment Common Anode Display (red).
Capacitors		
(1)	C6	0.1uF Monolithic, 0.1" Lead Spacing (non-polarized).
Resistors		
(10)	R23-29	200-220 Ohm Axial 0.25W 5% Resistors.
Other / Mechanical / Cable / Wiring / Power		
(1)	N/A	XGS Pico Edition PCB.
(5)	J7-11	SPST Momentary Switch, 0.3" Spacing.
(2)	J3, J12	SPDT Micro Switch 3mm Spacing.
(1)	J2	DB9 Male Connector for Joysticks PCB Mount.
(1)	J13	15-Pin 0.1" sip headers to export out pins.
(1)	J5	Video RCA JACK R/A YELLOW PCB mount, thru hole.
(1)	J6	Audio RCA JACK R/A WHITE PCB mount, thru hole.
(1)	J1	2.1mm R/A PC Mount Power Jack.
(4)	N/A	Self Adhesive Rubber Feet.
(1)	N/A	Screw and Nut Assembly for Voltage Regulator.

12.4 Building the XGS Pico Edition PCB

Now you're ready to build the XGS Pico Edition PCB Version! Unlike the solderless breadboard version where the unit was wired system by system, when soldering the unit together we are going to solder the parts that are most convenient and keep the work space on the board as free as possible. For example, you don't want to solder in one part that might make it hard to solder another part in later. Thus, I have found an *"order of operations"* that is most optimal to construct the Pico PCB with the least amount of problems. There are exactly *16 steps* to build the system. With that in mind, you should have all your parts ready and accessible, your soldering iron hot, safety goggles' on, and a good supply of solder (the unit takes about 10 feet of solder to complete).

Video	There is a video of the construction of the XGS Pico by *Joshua Hintze* for reference on the CD. The construction doesn't follow the same assembly order as we do in the written version here, but the video is good for reference. You can find it on the root of the CD: **xgs_pico_pcb_assembly_01.wmv.** It's about 22 minutes and worth watching since seeing someone go thru the assembly process is very helpful. But, it's not a requirement.

Review Figures' 12.1 and 12.2 for design reference. Also, you might want to print out the XGS Pico Edition PCB design from this file:

CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_PCB_01_2.JPG

Or you can load the Proteus design file in, so you can scroll and zoom:

**CDROOT:\XGSME_HW_CD\Datasheets\xgs_pico_edition_PCB_01_2.DSN**

In any event, it's good to have a photo or image of the final PCB and the design in front of you to work with. Now, without further ado, let's build this thing! We are going to build the Pico PCB in steps rather then specific functional blocks. This is because the placement of parts and soldering them doesn't necessarily map to functional blocks in all cases. Thus, the construction will be illustrated in a step by step fashion with details each step and photos. So, just follow along and you should have no problems. However, we will begin with the power supply since we need to confirm this functional block first. Lastly, make sure to read each step completely before performing it.

TIP	If you haven't soldered a lot then remember; you want clean silver solder joints. Not cold and, dull. So, when soldering *first* apply the soldering iron to the component that you are soldering, make good contact with the PCB hole as well, get it good and hot (0.5-1 seconds). Then start introducing your solder into the interface where its hot, let it flow. Then remove the solder first, *then* the soldering iron second. This way after you remove the solder, the soldering iron heats the joint a little more, burns off more of the flux and you get a nice shiny solder joint. Each solder joint should take about 2-3 seconds and you should count them off in your head; 1…2…3. Each time, this way it's all consistent.

Figure 12.17 – The Completed Power Supply.
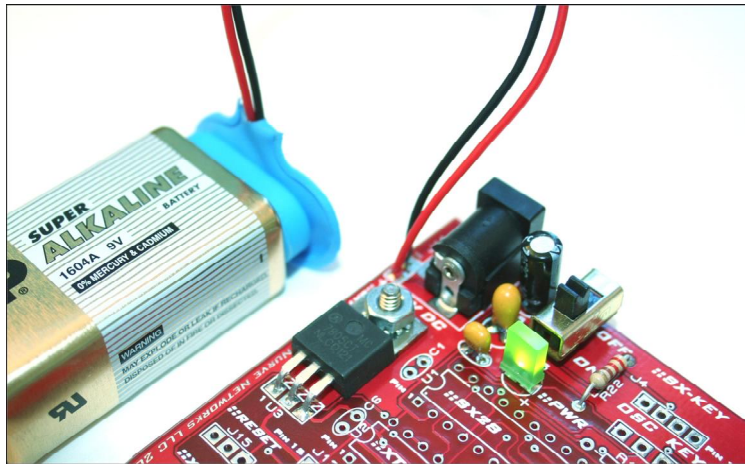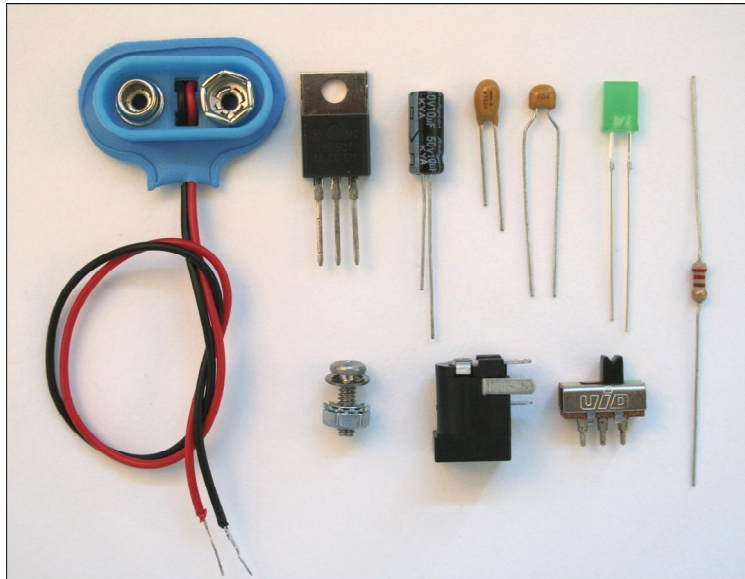

12.4.1 Assembly Part 1: The Power Supply

Figure 12.17 shows the completed power supply with the 9V battery plugged in and the Pico Edition powered up. This is what we are going to build first. After this first system we are going to pick up the pace a bit. Let's begin by collecting the parts for the power supply. They are listed in Table 12.5 and shown in Figure 12.18.

Table 12.5 – Parts for Pico PCB Power Supply.

Reference Designator(s)	Description
U3	LM7805 Voltage Regulator, TO220 Package.
N/A	Screw and Nut Assembly for Voltage Regulator.
J14	9V Battery Clip/Strap W/4" Leads.
J12	SPDT Micro Switch 3mm Spacing.
J1	2.1mm R/A PC Mount Power Jack.
D5	Green LED (Light Emitting Diode).
R22	200-220 Ohm Axial 0.25W 5% Resistor.
C2	10uF Electrolytic Radial, 0.1" Lead Spacing (polarized).
C5	1uF Tantalum Dipped, 0.1" Lead Spacing (polarized).
C4	0.1uF Monolithic, 0.1" Lead Spacing (non-polarized).

Note: Only **J12** and **J14** come from the Pico Add-On Kit, the all the other parts are from the basic Pico Kit.
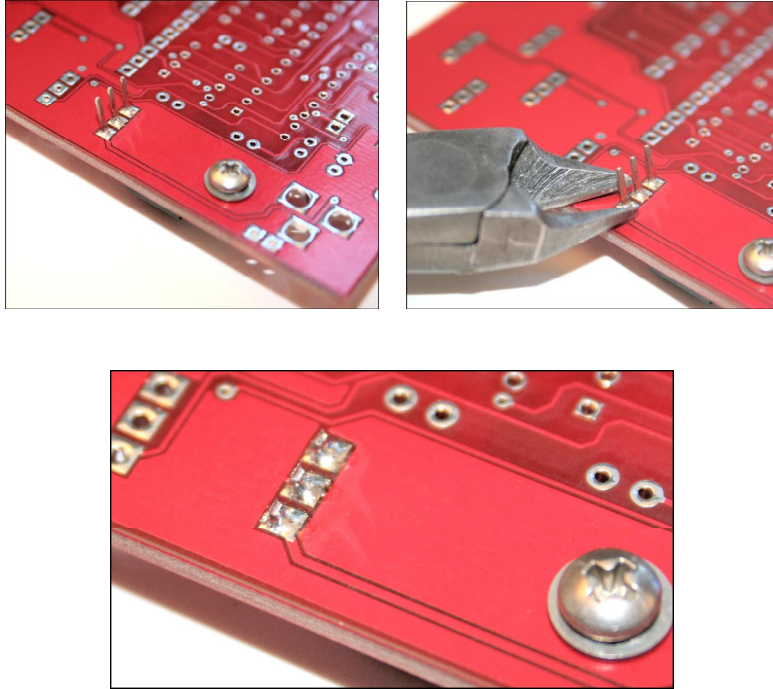
Figure 12.18 – The Parts for the Power Supply.

Once you have collected all the parts, you are ready to begin. Of course you will need the Pico PCB itself. Here are the steps to build the power supply:

**Figure 12.19 – The LM7805 Voltage Regulator and Screw Assembly.**

Step 1: Insert the **LM7805** voltage regulator as shown in Figure 12.19, notice that there is a footprint for the part named **U3**, and pin 1 is labeled (to the left). After you insert the LM7805 regulator, go ahead and bend it over gently until the hole in the regulator lines up with the hole in the PCB. Then take the screw assembly and insert the screw thru the bottom of the PCB and fasten the nut on top. Make it good and tight, if you have some pliers you might give it a quarter turn extra. The point is to make a good mechanical contact, so that the regulator's built in heat sink conducts heat to the PCB.

Figure 12.20(a)(b)(c) – The LM7805 after Soldering.

Step 2: Turn the PCB over and solder each of the 3 leads from the LM7805 into the PCB. Make sure to make good contact with the PCB and each lead with the soldering iron. You want these contacts soldered with a good amount of solder since all the current for the system has to go in, out, and to ground thru these 3 conductive pads. The solder joints should look something like that shown in Figure 12.20(a).

Step 3: The leads from the LM7805 must be cut. So go ahead and take your wire cutters as shown in Figure 12.20(b), and cut each lead at the base. When you make the cuts be sure to hold onto the lead with your hand since the force of the cut will probably send the lead flying into the air. Be sure to wear your safety goggles! When you are complete, the underside of the PCB where the LM7805 goes should look like that shown in Figure 12.20(c).

Figure 12.21(a)(b)(c) – The 2.1mm Power Jack Inserted into the PCB.
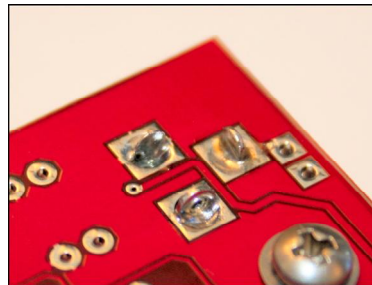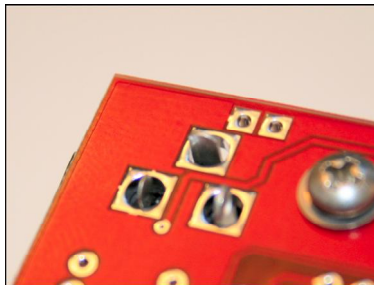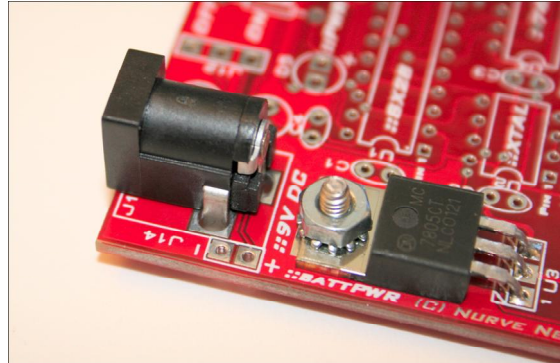
Step 4: Next, we are going to solder in **J1** the **2.1mm** power jack that is used for external power supplies. First step is to insert the component into the board. It will only go one way as shown in Figure 12.21(a). While holding the component, flip the board over and prepare to solder it.

Step 5: Referring to Figure 12.21(b) you can see there are 3 leads coming out from the 2.1mm jack and the leads are pretty big. Moreover, these leads connect to large conductive paths and *"ground fills"* in the PCB, so they are going to be a little harder to solder than normal. Go ahead and take your soldering iron, heat up one of the leads and interface to the PCB and then start soldering, keep the solder moving until the entire hole fills up. Then remove the solder, then remove the soldering iron. Continue the process until all 3 leads are soldered as shown in Figure 12.21(c). Also, if you're wondering why we are filling up the holes, not only do we want a good *electrical contact*, but we want a strong *mechanical contact* since users maybe inserting and removing the power line a lot. Thus, there will be a lot of mechanical stress on this connector hence a good strong solder joint is absolutely necessary for mechanical integrity. In this case the leads from the jack are both for electrical signals and mechanical fastening this is why we care.

**Figure 12.22(a)(b)(c) – The 9V Battery Clip Inserted into the PCB.**
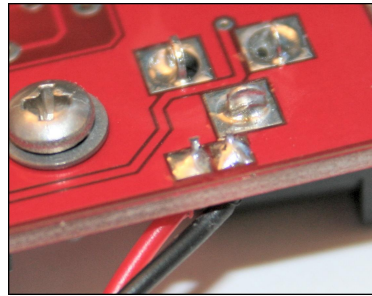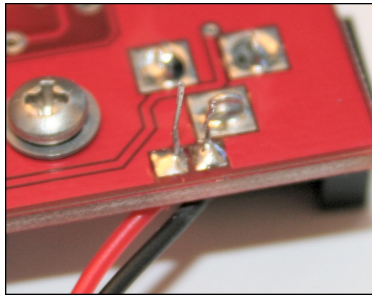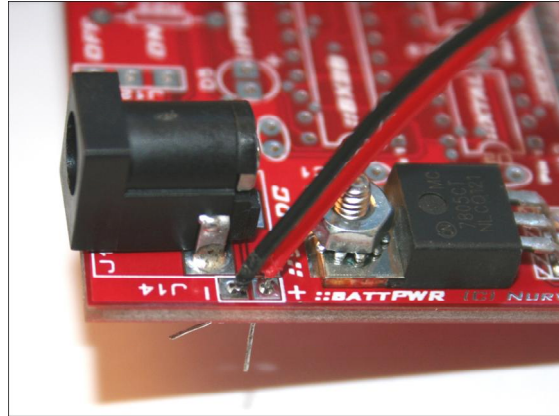
Step 6: The next thing we are going to solder in is **J14** the 9V battery clip into the PCB. The 9V battery clip is over course polarized; *red* for *positive*, *black* for *negative*. These must be inserted into the PCB right next to the 2.1mm power jack. You will see a little "+" and "-" sign to indicate polarity also shown in Figure 12.22(a). Also, before you solder the leads, make sure you twist them tight and *"tin"* them with solder, so they are strong. The leads are made of *stranded* wire and tend to break off with stress, so twisting and tinning can strengthen them more. In any case, insert the battery clips leads as shown in Figure 12.22(a).

Step 7: Solder the battery leads in as shown in Figure 12.22(b) and then cut the excess leads off with your wire cutters as shown in Figure 12.22(c).

Figure 12.23(a)(b)(c) – The Placement of the Power Supply Filtering Capacitors.
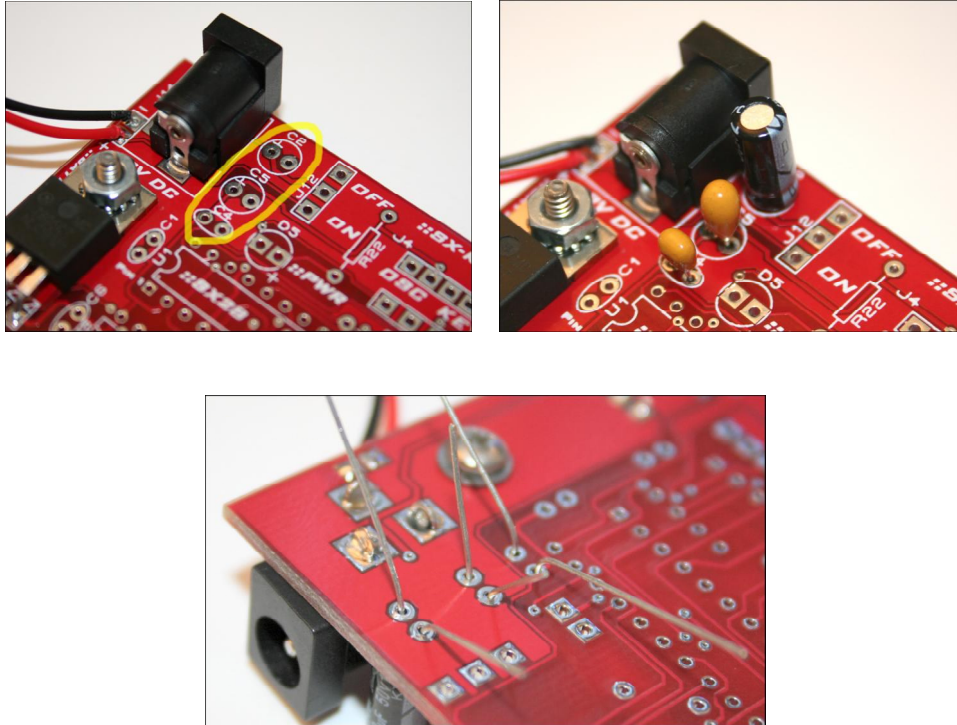
*Figure 12.24 – Close-up of the Electrolytic (left) and Tantalum (right) Capacitors.*
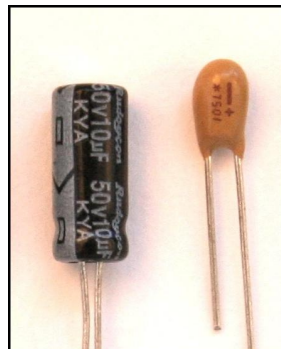
Step 8: Next we are going to solder in the (3) power supplying filtering capacitors. Figure 12.23(a) shows the site outlined on the PCB that they will go in. The capacitors we are soldering are **C2** which is a **10.0uF** electrolytic, **C4** which is a **0.1uF** monolithic, and finally **C5** which is a **1.0uF** tantalum. Both, **C2** and **C5** are *polarized* which of course means it *matters* which way they go in. If you aren't sure how to read the polarity of them take a look at Figure 12.24, it shows a close up of the electrolytic capacitor **C2** (left side) and the tantalum capacitor **C5** (right side). Notice that the *electrolytic* has the *minus* (-) lead marked, while the *tantalum* has the *plus* (+) lead marked. Match the polarization of the capacitors to the PCB silk screen which have a (+) to the left of both the foot prints of **C2** and **C5**. Lastly, **C4** is *monolithic*, thus it has no polarity, so you can insert it either way, I prefer to insert it with the label facing the front of the board though. In any event, go ahead and insert all (3) capacitors into the PCB as shown in Figure 12.23(b), turn the board over and bend the leads over as shown in Figure 12.23(c).

Step 9: You are ready to solder the capacitors, the point of bending the leads over is so you can solder them without them falling out. Solder each capacitor in and then when you are complete cut all the excess leads off. Remember to hold the leads as you cut them, so they don't go flying into the air or your eyes!

Figure 12.25(a)(b)(c) – The Power Switch, LED, and Current Limited Resistor.
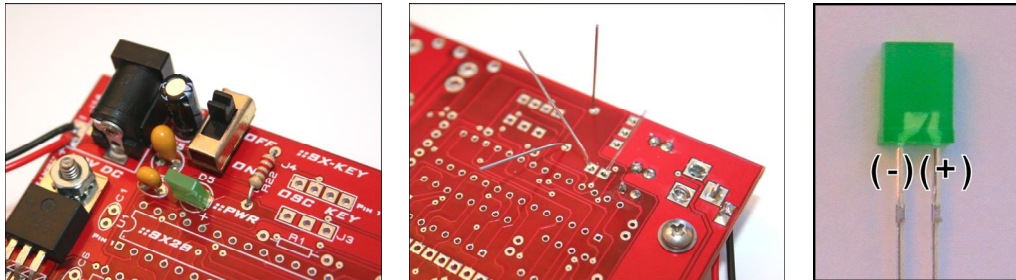
Step 10: Next we are going to insert **J12** the power switch, **D5** the green LED, and **R22** the 200-220 Ohm current limiting resistor. Figure 12.25(a) shows all the components inserted into the PCB. Also, make sure you use one of the 200-220 Ohm resistors from the basic Pico kit and *not* the Pico Add-On, since either kit might have 200 or 220 Ohm resistors, but to keep things consistent we want to use all 200 Ohms in one place and 220 in another if possible. So keep the 200 Ohm resistors from the Pico PCB Add-On Kit separate until we get to the 7-Segment display soldering. In any case, insert the components into the PCB. The switch might not stay in the board, if not press harder. You just need it to stick long enough to turn the board over and solder it. Also, remember LEDs are polarized, that is they are really diodes with a cathode (-) and an anode (+) and must be inserted correctly. If you look at the PCB where the LED inserts at D5 (circular silk screen for generic LEDs) you will see a little (+) to the lower right, this is where the positive lead of the LED goes. If you don't know which lead is (+) and (-) on LEDs there are a few ways to tell. First, you can always try the LED with 2-3V and see which way it lights up. This won't hurt the LED and you can be certain. The 2nd way is to look at the length of the leads from the LED, the short lead is usually (-). However, this can be wrong if someone cut the leads or a manufacturer reversed them. The 3rd way to tell which lead is which is to look *inside* the LED itself, the larger upside triangular platform is usually (-) in all cases, and the smaller side is of course (+). Figure 12.25 (c) shows this graphically. But, once again, I have actually seen LEDs that are reversed as well internally. So there is no substitute to testing your LEDs or knowing the exact manufacturer and thus lead orientation. But, with the Pico Kits the LEDs do follow these rules, so look inside the LED for the larger upside down triangle and this will be the negative (-) lead. Figure 12.25(b) shows the components on the underside of the PCB, simply solder and then cut the excess leads as usual.

Congratulations! You have finished the power supply, not let's test it.

Figure 12.26(a)(b) – The XGS Pico PCB Edition Powered up with the 9V Battery and External Power Supply.
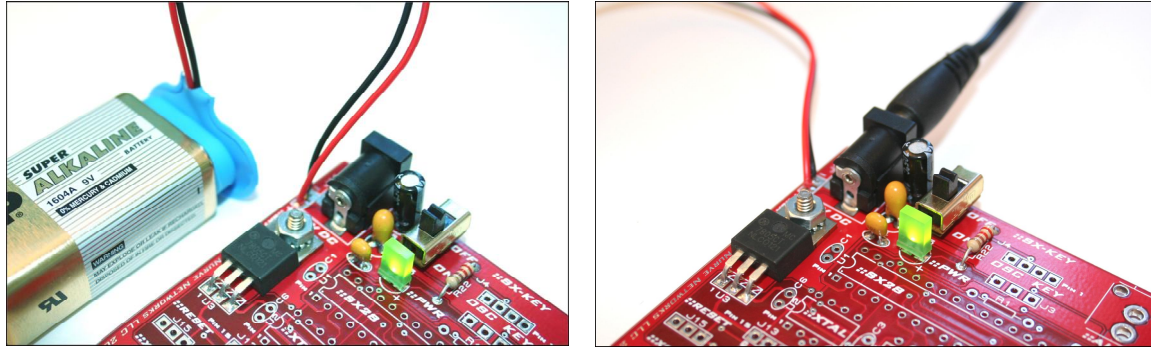
Step 11: Let's test the power supply and voltage regulation of the system now. First, make sure that the power switch is **OFF**. And blow away any loose solder, or metallic particles from the board with compressed air (preferably). Also, visually inspect all your solder connections to make sure there are no shorts. If all looks good, go ahead an clip in the 9V battery and turn the power switch **ON**. You should see the green power LED turn on and all should be well. This is shown in Figure 12.26(a). If you have an external 9V 300+ mA power supply with a 2.1mm barrel connector with tip (+) and ring (-), then go ahead and give it a try as well. First make sure to *unplug* the 9V battery. This is shown in Figure 12.26(b). Lastly, if you have a multimeter, put it on DC voltage and test the voltage at pin 3 of the LM7805 and ground (pin 2), it should be around 4.95-5.0V if all is well.

Go ahead and turn off the XGS Pico, unplug all the power sources, and let's continue. From now on, we are going to go a little bit faster, and in order of convenience to make soldering easiest.

Figure 12.27(a)(b)(c) – The 4-Pin Programming Header, Mode Switch, and Pull-up Resistor.
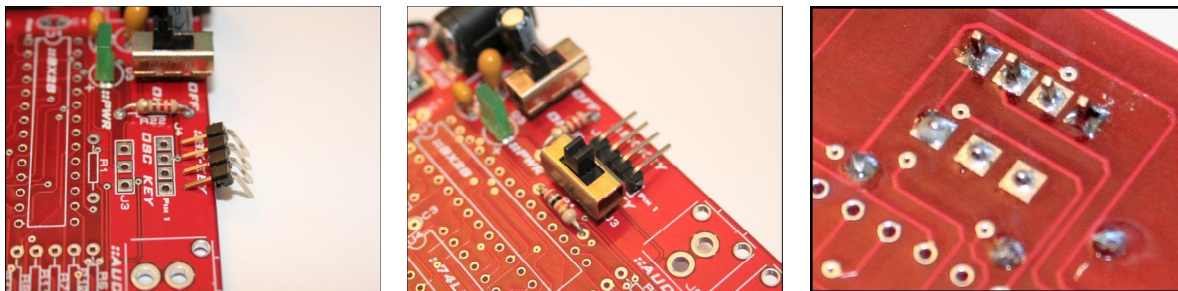
12.4.2 Assembly Part 2: SX-Key Programming Port, Programming Mode Switch, and Reset Pull-up

In this part, we are going to solder in **J4**, the 4-pin right angle programming header for the SX-Key. **J3** the programming mode switch, and finally **R1**, the **10K** Ohm pull-up resistor for the SX28 reset circuit. Table 12.6 shows the parts and designators for reference.

Table 12.6 – Parts for Pico PCB Programming Port, Mode Switch, and Reset Pull-up.

Reference Designator(s)	Description
J4	4-Pin Right Angle Header for SX28 Programming.
R1	10K Ohm Axial, 0.25W 5% Resistors.
J3	SPDT Micro Switch 3mm Spacing.

Step 1: Let's begin with the 4-pin header **J4**. Referring to Figure 12.27(a), you can see the header next to its insertion point. The important thing is to *solder* in the *short leads* and leave the long leads for the SX-Key as shown in the figure. Once you have that part inserted, you can insert the other (2) components the **switch J3** and the **10K resistor R1** as shown in Figure 12.27(b).

Step 2: Turn the PCB over and solder the components. You may have to hold **J4** with your hand or lay the PCB on a piece of foam to keep it in place. Also, you can try soldering one lead from the top of the PCB with component **J4** to keep it in place, then turn over the board and solder all the rest of the leads. When complete soldering cut all the leads as shown in Figure 12.27(c).

Figure 12.28 – 0.1uF Decoupling Capacitors Inserted at C1, C3, and C6.
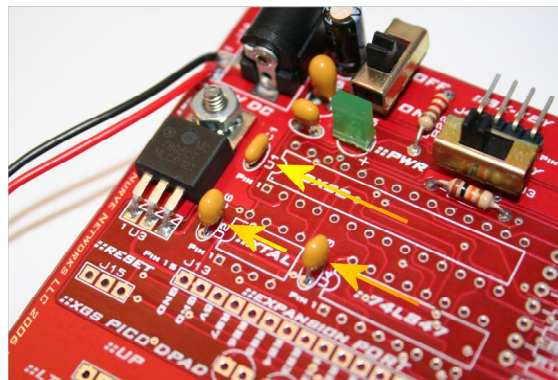
12.4.3 Assembly Part 3: Integrated Circuit De-Coupling Capacitors

In this section we are going to solder in the integrated circuit *"de-coupling"* capacitors. The de-coupling capacitors help filter out power supply fluctuations, plus give each chip a local supply of power when high currents are demanded. Figure 12.28 shows the (3) capacitors inserted into the PCB at **C1**, **C3**, and **C6**. They are all **0.1uF** monolithic (non-polarized), so you can insert them anyway you like. However, I suggest that you insert them all the same way with the writing facing the same direction just to be consistent. Table 12.7 lists the parts and designators for reference.

TIP	Each of the 0.1uF caps should have a small "104" written on them, this mean 104 nano-Farads which is 0.1uF. This is the convention used to indicate the value of small ceramic capacitors like this.

Table 12.7 – Parts for Pico PCB Programming Port, Mode Switch, and Reset Pull-up.

Reference Designator(s)   **Description**

C1, C3, C6   0.1uF Monolithic, 0.1" Lead Spacing (non-polarized).

Step 1: Insert the capacitors as shown in Figure 12.28, bend the leads on the under-side of the PCB and solder all the components in. When complete, cut the excess leads from the capacitors, turn the PCB over and make sure all the caps are nice and straight.

Figure 12.29(a)(b)(c) – The SX28 IC Socket.
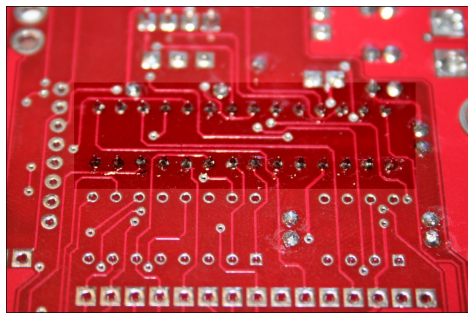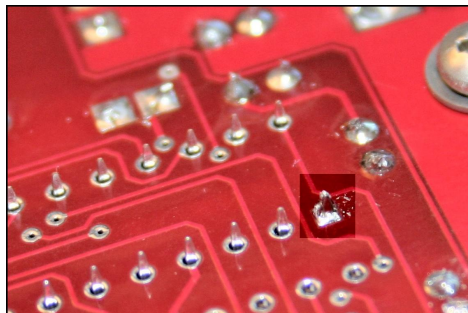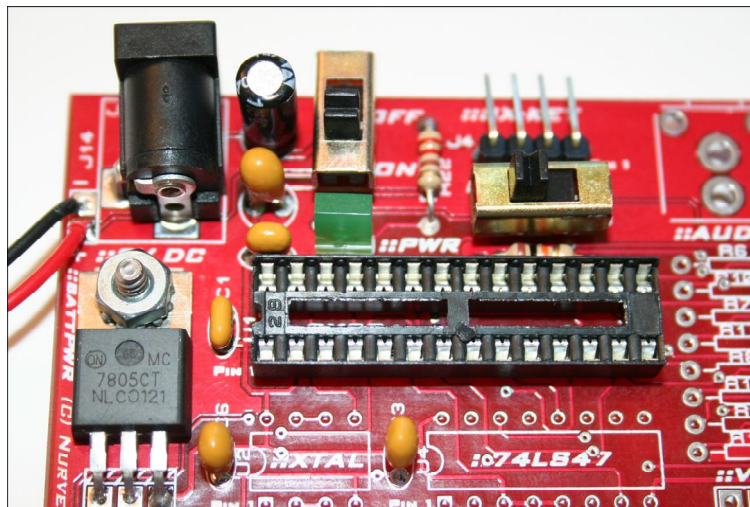
12.4.4 Assembly Part 4: The Integrated Circuit Sockets

In this part, we are going to solder all the integrated circuit sockets into place. There are (3) of them; one for the **SX28** itself **U1** (28-pin), one for the **80.000 MHz** or **78.750MHz** oscillator at **U2** (8-pin) and finally the 16-pin socket for the **74LS47** that goes into **U4**. Table 12.8 lists the socket parts.

Table 12.8 – Parts for Pico PCB IC Sockets.

| Reference Designator(s)		Description
(1)	U1 Socket	28-Pin Dip Socket 0.3" Width for SX28.
(1)	U2 Socket	8-Pin Dip Socket 0.3" Width for Oscillator.
(1)	U4 Socket	16-Pin Dip Socket 0.3" Width for 74LS47.

Note: All these parts come from the Pico Add-On Kit.

Step 1: First, locate the 28-pin 0.3" wide IC socket and insert it into the SX28 footprint at **U1** in the PCB as shown in Figure 12.29(a). Notice, the orientation of the *"pin 1 tab notch"* to the left. It is convention to insert IC sockets with the tab notch at pin 1.

Step 2: Turn the PCB over while holding the IC socket with your fingers or placing the board on a piece of foam. Now, solder a single corner lead to hold the socket in place. Make sure the socket is all the way flush with the PCB. Figure 12.29(b) shows this and highlights the single soldered pin.

Step 3: Continue soldering all the remaining contacts, be sure not to short contacts or to solder them for more than 2-3 second since the heat can damage the socket itself and melt it if you hold the soldering iron on a contact more than 3 seconds or so. When complete you should have nice silvered solder joints as shown highlighted in Figure 12.29(c).

Figure 12.30 – The Remaining 8-pin and 16-pin IC Sockets.
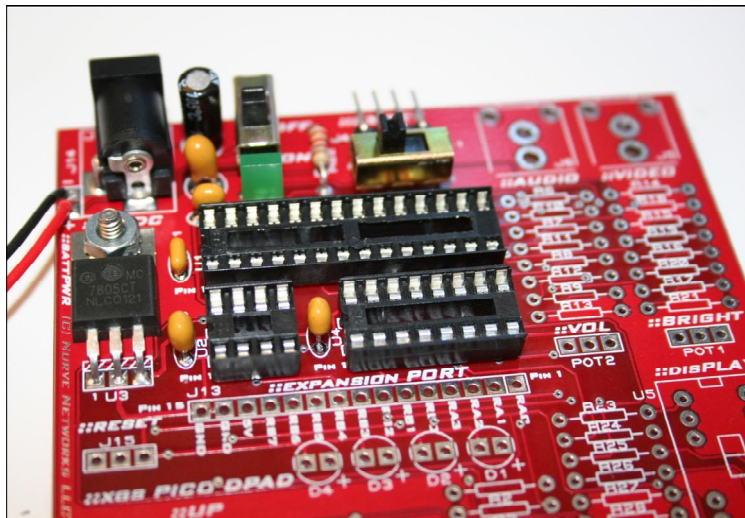
Step 4: Now Insert the remaining (2) IC sockets into the footprints for **U2** (8-pin oscillator) and **U4** (16-pin 74LS47). Make sure to keep the **pin 1** orientation notch to the **left**. Your board should look like the image shown in Figure 12.30. Turn the board over, solder (1) pin on each socket and then adjust and make sure the sockets are flush with the board and solder all the remaining pins. You might find that you can only do one socket at a time, this is fine. Also, when you are done, review all your solder joints, its easy to miss one or make a cold solder joint with so many to do. Remember, the entire soldering cycle each pin should take about 2-3 second each. Also, review the SX28 socket while you're at it.

Figure 12.31 – Insertion of the 15-Pin Header and SX28 Reset Switch.
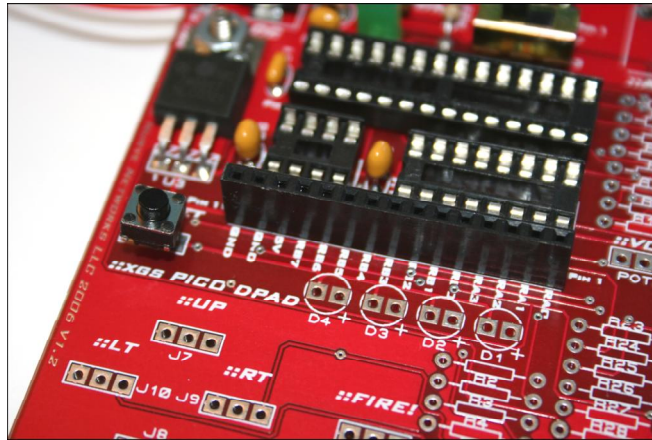
12.4.5 Assembly Part 5: 15-Pin Expansion Header and Reset Switch

In this part we are going to solder in the 15-pin expansion header **J13**, as well as the SX28's momentary reset switch **J15**. Figure 12.31 shows the components inserted into the PCB. Table 12.9 lists the parts once again for reference.

Table 12.9 – Parts for Pico PCB 15-Pin Expansion and Reset Switch.

Reference Designator(s)	Description
J13	15-Pin 0.1" SIP header to export out pins.
J15	SPST Momentary Micro Switch 2-Lead.

Step 1: Insert the momentary switch into footprint **J15** (use the black one, so when we get to the Dpad they will all be white) and bend the leads over behind the board. Now, solder the momentary switch in.

Step 2: Insert the 15-pin header into **J13**, notice that it has no orientation, but *"pin 1"* is to the **right** side of the PCB, thus if there is any kind of pin 1 notch or marketing on your particular 15-pin you might as well align it with pin 1 on the board. Hold the 15-pin header into the board and solder a single pin, then make sure to align the header and solder the remainder of the pins. The one thing you don't want to do with these headers is solder pins half way in, looks unprofessional. The idea is to get the component nice and flush to the PCB board.

Figure 12.32(a)(b) – The LED Indicators inserted into the PCB.
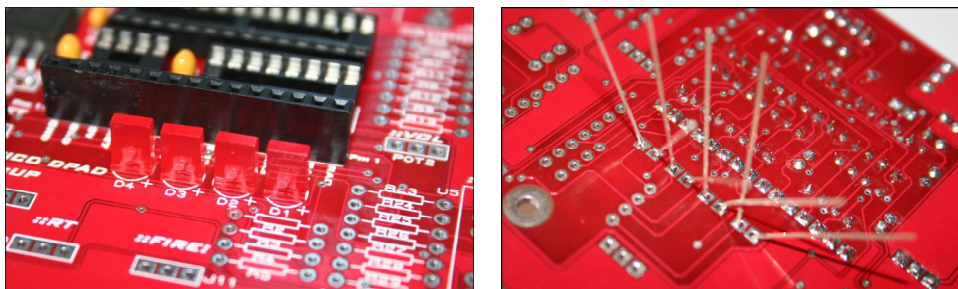
12.4.6 Assembly Part 6: LED Indicators

Now we are going insert the (4) red LED indicators into **D1-D4** on the PCB. The locations of the LEDs are right under the 15-pin expansion header as shown in Figure 12.32(a). Pay close attention to the polarization of the LEDs, the (+) leads are to the right of each footprint. If you look closely at the figure you can actually see inside the LEDs to confirm you have them in correctly. The parts for this step are shown in Table 12.10.

Table 12.10 – Parts for Pico PCB 15-Pin Expansion and Reset Switch.

Reference Designator(s)	Description
D1-D4	Red LEDs (Light Emitting Diode).

Step 1: Insert the (4) red LEDs into **D1-D4** as shown in Figure 12.32(a). As you insert each LED, bend the leads over a bit behind the board to keep the LEDs in the PCB. This is shown in Figure 12.32(b). Solder the LEDs in and cut the excess leads with your wire cutters. Take you time with this step, so you get all the LEDs nice and flush with the PCB.

Notice, that we are only soldering the LEDs in this step, *not* their current limiting resistors we will come back to this in a moment…

Figure 12.33 – The Directional Gamepad.
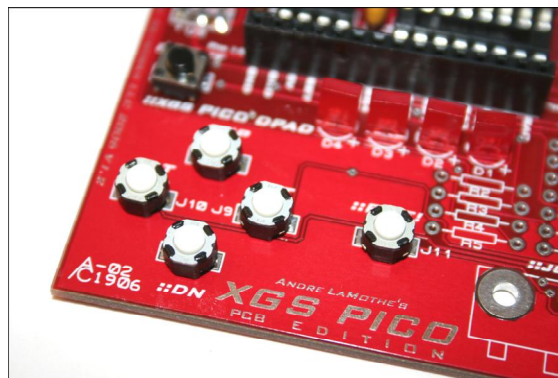
## 12.4.7 Assembly Part 7: The Directional Gamepad

The directional gamepad or **"Dpad"** for short is one of the coolest things about the XGS Pico PCB Edition. Game controllers are expensive, so we decided to just build one in! Of course, holding a square piece of laminated PCB isn't as nice as a new XBOX 360 controller, but it's nice to have the controller built it. The Dpad has up, down, left, right, and fire. Figure 12.33 shows the momentary switches inserted into the PCB at **J7-J11**. These momentary switches are all from the Pico Add-On kit and should all have white buttons (so they look uniform). Table 12.11 shows the exact part and descriptions.

Table 12.11 – Parts for Pico PCB 15-Pin Expansion and Reset Switch.

Reference Designator(s)	Description
J7-11	SPST Momentary Switch, 0.3" Spacing.

Step 1: Insert the momentary switches as shown in Figure 12.33, you will notice that each footprint **J7-J11** has (3) holes, but each switch only has (2) leads. The switch needs to go in the *outside right/left holes*, the middle hole is unused for each switch. Solder all the switches into place, make sure they are all flush and not rotated. Then cut the excess leads from each.

Figure 12.34(a)(b) – LED Indicator Current Limiting Resistors.
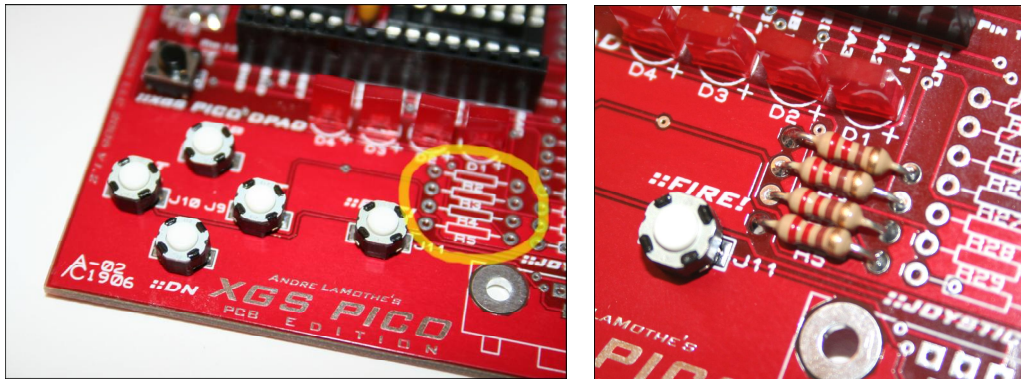
12.4.8 Assembly Part 8: LED Indicator Current Limiting Resistors

The LED indicators also need current limiting resistors, so they don't burn out. These resistors are selected to be around 200-220 Ohm which is either (red, black, brown – 200 Ohm) or (red, red, brown – 220 Ohm). And go into the PCB at **R2-R5**. Figure 12.34(a) shows the location of these current limiters and Figure 12.34(b) shown them after insertion. Note that you should use the **200-220** Ohm resistors from the **basic Pico Edition Kit**, don't use the (10) 200 Ohm resistors from the Pico PCB Add-On kit. The point being to keep values as similar as possible in sub-sections. Table 12.12 lists the parts for reference.

Table 12.12 – Parts for Pico PCB LED Indicator Current Limiters.

Reference Designator(s)	Description
R2-R5	200-220 Ohm Axial 0.25W 5% Resistors.

Step 1: Insert the (4) resistors into the PCB at R2-R5, bend the leads over under the PCB and then solder all the resistors into place.

Step 2: Cut all the excess leads from the (4) current limiting resistors, make sure to hold each lead as you cut it, so it doesn't go flying.

Figure 12.35(a)(b) – 7-Segment Display Current Limiting Resistors.
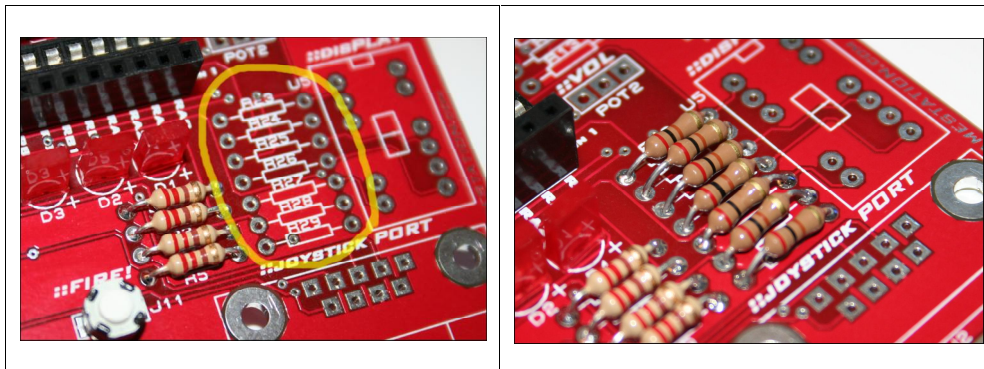
12.4.9 Assembly Part 9: 7-Segment Display Current Limiting Resistors

The 7-Segment display (**U5**) needs current limiters serially from the 74LS47's (**U4**) outputs. We are going to use the 200 Ohm (red, black, brown) resistors from the Pico PCB Add-On kit (these should always be 200 Ohm in the kit and won't vary like they do in the basic Pico Kit). Anyway, the resistors go into the locations **R23-R29** as indicated in Figure 12.35(a). Table 12.13 lists the parts for reference.

Table 12.13 – Parts for Pico PCB 7-Segment Display.

Reference Designator(s)	Description
R22-R29	200 Ohm Axial 0.25W 5% Resistors.

Step 1: Insert the (7) resistors into the PCB at **R23-R29**, bend the leads over under the PCB and then solder all the resistors into place as shown in Figure 12.35(b).

Step 2: Cut all the excess leads from the (7) current limiting resistors, make sure to hold each lead as you cut it, so it doesn't go flying.

Figure 12.36(a)(b)(c) – The Video and Audio R2R Ladders.
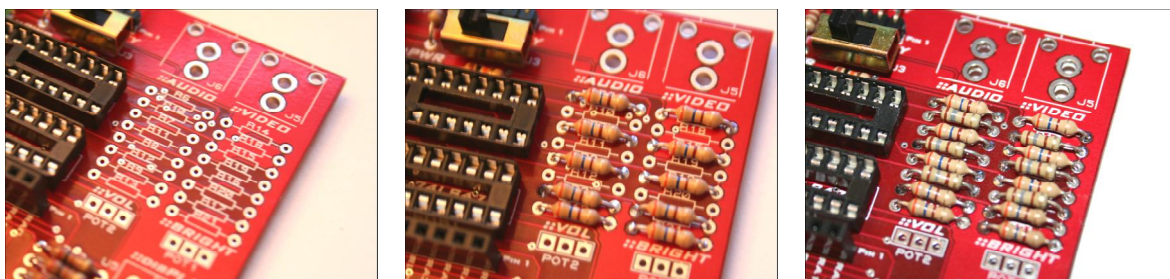
12.4.10 Assembly Part 10: The Video and Audio R2R Ladders

In this part, we are going to assemble the **R2R ladders** which are responsible for the digital to analog conversion for the video and audio generation. Referring to Figure 12.36(a), this is the PCB location where the resistor networks are going to be placed. We are going to build both ladders at the same time and stuff **R14-R21** as well as **R6-R13**. We only need two values of resistors; **180 Ohm** (brown, grey, brown) and **360 Ohm** (orange, blue, brown). These components are all from the basic Pico Kit and not the Add-On. Also, you might have slightly different resistance values like 182 Ohm and 365 Ohm if you happen to have precision resistors. The point is, use the values that are closest to 180 and 360 Ohm. Table 12.14 lists the parts for reference.

Table 12.14 – Parts for Pico PCB Video and Audio R2R Ladders.

Reference Designator(s)	Description
R6-9, R13, R14-17, R21	360 Ohm Axial, 0.25W 5% Resistors (10 pieces).
R10-12, R18-20	180 Ohm Axial, 0.25W 5% Resistors (6 pieces).

Step 1: First we are going to solder in the (10) **360 Ohm** resistors into locations **R6-R9**, as well as **R13** along with **R14-R17**, and finally **R21**. Take a look at Figure 12.36(b), it shows these resistors inserted into the board. Make sure you solder all the resistors flush into the PCB and you cut the leads off nice and short.

Step 2: Now, the spaces in the R2R ladders that we didn't insert 360 Ohm resistors are where the remaining **180 Ohm** resistors go. There are (6) of them at **R10-12** and **R18-20**. Go head an insert those components into your PCB now, solder, and cut the leads. The results should look like Figure 12.36 (c).

Figure 12.37 – The Pico PCB so far.
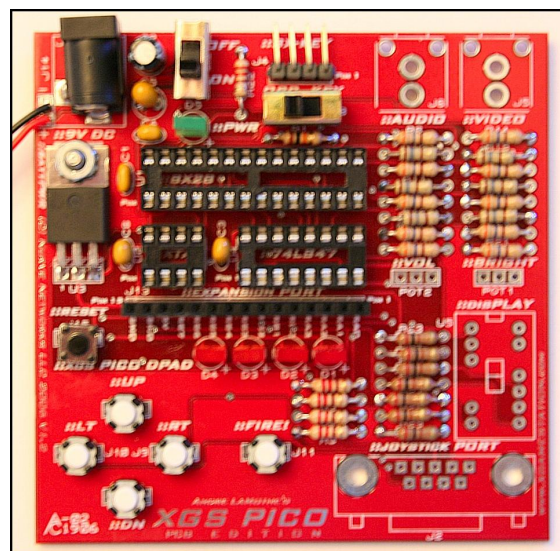
Step 3: Review the Pico PCB, it should look something like that shown in Figure 12.37. Only the 7-Segment display, POTs, RCA connectors and DB9 connector are left to solder. We are almost done!

Figure 12.38(a)(b)(c) – Video and Audio RCA Connectors.
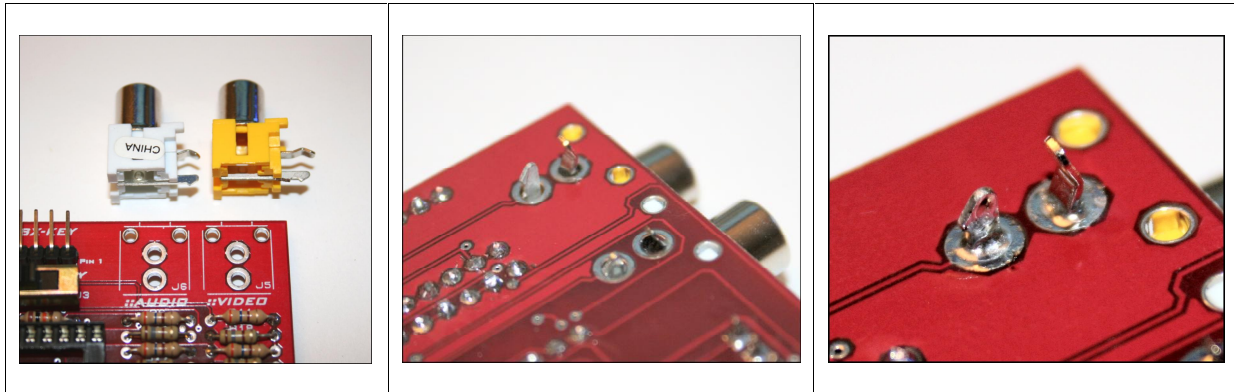
12.4.11 Assembly Part 11: Video and Audio RCA Connectors

In this part, we are going to solder in the white and yellow RCA connectors. Figure 12.38(a), shows a close up of the connectors lying next to each of their insertion points at **J5** (video/yellow) and **J6** (audio/white). There is nothing functional about the white or yellow, so you could swap them if you wanted, but since A/V cables are color coded with red/white as audio and yellow as video, we might as well follow this convention as best as possible. If you happen to have a kit that has two red, or two white, or even one blue and one green RCA connector they will work just as well. Table 12.15 lists the parts for reference.

Table 12.15 – Parts for Pico PCB Video and Audio RCA Connectors.

Reference Designator(s)	Description
J5	Video RCA JACK R/A YELLOW PCB mount, thru hole.
J6	Audio RCA JACK R/A WHITE PCB mount, thru hole.

Step 1: Insert each of the RCA connectors into their mounting holes at J5 (video/yellow) and J6 (audio/white). You will find that you have to align the plastic seating pins at the front of the connectors as well and **"click"** the connectors in. Don't press too hard, and you might have to **"wiggle"** the connector in to get it seated properly. Once you have both connectors in then you should turn the PCB over and see the lead pins sticking out as shown in Figure 12.38(b).

Step 2: The lead pins for the RCA connectors are not only for electrical conductivity, but for mechanical stability as well, that's why they are so big. The leads along with the plastic seating pins insure that the repeated insertion/removal of RCA cables doesn't rip the RCA connector off the PCB. With this in mind, solder the connectors to the PCB and make sure to fill the holes up completely as shown in Figure 12.38 (c). Do **NOT** solder the plastic seating pins! They are there for alignment only.

Figure 12.39 – The Audio and Video 500 Ohm Adjustment POTs Inserted into PCB.
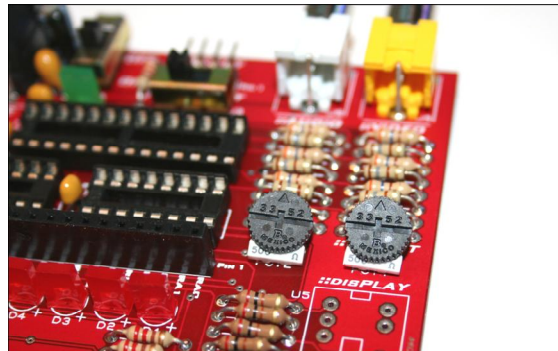
12.4.12 Assembly Part 12: The Audio / Video Adjustment POTs

We are almost here! Now, let's solder in the (2) **500 Ohm** Audio/Video Adjustment potentiometers at **POT1** and **POT2** (right under the R2R ladders). Figure 12.39 shows the insertion of these components. **Note:** you might have a kit with 1K or 2K POTs, these will work fine as well. Table 12.16 lists the parts for reference.

Table 12.16 – Parts for Pico PCB Video and Audio POTs.

Reference Designator(s)	Description
POT1, 2	500 Ohm Potentiometers.

Step 1: insert the (2) 500 Ohm potentiometers into the PCB at POT1 and POT2 a shown in Figure 12.39. You can actually insert them facing forward or backward, but forward as shown in the figure is more convenient, since the other way they might make contact with the 7-segment LED display. Each POT has (3) leads, so make sure all (3) leads are soldered properly.

Step 2: Go ahead and set the value of each POT to the half-way mark, meaning, turn the little black dial until the arrow on each dial is pointing straight up this means 50% value or 250 Ohms will be the starting point. A good place to set them at initially.

Figure 12.40(a)(b) – The 7-Segment LED Display.
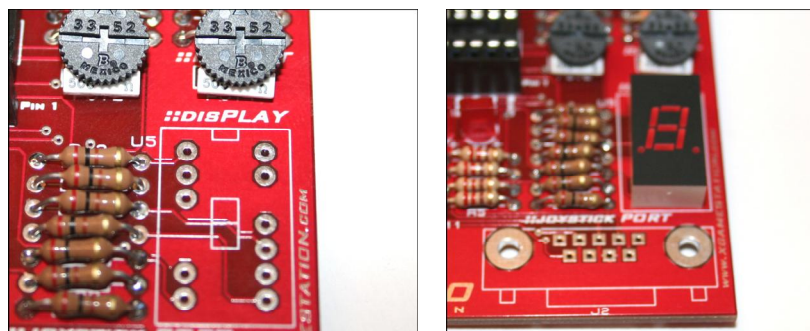
12.4.13 Assembly Part 13: The 7-Segment Display

Now let's solder in he 7-segment display at U5. The 7-segment display that is used in the Pico PCB is a common anode display with 2-decimal points. Although, the XGS doesn't use either of the decimal points for rendering. Figure 12.40(a) shows the location where the display should be inserted into the PCB, as you can see there is a little foot print with an "8" on the PCB and a rectangular notch painted on the foot print as well to indicate "top". Pin 1 is to the top left for reference. Table 12.17 lists the parts for reference.

Table 12.17 – Parts for Pico PCB ICs for Final Insertion.

Reference Designator(s)	Description
U5	LT312 7-Segment Common Anode Display (red).

Step 1: Insert the 7-segment display as shown in Figure 12.40(b), it will only go in one way due to the pinout. Flip the PCB over and solder.

Figure 12.41(a)(b)(c) – The DB9 Male Atari 2600 Joystick Connector.
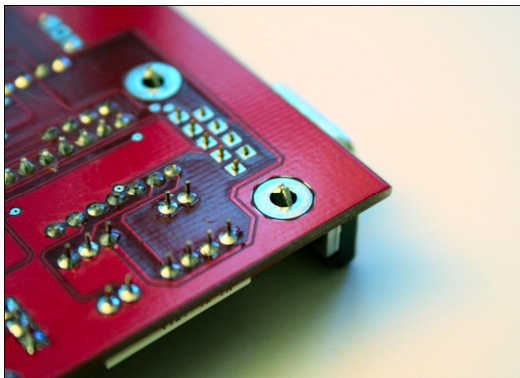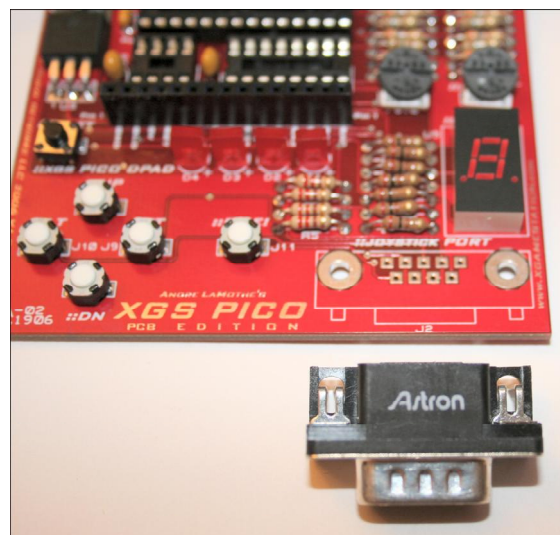
12.4.14 Assembly Part 14: The Atari 2600 Compatible DB9 Connector

This is the last step to the soldering of the PCB, so we are almost done! Take a look at Figure 12.41(a), here you see the DB9 make solder thru-hole connector set near its foot print on the PCB. This is component **J2**, locate it and follow the steps. Table 12.18 lists the parts for reference.

Table 12.18 – Parts for Pico PCB DB9 Male Atari Joystick Interface.

Reference Designator(s)	Description
J2	DB9 Male Connector for Joysticks PCB Mount.

Step 1: Insert the DB9 male socket into the PCB at J2. The socket has two mechanical inserts as well, so take your time. When the socket inserts into the PCB, you will hear a nice solid click. Turn the PCB over, you should see something like that shown in Figure 12.41(b).

Step 2: Solder the socket into place, first all the signal leads and then the mechanical inserts. Make sure to fill the entire holes up with solder for the mechanical inserts since these are used to hold the socket into the PCB. There will be a lot of stress on the connector when joysticks are inserted and removed. The soldering should look like that shown in Figure 12.41 (c).

Figure 12.42 – The XGS Pico with the Remaining Parts.
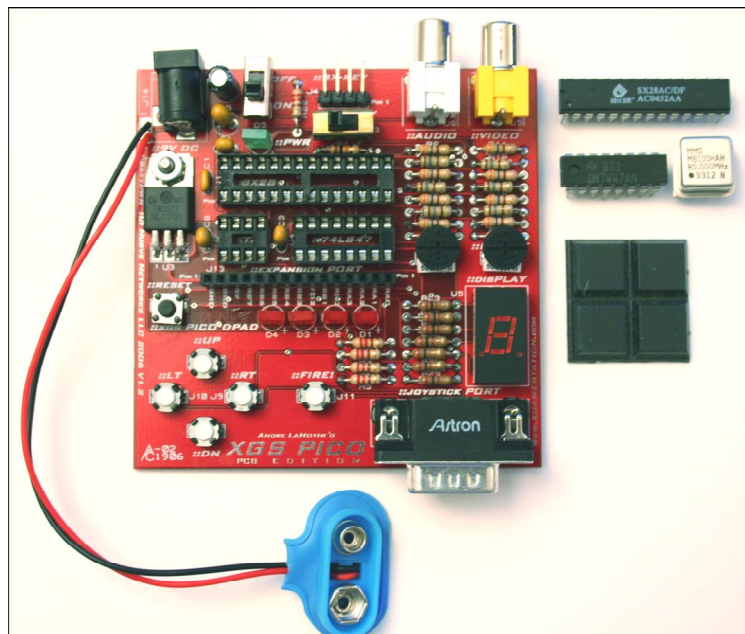
Figure 12.43 – The Final Parts inserted into the XGS Pico Edition.
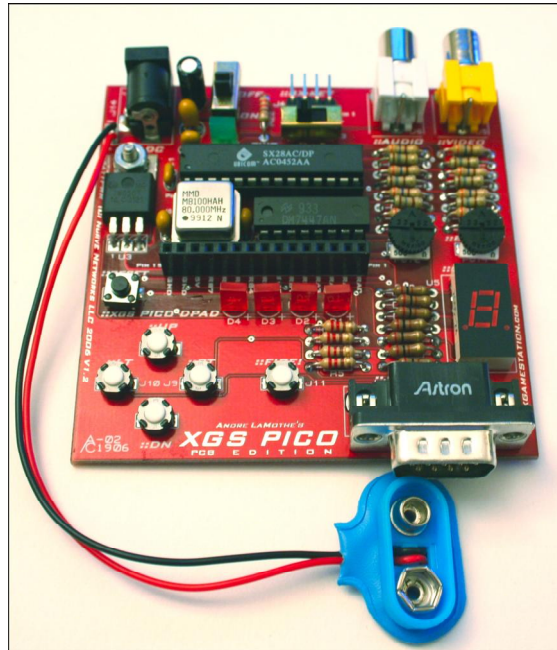
12.4.15 Assembly Part 15: Inserting the ICs and Oscillator

We are nearly complete, the last steps are to insert the ICs into the sockets and put the rubber feet on the underside of the PCB. Take a look at Figure 12.42, it shows the Pico PCB so far with the remaining parts to the right. There is the SX28, 74LS47, and 80.000 MHz oscillator along with (4) rubber feet. In this part, we are going to insert the chips into the sockets. Table 12.19 lists the parts for reference.

Table 12.19 – Parts for Pico PCB ICs for Final Insertion.

| Reference Designator(s)	Description
U1	Ubicom SX28 Microcontroller DIP 28.
U2	80.000 MHz 8-Pin DIP, Half Size Oscillator.
U4	74LS47 7-Segment Display Driver.

Step 1: Insert the SX28 processor into the 28-pin socket at U1 on the board. Make sure to align pin 1 of the SX28 with pin 1 of the socket, lower left hand corner. Be very careful inserting the SX28, if it won't go in, try straightening the pins a bit on a flat surface and bending them inward slightly, so they go into the socket more easily. Once the chip inserts into the socket, give it a good push to make sure it's seated. Inspect the pins to make sure all 28 are in the socket and there are no bent pins. If there are bent pins, gently remove the SX28 by prying under it with a small flat head screwdriver, each side, until it comes free. Then bend the pin flat with some pliers and try again. Refer to Figure 12.43(b) to see what the inserted chip looks like.

Step 2: Insert the 74LS47 into the socket at U4. Again, watch the orientation of the chip, pin 1, should be to the bottom, left.

Step 3: Finally, insert the 80.000 MHz oscillator into the socket at U2. Be careful that you locate pin 1 of the oscillator properly, it might be very faint, but you will see a little dot either printed or impressed into the metal. This

indicates pin 1 which once again inserts into the bottom left of the socket. Note: you might find that the oscillator stands off the socket about a 1/4th or 1/8th of an inch, you might want to clip the leads a 1/8th of an inch and reseat the oscillator. Just don't press too hard, you will bend the leads on insertion! Review, Figure 12.43, your Pico Edition should look like this which is the final XGS Pico Edition PCB.

*Figure 12.44(a)(b) – Adding the Rubber Feet to the Pico Edition.*
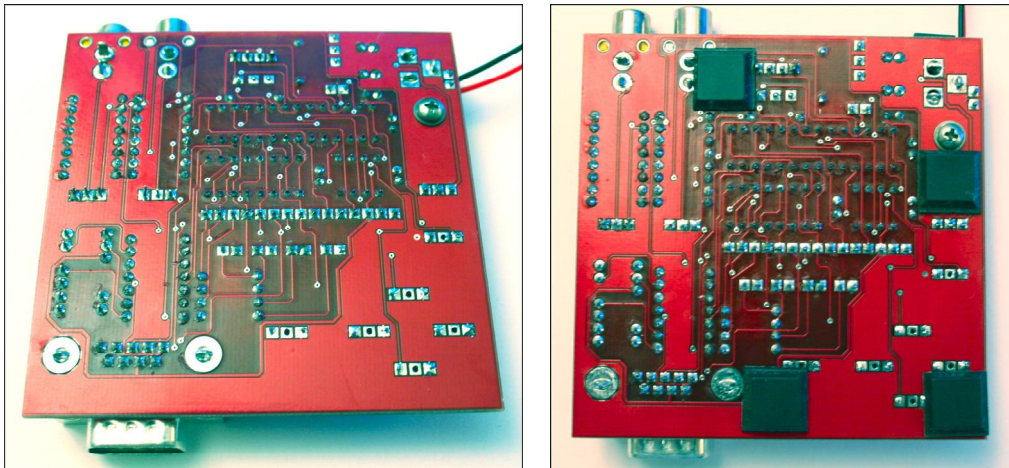
12.4.16 Assembly Part 16: Adding the Rubber Feet

There is one last step to finishing the XGS Pico Edition PCB and that is adding the rubber feet! Take a look at Figure 12.44(a) which is the underside of the PCB. The idea here is that we want to place the feet on the (4) corners of the PCB ideally, but there are various mechanical leads protruding out, so this makes it hard. Thus, we can settle for applying the (4) rubber feet to any flat areas on the board that distributes the weight reasonably. Or you can cut and mold the feet and try and put them on the four corners of the board. Figure 12.44(b) shows one possible configuration for the rubber feet.

The idea is that the feet keep the leads under the PCB from touching the surface and thus helps the system not short if you have happen to place it on a conductive surface like the top of your DVD player etc. Anyway, no steps here, just add the feet to the board and you're done!

Figure 12.45 – Hooking the XGS Pico Up for Testing.
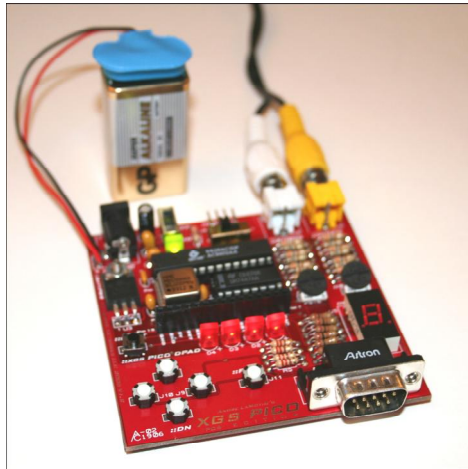
12.5 Testing the XGS Pico Edition PCB

Congratulations! Your XGS Pico PCB should be completely assembled if you made it here. Now, we are going to power the Pico up and make sure it all works. Here's a series of initialization steps to perform:

Step 1: Inspect the PCB once again, check for any debris, metal shavings, solder, or short circuits.

Step 2: Insert the A/C cable into the A/V ports on the Pico Edition PCB, remember yellow goes to video and white goes to audio on your TV set.

Step 3: Make sure the main **power switch** is **"OFF"** at **J12** (up position) and that the **mode switch** is in **"OSC mode"** at **J3** (to the right).

Step 4: Insert your external 9V power supply or 9V battery as shown in Figure 12.45.

Figure 12.46 – The Pre-Loaded "Racer City" Demo Running on the XGS Pico PCB.
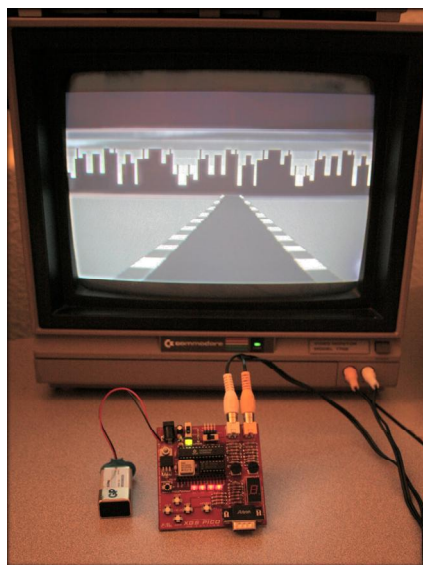
Step 5: Make sure the TV monitor is **ON** and go ahead and turn the XGS Pico Edition **"ON"** with the main power switch. You should see something like the image shown in Figure 12.46 (at the time of this writing there is a racing demo pre-loaded on the SX28). And the green power LED should be illuminated. The game demo *"Pico Racer City"* that was pre-loaded as firmware is located on the CD here for reference:

CDROOT:\XGSME_HW_CD\XGME_Sources\racer_city_pico_01.SRC

Step 6: You should see all (4) of the LED indicators on, but the 7-segment display should show nothing. The image on the screen should be stable and not moving. Go ahead and adjust the video brightness with labeled *"::Bright"* with the potentiometer at *POT1* until the image is pleasing on your screen. It should be **black and white** with shades of gray, this is *not* a color demo.

Step 7: If all is well, go ahead and use the Dpad button to drive the racing game. Forward will accelerate you, while backward or down will decelerate you. Right and left turn the 1st person car. The fire button does nothing. As you drive the simulation you should see the (4) LED indicator lights change, as the bit pattern changes on the LED indicators any binary number represented by them from 0…9 will show up on the 7-segment display.

Troubleshooting: If you don't see anything or there are problems with the control then there are a number of possibilities to test:

ü   Make sure the PCB has power.

ü   Make sure the 80.000 MHz OSC is inserted into the board in the correct way (pin 1, bottom left of socket).

ü   Make sure the **SX mode switch** is in *"OSC"* mode to the *left*, this gates the oscillator into the SX28 chip.

ü   Check that you have the audio / video feeds plugged in properly and that you have audio going into audio on your TV and video going into video. Make sure the TV is set to *"Video Input"* mode.

ü   It's possible that thru static discharge or previous experiments you damaged or erased the pre-loaded racing demo firmware. If this is the case then you will have to load the firmware again into the SX28 with the SX-Key programmer (sold separately). See section 12.6 for more information.

ü   Finally, if all else fails re-check all your soldering, check all your chips, make sure you didn't bend any pins or something isn't connected well.

ü   Contact us at **support@nurve.net** or log onto our website at **www.xgamestation.com** and post in the forums and ask for help.

Switching the Oscillator

The majority of XGS Pico demos are written for the **80.000 MHz** oscillator, but as noted in Chapter 11.16 on the Pico Edition, the kit also comes with a 78.750MHz oscillator as well which is a multiple of the NTSC color burst. Use this oscillator when you want to write your own color demos and make timing calculation easier since each clock on the SX28 is related to the color burst.

12.6 Loading programs into the XGS Pico Edition PCB

The whole idea of the XGS Pico Edition PCB version is after you build it to program it! To do this, you of course need an SX-Key, SX-Key IDE version 3.0 (for our demos) loaded, and a serial cable for the SX-Key itself. The XGS Pico PCB has two modes of operation:

§   Programming Mode

§   Run Mode.

Programming mode is set by selecting the SX-Key signal input via the mode switch at **J3**, move it to the *right* and select the *"KEY"* mode for this operation. Then the SX-Key hardware and software will control the clock and programming of the XGS Pico Edition.

Run mode is set by selecting the on-board oscillator signal input via the mode switch at **J3**, move it to the *left* and select the *"OSC"* mode for this operation. Then on-board oscillator will control the clock the XGS Pico Edition. This is used when you want to run the XGS alone without the SX-Key generating the clock signal.

To load any particular program into the XGS Pico Edition, please review Chapter **11.16**, specifically section **11.16.6** which outlines the software, interface, and programming procedures.

12.7 XGS Pico Edition PCB Version Accessories

In this section, we will briefly discuss a few add on accessories for the XGS Pico Edition PCB.

Figure 12.47 – The SX-Key Plugged into the XGS Pico PCB.
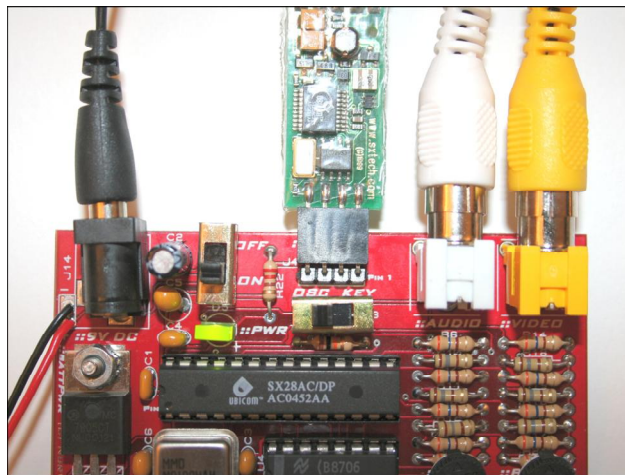

SX-Key

One of the cool things about the XGS Pico PCB version is that is easy to re-program the SX28, you will need an SX-Key and of course the SX-Key software. This is outlined in the previous chapter section 11.16 in detail. You can purchase SX-Keys at www.xgamestation.com, Parallax, as well as other distributors. Figure 12.47 shows an SX-Key plugged into the XGS Pico PCB's programming port.

Power Supply

Although, the 9V battery is cool for making the XGS Pico PCB portable, it will only last so long, thus if you're going to do any serious development on the Pico (which I hope you do), then you should pick up a 9V DC power adapter. The specs are 300 mA (or better), 9V DC, with a 2.1mm male lead with tip (+) and ring (-). You can pick these up at the XGamestation site, Digikey or any good distributor.

Figure 12.48 – Atari 2600 Compatible Joystick Plugged into Pico Edition PCB.
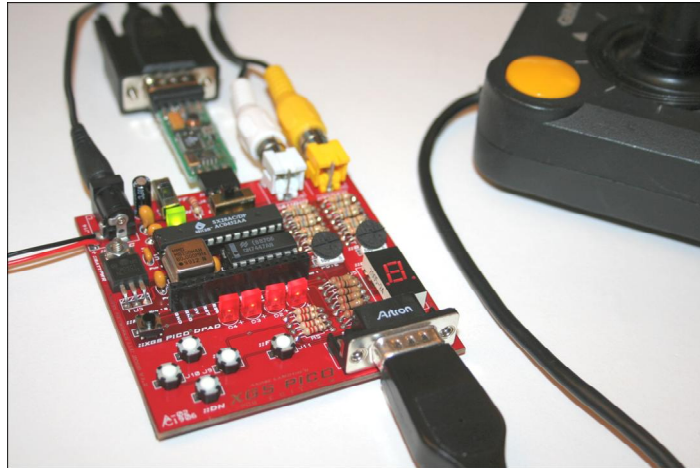
Atari 2600 compatible joystick

The XGS Pico Edition (both versions) support Atari 2600 compatible joysticks (Figure 12.48 shows the Atari compatible *"Gemstik"* plugged into the Pico PCB). These are reasonably easy to find on eBay or from your old 8-bit computer systems. The XGamestation site sells these as well.

Summary

Well, hopefully you had a lot of fun assembling the **XGS Pico Edition PCB**. It gives you an appreciation of all the early pioneers that literally soldered and wire wrapped thousands to tens of thousands of connections to design the first integrated circuits with TTL or transistor based discrete implementations! Not to mention how difficult surface mount soldering is! The XGS Pico Edition PCB is not only a small game console, but a full blown portable embedded system that is *very* powerful. The SX28 at 80.000 MHz is putting out 80 MIPS of computational work and that's a lot to work with. See if you can re-create some of your favorite retro games on the system. Of course you will need an SX-Key programmer and IDE which can be purchased from distributors around the world including **www.xgamestation.com**, **www.parallax.com** and others. Good luck! And be sure to upload your demos to the XGamestation site at **www.xgamestation.com** as well as visit our forums. Lastly, make sure to really investigate the CD-ROM that comes with the kit(s), there are a lot of interesting demos, articles, and source files to work with.