Homebuilt Balloon Instrument Package

This project is intended for the balloon builder looking for new vistas. I've been building balloons for 22 years. Sometime back I decided to learn to program microcontrollers—those smart chips found in everything from tooth brushes to TV sets. My first balloon project was a temperature gauge for my wife's balloon. The LCD screen displays temperature or elapsed time. When first turned on it also pages through a preflight check list.

Following that, I decided to build a full balloon instrument package. This is my fourth version. The instrument package displays altitude, envelope temperature, rate of climb and elapsed time on a graphic LCD (GLCD). The display has large font. The hardware design is simple.

The current design is aimed at ease of construction. The builder is required to fit the display, and other controls, into a case by cutting and drilling. The circuit board requires inserting electronic components through holes and soldering them in place. A pair of wires is assembled, to connect the envelope sensor to the case. There is one tiny surface mount component. But it is mounted on a plug-in sub-board available from the author, as is the preprogrammed microcontroller.

This article stresses construction using a professionally etched circuit board. However, on page 18 a template is provided for a builder who wants to etch his own board.

Above is a picture of the actual display. Its about 2.75 inches wide. The large font is almost half an inch tall. At the top left, the altitude is displayed from 0000 to 9999 feet. Below that is an elapsed time timer which displays up to 256 minutes with seconds. It currently displays 0 minutes and 45 seconds. Middle right is the envelope temperature. A bar graph displays rate of climb; it currently displays a climb of about 370 feet per minute.

Information on ordering parts is on pages 15, 16.

Introduction to the Design

The board has a simple design. A minimal amount of circuitry is used to achieve the various functions.

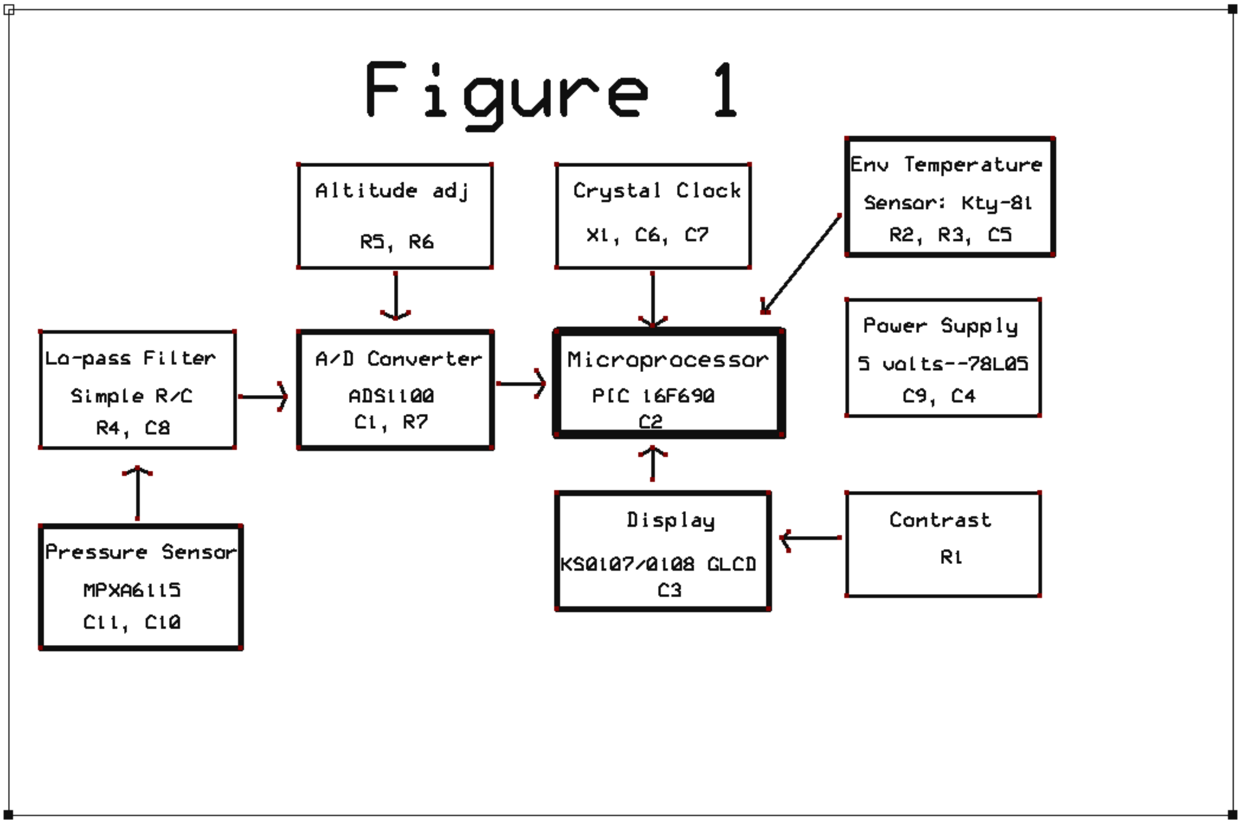


Figure 1 shows the design blocks. The blocks are in the same general location as shown in the actual board shown on page 4 in *Figure 2.*

Atmospheric pressure is read by a MPXA6115 pressure sensor. The sensor output is a voltage that varies with changing pressure.

This output voltage is fed through a low pass filter consisting of one resistor and one capacitor. This low pass filter helps remove electrical noise from the pressure sensor output.

The altitude voltage is read into an ADS1100, 16 bit analog-to-digital (AD) converter. This converter translates the voltage into a number that can be read by the microcontroller.

The AD converter has two inputs. One input comes from the pressure sensor. To adjust for variations in barometer settings, a ten turn potentiometer feeds into the other input. This is used to adjust the surface altitude. There is no barometric pressure display in the instrument. The instrument is either set to zero for takeoff altitude, or the altitude, MSL, of the takeoff point.

The AD output is fed to the PIC. The PIC 16F690 microcontroller is the heart of the instrument. It performs all math calculations and provides information and instructions to the display. While the display will display from 0 to 9999 feet of altitude, altitudes less than zero are displayed as four dashes. Altitudes above 9999 are displayed without the leading “1” for 10,000 feet. The display limits out at about 11,500 feet.

When compared with two other aircraft grade altimeters this instrument has proven accurate. At 5,000 feet it was within 30 feet of the other altimeters.

A KTY-81 sensor was selected for envelope temperature readings. The resistance of the sensor varies with temperature changes. This sensor is wired, as one side of a voltage divider, into a 10-bit AD converter build into the PIC 16F690. The voltage divider translates the changing sensor resistance into a voltage which is displayed as temperature. The voltage divider includes a resistance trimmer to adjust the temperature displayed. I set my instrument by dipping the sensor in boiling water. The instrument displays a temperature range of 32-285 degrees F.

The 16F690 clock is a crystal oscillator to permit an accurate timer. The timer starts when the unit is turned on and will read to 255 minutes and 59 seconds before it rolls over to zero.

The power supply is driven by a 9 volt battery through a 78L05 voltage regulator. The instrument pulls about 15 ma for long battery life.

The envelope temperature and altitude are displayed in large bold font. The elapsed time is displayed in large regular font. The rate of climb is displayed as a bar graph.

The graphic liquid crystal display (**GLCD**) is a common 128 pixel by 64 pixel display using KS0107 and KS0108 controllers. This display is actually two 64 x 64 pixel displays driven side by side.

A schematic diagram of the instrument and a parts list are displayed on pages 16 and 17.

Circuit Board

The circuit board was designed using software provided by ExpressPCB. My circuit boards are also produced by this company.

ExpressPCB.com is a circuit board manufacturer marketing to the experimenter. One downloads their software, designs a board, and sends a file to the company. In a couple of days the boards appear at your door.

Figure 2, next page, displays the circuit board as it appears in ExpressPCB software. The green layer is the trace layer on the back of the board. The red layer is the trace on front of the board.

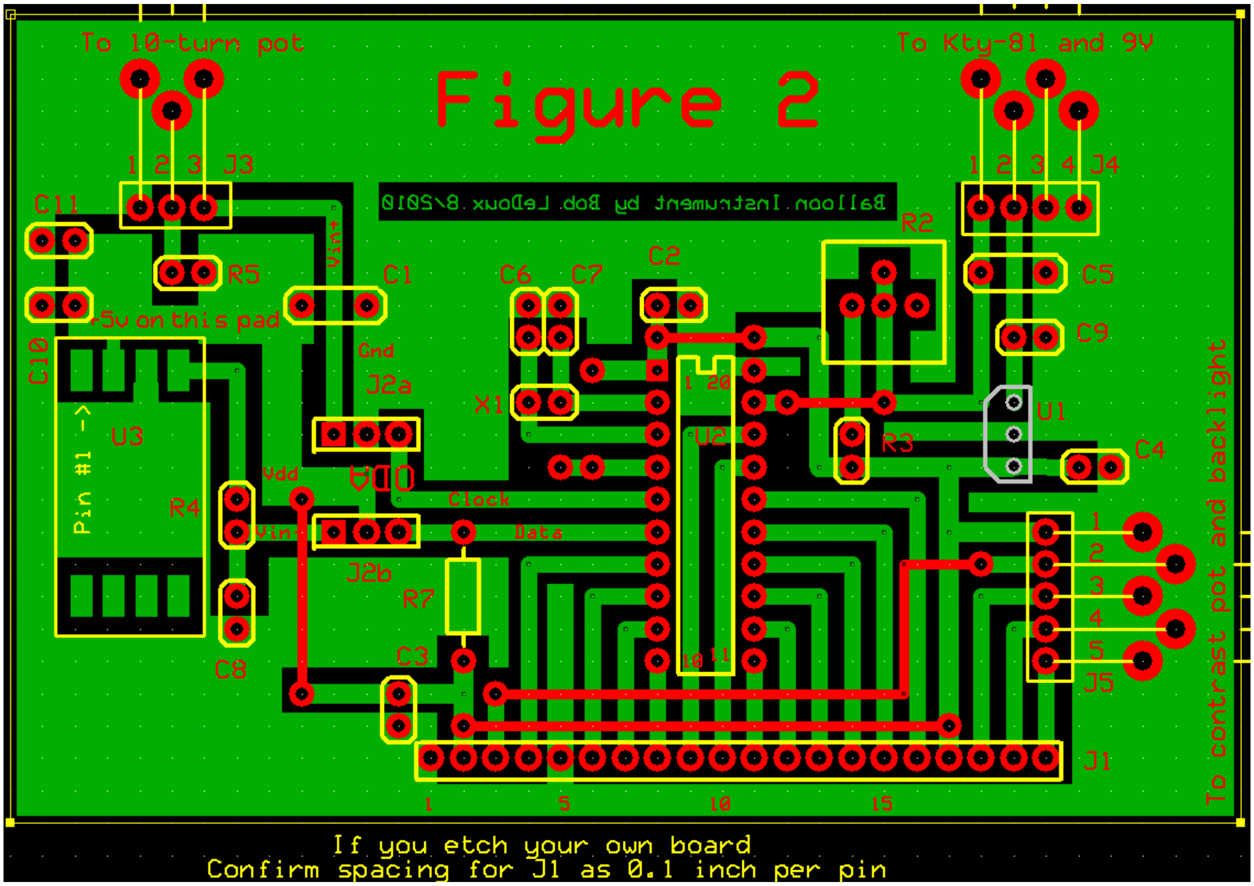
Below *Figure 2* is a picture of the *Figure 2* board with components added, and mounted in the case cover. The case cover holds all the components. The case itself, acts as a cover for the instrument.

Refer to the photograph on the next page. The *Figure 2* board mounts on top of the graphic display. A 20-pin connector is shared between the two units. On the upper left is a jack for the temperature sensor. Below it is the altitude adjustment control. Upper right is a 9-volt battery. Below the battery is the display contrast control that also includes the on/off switch.

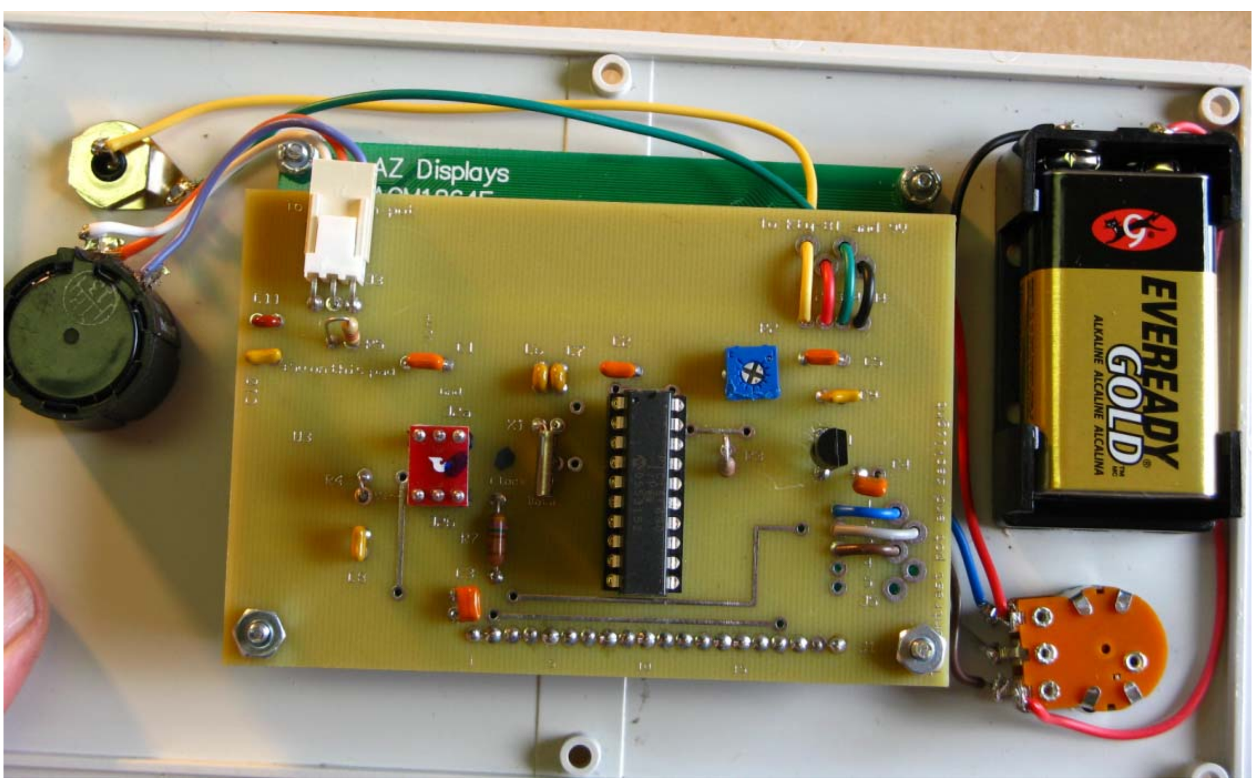
The ExpressPCB board is 2.5 inches by 3.8 inches. I sell this board. This is a standard size board, provided at the lowest cost. Three boards are delivered to USA addresses for $60 total. The size is such that only two of the GLCD mounting holes are be used to support the board.

If a board is ordered from ExpressPCB it will be a two layer board with the red and green areas etched onto the final board. If the user decides to make a board I suggest making a single layer board using the green layer. The red lines on the front become wire jumpers. Holes are provided in the board design for the jumpers. The ExpressPCB circuit board file is available from the author.

The figures on the next page are referred to throughout this manual. Study them carefully.



The picture below is the board, mounted in its case with the components soldered in place. Note the photo shows two mounting screws at the bottom of the board. Note direction of U1, the voltage regulator as seen in gray, above, and in the photo, below.



Construction Process

See the parts list on page 16. Construction involves the following steps:

First; the case cover is cut and drilled to mount the display and other controls. The *Figure 2* circuit board is also drilled to mount in the case.

Second; the *Figure 2* circuit board is populated with its components and wires.

Third; Controls are mounted to the case cover. The display is mounted by four corner screws. Two of these screws also support the *Figure 2* circuit board. The battery case is mounted using screws or double sided tape.

Fourth; wires are connected from the *Figure 2* board to the controls.

Fifth; the envelope sensor wire is assembled and temperature reading adjusted.

Sixth; the unit is powered and closed up.

The components list shows recommend items. Other components might be used. Read the "Component Options and Clarification" section below to get a fuller understanding of component specs and builder options.

Step 1: Preparing the Case and Mounts

The term "GLCD" refers to the display and its attached circuit board. In the following instructions the prototype was built using the "LCD1030" board purchased from BG Micro. I have not tested other GLCD's built to the same specs.

The GLCD is supported by screw holes in each corner of the circuit board. These holes fit 2.5mm screws. Drill these holes out to 1/8th inch. You will not damage the nearby traces on the board. This allows us to use common #4-40 screws for my assembly. My LCD1030 board had plated through holes at each corner. As long as a bit of the plated edge continues to show after drilling then the hole is not too large.

Twelve wire connections are made to the *Figure 2* circuit board at J3, J4, and J5. One way to make these connections is to solder wires to the connection points and run the loose wire ends through the strain relief holes. These strain relief holes are the larger zigzag holes near the edge of the board. If you choose this option try passing a length of your hookup wire through the strain relief holes. If necessary, enlarge the holes to pass the insulated ware. I used this wiring technique for connectors J4 and J5.

Another option is to use connectors for J3 through J5, like Molex series "KK" connectors. This allows the wires to be unplugged from the board. With this option the strain release holes are unused. My example used "KK" connectors for J3.

Step 1a: Cutting Out the Case Cover

Refer to the "Case Template," below. The placement and cutting are not critical so long as everything fits. The Case Template is the layout used on the prototype instrument.

Print out the case template so it is full size. Note the width dimension of 7.1 inches.

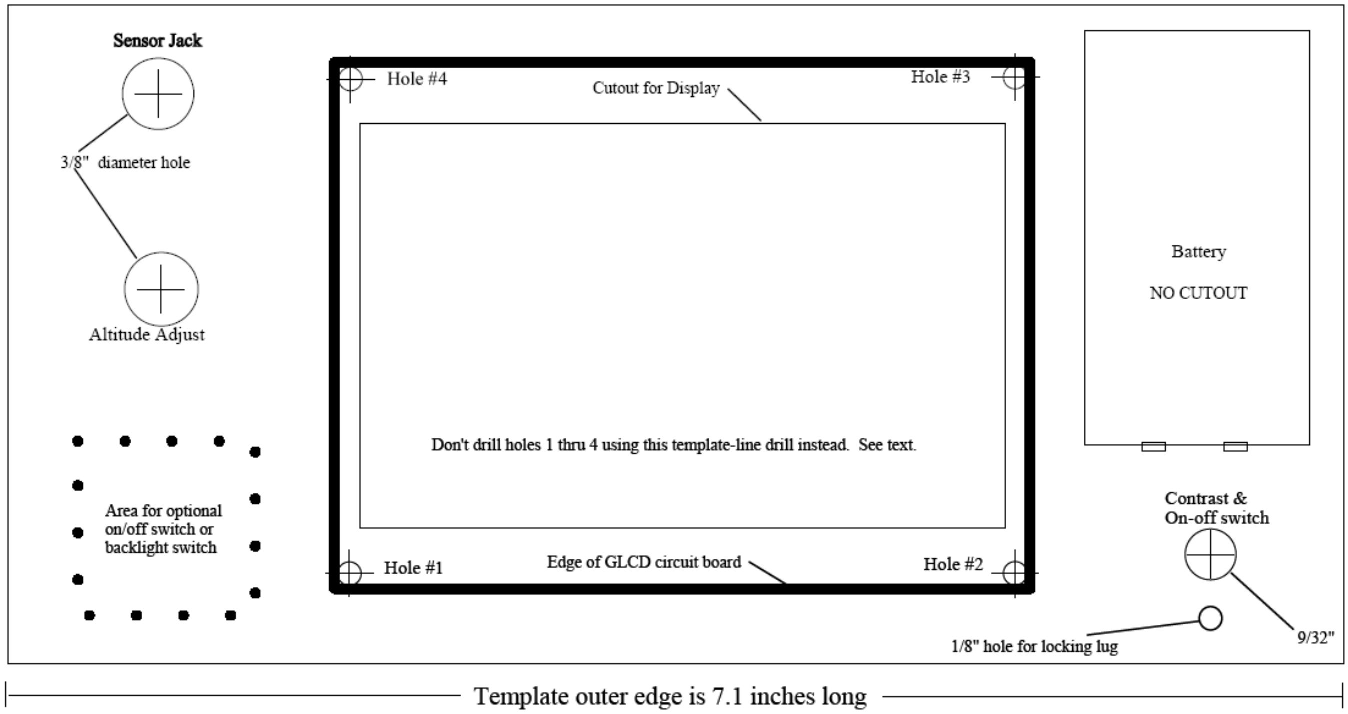
Cut the template out on the outside line. Attach the case template to the inside case cover using adhesive. I used 3-M Super 77 spray adhesive. The template is sized to fit between the left and right edge of the case cover, and between the pads for the case screws.

Refer now to the center of the template drawing. The light line is the cutout for the actual LCD1030 display.

The heavy line is the circuit board mounted on the back of display. It has a hole in each corner for mounting. Do not cut on this dark line!

Case Template

Back View of Case Cover



Ensure the template cutout for the GLCD matches your display.   If necessary redraw lines for your display cutout—the light line on the template. Cut out the light line to form the square hole in which the GLCD screen is fitted.

Line drill the case cover. Fit the GLCD into the hole in the case cover. Run a 1/8th inch drill through the four GLCD mount holes and drill out the case cover. Now, a screw slipped through a case cover hole will fit through the GLCD mount hole.

Drill the holes in the case cover for the contrast control, altitude adjust, the sensor phono jack, and extra power switch, if used. Check to see your controls fit in the hole size listed.

Step 1b: Install Connector between Circuit Boards

Let’s begin with some terminology. The top of the *Figure 2* board is the end with 3 holes and 4 holes in a diagonal pattern The bottom of the board is the end that has the 20 pin connector. If you are looking at the front of the board you see the lettering for component placement. This is the red side of the board in *Figure 2*. Looking at the back of the board the board title and author’s name appear in normal font. This is the green side of the board with most of the traces. You might want to mark ‘top,’ ‘bottom,’ ‘front,’ ‘back’ on your board with a felt tip pen

We now install the 20 pin socket and header to mate the GLCD to the *Figure 2* circuit board. When finished the boards stack as shown in the photo on page 4.

Cut a length of header, the male component (part 517-834-03-36) with a length of 20 pins. A diagonal pliers (dikes) or a fine tooth craft saw may be used to make the cut.

Take the 20 pin header. Insert into the *Figure 2* circuit board: Insert the short pins into the circuit board from the back. Solder the pins on the front of the board. Make certain the pins are perpendicular to the board surface. This is the only component soldered from the front on the *Figure 2* circuit board. You can see the twenty solder joints in the picture on page 4.

Cut out a 20 pin socket. The socket in the part list, part 517-974-01-21, only comes in a 21 pin length. Remove one pin to make it 20 pins long. Don't try to cut between the teeth. Use sandpaper, file or a Xacto knife to remove the excess pin. Insert the socket into the holes in the GLCD from the back. Solder the pins from the front. Make certain the socket is perpendicular to the GLCD.

At this point you can plug the GLCD and *Figure 2* circuit board together. It should stack like that shown in the photo on page 4.

Step 1c: Line Drill Figure 2 Circuit Board

You drilled out the GLCD mount holes to match the case cover in Step 1a. Take the stacked GLCD and *Figure 2* board from Step 1b. Pass a 1/8th inch drill through each of the two bottom mount holes on the GLCD and drill out the *Figure 2* Board. See the picture on page 9 to see the result.

Populate the *Figure 2* Circuit Board

Remember that all the remaining components, except for the pressure sensor, U3, are placed through the holes from the front of the board and soldered on the back.

After soldering components, clip off the extra wire beyond the solder joint on the back of the board.

Solder the pressure sensor to the back of the board. See photo below. Turn the board over on its back side up. The 3 diagonal strain relief holes should be towards your left. Locate the notch on pin 1 of the pressure sensor. This maybe the pin next to the cut off corner on the chip. Place this pin on the upper left. Set the sensor on the 8 pads on the board.

The sensor is now properly aligned for soldering. Set the sensor to the side and heat a bit of solder on pin number 1 pad on the board. Set the sensor in place and heat the pad to solder the sensor in place. Check to see all 8 pins are in good alignment. Now solder the pin and pad on the opposite corner. Check the alignment. Solder the remaining pins in place. *Figure 2* notes the placement of pin #1, as seen from the front of the board.

Now to solder components on the front of the board:

Turn the board over. Solder the U2 PIC socket in place. The socket has a notch in the end. This notch faces toward the top of the board. Place the socket in the holes marked U2. When properly placed you will see a trace, and two holes running beyond the top end of the socket. Solder the socket in place from the back. See *Figure 2* for clarification. (While the 16F690 can be soldered directly to the board the socket makes it easier to re-program the chip.)

Install J2. If you purchased the sub-board assembly simply plug the board into the six holes on the front of the *Figure 2* board. Solder the 6 pins sticking out the back of the board. Gently pull off the sub-board and set it aside.

For the following components refer to the parts list, page 16, for component values. Refer to *Figure 2* for proper component placement. *Figure 2* allows you to see through the board, something you can't do with the real board.

Insert R2, the 1k ohm trimmer, into the front of the board. Solder R2 in place.

Mount voltage regulator, U1 in place. This must be soldered in correct alignment. Looking from the top there is a flat side and curved side. Insert the unit into the three holes according to the image in *Figure 2*.

The following resistors may be 1/8th or ¼ watt rating. Those listed in the parts list are 1/8th watt. Radio Shack is a supplier of resistors. I used ¼ watt resistors for better photo visibility.

Solder resistors R3, R4, and R5 in place. Hold each resistor so the wires are vertical. Bend the top wire down the side of the resistor. Insert the resistor in the board so it stands upright and solder in place. The holes are 0.1 inches apart. Refer to the photo on page 4.

Solder resistor R7 in place. Bend the leads so this resistor can lay flat on the board. The mounting holes are 0.4 inches apart. Solder in place.

Solder the crystal, X1 in place. Bend the leads sideways so the crystal can lay flat on the board. Insert the leads and solder in place.

Solder the two crystal capacitors, C6 and C7 in place. These are installed, side by side. Check *Figure 2* for correct orientation.

Solder the bypass capacitors C1, C2, C3, C4, and C5 in place. The leads on C1 and C5 have to be bent to fit the holes which are 0.2 inches apart. The other 3 capacitors should fit straight into the holes without lead bending.

If C8, or C9, or C10 are tantalum or electrolytic capacitors see notes in following section about cautions.

Solder the .33uf capacitors, C8 and C9 in place.

Solder the two capacitors C10, 1 mfd, and C11, .01 mfd in place.

All the components are now soldered in place on the board.

Mounting the 12 wires: If you choose to use Molex KK type connectors I recommend that right angle connectors be used. Straight connectors may be too long to fit in the chosen case.

If you are wiring directly to the J3, J4, and J5 connection points, cut excess lengths of wire and strip about ¼ inch off one end. Insert this end into the front surface of the board and solder from the back. Pass the free wire end through the strain relief hole. Leave a small loop between the solder point and passage through the strain relief hole.

At this point all of the components and wires are mounted to the circuit board. Clip off the wire ends that stick out the back of the board. There will remain empty holes in the circuit board.

Backlight

Note the parts list does not include battery and switch for back light. I have not wired the backlight on my instrument, wires J5-4 and J5-5. If you desire to use a backlight check the current requirements for your GLCD. The LCD1030 GLCD requires over 400ma to power the back light. This current cannot be taken from the 9 volt battery. An extra battery must be supplied. A momentary switch must be used as the backlight will discharge even “AA” batteries very rapidly.

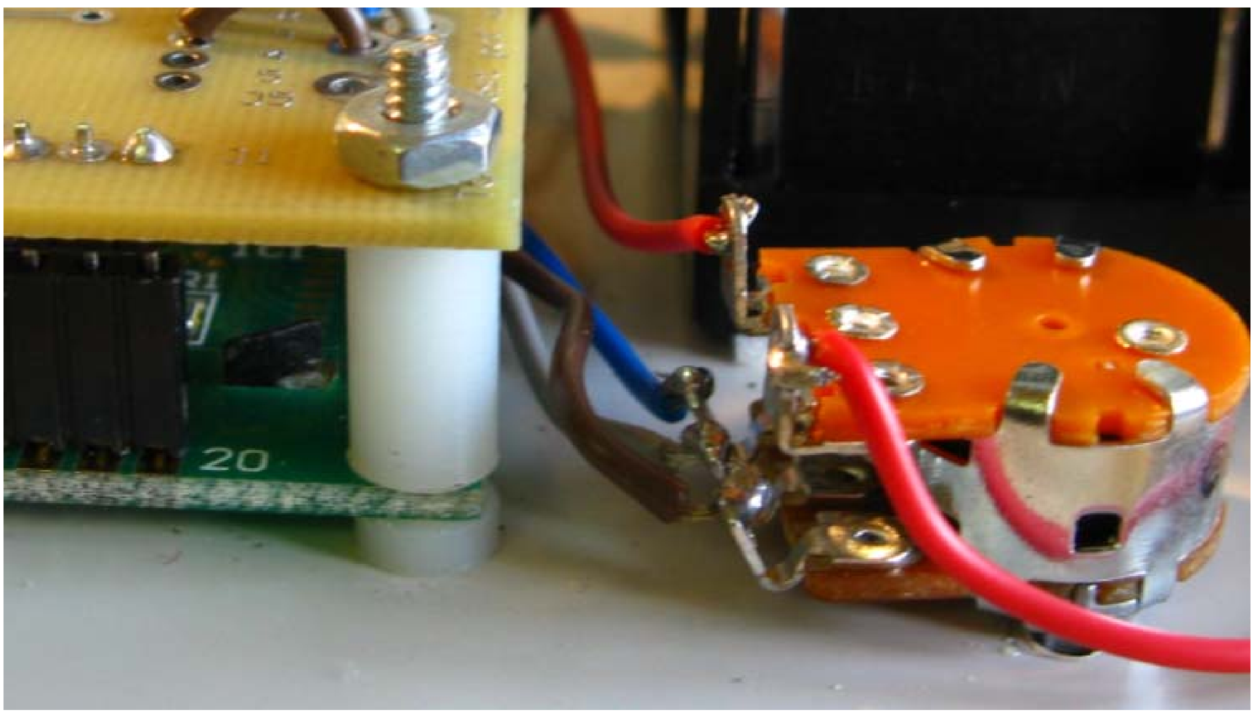
Step 3a: Mounting GLCD and *Figure 2* Board in the case.

Assemble the GLCD and *Figure 2* circuit board sandwich and screw it to the case front, with spacers in place. The photo below shows this sandwich:

In Step 1c the two boards were line drilled. A 1 inch long #4-40 screw comes up through the case cover. A #4-40 nylon washer is next, followed by the GLCD, the green board. Next, is a 3/8 inch long nylon spacer and the *Figure 2* circuit board. A nut tightens everything in place. You should secure the nut in place. Use finger nail polish on the exposed thread at the nut.

The short bottom spacer provides clearance for the solder points on the front of the GLCD.

The long nylon spacer might be a bit short, as evidenced by the top circuit board bowing when the nut is tightened up. In this case add a nylon washer below the top board.



Also note the photo shows wire connections to the switch and contrast control described later.

In the following steps the wires will be connected to controls. Cut the wires to appropriate length before soldering the free end.

Step 3b: Mount Sensor Jack

Refer now the photo on page 4. I have used different colored wires for identification:

Place the sensor jack through its mount hole. The washer with the solder tab is on the inside of the case cover, pointing toward the boards. Tighten in place using the nut.

Solder the wire from J4-1 (wire #1 on J4), the yellow wire, to the center tab of the sensor jack.
Solder the wire from J4-3, the green wire, to the side tab on the sensor jack.

Step 3c: Mount the Altitude Adjust Control

Check the control mount. There may be a lug or knob intended to keep it from turning in its hole. If so, create a matching hole or dimple in the case cover. You could also use double stick tape between the control and the case.

Solder the wire from J3-1, White wire, to the wiper or slider terminal on the control.
Solder the wire from J3-2, orange wire, to the CW (clockwise) or T2 terminal of the control
Solder the wire from J3-3, purple wire, to the CCW or T1 terminal of the control.
Slip the control into its mount hole and tighten up the nut.

Step 3d: Mount Battery Case

Mount the battery case in position using small screws or double stick tape. If you use screws make certain they don't impede installation of the battery into the case. The two solder contacts should face towards the top of the inside case cover.

Step 3e: Mount the Contrast Control and Power Switch

See details on the photo above, and photo on page 4.

The control has two rows of solder lugs. The two solder lugs near the back are for the switch. The three solder lugs in the middle are for the contrast control. We solder the middle three wires and then install the control.

For ease of fit I gently bent the three terminals in a row out by 45 degrees.

Solder the three wires in place. Refer to the photo on page 9. They solder in the order shown. The blue wire (J5-1) goes to the away terminal; the gray wire (J5-2) to the center terminal and the brown wire (J5-3) to the near terminal.

Insert the control and tighten the nut to install in place. The control has a lug that fits in the small hole to keep it from turning.

Now to solder the switch:

Take the **black** wire (J4-4) and solder it to the left, negative terminal of the battery case.

Take the red wire coming out of J4-2 and solder to the top switch solder lug on the contrast control.

Take a length of hookup wire and solder it to the bottom switch lug. (This is the other red wire in the photo above.) Solder the other end of this wire to the positive terminal, the right terminal on the battery case.

Step 3f: Plug in's

Insert the PIC 16F690 into socket for U2. The PIC has a dimple at one end that signifies the #1 pin. This dimple is towards the top of the board, away from the 20 pin connector.

The breakout board that plugs into J2 has a mark or notch on one end of the board. Insert this so the it faces toward the U2-16F690 socket. It is essential the breakout board is plugged in correctly. If it is reversed, power to the chip will be reversed and the A/D chip will burn out.

Install a 9 volt battery and close up the case.

This completes assembly of the unit.

Step 4: Sensor Wire Assembly

In this section the sensor wire is prepared and the unit is calibrated.

Read about sensor wire in the section below.

Having selected your sensor wire solder the KTY-81/110 sensor to one end of the wire pair. I suggest placing a 2 inch length of shrink wrap tubing over each wire before soldering. After soldering slide the tubing over the exposed wire and shrink it using a hair drier. Be a bit cautious as the sensor is limited to a maximum temperature of 302 degrees F.

I place a second shrink wrap cover over my sensor end. I take a piece of shrink wrap tubing that barely fits over the sensor and run it two inches from the end of the sensor over the wires. I'm careful about applying heat for the reasons stated above.

Solder the other end of the wire pair to your hookup plug, the one that matches your jack in the instrument case cover.

Step 4b: Sensor calibration

Plug the sensor into the instrument. Heat a pan of water to a rolling boil. Turn on your instrument. Slowly move the sensor towards the water as the temperature reading increases. This avoids temperature shock to the sensor. Dip the sensor into the boiling water. Don't let it touch the sides or bottom of the pan. Wait for the display to reach a final reading. Adjust R2 until the display reads 211-213 degrees F. Calibration is finished.

The instrument is ready to use.

Please advise the author of errors or corrections.

Component Options and Clarification

This section provides further detail of components and discusses options for the builder. This is intended for the builder interested in further details. There is more detail here than needed to built a successful project.

The parts list uses Mouser.com, a major electronic supply house with fast service and no minimum sale amount. Mouser provides data sheets and drawings for its components.

A 15 volt rating is minimum for all capacitors.

All resistors can be 1/4th or 1/8th watt rated. The parts list has 1/8th watt units. I used 1/4th watt resistors in the prototype for better visibility in the photos.

Bypass capacitors, C1, C2, C3, C4, C5. are standard ceramic caps of 0.1 microfarads.

Regarding C8, C9 and C10. These are large value capacitors available as ceramic, tantalum or electrolytic versions. If electrolytic or tantalum units are chosen be aware these are polarized caps. These units must be soldered in the correct direction. The negative pole must be soldered towards the edge of the board, not towards the middle of the board. The edges of the board contain the ground planes. All broad trace areas are ground with one exception. There is a plane above the J2 header that is at +5 volts. It is so marked on the front of the *Figure 2* board.

Bypass capacitors C8, C9, are 0.33 microfarad capacitors.

Sensor capacitors C10, C11 are 1 microfarad and 0.01 microfarad respectively. These caps provide clean power to the altitude pressure sensor. If you inadvertently put C10 in the place of C11 or vice versa, don't worry about it.

Contrast adjustment R1. This is a panel potentiometer in the range of 10k to 20k ohms. This control sets the contrast of the visual display. Temperature and light may affect screen visibility so a control on the front panel is recommended. The component in the parts list also includes a switch. When turned fully counterclockwise the switch turns off. This allows the rotary switch to be used to turn power off and on to the entire instrument.

Power control can be made using a separate switch. My experience is a rotary switch is less likely to be accidently turned on while packing up a balloon. A toggle switch has a sharp point that can cause injury in a hard landing. Separate rotary switches are expensive.

Temperature adjustment R2 and R3: R2 is a one turn 1k ohm trimmer. It is used in series with R3, which is 3900 ohms, to create a voltage divider for the temperature sensor. The series resistance of R2 and R3 should be adjustable on either side of 4700 ohms.

R2 can also be replaced with a 5k or 10k ohm, 10-15 turn trimmer. In this case install a wire jumper in place of R3.

Noise filter resistor: R4 is a 750 ohm resistor. It along with C8 creates a low pass filter with a cutoff frequency of about 700 Hertz. As an alternative, R4 can be replaced with a 330 ohm resistor if C8 is replaced with a 1 microfarad capacitor, similar to C10.

Altimeter setting components R5 and R6. These two components create a voltage divider to provide 0.0 to 0.22 volts to the AD converter in J2. This voltage allows setting the altitude display. R6 is a ten turn panel mount potentiometer of 1K ohms maximum value. R5 could be made smaller and R6 larger as long as the desired voltage range is obtained. However be aware of changes in sensitivity to adjustment of the altitude.

Also be aware that the ADS1100 inputs are specified as "low impedance." So large resistances for R5 or R6 may impair the ADS1100 readings.

I did not put a knob on altimeter setting control R6. This is to remind me that this control is not touched after setting it before flight. It also reduces the possibility the control will be knocked out of adjustment during a landing.

J2 holds the A/D converter sub-board. J2a and J2b are made from two, three pin headers. The headers plug into a socket on the sub-board. The ADS1100 is a small surface mount chip about the diameter of a pencil eraser. The chip is mounted on a breakout board "BOB-00717" purchased from SparkFun Electronics. The ADS1100 and the sockets are mounted on the bottom of the breakout board

If you turn over the sub-board you will see the little chip, the ADS1100. If you look at the chip with a magnifying glass you will see the letters "AD0," which identifies the chip and its communications (I2C) address. When properly plugged into the board, the number "0" will be the nearest letter towards the 16F690 microcontroller.

I chose the ADS1100 A/D converter because it is simple and low cost. It is not available in a DIP (through hole) package. If you order your own chip make certain it is the same part number listed in the parts list. It comes in packages marked AD0 through AD7. The last digit is the communications address. The PIC software is programmed for AD0 chip address.

Battery holder: I have not been really happy with any of the 9 volt battery holders I've found. Secure battery mount tends to be a problem. You might be happy with a spring clip battery holder and snap on wire connector found at Radio Shack

Connectors: In the photo on page 4, a KK connector is used for J3. A special tool is needed to crimp KK wires to connectors. allelectronics.com sells KK female connectors crimped to wires. Check items CON-243, CON-244 and CON-55.

GLCD: The graphics liquid crystal display is a 64 pixel by 128 pixel unit using KS0107 and KS0108 controllers. The circuit board requires the display to use a 20 pin out on 0.1 inch centers. The pin connector must be below the display area. Different GLCD's vary in pixel size. So some boards will have a bit smaller active display area than others. Variation in displays may result in a different size cutout in the case.

The display will cost between $10 and $40. One of the largest and least expensive displays is the LCD1030 unit sold by BGMicro for $9.95. Mouser Electronics sells several displays with these controllers made by "New Haven." I like 763-12864AZ-FSW-GBW. CrystalFontz has a wide selection of boards though they tend to be more expensive when purchased in unit quantities.

If you choose a different GLCD confirm it has the same pin out sequence as in the LCD1030 used on the prototype. See the parts list for pin definitions, page 16. Check to see if pins "CS1" and "CS2 or reversed."

Temperature Sensor: I chose a KTY-81/110 silicon sensor after a number of tests. It is non-polarized and linear over the large temperature range. If it responds too quickly to the burner I apply extra shrink wrap over the sensor as increased thermal insulation.

In the past I have used pyrometers, silicon junctions, various dedicated temperature sensors, both analog and digital. The digital units are generally limited to 255 degrees F. They also require three wire feed. The other analog choices have not been reliable over broad temperature ranges, or require special output (opamp) treatment. Some sensors are also expensive.

Sensor Wire: The temperature sensor is mounted above the equator of the balloon envelope and the instruments are down in the basket. A pair of wires connect the two. The wire must be capable of surviving the elevated temperatures as well as surviving constant bending from packing up the envelope. Typically, a stranded wire of size #18 to #22 is used. To take the heat the wire must have a heat resistant cover like Teflon. In a perfect world, two Teflon insulated wires would be contained in a Teflon cover to create a cable. This setup is often found in manufactured balloons.

Such a Teflon cable is special order. If one goes to a wire manufacturer and seeks a quote for this cable one is likely to see a minimum order of 1000 feet at $2.50 a foot. That's why factory made temperature harnesses are so expensive.

Teflon wire is commonly on sale on eBay. Occasionally even Teflon cables are found. Remember that the sensor will require two wires of about 75 feet, each for this application.

Another option for the experimenter to consider is motor rewind wire. Electric motors are wound with solid copper wire with a hardy varnish surface called Formvar. The coating is good to hundreds of degrees F and it retains its flexibility over time. About a quarter pound of #21 or #22 wire should be sufficient for a sensor line. At the right shop it should cost $5-$10. Solid wire won't last as long as stranded wire, but motor rewind wire has a lower cost.

Teflon wire is "springy" and can develop a rats' nest if not contained. My wire harness has the basket end to the envelope mouth covered with heat shrink tubing or fiberglass tube. Motor rewind shops carry the fiberglass tube. Inside the envelope the wire runs through a narrow tube constructed of envelope fabric sewn to a vertical tape.

Sensor wire connector. The sensor wire plugs into the instrument package. Any reliable two wire connector may be used. The parts list displays a phono jack that mounts from the rear of the panel. The rear mount allows removing the circuit board from the panel without unsoldering the wire to the connector.

The phono jack was chosen because it provides reliable friction contact. I've also used DC power plugs and jacks, like those found on the end of "wall warts," plug-in power supplies. They have the advantage that they are available in right angle connectors. Right angle plugs molded into wires are also available for a robust connector. See Mouser part 172-7445-E as an example.

Power Supply. The plans call for a single 9 volt alkaline battery. A battery should last about 30 hours of operation. No provision is made for a backup battery. This is because I don't' consider my instruments critical for flight safety. Additional batteries could be mounted in the case with a battery selection switch. Another option is to use a 6-cell battery holder for "AA" or "AAA" batteries which should last a season. At least 7 volts is required input for the voltage regulator.

Instrument Case: The case shown here was chosen to complete the instrument. You have to figure out how to mount the instrument in the balloon. If you have a better display case solution share your ideas.

Etching your own board.

On page 18 in this manual is a template for etching your own board. Don't expect the pdf file to precisely match the final pattern size. The 20-pin connector spacing is a critical dimension. These pins are spaced on a precise 0.1 inch spacing. This spacing must be maintained in the final board or the header won't fit. I use a piece of pre-drilled prototype board over the circuit board for a drilling template.

Overview of Software Design

The software is written in Microchip assembler. Because the GLCD has no internal font, the assembler code includes font tables. The software is about 2400 lines of code, which about 1400 are font and altitude conversion tables. Contact the author for details.

The program operates on a 0.5 second loop. During each loop new altitude and temperature reading are made. On every other loop the seconds timer is updated.

The altitude is read using less than the 16 bits range available on the ADS1100 A/D converter. This avoids the use of op amps or other intercept/slope altering methods. Because the pressure-altitude relationship is not linear, the altitude is calculated using tables. Refer to Microchip Application note AN942 on Piecewise Linear Interpolation to understand the process.

Rate of climb is calculated using a six byte first in last out table. Every second the current altitude is read and the difference taken with the altitude one second ago. This difference goes into the top of the stack. This difference, taken seven seconds ago, drops off the bottom of the stack. The stack is summed to generate a total change over six seconds. This result is multiplied by ten to generate a rate of climb per minute.

The rate of climb bar graph is not linear. It is more sensitive at low rates of change. This decision was made because most pilots are more interested in precision when maintaining level flight. A table is used to translate computed rate of climb into bar length. Any rate of climb greater than 800 feet per minute displays as 800 fpm. The bar graph is table driven so it can be changed.

Pins 1 and 4, on U2 show unfilled holes in *Figure 2*. These are available for the advanced experimenter. The extra hole on pin 1 is to the 5 volt power supply. Pin 4 is the MCLR pin on the microcontroller. (MCLR function is “internal” in this application.) Pin 4 can be used as an input. A pull-up resistor can be run between pin 1 and pin 4. The other hole on pin 4 can run to a switch that grounds pin 4. This would allow pushing a switch to alter the computer program. Contact the author if you wish to pursue this option.

Ordering Parts

Send me an email before ordering parts.

Here is a list of unique parts and costs, delivered in the USA. The list for standard parts is on the next page:

Microcontroller, preprogrammed 16F690: $5.00
ADS1100 sub-board, assembled: $10.00
Professionally etched circuit board: $22.00
Kit of 16F690, sub-board and circuit board: $35.00

Contact me at: bobledoux@proaxis.com

Bob LeDoux
PO Box 1306
Jefferson, OR 97352

Parts List

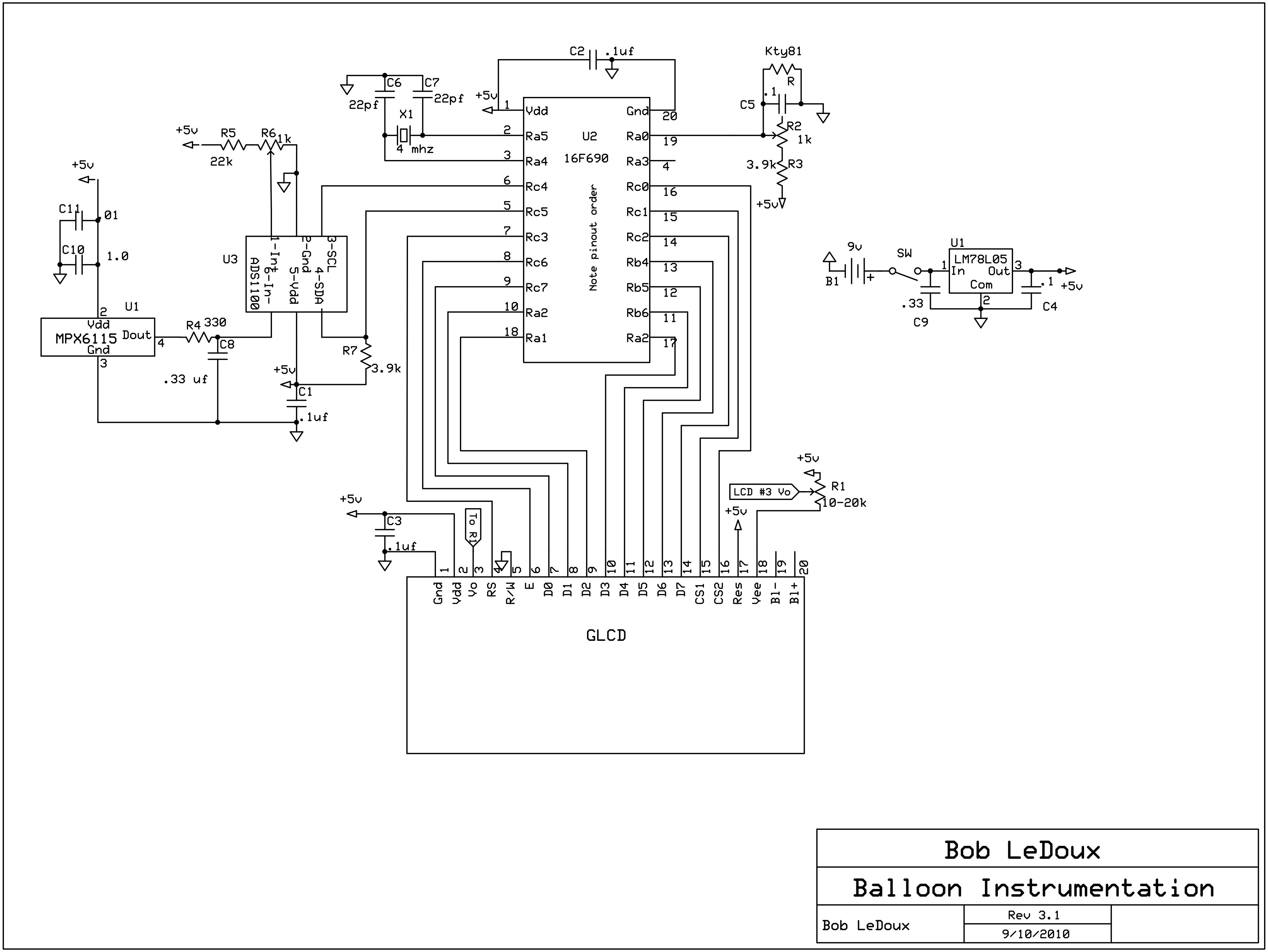
Bold items are for sale by the author

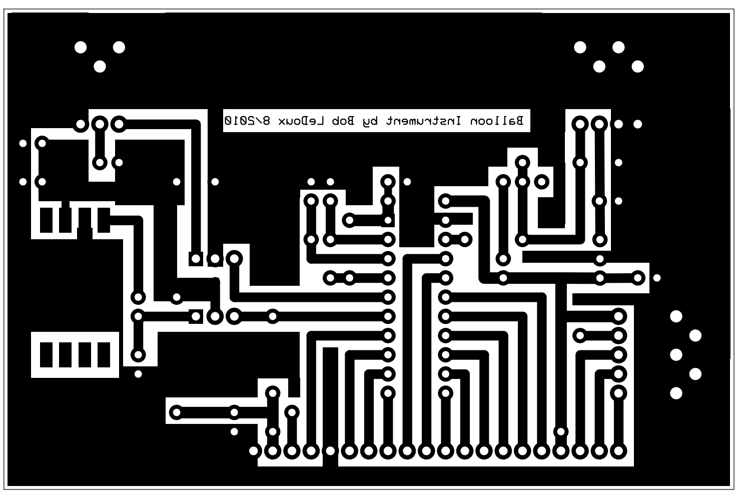
Item ID	Item	Mouser Parts Number
Sensor Wire	See builders' notes.	
Circuit Board	See builders' notes.	
GLCD Display	See builders' notes.	
C1, C2, C3, C4, C5	0.1 uf bypass caps	581-SR215C104KAR
C6, C7	18pf load caps for crystal	581-SR215A180J
C8, C9	0.33 uf	581-SR215E334MAR
C10	1uf cap	581-SR215E105MAR
C11	0.01uf cap	581-SR221E103M
R1	10-20k single turn contrast w/switch	313-1100F-10k
R2	1k trimmer	72-T70XH-1K
R3, R7	3900 ohm resistor	71-RN55D3901F
R4	750 ohm resistor	71-RN55C-F-750/R
R5	22k resistor	71-RN55D-F-22K
R6	1k 10 turn pot	652-3590S-1-102L
J1, J2	21 pin socket	517-974-01-21
	40 pin header	517-834-03-36
	20 pin DIP socket	575-11044320
U3	MPX6515 pressure sensor	841-MPXA6115A6U
U2	**16F690 Microchip controller**	579-PIC16F690-I/P
U1	78L05 Voltage Regulator	512-MC78L05ACHX
X1	4 mhz crystal	815-AB308-4.000
S1	Kty-81	771-KTY81/110
J3, J4, J5	See builders' notes.	
J6	Phono Jack for sensor wire	161-0351
	9 volt battery holder	12BH615D-GR
	Case	546-1591ES-BK
U4	**A to D convertor**	595-ADS1100A0IDBVT
	Breakout Board	BOB-00717-from Sparkfun Electronics

Minor items like screws and spacers are not listed. Local hobby shops cary these items

GLCD Pinouts Required on Figure 2 Board

1 Ground
2 Vdd-5 volts from 78L05
3 Vo-Contrast voltage from R1
4 Select input is data or instruction
5 Read/write pin; grounded as no reading from GLCD is required
6 Enable pin; pulses high to tell GLCD to read data lines
7 Data pin 0
8 Data pin 1
9 Data pin 2
10 Data pin 3
11 Data pin 4
12 Data pin 5
13 Data pin 6
14 Data pin 7
15 CS1; selects controller for left side of screen
16 CS2; selects controller for right side of screen
17 Reset GLCD; pulled high to Vdd because function is not used
18 Power supply GLCD contrast
19 Goes to positive terminal of separate battery for backlight J5-4
20 Goes to negative terminal of separate battery for backlight J5-5


Bob LeDoux
Balloon Instrumentation
Bob LeDoux
Rev 3.1
9/10/2010
U2
16F690
Note pinout order
Vdd
Ra5
Ra4
Rc4
Rc5
Rc3
Rc6
Rc7
Ra2
Ra1
Gnd
Ra0
Ra3
Rc0
Rc1
Rc2
Rb4
Rb5
Rb6
Ra2
GLCD
Gnd
Vdd
Vo
RS
R/W
E
D0
D1
D2
D3
D4
D5
D6
D7
CS1
CS2
Res
Vee
Bl-
Bl+
U1
LM78L05
In
Out
Com
U1
MPX6115
Vdd
Dout
Gnd
U3
ADS1100
1-Int
2-Gnd
3-SCL
4-SDA
5-Vdd
6-In-
C2 .1uf
C6 22pf
C7 22pf
X1 4 mhz
Kty81
R
C5 .1
R2 1k
R3 3.9k
R5 22k
R6 1k
C11 01
C10 1.0
R4 330
C8 .33 uf
R7 3.9k
C1 .1uf
C3 .1uf
To R1
LCD #3 Vo
R1 10-20k
B1 9v
SW
C9 .33
C4 .1
+5v



Board width is 3.8 inches

Confirm spacing for J1 as 0.1 inch per pin