Tucson Amateur Packet Radio
Internet: tapr@tapr.org www.tapr.org
Non-Profit Research and Development Corporation

T-238+
Main Board (Rev B)

T-238+ Assembly Manual

Introduction

The TAPR T-238+ is a general-purpose Motorola MC68HC908GP32 microcontroller-based board intended for use with the Dallas Semiconductor 1-Wire™ bus and any device that needs a serial connection. Four push-button switches and a Liquid Crystal Display (LCD) module provide the user interface. The initial application for the T-238+ is a controller that accepts data from a Dallas Semiconductor weather sensor unit, processes it, and formats it for sending either directly to a Terminal Node Controller (TNC) for broadcasting in APRS™ format, to a computer for processing and/or logging, or directly to a radio using the MX-614-based modem. There is nothing about this board that makes it solely a weather station. This just happens to be what the software for it does.

The T238+ is a successor to the TAPR T238. Software releases will work with both projects. The software will determine at startup which project it is running on and configure itself as needed.

For the most current information on weather sensors, software downloads, development tools, and related information, visit http://www.tapr.org and follow the links to the T238+ project, or visit http://www.beals5.com/wx.

The TAPR T-238+ is a ready-made kit with the weather application pre-programmed into the microcontroller. It is possible to develop your own applications on the T238+ as well. A debug header is provided that with a small adapter circuit can enable a fully functional development system.

The T238+ kit does not come with an enclosure or power supply. It is left up to the builder to find an enclosure that is suitable. For power, the user can use an ordinary wall transformer ("wall wart") that provides 8-12 Volts DC at 300 milliamps. The T238+ is fitted with a 2.1mm power coaxial connector arranged so that the center conductor is positive. Unlike the original T238, the T238+ contains analog circuitry that is much more susceptible to RFI, making the need for a metal case much greater.

As T-238 programs are developed they will be displayed or linked from the TAPR Weather web site. Go to http://www.tapr.org and follow the T238 links. If you've created a project and would like to display or link to it, contact TAPR at tapr@tapr.org.

The T238+ is discussed on the Weather Special Interest Group (WXSIG). You can join WXSIG via the web at http://www.tapr.org/cgi-bin/lyris.pl?join=wxsig.

Acknowledgments

The Tucson Amateur Packet Radio T238+ Kit was made possible by the efforts of (in alphabetical order):

William Beals, N0XGA - Conceptual Designer
John Bennett, N4XI – Sensors
Joe Borovetz, WA5VMS - Parts Management
Russ Chadwick, KB0TVJ - Conceptual Designer
John Koster, W9DDD – Project Manager

The T238+ consists of three boards. These are the main board, the modem board, and the button board. The main board and button board are kitted together and the modem board is separately. The LCD module is a board too, but is listed as a component for the main board. You will assemble the main board and button board first, and check them out. Finally, you will assemble and check out the modem board.

T238+ MAIN BOARD

PARTS LIST

The parts list is organized by quantity and part type. Verify that all parts are present by checking in the [ ] as you locate the part in the list. You may wish to take this opportunity to sort the parts into a compartmented container such as an egg carton or muffin tin as you inventory them. This will aid you in kit building.

Resistors 1/4 w, 5% Carbon Film:

[ ] (1) 10 ? (brown-black-black)	R4
[ ] (1) 470 ? (yellow-violet-brown-gold)	R9
[ ] (1) 1.5K ? (brown-green-red-gold)	R16
[ ] (1) 2.2K ? (red-red-red-gold)	R1
[ ] (4) 10K ? (brown-black-orange-gold)	R2,3,8,18
[ ] (1) 20K ? (red-black-orange-gold)	R20

Resistor 1/2 w, 5% Carbon Film:

[ ] (1) 4.7 ? (yellow-violet-gold)	R5

Resistor Pack, 5% Carbon Film:

[ ] (1) 10K ?	R19

Resistor, Trimpot

[ ] (1) 10K ? Trimpot (103)	V1

Capacitors

Capacitors may be marked in various ways. The typical markings are listed but may vary. Find all that match and the remaining ones, if any, should become apparent by elimination.

Mylar or Monolithic

[ ] (5) 0.1uf (104)	C1,4,5,9,12

Electrolytic or Tantalum

[ ] (5) 1uF (1uf 50v)	C3,6,7,8,13
[ ] (2) 10uF	C2,14

Diodes

[ ] (2) 1N4148 Silicon Diode	D2, D3
[ ] (1) 1N5817 Schottky Diode	D1

Light Emitting Diodes

[ ] (1) Red LED	DS1

Inductors

[ ] (5) Ferrite Beads	L1,2,3,4,5

Integrated Circuits

NOTE: Do not handle the ICs at this time! Carefully remove the black foam carrier with ICs from the bag and verify the ICs against this list. Do not touch the ICs! Leave them in the protective foam.

[ ] (1) MC68HC908GP32CP	U2
[ ] (1) MAX232 or HIN232	U1

Voltage Regulators

[ ] (1) 7805 TO-220 case	U3

Crystal Oscillator

[ ] (1) 32.000MHz	U4

IC Sockets

[ ] (1) 16-pin DIP Socket	U1
[ ] (1) 40-pin DIP Socket	U2

Switches

[ ] (1) DPDT Momentary push button (6-pin)	SW1
[ ] (4) Round momentary switches	SW1-4 (button)

LCD Module

[ ] (1) 4 line by 20 character LCD module

Connectors

[ ] (2) 2x5-pin male header	J2, P1(button)
[ ] (1) 2x8-pin male header	J3
[ ] (1) 1x16-pin male header	J1
[ ] (1) 1x16-pin female header	J1(LCD)
[ ] (1) DE9 Female PCB right angle mount	J4
[ ] (1) 2.1 mm Coaxial Jack	J6
[ ] (1) 2.1 mm Coaxial Plug	J6
[ ] (1) RJ-11 6-6 connector PCB mount	J5

Miscellaneous
[ ] (1) Printed Circuit Board (Rev B)
[ ] (1) Main Board Assembly Manual (this document)
[ ] (1) Operations Manual
[ ] (1) TO-220 heat sink
[ ] (1) 6-inch 10-conductor IDC cable with connectors
[ ] (1) 4-40 x ¼" screw
[ ] (1) 4x40 nuts
[ ] (4) 6-60 3/8" spacers
[ ] (4) 6-60 1/2" spacers
[ ] (2) 6-60 5/8" spacers
[ ] (4) 6-60 screws

Construction Notes

You are now ready to begin construction of the T238+. Follow these standard construction practices when building the unit.

Use a temperature-controlled, fine-tipped soldering iron of relatively low wattage (25 watts maximum, 15 watts is ideal) and a good quality 60/40 or 63/37 rosin-core solder for construction. Keep the tip of your soldering iron bright and clean, wiping it frequently on a wet rag or sponge. Make solder joints carefully, but swiftly. Prolonged heat on a PC board pad can be as disastrous as it can ruin the PC board. Two to three seconds should be enough time to apply heat to any joint. Due to the proximity of some of the traces on the PC board, solder bridges are a very distinct possibility. Following the important points above could eliminate several hours of troubleshooting (or worse). This is good practice when working on any kit.

You will need small flush or semi-flush cutting pliers and small-tipped long nosed pliers. A magnifying glass may prove helpful to identify the values of the small components.

Pay careful attention to the directions that follow:

1) Read this entire document prior to starting construction of your kit.

2) Identifying the pins on headers and ICs - Pin 1 is identified by the square pad on silk-screen.

Electrostatic Protection
The Integrated Circuits and crystal oscillators are susceptible to static discharge. Observe anti-static precautions when assembling the T-238+. Never handle the ICs without grounding yourself, the protective foam, and the PCB first. Always touch and hold the foam first before touching the IC. Always touch and hold the printed circuit board before inserting the IC into its socket. Never leave the ICs on anything but the foam or the PCB.

Construction

Refer to the layout diagram for clarification of parts placement. All references to up, down, left, and right assume that you are looking at the PCB side with the TAPR logo with the text right side up. Sample pictures of the overall construction are available on the T238 website.

Resistors
Resistors have a lead spacing of 0.4" and should lie flat on the PC board. You may wish to use a lead former to pre-form the resistor leads for neatest appearance.

NOTE: – Save the clipped leads as you will use them to solder the ferrite beads in place in a later step.

Install the following 5 resistors:
[ ] R1 2.2K ? (red-red-red-gold)
[ ] R2 10K ? (brown-black-orange-gold)
[ ] R3 10K ? (brown-black-orange-gold)
[ ] R4 10 ? (brown-black-black-gold)
[ ] R5 4.7 ? 1/2 Watt (yellow-violet-gold)

NOTE: R5 is a ½ watt resistor. It is slightly larger than all the other resistors. The lead spacing is 0.5".

WARNING! – Be careful when clipping leads, as they have a tendency to fly towards your eyes! Take appropriate precautions (grasp leads and wear eye protection).

[ ] Solder and clip the leads (10 leads total)

[ ] R8 10K ? (brown-black-orange-gold)
[ ] R9 470 ? (yellow-violet-brown-gold)
[ ] R16 1.5K ? (brown-green-red-gold)
[ ] R18 10K ? (brown-black-orange-gold)
[ ] R19 10K ? RPACK (pin 1 closest to J2)

[ ] Solder and clip the leads (10 leads total)

[ ] R20 20K ? (red-black-orange-gold)

[ ] Solder and clip the leads (2 leads total)

Now check your work. All leads should be soldered. There should be no solder bridges or cold solder connections.

[ ] OK so far.

This completes the resistor installation. You should have no remaining resistors.

[ ] No resistors remaining.

Diodes
Diodes are polarity sensitive devices. Diodes are mounted flat near the surface of the board like the resistors previously installed. The cathode end of the diode is banded and corresponds to the banded silk-screen legend on the PCB.

Install the following diodes:
[ ] D1 1N5817 Schottky
[ ] D2 1N4148
[ ] D3 1N4148

[ ] Solder and clip the leads (6 leads total)

Now check your work. All leads should be soldered. There should be no solder bridges or cold solder connections.

[ ] OK so far.

This completes the diode installation. You should have no remaining diodes.

[ ] One LED remaining.

IC Sockets

NOTE: If any socket pins are bent, carefully straighten them with a pair of long-nose pliers before assembly. Some types of IC sockets have crimps in the pins to hold them in place when automatic wave soldering is performed. These sockets may be tricky to install if you are not familiar with them. If your kit contains these sockets, you may want to straighten the pins before attempting to insert them into the PC board.

When installing IC sockets double check to ensure that the socket is seated properly against the board with the notch matching the silk-screen. Pin 1 (nearest the socket notch) has a square solder pad. Be sure that all IC socket pins are showing on the solder side of the board. Next, tack-solder two diagonally opposite corners first (such as pins 1 and 8 on a 14-pin socket).

Then solder the remaining pins of that socket before proceeding to the next one. If you find a socket is difficult to install, remove it and double-check for a bent pin.

CAUTION! – Take care to avoid solder bridges!

Install the following IC sockets. Align the notch in the socket to the notch in the silkscreen outline:
[ ] U1 16-pin
[ ] U2 40-pin

Now check your work. All leads should be soldered. There should be no solder bridges (a blob of solder that shorts two adjacent soldered connections) or cold (gray and/or grainy looking) solder connections.

[ ] OK so far.

This completes the IC socket installation.

Ceramic Capacitors

All capacitors should be mounted as nearly flush to the board surface as practical without stressing the leads.
Install the following capacitors:
[ ] C1 0.1 uF (104)
[ ] C4 0.1 uF (104)
[ ] C5 0.1 uF (104)
[ ] C9 0.1 uF (104)
[ ] C12 0.1 uF (104)

[ ] Solder and clip the leads (10 leads total)

Electrolytic and Tantalum capacitors

Electrolytic and Tantalum capacitors are polarized. The positive lead goes in the hole on the board marked with a "+". Be careful! Typically the negative lead is marked but sometimes the positive lead is marked.

NOTE: The orientation of the positive "+" leads are not all the same.

[ ] C3 1 uF (105 or 1+35)
[ ] C6 1 uF (105 or 1+35) (text obscured, next to C7)
[ ] C7 1 uF (105 or 1+35)
[ ] C8 1 uF (105 or 1+35)
[ ] C13 1 uF (105 or 1+35)

[ ] Solder and clip the leads (10 leads total)

[ ] C2 10 uF (106 or 10uF)
[ ] C14 10 uF (106 or 10uF)

[ ] Solder and clip the leads (4 leads total)

Now check your work. All leads should be soldered. There should be no solder bridges or cold solder connections.

[ ] OK so far.

This completes the capacitor installation. You should have no remaining capacitors.

[ ] No capacitors remaining.

Ferrite Beads
Ferrite beads are cylinder shaped. String a clipped lead from a resistor or capacitor though the ferrite bead. Position the ferrite bead over the silkscreen outline and insert the leads into the solder holes. Solder the leads and clip them flush on the bottom of the PCB.

Install the following ferrite beads:
[ ] L1
[ ] L2
[ ] L3
[ ] L4
[ ] L5

[ ] Solder and clip the leads (10 leads total)

Trimpot
Align the trimpot according to the three solder leads. **Note:** the silkscreen is incorrect! It shows the small plastic tabs on the wrong side.

Install the trimpot:
[ ] V1 10K ohm

[ ] Solder and clip the leads (3 total)

Light Emitting Diode (LED)
LEDs are polarized components. The flat side on the base and the shorter lead identifies the cathode lead. Insert the LED according to the silk-screen outline, the flat side or short lead to the flat side of the silkscreen

NOTE – Mounting the LED on this board assumes you do not want to see the heartbeat LED once the project is put in a chassis. If you want to see the heartbeat after the project is in a chassis, save the LED and mount it on the button board instead.

[ ] DS1 Red LED (if you want it here)

[ ] Solder and clip the leads (2 total)

Male Headers
The male headers will be installed next. **Except for J1**, the plastic body of the part should rest flush with the top surface of the PC board. Note that the short end of the pins go into the PC board, the long end sticks up**. J1 is a special case that will be handled later.**

WARNING! – Do not hold these parts with your fingers while soldering. The pins get very hot.

Place a 2-pin jumper on the header to insulate your finger from the pins, hold the header in place and tack solder one pin. Check for proper alignment. If alignment is off, you can reheat the one pin to adjust. Once the alignment is correct, solder the rest of the pins and then reflow (reheat) the first pin you soldered.

NOTE: It is important to ensure these headers are installed flush so they come out of the board vertically. This is especially important for J3, the modem header.

[ ] J2 2x5 male header **DO NOT INSTALL J1 YET!**
[ ] J3 2x8 male header

Note that P2 and P3 are not stuffed for normal use, they are for debugging or special applications.

Connectors

Gently rock the RJ-11 connector until it snaps into place and solder the leads.

[ ] J5 RJ-11 connector

CAUTION! – Soldering the coaxial power connector is a bit tricky because of the large mounting holes.

[ ] J6 Coaxial power connector

Install the DE-9 connector. Solder in the tabs that hold the connector in place first, then solder the pins.

[ ] J4 DE-9 female

[ ] Solder the leads

Switches

Install the square DPDT momentary switch. The orientation matters! Look at the plunger to see which side has a depression in it. Install the switch with that depression "up" (toward the potentiometer).

[ ] SW1 DPDT Momentary Push Button

[ ] Solder the leads

Voltage Regulator
Refer to the assembly diagram at the end of this document for how to assemble the voltage regulator and heatsink. The regulator will be mounted with its metal tab against the heatsink and that in turn compressed to the component side of the board. This needs to be a secure mechanical connection for decent heat flow to both the heatsink as well as the main board as the ground plane is being used as a heatsink too. While heat sink grease would be nice, it is not necessary.

Temporarily mount the assembly to the board and note (with a pen or scribe) where to bend the regulator leads to line up with their solder pads. Remove the regulator, bend the leads 90 degrees, and then re-assemble the heatsink/regulator assembly.

Double-check the orientation per the paragraph above and the assembly diagram at the end of this manual.

[ ] Solder U3, 7805 assembly

[ ] Solder and clip the leads (3 total)

Crystal Oscillator

The crystal oscillator for this project looks like an IC except it has only four pins. Note the pin 1 marking on the IC (round dot) and make sure the oscillator is oriented correctly.

NOTE: If you feel you may want to develop software for the T238+, It may be a good idea to socket the oscillator. As we don't expect many people to become software developers, a socket is not being included in the kit.

[ ] Solder U4, Oscillator

LCD Module

Now you will prepare the LCD module for connection to the T-238+ PCB. The LCD module connects to the PCB via a male/female set of connectors. Refer to the assembly drawing at the end of this manual.

[ ] Carefully remove the LCD module from its protective packaging. If it has a protective plastic cover over the glass, leave it on.

You will be soldering a 1x16-pin male header to the back of the LCD module. During this time you need to protect the LCD screen from splatter while soldering.

[ ] Ensure the LCD screen is protected.

[ ] Loosely insert the 16-pin male header into the female header just to the point where there is some resistance to the insertion.

[ ] Install the four 1/2" spacers on the back of the LCD module. Install them by screwing in the 3/8" spacers into them. The 3/8" spacers should be on the front of the display with their threads going through the LCD module into the 1/2" spacer.

[ ] Install the male side of the 16-pin header pair into the back of the LCD module. Line up the two boards so the threads from the 1/2" spacers go into the holes of the main board. Squeeze the two boards together, compressing the headers until the spacers are firmly seated. This will ensure accurate alignment of the connector while soldering. Temporarily install two more 5/8" spacers on the threads of the 1/2" spacers to hold the two boards together for soldering. Verify your assembly looks like the assembly drawing at the end of this document.

[ ] Solder both 1x16-pin headers.

[ ] Unscrew the two temporary 5/8" spacers, separate the two boards, and put the LCD module back in it's protective packaging.

Almost done with the main board
At this point, these should be the only items left in the main board kit:

NOTE: Keep U3 and U4 in the antistatic foam

[ ] U3 MC68HC908GP32
[ ] U4 MAX23

[ ] 2.1 mm Coaxial Plug

Board Check
You are now complete with the soldering of the main board. Other than the two ICs mentioned, the only component locations that should not have anything loaded are P2 and P3. These are for debug and program development use and should not normally be needed.

DS1 may or may not be loaded depending on where you want your activity LED to be.

T238+ BUTTON BOARD

Construction

Refer to the layout diagram for clarification of parts placement. All references to up, down, left, and right assume that you are looking at the PCB with the text right side up. The front of the board is the side with button text (UP, DOWN, SEL, MENU). Switches and the LED are installed on this side; **the header is installed on the back side.**

Switches

Install the four SPST momentary switches. Note the flat end of the switch and line it up with the flat edge marked on the silkscreen. All four switches are the same.

[ ] SW1 SPST Momentary Push Button
[ ] SW2 SPST Momentary Push Button
[ ] SW3 SPST Momentary Push Button
[ ] SW4 SPST Momentary Push Button

Light Emitting Diode (LED)

LEDs are polarized components. The flat side on the base and the shorter lead identifies the cathode lead. Insert the LED according to the silk-screen outline, the flat side or short lead to the flat side of the silkscreen

NOTE – Mounting the LED on this board assumes you did not mount it on the main board.

[ ] DS1 Red LED (if you want it here)

[ ] Solder and clip the leads (2 total)

Male Header

The male header will be installed next. The plastic body of the part should rest flush with the top surface of the PC board. Note that the short end of the pins go into the PC board, the long end sticks up.

WARNING! – Do not hold these parts with your fingers while soldering. The pins get very hot.

NOTE: This header must be installed on the back of the board.

Place a 2-pin jumper on the header to insulate your finger from the pins, hold the header in place and tack solder one pin. Check for proper alignment. If alignment is off, you can reheat the one pin to adjust. Once the alignment is correct, solder the rest of the pins and then reflow (reheat) the first pin you soldered.

[ ] P1 2x5 male header

Power Up And Initial Testing:

In this section you will slowly power-up the board and check for assembly errors. It is important that you follow these steps carefully. You will systematically apply power to sections of the board while checking voltages to find and eliminate errors. Errors take two forms, the kind that damage components and the kind that don't. Neither is desirable, but the kinds that damage components will require you to find a replacement part.

Pre-Power Check

Before applying power for the first time, please check the following:

[ ] ICs U1 and U2 are NOT installed.

[ ] Check the polarity of all the electrolytic or tantalum capacitors. The capacitors negative side is usually identified with a white stripe and a big minus sign on it. The positive side is identified on the PCB. Not all the capacitors have the same orientation; double-check them with reference to PCB silkscreen.

[ ] Check the polarity of diodes D1, D2, and D3. The band on the diode should match the extra stripe on the PCB silkscreen.

[ ] Headers P1, J1, J2, j3, J4, J5 and J6 are not connected.

[ ] The LCD module should not be plugged in.

First Power Check

The T238+ kit does not come with a power supply. An ordinary wall transformer ("wall wart") that provides 7 to 12 Volts DC at 300 milliamps will work fine. The closer you can get to 7V the cooler the regulator will be. Be warned that may "12V" wall transformers supply much higher voltages even under load. The T238+ is fitted with a 2.1mm power coaxial connector such that the center conductor is positive and the outer barrel is ground. Measure this with a voltmeter to confirm both the voltage, but more importantly, the polarity. The Radio Shack Model 19-1120/19-1140, supplied with the HTX-202 and the HTX-404 handheld radios, is suitable for this application.

[ ] The voltage is between 9 and 12 Volts DC and the center conductor is positive and the barrel is ground

[ ] Apply power to the T238+. Observe anything unusual such as components heating up, smoke or smell. Remove power immediately if anything unusual appears.

Place the ground lead of your voltmeter on the center pin of U3. This will be the ground reference for the rest of the measurements.

[ ] Measure the voltage on the voltage regulator U3 on the pin closest to the edge of the board. It should be between 8 and 14V. If it is 0V, then L5 is likely missing or damaged. If it is low, the transformer may not be rated for the right voltage or something may be overloading it.

[ ] Measure the output voltage on the voltage regulator U5 on the pin furthest from the edge of the board. It should be between 4.90V and 5.10V. If it is low, something could be overloading the regulator and whatever is doing that should be hot.

WARNING! – Don't proceed until the voltages in and out of the voltage regulator U5 are within proper limits.

[ ] Voltages in and out of voltage regulator U5 are within limits.

NOTE – In the sections below, there are frequent references to +5V, which actually means the voltage between 4.90V and 5.10V that you measured above.

With the ground lead of your voltmeter still on U3 pin 2 (center), confirm all the following test points also measure +5V. Remedies are listed if you do not see +5V.

Check	Pin	Remedy if not +5V
[ ]	U1 pin 16	Check socket.
[ ]	J5 pin 5 (pin 1 closest to board edge)	Check socket and L4
[ ]	J5 pin 3	Check socket, L1, and R16
[ ]	U2 pins 1, 20, and 31	SW1 possibly installed backwards. Check by pressing SW1. If voltage appears when the switch is pressed, it is in backwards.
[ ]	U4 Pin 14 (above square pad)	Check socket

Next install the button board cable and the button board. Convention has the brown wire as Pin 1, but as long as you are consistent on both ends it does not matter. Check the four switches following the table below. For each pin on the microcontroller, verify that the voltage is normally 5V. Pressing the switch makes the voltage go to 0V. If the voltage is always low (regardless of the switch), then likely the RPACK is not installed correctly or the switch is installed 90 degrees off.

Check	U2 Pin number	Switch
[ ]	U2-26	SW1 (up)
[ ]	U2-27	SW2 (down)
[ ]	U2-28	SW3 (select)
[ ]	U2-29	SW4 (menu)

Using a small wire bent in a U-shape, short between U2 pins 20 and 21. Do this carefully so you don't damage the IC socket. The LED should light (wherever you installed it, main board or button board). If it does not, there is a problem with R12 or more likely, the LED is installed backwards.

[ ] LED works

[ ] Remove power from the T238+.

Board Level Power Check

Next you will insert U1 and U2. Make sure you match the pin-1 notch of the device to the notch on the silkscreen. There are a lot of pins on U2 and it can be difficult to line them up with the socket. Your best bet is to place the chip sideways on a flat static-safe surface so that one row of the pins all lie flat on the surface. Carefully rotate the body of the IC while pushing down on the pins. Rotating while pushing down should bend all the pins together and bend them where they protrude from the body. Bend the pins a little at a time, then check to see if they are perpendicular to the body of the IC. Once perpendicular, the pins of the IC will be much easier to insert as they line up with the socket.

Insert U2 slowly. Observe that all the pins slide into the socket. Watch for pins that bend under the IC or out of the IC socket.

[ ] Insert U2, MC68HC908GP32

[ ] The notch on the chip is aligned with the silkscreen notch on the PCB.

[ ] Apply power to the T238. The LED will flash three times quickly, and after approximately 10 seconds it should start flashing once per second. If it does not flash immediately, remove power and perform all the above checks again as a precaution. If they are all OK, the only thing left is the clock. Check U4 for any errors or soldering problems.

If the LED still does not start flashing, make sure that all the pins in U2 were inserted cleanly into the socket and check the following components carefully: R1, R2, R3, R8, and R11. Do not proceed until the LED flashes as described above.

[ ] LED flashing approximately once per second

Remove power, then insert U1 slowly. Observe that all the pins slide into the socket. Watch for pins that bend under the IC or out of the IC socket.

[ ] Insert U1, MAX232

With the ground lead of your voltmeter on pin 15 of U1, check the following voltages:

Check	Pin	Voltage	Remedy
[ ]	U1 Pin 16	+5	Check IC socket
[ ]	U1 Pin 2	Greater than 8V	Check IC socket, C3, C6, C7, C8
[ ]	U1 Pin 6	Less than -8V	Check IC socket, C3, C6, C7, C8

[ ] Remove power from the T238+.

Attaching the LCD Module

[ ] Re-install the LCD module as you did when you soldered the connectors and per the assembly drawing at the end of this document. Refer again to the assembly drawing at the end of this manual.

[ ] Adjust VR1 to mid-position.

[ ] Apply power to the T238+.

[ ] After about 1 second you should see the opening banners displayed on the LCD module. If the display is blank, shows overly dim or dark text, or nothing but blocks, adjust VR1, the LCD contrast adjustment. Adjust VR1 for best viewing. If text cannot become visible, check the soldering on U2 and J1 as well as the header on the LCD module.

[ ] LCD showing legible text.

During the power up sequence, you will get a message saying that there is a 1-wire bus error. If the error says no devices are present, you are OK. If it says the bus is shorted to ground, then check U2, J5, D2, D3, R16, and L1 for assembly problems or bad solder joints.

Do not select any menu options at this time.

[ ] Received error message saying no devices on the bus.

If you haven't already done so…

[ ] Assemble the 1-Wire™ Weather Sensor using the instructions that came with the unit.

[ ] Confirm that the Weather Sensor is operational with the PC software that came with the unit.

[ ] Weather station sensor unit works with PC software

[ ] Remove power from the T238+.

[ ] Plug in the telephone cable supplied with your sensors into J5 and in the weather sensor unit. A good approach here is to mount the 1-Wire weather station sensor unit to a bench with a C-clamp, aligning the aluminum square tube in the same orientation that it will have when the weather sensor unit is permanently mounted outside.

[ ] Apply power to the T238+. The LCD should display its opening messages. If the LCD says that the 1-Wire bus is shorted then most likely the 1-Wire bus has been reversed. This will not cause any damage, but the unit will not work. Some telephone cables reverse the wires and some do not. In addition, some telephone cord couplers reverse the wires. Find a telephone cable that does not reverse the wiring.

Hold the connectors side-by-side and oriented the same direction (for example, the clip tab down on both connectors). Observing the red and green wires inside. If, for example, red is on the left in both connectors, then this cable does not reverse the signal wires.

When the weather sensor is connected properly, the LCD will not give any bus error messages. It will complain that it has no IDs and will ask that your run setup. This is perfectly normal.

[ ] No bus error messages at all when powering up the weather station.

Congratulations!

You have completed the power-up and initial testing phase! If you do not intend to use the T238+ modem you are done and can start to operate the T238+. Otherwise, continue with the modem board assembly next.

To learn how to setup and operate your T238+ APRS™ 1-Wire™ Weather Station, please refer to the T238 Operations Manual. It is available on the T-238 web site at

http://www.beals5.com/wx
or
http://www.tapr.org/

and follow the links for the T238+.

Assembly Drawings: