AMERICAN QRP CLUB

~ Assembly Manual ~

Micro908 Antenna Analyst

The Micro908 is a flexible and re-usable control platform for ham radio projects. The Micro908 platform is designed to be easily operated on the bench as well as in the field. It is comprised of a single 5" x 5" printed circuit board containing all components, connectors, controls, LCD, and two daughtercards. The plastic enclosure contains an 8-cell AA battery back enabling convenient field use. A number of standard ham radio peripherals may be connected to the Micro908: antenna, paddles, a PC-style keyboard, headphones, an audio line to drive an SSB transceiver, a keyline to drive a transmitter, your rig's audio in/out signals, and custom control lines via an auxiliary jack. The first major software available for the Micro908 platform is the ***Antenna Analyst*** *– an instrument that automatically determines SWR and complex impedance characteristics of an HF antenna system. Advanced features of DDS frequency control, LCD tuning, PC data collection and plotting, numerous operating modes and easy software upgradability make this instrument attractive for homebrewers and antenna enthusiasts.*

CONTENTS

Section 1: Introduction

Welcome to the Micro908 Antenna Analyst Kit, a reprogrammable and multi-use microcontrolled instrument that will provide years of reliable service in both the shack and the field when constructed according to this assembly guide.

This project involves the soldering of small surface mount technology (SMT) parts and other delicate components to a printed circuit card, and it will take about six hours to assemble – perhaps several evenings or over a weekend. Once the kit is assembled, the pre-programmed controller (the HC908 Daughtercard) may be inserted into place and immediate indication of product operation will be seen.

This Assembly Manual provides detailed, step-by-step instructions on preparation of the components, installation of them to the printed circuit board and enclosure, and basic calibration and operation of the instrument. Other documents provided on the enclosed CDROM describe the technical overview and more detailed usage. Several useful software programs are also contained on this Micro908 Resource CDROM, such as a software loader application and a terminal program. Additionally, the CD contains information and full software source code that will be useful for those wishing to develop custom software for the Micro908 platform.

Please regularly visit the Micro908 project website (**www.amqrp.org/kits/micro908**) to download updated versions of the software, manuals and schematics. We are also maintaining a list of frequently asked questions (and answers) that will surely be of help to you in building and using this kit. An even more complete and up to date Assembly Manual, with additional photos (all in color) may be found at the Micro908 online site.

We've made every effort possible within reason to make construction and use of the Micro908 Antenna Analyst a success for the builder. Please let us know how it works out for you or if you have any questions along the way. We thank you for purchasing the Micro908 Kit from the AmQRP Club and we wish you good luck in building and using it!

Sincerely yours,

"The Micro908 Development Team"

George Heron, N2APB *n2apb@amqrp.org*

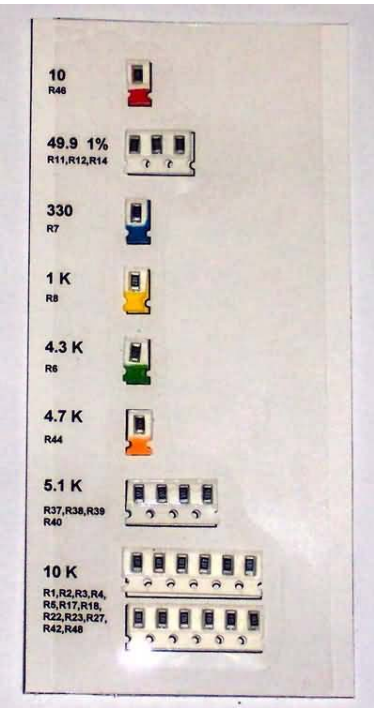
Joe Everhart, N2CX *n2cx@amqrp.org*

Tom (W8KOX) & Nancy (NJ8B) Feeny, *w8kox@amqrp.org*

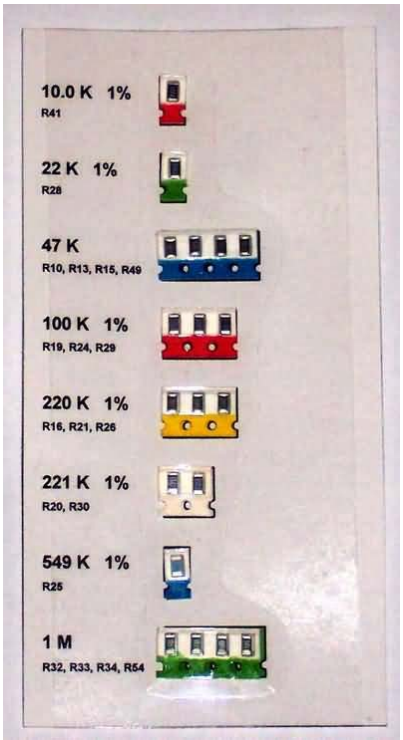
Paul Maciel, AK1P *ak1p@amqrp.org*

Section 2: Parts Inventory

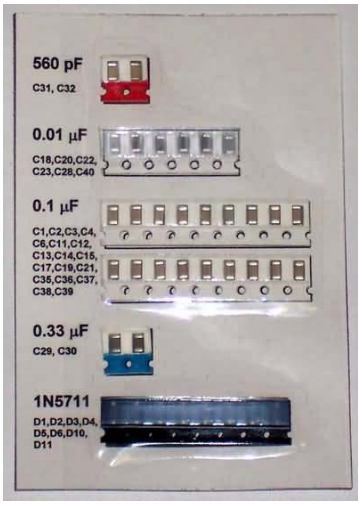
Carefully review the contents of each parts bag and component supplied in the kit to ensure that you have everything needed at the start of the project. If a part is missing, please contact us by email and we'll get it out to you right away.



SMT Cards Bag		
Resistor Card 1		
1	R46	Resistor, 10, SMT, 1206
3	R11, R12, R14	Resistor, 49.9, SMT, 1206, 1%
1	R7	Resistor, 330, SMT, 1206
1	R8	Resistor, 1K, SMT, 1206
1	R6	Resistor, 4.3K, SMT, 1206
1	R44	Resistor, 4.7K, SMT, 1206
4	R37, R38, R39, R40	Resistor, 5.1K, SMT, 1206
12	R1, R2, R3, R4, R5, R17, R18, R22, R23, R27, R42, R48	Resistor, 10K, SMT, 1206

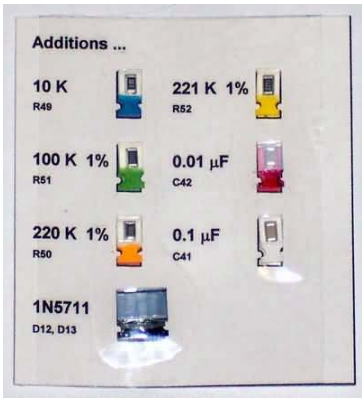


SMT Cards Bag		
Resistor Card 2		
1	R41	Resistor, 10.0K, SMT, 1206, 1%
1	R28	Resistor, 22K, SMT, 1206, 1%
4	R10, R13, R15, R49	Resistor, 47K, SMT, 1206
3	R19, R24, R29	Resistor, 100K, SMT, 1206, 1%
3	R16, R21, R26	Resistor, 220K, SMT, 1206, 1%
2	R20, R30	Resistor, 221K, SMT, 1206, 1%
1	R25	Resistor, 549K, SMT, 1206, 1%
4	R32, R33, R34, R54	Resistor, 1M, SMT, 1206



SMT Cards Bag		
Capacitor & Diode Card		
2	C31, C32	Capacitor, 560 pF, SMT, 1206
6	C18, C20, C22, C23, C28, C40	Capacitor, 0.01 uF, SMT, 1206
18	C1, C2, C3, C4, C6, C11, C12, C13, C14, C15, C17, C19, C21, C35, C36, C37, C38, C39	Capacitor, 0.1 uF, SMT, 1206
2	C29, C30	Capacitor, 0.33 uF, SMT, 1206
8	D1, D2, D3, D4, D5, D6, D10, D11	Diode, Schottky, 1N5711, SMT

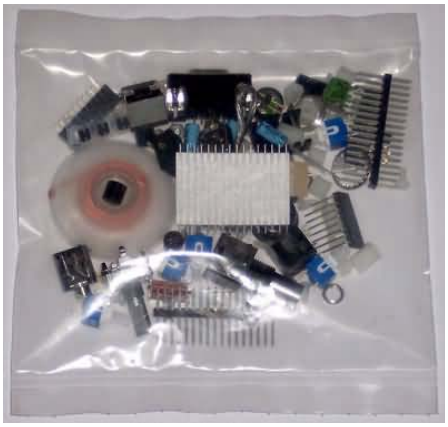
Note: 'C35' shown in the 0.1 uF line on the card should read 'C25'.



SMT Cards Bag		
Additions Card		
1	R49	Resistor, 10K, SMT, 1206
1	R51	Resistor, 100K, SMT, 1206, 1%
1	R50	Resistor, 220K, SMT, 1206, 1%
1	R52	Resistor, 221K, SMT, 1206, 1%
1	C42	Capacitor, 0.01 uF, SMT, 1206
1	C41	Capacitor, 0.1 uF, SMT, 1206
2	D12, D13	Diode, Schottky, 1N5711, SMT

NOTE: The 'R49" component on this card is a duplicate and will not be used.

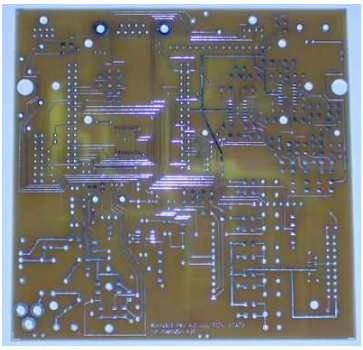
Battery Holder Bag		
1	LCD	Display, LCD, Hantronix, 16x2 STN, GRAY
1	J3	Battery holder, 8-AA cells
1		Flux Pen, No-Clean



Controls & Connectors Bag		
1	J1	BNC, pcb mount
1	J2	Serial port connector, DB9F
1	ENC	Rotary Encoder
1	J4	Coaxial pow er connector, 2.1mm
1	J8	Socket, 1x12 pos'n (DSP out)
1	J9	Socket, 2x10 position, (DSP in)
1	J11	Mini-DIN, 6 pos'n (KBD)
3	J6, J7, J12	Audio jack, 1/8", pcb mount
1	J10	Socket, 1x8 pos'n, right angle
1	J14	Mini-Din, 8 pos'n (AUX)
1	W1	Jumper, Flexstrip, (LCD)
2	P1, P2	Pinheader, 2x34 (HC908) (cut 2 from 2x72 strip)
2	P3, P4	Pinheader, 1x2 pos'n (MON) (cut from 2x72 strip)
3	C7, C33, C34	Capacitor, 1 uF, Electrolytic, SMT
2	C5, C26	Capacitor, 10 uF, Electrolytic
1	C27	Capacitor, 47 uF, Electrolytic
1	C24	Capacitor, 100 uF, Electrolytic
1	R45	Potentiometer, 10K, miniature, pcb mount
1	S2	Slide sw itch, pcb mount, SPST
5	PB1, PB2, PB3, PB4, PB5	Pushbuttons, SPST, momentary contact
1	SPKR	Speaker, miniature, 32-ohm
5		Pushbutton caps (black)
1		Heatsink - TO220
1		Knob, 1/4"-dia shaft, (ENC)
1		Nut, zinc plated, 4-40 (U1)
1		Machine screw , zinc plated, round head, 4-40 x 0.5" (U1)
4		Spacer, nylon, hex tapped, 4-40x0.25" (LCD)
6		Spacer, nylon, hex tapped, 4-40x0.375" (PCB+DDS)
16		Machine screw , pan slotted, #4-40x0.25" (PCB+DDS+LCD)
4		Machine screw , flat slotted, #4-40x0.25" (PCB-cover)
1		Desoldering Braid
1		Solder, 63/37 Eutectic, No-Clean Flux, @28 ga.

Overlay Bag (Included with Enclosure Option)		
1	Front panel overlay	
1	Side panel overlay	
1	End panel overlay	
1	End panel (drilled)	
1	End panel (undrilled)	
4	Enclosure screw s	
4	Enclosure rubber feet	

Semiconductor Bag		
1	U1	Voltage regulator, 3-terminal, 7805
1	U2	Memory, SEEPROM, 512Mb
2	U3, U4	IC, Op Amp, LMC6484, SOIC
1	U5	IC, Audio Amp, LM386, SOIC
1	U6	IC, Level Translator, TC7SET08F, SOIC
1	LED1	LED, T1-3/4 (BUSY)
2	Shunt	shunt, 0.1", 2 pos'n
3	D7, D8, D9	Diode, Schottky, 1N5817, DO-41
1	VR1 - DDS Option	Voltage Regulator, 78L08, 8V (optional, for use with regarga
1	Q1	Transistor, NPN, 2N3904, TO92



PCB Bag		
1	PCB	PC Board

HC908 Daughtercard Bag (Optional)		
1	HC908	HC908 Daughtercard assembly

DSPx Daughtercard Bag (Optional)		
1	DSPx	DSPx for Micro908 assembly

DDS Daughtercard Kit Bag (Optional)		
1	DDS Kit	DDS Daughtercard Kit

Keyboard Cable Bag (Optional)		
1	Cable	Keyborad extension cable (6')

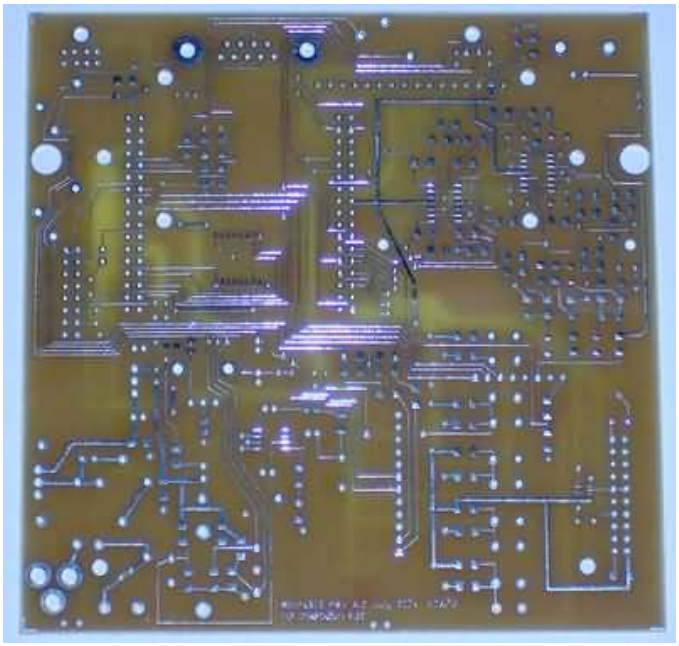
Keyboard (Optional)		
1	Keyboard	Dauphin Mini Keyboard

Enclosure (Optional)		
1	Enclosure	Pac-Tec LH-57 Enclosure (pre-drilled)

Section 3: PC Board Preparation

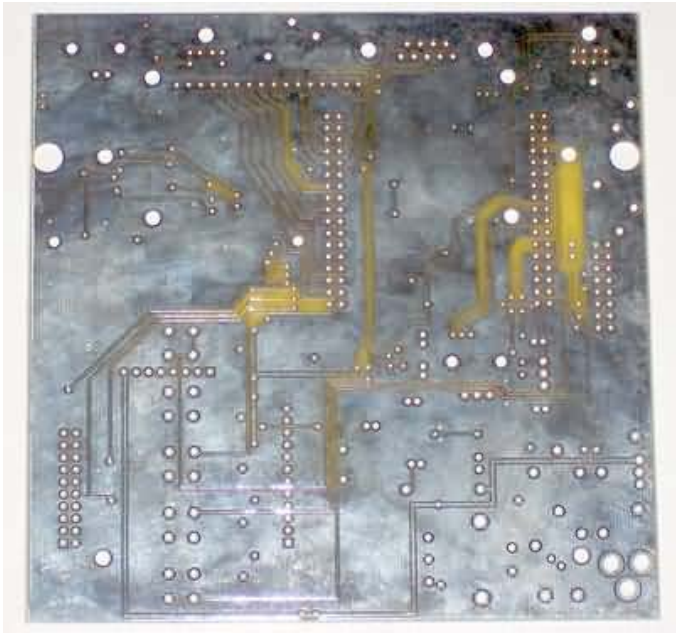
Meet the Micro908 PC Board! You should become familiar with the orientation nomenclature that we'll be using throughout this manual. (Note: The pc board shown here is one of the prototypes … the production board provided in the kit has green solder mask and white component legends to help with location identification and soldering.)

Top or "Component" Side



← Left Right →

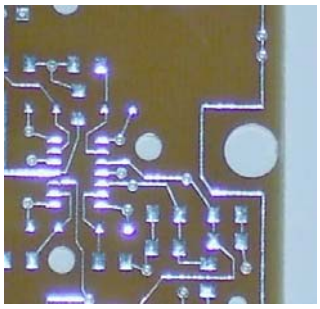
Bottom, or "Ground" or "Controls" Side



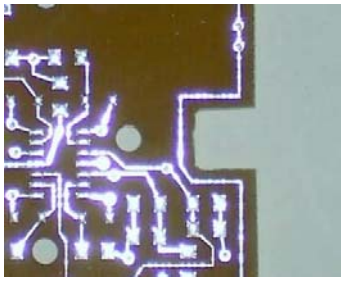
← Left Right →

If you intend on installing the pc board in the optional Enclosure, you will need to slightly modify it before starting assembly.

Using a flat file, convert the two largest holes on the left and right sides of the board as shown to allow the board to fit over the mounting posts on the inside of the enclosure. (If you are not using the Enclosure option, this step will not be necessary.)



Original hole (right side)

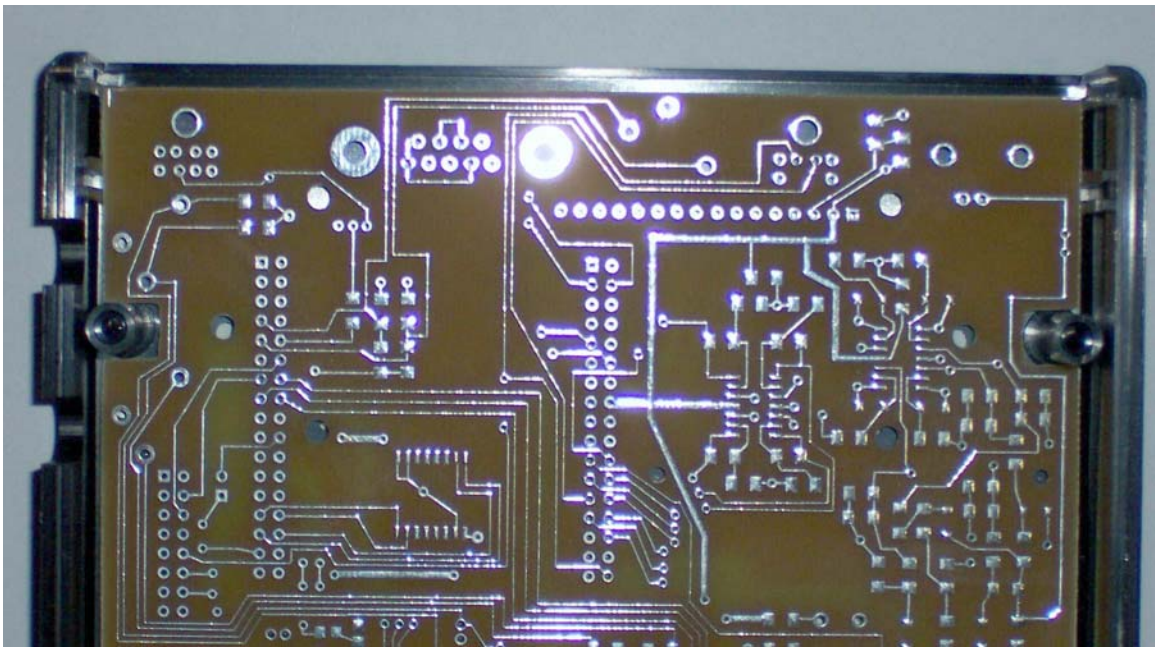


Modified hole (right side)

Do this for the left side hole as well. It is not critical as to exactly how large you make the rectangle – just don't file into the top side traces! (Actually, the larger the better, which will allow easier alignment when you assemble the pc board into the enclosure.

Using the flat file again, file the four outside edges of the overall pc board. Take off only about 1/32" all around the board in order to allow it to fit into the enclosure without any resistance.

When properly done, the pc board should easily drop into the top half of the enclosure as shown below. Note that the front edge of the board should reach up to the inside edge of the slot at the top. If the board extends any further than this, the end panel will not fit properly. File that front edge of the board to have the edge just meet the inside part of the slot. (You could take one of the enclosure end panels and drop it into the slot to ensure proper sizing of the pc board.



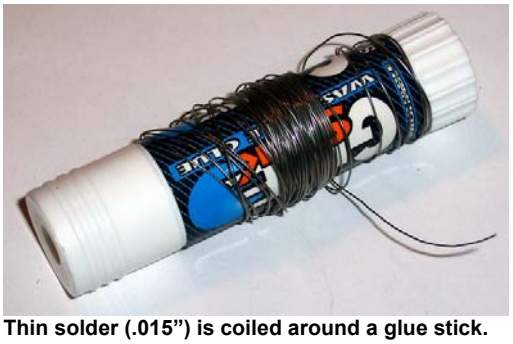
Section 4: Installing the Surface Mount Components

Preparing for the job

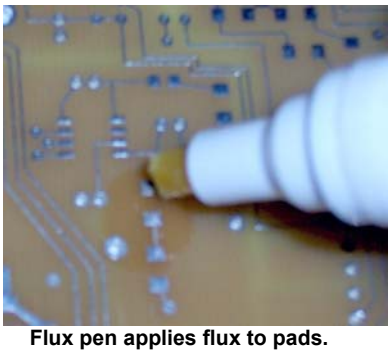
The key to being successful with any construction project is selecting and using the proper tools. For projects using SMT (Surface Mount Technology), the tools are easy to find**.** A magnifying lamp is essential for well-lighted, close-up work on the components. Tweezers or fine-tipped pliers allow you to grab the small chip components with dexterity. Thinner solder (.015") than you might normally use is preferred because of its being quicker to melt and smaller in solder volume on the component lead. Use of a super fine-tipped soldering iron make soldering the leads of these small parts straightforward and easy. A clean work surface is of paramount importance because SMT components often have a tendency to fly away even when held with the utmost care in tweezers – you'll have the best chance of recovering your wayward part if your table is clear. When the inevitable happens, despite your best efforts of holding an SMT part in your tweezers, you'll have lots of trouble finding it if it falls onto a rug-covered floor covered. It's best to have your work area in a non-carpeted room, for this reason as well as to protect static-sensitive parts.

Attaching SMT Components to the PC Board

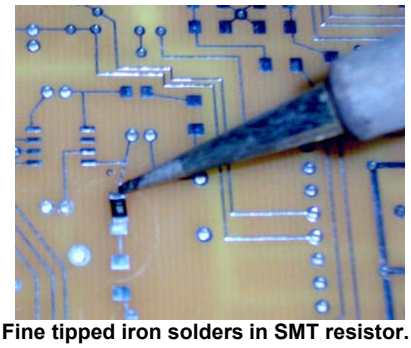
We've supplied two items in this Micro908 Kit that will greatly help you successfully solder these small surface mount components to the pc board. The first is a small coil of **.015" solder**. As described above, this thin solder is perfect for soldering small SMT parts. Just wrap the solder around a convenient tube as shown below on the left) so you can easily play out the solder as you go along in the board assembly. The other helpful item is the **Flux Pen**. By making the pads to be soldered wet with the liquid flux, you are greatly helping the joint be clean and ready to accept the soldered component. (Just press down a little on the tip of the flux pen and the liquid flux will start flowing out through the sponge tip.) In order to solder these small components in place, having a fine-tipped, 600-700 degree soldering iron is essential, as illustrated in the rightmost photo below.



Thin solder (.015") is coiled around a glue stick.



Flux pen applies flux to pads.

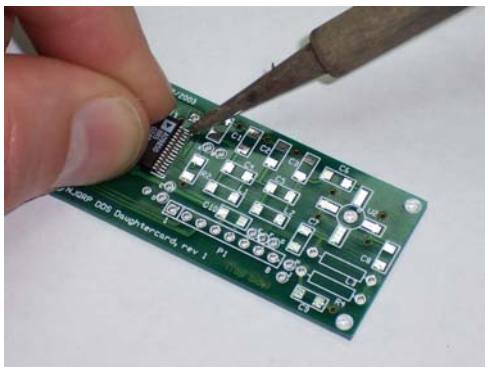


Fine tipped iron solders in SMT resistor.

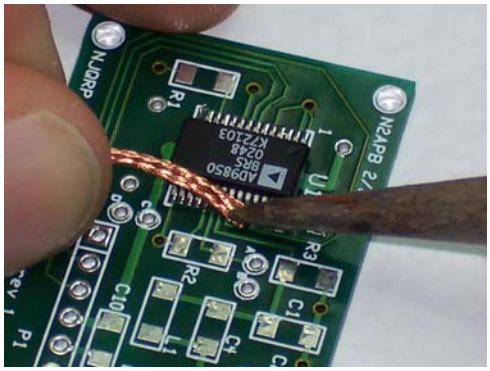
The trick to soldering surface mount devices to pc boards is to (a) pre-solder one of the pads on the board where the component will ultimately go; (b) hold the component in place with needle nose pliers or tweezers on the tinned pad; (c) re-heat the tinned pad and component to reflow the solder onto the component lead, thus holding the component in place; and lastly (d) solder the other end of the component to its pad.

Attaching a Surface Mount IC

There are four surface mount integrated circuits to attach on the Micro908 pc board: U2 (memory chip), U3 & U4 (op amps), U5 (audio amp) and U6 (a pretty tiny driver IC). Pre-solder the pad in one corner of the given layout then carefully position the leads of the IC over its set of pads on the pc board. I generally use my fingers to carefully align the IC over all its pads and then reheat the corner pad to reflow the solder onto the IC pin. This should leave the IC attached by that pin. Again making sure the IC pins are aligned over all pads, carefully solder the opposite corner lead to its pad. This should leave all other pins of the IC aligned over their respective pads, making it easier to solder them. Next solder each of the other pins to their respective pads, being careful not to bridge solder across any adjacent pads or pins. If this does happen, that's okay! Just grab some **solder wick** (also supplied in the Micro908 Kit) and use it to draw off the excess solder, which should be fairly easy and clean because of the solder mask on the circuit board.



First corner pin of surface mount IC being attached.



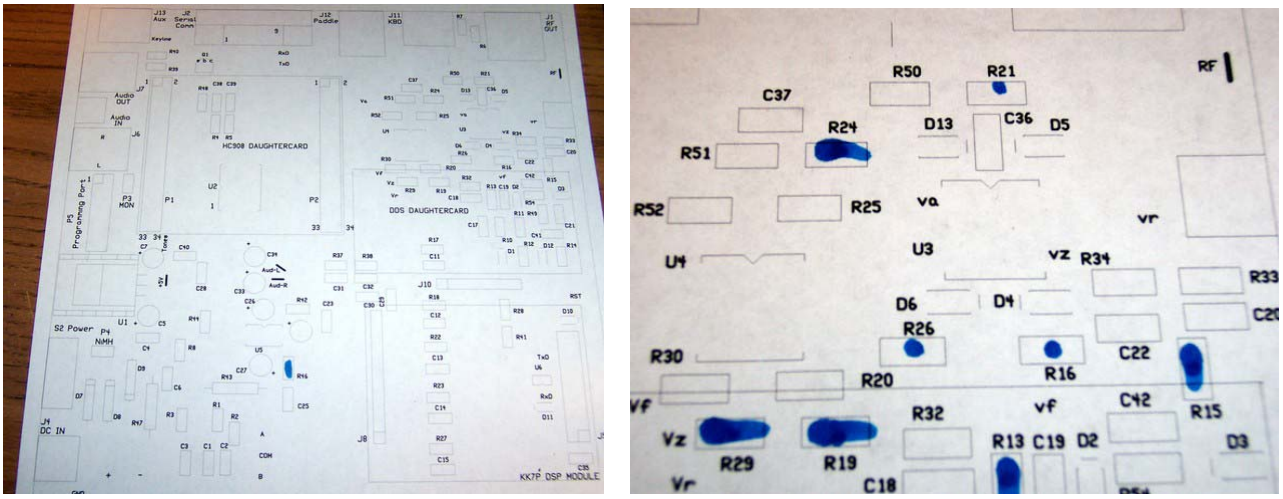
Solder wick easily absorbs excess solder between pins.

(IC shown being attached here is the DDS chip onto the DDS Daughtercard. The techniques are the same for the Micro908 ICs.)

Using the Component Layouts During Assembly

A helpful practice to develop is to mark the supplied Layout diagram as you install each component. As you go along, the diagram will fill up with more and more marks, enabling you to more easily find the location of the remaining components, and have confidence that you haven't omitted installation of a component along the way.

A useful marking technique is to identify with a "dot" the location of the parts you are about to install, making it easier for you to place and solder the part in the right spot. Once soldered in place, going back to the diagram and placing a full line in that same spot will indicate that you've soldered it in place, as shown in the photo on the right.



The component layout diagram is a useful tool if you mark it up as you proceed along in installing the components.

1) Install components from Resistor Card 1

Using the Component Layout Diagram in Appendix A as a guide, install the SMT components from the Resistor Card 1. Check off each row as you complete installing those components.

	QTY		
[ ]	1	R46	Resistor, 10, SMT, 1206
[ ]	3	R11, R12, R14	Resistor, 49.9, SMT, 1206, 1%
[ ]	1	R7	Resistor, 330, SMT, 1206
[ ]	1	R8	Resistor, 1K, SMT, 1206
[ ]	1	R6	Resistor, 4.3K, SMT, 1206
[ ]	1	R44	Resistor, 4.7K, SMT, 1206

[ ]	**4**	**R37, R38, R39, R40**	**Resistor, 5.1K, SMT, 1206**
[ ]	**12**	**R1, R2, R3, R4, R5, R17, R18, R22, R23, R27, R42, R48**	**Resistor, 10K, SMT, 1206**

2) Install components from Resistor Card 2

Using the Component Layout Diagram in Appendix A as a guide, install the SMT components from the Resistor Card 2. Check off each row as you complete installing those components.

	QTY		
[ ]	**1**	**R41**	**Resistor, 10.0K, SMT, 1206, 1%**
[ ]	**1**	**R28**	**Resistor, 22K, SMT, 1206, 1%**
[ ]	**4**	**R10, R13, R15, R49**	**Resistor, 47K, SMT, 1206**
[ ]	**3**	**R19, R24, R29**	**Resistor, 100K, SMT, 1206, 1%**
[ ]	**3**	**R16, R21, R26**	**Resistor, 220K, SMT, 1206, 1%**
[ ]	**2**	**R20, R30**	**Resistor, 221K, SMT, 1206, 1%**
[ ]	**1**	**R25**	**Resistor, 549K, SMT, 1206, 1%**
[ ]	**4**	**R32, R33, R34, R54**	**Resistor, 1M, SMT, 1206**

3) Install components from Capacitor & Diode Card

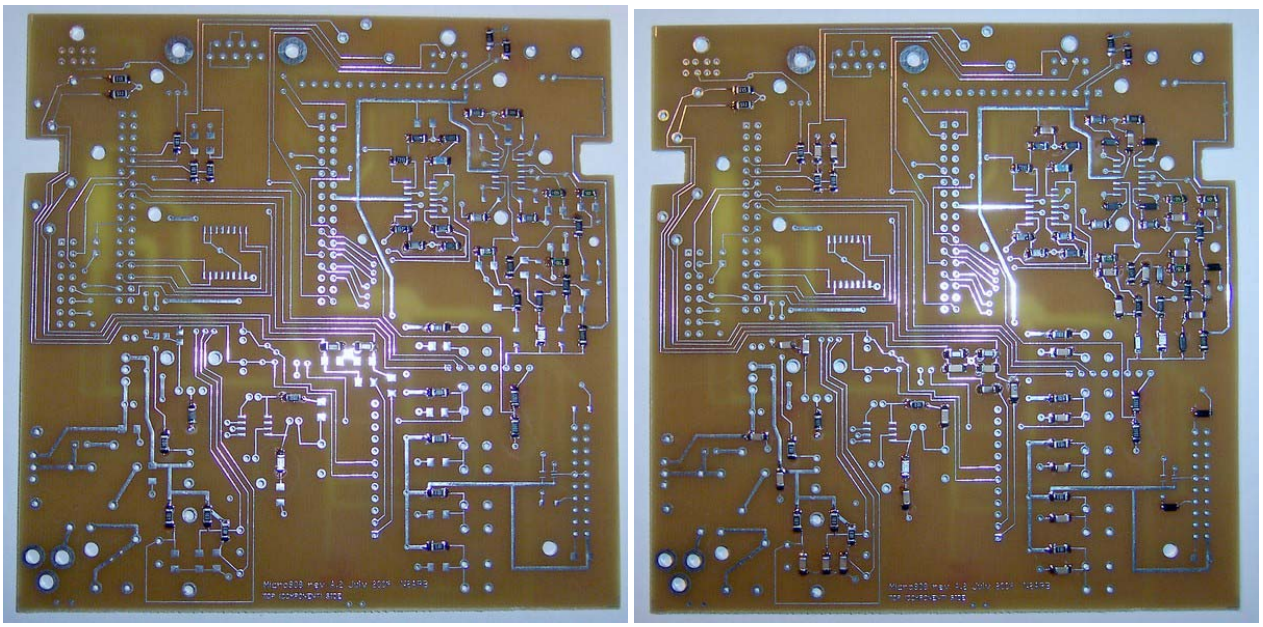
Using the Component Layout Diagram in Appendix A as a guide, install the SMT components from the Capacitor & Diode card. Check off each row as you complete installing those components. Be careful to identify the cathode of the diodes on this card. The cathode is the side of the diode with a single straight line on the schematic symbol, and with a (faint) single straight line on the package. You will surely need to use your magnifying glass to see this mark. Orient the end of the diode with the single straight line onto the pc board with the diode outline also containing the straight line indicating the cathode.

[ ]	**2**	**C31, C32**	**Capacitor, 560 pF, SMT, 1206**
[ ]	**6**	**C18, C20, C22, C23, C28, C40**	**Capacitor, 0.01 uF, SMT, 1206**
[ ]	**18**	**C1, C2, C3, C4, C6, C11, C12, C13, C14, C15, C17, C19, C21, C25, C36, C37, C38, C39**	**Capacitor, 0.1 uF, SMT, 1206**
[ ]	**2**	**C29, C30**	**Capacitor, 0.33 uF, SMT, 1206**
[ ]	**8**	**D1, D2, D3, D4, D5, D6, D10, D11**	**Diode, Schottky, 1N5711, SMT** ***(These diodes have faint cathode markings)***

4) Install components from the Additional Card

Using the Component Layout Diagram in Appendix A as a guide, install the SMT components from the Additional Card. Check off each row as you complete installing those components.

	QTY		
[ ]	1	R49	**Resistor, 10K, SMT, 1206**
[ ]	1	R51	**Resistor, 100K, SMT, 1206, 1%**
[ ]	1	R50	**Resistor, 220K, SMT, 1206, 1%**
[ ]	1	R52	**Resistor, 221K, SMT, 1206, 1%**
[ ]	1	C42	**Capacitor, 0.01 uF, SMT, 1206**
[ ]	1	C41	**Capacitor, 0.1 uF, SMT, 1206**
[ ]	2	D12, D13	**Diode, Schottky, 1N5711, SMT** ***(Faint cathode marking)***



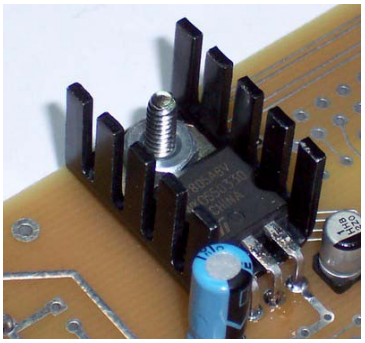
PCB with all SMT resistors mounted (shown on left), and then with all SMT capacitors and SMT diodes added.

5) **Install components from Semiconductor Bag**

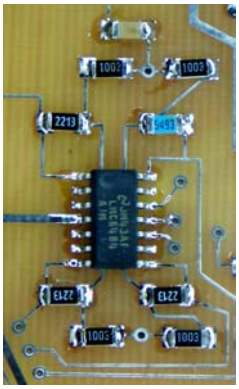
Using the Component Layout Diagram in Appendix A as a guide, install the components from the Semiconductor Bag. Check off each row as you complete installing those components.

	QTY		
[ ]	**1**	**U1**	**Voltage regulator, 3-terminal, 7805**
[ ]	**1**		**Heatsink - TO220** ***(From Controls Bag)***

Position the voltage regulator over the mounting hole to determine where to bend the three leads so they can be inserted to the pc board as shown below. Before soldering in place, orient the heatsink as shown and use the metal screw & nut to connect the two components to the pc board. Then, when aligned nice and straight, solder the three leads of U1 to the pads.

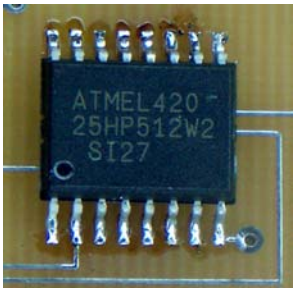


[ ]	**1**	**U2**	**Memory, SEEPROM, 512Mb** ***(16-pin surface mount IC.)***
[ ]	**2**	**U3, U4**	**IC, Op Amp, LMC6484, SOIC** ***(14-pin surface mount IC packages.)***
[ ]	**1**	**U5**	**IC, Audio Amp, LM386, SOIC** ***(8-pin surface mount IC)***
[ ]	**1**	**U6**	**IC, Level Translator, TC7SET08F, SOIC** ***(Pretty small 5-lead surface mount IC.)***
[ ]	**3**	**D7, D8, D9**	**Diode, Schottky, 1N5817, DO-41** ***(Faint cathode marking)***
[ ]	**1**	**Q1**	**Transistor, NPN, 2N3904, TO92** ***(Familiar 3-lead thru-hole package.)***

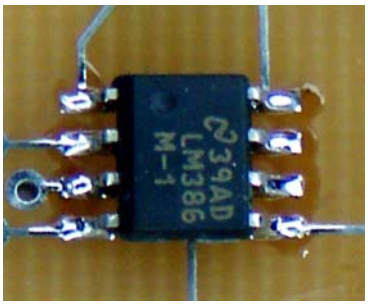


U4 op amp

U6 translator



U2 SEEPROM Memory



U5 LM386 Audio Amp

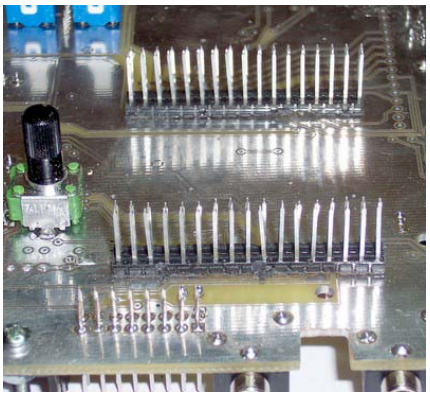
Section 5: Installing Parts from the Controls and Connectors Bag

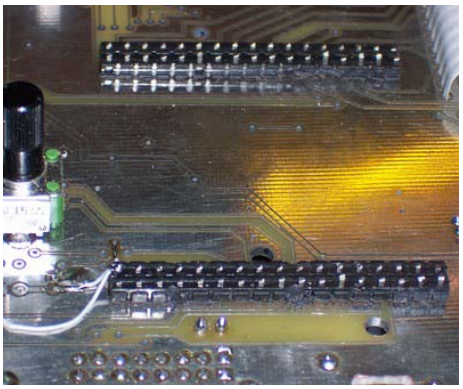
1) Install Pinheaders & Sockets

Using the Component Layout Diagram in Appendix A as a guide, install all pinheaders and strip sockets on the Component side of the board …

	QTY		
[ ]	2	P1, P2	**Pinheader, 2x34 (HC908)**

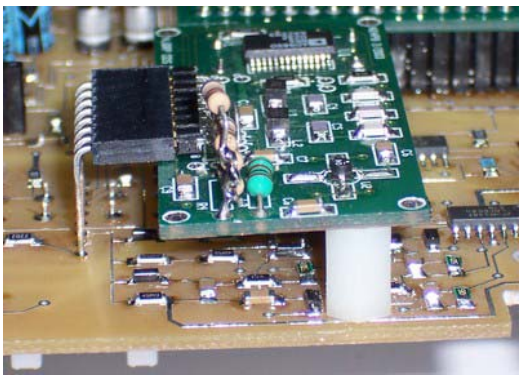
It's really important to insert the **longer-side pins** of P1 & P2 into their respective holes **from the Bottom/Controls side of the pc board.** Then, since the black plastic part of the connector body is on the bottom, you'll need to carefully solder the pins on the Top/Component side. When you solder the pins, take care to not let the solder wick up the pins, as the HC908 Daughtercard sockets will need to fit down onto these pinheaders. (Just heat the pad and very lowest part of each pin, then quickly apply a *small* amount of solder.) When all the pins have been solders, snip off the pins on the Bottom/Controls side of the pc board. See photos below for reference.





[ ]	2	P3, P4	**Pinheader, 1x2 pos'n (MON & NiMH)**
[ ]	1	J8	**Socket, 1x12 pos'n (DSP out)**
[ ]	1	J9	**Socket, 2x10 position, (DSP in)**
[ ]	1	J10	**Socket, 1x8 pos'n, right angle (for DDS Daughtercard)**

This is the right-angle connector for the DDS Daughtercard and it will be important to get it soldered in at the correct height above the pc board. First install two nylon spacers to the Top/Component side of the pc board located at the top two corners of the silscreen indication of the DDS card. (These two holes are located near components R30 and R15.) Use nylon screws to hold them in place. These will be the resting points for the DDS card once it is installed. Next, stick J10 on the end connector of your DDS Daughtercard with the pins of J10 extending down toward the bottom/ground side of the DDS card. Insert the pins of J10 into the pc board from the Top/Component side such that the attached DDS card is resting on the just-installed nylon spacers. Solder the J10 pins from the bottom side while holding the DDS card steady and parallel to the pc board, and snip off the pins. Proper orientation will be as shown below

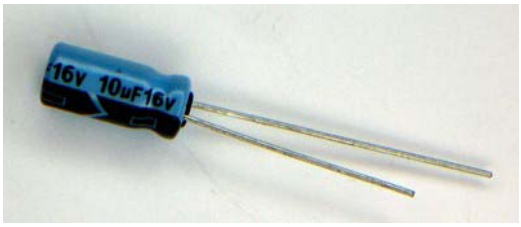


2) Install Thru-hole Components

Using the Component Layout Diagram in Appendix A as a guide, install all thru-hole components on the Component side of the board …

	QTY		
[ ]	**2**	**C5, C26**	**Capacitor, 10 uF, Electrolytic**

When installing these radial-lead, thru-hole electrolytics, **be sure to properly identify the component polarity**. As shown in the photo below, the longer lead is the positive and the shorter is the negative (which is also identified with the black stripe on the side of the component.) Be sure to insert the positive/longer lead in the pc board hole closest to the silkscreened '+' sign.



[ ]	**1**	**C27**	**Capacitor, 47 uF, Electrolytic**
[ ]	**1**	**C24**	**Capacitor, 100 uF, Electrolytic**
[ ]	**1**	**R43**	**Resistor, 10, 1/2W**
[ ]	**1**	**R47**	**Resistor, 12, 1/2W**

3) Install Connectors

Using the Component Layout Diagram in Appendix A as a guide, install most connectors on the Component side of the board. **It is very important to mount these components on the Top/Component side of the pc board.** Most of these connectors and their pads are symmetrical, so you could mistakenly mount them on the wrong side. Double-check the Completed PC Board Assembly photos in Appendix E to ensure that you are inserting these connectors to the proper side of the pc board.

	QTY		
[ ]	**1**	**J1**	**BNC, pcb mount**
[ ]	**1**	**J2**	**Serial port connector, DB9F**
[ ]	**1**	**J4**	**Coaxial power connector, 2.1mm**
[ ]	**1**	**J11**	**Mini-DIN, 6 pos'n (KBD)**
[ ]	**3**	**J6, J7, J12**	**Audio jack, 1/8", pcb mount**
[ ]	**1**	**J14**	**Mini-Din, 8 pos'n (AUX)**
[ ]	**1**	**S2**	**Slide switch, pcb mount, SPST**

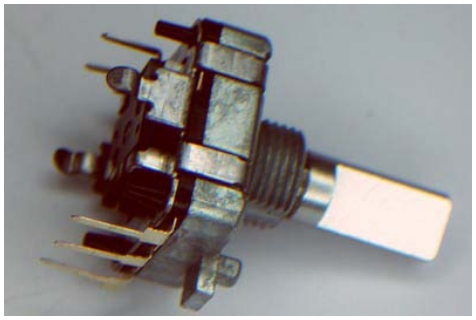
4) Install Controls

Using the Component Layout Diagram in Appendix A as a guide, install all controls on the Bottom/Controls/Ground side of the pc board. It is really important to install these components on the Bottom/Controls side of the pc board. Double-check the Completed PC Board Assembly photos in Appendix E to ensure that you are inserting these connectors to the proper side of the pc board.

QTY

[ ] 1 ENC Rotary Encoder

The encoder needs a little preparation before you assemble it to the pc board. Using some side cutters/diagonals (and preferably heavy-duty ones), you must first snip off the zinc cast alignment nub that sticks up in the same direction as the shaft. Taking off this small piece will allow the encoder to sit flush up underneath the front panel of the enclosure. Also, the encoder shaft is too long for our purposes here, so you'll need to use a hack saw (or equiv) to saw off about ½" of the flat-sided end of the shaft.



[ ] 1 R45 Potentiometer, 10K, miniature, pcb mount

[ ] 5 PB1, PB2, PB3, PB4, PB5 Pushbuttons, SPST, momentary contact

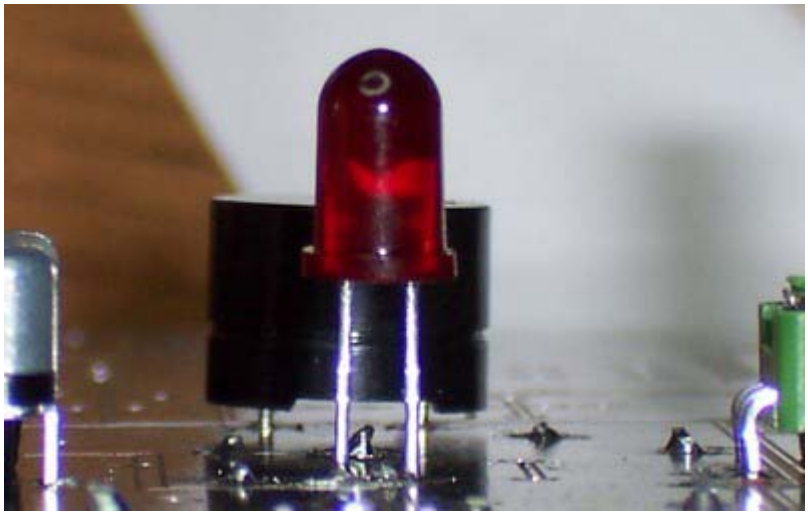
When installing these pushbuttons, it's very important to orient them as shown in the photo below. Each pushbutton must have the while tabs on the side of its body oriented to the right/left of the board (i.e., toward the speaker and J9). Otherwise the switch input lines to the MPU will always be shorted.

[ ] 1 SPKR Speaker, miniature, 32-ohm

When soldering this component in place, be careful to orient the leads so the pin marked with a '+' is placed in the hole clsest to the silkscreened '+'. Be sure not to apply too much heat while soldering, as the plasic of fthe speaker body can easily melt and deform. Lastly, leave the speaker a little elevated (about .04") from the pc board, as this will allow it to better reach up to the front panel hole. See photo below for reference.

[ ] 1 LED1 LED, T1-3/4 (BUSY) ***(From Semiconductor Bag)***

The the cathode of the LED is indicated by the shorted lead and a slight notch in the side of the red plastic body. The catchode must be mounted toward the upper end of the board, toward the straight line indication on the silkscreen. In order for the LED to reach up through the front panel, it should be mounted about 0.3" up off the pc board. See the photo below for reference.



4) Install Test Points

We provided for some important signals to be readily accessible to your DVM or oscilloscope probes during the instruction checkout. These "Test Points" are merely small "loops" of wire made from of scrap component leads that are soldered onto adjacent pads on a trace of a specific signal.

[ ] Install Test point 'GND' located in lower left corner of the board.

[ ] Install test point '+V' located in lower-middle area of the board.

[ ] Install test point 'RF' located in the top-right corner of the board.

[ ] Install test point '+5' located to the right of regulator U1.

[ ] Install test point 'Aud-R' located in the middle of the board near C33.

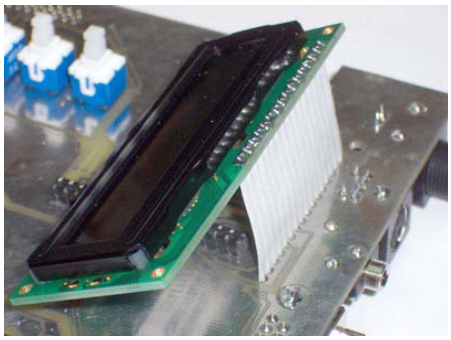
[ ] Install test point 'Aud-L' located in the middle of the board near C34.

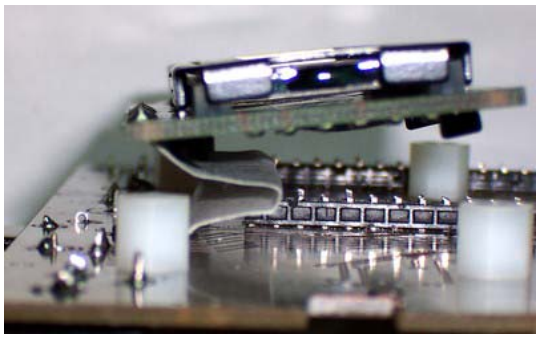
5) Install LCD

Using the Component Layout Diagram in Appendix A as a guide, install the LCD on the Bottom/Controls/Ground side of the pc board …

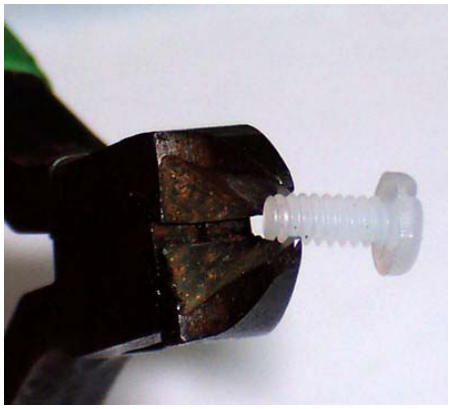
	QTY		
[ ]	**1**	**LCD**	**Display, LCD, Hantronix, 16x2 STN, GRAY**
[ ]	**1**	**W1**	**Jumper, Flexstrip, (LCD)**

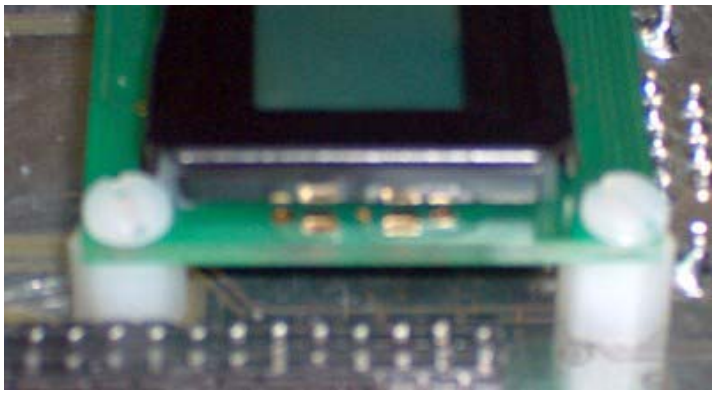
You'll use the 16-wire flexible jumper W1 to connect the LCD to the pcb, as shown in the photos below. Bend the wire jumper at the midpoint and fold it under the LCD as shown below in preparation for attaching the LCD to the nylon standoffs on the pc board.





Locate eight nylon screws and using your wire cutters/diagonals, nip 1/16" off the end of each. This amounts to about 1.5-2 "threads" as you position the cutters on the shaft of the screw, as shown below. We have to do this because the spacers we'll use in the next step are short and the screws will not insert far enough to seat the LCD on one side and the pc board on the other.





Locate the four shorter, 0.25" nylon spacers. Be sure these are the shorter ones supplied in the kit. Attach all four spacers on the Bottom/Controls side of the pc board using four of the shortened nylon screws, as shown in the upper-right photos. Be careful not to over-tighten the nylon hardware, as you'll strip out the threads.

Next, you will screw the four remaining shortened nylon screws through the corner holes of the LCD and into the threaded spacers mounted on the board. NOTE: The four holes in the corners of the LCD are smaller than the screws, but by applying a little pressure while turning them into the holes, you will "thread" them into the holes and subsequently into the spacer below. (If you have trouble with this, you can use an appropriately-sized drill bit to carefully enlarge the LCD mounting holes.) Again, be careful not to over-tighten the nylon hardware. When complete, the LCD should look as shown in the upper-right photo.

Section 6: Power-up and Test

[ ] Prepare for the Tests

When first ready to apply power to the newly-assembled Micro908, position the pc board on the bench (without the enclosure) and with the LCD and controls side facing up. This will enable you to more easily see the components and you can spot gross problems (like exploding capacitors, smoking regulators, etc.) as soon as they occur.

Make sure the shunt (small, black 1x2 position jumper socket) is removed from P3.

Install the HC908 Daughtercard. Make certain the Heartbeat LED is oriented toward the bottom of the Micro908 pc board, as shown in Appendix E: Fully-Assembled PC Board

Do not install the DDS Daughtercard yet.

Do not install the DSP Daughtercard yet.

It's a good practice to use a current-limited external power supply when first testing out projects like the Micro908. You can set the maximum current to be about 500ma and be assured that a short circuit will not blast away circuit traces in the process of first applying power. A good alternative is to power the project from a battery on the bench. The similar effect can be gained by the natural limits of a battery to supply lots of current.

[ ] Apply power and see display on LCD

After plugging in the power supply and moving the Power slide switch to the ON position (up), you will see the approximate power supply voltage displayed in the LCD for about ½ second, then the LCD will display the "main menu" message of "AA908 v2.0" on the first line, and "Select Command" on the second line.. If you don't see this indication on the LCD, proceed to the Troubleshooting section.

[ ] Press MODE Pushbutton and see Frequency, SWR, R and X display on LCD

Pressing MODE pushbutton (also called the Manual Mode) will the displayed values for frequency, SWR, R and X displayed on the LCD. (You might need to turn the Dial a bit to see the full display appear.) The readings for SWR, R and X will be meaningless at this point since you do not yet have the DDS Daughtercard (i.e., the RF signal source) installed.

[ ] Turn Dial to Change Frequency

Turning the Dial will result in changing the displayed frequency, starting at the digit that has the underline cursor beneath it. This digit will be the 10 kHz digit.

[ ] Press and Turn Dial to Change Cursor Position

When you press-and-turn the Dial, the underscore cursor will move to the next-higher (or next-lower) digit in the frequency display, thus allowing you to change frequency at a more (or less) rapid rate. The cursor will stop at the rightmost digit despite further clockwise rotation of the Dial, thereby providing 10 Hz increments as the smallest allowable. Similarly, the cursor will stop at the leftmost position despite further counterclockwise rotation of the Dial, thereby providing 10 MHz as the greatest incrementing value of frequency.

[ ] Press the BAND Pushbutton to Select Band

You can press the BAND pushbutton at any time to select one of four bands segments to automatically scan when you later press the SCAN pushbutton. The band segments are 1-10 MHz, 10-20 MHz, 20-30 MHz and "Custom Band". (The Custom Band uses start/end limits that were previously set in the Configuration menus, discussed later.) The different Band segments are sequentially displayed while turning the Dial. A band is selected by pressing the Dial when the desired band is displayed. Once a band is selected, the display will show the Main display ("AA908 v2.0" / "Select Function".)

[ ] Press the SCAN Pushbutton to Scan the Band

When you press the SCAN pushbutton, you initiate an automatic scan of the selected band. The LED is illuminated and the internal DDS signal generator is automatically set to the Start frequency for that band and the SWR is determined. The frequency is incremented by the Step value (default of 10 kHz and user-settable in Config) and the SWR is determined at this point. This sequence continues throughout the entire band until the End frequency is reached. Throughout the scanning, the software saves the frequency that yielded the lowest SWR and when the scan is complete, control is given to the Mode function that displays that point of lowest SWR. The display will show the frequency, SWR, R and X values for that point and the user is able to manually control the instrument as described in the Mode section above. Since the DDS signal generator is not yet installed, the Scan will likely exit showing a frequency of 1,000.00 MHz and an SWR of '>10'.

This is okay for now. If the LED did not illuminate at the start of the Scan, and turn off at the end of the Scan, make a note to visit the Troubleshooting section later on.

[ ] Press the CONFIG Pushbutton to get into the Configuration Menus

The CONFIG pushbutton may be pressed at any time to access the Configuration menus. Please refer to the Micro908 Technical Reference and Operation Guide for a complete description of the available functions in this mode. But in summary, the user is allowed to specify Start/End frequencies and Step size for a Custom Band, turn the Tones On/Off, Update the Software, turn on DEBUG Mode, access the low-level Debug Monitor (HCmon), Display the battery voltage, Calibrate the instrument, and Exit back to the Main display.

[ ] Select TONE ON to hear Audio Indication of SWR

While in the Configuration menus, turn the Dial until 'Tone ON' is displayed and then press the Dial to select it. Rotate the Dial until 'Exit' is displayed and press the Dial to exit. A message on the second line of the LCD will indicate that the changed settings are being saved (to nonvolatile EEPROM memory) and then the Main display will show once again. Press the MODE pushbutton to see the frequency display in Manual Mode and immediately hear a high-pitched tone coming from the speaker, indicating that a high SWR reading is currently being displayed. If you do not hear a tone, make a not of later checking it out in the Troubleshooting section. When we later install the DDS Daughtercard and an antenna (or dummy antenna), you will note a variable pitch being generated when the Dial is tuned through a resonance. A lower tone indicates a lower SWR being measured. In order to preserve your sanity (and hearing) during the remainder of the tests, go back into Configuration menu to turn the Tone OFF.

[ ] Select 'Debug Monitor' from Configuration Menus

You will test the RS232 serial port on the Micro908 now, so connect a 9-pin, "straight-through" serial cable (not a "null modem cable") between your Micro908 pc board (connector J2) and a PC running a dumb terminal program such as HyperTerm. Configure the terminal program to run with settings of "9600 N81" (9600 baud, No parity, 8 bits, 1 stop bit). In the Micro908 Configuration menu, select 'Debug Monitor" and see a line of data displayed on the PC screen that shows "HCmon>" and intelligible letters and numbers following it. If you do not see this indication on the PC screen, make a note to check it out in the Troubleshooting section.

[ ] Turn the Micro908 power switch S1 to OFF

[ ] Install the upgraded DDS Daughtercard

After making the improvements to the DDS Daughtercard (per the simple instructions provided in Appendix E), install the DDS card into its connector J10.

[ ] Turn on the power to the Micro908

After siding S1 back to ON, check that there is no smoke, sizzle or unexplained flashes of light coming from your DDS Daughtercard. The DDS chip, the MAV amplifier and the R4 bias resistor will indeed get warm to the touch, but not excessively so. If they do, power down immediately and go to the Troubleshooting section.

[ ] Attach Dummy Antenna to BNC connector J1

We're going to measure some antenna characteristics here so you'll need a load. Ideally at this point, you should connect a "dummy antenna" that you might have previously prepared, per Appendix F. Alternatively, you can connect an antenna that is known-resonant at some frequency within the HF bands.

[ ] Get into Manual Mode (MODE Pushbutton) and See Measurements Change

When you get into the Manual mode (by pressing MODE pushbutton), move the frequency up/down until you see a dip occur in the SWR display, indicating that you are approaching the resonant point of the dummy antenna. When you have found the minimum SWR "dip", move the cursor to the next lower position (by press-and-turning the Dial to move the underscore cursor) and continue turning the Dial to get finer resolution of the minimum SWR point. If you do not see this SWR dip condition happening, go to the Troubleshooting section to dig into the problem.

[ ] Select the Band that has the Resonance and SCAN

Make sure you have selected the Band that contains the known-resonant frequency (1-10, 10-20 or 20-30 MHz) and press SCAN. The instrument will scan the band, and at the end it will display the antenna characteristics (SWR, R and X) found at the point of minimum detected SWR.

[ ] Calibrate the Micro908 reflectometer channels for gain

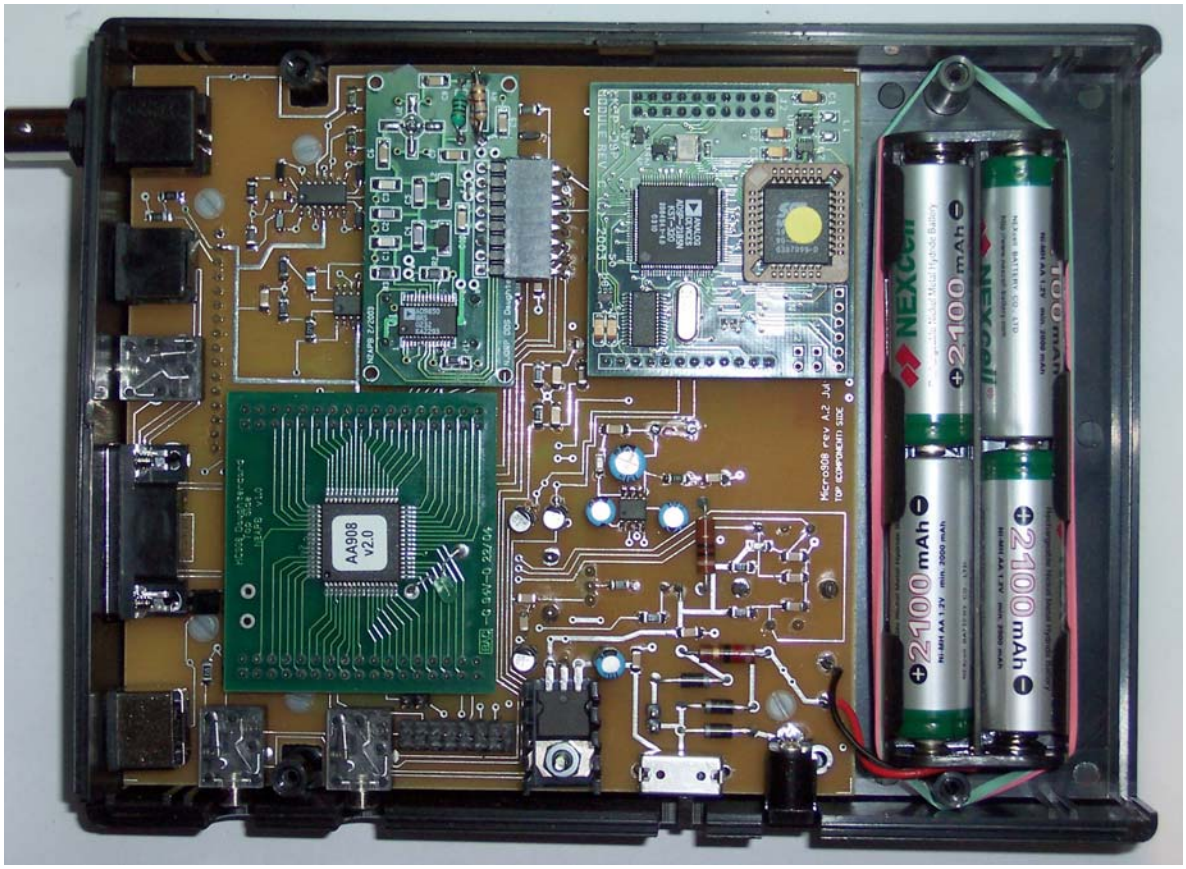
The "calibrate" function is only available starting with version 3.0, so if you don't yet have this latest software, please download and install it per the instructions given in Appendix G. Once this is done, you can select the Calibrate function from the Config menus and just follo the directions given inn the LCD. The calibration steps include:

1) Ensuring that no channel exceeds the range of the A/D converter for an open circuit (i.e., no load on J1). You will be instructed to adjust the trim pot on the DDS card such that none of the open circuit voltages displayed on the LCD are approaching FF.
2) You will place a 50-ohm load on J1 and press the Dial to continue.
3) You will place a 100-ohm load on J1 and press the Dial to continue.
4) You will place a 270-ohm load on J1 and press the Dial to continue.

Section 7: Installing the PCB in the Enclosure

Now that the functional tests are complete, you can finish the final assembly of the Micro908 by installing it into the enclosure. (NOTE: The enclosure is an optional item and may not have been purchased. If this is the case, you are done!) Refer to the Mechanical Assembly diagram in Appendix E for a detailed "exploded view" described in this section.

When this section is complete, your Micro908 will look something like the photo below.



Fully-Assembled Micro908 PCB mounted in the Top Shell of the Enclosure

[ ] Apply Overlay Label to Drilled End Panel

Locate the black overlay with white labels and carefully apply it to the drilled end panel. Carefully peel back the protective paper from the sticky side off the overlay and lay it down on the side of the end panel that is "raised" around the edges – that is, the fully-flat side of the end panel will end up being on the inside of the Micro908, so you want to apply the overlay label to the outside surface. Be careful not to misalign the label, as it is very difficult to remove.

Pre-Drilled End Panel with Overlay Label attached

[ ] Install spacers on inside of top shell

Locate four of the longer nylon spacers and the four flathead nylon screws. (Flathead screws have an angled head that allows them to be countersunk in the plastic to lie flat with the surface of the enclosure and thus provide a smooth overall finish.) Connect the four nylon spacers to the inside of the top shell using the flathead nylon screws at the countersunk positions in the enclosure shell.

[ ] Apply front Panel Overlay Label to the Top Shell

Once again, peel back the protective paper from the large yellow front panel overlay label and carefully align it over the holes and to the edges of the enclosure shell. Once in place, the overlay label will cover the nylon spacers just installed, so ensure they are snug before covering over the flathead screws.

[ ] Apply Side Panel Overlay Label to the Side of the Top Shell

Locate and peel back the protective paper from the long, black (with white letters 'ON', 'OFF', etc.) side panel overlay label and carefully align it over the holes and to the edge of the side of the enclosure shell.

[ ] Place Drilled End Panel on Connector End of the PC Board

Place the end panel on the connectors at the edge of the Micro908 pc board, with the label side facing out/away from the board. The end panel will only go on one way and it can remain loosely in place.

[ ] Install the PCB Assembly into the Top Shell

Carefully slide the pc board into the top shell, making sure to guide the loose end panel into the slots at the top end of the enclosure shell while also guiding the edge connectors and front panel controls through the appropriate holes. Although it may seem like a tight fit, it should drop into position real nice once everything is aligned. If you have problems with a fit being too tight, you might not have filed the side post holes in the pc board large enough, or you might not have filed the top and side edges of the pc board enough to allow the fit. Once the PCB assembly is in place, use the four remaining nylon screws to connect the pc board to the nylon spacers previously installed on the inside of the enclosure top shell. The holes for these screws are located near J6, J4, U3 and D11.

[ ] Install the Battery Holder

If not already done, solder the wires from the battery holder to the pc board at the points marked 'BATT'. The red wire goes to the '+' pad and the black wire goes to the '-' pad. The battery holder (and heavy batteries) can effectively be held in place with some rubber bands holding the assembly to the two molded enclosure posts, as shown in the photo at the top of this section. Additionally, a strip of the small bubble wrap supplied in the Kit may be cut off and affixed to the inside of the enclosure above/below the battery holder to further act as a cushion for the battery assembly.

[ ] Install the other End Panel

Slide the uncut plastic end panel into position at the bottom end of the enclosure shell. Be sure that it is contained in the small slot at the end of the shell.

[ ] Install the Bottom Shell of the Enclosure

Place the bottom black plastic shell of the enclosure in place and use the four ¾" self-tapping screws to secure it to the top shell.

[ ] Install Rubber Feet

Peel off the four rubber feet and apply them to the corners of the enclosure's bottom shell.

[ ] It's "Miller Time"!

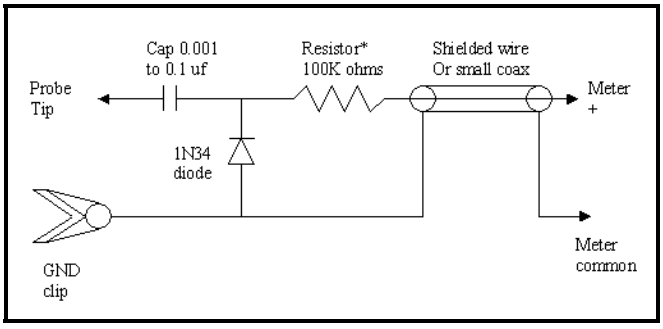
Assembly of the Micro908 is complete.

Section 8: Troubleshooting

In this section we'll help you get your Micro908 instrument working if it doesn't behave as described when power is first applied. You'll need some basic equipment, the schematic from Section B and the pc board layouts from Section C.

Equipment Needed

If you have an oscilloscope that's good to 100 MHz, you'll be all set to probe at all the signals inside the Micro908. But at a minimum, you'll need a volt-ohmmeter (DVM) for DC voltage and resistance measurements. You'll also need an RF probe in order to see the RF voltages that we're dealing with in this instrument. If you don't have an RF probe, like the AccuProbe from the NorTex QRP Club, it's pretty easy to make a simple one. The one shown below can be made up in less than an hour from common junk box parts. If you want to get fancy you could install the components on a narrow piece of perf board and slide it into an old metal cigar tube for shielding. Otherwise just have the probe made from stiff wire affixed to the end of that narrow perf board and an alligator clip for ground. Then with your DVM probe connected to the back end, you can hold the perf board with the "probe tip" at the front end and use it to probe around the circuit.



Simple RF Probe (by Phil DeCaire, WB7AEI, described in QRP Homebrewer #9)

Basic Tests

We'll assume you haven't seen any smoke coming from the circuit board or components. That's always a good starting point for a successful repair of a unit. Otherwise, you'll obviously need to look in the area when the smoke came from, or where the board and components look charred – for that is the place where a component was mis-installed or the place where the problem surfaced from something else (e.g., reversed power supply, etc.)

Power Supply

If you are operating from batteries, be sure the terminal voltage on the battery holder is at least 8.5V. If the level is below that, the RF signal will not be generated properly, or at all, and you will not get expected readings displayed on the LCD.

If operating from an external source, make sure the voltage level does not exceed 16V DC. Beyond this maximum specification, the regulators will be required to dissipate too much power and they'll get too hot for normal operation. Further, the "battery monitor" voltage divider R28 and R41 will present more than 5V to the A/D on the MPU , which would not be too nice.

Make sure you have your battery polarity proper with the wires going to the circuit board, and be sure the connector supplying external power is wired with positive-to-center on the mating plug to J4. If either connection is reversed, your board will not be damaged, but it won't receive the necessary voltage to make it work. In that case … fix it!

With the power switch ON, and the ground probe of your DVM attached to the board ground at the GND test point (located at the bottom-left of the pc board), measure your supply voltage at the +V test point at the +V test point (located at the bottom-center of the pc board). If you don't see the proper voltage, your problem is in the diode arrangement around the battery or power connector and power switch S2.

Measure +5V at the test point located just to the right of regulator U1. If you don't see +5V here, there is likely something pulling down the 5V bus on the board. In this case, U1 is likely getting pretty hot. Power down the unit and find the cause of the problem, which is likely a solder short on any of the closely-spaced terminals of the components or connectors. Also, make sure that the HC908, DDS and DSP daughtercards are oriented properly within their sockets. If they are plugged in backwards (180-degrees out), or shifted up/down one position, chances are this is causing the power supply problems. It's also likely that the daughtercard is damaged (or at least suspect), unfortunately.

Specific Problems

Problem: "The frequency doesn't increment or decrement properly when I turn the Dial"

The fact that the Dial has some effect at all is a good thing and indicates that your board is very likely working. We needed to change the pc board artwork after the HC908 Daughtercards were programmed with "version 2" and you will need to download the latest software (version 3.0) from the project website and load it into your Micro908 using either of the techniques described in Appendix G: "Loading New Software into the Micro908." There will be many occasions of loading new software in the future, so might as well get into it right now! We're sorry that were not able to

Problem: The Serial port isn't sending data to my PC during the 'Debug Monitor' tests.

The RS232 serial port signals of the HC908 Daughtercard were tested at assembly time, so the only possible causes of this condition are the serial cable (needs to be a "straight-thru" cable, not a null modem), or the setup of your terminal program (HyperTerm or whatever). Please check the appropriate section describing these settings to ensure that you have configured the program properly on your PC and that you have the serial port free of any other program trying to control it.

Problem: The LED is not illuminated during a Scan.

You should see a low voltage level at the top of R8 during a Scan and a high one (~5V) when the Scan is complete, corresponding to the LED turning on and off, respectively. If the signal levels are okay, but still no LED illumination, you may have the LED oriented backwards. See the appropriate assembly section to review details for proper orientation.

Problem: No tone is heard when reading SWR with the Tone enabled from the Config menu.

You should see an approximate 3V audio frequency signal on the left side of C40, and be able to trace that through R45, U5, C24 and then on to the the speaker itself. If the signal is there, but still there is no sound, you might have damaged the speaker during installation.

Problem: The readings for SWR, R and X are not changing, or are way off, when I move the Dial or do a Scan.

There may be several causes for this condition.

1) Ensure that you have downloaded the latest version of software (Version 3.0) from the Micro908 web pages and have performed gone through the calibration steps (located under the CONFIG pushbutton).

2) You may not have proper signal levels coming from the DDS Daughtercard. With no load connected to the RF Out jack (J1), dial up a frequency of 1,000.00 and probe the output of the DDS card (J10 pin 6). You should see about 4 Vp-p, or about 1.4 Vrms using an RF Probe. If you do not see these approximate signals, nothing "downstream" will work right and you will need to find the cause of the problem before proceeding.

3) You may have a component problem (wrong part, solder short, etc.) in the reflectometer or buffer amplifiers. Use the following charts to determine if you have the proper levels at the specified points in the circuitry under conditions on J1 of: open circuit, short circuit, and 50-ohm loads. Try to narrow the problem down to a specific channel (Vf, Vr, Vz and Va), and then to a specific point in that signal chain.

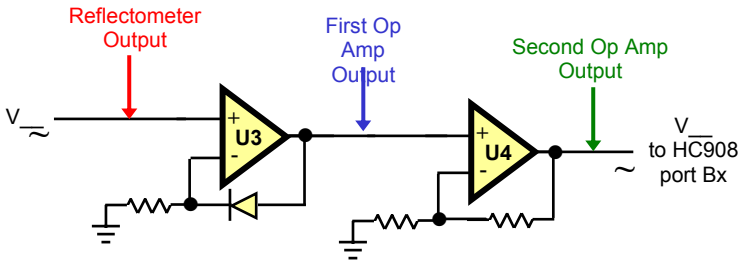
Voltage Charts

Typical LCD display of Reflectometer readings in Calibration for the three calibration conditions of **Open** circuit load, **Short** circuit load, and **50-Ohm** circuit:

	Vf	Vr	Vz	Va
Open	FB	FB	FB	01
Short	8C	8F	02	8D
50-ohm	C5	04	67	60

Typical voltages for the four Reflectometer op amp channels in the three calibration conditions of **Open** circuit load, **Short** circuit load, and **50-Ohm** circuit:

	VA	VZ	VR	VF
OPEN CIRCUIT				
Reflectometer output	U3 pin 3 = **0**	U3 pin 10 = **1.9**	U3 pin 12 = **0.8**	U3 pin 5 = **1.8**
First op amp output	U3 pin 1 = **0**	U3 pin 8 = **2.1**	U3 pin 14 = **0.9**	U3 pin 7 = **1.9**
Second op amp output	U4 pin 1 = **0**	U4 pin 8 = **4.8**	U4 pin 14 = **4.8**	U4 pin 7 = **4.8**
SHORT CIRCUIT				
Reflectometer output	U3 pin 3 = **0.7**	U3 pin 10 = **0**	U3 pin 12 = **0.8**	U3 pin 5 = **0.7**
First op amp output	U3 pin 1 = **0.8**	U3 pin 8 = **0**	U3 pin 14 = **0.3**	U3 pin 7 = **0.8**
Second op amp output	U4 pin 1 = **2.7**	U4 pin 8 = **0**	U4 pin 14 = **2.7**	U4 pin 7 = **2.7**
50-OHM CIRCUIT				
Reflectometer output	U3 pin 3 = **0.5**	U3 pin 10 = **0.5**	U3 pin 12 = **0**	U3 pin 5 = **1.0**
First op amp output	U3 pin 1 = **0.6**	U3 pin 8 = **0.6**	U3 pin 14 = **0**	U3 pin 7 = **1.2**
Second op amp output	U4 pin 1 = **1.9**	U4 pin 8 = **1.9**	U4 pin 14 = **0**	U4 pin 7 = **3.8**



APPENDIX A: Micro908 Parts List

SMT Cards Bag		
Resistor Card 1		
1	R46	Resistor, 10, SMT, 1206
3	R11, R12, R14	Resistor, 49.9, SMT, 1206, 1%
1	R7	Resistor, 330, SMT, 1206
1	R8	Resistor, 1K, SMT, 1206
1	R6	Resistor, 4.3K, SMT, 1206
1	R44	Resistor, 4.7K, SMT, 1206
4	R37, R38, R39, R40	Resistor, 5.1K, SMT, 1206
12	R1, R2, R3, R4, R5, R17, R18, R22, R23, R27, R42, R48	Resistor, 10K, SMT, 1206
Resistor Card 2		
1	R41	Resistor, 10.0K, SMT, 1206, 1%
1	R28	Resistor, 22K, SMT, 1206, 1%
4	R10, R13, R15, R49	Resistor, 47K, SMT, 1206
3	R19, R24, R29	Resistor, 100K, SMT, 1206, 1%
3	R16, R21, R26	Resistor, 220K, SMT, 1206, 1%
2	R20, R30	Resistor, 221K, SMT, 1206, 1%
1	R25	Resistor, 549K, SMT, 1206, 1%
4	R32, R33, R34, R54	Resistor, 1M, SMT, 1206
Capacitor & Diode Card		
2	C31, C32	Capacitor, 560 pF, SMT, 1206
6	C18, C20, C22, C23, C28, C40	Capacitor, 0.01 uF, SMT, 1206
18	C1, C2, C3, C4, C6, C11, C12, C13, C14, C15, C17, C19, C21, C35, C36, C37, C38, C39	Capacitor, 0.1 uF, SMT, 1206
2	C29, C30	Capacitor, 0.33 uF, SMT, 1206
8	D1, D2, D3, D4, D5, D6, D10, D11	Diode, Schottky, 1N5711, SMT
Additions Card		
1	R49	Resistor, 10K, SMT, 1206
1	R51	Resistor, 100K, SMT, 1206, 1%
1	R50	Resistor, 220K, SMT, 1206, 1%
1	R52	Resistor, 221K, SMT, 1206, 1%
1	C42	Capacitor, 0.01 uF, SMT, 1206
1	C41	Capacitor, 0.1 uF, SMT, 1206
2	D12, D13	Diode, Schottky, 1N5711, SMT

Controls & Connectors Bag		
1	J1	BNC, pcb mount
1	J2	Serial port connector, DB9F
1	ENC	Rotary Encoder
1	J4	Coaxial pow er connector, 2.1mm
1	J8	Socket, 1x12 pos'n (DSP out)
1	J9	Socket, 2x10 position, (DSP in)
1	J11	Mini-DIN, 6 pos'n (KBD)
3	J6, J7, J12	Audio jack, 1/8", pcb mount
1	J10	Socket, 1x8 pos'n, right angle
1	J14	Mini-Din, 8 pos'n (AUX)
1	W1	Jumper, Flexstrip, (LCD)
2	P1, P2	Pinheader, 2x34 (HC908) (cut 2 from 2x72 strip)
2	P3, P4	Pinheader, 1x2 pos'n (MON) (cut from 2x72 strip)
3	C7, C33, C34	Capacitor, 1 uF, Electrolytic, SMT
2	C5, C26	Capacitor, 10 uF, Electrolytic
1	C27	Capacitor, 47 uF, Electrolytic
1	C24	Capacitor, 100 uF, Electrolytic
1	R45	Potentiometer, 10K, miniature, pcb mount
1	S2	Slide sw itch, pcb mount, SPST
5	PB1, PB2, PB3, PB4, PB5	Pushbuttons, SPST, momentary contact
1	SPKR	Speaker, miniature, 32-ohm
5		Pushbutton caps (black)
1		Heatsink - TO220
1		Knob, 1/4"-dia shaft, (ENC)
1		Nut, zinc plated, 4-40 (U1)
1		Machine screw , zinc plated, round head, 4-40 x 0.5" (U1)
4		Spacer, nylon, hex tapped, 4-40x0.25" (LCD)
6		Spacer, nylon, hex tapped, 4-40x0.375" (PCB+DDS)
16		Machine screw , pan slotted, #4-40x0.25" (PCB+DDS+LCD)
4		Machine screw , flat slotted, #4-40x0.25" (PCB-cover)
2	Shunt	shunt, 0.1", 2 pos'n
1		Desoldering Braid
1		Solder, 63/37 Eutectic, No-Clean Flux, @28 ga.

Battery Holder Bag		
1	LCD	Display, LCD, Hantronix, 16x2 STN, GRAY
1	J3	Battery holder, 8-AA cells
1		Flux Pen, No-Clean

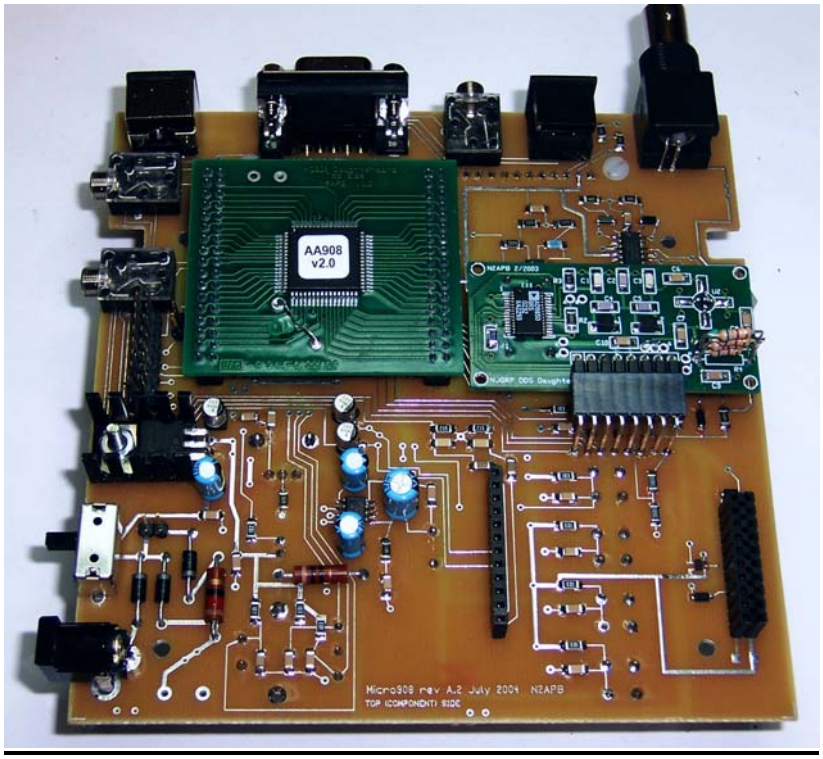
Semiconductor Bag		
1	U1	Voltage regulator, 3-terminal, 7805
1	U2	Memory, SEEPROM, 512Mb
2	U3, U4	IC, Op Amp, LMC6484, SOIC
1	U5	IC, Audio Amp, LM386, SOIC
1	U6	IC, Level Translator, TC7SET08F, SOIC
1	LED1	LED, T1-3/4 (BUSY)
3	D7, D8, D9	Diode, Schottky, 1N5817, DO-41
1	Q1	Transistor, NPN, 2N3904, TO92

PCB Bag		
1	PCB	PC Board
HC908 Daughtercard Bag (Optional)		
1	HC908	HC908 Daughtercard assembly
DSPx Daughtercard Bag (Optional)		
1	DSPx	DSPx for Micro908 assembly
DDS Daughtercard Kit Bag (Optional)		
1	DDS Kit	DDS Daughtercard Kit
Keyboard Cable Bag (Optional)		
1	Cable	Keyborad extension cable (6')
Keyboard (Optional)		
1	Keyboard	Dauphin Mini Keyboard
Enclosure (Optional)		
1	Enclosure	Pac-Tec LH-57 Enclosure (pre-drilled)
Overlay Bag (Included with Enclosure Option)		
1	Front panel overlay	
1	Side panel overlay	
1	End panel overlay	
1	End panel (drilled)	
1	End panel (undrilled)	
4	Enclosure screw s	
4	Enclosure rubber feet	

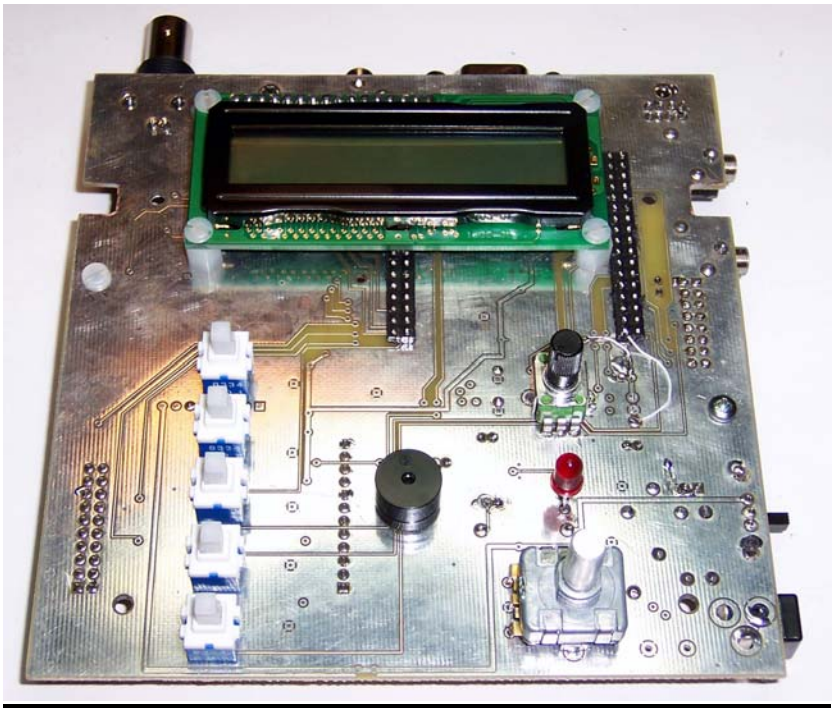
APPENDIX B: Micro908 Schematic

APPENDIX C:
PC Board Component Layout – TOP & BOTTOM

APPENDIX D: Completed PC Board Assembly

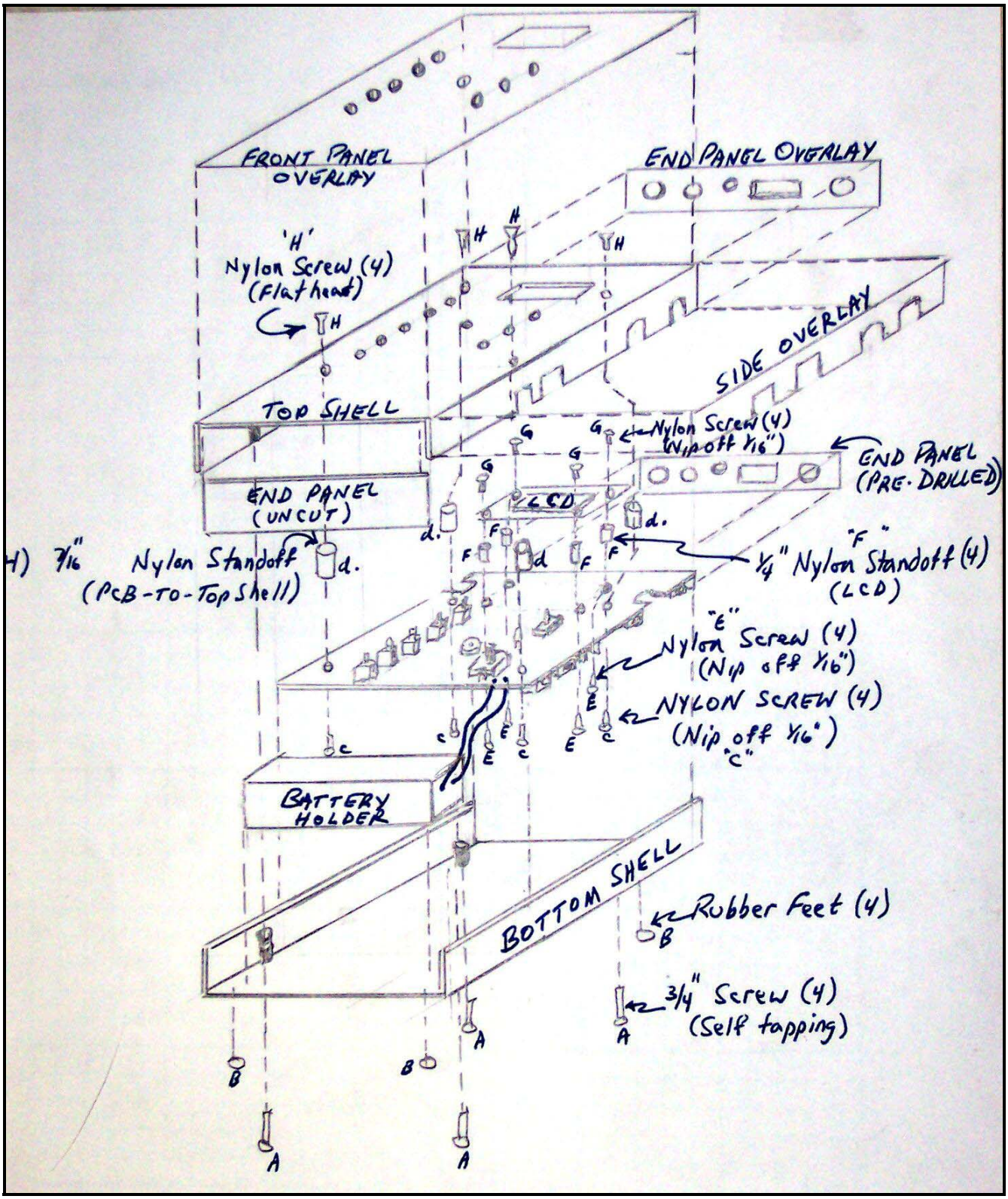


Top/Component Side



Bottom/Controls Side

APPENDIX E: Mechanical Assembly



APPENDIX F: Quick Reference Sheet

APPENDIX G
Loading New Software into the Micro908

This section overviews two ways you can load software updates into your Micro908.

BACKGROUND
Increasingly today, microcontrolled kits and projects have an ability to be "field updated" with new capabilities and software updates made available by the designer. So, instead of needing to send your instrument back for re-programming to get these new features, you can now simply download the program update from the Internet and send it to the target hardware and the device automatically updates its internal memory with the new program. What a great way to keep your project completely up to date with the latest features!

m908 Loader is a PC application that simply and conveniently loads new program files onto the Micro908. It runs on a Windows-class PC connected to the target hardware via a 9600 baud serial port. All you need to do is connect a serial cable between your PC and the Micro908, command the current Micro908 software to do an "auto update" and then run the m908 Loader program on your PC. The new program file you previously downloaded from the Internet web site is then selected in m908 Loader and is sent to the Micro908 over the serial port. Within seconds the transfer of the new program to your project will be complete.

PC REQUIREMENTS
- Windows 95, Windows 98, Windows 2000, and Windows XP. It may also work on other Windows versions.
- Works with computers ranging from 33 MHz Intel 486 processors up to 2.4 GHz P4 processors.
- An available RS-232 serial port. (USB-to-RS232 adaptors may work for those computers having only USB-based serial ports.)

INSTALLATION
1. Copy the **m908installation.zip** file from the Micro908 Resource CDROM to a temporary folder on your PC.
2. Extract (Unzip) the file and a new folder called m908 Loader will be created in that temporary folder.
3. In that m908 Loader folder, run **setup.exe**. Accept the default location for the program files (C:\m908loader) and the default program name.
4. At the conclusion of the installation process, you will have a folder called **m908loader** in the C: drive containing:
    a. **m908 Loader.exe** – The program.
    b. **config.dat** – File used to hold the default and last-selected selections for Serial Port and Delay.
    c. **st6unst.*** – Several files used in uninstalling the program from your computer.
    d. ***.s19** – "S-record" files are present with the "s19" filename extension. S-record files represent the binary images of new programs to be serially loaded into the HC908-based target hardware. These S-record files are presented for selection by the user during the Load process. When a new program, or a program update is made available for use on the HC908 Daughtercard-based hardware, you should place that new S-record file in this folder so it may be selected for load when the m908 Loader program is run. The S-record files are provided in the initial installation include:
        - **aa908_v1-0.s19** -- Micro908 Antenna Analyst (and other .s19 files)
5. When the installation has successfully concluded, you may delete the files used in your temporary folder.

USAGE
1. Connect your Micro908 device to the PC using a standard, straight-through male-female DB9-type serial cable. (Do not use a null modem cable that swaps pins 2 and 3 from end-to-end.)
2. Issue the Auto Update command on the Micro908, located as an option under the Config menu. Confirm the operation by selecting "Yes".
3. Start up the **m908 Loader** program on your PC. (Be certain that no other program has control of the serial port – if your Palm HotSync program is running, or if you have a terminal program like Hyperterminal running, you must exit that program to release the serial port for the **m908 Loader**.)
4. Specify the serial port you have on your computer, using the **Select Port** pull-down menu in m908 Loader. If you select the wrong COM port number, an error will be shown when you initiate the Load operation. In that case, just select another port number until you find the available one on your computer. This setting is saved in the **config.dat** file in the m908loader folder, so you only need specify the port number once.
5. Specify the appropriate end-of-line delay to be used for your computer, using the Delay pull-down menu. (The S-record lines of data are sent "open loop" to the Micro908 and a slight delay is required after each line of data is sent to give the HC908 Daughtercard time to program that line of data. The amount of delay depends on the speed

of your computer. The default setting of 40 milliseconds works well even on computers with 2.4 GHz processors, but you may try longer delays if you have faster computers.)

6. Click the Load button to bring up the list of S-record files present in the m908loader folder. (If you have a newly-downloaded program to send to the Micro908, just place it in this folder beforehand.) Selecting the desired filename (e.g., aa908_v1-0.s19) and pressing OK will initiate the loading sequence.
7. You will see the line counter increment in the main m908 Loader window, indicating that lines of S-record data are being sent to the Micro908 device. When complete, END and then READY will be displayed in the window.
8. Disconnect the serial cable and restart the Micro908 (i.e., turn the power off and on again) to start the newly-loaded program. You should note the new version number presented in the LCD sign-on message!

ALTERNATE (MANUAL) LOADING OF SOFTWARE UPDATES

If you have problems using the **m908 Loader** program, you may follow these simple "manual" steps for getting new software onto your Micro908 instrument.

2) Be certain that your serial port is "clear", meaning that no other program is using it. A typical other use of the serial port is the HotSync manager for connecting to PDAs. This program may be turned off by right-clicking its tray icon and selecting "Exit". This will release the serial port for other programs, like Hyperterm, to us.

3) Attach serial cable to PC between the PC and your Micro908, and bring up HyperTerminal, or the TeraTerm program (supplied on the Micro908 Resource CDROM. If not already set up, configure the terminal program as 9600 8N1, no flow control. Make sure that you have the terminal program configured to be using the active serial port in your PC.

4) Get into the Configuration menus of the Micro908 and select "Debug Monitor". See HCmon sign-on message and prompt "HCmon >".

5) Type "C" to clear out memory

6) Type "L" to load new program. See message ""…waiting …"

7) Pull down the Transfer menu and select "Send text file ..."

8) Navigate to where you saved the *.s19 binary program file (i.e., the binary file with a ".s19" extension you are looking to load into the unit.)

9) Once in the folder that you know contains AA_908v3-0.s19 file, pull down the "Files of Type:" item selection and select "All Files (*.*) to see the target file.

10) Select that binary file and click the Open button. (Be careful to only load .s19 files! If you mistakenly select any other file extension, it is highly likely that you'll mess up the bootstrap loader program.)

11) See line after line of ASCII data displayed in short one-second bursts. This will continue for about 2 minutes until the entire program has been sent to the Micro908 and flashed into the MPU. When it is complete, the "HCmon >" command prompt will be displayed again.

12) Reboot the Micro908 (cycle the power) and see the new version of loaded software reflected in the Main display

SUPPORT

Please check the m908 Loader web page at www.amqrp.org/m908loader for the latest information, capabilities, FAQ list, and newer versions of the m908 Loader program. You may also write to George Heron, N2APB at n2apb@amqrp.org and we'll do our best to answer your questions or help solve any problems you might have using the m908 Loader.

ACKNOWLEDGEMENTS

The American QRP Club would like to thank the designer of this program, Bob Hillard, WA6UFQ, for his ready and able willingness to help out with the **m908 Loader** project.

APPENDIX H: DDS DAUGHTERCARD UPGRADE

The following page describes several changes that should be made to your "stock" DDS Daughtercard Kit" in order for it to operate properly with the Micro908. We've upgraded the RF amplifier so the card can produce enough signal reliably throughout the frequency range of interest in the Micro908, and we've added another onboard voltage regulator so that RF amplifier's output signal level will not be dependent upon the supply voltage. These two features are significant and will produce reliable operation of the Micro908 even when battery-powered.

[ ] Make the DDS Upgrade Kit" mods to the DDS Daughtercard

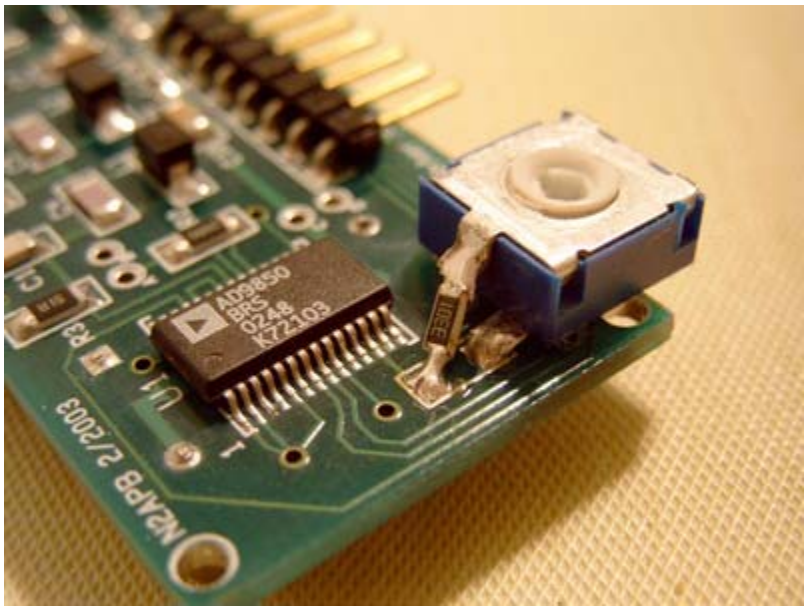
These changes are outlined on the following sheet entitled "DDS Upgrade Kit". The parts for these modifications are provided in this Micro908 Kit and they must be added in the prescribed manner to the DDS Daughtercard.

Another change is required to allow variable tweaking of the signal level provided by the DDS Daughtercard. This is a needed capabililty that was discovered toward the end of the Micro908 development effort when we discovered that some of the upgraded DDS cards were "hot", or providing a signal level greater than we expected. Thus it is necessary during the Micro908 calibration steps to adjust a trimpot on the DDS card to ensure that the signal level provided to the reflectometer circuitry is not too great. This "trimpot" addition did not make it to our standard "DDS Upgrade Kit" instructions, so we'll describe this simple step here.

[ ] Add the Trimpot to the DDS Daughtercard

Locate the trimpot supplied in the Micro908 Kit and orient as shown in the photo below. You may use Super Glue, RTV or a small daub of epoxy to hold the component in place. Be careful not to use too much adhesive, as it may leech up into the component and prevent the wiper from moving.

The idea is to put this 10K-ohm trimpot in series with the new value of R1 supplied. Solder one leg of the trimpot to the lower R1 pad. Solder the SMT resistor R1 between the upper ("middle" or wiper) lead of the trimpot and the other R1 pad, literally having R1 angled up in the air reaching out to the trimpot lead.



This technique has worked well in all of our betas and the trimpot can be adusted during calibration to have the DDS chip generate the required signal level to the RF amplifier.

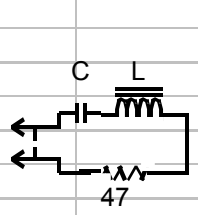
APPENDIX I: Making a Dummy Antenna for Testing

You will find it convenient to have a "dummy antenna" to use in testing the Micro908 Antenna Analyst operation. A dummy antenna is simply a 3-component circuit that is resonant at a specific frequency and has a purely resistive characteristic at that resonant frequency. The dummy antenna can be used in place of having an actual antenna during the initial tests of the instrument.

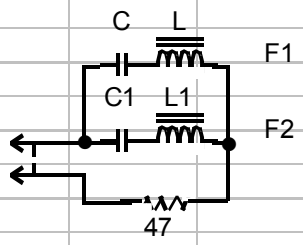
The dummy antenna described here consists of a resistor, inductor and capacitor in a series circuit, constructed on the end of a BNC connector that can be applied to the RF output of the Micro908. You can select the component values indicated below to create a resonant dummy antenna at any of the specified frequencies.

Dummy Antennas

Freq. (Mhz)	C (pf)	L (uH)	Core	# Turns
7.1	82	6.8	T50-2	37
10.1	47	5.28	T50-2	33
14.1	27	4.7	T50-2	31
18.1	22	3.51	T37-2	30
21.2	27	2.08	T37-2	23
28.5	15	2.08	T37-2	23



Single Resonance



Double Resonance

