Here is an evaluation report of LCD HF-SWR Meter, carried out by Jeroen/PE1RGE. My answers to his technical points are given in Blue. This document is presented here so that kit builders get a good idea on assembly and alignment of this SWR meter.

73s/Dinesh

Evaluation report by: Jeroen PE1RGE

Subject: Kit evaluation "report" on the LCD HF-SWR-meter construction!

I started to make notes when soldering the kit together. The micro controller board was the first one. I will give you the remarks in the order I made them during the building process:

1. Resistors R6 and R7 are 2.2Ohms instead of the 2.5 Ohms.
   (Will change in schematic)

2. The X-Tal (X1) has no insulating spacer, so I mounted it slightly elevated above the PCB in order not to get short circuits.
   (Kits will not be supplied with spacers; it would be best to keep it elevated by 0.5mm from board)

3. Why did you call the resistor connected between C7 and U2: "RZ"?
   (Name RZ was given because that R must be determined according to use of 2.5V Regulator like 1009 or using a zener. Actual kit packing list will mention this clearly)

4. In general: to improve the readability of your schematics use a ground-symbol instead of the long lines connecting al the GND-points together.
   (The ARES program that I use, do not have that symbol. May be you are right to do so. I will create one nice ground symbol soon)

5. Make a check box on the components list to mark if the 2.5V Zener is supplied or not. All associated components can than be installed
   (Yes, this will be done)

6. Give each capacitor its own symbol. E.g. there are three capacitors C15, 16 and 20 associated with one symbol. I think it will be more clear when each capacitor has its own symbol.
   (Will check in program if they are available. Else, must create them)

7. The mounting holes for RFC1 and RFC2 on the microprocessor board are too close to each other.
   (Yes, can't correct it now because production boards are made. May be next lot!)

8. The supplied screws for the stand-offs are too short! Furthermore, I think standard M3 material would be a better choice
   (This will be done)

9. Add two extra holes for solder pins near the power connector. In such a way, one can make a choice between "hard wired" power supply leads to the pins or using the adapter connector. (Good suggestion. This too for next run of boards.)

10. Resistor R10 and the spacer next to it are very close together; a little bit more space would be convenient. The same counts for RV1 and the spacer next to it.
    (Thanks. Will keep in mind when reordering next lot of PCBs)

For the RF-Bridge board:

1. I couldn't fit the supplied N-connectors to the board, without using a drill to make the holes bigger.
   (Kits will be supplied with SO-239 connectors. I think they fit well, as I used them on my first prototype board.)

2. Some of the supplied screws for mounting the connectors didn't have a slot(!) to mount them. The head of the screw was not finished.
   (I will watch for this & keep supplier tight!!)

3. There is two times a C8 in the schematic of the RF-board. Only the 82pF capacitor is C7, the 15pF capacitor is C8.
   (Thanks, this really went un-noticed. I will amend it soon)

4. Extra tip: I wound the toroid and placed a piece of heat-shrink tubing over it. (In such a way that the coax still fits through the holes;-)). The tubing keeps the wires nicely in place and protects the as well.
   (I think, I will supply Teflon tapes with toroid for same purpose)

5. I added a piece of Teflon sleeving over the pin of C13 connecting to the PCB (to protect it from short-circuits against the mounting-hardware of the RF-connectors).
   (Will look for Teflon sleeves. If available, they will be in kits)

After assembly the whole unit functioned almost directly.

I adjusted the contrast of the display, and I was able to "scroll" through the menus.

I then adjusted RV1 in such a way that 0,65Volts was supplied to the diodes.

RV2 and RV3 were left at their factory setting.
Then I connected the transmitter and started with low power on the 80m band.

Then I noticed the reading for the forward power and the reflected power were reversed.
I had to change the wires, connected to R and F.

The wiring scheme became:

Micro-board RF-board
P ---------- P
R ---------- F
G ---------- G
F ---------- R

(Above Errors will be corrected on the Bridgeboard Silk)

For the rest the circuit performed very well!

If I'm correct the meaning of the LEDs is as follows:

Green led of red led off: SWR <2
Green led on, red off: 2<SWR<3
Green led on, red on: SWR >3

Some hints on setting the Potentiometers:

Here is how I did it:

I finally tuned RV2 and RV3 for the same voltage at ascertain power-level of the transmitter. So, measuring V-forward, note the value. Then reverse the antenna and transmitter connection, measure V-reverse and adjusting RV2 and RV3 in such a way that the voltage would be the same.

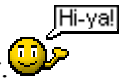
In general, some parts were difficult to solder due to the fact that there were no thermal relieves. But we already discussed that part

Finally:

The kit performs very well!
The response of the display is fast enough.
The dimmer for the background is a very nice feature!

The list above might seem a bit pessimistic, but the final remark would be that there are no major mistakes.

Only few small things to change, to make it even better.



Most of the remarks are more about the schematic, than about the real electronics.

Here is my report at this time. Read it, if you have further questions or remarks please let me know.

By the way, people at the local ham-club (PI9SRS (www.PI9SRS.eu)) were very enthusiastic about the project! I even got requests from people to bring it with me again next time!

Thank you for the opportunity to help you with this kit, "mike back to you",
(Thanks Jeroen, without your help, this project was going to be in cooler for a long!!)

73 Jeroen PE1RGE

22nd April 2008

For more information on this project and other projects & kits, please visit:
http://www.foxdelta.com