FE-5680A Breakout board

I. Muir   MW0XEN/p March 2012
Board Revision V1.1
cadbloke@hotmail.com

Like many, I picked up one of the current batch of cheap EF-5680A's.
Not being clear where I would end up using it, I laid out a breakout board to make it easy to play with on the bench, but with the expectation that it would remain attached when I got around to mounting the Rb in a box (or where ever).

Summary:
- LM1084Adj for 15V (pin compatible with LM317, but rated for 5A).
- LM1117-5 for the 5V.(rated 800mA)
- Both VRegs have thermal and overload protection.
- LDO voltage for both VRegs is under 1V for the given load.
- The VRegs can be fed individually or jumpered to run the '1117 from 15V. This works, but both regs run fairly hot.
- 1PPS and LD are brought out to individual headers.
- 10MHz is brought out to an SMA connector.
- RS232 is given its own DB-9 wired for 2, 3, & 5 (TX, RX, Gnd) for a standard cable.
- If it is desired, solder pads are provided to allow a straight through cable to appear as a Null Modem.
- V1.1 board differs from V1.0 only in the component placement and minor additional heat sinking for the regulators. The pinouts remain unchanged.

One thing to note is that this board is designed for the current batch of cheap FE-5680A with the following pinout:

1. +15V (~1800mA cold: ~800mA hot).
2. Gnd.
3. Rb Lock (lock = low).
4. +5V (~85mA).
5. Gnd.
6. 1PPS (high until Rb Lock goes low).
7. 10 MHz.
8. RS323-RX.
9. RS232-TX.

I've seen some refer to 'Pin 5' as Sig Return, but on mine, pins 2, 5 and the case are all common.

There is nothing exotic about the circuit design. In fact I've tried to be as "general purpose" as possible.
IC1 (LM1084-adj) has its output voltage set by R1 (240R) & R2 (2k7). The set voltage should be 15.3V
If the 0.3V bothers you, $V_{OUT} = 1.25V \left(1 + \frac{R2}{R1}\right)$ soldering a 120k resistor in parallel with R2 will get close to 15.0V.
C7 (LM1084 –Adj pin) is optional, but suggested on the data sheet for improved ripple rejection. Although I've specified it as 0.1uF, it should be tailored to suit the upstream PSU to get the best effect. As that will be a personal choice, you can adjust as appropriate.
Capacitors C8, 9 10 are Tantalum selected for low ESR to help reduce noise.
R3 (current limiting 5V on Molex LD) is a pretty arbitrary value and chosen to be the same as R1 for convenience (also convenient is that it limits the current to 20mA).
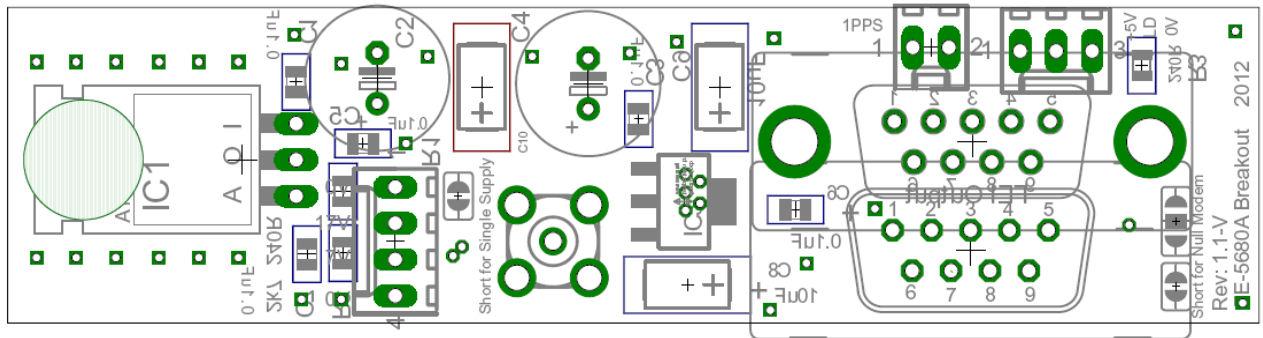
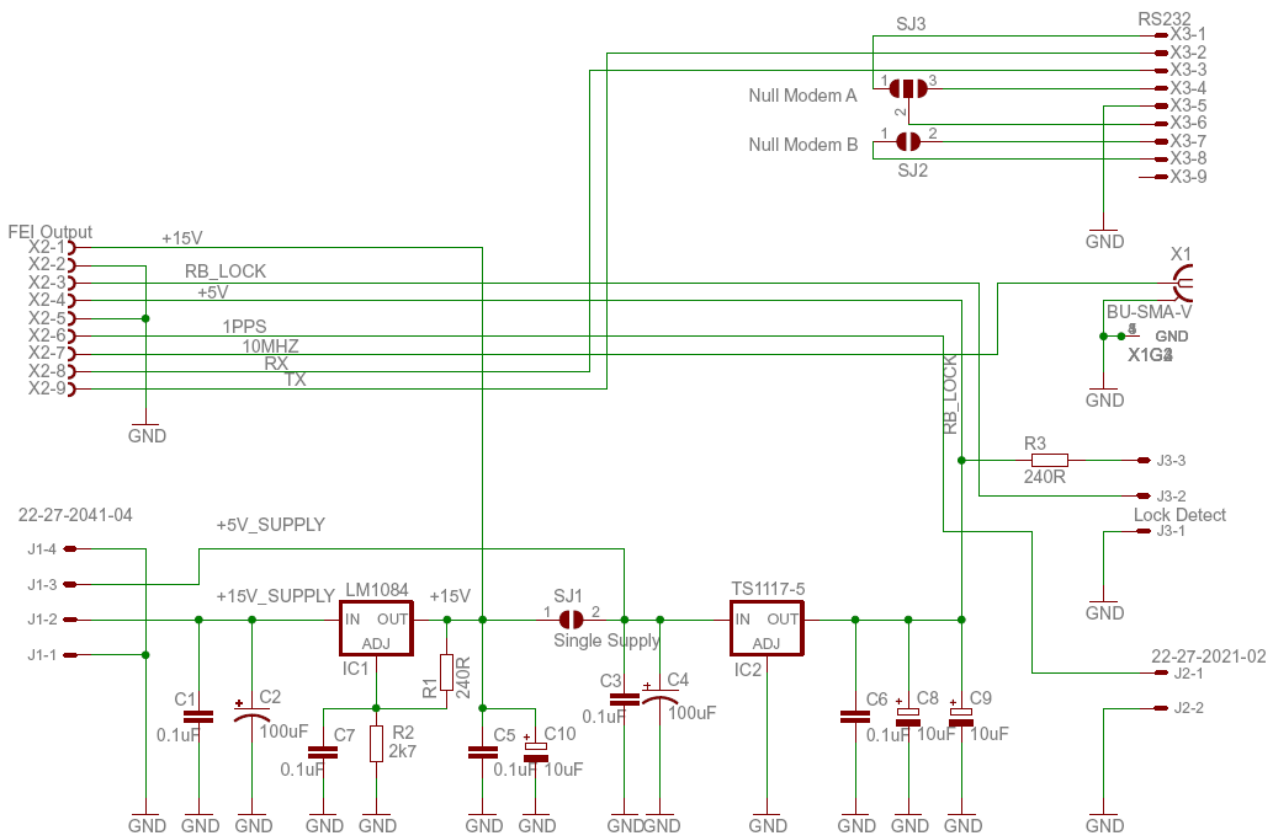**Fig.1   Component layout: X-ray View.**

Fig.2   Schematic

Build order is straight forward – SMD first, Molex headers last.
If no additional heat sinking is planned, soldering the LM1084 to the PCB works as well as a screw. If a heat sink is added, be aware the tab is at V_{out} (15V)

Hint: If your '5680A was supplied on a PCB, you'll have a DB-9 with retaining clips fitted. Recover the clips and fit to the DB-9F *before* soldering to the board. Holes are provided to give screw driver access through the board.
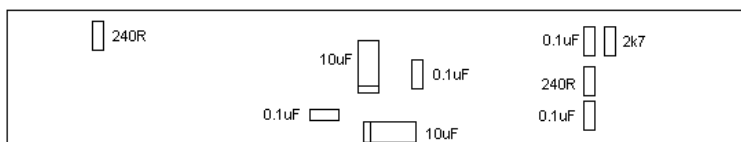

Fig 3 SMD Positions

Data Sheets
http://www.ti.com/lit/ds/symlink/lm1084.pdf
http://www.ti.com/lit/ds/symlink/lm1117-n.pdf