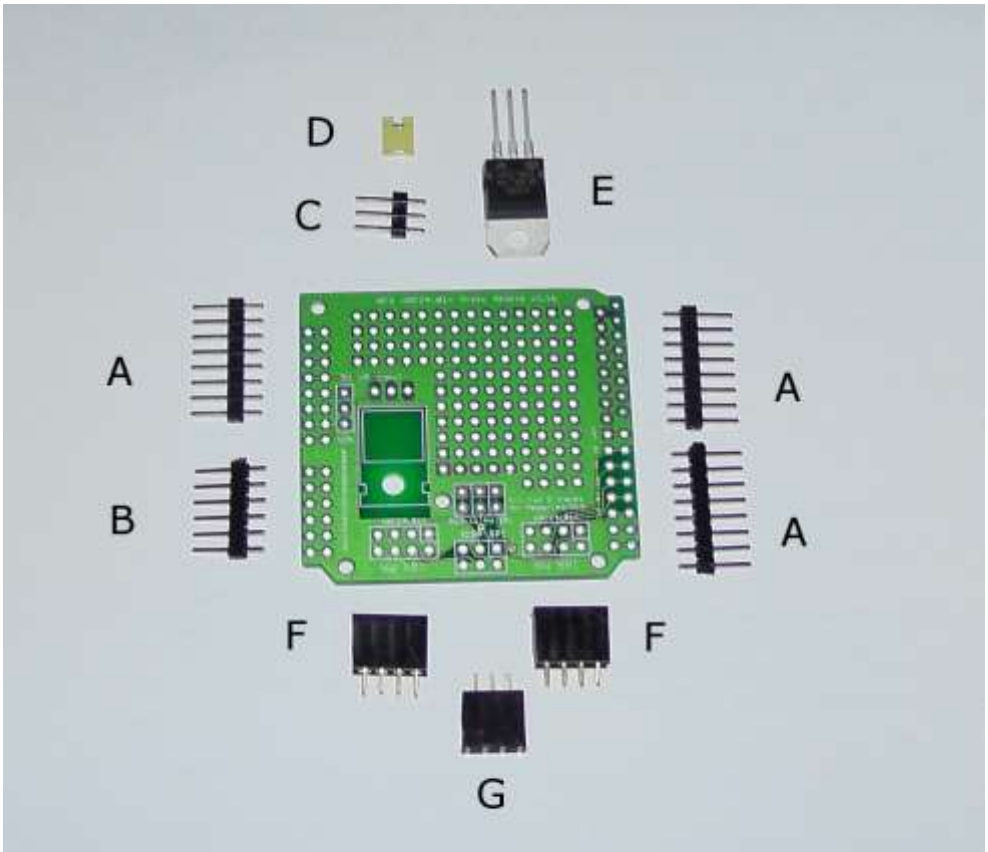
## RFX nRF24L01+ proto shield assembly instructions. (v1.1bc, Sept 2013, updated June 2014, embeddedcoolness.com)

(Note: If assembling shield for use with a PJRC Teensy 3.x dev board, please read “Addendum: Deploying the RFX nRF24L01+ proto shield with a PJRC Teensy 3.x dev board” at the end of this document first.)




**RFX nRF24L01+ proto shield kit components part list:**

**A** - 8 pin male header (3pcs)
**B** - 6 pin male header (1pc)
**C** - 3 pin male header (1pc)
**D** - pin header jumper (1pc)
**E** - 3v3 low drop voltage regulator (1pc)
**F** - 4x2 pin female header (2pcs)
**G** - 3x2 pin female header (1pc)

Note: The 8 pin male header that covers D8-D13,Gnd, Aref (shown top right as **A**) can be also substituted with a 10 pin male header if soldering access to the SDA and SCL pins in this position if required. Kits will have a 10 pin header supplied to allow for this option. If an 8 pin male header is preferred, simply cut the 10 pin header down to an 8 pin header.

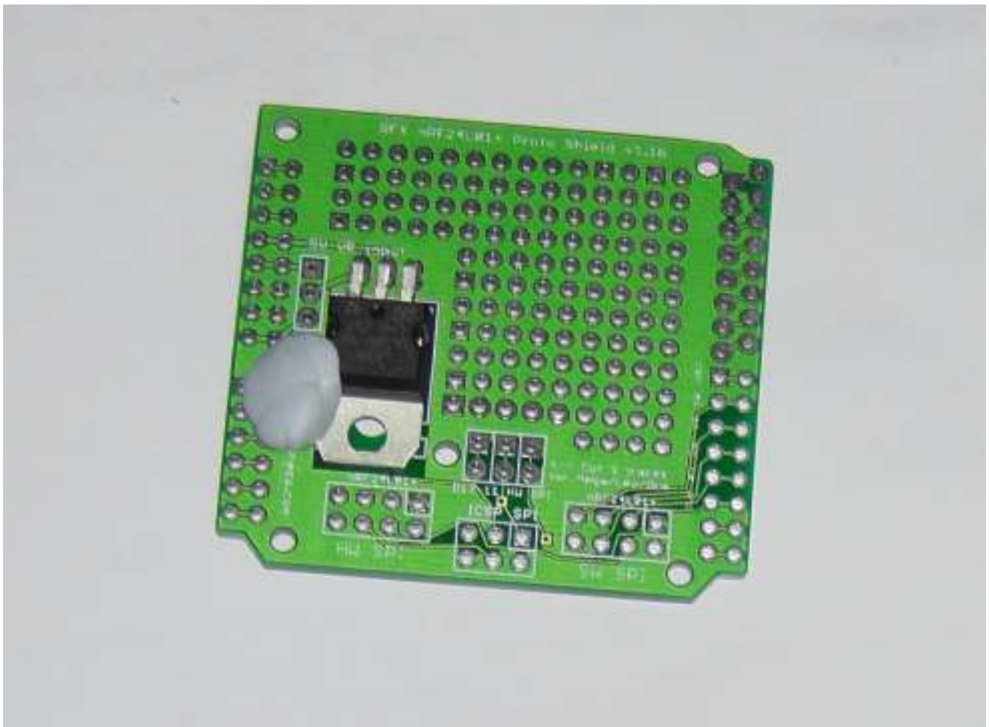
**Recommended assembly steps:**

Step 1)

Mount 3v3 voltage regulator (E)

Tip: Bend pins on regulator $90^{o}$ so the component lies flat on shield before soldering pins. (It doesn't matter if hole on regulator and board aren't exactly lined up, but part should be straight and flat on board.)

Tip: A blob of "BluTack" (or similar poster mount putty) is an easy way to hold regulator on board and lined up straight while turning board over to solder pins.

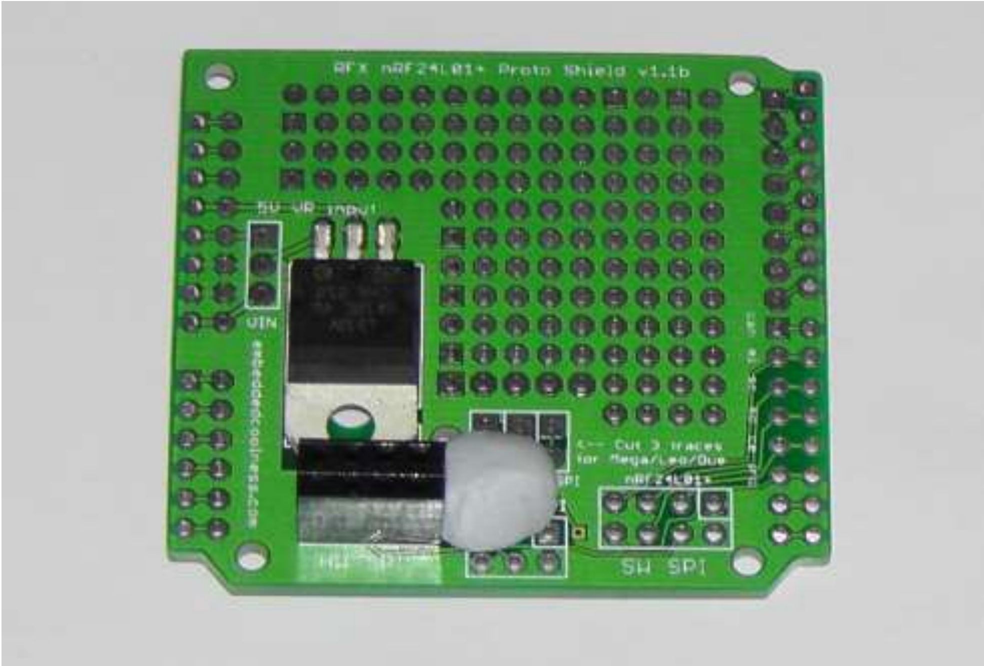


Tip: Start by soldering just one pin of regulator, then turn over board to check the part is still straight and flat. This makes it easy to resolder if needs straightening. Once straight, solder remaining 2 pins.

Step 2)

Mount 4x2 female pin headers (F)

Tip: "BluTack" tip above applies.

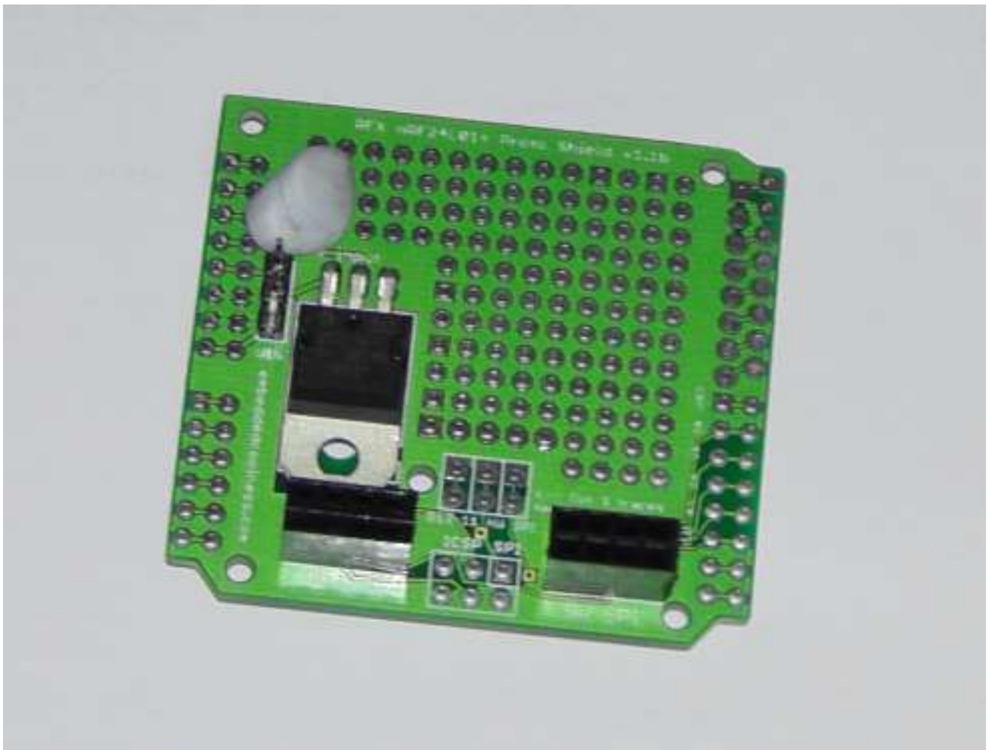



Tip: Only solder one pin and then turn over to check for straightness. This makes it easy to resolder if needs straightening. Once straight, solder remaining 7 pins.

Step 3)

Mount 3 pin male header (C)

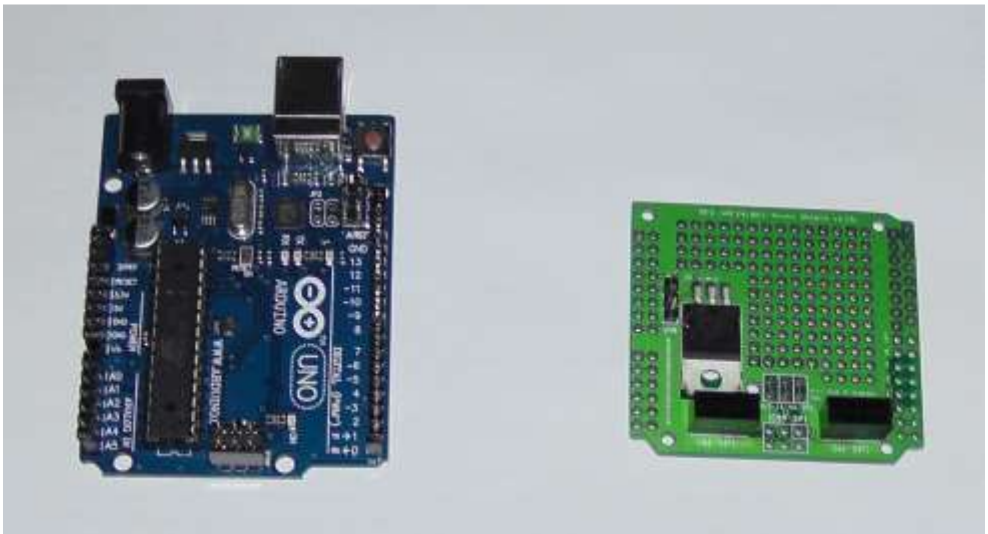
Tip: "BluTack" tip above applies.



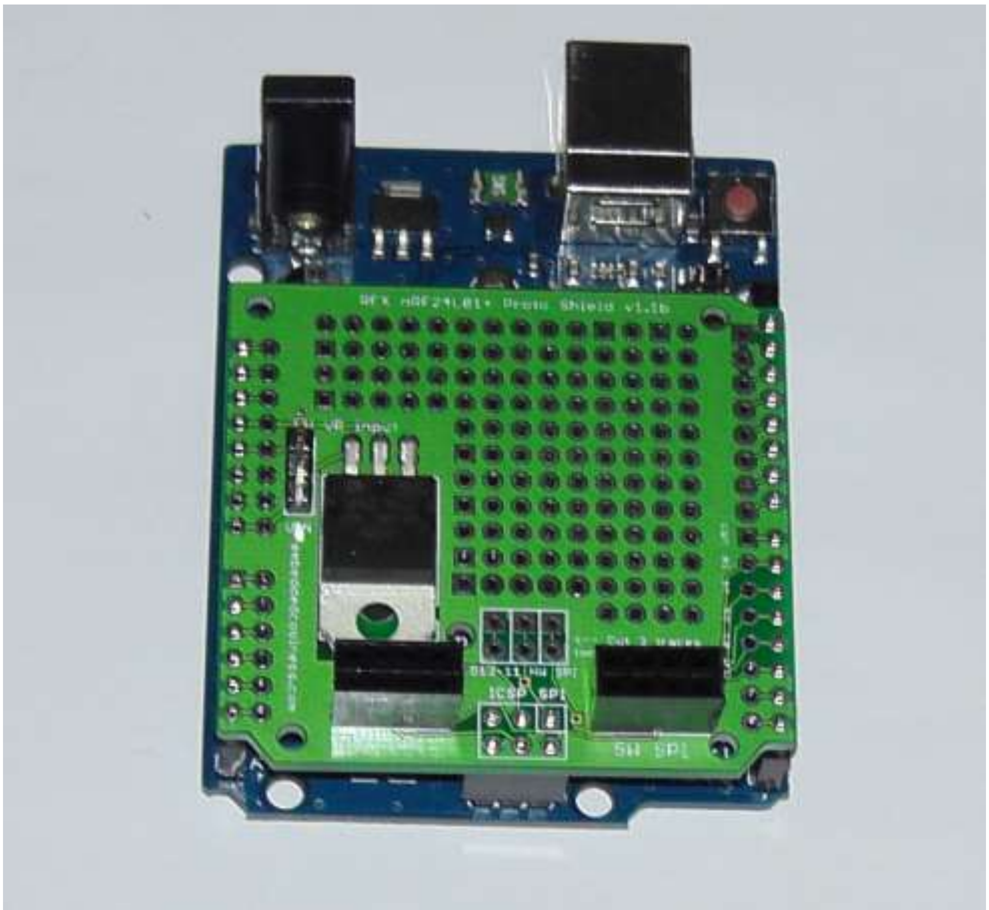
Step 4)

Mount male single row pin headers (A and B) and 3x2 female pin header (G)

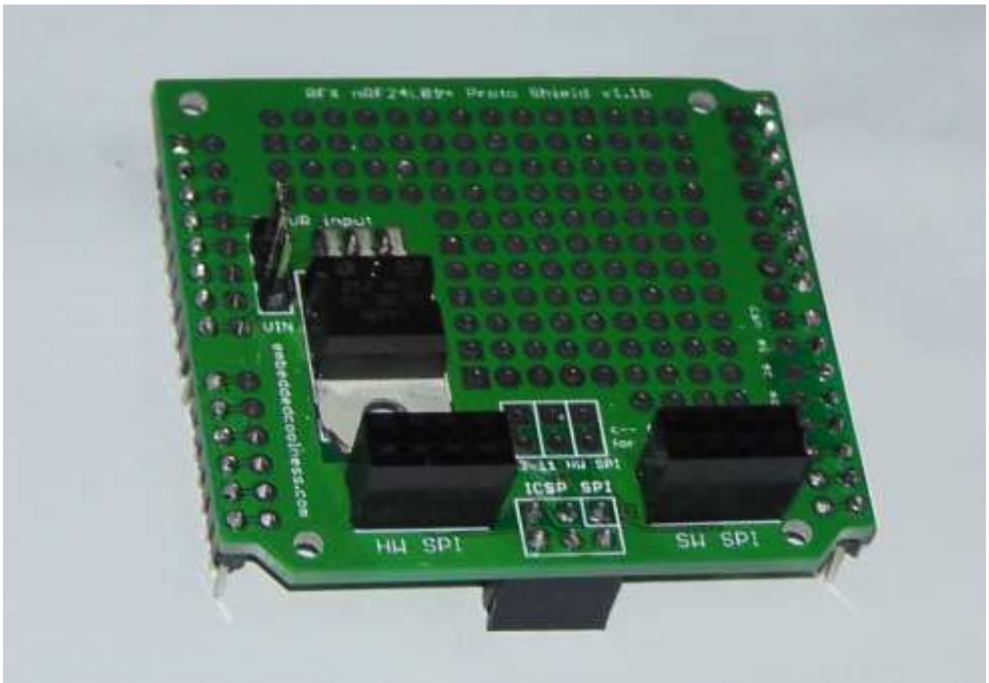
Tip: ensure all headers are lined up and straight by plugging loose headers into a suitable target board first.



Place shield on headers.




While shield is sitting on headers, solder two pins on each header to keep headers in place. Remove shield from board to finish soldering header pins on shield.



Shield is now complete and ready for deployment.

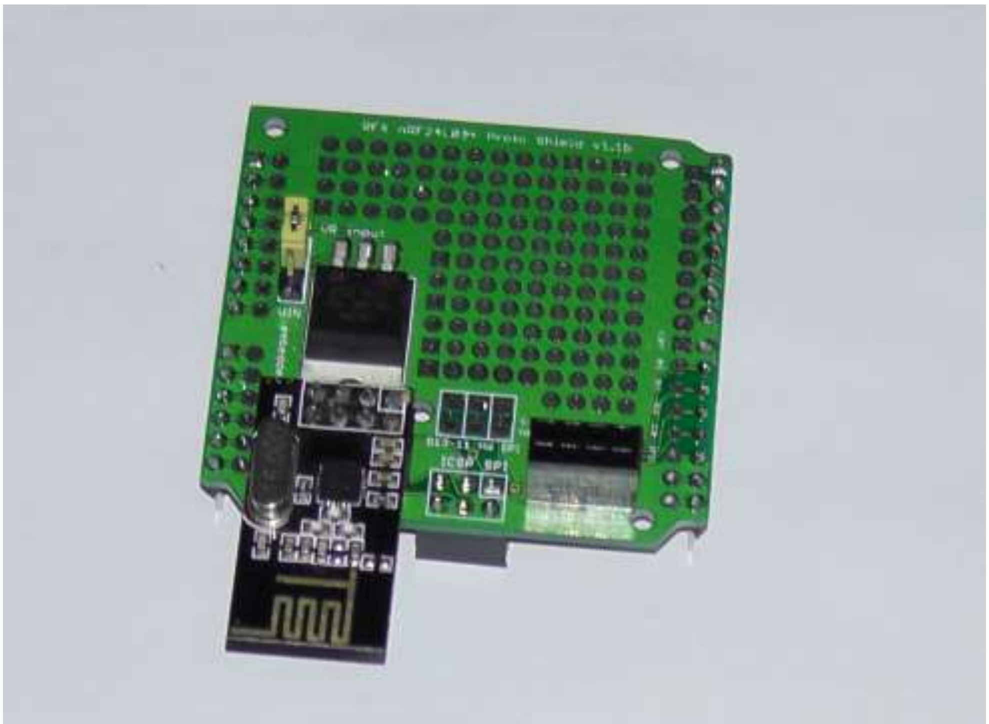
## Deployment:

- Select input source for voltage regulator, Vin or 5v pin, positioning jumper (D) on 3 pin header (C). Jumpering pins 1 & 2 selects 5V pin as input source, 2 & 3 selects Vin as input source. Selecting 5V draws slightly less current overall, but also reduces available supply from 5V regulator on dev board. Will supply cleaner input if supply on Vin is not well filtered and regulated. Selecting Vin draws slightly more current overall, but doesn't place demand on 5V rail. (In most situations the choice won't matter, but it is there is if needed.)

- Plug nRF24L01+ radio module into either HW SPI header or SW SPI header.

- If using the shield with HW SPI on Mega, Leonardo, Due, or other dev board that *doesn't* have HW SPI connected to D11-D13, on the v1.1b shield it is necessary to cut traces between three pairs of connected through holes, where marked on board (above the 3x2 ICSP header). They are labeled D13-D11 HW SPI. (On the v1.1c shield, they are not connected by default, so cutting is unnecessary. If you wish to connect these, follow the instructions below.)

If you wish to connect or reconnect these traces for a board that *does* have HW SPI connected to D11-D13 (e.g., Uno, Teensy3.x), you can do so simply by soldering in three links to reestablish the connections. This might be necessary, if, for example, access to the dev board ICSP pins weren't available on a particular dev board, or because another shield is using them.



Shield shown above configured with low power nRF24L01+ module using HW SPI, 5V voltage regulator input source selected.

Shown above configured with high power nRF24L01+ module using SW SPI, Vin voltage regulator input source selected.

## Additional decoupling caps for use with high power modules:

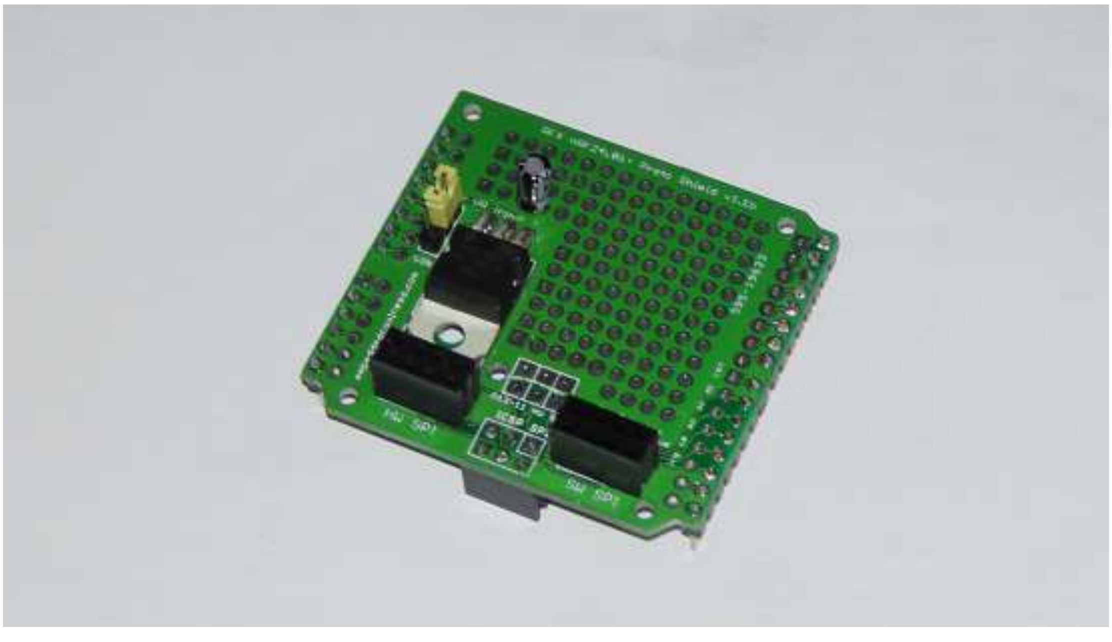
Some of the high power modules show better performance if two additional decoupling caps are added across the voltage regulator. (The low power modules seem to operate perfectly well without this modification, however, presumably because of their significantly lower current requirements.)

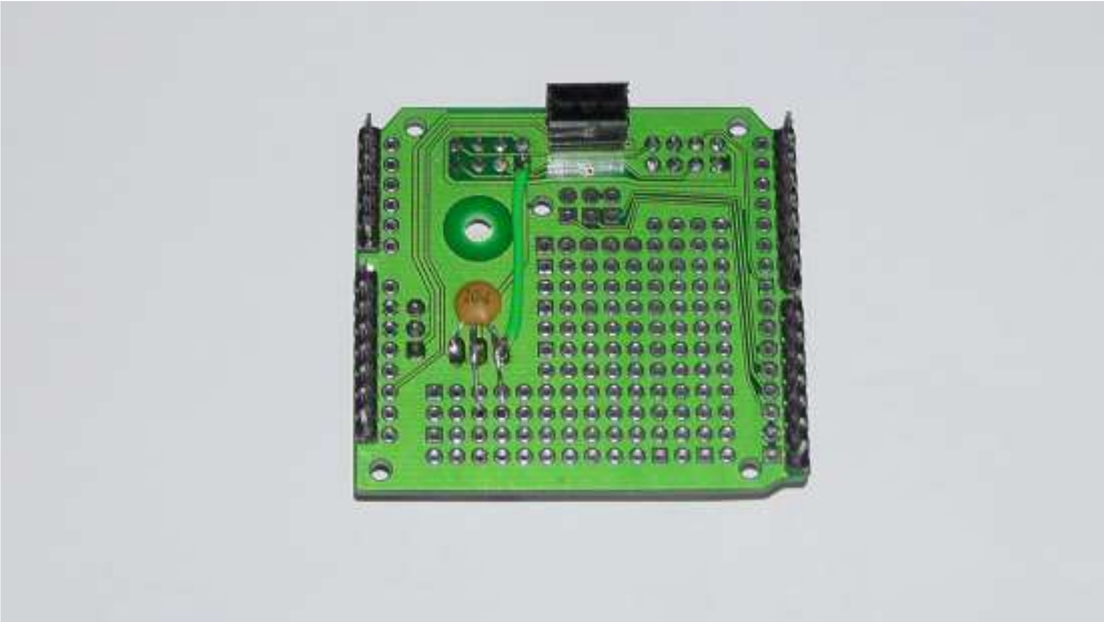
The performance benefit for the high power modules is not uniform, even amongst modules of the same type from the same source, so this step may or may not be required, depending on individual characteristics of the particular modules. It is an easy step to try if you suspect a high power module is performing less than optimally, however.

Use the datasheet as a guide for the values for the specifics regulator used (a low drop 3v3 regulator such as the LD1117 will be needed if using the 5V dev board pin as the supply input.) If you bought the kit with the components option the regulator will be a LD1117.

The LD1117 datasheet suggests a 0.1uF ceramic cap across the supply input and ground pins of the regulator, and a minimum 10uF electrolytic cap across the output and ground pins of the regulator (make sure to observe correct polarity for the electrolytic.) The precise value of the electrolytic does not seem to be particularly critical for the LD1117… values between 10uF and 470uF will work fine, although some experimentation on the optimal value for a particularly finicky module may be indicated. Also, the source of the input supply to regulator may be a consideration; the characteristics of the supply will vary even depending on the dev board being used (the “megas”1280/2560 boards being notoriously noisy.)

Finally, connecting an extra wire directly from the ground pin of the module header to the ground pin of the regulator improves the current return path for the module, which can also improve decoupling characteristics of the circuit. A shield with all these mods is shown below. (Note the mounting of the ceramic 0.1uF cap under the board to keeps the leads as short as possible.)





**Reference:**

HW SPI header connections:

nRF24L01+ module SPI pins connected to D11-D13, and also ICSP header SPI pins

D11- MOSI
D12 - MISO
D13 - SCK

SW SPI header connections:

nRF24L01+ module SPI pins connected to D4-D6

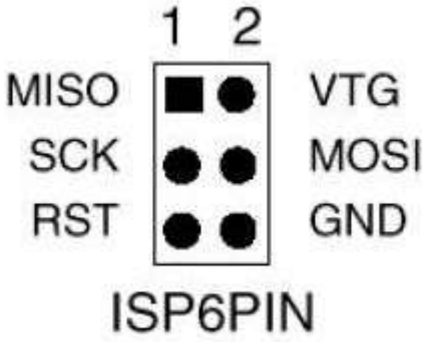
D4 - MOSI
D5 - SCK
D6 - MISO

For both HW SPI and SW SPI headers:

D2 IRQ
D3 CE
D7 CSN

Note: If running software that does not use nRF24L01+ IRQ, then D2 can be freed up by cutting trace on top layer of board between “inner” (0.1” pitch grid aligned) and “outer” (standard Arduino shield) headers for D2, if required.

6-pin ICSP pinout:




8-pin nRF24L01+ pinout:

## Addendum: Deploying the RFX nRF24L01+ proto shield with a PJRC Teensy 3.x dev board

Although not designed primarily as a Teensy 3.x "carrier board", the shield can be adapted to compactly host a Teensy 3.x dev board in a fairly straightforward way, described below.

The assumptions regarding the header configuration on the shield will be different, however, so if deploying with Teensy 3.x board, please read this addendum first, before referring to the general assembly instructions. In particular, note that

a) mounting a 4x2 female header block to the SW SPI will definitely not be applicable (mounting this header block will obstruct mounting the Teensy on the shield. The Teensy will need to use the HW SPI header to connect to a nRF24L01+ module).

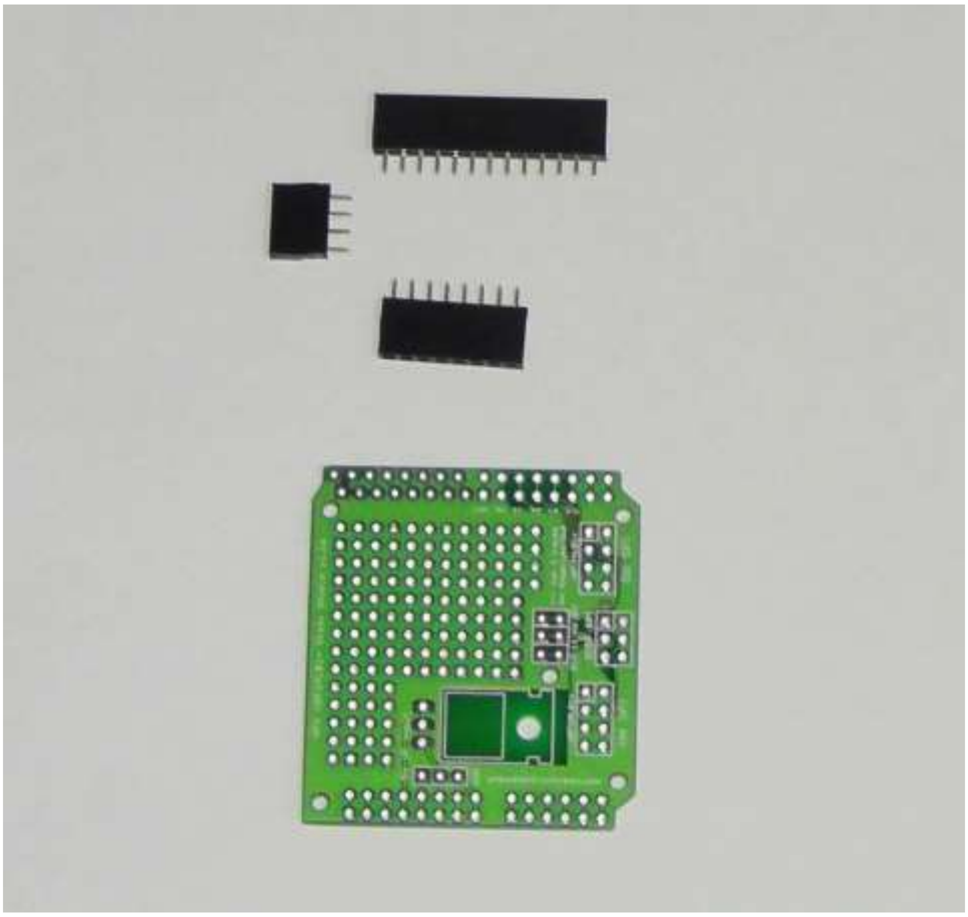
Also,

b) the 3x2 ICSP SPI female header block, and
c) the Arduino shield male pin headers for connectivity to a stacked shield/board under the nRF24L01+ proto shield, may or may not be applicable, depending on the project design.
d) For a v1.1c shield, the links connecting HW SPI to pin D11-13 will need to be installed (refer "Deployment" p5). For v1.1b shields, the connections are already in place by default, so installation will be unnecessary

To set up a Teensy 3.x on the shield as in the photo above, you will need three lengths of 0.1" female header (13, 4 and 8 pin lengths).

This will break out Teensy 3.x IO pins 0-20 (corresponding to D0-D13, A0-A6), as well as Reset (or A14/DAC on Teensy 3.1), Program, GND, and 3v3.

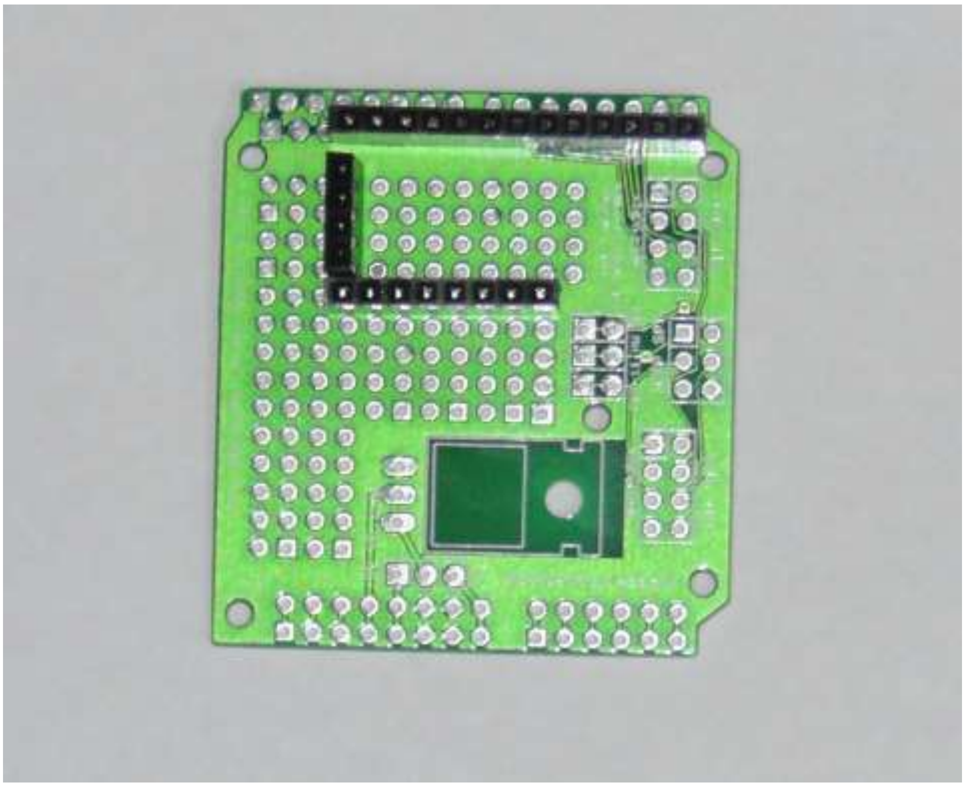
While this will be a usable subset of pins for many applications, please note a limitation of this arrangement is that Teensy 3.x pins VBAT, VUSB, AREF, AGND, and A7-A11 aren't broken out on the headers, as aren't any of the under-board IO "pads" D24-D33, and A12-A13. Access to these pins and pads, if needed, will require a less convenient solution, likely involving fixed wiring.



1) Solder the 13 pin length piece of female header on to the D0-D12 "inner" (0.1" grid aligned/breadboard compatible) header holes.

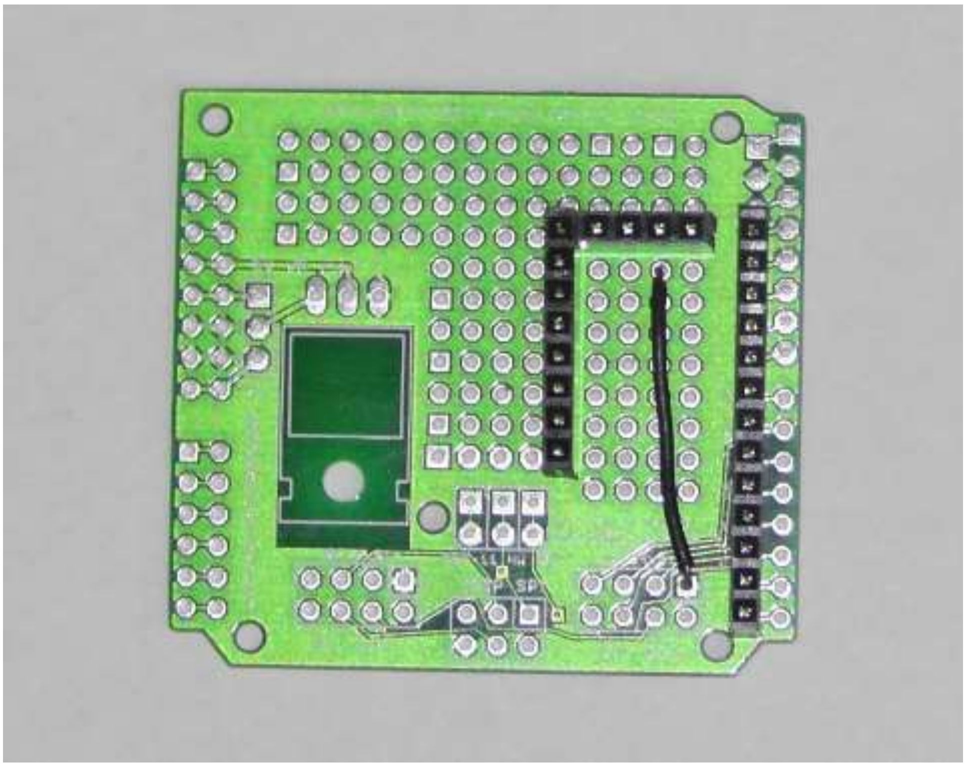
2) Solder the 4 pin length header on to the prototype area, perpendicular to the first header piece, lined up with the D12 pin position. (Note the gap between these header pieces).

3) Solder the 8 pin length header parallel to the first header piece, abutting the second header piece.

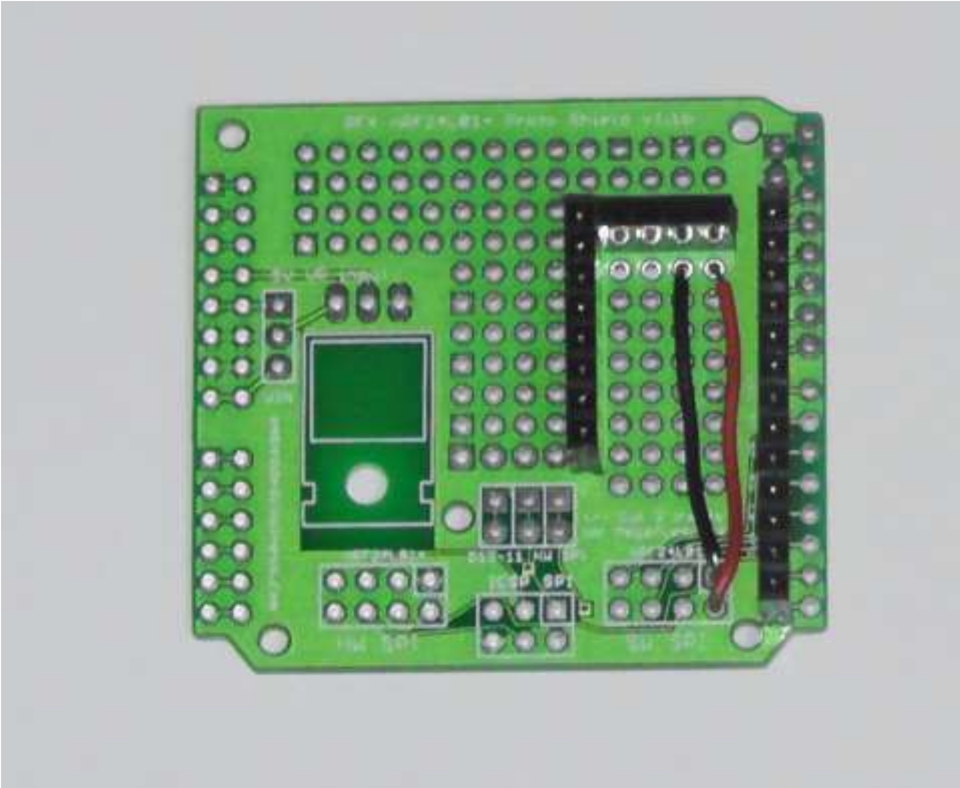


Next, using three pieces of hook-up wire:

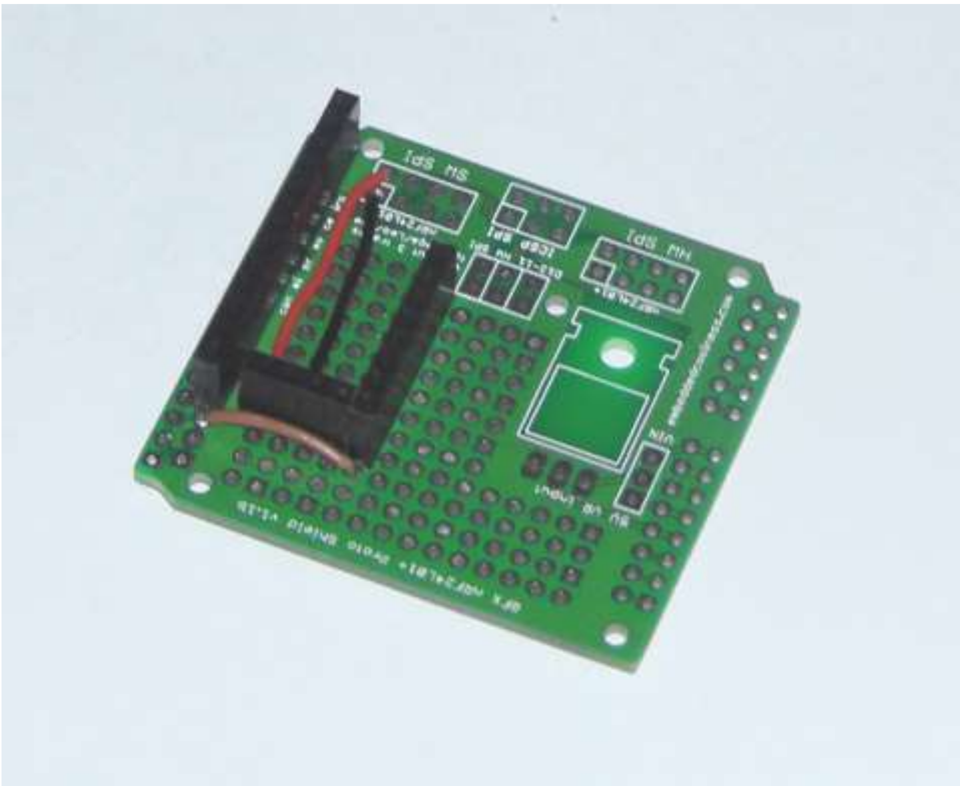
1) Connect Gnd on Teensy female header to Gnd connection on SW SPI header.



2) Connect 3v3 on Teensy female header to Vcc connection on SW SPI header.



3) Connect D13 on Teensy female header (first pin on 8 pin header piece) to the D13 hole on “inner” (0.1” grid aligned/breadboard compatible) header. (This completes the connection of Teensy pins D0-D13 to the corresponding Arduino digital IO pin headers.)

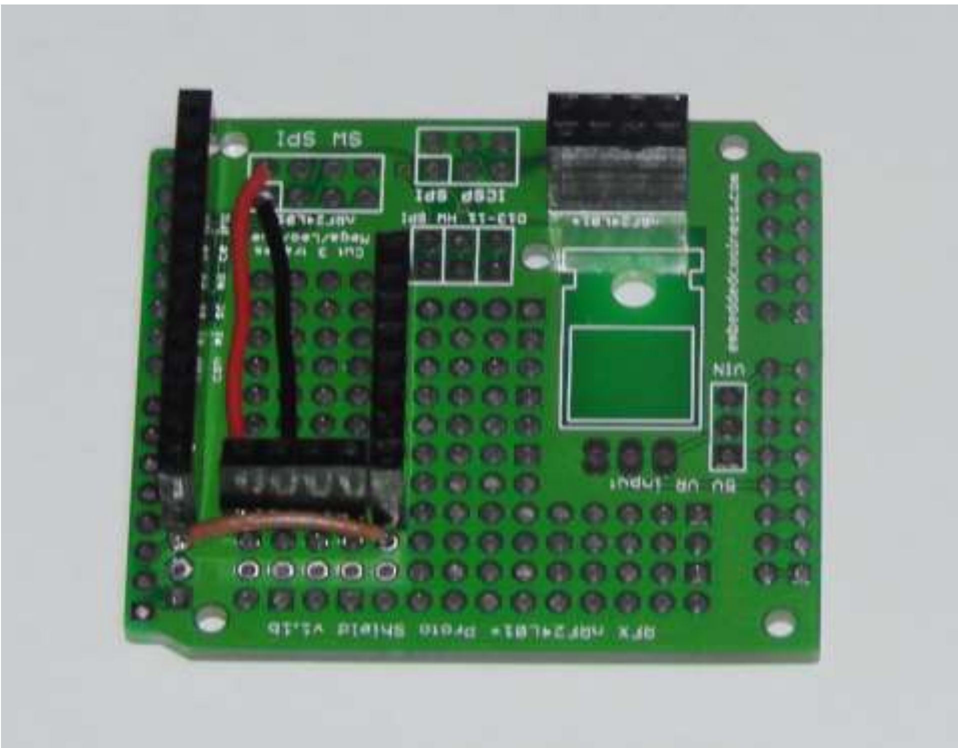


With the shield configured this way, the Teensy 3.x will now connected via the nRF24L01+ HW SPI header pins as follows:

D2 IRQ
D3 CE
D7 CSN
D11 MOSI
D12 MISO
D13 SCK

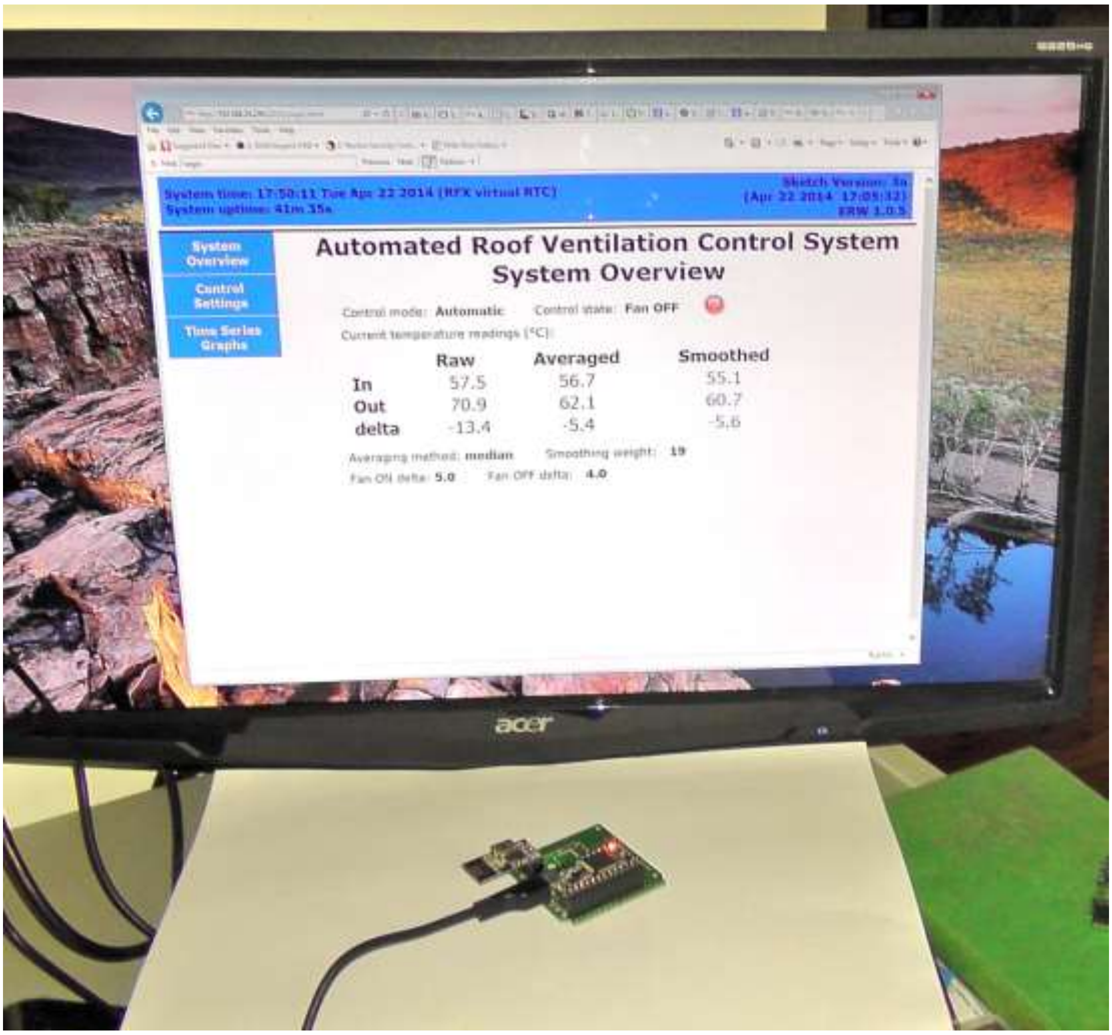
This is sufficient to drive a nRF24L01+ module attached on the HW SPI header.

Attach a 4x2 female header block to the HW SPI header, and the RFX bRF24L01+ proto shield is now in the minimal deployable configuration for a Teensy 3.x!



Powered via a USB cable connection, the Teensy's on-board voltage regulator can provide 100mA excess capacity, which is sufficient to power a low-power nRF24L01+ module. (A high-power mRF24L01+ module with separate PA+LNA stages will exceed this, however. Using one of the externally powered 3v3 rail options is recommended for that application.)

Below is a photo of a Teensy 3.0 on a RFX nRF24L01+ proto shield in the minimal configuration running a demo control application with embedded web server over the nRF24L01+ radio link (see: RFX demo "Automated Roof Ventilation Control System" -- stage 3a).




The minimal configuration can be extended in various ways, as required. External power options have already been touched upon. Additional discussion on various power options using the shield's optional 800mA LD1117v33 voltage regulator is in the "Deployment" section of the general assembly instructions.

If the Teensy will be connected to a USB source, accessing 5V from VUSB to supply input to the LD1117v33 may be an option. (Consideration of the Teensy's "poly fuse" between the USB connector and the VUSB pin, limiting current to 500mA, must be given for such an arrangement.) VUSB could also potentially supply 5V to the 5V pin on the Arduino power header in order to provide a 5V source to connected shields.

If connectivity to other Arduino shields is desired via the shield headers, in this configuration, it may be more straightforward (due to height clearances) to have the nRF24L01+ proto shield at the top of the shield stack, rather than at the bottom or in the middle. Also, note that while to digital pins D0-D13 are already connected to the Arduino shield headers in this configuration, the analog pins A0-A5 won't be, and will be need to be explicitly hooked up if this inter-shield connectivity is required.