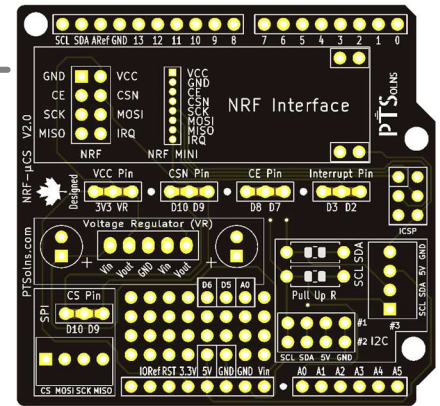


Overview

The *NRF-Shield* is an interface board intended to be used with the common NRF24L01+. This board can work with any of the three common NRF packages: the standard size, the mini and the +PA+LNA. All three of which are simply referred collectively as “NRF”. Hardware pin configurations give the user flexibility in implementing their design. The board offers I2C and SPI interfaces as well as a small prototyping section.

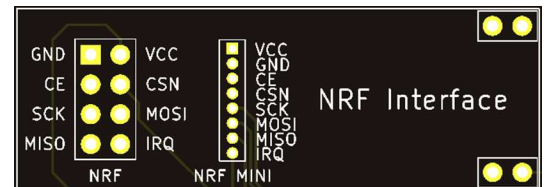


Compatibility

The physical PCB outline has the same width as compared to the Arduino Uno R3 or Leonardo, etc., but is shorter in length. Three of the four mounting holes on the Uno R3/Leonardo, etc. are found on the *NRF-Shield*, with the same position and diameter. The fourth mounting hole was sacrificed to add an interface (see below). Although the shield is stackable, for best RF communication, this board should be installed as the topmost shield. The *NRF-Shield* is compatible with the Arduino Uno R3, Leonardo, Mega R3, Due, and any other similar microcontroller.

NRF Interface

The *NRF-Shield* is designed to work with the common NRF24L01+ standard and mini sizes, as well as the NRF24L01+PA+LNA. Footprints for all three are built into the design. Note that the PCB underneath the antenna is clear of traces and copper planes to reduce any undesirable impact. The antenna should be as far away from other components as possible, and hence this board is intended to be the top board on a stack to provide as much clearance as possible.



The user has several options as to how the pins of the microcontroller are connected to the NRF. Using male headers and 2-pin jumper connectors (or simply soldering across the respective holes) the user can make the following selections:

- IRQ Pin on NRF → D2 or D3 on microcontroller
- CE Pin on NRF → D7 or D8 on microcontroller
- CSN Pin on NRF → D9 or D10 on microcontroller (see next section)
- VCC Pin on NRF → 3.3V from microcontroller or Vout from onboard voltage regulator.

These selections allow for more flexibility in the corresponding sketch. Note that the user must define whichever pins are selected in the sketch. If the Vout option is selected, the user needs to solder the respective voltage regulator and input/output smoothing capacitors on the footprints provided. The input Vin to the voltage regulator is connected to the Vin of the microcontroller.

I2C & SPI Interfaces

The *NRF-Shield* includes I2C and SPI interfaces. To the bottom-right of the board are three I2C interfaces, including SDA and SCL pull-up resistor footprints. Note that the logic level voltage of the I2C depends on the microcontroller. In particular for the Uno R3, Mega R3 and Leonardo the logic level voltage is 5V, whereas for the Due it is 3.3V. Only I2C devices with the matching logic level voltage will work. Otherwise a logic level shifter is required. To the bottom-left of the board is an SPI interface. The CS pin (AKA SS pin) can be configured to be either D9 or D10 on the microcontroller.

