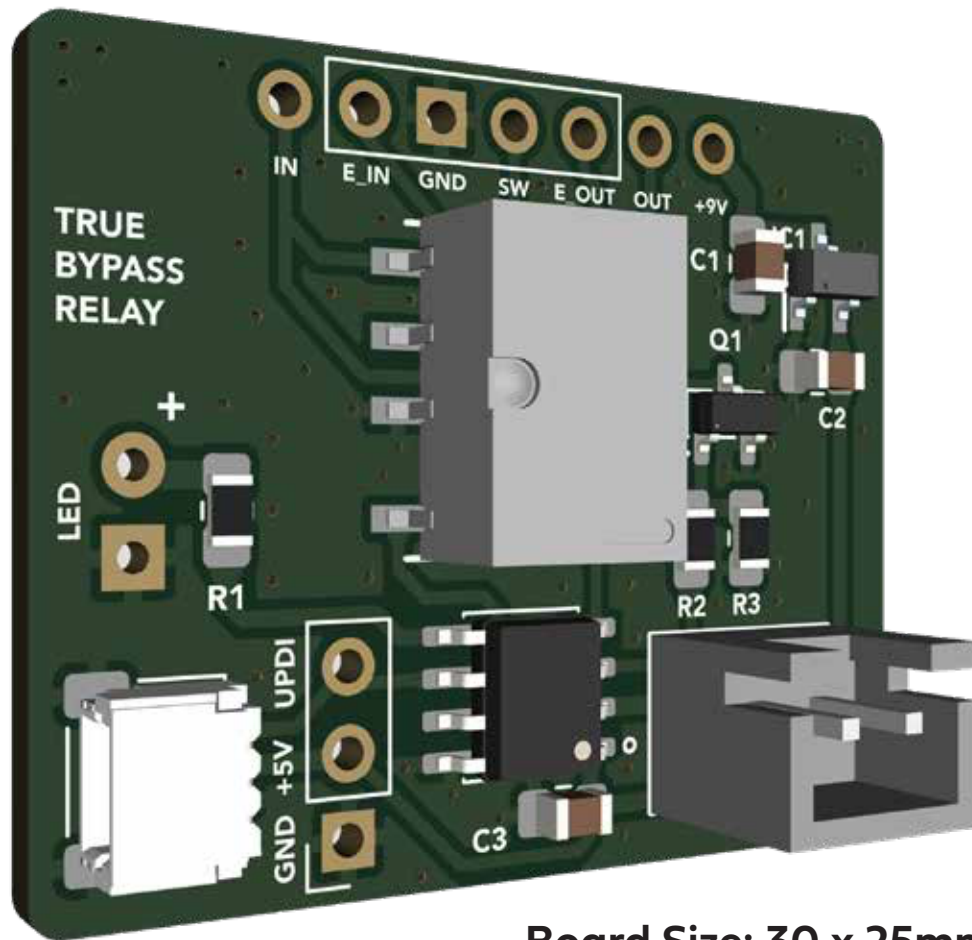


# OVERVIEW



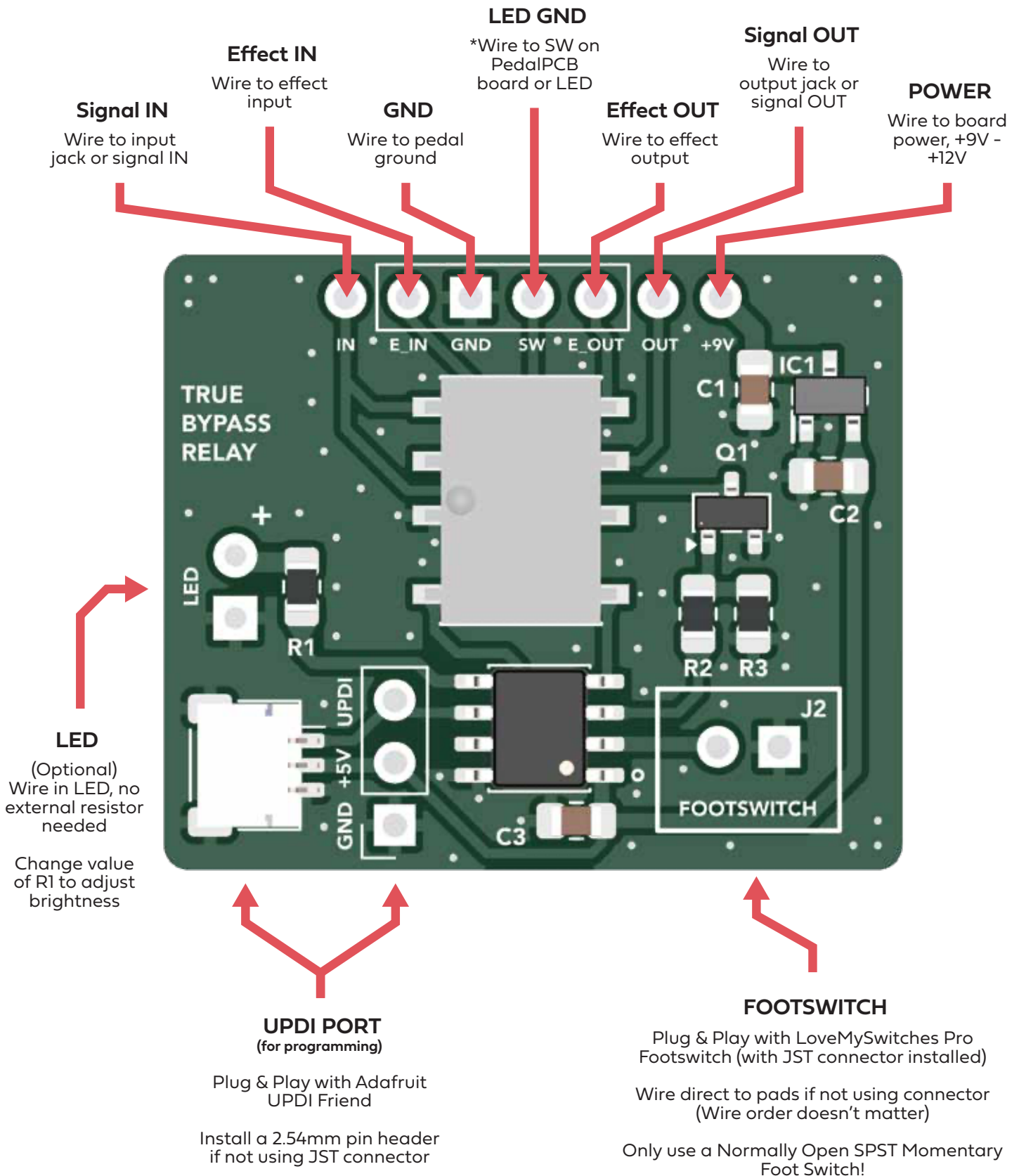
Board Size: 30 x 25mm

## COMPONENTS:

- C1 - 1 $\mu$ F
- C2 - 2.2 $\mu$ F
- C3 - 100n
- LDO (IC1) - AP2210N
- Footswitch Connector (J2) - B2B-XH-A-GU (Optional)
- UPDI Connector (J4) - SM03B-SRSS-TB\_LF\_\_SN\_ (Optional)
- Relay (K1) - G6KU-2F-Y-TR\_DC5
- MOSFET (Q1) - 2N7002
- R1, R2 - 220R
- R3 - 100K
- MCU (U1) - ATtiny412-SSF
- Footswitch - SPST Momentary Normally Open
- Optional - 5mm or 3mm LED
- Optional - 2.54mm header pins

See BOM for full component details

# WIRING DIAGRAM

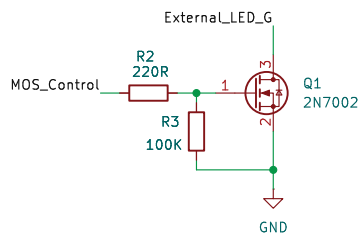
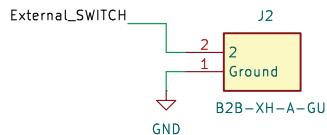
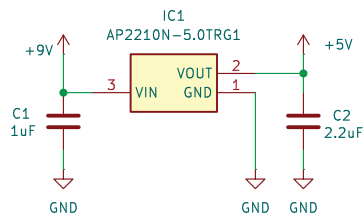


**LED GND** \*Main PCB must have a resistor between LED and ground. Connecting direct to ground will burn out LED.

# SCHEMATIC

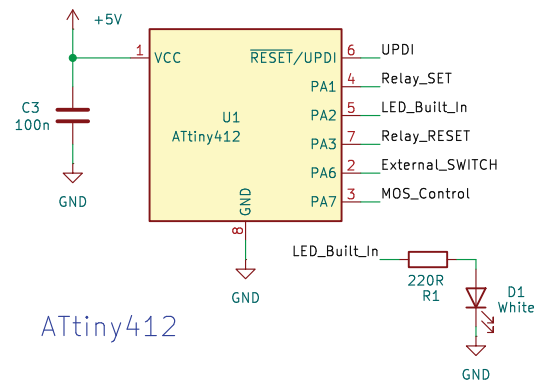
V1.0 2/9/2026

## Power

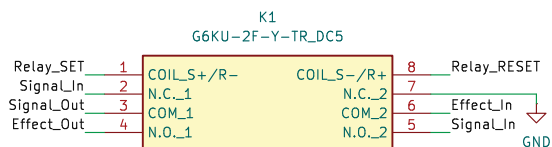


## Pedal LED Control

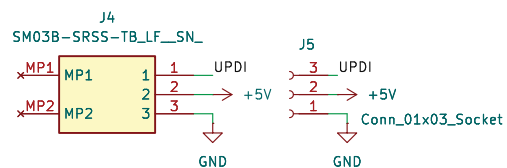
## PedalPCB Interface



## ATtiny412



## Relay



## UPDI Programming