

~~shift down 2 bit after~~

## PCB Footprints

✓ Footprint → 6x Button 5x ✓ 10k pull down

✓ Footprint → 2x LED 2x 220Ω Res

✓ PINS → GND + Power IN (Pins) From Charge Con. or sockets?

✓ PINS → Light Sensor IN/OUT (Pins or Sockets?)  
GND, 3.3, SCL, SDA

✓ Footprint → OLED → 4 Pin ~~footprint~~ (Pins or Sockets)  
GND, 3.3, SCL, SDA

✓ Footprint → BME → same as OLED  
GND, 3.3, SCL, SDA

✓ PINS → PH sensor → 3 Pins  
GND, 3.3, AI

✓ PINS → Moisture Sensor → 3 Pins  
GND, 3.3, AI

Resistor Footprint: Resistor-THT  
R-Axial-DIN0204-L3,6mm-  
D1,6mm-P5,08mm-Horizontal

Button Footprint:

Button-Switch-THT SW-PUSH-6mm-H5mm (6mm short dist between feet, 5mm tall feet)

LED Footprint:

LED-THT LED-D3,0mm (3mm between feet)

Pins on board:

Connector-Pin Header-2.54mm

Pin Header-1x(##)-P2.54mm-Horizontal  
# of pins

Ask Brian about

P2 Footprint

Sockets on board:  
same principal as headers