

Lab 05: Prepare Schematic for PCB Layout

MedTech Prototyping Skills (BME290L)

2023-03-23

1 Update Board to v2.1.1

- Upgrade circuit schematic to v2.1.1 (<https://mlp6.pages.oit.duke.edu/MedTech-Prototyping-Skills/Prototyping-S23-one-shot-blinking-light-v2.1.0.pdf>)¹
- Note the CHANGELOG on the schematic.

```
# CHANGELOG
## v2.0.0
* Change astable to fixed 50% DC
* Adjust RC values for both timers
* Add annotations & pinouts / desc
* Add NetTie (astable)

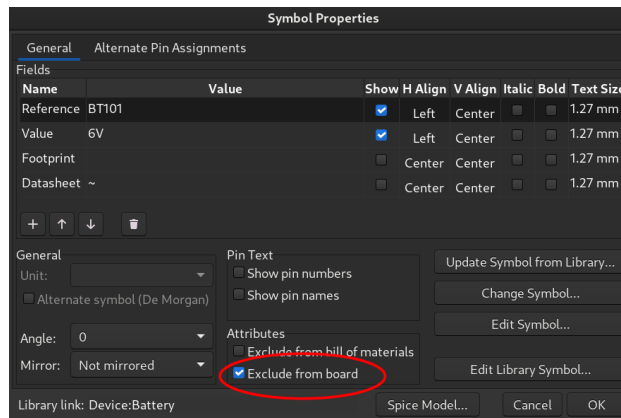
## v2.0.1
* Fix oneshot eqn

## v2.1.0
* Add BT connector
* Consolidate connectors/peripherals
* Change to male connectors
* Change AND gate to quad
* Assign footprints
* Assign all parts

## v2.1.1
* Fix R footprints
```

¹v2.1.1 and v2.1.0 have the same rendered schematic.

2 Exclude Peripheral Parts from PCB



3 Assign Footprints

- Tools -> Assign Footprints...
- Can also assign footprints directly to each part (Properties)

Symbol	Footprint Assignments
1	C101 - 10u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
2	C102 - 10u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
3	C201 - 100n : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
4	C202 - 0.01u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
5	C203 - 0.01u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
6	C204 - 10u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
7	C205 - 1u : Capacitor_THT:CP_Radial_D10.0mm_P5.00mm
8	D101 - RevPol_PwrStatus : LED_THT:LED_D5.0mm
9	H101 - MountingHole : MountingHole:MountingHole_3.2mm_M3
10	H102 - MountingHole : MountingHole:MountingHole_3.2mm_M3
11	H103 - MountingHole : MountingHole:MountingHole_3.2mm_M3
12	H104 - MountingHole : MountingHole:MountingHole_3.2mm_M3
13	J101 - Push Button : Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical
14	J102 - LED : Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical
15	J103 - Power Switch : Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical
16	J104 - BATT : Connector_PinHeader_2.54mm:PinHeader_1x02_P2.54mm_Vertical
17	NT1 - To_Button : NetTie:NetTie-2_THT_Pad1.0mm
18	R101 - 1k : Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P12.70mm_Horizontal
19	R200 - 455k : Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P12.70mm_Horizontal
20	R202 - 42.2k : Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P12.70mm_Horizontal
21	R203 - 10k : Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P12.70mm_Horizontal
22	R204 - 100k : Resistor_THT:R_Axial_DIN0309_L9.0mm_D3.2mm_P12.70mm_Horizontal
23	TP101 - VCC : TestPoint:TestPoint_Pad_D4.0mm
24	TP102 - AstableOut : TestPoint:TestPoint_Pad_D4.0mm
25	TP103 - MonoOut : TestPoint:TestPoint_Pad_D4.0mm
26	TP104 - LightOut : TestPoint:TestPoint_Pad_D4.0mm
27	TP105 - GND : TestPoint:TestPoint_Pad_D4.0mm
28	U101 - LM7805_T0220 : Package_TO_SOT_THT:TO-220-3_Vertical
29	U102 - 4081 : Package_DIP:DIP-14_W7.62mm_Socket_LongPads
30	U201 - LM556 : Package_DIP:DIP-14_W7.62mm_Socket_LongPads

4 Augment Functionality of Your Circuit

- Modify your circuit to use potentiometers to adjust the LED on-time duration and the blink rate.
- Determine the range of your functional modulation with those potentiometers.
- Choose your own adventure:
 1. Change the output stage of the circuit toggle 2 LEDs out of phase with one another, or
 2. Up the ante and come up with a more creative / complex output scheme. Complexity will be rewarded...but not at the expense of functionality (from a grading perspective).

5 Generate Bill of Materials

- Tools -> Generate BOM... (include footprints)

6 What to Submit

A single PDF containing:

- The sheets of your augmented schematic, versioned at v3.x.x.
- Rendered BOM with footprint descriptions.
- Your analysis of the range of functional modulation of monostable duration and astable frequency using your potentiometers.