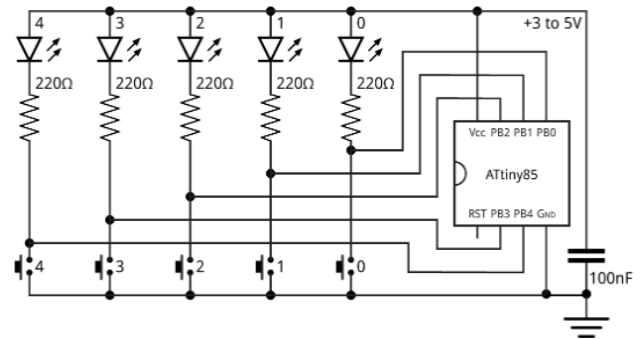


ICS4C - KiCAD Final Assessment

You are building a small LED & pushbutton puzzle board. Your main goal is to make a small board that can be used for a puzzle game. Do not concern yourself with the game or how it works, simply recreate the circuit and design the board.

Requirements:

- Use the schematic below to create the design
- Use the parts list below for exact parts and footprints
- The buttons should be laid out underneath the LEDs
- The battery holder should be located on the back of the board
- **The entire board should fit inside a 65mm x 26mm rectangle**



Parts List

- 5 LEDs
 - KiCAD Component: *LED*
 - KiCAD Footprint: *LED_THT:LED_D5.0mm*
- 5 Resistors
 - KiCAD Component: *R*
 - KiCAD Footprint: *Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal*
- 5 Buttons
 - KiCAD Component: *SW_Push*
 - KiCAD Footprint: *Button_Switch_THT:SW_PUSH_6mm*
- 1 Capacitor
 - KiCAD Component: *C*
 - KiCAD Footprint: *Capacitor_THT:C_Disc_D5.0mm_W2.5mm_P2.50mm*
- 1 Battery Holder
 - KiCAD Component: *Battery*
 - KiCAD Footprint: *Battery:BatteryHolder_Keystone_2468_2xAAA*
- 1 Microcontroller
 - KiCAD Component: *ATtiny85-20P*
 - KiCAD Footprint: *Package_DIP:DIP-8_W7.62mm*