PCB Badge Challenge

Christmas PCB Documentation

Guilherme Gomes, 2020

Contents

1 Initial Project

The project idea is to make a PCB Badge, a Christmas ornament. Its implementation is made with a Johnson decade counter IC, the CD4017, to create a sequencer. The clock circuitry was made by transistors, creating an Astable Multivibrator.

2 Bill of Materials (BOM)

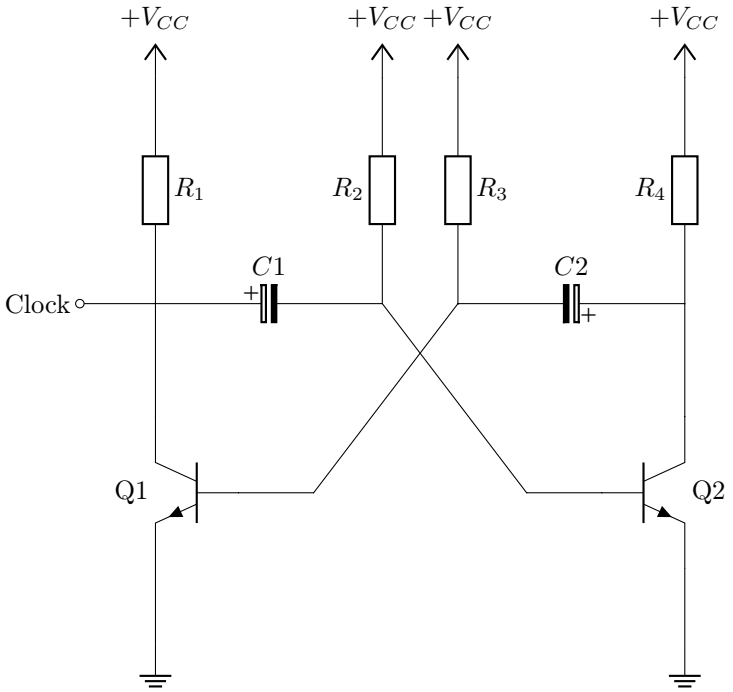
Suggestion: buy the components first and check the project for the right footprints.

Reference	Quantity	Value	Package
BT1	1	CR2016/CR2025/CR2032	Coin cell
BT	1	Battery Holder	-
C1	1	10uF	Electrolytic SMD
C2	1	47uF	Electrolytic SMD
D1 - D10	10	Red LED	SMD 0805
D11 - D15	5	White LED	SMD 0805
Q1, Q2	2	BC847	SOT-23
R1, R4	2	330R	SMD 0805
R2	1	47k	SMD 0805
R3	1	22k	SMD 0805
R5 - R19	15	100R	SMD 0805
SW1	1	DPDT Switch	8x8mm toggle switch
U1	1	CD4017	SO-16

3 Calculations and how it works

The CD4017 IC needs a clock pulse to increment the counter, which was generated by the Astable Multivibrator. This circuit was chosen rather than other simple oscillators like the 555 IC or CD4093 IC, because of the possibility of choosing the hold time of the clock pulse without using a microcontroller.

3.1 Astable Multivibrator



The conduction time of each transistor is controller by the value of the capacitors and resistors. The equation to calculate this is given by

$$t_c = 0,69 \cdot R \cdot C$$

where R is the resistance in ohms (Ω) and F is the capacitance in Farads (F). The components that controls the transistors are R_2, R_3, C_1 and C_2. The control is made in a cross-way, with means that the combination of R_2 and C_1 controls the timeout of the circuit, since they are connected to the base of Q_2, while R_3 and C_2 controls the clock pulse, controlling the polarization of Q_1. R_1 e R_4 are resistors to control the current flow of transistor. To the PCB, the following values were used:

R_1 e $R_4 = 330\Omega$
$R_2 = 47k\Omega$
$R_3 = 22k\Omega$
$C_1 = 10\mu F$
$C_2 = 47\mu F$

Doing the math, we have:

$$t_{timout} = 0,69 \cdot 47 \times 10^3 \cdot 10 \times 10^{-6} = 0,3243 = 324,3\text{ms}$$

$$t_{clock} = 0,69 \cdot 22 \times 10^3 \cdot 47 \times 10^{-6} = 0,71346 = 713,46\text{ms}$$

This clock pulse is sent to the counter.

3.2 Sequential circuitry

Every rising edge of the clock pulse increments the 4017 output. When turned on, the Q0 output is set to a high logic level, while the others are set to low. At the first clock pulse, Q0 is set to low, and Q1 goes to high. And that continues at every clock pulse, as described on the component datasheet:

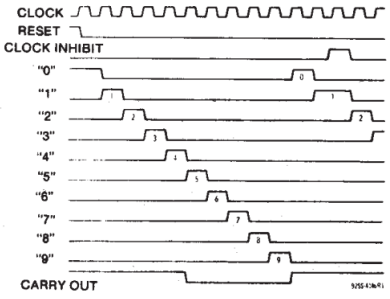
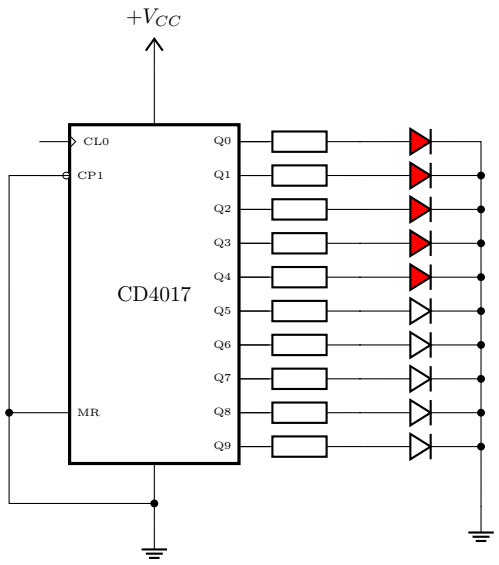


Fig. 2 – Timing diagram for CD4017B.

The LEDs are connected to the counter output. In the final circuit, every red LED shown in the picture below are two red LEDs in parallel.



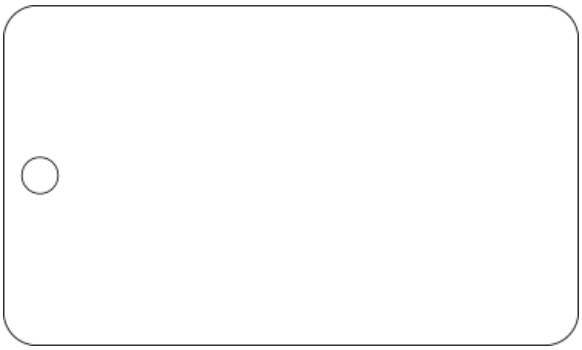
4 Designing the PCB

To make the board, the chosen option is SMD components with 0805 package, to keep the circuit small, yet with a comfortable size to be hand-soldered. The initial design was made using *Inkscape*, to draw **.svg** files, that are exported as a **.dxf** file, natively recognized by the *Kicad* software.

4.1 Initial design

After the ideas, given the size limits (100 x 100 mm), the design below was considered to be the board edge, with the following design to be made on the silk screen layer.

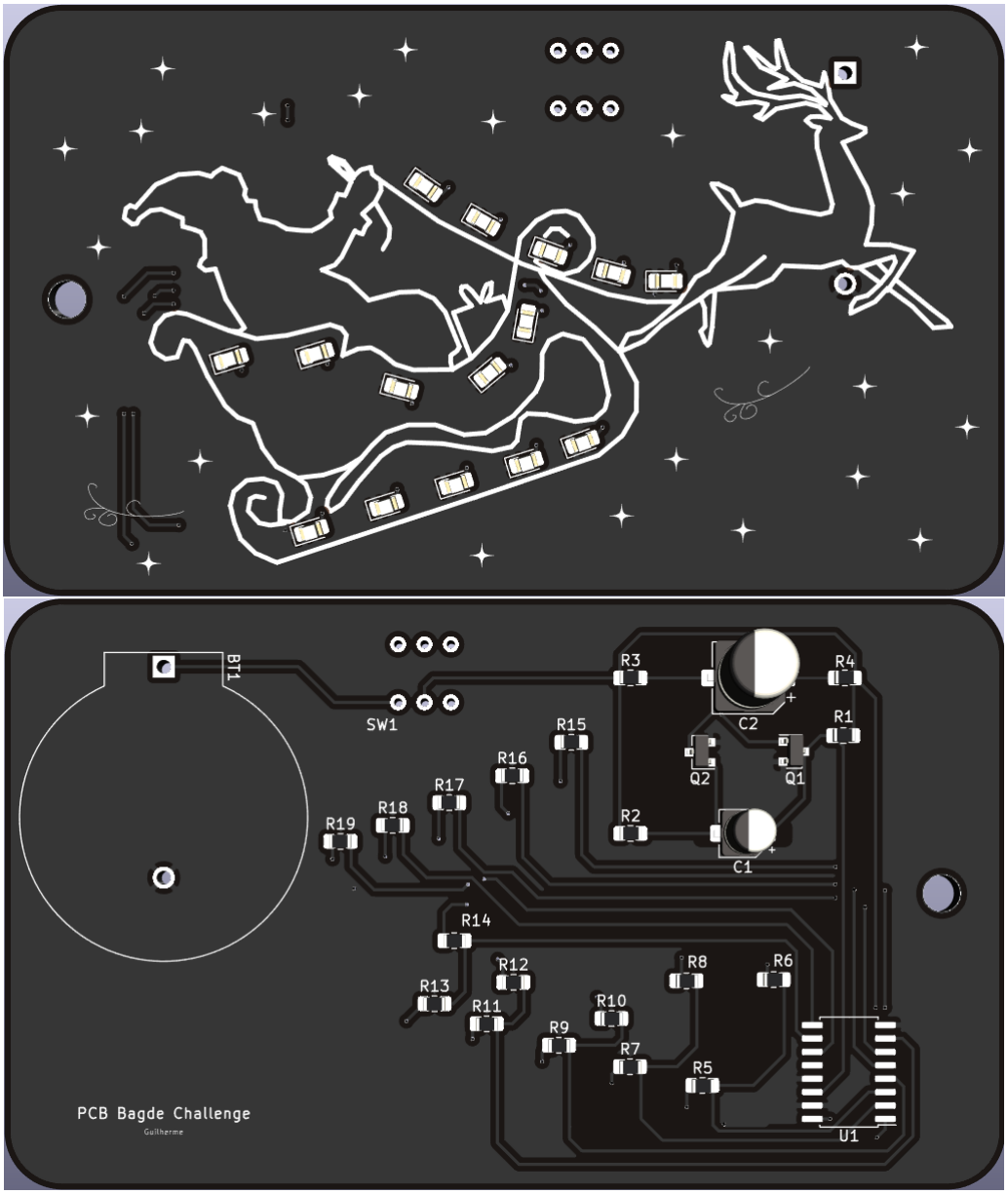
While trying to design the PCB, the size has proven to be too small, and the board edge design was changed to be a Christmas tree ornament, or to be used as a backpack tag, with a new size of about 95 x 55 mm.



4.2 Final design and layers

Since the intention is to be a pretty circuit, it was filled with more ornaments, like the stars and the snow to give movement to the Santa and his sleigh. To keep the board front layer as clean as possible, all routing was made on the backside, with the front copper layer only getting small jumper trails. The solder mask will be black, to represent the night sky, while the Santa figure and the stars are on the pad layer, to receive a shining silver finish. The snow is at the silk screen layer, since it going to be white, and contrasts with the black background. The hole on the board was made with a bigger via, with 3.8 mm of diameter with a 3.5 mm hole. Via is the choice instead of a simple board edge cut because of the internal copper finish that the via receives.

After all the changes, this is a 3D visualization of the board:


BT1
SW1
R3
R4
C2
R1
R15
Q2
Q1
R16
R17
R18
R19
R2
C1
R14
R12
R8
R6
R13
R11
R10
R9
R7
R5
U1
PCB Bagde Challenge
Guilherme