**Recommendations:**
- The connection of the battery charger U2 is incorrect. With the current configuration, the Vbus voltage from the USB (+5V_USB) is being routed to both the input and the output of the charger, which is not correct. Essentially, the input and output of the charger are short-circuited. The Vbus voltage from the USB should only be connected to the input of the charger and not to its output. The circuit should be powered from the output of the charger U1. Therefore, changes need to be made to the wiring of the Voltage Regulator U2 so that it is powered by the output of the charger. The correct connection of the charger is shown in the diagram below:

Typical Application Circuit

MCP73871 Typical Application
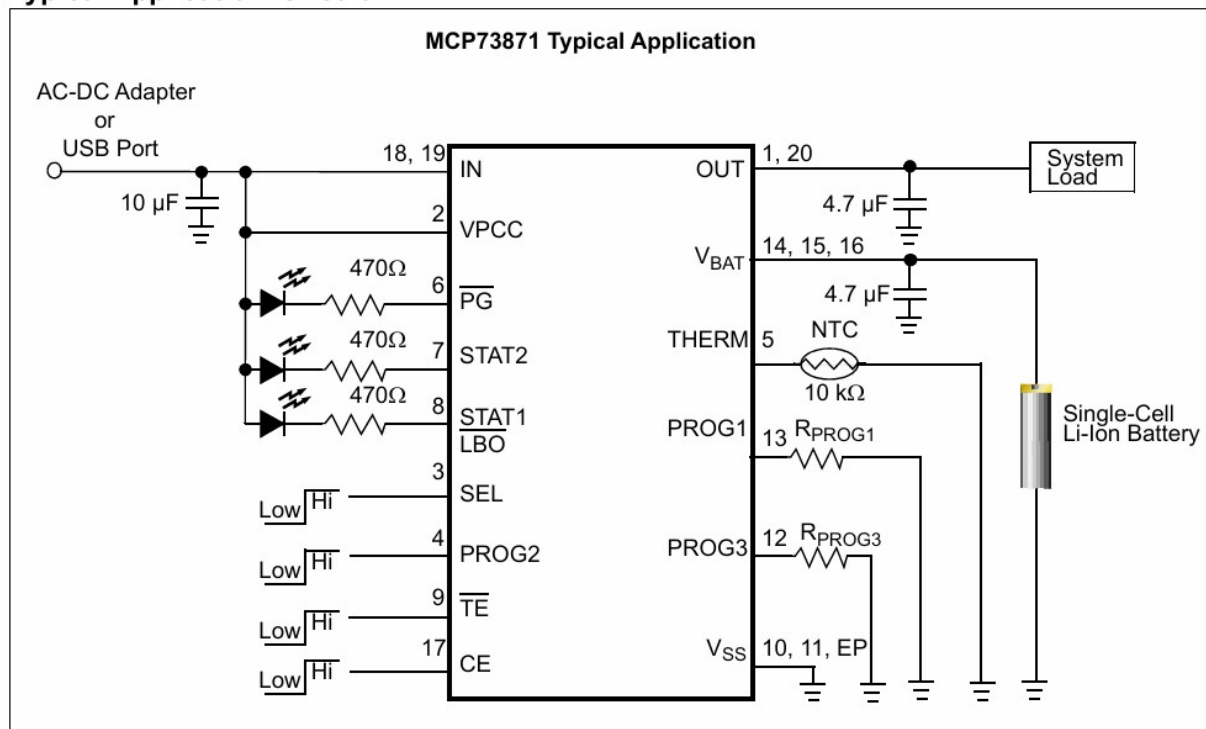
- It is recommended to add 100nF capacitors in parallel with the 10uF capacitors at the input and output of the Voltage Regulator.
- Node VBUS at page2, is not connected to the VBUS node in Root Page1. Global label is missing at page1.
- At the current design the USB signal lines are protected from electrostatic discharge by diodes D6 and D7. However, the capacitance of these diodes is very high, up to 200 pF max, and they are not recommended for use in high-speed communication signals like USB-C.
- The capacitance of capacitor C9 is too high at 100uF. Additionally, parallel capacitors C4, C5, and C6 increase the total capacitance to 119.4uF. This capacitance is very large and not recommended for devices powered by USB, as it can cause a significant inrush current, potentially damaging the USB. The maximum recommended capacitance on a USB-B bus is 10uF. Therefore, it is advised to remove capacitors C9, C4, C5, and C6.
- Capacitor C3 (100uF) can be removed.
- It is recommended to remove resistors R10 and R2. The CE and Prog2 inputs of the charger can be connected directly to the +5V_USB voltage.
- It is recommended to remove resistor R9. The SEL input of the charger can be connected directly to GND.
- In the current design, diode D2 is connected in reverse. As a result, the circuit cannot function because, due to the reversed orientation, the current from VBUS will not be able to pass through the diode to the rest of the circuit. However, using diode D2 for reverse current protection is unnecessary since the charger already prevents reverse current flow from the battery to the USB.

- It is recommended to supply power to the USB bridge with the VBUS voltage after the fuse, not before.

- It is recommended to reduce the values of resistors R4, R5, R6, R19, and R20 to increase the current through the LEDs to approximately 10mA.

- It is recommended to replace C22 with a 1uF capacitor instead of 0.1uF.

- The USB bridge U4 is not connected correctly. With the current configuration, the output of the integrated voltage regulator of the USB bridge (VDD) is connected to the output of the Voltage Regulator U2. Since we are using the external Voltage Regulator U2 in the circuit, the use of the internal voltage regulator of the USB bridge U4 is not necessary. Therefore, according to the datasheet, the circuit must be modified as shown below:

The figure below shows a typical connection diagram for the power pins of the CP2102N devices when the internal 5 V-to-3.3 V regulator is not used.
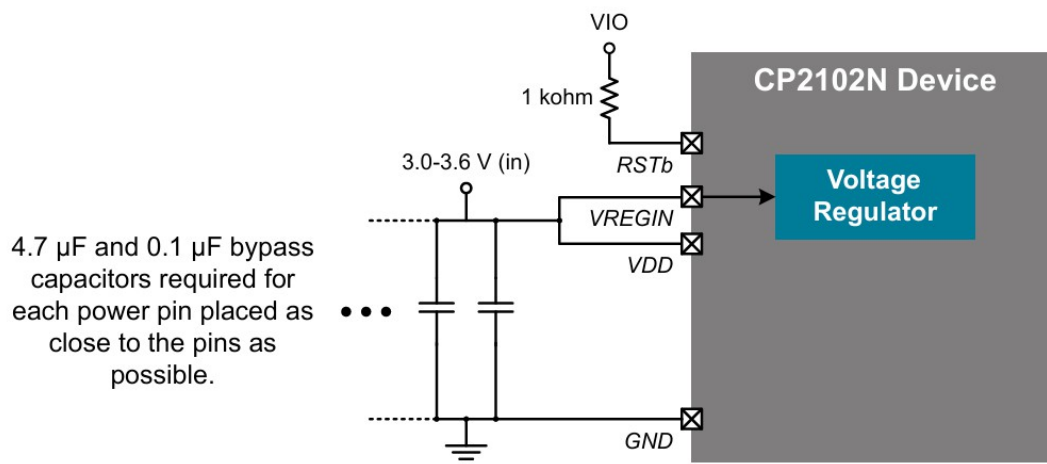
As a result, capacitor C10 can be removed. .

- It is recommended to place a 1uF capacitor at the Vbus input of the USB bridge U4.

- It is recommended to connect the RST of U4 to VDD (+3.3V) instead of VBUS.

- It is recommended to remove resistors R15 and R16.

- According to the current design, the BOOT signal is connected to GPIO0 of the ESP32, but this is a design mistake. The BOOT signal should be connected to GPIO9 because GPIO2, GPIO8, and GPIO9 are the strapping pins and control the boot mode after the reset is released.

- Resistor R36 can be removed because GPIO9 has an internal pull-up resistor.

- It is recommended to add a 100nF capacitor in parallel with C8.

- It is recommended to remove C16. .

- VCC pin of U7 (MAX4466EXK) is connected to 3.3V through a resistor R33 2K2. R33 must be removed and the VCC pin should connected directly to 3.3V

- The connection of the Microphone Preamplifier U7 (MAX4466EXK) is incorrect. The following wiring, as recommended by the manufacturer, is suggested for operation with an ELECTRET CONDENSER MICROPHONE.
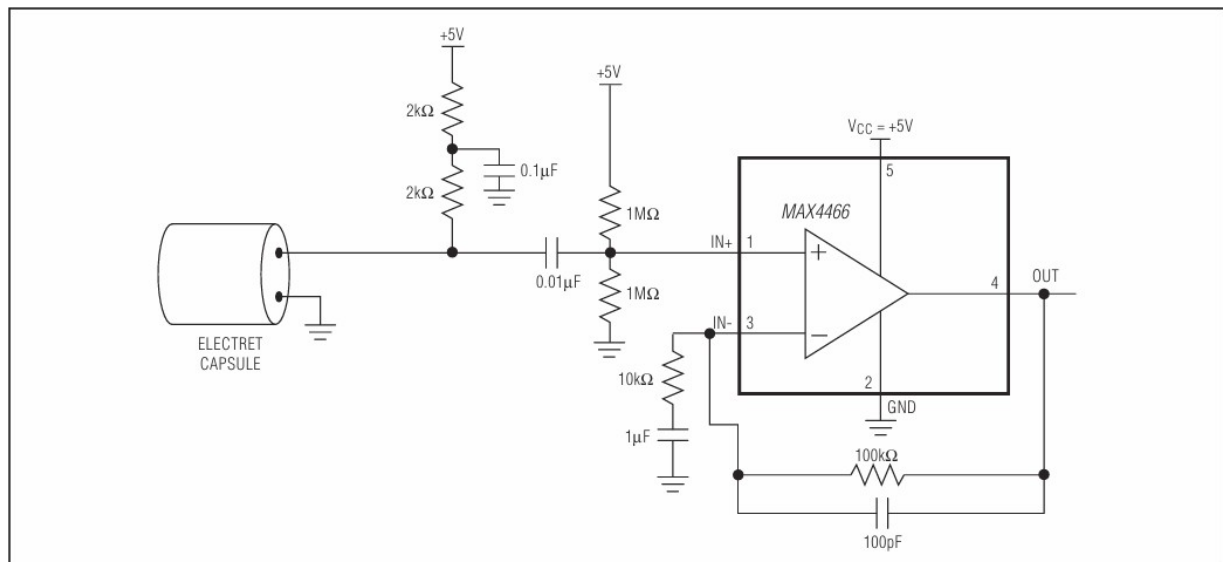
Figure 2. MAX4466 Typical Application Circuit

- It is recommended to remove resistor R22. The CS pin of the Flash memory is only a chip select signal and is not a high-frequency signal bus.
- It is recommended to remove resistor R29.

- It is recommended to add a pull-up resistor to the CD pin of the SD card. If the internal pull-up of the ESP32 is enabled, then it is not necessary. .

- It is recommended to place ESD diodes for protection of the SD card signals, This modification will be implemented only if requested by you.

Modifications:
1. Correction of the U1 connection.
2. Correction of the U2 connection.
3. Addition of 100nF capacitors in parallel with the 10uF capacitors at the input and output of the voltage regulator.
4. Addition of a global label VBUS at page 1 at node VBUS.
5. Replacement of diodes D6 and D7 with the USB protection diode USBLC6-2SC6, which has protection
for both USB bus lines and the VBUS voltage.
6. Removal of diode D1 (PESD5V0S1UL_315) since the USBLC6-2SC6 diode already provides protection for VBUS.
7. Removal of capacitors C9, C4, C5, and C6.
8. Removal of capacitor C3 (100uF).
9. Removal of resistors R10 and R2. Connect the CE and Prog2 directly to the +5V_USB voltage.
10. Removal of resistor R9. Connect the SEL pin directly to GND.
11. Removal of diode D9.
12. Powering the USB bridge after the fuse.
13. Replacement of C22 with 1uF instead of 0.1uF to create a delay in the start of the EPS32.
14. Set the value of the USB bridge IC with part number: CP2102N-A02-GQFN20.
15. Correction of the U4 connection, with the 3.3V voltage from the output of the voltage regulator U2 applied to the VREGIN and VDD pins of the USB bridge U4.
16. Removal of capacitor C10.
17. Placing a 1uF capacitor at the VBUS input of the USB bridge U4.
18. Connecting the RST pin of U4 to VDD (+3.3V) instead of VBUS.
19. Removal of resistors R15 and R16.
20. Connecting the BOOT signal to GPIO9 and the SC_CD signal to GPIO0 pin.
21. Removal of R36.
22. Adding a 100nF capacitor in parallel with C8.

23. Removal of C16.
24. Removal of R33. Connection of VCC directly to 3.3V.
25. Redesign of the pre-amplifier U4.
26. Removal of resistor R22.
27. Removal of resistor R29.
28. Adding a pull-up resistor to the CD pin of the SD card.