

USB Charger PCB

Overview

This project focuses on designing and building a USB Charger PCB. The primary function of this PCB is to provide a regulated 5V DC power supply to charge USB devices such as smartphones, tablets, and other portable electronics.

How it Works

The design takes 120V AC power from a standard wall outlet and steps it down to a safe level using a transformer. The AC voltage is then rectified into DC using a bridge rectifier. Filter capacitors smooth out the rectified DC voltage, reducing ripple. Finally, a voltage regulator provides a stable 5V DC output suitable for charging USB devices.

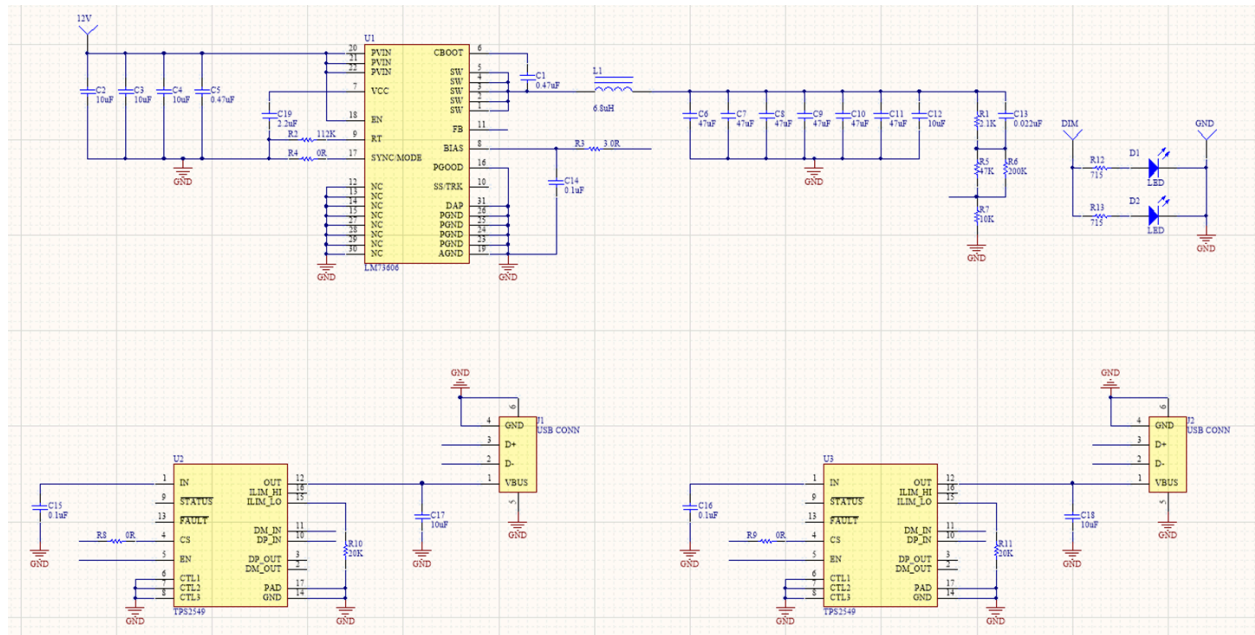
Objectives

- Design PCB using KiCAD
 - Begin with schematics
 - Assign Footprints
 - Run and pass ERC
 - Complete layout (PCB)
 - Run and pass DRC
 - Generate Gerber files
 - Generate BOM
- Submit PCB to Oshpark for fabrication
- Order parts from DigiKey
- Assemble PCB
- Test PCB
- Profit

Specifications

1. 12 V input
2. 15 W maximum power output at 5 V (3 A maximum current) per port

Schematics



The schematic should include:

- An AC-DC converter (likely using a bridge rectifier and filter capacitors) to convert the 120V AC input to DC.
- A switching regulator (e.g., based on the LM73606 and TPS2549 ICs) to step down and regulate the DC voltage to 5V.
- An inductor (L1) and capacitors (C1-C19) to form the switching regulator's energy storage and filtering components.
- USB A connectors (J1, J2) for the output.
- LED indicators (D1, D2) to show power status.
- Resistors (R1-R13) for various functions (e.g., current limiting, feedback).

Components

- Capacitors (various values) - 19 total
- LEDs (BLUE CLEAR SMD R/A) - 2
- USB Connectors - 2
- Inductor (Iron-core) - 1
- Resistors (various values and types) - 13 total
- LM73606 switching regulator IC - 1
- TPS2549 IC (likely a synchronous rectifier driver) - 2

Assembly Tips

- Start by soldering the smallest components (diodes, resistors) first.
- Be careful with the polarity of polarized components (diodes, electrolytic capacitors).
- Ensure the voltage regulator is properly mounted to the heatsink.
- Double-check all connections against the schematic before powering on the circuit.
- Use heat shrink tubing to insulate exposed wires and connections.
- Test the circuit with a multimeter before connecting any USB devices.

Resources

1. [Building a USB Charger Circuit](#)
2. [USB Charger Project in Altium Designer: Part 1](#)
3. [USB Charger Project in Altium Designer: Part 2](#)
4. [USB Charger Module](#)