Homework 6: Printed Circuit Board Layout Design Narrative

Team Code Name: Every1 DJ **Group No.** 2

Team Member Completing This Homework: Travis Jefferson

E-mail Address of Team Member: jefferst@ purdue.edu

NOTE: This is the third in a series of four "design component" homework assignments, each of which is to be completed by one team member. The body of the report should be 3-5 pages, **not** including this cover sheet, references, attachments or appendices.

Evaluation:

SEC	DESCRIPTION	MAX	SCORE
1.0	Introduction	10	
2.0	PCB Layout Design Considerations - Overall	20	
3.0	PCB Layout Design Considerations - Microcontroller	15	
4.0	PCB Layout Design Considerations - Power Supply	15	
5.0	Summary	10	
6.0	List of References	10	
Attach	PCB Layout Design Files (include in .zip archive)	20	
	TOTAL	100	

Comments:

Comments from the grader will be inserted here.

1.0 Introduction

The proposed design is an interactive DJ party system with an accompanied web application for users to add and vote on songs to be played. The main control unit of the Every1 DJ system is a custom printed circuit board (PCB). The current circuit board design includes power supply and voltage regulation circuitry, audio circuitry, motor circuitry, LED driver circuitry, decoupling capacitors and connections to other major components in the design. These components include: Raspberry Pi, microcontroller, LED's, LCD and push buttons. Some of the major PCB layout issues include correctly routing the board to avoid overlapping traces, placing parts to minimize trace conflicts and keeping like parts within close proximity. A few more design challenges become apparent at the design level: minimize/eliminate noisy signal proximity, supply adequate operational power to a dynamic load, and eliminate any interference from analog audio circuitry.

Updated PSSC's:

1. An ability for a user-created web application to communicate a selected song to the Raspberry Pi through an http request (API poll).
2. An ability for a computing PC (Raspberry Pi) to stream music through the Grooveshark API and communicate the song information with the microcontroller.
3. An ability to create a 'lights' show corresponding the music beat, which will be sampled from the auxiliary out of the Raspberry Pi via the microcontroller ADC.
4. An ability to control the direction and rotation speed of the LED package through a motor controlled via the micro PWM peripheral.
5. An ability to utilize an LCD to display user-selected (via a control device) metadata (e.g. audio options, light settings, and track information)

2.0 PCB Layout Design Considerations - Overall

The standard PCB design limits are outlined from Advanced Circuits in Table 1, with a comparison to the actual trace and drill sizes used in the Every1 DJ PCB design [1]. The PCB designed features larger trace and drill sizes to allow for optimal manufacturing.

Metric	Minimum Size Required	Actual Size Used
Trace Thickness	5 mil	10 mil (minimum)
Trace Spacing	5 mil	14 mil (minimum)
Drill Diameter	10 mil	23 mil
Drill Tolerance	5 mil	20 mil (approximate)

Table 1: PCB Specifications [1]

Another consideration for the PCB design includes the minimization of trace angles at or greater than 125 degrees. Any angles less than 125 degrees could lead to the creation of acid pits during manufacturing. Careful consideration went into part placement before routing the PCB traces. This is a necessary step to maximally facilitate trace routing. In the cases where overlapping wires could not be resolved, the traces were routed to the bottom of the board through a via. Certain location constraints were required, such as edge-mounted connectors like the power supply and RJ-11 programming port. Physical access was also a criterion in placing some components, especially connectors to package-mounted devices like the LCD or pushbuttons. Components with similar or related functionality were grouped near one another. Due to this grouping, higher frequency signals like I^2C and SPI clocks naturally did not route within close proximity of one another. Minimizing the interactions of noisy signals like clocks is an additional design goal for the Every1 DJ PCB layout.

3.0 PCB Layout Design Considerations - Microcontroller

The two most impactful decisions on the PCB layout were that of microcontroller orientation and its respective pin assignments to peripheral components. Once an initial orientation was chosen, peripheral component clusters were arranged in a manner which allowed for short and simple traces to be drawn to each group. At that point, the schematic was modified to select the most convenient GPIO pins for each peripheral. These decisions became especially important when more complicated signals like power and ground were routed and attached. A heartbeat LED was included for some simple debug feedback.

In order to sufficiently supply power to the microcontroller, maximally sized power and ground planes were designed and four decoupling capacitors were selected and routed. The 0.1μF decoupling capacitors were placed directly underneath the microcontroller (within the 250 mils) to satisfy the manufacturer's specifications[2].

Programming clock and data pins were traced out to an RJ-11 connector to allow for programming and in-circuit debugging after fabrication. In order to complete the programming interface, the MCLR pin was connected to an additional 0.1μF decoupling capacitor, a pull-up resistor (10kΩ), and a series resistor (250Ω)[2].

A simple filtering circuit was included for audio input from the Raspberry Pi. The filtered, mono audio output is then traced to an ADC peripheral input pin on the microcontroller. Due to the analog nature of this signal, a separate ground signal (AGND) was maintained for the duration of the analog circuitry and connected to the digital ground plane at a single location.

An external oscillator was not included in the design due to minimal clocking restrictions; all the communication protocols are synchronous, timing insensitive signals and sampling of the Raspberry Pi audio for simple beat detection is non-critical and operates at a relatively low frequency.

4.0 PCB Layout Design Considerations - Power Supply

The power supply portion of the PCB was designed to withstand a medium sized, dynamic load and a few smaller, lower-power loads. Current spikes of approximately 2A are expected during heavy motor/LED usage, and stable supply voltages area necessity for lower-power devices like the microcontroller and other digital circuitry. To meet these demands, power and ground planes were maximally sized. Additionally, the power and ground traces were kept to 50

mils in all locations save for those restricted by device pinout and package dimensions.

Since it is anticipated that demand for power will not exceed that which is supplied by the power supply, bulk capacitors are not included in the product design. All bypass capacitors were placed within manufacturers' specifications. A voltage regulator is included to convert 12V down to 5V for the Raspberry Pi. 5V circuitry was placed near this regulator to minimize trace length for these components. Similarly, an LDO was included to provide 3.3V power to the remaining digital circuitry, which was grouped as closely as possible to minimize routing complexity. The power connector for the device was placed near the regulators to meet this same design criterion.

5.0 Summary

The Every1 DJ provides a webapp-controlled DJ and lighting system to partygoers. To provide power to the visual effects system as well as maintain effective and operational digital components, a custom PCB was developed with wide power and ground traces as well as expansive supply and ground planes. To ensure smooth operation, manufacturer specifications were followed with respect to bypass capacitor values and their locational restraints, component and power/ground trace widths, and (locational) isolation of noisy communication clock signals. Analog and digital components are separated spatially and their respective ground signals meet only in a single location. Trace overlap and via placement were minimized by clustering like components and selecting convenient microcontroller GPIO pins, which can be confirmed upon inspection of the completed PCB layout in Appendix A.

ALL TEAM MEMBERS should read Motorola Application Note AN1259 (posted on course web site) before you begin your PCB layout.

6.0 List of References

[1] "Standard Spec vs. Custom Spec PCB's: What's the Difference?," Advanced Circuits, 2014. [Online]. Available: http://www.4pcb.com/standard-custom-order-pcbs/. [Accessed 28 February 2014].

[2] Microchip, "PIC18F87J11 FAMILY," Microchip, 2012. [Online]. Available: http://ww1.microchip.com/downloads/en/DeviceDoc/39778e.pdf. [Accessed 28 2 2014].

IMPORTANT: Use standard IEEE format for references, and CITE ALL REFERENCES listed in the body of your report. Provide "live" links to all data sheets utilized.