Homework 6: Printed Circuit Board Layout Design Narrative

Team Code Name: BeatSquare **Group No. 01**

Team Member Completing This Homework: Brennan Tran

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

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:

1.0 Introduction

The BeatSquare is a simple approachable audio sequencer based off of popular flash applications like Muzy Music Grid and Tonematrix Audiotool. At a high level the BeatSquare will consist of an 8x8 illuminable grid with a configuration section above containing an LCD and controls. A TI Tiva TM4C123GH6PM microcontroller (referred to hereafter as Tiva or microcontroller) will be utilized to interface with the system.

Due to the complexity of the circuit and the need to fit the system into relatively small packaging, a printed circuit board (PCB) will be designed for the device. With the integration of a PCB, several issues arise. Some major PCB considerations include the separation of different circuits based on signal type, electromagnetic interference (EMI), trace widths and lengths, and some power supply considerations.

Project Specific Success Criteria are as follows:

1. An ability to poll an array of pushbuttons to manipulate the musical configuration and display a visual pattern using LEDs.
2. An ability to adjust the beats per minute.
3. An ability to output multi-tone audio.
4. An ability to save audio output as a MIDI file on an SD card.
5. An ability to save and load musical configurations to and from the flash memory of a microcontroller.

2.0 PCB Layout Design Considerations - Overall

The first and foremost constraint of the PCB layout is the physical geometry. The board must fit into the size of the product's packaging, which entails having a square area less than or equal to 23 cm by 33 cm. Obviously, the maximum allotted size is far too large for this scale of application. Furthermore, it is important that the board itself be placed closest to the majority of peripherals that it will interface with, especially components that will be mounted on the board. In this case, the majority of the peripherals that will be interfaced with the board take up a square area of about 23 cm by 10 cm towards the upper end of the device, and the components that need to be mounted to the board (the SD card connector and the power jack) are located at the back end of the device. Thus, it makes logical sense to place the PCB in this vicinity.

Another consideration about the board's geometry is EMI from the speakers. While these speakers are only rated at 12 watts and are not especially powerful, the PCB's shape was designed to accommodate the speakers without increasing the height of the device by simply leaving space around the speakers. To further reduce EMI emissions in the circuitry, a number of decoupling capacitors were placed as close as possible to the microcontroller and integrated circuits.

Components on the board were placed close to where the peripherals will be located in the device. The SD card connector, power jack, and headers for the reset and power buttons are located towards the upper side of the board. The voltage regulators and power circuitry was also placed close to this area. Shift registers for the LED and button matrices are spread across the lower end of the board, across the "wings" so that each set of headers (as well as current-limiting resistors and the associated shift register) has sufficient space. The audio circuitry was kept apart from the digital logic on the board to further reduce EMI. Finally the microcontroller was placed close to the center of the board so that all traces could be routed from the micro to the peripherals with relative efficiency. Unfortunately, this did not entirely eliminate long traces from the design, so these traces were made larger so as to reduce the resistance in the trace.

Since many of the components used in this project will not be mountable on the PCB, many pin headers were placed on the board in order to interface with them. These components include the LED breakout boards (LEDs and push buttons), the LCD, the rotary pulse generators, the speakers, the menu push buttons, the power and reset inputs, and the programming and debugger connections. These headers are all through-hole components, which add some difficulty in routing the traces around them but are still small enough to fit on the board and warrant their use. The placement of the headers mainly corresponded to the IC controlling them but varied when the input was from the microcontroller itself.

3.0 PCB Layout Design Considerations – Microcontroller

One of the main concerns with the microcontroller is the bypass capacitors. Due to having multiple source pins, multiple bypass capacitors will be needed, as laid out in figure U2-B in the Tiva Launchpad Manual ([1], p22, lower left figure). Note that the VDDC pins, as specified in the Tiva's datasheet ([2], p1367, Table 24-12), also require bypass capacitors situated close to pin 56. This makes for a total of ten required capacitors around the microcontroller, which leaves the area tight for routing space. Although pins are available for connecting an external oscillator to increase clock accuracy, such a circuit was not included in the design because such accuracy is not required for the SPI or I^2C modules that will be utilized.

The Tiva comes in a small LQFP package with pins averaging between 0.17 and 0.27 mm in pitch. Because the package's pin layout is very dense, traces connecting these pins to other peripherals were made10 mils thick to match the pitch. Once far enough away from the micro, logic signal traces were made12 mils thick in consideration of spacing and lower resistance. Power supply rails to the micro were made to gradually decrease in size until they too matched the pitch of the pins.

4.0 PCB Layout Design Considerations - Power Supply

Power enters the board via a barrel jack connected to the circuit board itself. The power will come from a 5 volt unregulated A.C. adapter, pass through a 5V regulator, and then pass through a 3.3V converter. Both the 5V and the 3.3V lines will power different components throughout the board. The 3.3V source will power the SD card, the push button grid shift registers, and the microcontroller, and the 5V source will power all other components. Since the potential power dissipation will be large, the power traces were made wider to allow higher current to flow and offer less resistance. Using an online trace width calculator, the minimum width required for the 0.5A that the system is capable of providing is at least 12 mils. However, to ensure proper flow of current in this system, 32 mils was the typical width used for power and ground traces. The traces keep this width as long as possible, often times leading up next to the components they are powering before scaling down to a smaller width. They also run through the center of the board in order to minimize the trace length.

Several bulk capacitors were placed near the two voltage regulators in order to supply charge for the bypass capacitors as they discharge. The bypass capacitors were placed as close as possible to the IC's they are decoupling. In most cases this was next to the IC on the top layer, but in the case of the microcontroller, some capacitors were placed on the bottom layer and the traces connected to the top side through vias. Even with the capacitors on the bottom, they were situated as closely to the source pins as possible.

A ground plane was created in order to maximize the grounding capability of the source and to make routing easier. In most cases the pins needing a ground were already floating on the ground plane making the connection simple. In a few cases though, the pin was isolated due to other traces cutting off the ground plane in front of it. To correct this, the isolated area was connected to the ground plane through vias and a trace on the underside of the board.

5.0 Summary

In summary, many issues and concerns arose when designing the printed circuit board. Being wary of the shape of the board as well as the different circuit sections helped the organization of the traces and components. The microcontroller's small package and pitch provided additional difficulties in placement of parts and routing of traces to peripherals. In the design of the power supply, trace width was a major concern and was made as wide as possible without sacrificing efficiency of layout. Finally, a ground pour was included to provide easy grounding for most ICs on the board.

6.0 List of References

[1] Tiva™ C Series TM4C123G LaunchPad Evaluation Board User's Manual, *Texas Instruments*, [Online]. Available: http://www.ti.com/lit/ug/spmu296/spmu296.pdf (Accessed: 28 February 2014).

[2] TM4C123GH6PM Data Sheet, *Texas Instruments*, [Online]. Available: http://www.ti.com/lit/ds/spms376c/spms376c.pdf (Accessed: 28 February 2014).

[3] PCB Trace Width Calculator, *TheCircuitCalculator.com*,[Online]. Available: http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator/ (Accessed: 28 February 2014).