Homework 6: Printed Circuit Board Layout Design Narrative

Team Code Name: Drink Mixer **Group No.** 2

Team Member Completing This Homework: Adam Johnson

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

Evaluation:

SCORE	DESCRIPTION
10	***Excellent*** *– among the best papers submitted for this assignment. Very few corrections needed for version submitted in Final Report.*
9	***Very good*** *– all requirements aptly met. Minor additions/corrections needed for version submitted in Final Report.*
8	***Good*** *– all requirements considered and addressed. Several noteworthy additions/corrections needed for version submitted in Final Report.*
7	***Average*** *– all requirements basically met, but some revisions in content should be made for the version submitted in the Final Report.*
6	***Marginal*** *– all requirements met at a nominal level. Significant revisions in content should be made for the version submitted in the Final Report.*
*	***Below the passing threshold*** *– major revisions required to meet report requirements at a nominal level.* ***Revise and resubmit.***

** Resubmissions are due within **one week** of the date of return, and will be awarded a score of "6" provided all report requirements have been met at a nominal level.*

Comments:

1.0 Introduction

The drink mixer is an eight-channel audio mixer. Because it manipulates audio signals, the primary concern when designing printed circuit boards is noise immunity. Other major issues include fitting large analog circuits using the smallest possible board space, and placing user interface components in alignment with packaging.

2.0 PCB Layout Design Considerations - Overall

The main pcb required proper positioning for external ports such as the USB and RS-232. The LEDs and faders on the board also had to be properly positioned in order to make sure they line up for mounting on packaging. Another consideration were the vias required under the LED drivers for thermal dissipation[1]. The biggest consideration for the dual channel interface pcb was space. The board had to fit within an area of about 3" x 4.5". In addition to the space constraint, the RPG, LED bargraph, and pushbutton had to be on the top side of the board in the correct position for mounting on packaging.

The audioboard pcb also needed to be kept a reasonable size. Eventually, this pcb was split into two pcbs for economic reasons (two smaller boards cost less than one board which is over 60 square inches). In addition to that, the preamplifiers needed to remain close to the A/D pins and XLR input headers. There also needed to be separate grounds for the analog and digital ground, which were connected with a zero ohm resistor (which acts to reduce noise).

The power supply pcb has large traces and linear regulators that need to be lined up to attach to a heat sink. This pcb will be in a separate enclosure from the main packaging because of noise immunity and size considerations.

3.0 PCB Layout Design Considerations - Microcontroller

Using the Hammer simplifies our board layout because the ARM9, as well as its oscillator circuit, SDRAM, Flash, and power regulators are already contained within the prepackaged Hammer board. The DSP, however, was a different story. It has a JTAG interface that requires that trace lengths be the same. If the lengths of these traces are over six inches, resistors must be used as well[2]. An external oscillator was chosen for this microchip instead of a crystal in order to keep continuity with the development kit (no code changes will need to be made from prototype to pcb, which simplifies debugging). There were also special requirements for bypass

capacitor placement on the DSP. Certain bypass capacitors needed to be placed "as close as possible" to the DSP's reference pins[3].

The ATmega32A's were not as complicated to route as the DSP. Noise immunity is not critical for these chips, although bypass capacitors are still needed. All microcontrollers on our design have bypass capacitors on each side, and almost all of the microchips have extra pins routed to headers in order to reduce work if changes need to be made and these pins do need to be used later. The DSP and ARM9 also have reset pushbuttons, while the ATmega32a's have jumpered headers instead because of space considerations.

4.0 PCB Layout Design Considerations - Power Supply

When laying out pcbs, the first traces to be routed were usually power and ground. They needed to be routed to all areas of the board, and must be thick enough to withstand the maximum operating current. Nowhere was this more apparent than for the H-bridges used for fader motor operation. They consume high peak currents, so some of their traces are 30 mil for safety. Ground planes also reduce noise and help with thermal dissipation. The audio pcb has ground planes for audio and digital components which are tied together at a single point with a 0 ohm resistor. The resistor reduces noise between ground planes. Also, power to the DSP was provided by traces that connect to main power traces via 0 ohm resistors, also for noise reduction.

Bypass capacitors were also important in our design, and we had a lot of them since we had 12 microprocessors! Each side of every microprocessor had at least one bypass capacitor (most had two), and (as stated above) the DSP had special bypass capacitor requirements for its reference voltage pins[3].

Since we have sevaral printed circuit boards that require different power voltages, power is provided via 18 guage stranded wire through an umbiblical cord from the power supply pcb to the main packaging. From there, the various voltage rails are jumpered to each pcb as needed. On the individual channel pcbs, due to space considerations, traces were routed wherever they could fit. The Analog to Digital converter ICs on the audio pcb are actually routed so that different power rails go to different sections of the chip's footprint. This isolates digital and analog signals. The Atmega32A's have a ground pad underneath them, so ground pins were

routed to a via under the middle of the chip, which was connected to ground. Filter caps were also routed to this via.

5.0 Summary

When designing an audio mixer, the main concern is noise immunity. Other concerns include fitting large circuits onto small board space, and placing user interface components in alignment with packaging. Noise immunity practices including ground planes, bypass capacitors, and noise isolation jumpers (known elsewhere as 0 ohm resistors) were used. Circuit board design will mesh with packaging, and was designed with power needs in mind as well.

List of References

[1] "24-Channel, 12-Bit PWM LED Driver with Internal Oscillator," *Texas Instruments*, Sept. 2008. [Online]. http://focus.ti.com/lit/ds/symlink/tlc5947.pdf [Accessed: Oct. 7, 2009].

[2] D. Doyle, "Analog Devices JTAG Emulation Technical Reference," *Analog Devices*, EE-68, Apr. 15, 2008. [Online]. http://www.analog.com/static/imported-files/application_notes/ee-68_rev10.pdf [Accessed: Oct. 7, 2009].

[3] "ADSP-21261/ADSP-21262/ADSP-21266," *Analog Devices*, July 2009. [Online]. http://www.analog.com/static/imported-files/data_sheets/ADSP-21261_21262_21266.pdf [Accessed: Oct. 3, 2009].