EECE 690/890
Digital Radio Hardware Design

Team 4   Assignment 1

Common Tasks Due: Thurs 10/8/98
**Specific Tasks Due   Tues 10/13/98**

Introduction

This is the first of a series of assignments designed to guide you through the tasks needed to complete the audio circuit design, layout, and mechanical engineering tasks. The first part of the assignment is common for all team members and is designed to get each of you familiar with the process of creating PC board "artwork". Each team member should do the tasks under "Common Tasks" independently. Following these, specific tasks are detailed for each team member.

Attached to this assignment is an additional page giving a preview of future tasks that will need to be completed before the PDR. You are encouraged to work ahead if you can. This will ease your workload when things like tests stack up in other classes in the coming weeks.

Common Tasks for Both Team Members

Each team member should perform the following tasks on his/her own. You may help each other out, but each person should go through the full exercise and make their own printouts:

- Do the Layout Tutorial handed out in conjunction with this assignment. For this first exposure to the software, just place some components and get used to drawing traces anywhere you want. We'll do a specific layout in the following steps.
- During the tutorial, zoom in on the SOIC 14 footprint and the R2012 footprint and measure/record their dimensions. These will be helpful in doing draft layouts on paper (highly recommended for initial planning of your final layout!).
- Do two draft layouts of the circuit below. One should be done assuming that you can route on both sides of the board (i.e. no ground plane). The second should be done with a ground plane on the Solder side and traces on the Component side. For both, assume the circuit will go in the bottom right corner of a PC board and that Vcc comes in from a trace entering from the upper left.
- Generate printouts of the Component Side, and the Solder Side (separate pages).
- Generate the Photoplot (Gerber) and Drill files and print them out using WordPad.
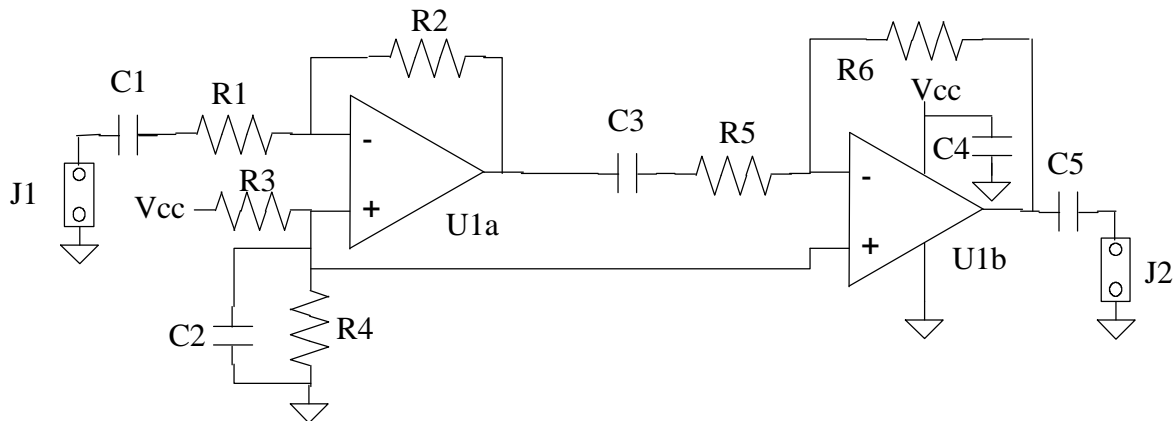
NOTES: U1 is an LM2902 in an SOIC14 package
All R, C are 0805 SMT components
J1, J2 are headers with
0.1 inch spacing, 0.64mm square pins

EDA/Mechanical Design Engineer 1 Tasks

- Familiarize yourself with connectors used in computer and other commercial products by a) looking at computer circuit boards, b) looking in the DigiKey and Mouser Electronics catalogs, and c) talking with Steve and Joe in the Electronics shop downstairs.
- Get started on the product design by determining what connectors will be needed to connect the analog/RF board to the digital board, and to connect the boards to the various external parts (antenna, LCD, handset, battery, etc.).
- Make a preliminary list of all areas of the design where connectors and cables will be needed, and specify what type of connector you expect to use (number of pins, physical dimensions (draw picture of each and label pin-to-pin spacings, etc.), manufacturer/vendor part numbers and catalog page numbers, and cost. Be certain that the connectors are easy to get, small, and low-cost! Again, the trick here is to see what is commonly used in other products and review the vendor catalogs carefully to understand the array of choices.

Your deliverables are listed below:

- The printouts specified in the common task above. Each team member should provide his or her own printouts.

- Your connector list specifed above.

EDA/Mechanical Design Engineer 2 Tasks

- Familiarize yourself with components such as microphones/speakers, switches, ringers (or buzzers), batteries and battery holders, etc. used in commercial products by a) looking at cordless phones, b) looking in the DigiKey and Mouser Electronics catalogs, and c) talking with Steve and Joe in the Electronics shop downstairs.
- Get started on the product design by determining what parts will be needed in our product, and how they will be mounted within the physical enclosures used for the base and mobile units. To help in this task, you should draw some sketches of how the base and mobile should be built. (Remember that we will display at Open House in the Spring, so it may be helpful to have the base unit fully exposed so that visitors can see all the work that went into it.
- Make a preliminary list of parts you will need. As a minimum, identify switches, ringers, and battery components, as well as any physical enclosures and/or fabrication of panels. Include manufacturer/vendor part numbers, catalog page numbers, and cost.

Your deliverables are listed below:

- The printouts specified in the common task above. Each team member should provide his or her own printouts.

- Your sketches of the product design and list of parts/prices/etc.

Team 3 Future Assignments

The following gives an overview of the tasks remaining after task 1. These will be broken down into assignments like task 1, with recommended subtasks and deliverables. We will also have periodic "mini-design reviews" in which your team will meet with the instructors. In these informal reviews, you will need to explain your design decisions, and we will try to find "holes" in them that may need to be addressed.

EDA/Mechanical Design Engineer 1

- Study the LCD hardware interface on the data sheet to determine connections needed and any supporting circuits needed (e.g. contrast adjust pot).

- Draw a schematic of the connections and circuits needed on the digital board, including the connector to be used and pin assignments. (Be sure to consider EMC)

- Add the connectors and cables needed to the parts list.

- Get DC voltage requirements and current consumption estimates from all teams and locate suitable voltage regulators to be used (low dropout voltage and acceptable power dissipation).

- Decide if we want to use Nicad batteries in the mobile and if so, find a suitable charge regulator.

- Draw schematic showing all power supply components (including connectors), and add these to the parts list.

- Work with digital PLD and software designers to determine pin assignments on the PLD and uC, and connections to the ADPCM codec, and generate a schematic of these circuits.

- Work with your teammate to generate a complete schematic of the digital board, including connectors.

- Do initial "floorplanning" (component placement diagram for digital PC board).

- Do digital circuit board layout. (Do this on paper first! Then enter the design in the Layout program - This will help keep you flexible and allow you to try several options.)

- Check the layout against the schematic by printing each out and highlighting nets (one at a time) on each to be sure they match.

- Do a final check, in conjunction with members of other teams to be sure nothing in the design has changed! Check the layout again!

- Generate Gerber and Drill files.

- Confirm that the tools called out in the generated files are acceptable to our PC board fabricator, and if not, modify them.

- View the photoplot and drill files in a third-party Gerber viewer to be sure it was generated correctly.

- Prepare the files for sending to the fabricator. (We may need to merge both the RF and digital layouts into a single design to lower fab cost, but this can be done with cut-and paste at the end).

EDA/Mechanical Design Engineer 2

- Study/test the handset to detemine the type and impedance of mic/speaker used.

- Study the ADPCM application note and design the opamp circuits (and speaker amp if desired) needed.

- Generate a schematic of these parts and add any ICs and connectors needed to the parts list.

- Refine your mechanical design from assignment 1, select cables needed, and generate any mechanical drawings needed for panel fabrication etc.

- Work with the RF team to get their schematic and layout designs.

- Work with your teammate to create a list of all IC sockets, connectors, and other special parts needed in the layout.

- Locate footprints for all the parts in the Professional version of PCBoards layout software, place the footprints on a page, and print out the page for reference.

- Create a list of all IC sockets, cables, etc. needed.

- Enter RF layout

- Check the layout against the RF team's schematic by printing each out and highlighting nets (one at a time) on each to be sure they match.

- Do a final check, in conjunction with members of other teams to be sure nothing in the design has changed! Check the layout again!

- Generate Gerber and Drill files.

- Confirm that the tools called out in the generated files are acceptable to our PC board fabricator, and if not, modify them.

- View the photoplot and drill files in a third-party Gerber viewer to be sure it was generated correctly.

- Prepare the files for sending to the fabricator. (We may need to merge both the RF and digital layouts into a single design to lower fab cost, but this can be done with cut-and paste at the end).