EECE 690/890
Digital Radio Hardware Design

Team 1 Assignment 4

Due: Thursday, 10/22/98

Introduction

This is the fourth in a series of assignments designed to guide you through the tasks needed to complete the RF transceiver design. In this assignment, both team members will be working on the RF layout. The assignment goal is to generate a hand-drawn layout at a 2:1 or 4:1 scale that you can supply to Team 4. Your hand-drawn layout should be neatly done and detailed/accurate enough that Team 4 does not need to make any decisions when entering the artwork into the computer.

Since the tasks for both team members are tightly coupled and basically similar, only one set of instructions is given below. You should do the layout for the components that pertain to your circuits but should coordinate closely to be sure that the final layout fits together well. In particular, it is critical that you work together on the "floorplanning" and communicate any changes needed to the floorplan so that your final designs can be integrated well into a single board.

Layout Design Tasks

- Visit the comm lab (297) and study some of the PC boards there to get a feel for how layouts are done, and what density of components can be achieved. Your goal will be to produce a layout that is aesthetically pleasing, relatively compact (no larger than 5"x8" total board space), and requires little or no traces on the ground plane side.
- Make a list of all of the components that you will use which are not simple two-terminal surface mount resistors, capacitors, or inductors. For each one, find the necessary component "footprint" (talk to team 4 to see understand what this means), and sketch it next to the component name. Then sketch the footprint for 0805 (.08x.05 inch) discrete components (resistors, caps, inductors). This will help you as you plan your layout.
- Buy a PC board layout template if you can find one at the bookstore/ etc. (This is a stencil that has holes you can use to draw the component footprints.)Decide on whether you will use a 2:1 or 4:1 scale for your layout and get some gridded paper to create your layouts on.

- Develop a layout floorplan. You may start with a 5x8 inch board outline, but should make your final design smaller than this if possible. The goal in component placement is to minimize board size and ground-plane traces. This is usually accomplished by placing major components (e.g. ICs) that have the most connections to each other close together, and orienting them to avoid the need for traces to cross other traces. Don't forget the power and control signals (sleep, synth clock/data, etc.). These need to be routed to one or two connectors and therefore their placement should be carefully planned before getting too deep into wiring up the components themselves. (Ground is no problem since the whole backside is a ground plane and all you need is a via to make a ground connection.)

  A possible basic floorplan is attached to this assignment to help you get started. You should study this suggested plan and modify it as you see fit.

- Determine if we will use 0.062 or 0.031 inch thickness board. This will determine the trace width needed for 50 Ohm lines in the RF sections of the circuit. (105 mils for 0.062" FR4 board or 49 mils for 0.031" board).

- If you are the transceiver designer, do a layout for the duplexer. Since we are using two separate filters, it is important to consider the impedance that each filter presents to the circuit at the frequency of the opposite filter. A Smith chart plot of input impedance of a dielectric resonator filter is attached to this assignment. Note that outside the passband, the filter exhibits either a low or high impedance. If it presents a low impedance, then it will load down the signal going to the opposite filter. To fix this you will need to place a quarter-wave line between the filter and where it connects to the opposite filter to convert the low Z to a high Z. See your instructor for additional details on how this can be done.

- If you are the synth designer, talk to team 4 and find out what connector type will be used. It is recommended that you decide on what signals will be on what pins and let team 4 follow your lead when they do the digital circuit layout, since they have more freedom to cross traces on their two-sided board.

- Begin drawing layouts for each section of the design. It is recommended that you work with no more than two ICs at a time, generating small layout sketches that you can later piece together into your final layout. If you did a good floorplan, this piecing together will go relatively smoothly and you can make refinements as you do it to improve the appearance of your final artwork.

- Work with your teammate to generate the final layout drawing to be given to team 4.

- Work with your teammate to generate an "assembly drawing" that shows where each component is located on the board. This will help when we get the boards back from the fabricator and begin "stuffing" it (placing parts on it).

Deliverables:

- A copy of your final layout and assembly drawings to both your instructor and team 4.

Team 1 Future Assignments

The following gives an overview of the tasks remaining after task 4.

RF Transceiver Design

- Carefully check your layout and report any corrections to team 4 immediately.
- Do test planning and add test points to block diagram, schematic, and layout.
- Work to get all parts on-order.

Possible Floorplan: