EEC 134 PCB 3 Guidelines

Instructor: Xiaoguang "Leo" Liu
lxgliu@ucdavis.edu

Last updated: February 15, 2020

In Lab 3, we will start putting multiple RF components together on a single PCB to prepare you for your Quarter 2 design. In particular, you will design a simple transmitter PCB according to Fig.1.

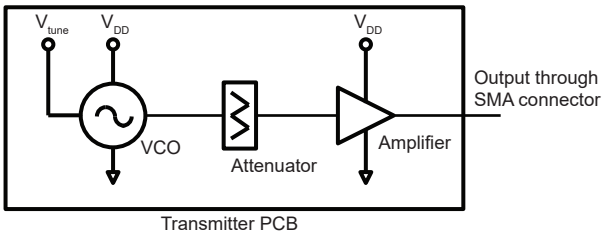

Figure 1: System block diagram for PCB 3.

1 Design Requirements

- The main components used in this design are
  1. VCO: Maxim MAX2750
  2. Attenuator: Analog Devices HMC653LP2E
  3. Amplifier: Analog Devices ADL5611
- Desired output frequency range: 2.3–2.5 GHz
- Desired output power: >15 dBm
- Output connector: SMA, same as in PCB 2.
- The PCB should be less than 1"×2" in size. The smaller the better.

2 Some tips

1. Read the datasheets of these components carefully to understand their characteristics. You should follow the manufacturer's recommended schematic and layout designs. If the manufacturer provides evaluation boards, see if you can obtain the designs files and use them as a reference.

2. Make sure to include dc-blocking capacitors at the input and output of the active components to ensure that the components can be biased correctly.

3 Test Report Requirements

In the test report, please include the following

- Circuit schematic
- PCB layout
- Measured output frequency vs V_{tune}
- Measured output power vs output frequency
- Discussion of the measured results.

4 Critical Dates

- Mar. 1st, 2020: PCB 3 design and review report
- Mar. 1st, 2020: PCB 3 design files
- Mar. 19th, 2020: PCB 3 test reports