Amplifier Design Description for ECE 432/532

Spring 2016

The experience you build up during this project will be useful in the 2nd project even though we will use packaged LNA for it. You are to design a "low noise amplifier" (LNA) with the following specifications:

1. Frequency range: 2.4 to 2.6 GHz (2.35 to 2.65 preferred)
2. Gain: > 10 dB
3. Noise figure: < 2.5 dB
4. Input return loss > 10 dB
5. Output return loss: > 10 dB
6. Supply voltage: 3.3 Volts
7. Supply current: < 100 mA

LNA is to be built on FR4 or some better substrate and manufactured in LID. You can use either SMD components or microstrip design. Biasing circuit should be included on the board. Make sure that you have ac-coupling capacitors on input and output. You will use SAV-541+ transistor by MiniCircuits. For SMD components you can use any manufacturer but remember that a selection of parts from Murata is stocked by IEEE store. Make sure that you consider connectors when laying out the circuit. Needless to say, you will have to use ADS in all stages of your design.

You will have to solder all of your components and test the final board. Your report should include comparison of measured data with simulated – try to do these on the same plot. One report per team is required (one submission). Your report should include description of any alternative designs that you considered, any items that were especially troublesome, and in your conclusions you should include a "lessons learned" section with suggestions on how you might to do this if you had to do it again.

In general, there will be three members of the team, with following "specializations" :

1. Circuit design and implementation (simulation + layout)
2. Test and calibration plan and implementation
3. Simulation based sensitivity analysis of the circuit and implementation (DC and ac)
4. Optional: using Emag simulation (Momentum or similar)

Note that every team member has to be able to explain any and all parts of the design. For example, if you worked on item 1. Above you still must be able to explain how the testing is done and why and what steps you took in analyzing sensitivity of your circuit.

You can (and, perhaps, should) follow the design procedure given in K. Payne's white paper [1]. Some additional literature on the topic will be posted. Rubrics for report assessment will also be posted (they are the same as for ECE 431/531 project). You will also have to demonstrate that your amplifier works and how you measured its characteristics. You will also have to archive your design from ADS and upload it to D2L as well.

Extra credit (must be identified in your report):

- Setup and measurements done on a probe station – I would really like you to attempt this. It is not as hard as it sounds.
- To get some competitive juices flowing, there will be extra credit given to the best design. I will take into account gain, power dissipation, and general artistic impression ☺ .
- Measurement(s) of 1-dB compression point and third order intercept point (IP3)
- Using series inductive feedback design procedure (some references are given below)
- Measuring LNA noise figure (to be done only if all other specifications are already met)
- Using Emag simulation

References:

[1] K. Payne, "Practical RF Amplifi er Design Using the Available Gain Procedure and the Advanced Design System EM/Circuit Co-Simulation Capability," Agilent Technologies (5990-3356EN), 2008.

[2] A. Victor and J. Nath, "An Analytical and Graphical Method for LNA Design with Feedback," *High Frequency Design*, pp. 16 – 28, June 2010.

[3] D.D. Henkes, "LNA Design Uses Series Feedback to Achieve Simultaneous Low Input VSWR and Low Noise," *Applied Microwave & Wireless*, pp. 26-32, October 1998.

[4] A. Harter, "LNA Matching Techniques for Optimizing Noise Figures," *RF Design*, pp. 20 – 30, February 2003.

[5] "Low-Noise Amplifier Stability Concept to Practical Considerations," parts 1 to 3, Maxim Integrated Products, Application Notes 1849, 1851 and 1852, available at www.maxim-ic.com/an1849 , etc. last accessed May 2012.