

Training Course Workbook

Advanced Design System

ADS 2003 Fundamentals



Agilent EEs of EDA - Customer Education

Santa Rosa, California USA

ADS 2003 Fundamentals

Contents: Slides and Lab Exercises

TOPICS (tab sections)

- 1: Circuit Simulation Fundamentals
- 2: System Simulation Fundamentals
- 3: DC Simulation and Circuit Modeling
- 4: AC Simulation and Tuning
- 5: S-Parameter Simulation and Optimization
- 6: Filters: Transient, DesignGuide, Momentum, DAC
- 7: Harmonic Balance Simulation
- 8: Circuit Envelope Simulation
- 9: Final Circuit / System Simulation (Ptolemy)

Appendix: additional material on simulators and examples

ADS Fundamentals - 2003

LAB 1: Circuit Simulation Fundamentals

Overview - This lab covers user interface basics, ADS files, schematic capture, simulation, and data display. In addition, tuning and ADS example files are also covered.

OBJECTIVES

- Create a new project and schematic designs
- Setup and perform S-parameter simulation
- Display simulation data and save files
- Tune circuit parameters during simulation
- Use the Examples files and node names
- Perform a Harmonic Balance simulation
- Write an equation in the data display



Agilent Technologies

Table of Contents

1. Start ADS on the computer.....	3
2. Create a new Project.....	4
3. Examine the files in your new project directory.....	5
4. Create a low-pass filter design.....	5
5. Setup the S-param Simulation.	8
6. Launch the simulation and display the data.	8
7. Save the Data Display Window.....	10
8. Tune the filter circuit.....	11
9. Simulate an RFIC with Harmonic Balance.....	14
10. Add a wire label (node name) , simulate, plot data	16



Agilent Technologies

LAB 2: System Design Fundamentals

Overview - This chapter introduces the use of behavioral models to create a system such as a receiver. This lab will be the first step in the design process where the system level behavioral models are simulated to approximate the desired performance. By setting the desired specifications in the system components, you can later replace them with individual circuits and compare the results to the behavioral models.

OBJECTIVES

- Use the skills developed in the first lab exercise.
- Create a system project for an RF receiver using behavioral models (filter, amplifier, mixer) where: RF = 1900 MHz and IF= 100 MHz.
- Use an RF source, LO with phase noise, and a Noise Controller.
- Test the system: S-parameters , Spectrum, Noise, etc.

Table of Contents

1. Create a New Project (system) and schematic.....	3
2. Build a behavioral RF receiver system.....	3
3. Set up an S-parameter simulation with frequency conversion.	5
4. Plot the S-21 data.	7
5. Increase gain, simulate, and add a second trace.....	7
6. Set up an RF source and LO with Phase Noise.....	8
7. Set up a HB Noise Controller.	9
8. Set up a HB simulation.....	10
9. Simulate and plot the response: pnmx and Vout.	11
10. OPTIONAL - SDD (Symbolically Defined Device) simulation.....	12

LAB 3: DC Simulations and Circuit Modeling

Overview - This chapter introduces parametric subnetworks: how to create and use them in hierarchical designs. Beginning with a device model, the lowest level subnetwork will also contain packaging parasitics to better model the device behavior. Also, a test template will be used to simulate curve tracer responses from which a bias network can be computed, built, and checked. The circuit in this lab exercise will be the foundation of the amplifier that will be used for the other lab exercises in this course.

OBJECTIVES

- Model a generic BJT with parasitics and save it as a sub circuit.
- Set up and run numerous DC simulations to determine performance.
- Calculate bias resistor values in the data display.
- Build a biased network based on the DC simulations.
- Test the biased network.



Table of Contents

1. Create a new project: amp_1900.	3
2. Set up a generic BJT symbol and model card.....	3
3. Add parasitics and connectors to the circuit.....	5
4. View the default symbol.....	6
5. Set up the Design Parameters and built-in symbol.....	6
6. Use a curve tracer template to test the bjt_pkg sub-circuit.....	8
7. Modify the template Parameter Sweep.....	10
8. Simulate at beta=100 and 160.....	10
9. Open a new design and check all your files in the Main window.	11
10. Set up and simulate the dc bias parameter sweep.....	12
11. Calculate bias values Rb and Rc for a grounded-emitter circuit.	13
12. Set up the biased network.....	14
13. Simulate and annotate the DC solution.	15
14. OPTIONAL: Sweep Temperature	16

LAB 4: AC Simulations

Overview - This lab continues the amp_1900 project and uses the same sub-circuit as the previous lab. This exercise teaches the basics of AC simulation, including small signal gain and noise. It also shows many detailed features of the data display for controlling and manipulating data.

OBJECTIVES

- Perform AC small signal and noise simulations.
- Adjust pin/wire labels.
- Sweep variables and write equations.
- Control plots, traces, datasets, and AC sources.



Table of Contents

1. Copy & Paste (Ctrl+C / Ctrl+V) from one design to another.	3
2. Modify the copied circuit and pin labels.....	4
3. Push and pop to verify the sub circuit.	5
4. Set up an AC simulation with Noise.....	5
5. Simulate and list the noise data.....	5
6. Control the output of equations and node voltages.....	6
7. Simulate without noise.....	7
8. Write a data display equation using a measurement equation.....	7
9. Work with measurement and data display equations.....	8
10. Plot the phase and group delay for the ac analysis data	9
11. Variable Info and the <i>what</i> function.....	10
12. OPTIONAL - Sweep Vcc (as if the battery voltage is decreasing).....	11



Agilent Technologies

LAB 5: S-parameter Simulation and Optimization

Overview - This exercise continues the amp_1900 design. It teaches how to setup, run, optimize and plot the results of various S-parameter simulations. Also, the optimizer is used to create the impedance matching networks.

OBJECTIVES

- Measure gain and impedance.
- Set up and use sweep plans, parameter sweeps, and equation based impedance.
- Calculate values for a matching network.
- Tune a matching network.
- Use optimization to meet design goals.
- Use Noise and Gain circles.



Table of Contents

1. Set up the simulation and circuit with ideal components.....	3
2. Simulate and plot data with marker readout modifications.....	4
3. Write an equation to vary the Term port impedance.....	4
4. Calculate L and C values in the data display.	5
5. Replace L and C with calculated values and simulate.....	6
6. Add matching components L and C, simulate, and plot the results.....	7
7. Tune the input matching values.....	8
8. Add output matching components.....	9
9. Set up an Optimization controller and Goals.....	10
10. Enable the components to be optimized.....	12
11. Plot the results.....	14
12. Update optimized values and disable the <i>opt</i> function.	15
13. Simulate the final matched circuit.	16
14. Stability equations with gain and noise circles.....	17
15. OPTIONAL – Read and Write S-parameter Data with an S2P file	19



Agilent Technologies

LAB 6: Filters: Design Guide, Transient, and Momentum Simulation

Overview - This chapter shows the fundamentals of creating filters in ADS and using the Transient simulator. The Design Guide is used to build a lumped component filter and Momentum is used to test a microstrip filter.

OBJECTIVES

- Build a 200 MHz IF low pass lumped filter using a DesignGuide
- Build a 1900 MHz RF bandpass filter in microstrip.
- Perform a Transient analysis on the microstrip filter.
- Simulate the microstrip filter in Momentum.
- OPTIONAL –DAC (data access component) exercise.



TABLE OF CONTENTS

1. Change projects and start the Design Guide.....	3
2. Insert the LPF Smart Component and design the filter.	3
3. Microstrip 1900 MHz bandpass filter.....	6
4. Transient Analysis on the microstrip filter.....	8
5. Momentum simulation from layout.....	10
6. OPTIONAL - Impedance response Data Access Component.	15

LAB 7: Harmonic Balance Simulations

Overview - This exercise continues the amp_1900 design and shows the fundamentals of using the Harmonic Balance simulator to look at the spectrum, analyze compression, calculate TOI, and perform other non-linear measurements.

OBJECTIVES

- Set up and perform a 1 tone HB simulation.
- Set up and perform a 2 tone HB simulation.
- Use variables for simulation and source control.
- Test Gain, Compression, Available Power, Noise Figure, IP3, and other specifications.
- Use the *ts* transform on HB data.
- Work with equations, plots, and the Mix table.



TABLE OF CONTENTS

1. Set up the circuit with a P_1Tone source.....	3
2. Set up a one-tone Harmonic Balance simulation.....	4
3. Write a measurement equation for <u>dBm of Vout</u> and simulate.....	4
4. Plot the spectrum, equation, and <i>ts</i> of the node voltages.....	5
5. Operate on Vout and Mix using functions and indexing.....	6
6. Simulate Delivered Power and Zin.	7
7. Test for Gain Compression using the XDB simulator.....	8
8. Simulate compression with a power sweep.....	9
9. Plot various gain, power, and line equations.....	10
10. Two-tone HB simulation with variables.....	11
11. Use equations to access and control HB data.	12
12. Simulate IP3 or TOI (Third Order Intercept).....	13
13. OPTIONAL - Sweep RF power against the TOI measurement	15

LAB 8: Circuit Envelope Simulation

Overview - This chapter covers the basics of Circuit Envelope simulation to simulate time and frequency of an output signal when the input is a pulsed or modulated source such as GSM, CDMA, etc.

OBJECTIVES

- Set up Circuit Envelope simulations using a behavioral amp
- Experiment with simulation parameters
- Test for distortion
- Use demodulation components and equations
- Simulate the 1900 MHz amp with a GSM signal
- Plot carrier and baseband data
- Operate on datasets in the frequency and time domain



TABLE OF CONTENTS

1. Set up a PtRF source and behavioral amp.....	3
2. Set up the Envelope Simulation controller.....	4
3. Simulate and plot the time domain response.	4
4. Add distortion to the behavioral amplifier.....	6
5. Set up demodulators and a GSM source.....	7
6. Set up the Envelope Simulation with variables.....	7
7. Simulate and plot the results demodulated results.....	8
8. Use a filter to simulate phase distortion.	9
9. Simulate and plot input and output modulation.	9
10. Simulate amp_1900 with a GSM source	10
11. Plot the GSM data and spectrum.....	11
12. OPTIONAL – Channel power calculations	14

LAB 9: Final System and Circuit Simulations

Overview – This last lab exercise brings together all the circuits built during the course: the amplifier and filters. They replace the behavioral system models used in the earlier exercise.

OBJECTIVES

- Create a sub-circuit for the 1900 MHz amplifier for use in the system.
- Use the Smart Simulation Wizard.
- Set up and run simulation a HB simulation with a GSM source.
- Set up and run a CE simulation using a CDMA source.
- Simulate ACPR and power specs using an example data display.
- Program Marker sliders to customize data displays.
- OPTIONAL - Co-simulations with minimal instructions.



Agilent Technologies

TABLE OF CONTENTS

1. Create the final AMP_1900 sub-circuit for the library.	3
2. Simulate AMP_1900 with the Smart Simulation Wizard.....	4
3. Create the final HB swept LO schematic and equations.....	7
4. Final HB simulation: 2 tones with swept LO power and noise.....	9
5. View and Plot: NF, Conv Gain, dbm_out, and IF_gain.....	11
6. Final Envelope simulation: CDMA source.	12
7. Open an Example DDS to plot ACPR and Power Specs.....	13
8. Plot the spectrum using a programmed marker slider.....	14
9. OPTIONAL - Co-simulation of the behavioral rf system.....	16