OSA90/hope™
Lab Manual

Version 3.0

August, 1994

Optimization Systems Associates Inc.

LIABILITY AND WARRANTY

Copyright

 OSA90/hope Lab Manual Version 3.0 first published in 1994.
Printed in Canada.

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario
Canada L9H 5E7

Tel 905 628 8228
Fax 905 628 8225

Trademarks of Optimization Systems Associates Inc.

OSA90
OSA90/hope
Datapipe
Empipe
Spicepipe
HarPE
FAST
Space Mapping

Other Trademarks

HP is a trademark of Hewlett-Packard Company.
Sun and Sun Workstation are trademarks of Sun Microsystems, Inc.
SunOS is a trademark of Sun Microsystems, Inc.
UNIX is a registered trademark of AT&T.
em and xgeom are trademarks of Sonnet Software, Inc.
PostScript is a trademark of Adobe Systems Inc.

Table of Contents

Lab 1 Simulation of an LC Transformer

Objectives

This lab is designed to help you set up linear circuit simulation using OSA90.

Circuit Diagram and Parameter Values

Consider the LC transformer circuit shown in Fig. L1. Use OSA90 to compute the branch currents, branch voltages and magnitude of the input reflection coefficient of the LC circuit. Use 21 uniformly spaced points in the frequency range 0.5 - 1.179 rad/s to obtain the reflection coefficient. Use frequency points 0.5 rad/s, 0.8 rad/s and 1.1 rad/s to obtain the requested voltages and currents. The parameter values are $E = 3$ V, $R_1 = 3\ \Omega$, $R_2 = 1\ \Omega$, $L_1 = 1.041$ H, $C_2 = 0.979$ F, $L_3 = 2.341$ H, $C_4 = 0.781$ F, $L_5 = 2.937$ H, $C_6 = 0.347$ F.

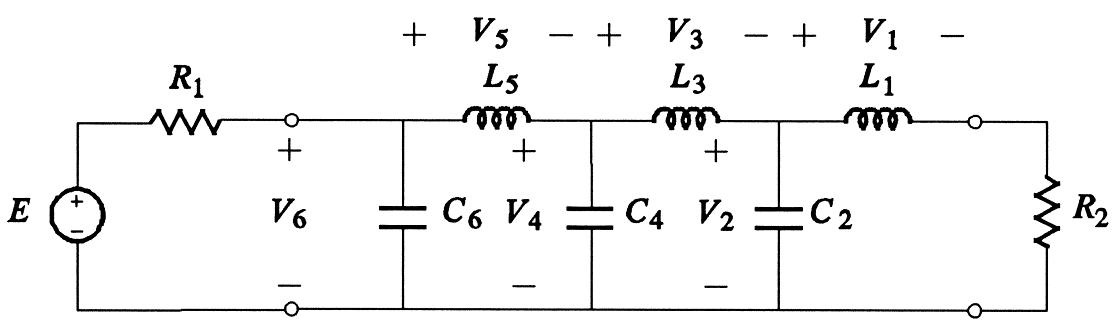

Fig. L1 A lumped element LC transformer.

Procedure

Create Your Circuit File

(1) Create a Control block to set the default unit system to Non_Microwave_Units (Manual Chapter 3, pages 3-24 to 3-25).

(2) Create a Model block to describe the circuit.

(a) Set the value of the angular frequency to 0.5 by assigning 0.5 to the label Omega.

(b) Define the elements R_2, L_1, L_3, L_5, C_2, C_4 and C_6 using the keywords RES, IND and CAP (Manual Chapter 6 - "Elements") and the topology of the circuit (Manual Chapter 6).

(c) Define the input port with the excitation (both the voltage E and the resistance R_1) (Manual Chapter 6 - "Ports"), e.g.,

```
PORT node1 node2 NAME=portname V=E R=R1;
```

(d) Define complex branch voltages V_1, V_2, V_3, V_4, and V_5 using Voltage Labels (Manual Chapter 6 -"Voltage Labels"). For example,

```
VLABEL node1 node0 NAME=V1;
```

Notice that this implies defining arrays: MV1[], PV1[], ..., MV5[], PV5[] where "M" stands for *magnitude* and "P" stands for *phase* of the complex voltages (Manual Chapter 6, page 6-31). In our case (linear circuit under sinusoidal excitation), only the fundamental components, namely, MV1[1], PV1[1], ..., MV5[1] and PV5[1], are nonzero.

(e) Complete the circuit description using the CIRCUIT statement (Manual Chapter 6 - "The Circuit Statement").

(f) You do not need to define the branch voltage V_6 by VLABEL, because it coincides with the input port. V_6 is automatically available as MV*portname*[1] and PV*portname*[1], where *portname* is the name of the input port you defined at Step 2(c).

(g) Use the MP2RI transformation (Manual Chapter 4 - "Built-In Transformations") to find the real and imaginary parts of the requested voltages. For example:

```
MP2RI(MV1[1], PV1[1], RV1, IV1);
```

where RV1 and IV1 are the real and imaginary parts of the voltage V_1. Apply Ohm's Law to compute the real and imaginary parts of the requested currents.

(h) Terminate your Model block with the "End" keyword.

(3) Define the Sweep block (Manual Chapter 9). Find the magnitude of the reflection coefficient MS11 and real and imaginary part of all branch voltages and currents over the specified range of frequency.

(a) AC simulation (Manual Chapter 9 - "Sweep Block"):

```
AC: Omega: from 0.5 to 1.179 n=20, FREQ=(Omega/(2*PI)), RREF=3, MS11;
```

AC simulation computes the small-signal S parameters. MS11 is one of them, namely the modulus of S_{11}

(b) HB simulation (Manual Chapter 9 - "Sweep Block"):

```
HB: Omega: 0.5 0.8 1.1 FREQ=(Omega/(2*PI)) RV1, IV1, ..., V6, IV6, RI1,
    II1, ..., RI6, II6;
```

HB simulation computes the branch voltages and currents.

(c) Terminate your Sweep block with the "End" keyword.

The Sweep block may have the following appearance.

```
Sweep
   AC: Omega: from 0.5 to 1.179 n=20  FREQ=(Omega/(2*PI)), RREF=3OH, MS11;
   HB: Omega: 0.5 0.8 1.1 FREQ=(Omega/(2*PI)), RV1, IV1, RV2, IV2, RV3,
       IV3, RV4, IV4, RV5, IV5, RV6, IV6, RI1, II1, RI2, II2, RI3, II3,
       RI4, II4, RI5, IV5, RI6, II6;
End
```

Perform Circuit Simulation

(1) Parse the circuit file by exiting the file editor (press <F7>).

(2) Press <D> to choose the "Display" option.

(3) Press <X> to choose the "Xsweep" option.

(4) Use the "Graphical" option to display the response diagrams (Manual Chapter 9 - "OSA90.Display Menu Option").

(5) Use the "Numerical" option to obtain the numerical outputs. To obtain all the responses, you need to toggle among the three sweep sets in the "Xsweep" menu.

Complete The Following Tables

You may find the Report Generation feature of OSA90 helpful in generating the tables. See the Appendix for details.

TABLE I REFLECTION COEFFICIENT

No.	ω (rad/s)	MS11= $\lvert\rho\rvert$
1	0.5	
2	0.8055	
3	1.179	

TABLE II BRANCH VOLTAGES*

No.	ω (rad/s)	V_1(V)	V_2(V)	V_3(V)	V_4(V)	V_5(V)	V_6(V)
1	0.5						
2	0.8						
3	1.1						

* The voltages should be presented in the form (Real(voltage), Imaginary(voltage)).

TABLE III BRANCH CURRENTS*

No.	ω (rad/s)	I_1(A)	I_2(A)	I_3(A)	I_4(A)	I_5(A)	I_6(A)
1	0.5						
2	0.8						
3	1.1						

* The currents should be presented in the form (Real(current), Imaginary(current)).

Appendix for Lab 1

Report Generation

The Report Generation feature in OSA90 helps the user create elegantly formatted numerical outputs directly from the OSA90 text editor.

For example, by adding the following Report block, a numerical report will be generated to help you in filling in the Tables I, II and III on the previous pages.

Report Block Template

```
Report

                 REAL PARTS OF BRANCH VOLTAGES
-----------------------------------------------------------------
   Omega   RV1      RV2      RV3      RV4      RV5      RV6
-----------------------------------------------------------------
${     $%3.1f$$Omega$Hz   $% -6.4f$$RV1$   $% -6.4f$$RV2$   $% -6.4f$$RV3$   $%
-6.4f$$RV4$  $% -6.4f$$RV5$  $% -6.4f$$RV6$}$
-----------------------------------------------------------------


              IMAGINARY PARTS OF BRANCH VOLTAGES
-----------------------------------------------------------------
   Omega   IV1      IV2      IV3      IV4      IV5      IV6
-----------------------------------------------------------------
${     $%3.1f$$Omega$Hz   $% -6.4f$$IV1$   $% -6.4f$$IV2$   $% -6.4f$$IV3$   $%
-6.4f$$IV4$  $% -6.4f$$IV5$  $% -6.4f$$IV6$}$
-----------------------------------------------------------------


                 REAL PARTS OF BRANCH CURRENTS
-----------------------------------------------------------------
   Omega   RI1      RI2      RI3      RI4      RI5      RI6
-----------------------------------------------------------------
${     $%3.1f$$Omega$Hz   $% -6.4f$$RI1$   $% -6.4f$$RI2$   $% -6.4f$$RI3$   $%
-6.4f$$RI4$  $% -6.4f$$RI5$  $% -6.4f$$RI6$}$
-----------------------------------------------------------------


              IMAGINARY PARTS OF BRANCH VOLTAGES
-----------------------------------------------------------------
   Omega   II1      II2      II3      II4      II5      II6
-----------------------------------------------------------------
${     $%3.1f$$Omega$Hz   $% -6.4f$$II1$   $% -6.4f$$II2$   $% -6.4f$$II3$   $%
-6.4f$$II4$  $% -6.4f$$II5$  $% -6.4f$$II6$}$
-----------------------------------------------------------------
```

```
MAGNITUDE OF THE REFLECTION COEFFICIENT
--------------------
    Omega   MS11
--------------------
${    $%5.3f$$Omega$Hz  $% -6.4f$$MS11$}$
--------------------
```

End

Please notice that it is not necessary to quit the OSA90 editor to generate the report. More details can be found about the report generation feature in the "New Features" chapter of the OSA90 manual.

The file "lab5.rpt" provides a template for the above report block. You can incorporate it into your circuit file using the mark and copy features of OSA90 (Manual Chapter 2 - "Block Operations").

Lab 2 Nonlinear Memoryless Circuits: DC and Time-Domain Simulation

Objectives

This lab is designed to familiarize you with DC and time-domain simulations of nonlinear memoryless circuits (i.e., containing no dynamic components such as capacitance or inductance) using the built-in simulators of OSA90. It is also intended to refresh your theoretical knowledge of frequency-domain representation of periodic signals (Fourier series).

Circuit Diagram and Parameter Values

A circuit with the diode bridge (full-wave rectifier) will be used throughout this lab. It is shown in Fig. L2.

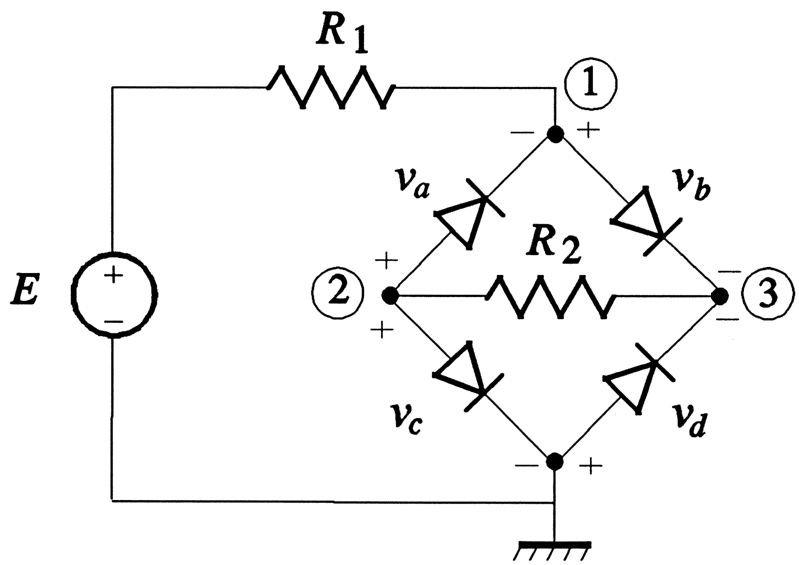

Fig. L2 A resistor-diode network.

The diodes are assumed memoryless, i.e., their capacitances are neglected. All diodes are identical and are fully described by the I-V characteristic given by the nonlinear equation

$$i_d = I_S[\exp(qv_d/(NkT)) - 1]$$

I_S, the saturation current, is such that the diodes are so called "10 mA" diodes: i_d is 10 mA for

$v_d = 0.7$ V (you will have to find the value of I_S). N, the emission coefficient, is 1, and T, the temperature, is 298°K. The parameters q and k in this equation are the electron charge and the Boltzmann constant, respectively. The expression kT/q is denoted by v_t and is called *thermal voltage*. Its value in this lab is constant and equals 25.67 mV. The aforementioned diode equation is implied when you define a diode within OSA90 using the keyword "DIODE". You can use the default values for all parameters except for I_S. Other parameters in the circuit are $R_1 = R_2 = 1$ kΩ, and various values for E.

Procedure

Preliminary Calculations

(1) Derive the expression to find the saturation current I_S for a "10 mA" diode.

The result is: I_S = ______________________________A.

(2) Derive the expression to find the conductance of a linearized (companion) model of the diode at a given $v_d = V_D$. The result is: g_d = ______________________________.

Create the OSA90 Input File

(1) Create the Model block. Describe the circuit elements and topology using "VSOURCE" (Manual Chapter 6 - "Circuit Models"), "RES" (Manual Chapter 8 - "Linear Elements"), and "DIODE" (Manual Chapter 7 - "Nonlinear Elements"). Define the nodal voltages v_1, v_2 and v_3 (with respect to ground) using Voltage Labels (Manual Chapter 6 - "Circuit Models"). Complete the circuit description using the CIRCUIT statement (Manual Chapter 6 - "Circuit Models"). Define all parameters either as constants or by labels.

(2) Create the Sweep block (Manual Chapter 9 - "Simulation"). Define the simulation type as "DC". Select E as the sweep parameter, initially with three values: 10 V, 1 V and -10 V. Define the responses to be simulated as the node voltages: v_1, v_2 and v_3.

(3) Show your input file to the instructor.

Simulation Experiment 1

(1) In the editor, you can click the left-hand mouse button to activate a pull-down menu. Parse the circuit file by selecting the option "Exit from editor" or by pressing the <F7> editor function key. Invoke the menu option "Display" by moving the cursor to highlight the option and click the left-hand mouse button or by pressing <D>. Select the "Xsweep" option by clicking the left-hand mouse button or by pressing <X>. Use the "Numerical" option to obtain the numerical output (Manual Chapter 9 - "Simulation"). Complete the following table.

TABLE I
VOLTAGE VALUES

E(V)	v_1(V)	v_2(V)	v_3(V)
10			
1			
—10			

(2) In the space below comment on the values of v_2 and v_3 at E = 10 V and E = —10 V.

(3) Show your results to the instructor.

Simulation Experiment 2

(1) Augment your input file to calculate the "output" voltage $V_{out} = v_3 - v_2$, and the conductance g_d of the linearized diode model for the diode denoted by the voltage v_b in Fig. L2. Resimulate the circuit and complete the following table.

TABLE II
OUTPUT VOLTAGE AND CONDUCTANCE

E(V)	V_{out}(V)	g_d(1/Ω)
10		
1		
—10		

(2) In the space below comment on your result

(3) Show your results to the instructor.

Simulation Experiment 3

(1) Define a new sweep parameter t (time) and sweep it through 51 points ($N = 50$) in one period of the sinusoidal source voltage

$$E(t) = E_m \sin(2\pi f t)$$

where $E_m = 10$ V and $f = 50$ Hz. Define the source voltage as a function of the sweep parameter t using the foregoing expression. Set up your sweep definition to display V_{out} together with the source voltage $E(t)$. Note: Use repetitive DC simulations to obtain results. In the space below sketch the time domain waveforms generated by the program on the screen.

(3) Show your results to the instructor.

Harmonic Analysis of the Rectified Voltage

(1) Invoke "Numerical" display option for the results of time domain analysis performed in "Simulation Experiment 3". Select V_{out} for the output. Edit the numerical output by removing any text and the values of time points, leaving only the numerical values of V_{out}. Save the results in the file "waveform.inc".

(2) Create a new simple input file for OSA90 consisting of the Expression and Sweep blocks. Define an array with the time domain samples saved in the file "waveform.inc". Use the DFT_TF built-in transformation to generate the frequency domain spectrum of V_{out}. Use the RI2MP built-in transformation to generate the magnitude spectrum of V_{out}. Display the magnitude spectrum using the "OSA90.Display>Array" menu option. In the space below sketch the results generated by the program on the screen.

(3) Show your results to the instructor.

Lab 3 Performance-Driven Design of Nonlinear Circuits

Objectives

This lab is designed to familiarize you with performance-driven design of nonlinear circuits using the built-in optimizers and simulators of OSA90. It is also intended to show you how to formulate and optimize user-defined responses derived from the basic built-in responses of OSA90.

Circuit Diagram and Parameter Values

A simple, single-stage amplifier will be used throughout this lab. It is shown in Fig. L3.

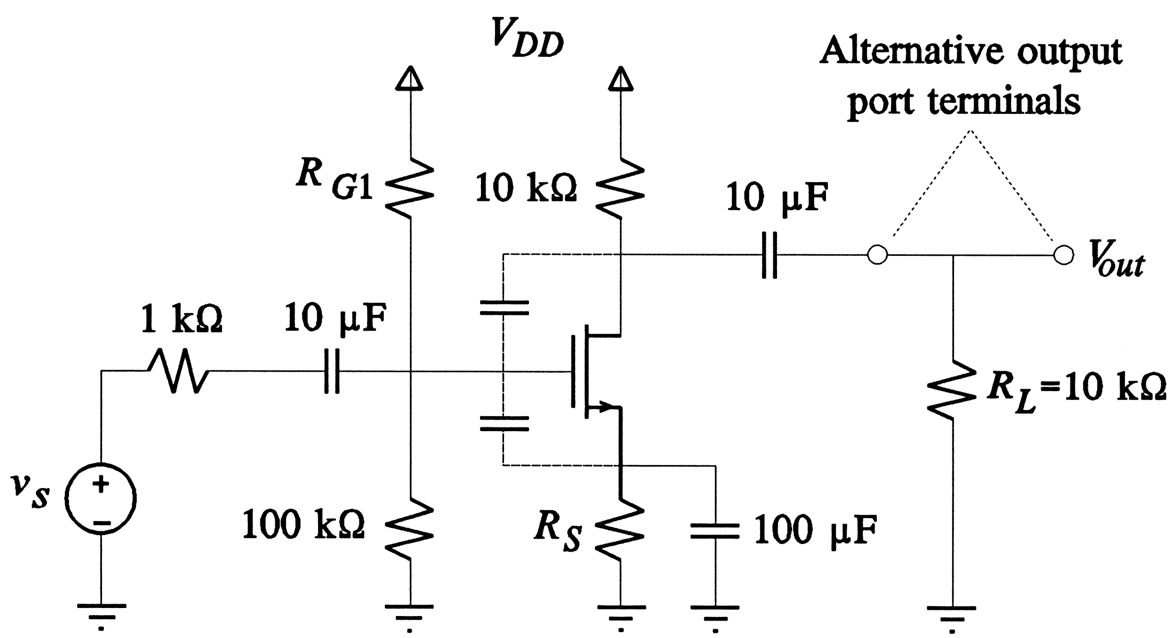

Fig. L3 A simple single-stage amplifier.

The amplifier is to be designed using a depletion type n-channel MOSFET with I_{DSS} = 4 mA and the pinch-off voltage V_P = —4 V. The parasitic capacitances of the FET are assumed as C_{gd} = 10 pF and C_{gs} = 0.1 μF. Other parasitic effects are neglected. The DC power supply is fixed at V_{DD} = 20 V. The resistances R_{G1}, R_S and the capacitance C_{gs} will be optimizable.

Procedure

Create the OSA90 Input File

(1) Define the Control block to use "Non_Microwave_Units".

(2) Create the Model block. Describe the circuit elements and topology using "RES", "CAP" and/or "PRC" (Manual Chapter 8 - "Linear Elements"), "FETM" (Manual Chapter 7 - "Nonlinear Elements"), "VSOURCE" (Manual Chapter 6 - "Circuit Models"), and "PORT" (Manual Chapter 6 - "Circuit Models"). Complete the circuit description using the CIRCUIT statement (Manual Chapter 6 - "Circuit Models"). The following hints should be helpful in setting up the circuit description.

(a) Use two separate DC bias sources: one for the gate and the other for the drain. Define a label such as V_{DD} and apply it to both sources. This will have the same effect as defining one source only. The advantage is that you will be able to calculate and optimize the gate and drain currents.

(b) Include the input source resistance in the "PORT" definition. Define the amplitude of the input voltage v_s through a label. This will allow you to sweep its value. The input source generates a sinusoidal voltage.

(c) The load resistance R_L may or may not be included in the output "PORT" definition. For some experiments it might be advantageous to include R_L in the 2-port while for others it might be better to define it as the terminating impedance (see Fig. L3).

(3) Create the Sweep block (Manual Chapter 9 - "Simulation"). Initially define one sweep set with the simulation type of "DC". No sweep parameters will be needed, so only numerical output will be available. Define the currents I_G and I_D through the gate and drain DC bias sources as the responses to be simulated.

(4) Show your input file to the instructor.

Experiment 1: Design of the FET Biasing Circuit

(1) Assume the following values for the unknown resistances: R_{G1} = 100 kΩ and R_S = 1 kΩ, and perform DC simulation. Your results are

I_G = ______________________ μA and I_D = ______________________ mA

(2) Define R_{G1} and R_S as optimization variables. Create the Specification block (Manual Chapter 11 - "Optimization"). Define one specification set of "DC" simulation type with the gate and drain currents as the responses to be optimized. Optimize the biasing circuit to get $I_G < 17\mu A$ and I_D = 1 mA. Use the ℓ_1 optimizer. Your results are

R_{G1} = ______________________ kΩ and R_S = ______________________ kΩ

(3) Resimulate the circuit and save the input file. Now, your results are

I_G = ______________________ μA and I_D = ______________________ mA

(4) Define R_{G1} and R_S as the sweep parameters and specify the sweep range from 100 kΩ to 1000 kΩ with a step of 100 kΩ, and 1 kΩ to 10 kΩ with a step of 1 kΩ, respectively. Define I_G and I_D as the response labels. Use 3D visualization to display the simulated responses.

(5) Show your results to the instructor.

Experiment 2: Small-Signal Simulation of the Amplifier

(1) Augment your Model block of the input file to calculate the voltage gain $V_{gain} = V_{out}/V_s$ where V_{out} is the voltage across R_L. V_{gain} can be calculated from the S parameters in the following way. If you select the reference resistance for the calculation of S parameters as the input resistance $R_{ref} = R_{in}$ and if you include R_L in the 2-port (i.e., the amplifier - see Fig. L3) then the voltage gain can be expressed as

$$V_{gain} = S_{21} \,/\, (1 - S_{22})$$

Alternatively, if R_L is excluded from the 2-port then (again for $R_{ref} = R_{in}$)

$$V_{gain} = S_{21} \,/\, [(1 - S_{22}) + (1 + S_{22})R_{ref}/R_L]$$

You can implement either of the two formulas, but be careful how you define R_L. In any case, remember that the S parameters are complex numbers. Since only the magnitude of V_{gain} will be of interest, define V_{gain} as the absolute value of the corresponding expression.

(2) Define a new sweep set in the Sweep block. The simulation type should be "AC". Define the exponential frequency sweep from 10 Hz to 10 kHz, 3 points per decade (Manual Chapter 9 - "Simulation"). Define the exponential sweep steps with "NEXP". Define an appropriate "RREF" and specify V_{gain} as the response to be simulated.

(3) Produce a logarithmic scale for frequency by typing "LOG" for the "Zoom scale" option "N x-axis ticks" (Manual Chapter 9 - "Log-Scale Display"). Complete the following table.

TABLE I
RESULTS FOR EXPERIMENT 2

f(Hz)	10	100	1000	10000
V_{gain}(V)				

(3) Show your results to the instructor.

Experiment 3: Optimization of the Frequency Response

Your previous experiment indicates that the 3dB bandwidth does not extend to 10 kHz. It is known that the parasitic capacitances of the FET are responsible for the high-frequency drop of the response. The question to answer through this experiment is: What is the largest allowable value of the parasitic capacitance C_{gs} for the 3 dB bandwidth to be extended to 10 kHz?

(1) Designate C_{gs} as the optimization variable. Modify the Specification block by commenting out the existing "DC" specification set and creating a new "AC" specification set. Define only one specification at 10 kHz in the form "Vgain > ...". In determining the value for this specification, assume that the frequency response in the pass band does not change much when C_{gs} is varied. Do not forget to specify "RREF".

(2) Optimize the circuit using the ℓ_2 optimizer. Your result is

C_{gs} < ________________________ nF

(3) Resimulate the circuit and save the input file. Now, the frequency response is tabulated as follows.

TABLE II
RESULTS FOR EXPERIMENT 3

f(Hz)	10	100	1000	10000
V_{gain}(V)				

(4) Show your results to the instructor.

Experiment 4: Large-Signal Simulation of the Amplifier

(1) Define a new sweep set in the Sweep block. The simulation type should be "HB". Define the amplitude of the input source voltage as the sweep parameter and specify the sweep range from 0.2 V to 2 V with a step of 0.3 V. Set the frequency of the input source to 1 kHz. The output port must now include R_L, so it should be excluded from the 2-port. Define MVout and PVout as the response labels. Define a graphical view to display simultaneously two periods of the output voltage waveforms corresponding to all values of v_s.

(2) Simulate the circuit and, in the space below, sketch the output voltage waveforms.

(3) *Optional:* Define the output power spectrum in an array POUT[0:N_Spectra] (N_Spectra is a built-in label, which in our case defaults to 4). Create an expression for the harmonic distortion h which is defined as the square root of the ratio of the power at all higher harmonics (P2 + P3 + ...) to the power at the fundamental frequency (P1). Include Pout and h in the sweep set.

(4) *Optional:* Complete the following table.

TABLE III
RESULTS FOR EXPERIMENT 4

v_s(V)	MVout[1](V)	Pout[1](mW)	h(%)
0.2			
0.8			
1.4			
2.0			

(5) Show your results to the instructor.

Lab 4 Device Modeling from Experimental Data

Objectives

This lab is designed to familiarize you with device modeling using the built-in optimizers and simulators of OSA90. At the completion of this lab you should be able to better understand the concepts of different models: small-signal, large-signal, linear, bias-dependent and nonlinear, and to apply optimization for extracting model parameters from measurement data.

Measurement Setup and Device Data

A microwave Schottky barrier FET (MESFET) has been measured in the setup shown in Fig. L4a.

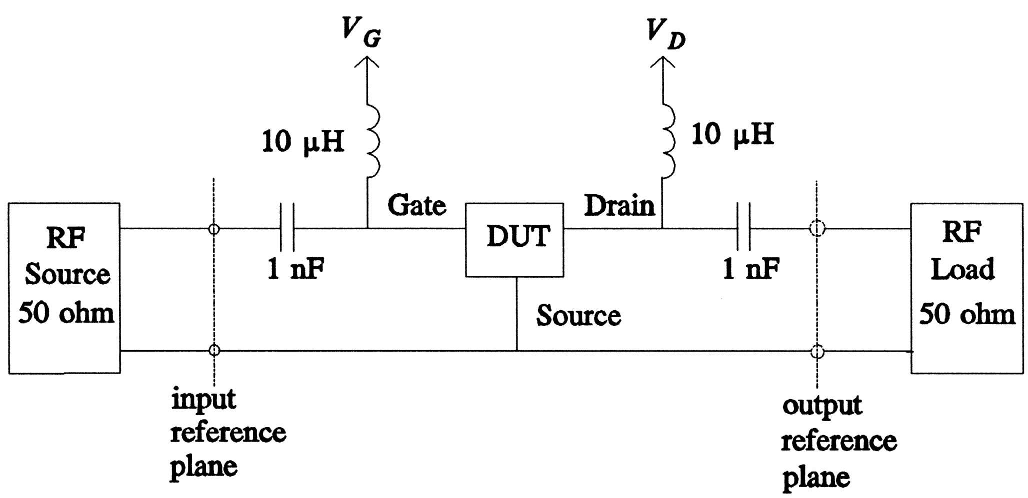

Fig. L4a MESFET measurement setup.

Measurements of S parameters with respect to 50 Ω were taken at the input and output reference planes for 17 frequencies in the range from 2 GHz to 18 GHz and for three values of the gate bias voltage V_G: 0, —1.74 and —3.1 volts. The drain bias voltage V_D was fixed at 4 V. The blocking capacitors of 1 nF and inductors of 10 μH were used to decouple the DC biasing circuit from the RF (or AC) circuit. The resulting S parameters are contained in an ASCII file generated by the

MicroCAT™ Test Executive system from Cascade Microtech. The file name is "cascade.dat" (see Appendix A).

Throughout this lab you will investigate two models for the FET. The first one is a simple small-signal model which may be suitable for simulating the FET around a specific operating point, determined and established separately. This model is shown in Fig. L4b. It consists of a voltage-controlled current source and four lumped, linear components. If the biasing of the FET is dropped out of the picture the resulting circuit is linear. The second model, shown in Fig. L4c, is a global, large-signal, nonlinear model which accounts for all nonlinearities, including biasing. In this lab you will use the Materka and Kacprzak built-in intrinsic nonlinear model (FETM) together with a built-in super-component "Extrinsic2", which consists of a number of linear elements connected internally in a predefined manner (Manual Chapters 7 and 8 - "Nonlinear Elements" and "Linear Elements").

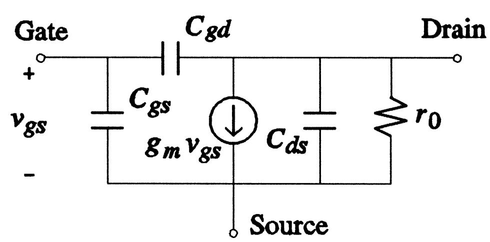

Fig. L4b A simple, small-signal FET model.

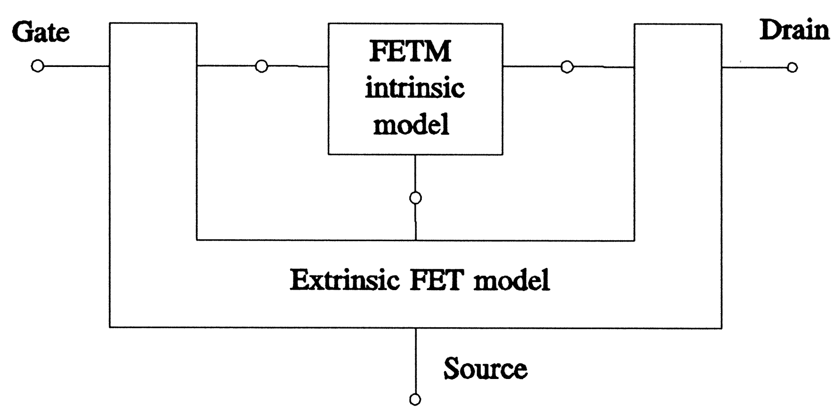

Fig. L4c Schematic diagram of a large-signal FET model.

Procedure

Experiment 1: Small-Signal FET Modeling

(1) Create the OSA90 Input File. It is recommended that you use default microwave units. Define the Model block. Describe the circuit of Fig. L4a with the model of Fig. L4b put in place of the device under test (DUT). Use "CAP" and "VCCS". You do not need to include the bias sources in this experiment, and you may exclude the blocking capacitors and inductors (if so, the overall circuit will simply reduce to that of Fig. L4b). Define the ports using "PORT" statements. Assume the following values for the model parameters.

$$C_{gs} = C_{gd} = C_{ds} = 1 \text{ pF}, \; g_m = 10 \text{ mA/V}, \; r_0 = 50 \; \Omega$$

All these parameters are to be optimizable. Be careful when you define the "VCCS". The orientation of the source in Fig. L4b is opposite to that in the Manual. Include r_0 in the definition of "VCCS". After completing the circuit description with the "CIRCUIT" statement, define the array "SP_SIM[8]" of simulated S parameters in the following order: Re(S_{11}), Im(S_{11}), Re(S_{21}), Im(S_{21}), Re(S_{12}), Im(S_{12}), Re(S_{22}), Im(S_{22}), which is the order of the data in each row in the file "cascade.dat".

(2) Define the array "SP_DATA[8]" and initialize it with the S parameters for $V_G = -1.74$ V at 2 GHz (copy the corresponding row, without the first number, from the data). Define the error functions in the array "SP_ERROR[8]" as the difference between the arrays "SP_SIM" and "SP_DATA". Close the Model block with the "End" statement.

(3) Define the Sweep block with one sweep set of the "AC" type to simulate the circuit at a single frequency of 2 GHz (only numerical output will be available). The responses of interest are "SP_SIM" and "SP_ERROR". Define the Specification block with one specification set of the "AC" type. Define a single specification of 0 for the whole array "SP_ERROR" at 2 GHz. Save your file.

(4) Perform simulation and then invoke the ℓ_1 optimizer. After optimization is completed, perform circuit simulation again, go to the file editor and save the back-annotated file under

a different name. Complete Table I.

TABLE I
S PARAMETERS

	S_{11}	S_{21}	S_{12}	S_{22}
Measurement Data				
Model Before Optimization				
Model After Optimization				

The extracted parameters of the model are

C_{gs} = _______pF, C_{gd} = _______pF, C_{ds} = _______pF, g_m = _______mA/V, r_0 = ________Ω.

(5) Show your results to the instructor.

Experiment 2: Checking the Validity of the Small-Signal FET Model

In this experiment you will investigate whether the model established in Experiment 1 is suitable for simulating device behaviour at a different operating point or in the whole frequency range.

(1) In the file saved after optimization performed in Experiment 1 replace the contents of the array "SP_DATA" with the S parameters for $V_G = -3.1$ V at 2 GHz (copy another row, without the first number, from the data). Repeat Step 4 of Experiment 1 without saving the resulting file. In the space below comment on the suitability of the model at different bias points.

(2) Retrieve the file saved after optimization performed in Experiment 1. Redefine your "SP_DATA" array to make it a two-dimensional array "SP_DATA[17,8]" and initialize it with the whole subset of the measurement data corresponding to $V_G = -1.74$ V. Do not forget to remove the first column of the data (frequencies). Define the new array "FREQs[17]" and initialize it with the frequencies at which the data was taken. If you use default microwave units enter the values in GHz but do not specify units (i.e., 2.0, 3.0, etc.). Now, the measurement data must be selected from the matrix "SP_DATA". You can do it by defining a frequency index "L", which will be swept, and selecting one row of the data for each specific value of "L" using the matrix function "ROW" (Manual Chapter 4 - "Array Function") to create a new array "SELECT_SP_DATA[8]". You will also need to redefine "SP_ERROR". Finally, extend the sweep and specification sets from a single frequency to all points by sweeping "L" from 1 to 17 and setting "FREQ" to the corresponding element of the array "FREQs". Repeat

Step 4 of Experiment 1. Save the files before and after optimization. In the space below comment on how the model fits the measurement data at 2 GHz and at all frequencies before and after optimization.

(3) Show your results to the instructor.

Experiment 3: Large-Signal FET Modeling

The preceding experiments indicate that a small-signal linear model is not capable of covering a large range of operating conditions, even for small-signal simulations. A linear model should at least have the parameter values that are bias-dependent (bias-dependent model). While such a model could be suitable for small-signal simulations under different bias conditions, only a true nonlinear model can be used as a global model. In this experiment you will use optimization to extract parameters of the Materka and Kacprzak model to fit simultaneously all measurement data.

(1) Retrieve either of the files used in Experiment 2, Step 2. Go to the secondary window of the editor by using the "Toggle file" command. Read in the file "lab4model.inc" (see Appendix B). Mark the whole contents of the file and copy it to the primary file. This is the model shown in Fig. L4c to be put in place of DUT in Fig. L4a and should replace the description of the small-signal model used in the preceding experiments. You may have to adjust the node numbers. You also need to include the blocking capacitors and inductors of Fig. L4a, and, using "VSOURCE", to define DC bias sources. When defining V_G, introduce a new array "VGS[3]" and a new bias index "K", which will be swept, and select the value of V_G using "VDC=VGS[K]". Adjust the number of rows of the matrix "SP_DATA" to accommodate all 51

lines of data from the file "cascade.dat". It is recommended that you edit that file (do not forget to delete the first column), save it under a different name such as "sp_data.inc" and use the "#include" directive in the input file. Now, to select the row of the data which corresponds to the Kth bias point and the Lth frequency you need the selector index "KL = (K - 1) * 17 + L". Finally, you need to define a double sweep in the Sweep and Specification blocks. Make "K" as the outer (first) sweep label and "L" as the inner (second) sweep label. Save your file and show it to the instructor.

(2) Perform simulation and view the error functions. Invoke the ℓ_1 optimizer. After optimization is completed, perform circuit simulation again, go to the file editor and save the back-annotated file under a different name. Again, view the error functions. Complete Table II with approximate values (orders of magnitude).

TABLE II
S-PARAMETER MATCHING ERRORS

	Average Error	Maximum Error
Model Before Optimization		
Model After Optimization		

(3) *Optional:* Define the parameter "NAME" for the drain bias source. Define the array "Ids_DATA[3]" and initialize it with the measured values of the current I_{ds} contained in the file "cascade.dat". Define the array "ID_ERROR" as the difference between the Kth element of "Ids_DATA" and the simulated DC current through the V_D source, multiplied by 50, the weighting factor for optimization. Include "ID_ERROR" in the Sweep and Specification blocks. Repeat Step 2 and comment on the result.

(4) *Optional:* Define a graphical view to display the fit of S_{21} for all frequencies and all bias conditions. Use continuous curves for displaying the simulated responses and "circles" to display measured data. You can define either a rectangular plot or polar plot. Sketch your results in the space below.

You can also generate an HPGL file for producing a hardcopy of the plot.

(5) Show your results to the instructor.

Appendix A for Lab 4

Data File "cascade.dat"

```
NOTE  FET S-parameter measurement for test with HarPE
NOTE  simulating the format by CASCADE MICROTECH
DATE  Nov 22, 1989
NOTE  the first bias
BIAS Vgs=0.0
BIAS Vds=4.0
BIAS Ids=0.177
S2RITABLE 17
 2.00E9     0.5952    -0.7405     -3.351      2.731    0.01574    0.02555     0.5699    -0.2847
 3.00E9     0.2782    -0.8682     -2.258      3.095    0.02697    0.02913      0.507    -0.3408
 4.00E9   0.003826    -0.8774     -1.332      3.088    0.03631    0.02841      0.448    -0.3896
 5.00E9    -0.2089    -0.8241    -0.6258      2.884    0.04307    0.02552     0.3927    -0.4319
 6.00E9     -0.367    -0.7442    -0.1137      2.599    0.04756    0.02184     0.3398    -0.4698
 7.00E9    -0.4831    -0.6561     0.2496      2.296    0.05027     0.0181     0.2879    -0.5043
 8.00E9    -0.5686    -0.5686     0.5039      2.002    0.05169    0.01465     0.2358    -0.5355
 9.00E9    -0.6318    -0.4851     0.6789      1.729    0.05217    0.01162     0.1829    -0.5634
10.00E9    -0.6788    -0.4068     0.7962      1.481    0.05197   0.009091      0.129    -0.5876
11.00E9    -0.7138    -0.3338     0.8709      1.258    0.05129   0.007052    0.07406    -0.6079
12.00E9    -0.7396    -0.2658      0.914       1.06     0.0503   0.005494    0.01827     -0.624
13.00E9    -0.7584    -0.2023     0.9333     0.8828     0.0491   0.004388   -0.03809    -0.6358
14.00E9    -0.7716    -0.1428     0.9347     0.7262    0.04779   0.003704   -0.09467    -0.6432
15.00E9    -0.7804   -0.08692     0.9226     0.5878    0.04644   0.003405    -0.1511    -0.6462
16.00E9    -0.7854   -0.03419     0.9002     0.4659    0.04512   0.003455     -0.207    -0.6448
17.00E9    -0.7874    0.01571     0.8703     0.3588    0.04386   0.003816    -0.2621    -0.6391
18.00E9    -0.7868    0.06306     0.8349     0.2652    0.04271    0.00445    -0.3159    -0.6294
ENDTABLE
NOTE  the second bias
BIAS Vgs=-1.74
BIAS Vds=4.0
BIAS Ids=0.08896
S2RITABLE 17
 2.00E9     0.7854    -0.5682     -2.667      1.616    0.01656     0.0352     0.5055    -0.2722
 3.00E9      0.577    -0.7439     -2.102      2.017    0.03109    0.04451     0.4452    -0.3342
 4.00E9     0.3559    -0.8389     -1.518      2.222    0.04608     0.0479     0.3823    -0.3876
 5.00E9     0.1491    -0.8697    -0.9818      2.265    0.05947    0.04668     0.3197    -0.4302
 6.00E9   -0.03086    -0.8564    -0.5255      2.195    0.07038    0.04242     0.2593    -0.4636
 7.00E9    -0.1815    -0.8155    -0.1548      2.058    0.07865    0.03644     0.2018    -0.4903
 8.00E9     -0.305    -0.7591     0.1372      1.885     0.0845    0.02968     0.1469    -0.5119
 9.00E9    -0.4053    -0.6947     0.3619      1.697    0.08829    0.02276    0.09385    -0.5294
10.00E9    -0.4864     -0.627     0.5311      1.507    0.09038    0.01605    0.04223    -0.5436
11.00E9    -0.5519    -0.5587     0.6554      1.322    0.09111   0.009802  -0.008427    -0.5546
12.00E9    -0.6049    -0.4915     0.7435      1.147    0.09076   0.004127   -0.05839    -0.5626
13.00E9    -0.6476    -0.4261     0.8028     0.9831    0.08957 -0.0009067    -0.1078    -0.5675
14.00E9    -0.6819    -0.3629     0.8389     0.8321    0.08775  -0.005273    -0.1567    -0.5695
15.00E9    -0.7093     -0.302     0.8566     0.6939    0.08546  -0.008975     -0.205    -0.5684
16.00E9    -0.7309    -0.2435     0.8596     0.5685    0.08284   -0.01203    -0.2527    -0.5643
17.00E9    -0.7476    -0.1873     0.8509     0.4553       0.08   -0.01447    -0.2995    -0.5571
18.00E9    -0.7601    -0.1332     0.8332     0.3539    0.07705   -0.01632    -0.3453    -0.5469
ENDTABLE
```

```
NOTE  the third bias point
BIAS Vgs=-3.1
BIAS Vds=4.0
BIAS Ids=0.0408
S2RITABLE 17
 2.00E9      0.835    -0.5042      -1.85      1.044    0.01928    0.04435     0.4908     -0.269
 3.00E9     0.6659    -0.6806     -1.509      1.335    0.03699    0.05763     0.4335     -0.333
 4.00E9     0.4752    -0.7942     -1.136      1.514    0.05638    0.06387     0.3715    -0.3904
 5.00E9     0.2856    -0.8521    -0.7732      1.587    0.07485    0.06398     0.3074    -0.4368
 6.00E9     0.1106    -0.8665    -0.4468      1.577    0.09087    0.05948     0.2439     -0.473
 7.00E9   -0.04368    -0.8502    -0.1682      1.511     0.1038    0.05193     0.1825    -0.5008
 8.00E9    -0.1762    -0.8135    0.06073      1.408     0.1136    0.04263     0.1235    -0.5221
 9.00E9    -0.2881    -0.7641     0.2435      1.285     0.1204    0.03251     0.0668    -0.5382
10.00E9     -0.382    -0.7073     0.3858      1.154     0.1247    0.02227    0.01213    -0.5501
11.00E9    -0.4604    -0.6467     0.4936       1.02     0.1268    0.01233   -0.04087    -0.5583
12.00E9    -0.5257    -0.5845     0.5727     0.8895     0.1272   0.002983   -0.09248    -0.5632
13.00E9    -0.5801     -0.522     0.6279      0.765     0.1261  -0.005595    -0.1429    -0.5649
14.00E9    -0.6251    -0.4603     0.6637      0.648     0.1239   -0.01331    -0.1922    -0.5636
15.00E9    -0.6624    -0.3997     0.6837     0.5396     0.1208   -0.02011    -0.2404    -0.5593
16.00E9     -0.693    -0.3405     0.6908     0.4402     0.1171     -0.026    -0.2875    -0.5521
17.00E9    -0.7179    -0.2829     0.6875     0.3497     0.1128   -0.03098    -0.3334    -0.5421
18.00E9    -0.7379     -0.227     0.6761     0.2681     0.1082   -0.03509     -0.378    -0.5294
ENDTABLE
```

Appendix B for Lab 4

Include File "lab4model.inc"

```
Extrinsic2  1 2 3 4 5
   RG: 0.0119   RD: 0.0006   RS: 0.33   CX: 1.5PF
   LG: ?0.1NH?
   LD: ?0.1NH?
   GDS: ?0.004?
   LS: ?0.1NH?
   CDS: ?0.1PF?;

FETM  1 2 3
   GAMMA: -0.1   KE: 0          SL: 0.2
   KG: -0.25     SS: 0          E: 2
   C10: 0.67PF   CF0: 0.023PF   KF: -0.12
   IG0: 5E-10    ALPHAG: 20     IB0: 8e-12
   ALPHAB: 1     VBC: 20        R10: 5.2
   KR: 0         K1: 1          C1S: 0.0048PF
   IDSS: ?0.2?
   VPO: ?-4?
   TAU: ?3PS?;
```

Lab 5 Statistical Design Centering

Objectives

This lab is intended to familiarize you with characterization of statistical variations of circuit parameters, Monte Carlo analysis and yield-driven design of nonlinear circuits using OSA90.

Procedure

Experiment 1: Monte Carlo Analysis of a Simple Voltage Divider

(1) Create the OSA90 input file for a simple voltage divider consisting of two resistors $R_1 = R_2 = 50\ \Omega$ connected in series. The input DC voltage source of 1 V is applied to both resistors and the output voltage is taken across R_2. Create the Model block using "VSOURCE", "RES", "VLABEL" and "CIRCUIT" statements. Define statistical variation of R_1 and R_2 using "UNIFORM" distribution (Manual Chapter 12 - "Uniform Distribution") within ±5% interval with respect to the nominal values ("TOL=5%"). Create the MonteCarlo block (Manual Chapter 12 - "MonteCarlo Block") with one sweep set of the DC type. Specify "N_outcomes = 100" and the output voltage as the response label.

(2) Invoke the OSA90.MonteCarlo menu option. After Monte Carlo analysis is finished view the Run Chart of the simulated output voltage. Select the numerical "Output form" in the Run Chart pop-up window. Create an array "OUTPUT[100]" to accommodate all outcomes of the MonteCarlo analysis.

Determine the minimum and maximum output voltages using "AMIN" and "AMAX" array function (Manual Chapter 4 - "Array Functions"). The spread of the output voltage is

min V_{out} = ________V, max V_{out} = ________V and variation of V_{out} = ± ________%

(3) Show your results to the instructor.

Experiment 2: Statistical Design Centering of a Small-Signal Amplifier (Fig. L5)

(1) Retrieve the input file generated during Experiment 3 of Lab 3 "Performance-Driven Design of Nonlinear Circuits". Assume that the nominal value of C_{gs} is 10 nF, which satisfies the specification $V_{gain} > 3.5$. Define statistical variations of "NORMAL" distribution (Manual Chapter 12 - "Normal Distribution") with the standard deviation of 5% ("SIGMA=5%") for all four resistors R_{G1}, R_{G2}, R_D and R_S, and for the capacitor C_{gs}. You should have 5 statistical variables altogether.

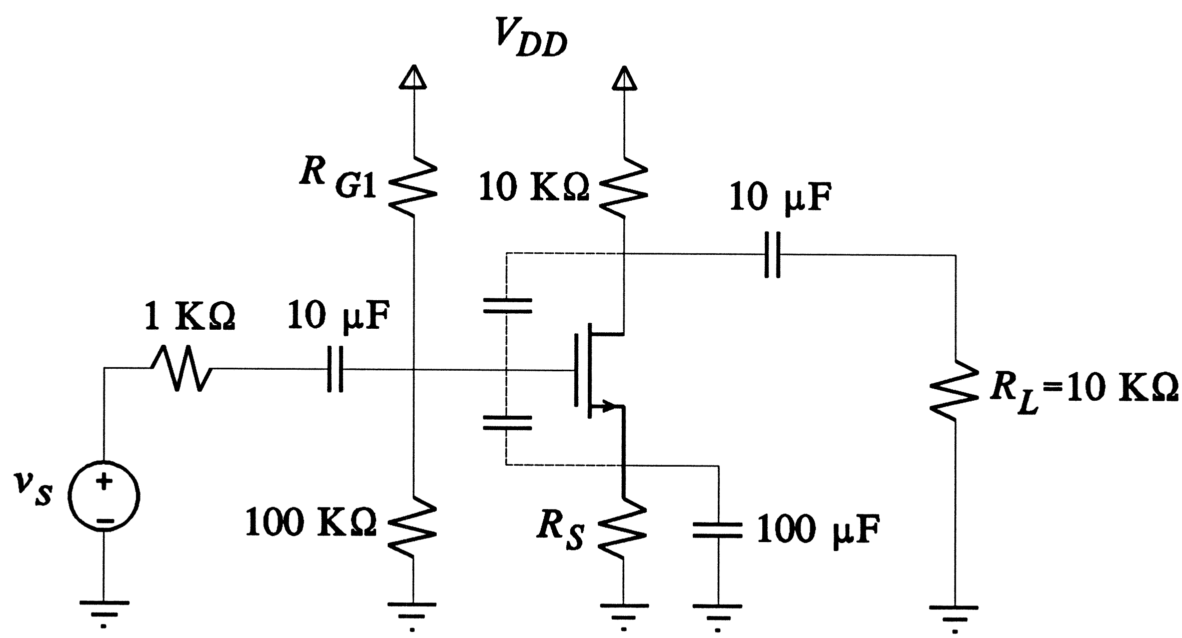

Fig. L5 A simple single-stage amplifier.

(2) Create the MonteCarlo block with one sweep set of the AC type by copying the existing sweep set from the Sweep block. Leave V_{gain} as the only response label. Use "N_outcomes = 100". Define design specifications $V_{gain} > 3.5$ and $V_{gain} < 5$ for yield estimation.

(3) Perform Monte Carlo analysis and invoke the OSA90.MonteCarlo.Xsweep menu option to simultaneously view all the responses corresponding to the 100 circuit outcomes generated according to the statistical distribution (statistical response). In the space below sketch the statistical response together with design specifications.

Initial yield = __________%

(4) Show your results to the instructor.

(5) Define the nominal values of R_{G1} and R_S as optimization variables. Modify the Specification block to define design specifications consistently with the MonteCarlo block (V_{gain} > 3.5 and V_{gain} < 5 for all 10 frequencies swept exponentially from 10 Hz to 10 kHz).

(6) Invoke the OSA90.Optimize menu option. Select "yield" as the optimizer. Once "yield" is selected, the pop-up window allows you to choose the number of outcomes to be involved in yield optimization. Select 50 outcomes. Accept defaults for all other options. Perform yield optimization. Then go back to the file editor and change the number of outcomes for Monte Carlo analysis to 300. Perform Monte Carlo analysis and view the statistical response. In the space below sketch the statistical response after yield optimization and report final yield and the values of optimization variables.

Yield after optimization = __________%

The values of the optimization variables after yield optimization are

R_{G1} = ____________________________kΩ and R_S = ______________________________kΩ

(7) Show your results to the instructor.

Lab 6 Simulation and Optimization Using Datapipe

Objectives

This lab is designed to familiarize students with simulation and optimization using the Datapipe.

Circuit Diagram and Parameter Values

Consider the LC transformer circuit shown in Fig. L6. Use OSA90 to carry out minimax optimization on the magnitude of the input reflection coefficient using all reactive components as optimization variables. Use 21 uniformly spaced points in the frequency range 0.5 - 1.179 (rad/s).

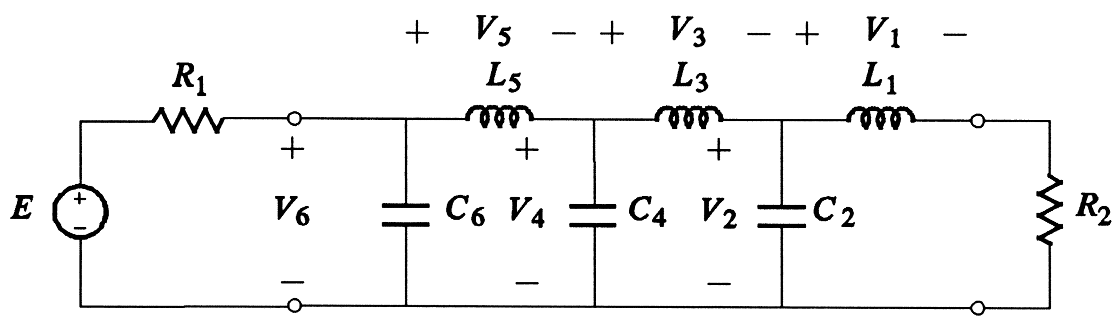

Fig. L6 A lumped element LC transformer.

The suggested starting point is

$$L_1 = C_2 = L_3 = C_4 = L_5 = C_6 = 1.$$

$E = 3$ V, $R_1 = 3\ \Omega$ and $R_2 = 1\ \Omega$. The solution, rounded to 3 decimal places, is

$$L_1 = 1.041$$

$$L_3 = 2.341$$

$$L_5 = 2.937$$

$$C_2 = 0.979$$

$$C_4 = 0.780$$

$$C_6 = 0.347$$

Procedure

Part I: Optimization Using the OSA90 Built-in Simulator

(1) Create the circuit file.

(a) Create a Model block to describe the circuit. Define the elements R_2, L_1, L_3, L_5, C_2, C_4 and C_6. Specify the parameters of L_1, L_3, L_5, C_2, C_4 and C_6 as optimization variables (Manual Chapter 6 - "Elements" and Chapter 4 - "Optimization Variables"). For example,

```
L1: ?1?;
IND n1 n2 L = L1;
```

or,

```
IND n1 n2 L = ?1?;
```

(b) Define the input port including both the voltage E and the resistance R_1 (Manual Chapter 6 - "Ports"):

```
PORT node1 node2 NAME=portname V=E R=R1;
```

(c) Complete the circuit description using the CIRCUIT statement (Manual Chapter 6 - "The Circuit Statement").

(d) Assign the initial value (0.5) to the label "Omega".

(e) Terminate the Model block with the "End" keyword.

(f) Define the Sweep block (Manual Chapter 9 - "Sweep Block"). This will enable OSA90 to display to magnitude of the reflection coefficient MS11 over the specified range of frequency, graphically or numerically (Manual Chapter 6 - "Small-Signal Responses" and Chapter 9 - "OSA90.Display Menu Option"):

```
Sweep
   AC: Omega: from 0.5 to 1.179 n=20, FREQ=(Omega/(2*PI)), RREF=3, MS11;
End
```

(g) Define the Specification block (Manual Chapter 10 - "Specification Block"). The goal for optimization is to minimize the magnitude of the reflection coefficient MS11 over the specified range of frequency.

```
Specification
   AC: Omega: from 0.5 to 1.179 n=20, FREQ=(Omega/(2*PI)), RREF=3,
       MS11=0;
End
```

(2) Save the circuit file you have created: press <F8>, enter the file name, and press <ENTER>.

(3) Perform minimax optimization.

(a) Exit the editor and parse the circuit file: press <F7>.

(b) Press <O> to choose the "Optimization" option.

(c) From the pop-up window, toggle to the "Optimizer" option and choose "Minimax". Change the "Number of iterations" option to "999".

(d) Press <ENTER> twice to start the optimization process.

(4) After the optimization is completed, press <F> to return to the circuit file. The optimizable parameters should have been updated with the minimax solution.

(5) Save the optimized circuit file for later comparison: press <F8>, enter a new file name and press <ENTER>.

DO NOT FORGET TO SAVE YOUR CURRENT FILE BEFORE THE NEXT STEP.