

Lab 0 – ADS 2009 Tutorial

*Due at the beginning of your week#3 lab session

INTRODUCTION:

This tutorial consists of four exercises intended to introduce you to the ADS 2009 software and give you some practice at using its basic features. Carefully follow the step-by-step instructions in each exercise and save all design files (.dsn) and display files (.dds).

Please bring these files with you to the first lab session (week#3) in room 329 EE East. At the beginning of lab, the Teaching Assistants will ask each student to open their LAB #0 files in order to receive credit.

Here is a list of the ADS files you should have saved and bring to lab:

Exercise 1:

- oneport_L.dsn
- oneport_L.dds
- oneport_C.dsn
- oneport_C.dds

Exercise 2:

- fish.dsn
- fish.dds


Exercise 3:


- ex3_before_OPTIM.dsn
- ex3_after_OPTIM.dsn
- ex3_after_OPTIM.dds

Exercise 4:

- ex4_MESFET.dsn
- ex4_MESFET.dds

ADS Documentation:

ADS 2009 comes with extensive documentation organized into an HTML-based catalog that can be accessed from the Start menu. Go to **Start → All Programs → Advanced Design System 2009 →  ADS Documentation**. This will open the catalog's top menu.

This tutorial contains numerous references to  **ADS Documentation** that offer thorough explanation of ADS features and functions. These references are included for your benefit and it is recommended that you follow them to gain further insight and understanding. You will be asked to use ADS for simulations in your homework assignments, lab activities, and for the design & layout of your final project, a balanced 2.4 GHz amplifier. The more comfortable you are with ADS, the better off you will be.

Some Background Information on ADS

Source: <http://www.home.agilent.com/agilent/product.jsp?nid=34346.0.00&lc=eng&cc=US>

Advanced Design System is the leading electronic design automation software for RF, microwave, and signal integrity applications. ADS pioneers the most innovative and commercially successful technologies, such as X-parameters* and 3D EM simulators, used by leading companies in the wireless communication & networking and aerospace & defense industries. For WiMAX™, LTE, multi-gigabit per second data links, radar, & satellite applications, ADS provides full, standards-based design and verification with Wireless Libraries and circuit-system-EM co-simulation in an integrated platform.

Key Benefits of ADS

- Complete, integrated set of fast, accurate and easy-to-use system, circuit & EM simulators enable first-pass design success in a complete desktop flow
- Application-specific DesignGuides encapsulate years of expertise in an easy-to-use interface
- Leading industry partners support ADS exclusively or months earlier than others

EXERCISE 1

Objective:


Perform a one-port simulation of a 795 pH inductor and an ideal transmission line with $Z_0 = 50$, $E = 45$ degrees at 10 GHz. Replicate the design of Figure 1.1. Simulate the design and produce the data display shown in Figure 1.2. Save this display as a display template.

Replace the inductor with a 318 fF capacitor and change port 2 termination impedance from 50Ω to $1 \text{ G}\Omega$ (\approx open circuit). Simulate the design and use the display template to view the simulation data.

Procedure:

1. Begin ADS by double-clicking the **Advanced Design System 2009** icon on the desktop. Alternatively, you may launch the program from the start menu.
2. If the **Getting Started with ADS** window opens, click **Create a New Project**. Otherwise, go to the **Advanced Design System 2009 (Main)** window and choose **File → Create a New Project**.
3. Name your new project "**LAB0**". Specify an appropriate file location such as your network P:\ drive or a USB flash drive (example: **P:\EE432\LAB0**).
 - NOTE: By specifying the file location P:\EE432\LAB0, you are telling ADS to create a project called "LAB0" at the location P:\EE432\. If there is already a "LAB0" folder at that location, ADS will ask you to specify a different location.

Change the standard length unit to millimeters and click OK. ADS will then create a project directory named "**LAB0_prj**" in the specified location. This folder will serve as the directory for all files associated with your "LAB0" ADS project.

- For more on how ADS uses projects to organize information:
 [ADS Documentation](#): under **Getting Started**, select **Quick Start** and click **Using Projects**.
4. After creating the "LAB0" project directory, an untitled schematic window will open along with the **Schematic Wizard**. The schematic wizard can be used to quickly set up a design or simulation by automatically placing some necessary components into your blank schematic window. However, you should also know how to find these components in the appropriate **Component Palette** along the left side of the workspace.

Using the Schematic Wizard:

- a) Select **Simulation**, click Next.
- b) Select **Linear Circuit, 2-port**, click Next.
- c) Select **I will design my own circuit**, click Next.
- d) Select **Linear Frequency Sweep**. The Description field below will then list the components that will be automatically inserted into your untitled schematic window. Click Finish.

- For more on creating and using designs:
 [ADS Documentation](#): under **Getting Started**, select **Quick Start** and click **Using Designs**.

5. Your schematic window will now contain the **S-Parameter Simulation Controller**, **Two Port Terminations**, and a **Display Template**. Alternatively, these components can be found in their corresponding **Component Palette** (the vertical button-bar on the left) by selecting the appropriate palette from the dropdown box directly above it.

To place an **S-Parameter Simulation Controller**—needed for all S-parameter simulations—select **Simulation-S_Param** from the component palette dropdown box. The S-Parameter Simulation Controller is the first button on the top right of this palette (labeled **S P**). To add this component, click the **S P** button once and move your cursor over to the workspace. Click on the workspace to place the component. Press the Escape key to exit the place component mode.

Scattering parameter (S-parameter) measurements are taken from two or more port terminations. The ADS name for these components is **Term**. The schematic wizard has already added two **Term** components to your design, but they are also available in the **Simulation-S_Param** palette. The number specified for each termination component (e.g. **Num=1** or **Num=2**) corresponds to the port number of the S-Parameter simulation controller.

- For more information on the S-Parameter Simulation controller:
 [ADS Documentation](#): under **Simulation**, select **Analog/RF → S-Parameter Simulation** and click **About S-Parameter Simulation**.

The schematic wizard has also placed a **Display Template** in your design workspace. A Display Template can be used to store data display items (such as pre-configured plots and other graphical items) that you may choose to use for future simulations. For now, you can delete this component. After simulating the circuit of Figure 1.1 and replicating the data display of Figure 1.2, you will create a display template and use it to view the simulation data of a similar circuit.

S-Parameter Simulation

Linear Frequency Sweep

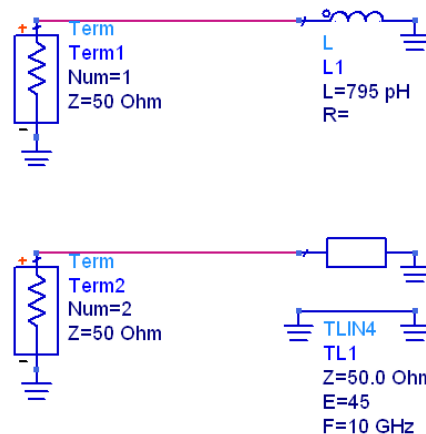
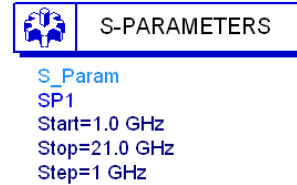







Figure 1.1 One-port simulation of inductor and ideal transmission line.

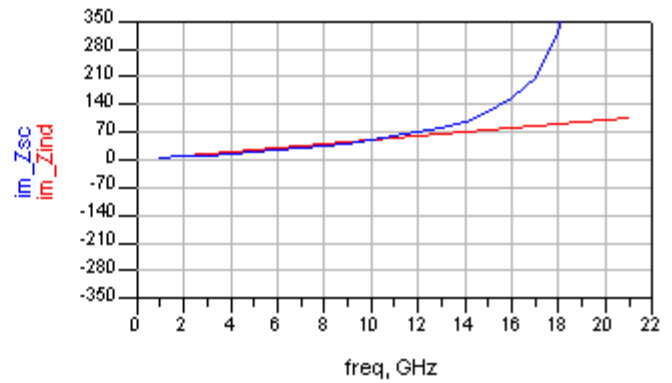
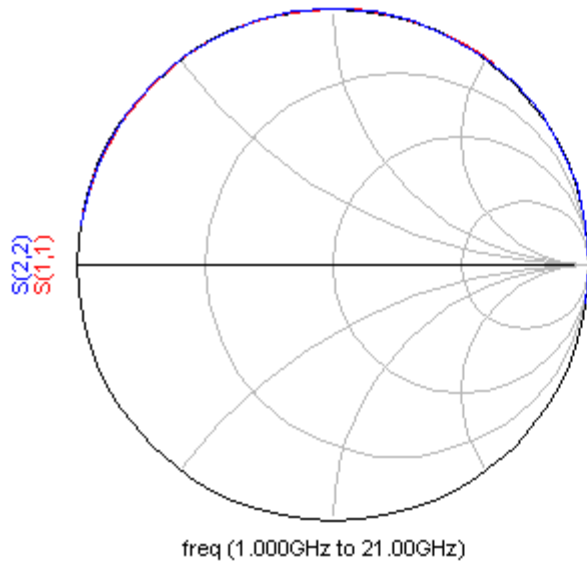
- Replicate the schematic design of Figure 1.1. To place the **Inductor**, go to the component palette dropdown box and select the **Lumped-Components** palette. Locate the inductor (**L**) on the left and place one in your workspace. Once you have placed the inductor, double-click it to open the component properties window and change the inductance value to **L = 795 pH**. The inductance value can also be changed by simply clicking on the value listed under the inductor in the schematic window.
 - For details on each of the Lumped Components:
 [ADS Documentation](#): under **Components**, select **Analog/RF** → **Introduction to Circuit Components**. There you will see a list of all Lumped Components.
- Locate the **Ideal Transmission Line** by selecting the **TLines-Ideal** palette from the dropdown box. Place the **TLIN4** component in your workspace. Change the electrical length to **E = 45 degrees** and the reference frequency to **F = 10 GHz**.
 - For details on each of the Transmission Line Components:
 [ADS Documentation](#): under **Components**, select **Analog/RF** → **Distributed Components**. Scroll down to find a list of all Transmission Line Components.
- As you assemble the circuit, use the **Insert Wire**  and **Insert GROUND**  buttons (along the top of the screen) as needed. Pressing **Ctrl+R** will rotate a selected component(s).
- If you have not already done so, change the **Start & Stop frequencies** and the **frequency Step** increment of the S-Parameter simulation controller.

10. When you have finished building the circuit of Figure 1.1, **Save** your design as “**oneport_L**”. ADS designs are given the file extension “**.dsn**” and are stored in the **networks** folder of your project directory.
11. Simulate your design by selecting **Simulate** → **Simulate** from the top menu, or by pressing **F7** on the keyboard. A status window will open while ADS executes the simulation. When the simulation is complete, the resulting data will be stored in an ADS **dataset** file and a blank **Data Display window** will open. The data display window allows you to view and analyze a dataset using a variety of graphical and numerical diagrams.
 - For more on using the data display window:
 -  [ADS Documentation](#): under **Design and Display**, select **Data Display** and click **Data Display Basics**.
12. Replicate the data display of Figure 1.2. Display items (plots, equations, text, etc.) may be inserted into your display page by choosing the **Insert** menu at the top or by right-clicking on the blank display page and selecting **Insert**. The vertical bar on the left also has buttons for inserting specific display items.

To insert the first plot, choose **Insert** → **Plot**. Move your cursor over the blank page and click to insert. The **Plot Traces & Attributes** window will open. Here you can select the plot type (rectangular, polar, etc.) and select the dataset variables to be plotted. Select **Smith Chart** by clicking on the appropriate button graphic.

13. The next step is to select variables to include in the plot. On the left side of the Plot Traces & Attributes window you will see the **Datasets and Equations** dropdown box followed by a list of dataset variables. To add variables as traces on the plot, simply highlight them and click **Add**. This will add them to the list of traces on the right.

For the first plot, add the S-parameter variables **S(1,1)** and **S(2,2)**. Click OK. The Smith chart plot of S_{11} and S_{22} should now appear in your display window. Before you create the remaining plots, you must use your simulation dataset to calculate additional circuit parameters.



Eqn $Z_{ind}=50*(1+S(1,1))/(1-S(1,1))$

Eqn $im_Z_{ind}=imag(Z_{ind})$

Eqn $Z_{sc}=50*(1+S(2,2))/(1-S(2,2))$

Eqn $im_Z_{sc}=imag(Z_{sc})$

Eqn $Zl=0$

Eqn $theta=45*(pi/180)$

Eqn $omega=2*pi*10.0E+9$

Eqn $length=theta/omega$


Eqn $Z_{sc_calc}=50*(Zl+j*50*tan(2*pi*freq*length))/(50+j*Zl*tan(2*pi*freq*length))$

freq	Zind	Zsc	Zsc_calc
1.000 GHz	-277.6a + j4.995	-999.2a + j3.935	0.0000 + j3.935
2.000 GHz	-444.1a + j9.990	666.1a + j7.919	0.0000 + j7.919
3.000 GHz	0.0000 + j14.99	1.332f + j12.00	0.0000 + j12.00
4.000 GHz	1.776f + j19.98	2.665f + j16.25	0.0000 + j16.25
5.000 GHz	0.0000 + j24.98	2.665f + j20.71	0.0000 + j20.71
6.000 GHz	-1.776f + j29.97	1.776f + j25.48	0.0000 + j25.48
7.000 GHz	0.0000 + j34.97	3.553f + j30.64	0.0000 + j30.64
8.000 GHz	-3.553f + j39.96	3.553f + j36.33	0.0000 + j36.33
9.000 GHz	0.0000 + j44.96	0.0000 + j42.70	0.0000 + j42.70
10.00 GHz	7.105f + j49.95	3.553f + j50.00	0.0000 + j50.00
11.00 GHz	-3.904f + j54.95	-8.319f + j58.54	0.0000 + j58.54
12.00 GHz	0.0000 + j59.94	9.780f + j68.82	0.0000 + j68.82
13.00 GHz	4.614f + j64.94	-5.798f + j81.59	0.0000 + j81.59
14.00 GHz	-9.938f + j69.93	34.86f + j98.13	0.0000 + j98.13
15.00 GHz	10.65f + j74.93	25.73f + j120.7	0.0000 + j120.7
16.00 GHz	0.0000 + j79.92	-16.40f + j153.9	0.0000 + j153.9
17.00 GHz	-24.13f + j84.92	103.6f + j208.3	0.0000 + j208.3
18.00 GHz	-25.55f + j89.91	-201.9f + j315.7	0.0000 + j315.7
19.00 GHz	13.49f + j94.91	-270.8f + j635.3	0.0000 + j635.3
20.00 GHz	14.20f + j99.90	-50.00 + j816.6E15	0.0000 + j816.6E15
21.00 GHz	44.72f + j104.9	-112.9f - j635.3	0.0000 - j635.3

Figure 1.2 Data display for one-port simulation of inductor & transmission line.

14. Insert the equations of Figure 1.2 into your data display. To do this, choose **Insert** → **Equation**. Click on the display window where you want to put your equation. This will open the **Enter Equation** window. Type out the first equation as it appears in Figure 1.2. You can insert variables into the equation as needed using the variable list on the right. Alternatively, you can type a variable's name directly into the equation line as long as you type the name exactly as it appears on the variable list.

When you are done, click OK. Do this for each of the remaining equations shown in Figure 1.2.

- For more on using expressions and functions in ADS:
 [ADS Documentation](#): under **Design and Display**, select **Functions** → **Measurement Expressions** and click **Introduction to Measurement Expressions**.

15. Plot the imaginary components of Z_{sc} and Z_{ind} versus frequency on a rectangular plot, as shown in Figure 1.2. To add **im_Zsc** and **im_Zind** traces to your plot, go to the **Datasets and Equations** dropdown box in the Plot Traces & Attributes window and select **Equations**. The variable list should now contain the nine parameters you defined in your equations.

Highlight **im_Zsc** and press the **>>Add Vs...>>** button. You will be prompted to **Select Independent Variable**. From the dropdown box choose your dataset (**oneport_L**), then highlight **freq** and click OK. Repeat this procedure for the **im_Zind** trace.

Under the **Plot Options** tab, specify the **Min**, **Max**, and **Step** of the **x-axis** to match Figure 1.2. Click OK.

16. Insert a table plot (**List**) of data values vs. frequency for **Zind**, **Zsc**, and **Zsc_calc**. Note the accuracy of your calculated Z_{sc} data (Z_{sc_calc}) compared to your simulation data for Z_{sc} .

The **Complex Data Format** of a variable listed in the table can be changed by double-clicking the variable at the top of the column. Your complex data values are probably displayed in **Mag/Degrees** by default but change the format to **Real/Imaginary** as in Figure 1.2.

You can also change the number format (e.g. the number of significant figures) by going to the Plot Options tab in the Plot Traces & Attributes window. Double-click on the plot to open the Plot Traces & Attributes window and change the format to **Engineering** with **3 Significant Figures**.

17. Go to **File → Save** and save your data display in your project directory under the name “**oneport_L**”. ADS data display files are given a “.dds” file extension and should be stored in your project directory (...\\LAB0_prj\\).
18. Now go to **File → Save As Template**. This saves your formatted data display items (plots, equations, etc.) as a template that can be used to easily view and analyze other datasets. Save your display template under the name “**exercise1_template**” and save it in the **user** folder.
19. Now return to your “oneport_L” schematic window and open a **New Design**. Call it “**oneport_C**”. Create this new design in a **New Schematic Window** and un-check the Schematic Wizard box. Select all components in your “oneport_L” schematic window then copy and paste them into your new “oneport_C” design.
20. In your “oneport_C” schematic window, replace the 795 pH inductor with a **318 fF capacitor**. The capacitor can be found in the **Lumped-Components** palette. Change the value of the **termination impedance** at **port 2** from 50 Ω to **1 GΩ**. This allows you to treat the termination as an open-circuit (infinite impedance).
21. **Simulate** the circuit. A blank **data display window** will open for your new simulation dataset. In this display window, go to **Insert → Template** and select “**exercise1_template**” located in the user folder. Click OK and all of the data display items from your template will be inserted into the new display window.

Alternatively, you can include a **display template component** in your **schematic window**, as was originally done by the schematic wizard. In this scenario, the display template would be automatically inserted into the new data display window that opens after a simulation.
22. **Save** your “**oneport_C**” display before closing the display window.

EXERCISE 2

Objective:

Simulate the simple transmission line circuit of Figure 1.3 and plot S_{11} on a Smith chart.

Procedure:

1. Open a new schematic design and call it “fish”.
2. Replicate the transmission line network of Figure 1.3. **Save** your design.
3. Simulate the circuit and plot S_{11} on a **Smith Chart**.
4. Based on this plot, it should be clear as to why we had you call this design “fish”.
5. **Save** your data display under the same name, “fish”.

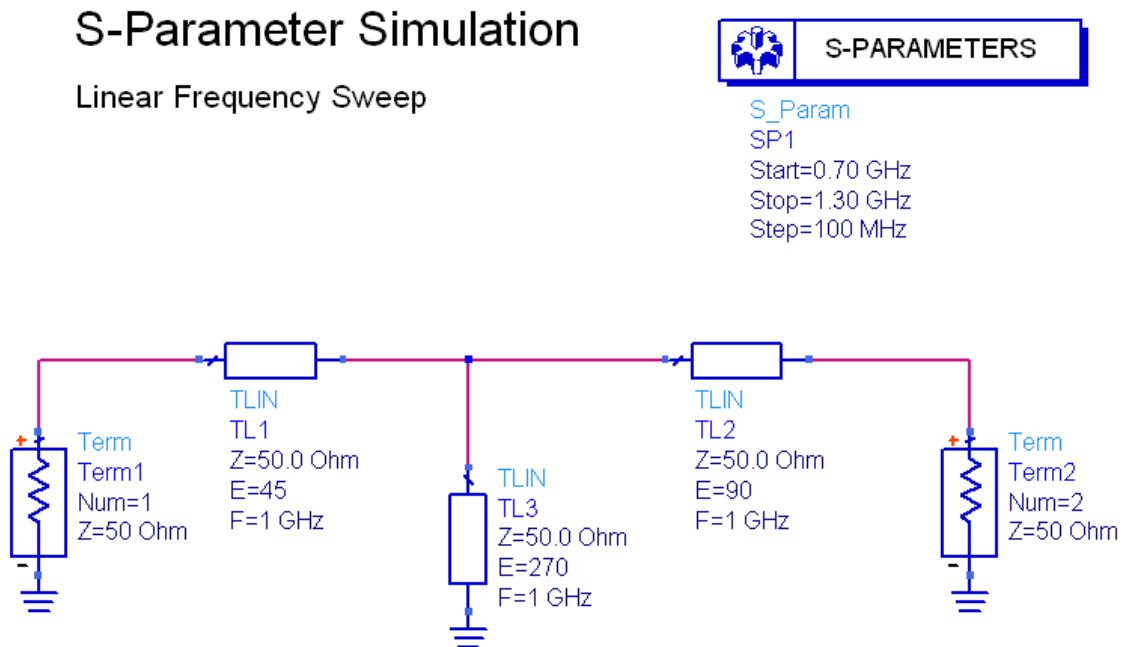


Figure 1.3 Simple transmission line circuit.

EXERCISE 3

Objective:

Use the ADS Nominal Optimization tool to optimize a 100 mm section of microstrip transmission line to have a 50Ω impedance at 1.5 GHz. Simulate the circuit and verify $|S_{11}| \approx 0$ at 1.5 GHz.

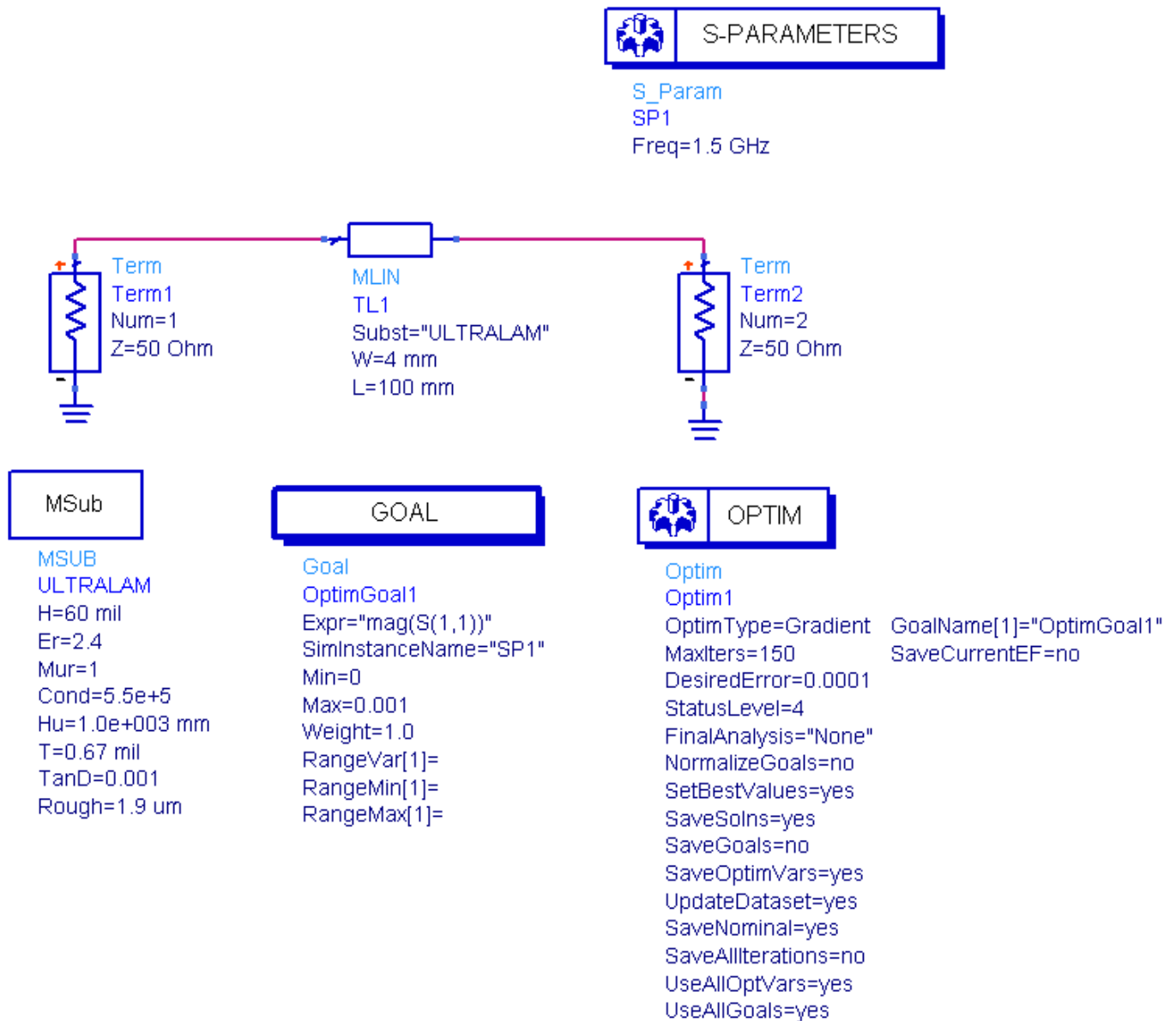





Figure 1.4 Circuit for Nominal Optimization of microstrip transmission line.

Procedure:

1. Replicate the circuit of Figure 1.4. The Microstrip Line component (**MLIN**) and Microstrip Substrate (**MSUB**) are found in the **TLines-Microstrip** component palette. The Optimization Controller (**Optim**) and Optimization Goal (**Goal**) are found in the **Optim/Stat/Yield/DOE** palette.


To set up the S-parameter simulation controller (**S_Param**), double-click on it and change the frequency **Sweep Type** to **Single Point** and specify **1.5 GHz**. Under the **Display tab**, uncheck the Start, Stop, and Step parameters and check the Freq parameter only. The frequency parameter should now be the only parameter listed below the component in the schematic window.

Change the parameter values under each component to match figure 1.4 exactly. As was already mentioned, a component parameter can be changed in the component properties window or by simply clicking the parameter value in the schematic window (if listed under the component) and typing in the new value.


- For details on Microstrip Line (**MLIN**) and Substrate (**MSUB**):
 [ADS Documentation](#): under **Components**, select **Analog/RF** → **Distributed Components**. Scroll down to find a list of all Microstrip Components.
 - For details on Setting Optimization Goals (**GOAL**):
 [ADS Documentation](#): under **Simulation**, select **Analog/RF** → **Tuning, Optimization, and Statistical Design** and click **Performing Nominal Optimization**. Scroll down to Setting Optimization Goals.
 - For details on Setting Optimization Job Parameters (**OPTIM**):
 [ADS Documentation](#): under **Simulation**, select **Analog/RF** → **Tuning, Optimization, and Statistical Design** and click **Performing Nominal Optimization**. Scroll down to Setting Optimization Job Parameters.
2. Once you have replicated the circuit of Figure 1.4, you must initiate a **Nominal Optimization** of the MLIN **width** parameter (**W**) in order for ADS to determine the “optimum” value that will satisfy the design goal of $|S_{11}| < 0.001$ at **1.5 GHz**.

Begin by double-clicking the MLIN component. Under Select Parameter, highlight **W=4 mm** and click the **Tune/Opt/Stat/DOE Setup** button. Under the Optimization tab, change the **Optimization Status** to **Enabled**. Specify the Minimum and Maximum Values to be **1 mm** and **20 mm**, respectively, then click OK → OK. You are now ready to perform the optimization.

3. Before you proceed with optimization, **Save** your design as “**ex3_before_OPTIM**”.

4. Now **Simulate** the circuit. The simulation status window will open and display the progress of the optimization. When the optimization process finds a value of **W** that satisfies the design goal, the simulation will terminate and a new data display window will open. Close the data display window and return to the schematic window.
5. Go to **Simulate** → **Update Optimization Values**. This updates the value of **W** to the new optimized value generated by the nominal optimization. Your optimized value will probably be near 4.4 mm. If not, inspect your schematic to make sure it matches Figure 1.4.
 - For a complete overview of ADS Nominal Optimization:
 [ADS Documentation](#): under **Simulation**, select **Analog/RF** → **Tuning, Optimization, and Statistical Design** and click **Performing Nominal Optimization**.

You should simulate the circuit to test your results, but first you must clear the optimization feature from your design. Double-click the **MLIN** component (as you did in step 2) and change the **Optimization Status** of parameter **W** back to **Clear**. Now that there is no longer a parameter enabled for optimization, you may delete the Optimization Controller (**OPTIM**) and Goal component (**GOAL**) from your design.

6. **Simulate** the circuit from 300 MHz to 3 GHz with a 100 MHz frequency step. You will have to change the **sweep type** of the S-Parameter simulation controller back to **Linear** and define Start, Stop, and Step frequencies.
7. In the data display window, create a table plot (**List**) of **S₁₁** vs. **frequency**. Make sure your values of **S₁₁** are displayed in Mag/Degrees so that you may easily compare the magnitudes.
8. **Save** your data display as “**ex3_after_OPTIM**” and save your schematic design (with the optimized width value) under the same name.
9. You might notice that your value of **|S₁₁|** at 1.5 GHz does not directly satisfy the specified design goal of **|S₁₁| < 0.001**. Instead, you probably have a value of **|S₁₁| = 0.006**, or as high as 0.015. This is because the **Desired Error** of the Optimization Controller was specified to be 0.0001. Based on the method by which the Optimizer calculates **Error Factor**, the value **|S₁₁| = 0.006** does satisfy your design goal + desired error.
 - For details on how the Error Factor is formulated:
 [ADS Documentation](#): under **Simulation**, select **Analog/RF** → **Tuning, Optimization, and Statistical Design** and click **Summary of Optimizers**. In the Contents frame, click **Error-Function (EF) Formulation**.

You may wonder if setting the Desired Error to zero will generate an optimized width parameter that is closer to $|S_{11}| = 0$ at 1.5 GHz. You are encouraged to experiment with this for yourself, but it appears that $|S_{11}| = 0.006$ may be the closest that the random-type optimizer can get to meeting the ideal goal of zero reflection ($|S_{11}| = 0$) at 1.5 GHz.

EXERCISE 4

Objective:

Simulate the small signal model for a high frequency MESFET (Figure 1.5). Perform an S-parameter simulation and use the resulting data to calculate the hybrid matrix (H-parameters), Rollett stability factor (K), and Mason's invariant (U). Replicate the data display of Figure 1.6.

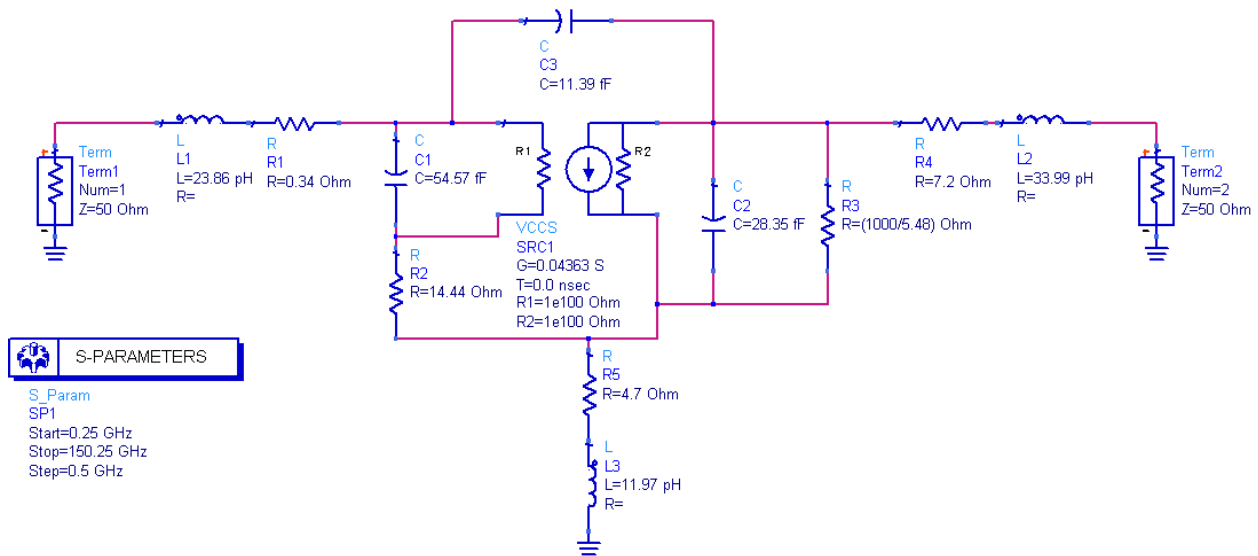

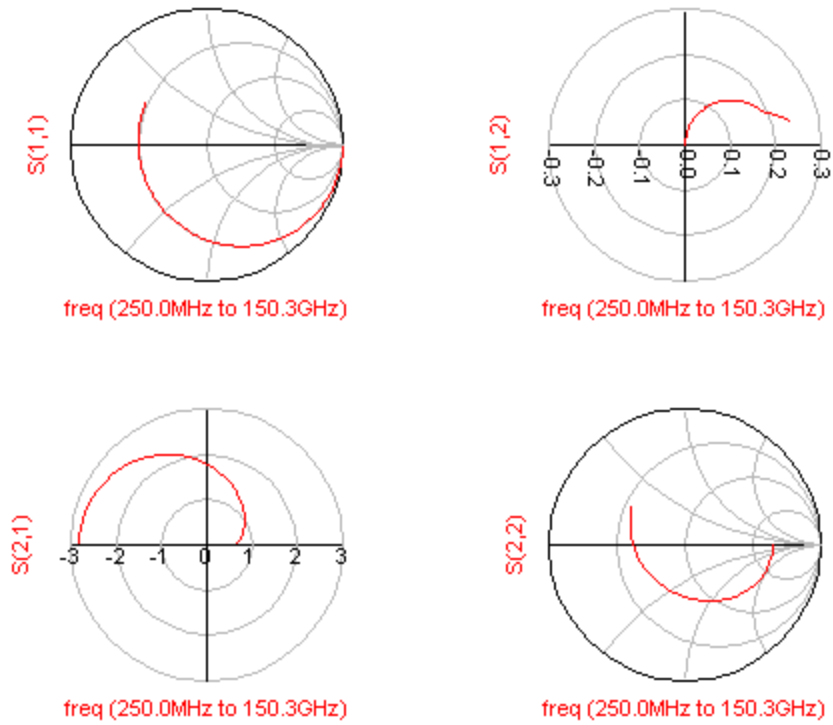


Figure 1.5 Small signal model of a high frequency MESFET device.

Procedure:

1. Replicate the circuit of Figure 1.5. The Voltage Controlled Current Source (VCCS) is available in the **Sources-Controlled** palette.
2. Before simulating the design, **Save** it as "**ex4_MESFET**".
3. **Simulate** the circuit from 250 MHz to 150.25 GHz.
4. Replicate the data display page of Figure 1.6. Insert equations to calculate the hybrid matrix (H-parameters), the Rollett stability factor (K), and Mason's invariant (U). The meaning and significance of these parameters will be covered later in the course; for now, they are presented only to give you practice in using ADS.
 - For details on using expressions and functions in ADS:
 - **ADS Documentation:** under **Design and Display**, select **Functions** → **Measurement Expressions** and click **Introduction to Measurement Expressions**.
 - Also scroll down to see a list of mathematical operators (such as **)

- For details on the **stoh(·)** function:
 -  [ADS Documentation](#): under **Design and Display**, select **Functions** → **Measurement Expressions**.
 - The **stoh(·)** function is listed under **S-Parameter Analysis Functions for Measurement Expressions**.
- 5. Plot the functions U and H_{21} on a logarithmic scale and plot K on a linear scale, as shown. To insert logarithmic expressions (such as $10*\log(U)$) click the **Advanced** button under the Plot Type tab of the **Plot Traces & Attributes** window. This opens the **Advanced Trace Entry** window in which you can enter any expression and add it to the list of traces for that plot. Press the **Help** button to see a list of ADS measurement expressions.
- 6. When you have successfully replicated the data display of Figure 1.6, **Save** your display as "**ex4_MESFET**".



$$\text{Eqn newh} = \text{stoh}(S)$$

$$\text{Eqn } K = (1 - \text{abs}(S(1,1))^2 - \text{abs}(S(2,2))^2 + \text{abs}(\text{delta})^2) / (2 * \text{abs}(S(1,2)) * S(2,1))$$

$$\text{Eqn } \text{delta} = S(1,1) * S(2,2) - S(1,2) * S(2,1)$$

$$\text{Eqn } U = \text{abs}(S(2,1) / S(1,2) - 1)^2 / (2 * K * \text{abs}(S(2,1) / S(1,2)) - 2 * \text{real}(S(2,1) / S(1,2)))$$

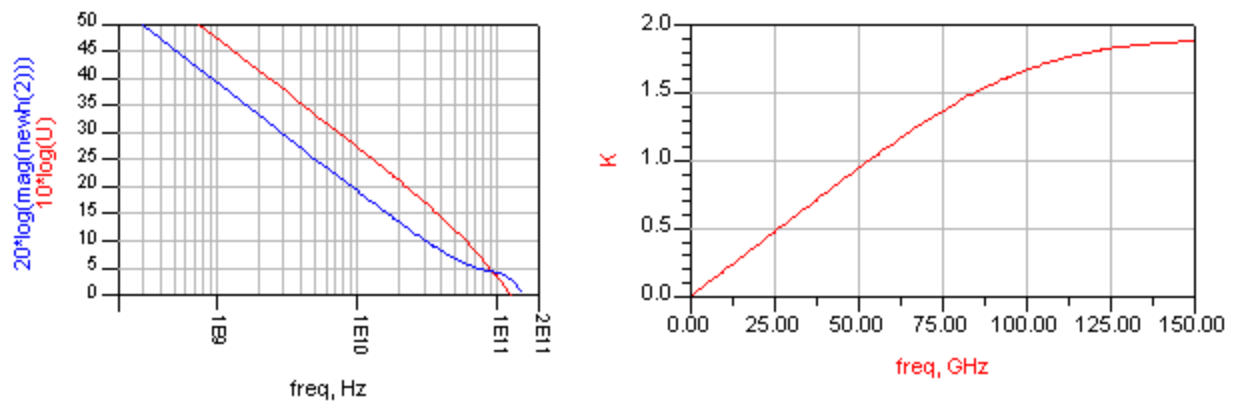


Figure 1.6 Data Display for MESFET small signal model simulation.