

Getting Start with Agilent - ADS Software.

Dr. Haim Matzner, Shimshon Levy and Dafna Ackerman.

July 2008.

CONTENTS

0.1	Objectives	3
1	Background	5
1.1	The Searching Option	5
2	Experiment Procedure	9
2.1	Required Equipment	9
2.2	T Attenuator Simulation	9
2.3	Transmission Line Simulation	12

0.1 Objectives

Upon completion of the study, the student will become familiar with the following topics:

1. Understanding the ability of the Agilent-ADS software.
2. Designing basic lumped circuits by applying the Agilent-ADS software.

1. BACKGROUND

Agilent ADS is a leading software for designing and optimizing RF and microwave circuits. When applying the software one can draw a circuit, choose components from the software data base or data-sheet of many manufacturers, simulate and optimize the circuit and draw output S parameters or other graphs of the design.

1.1 The Searching Option

For finding a specific component, click on the 'Display Component Library List' button, as shown in Figure 1.

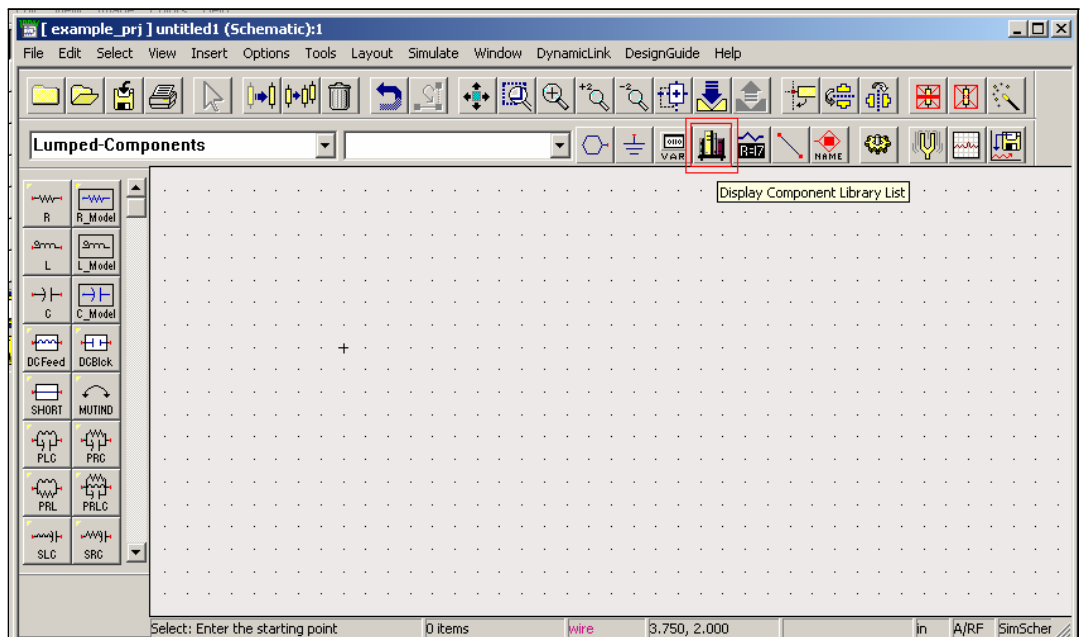


Figure 1 - 'Display Component Library List' button.

A component library window should be opened. Press on the 'Find Component(s)' button, as shown in Figure 2.

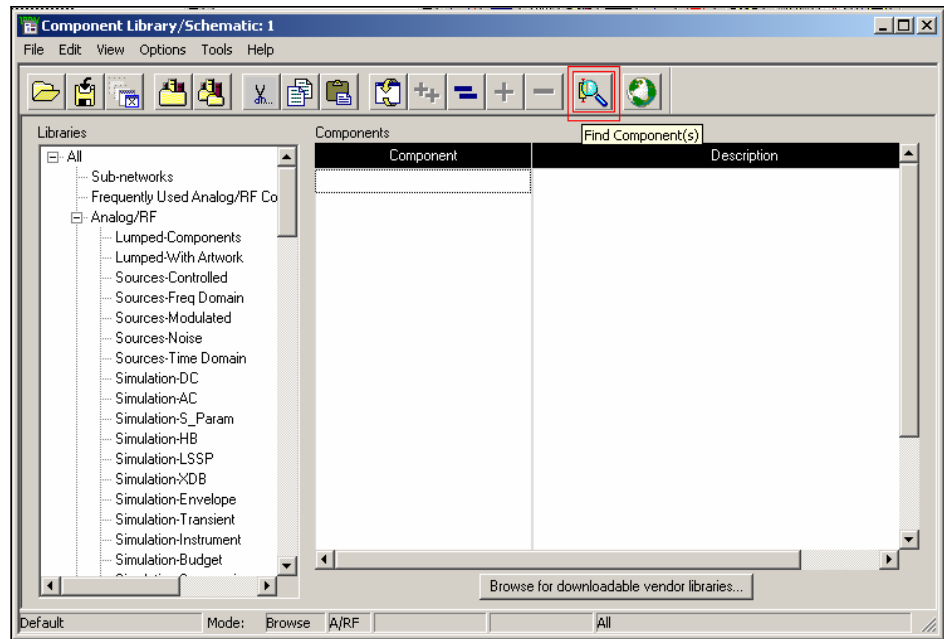


Figure 2 - The component library window

A 'Find' window should be open, as shown in figure 3. Type the name of the desired component in the 'Find' tab. Pay attention that the 'Look in:' tab is marked as 'All' (searching option for all ADS directories).

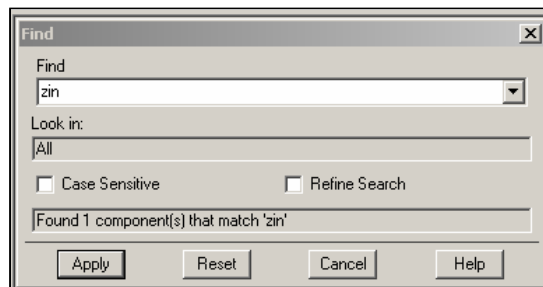


Figure 3 - Find component.

Another way for searching a component is to write its exact name in the components tab, as shown in Figure 4.

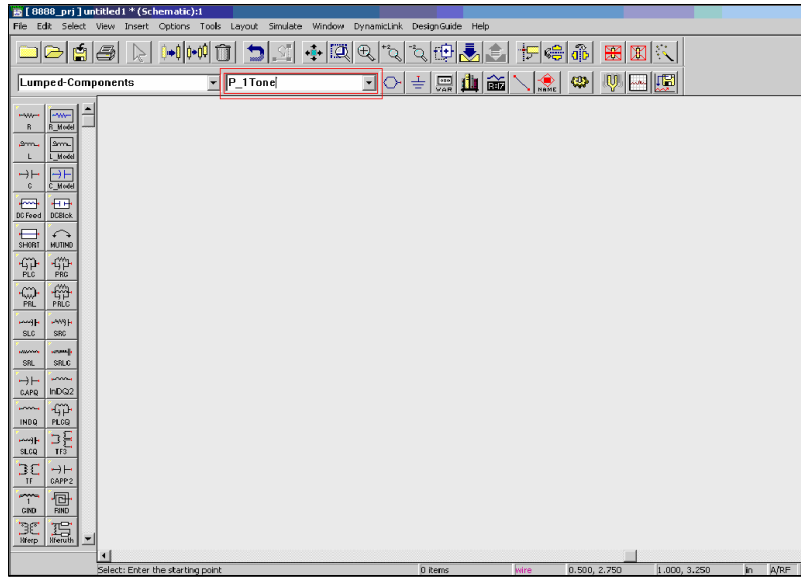


Figure 4 - Searching a component by its exact name.

A manual searching option is also available by searching the component in the components categories, as shown in Figure 5.

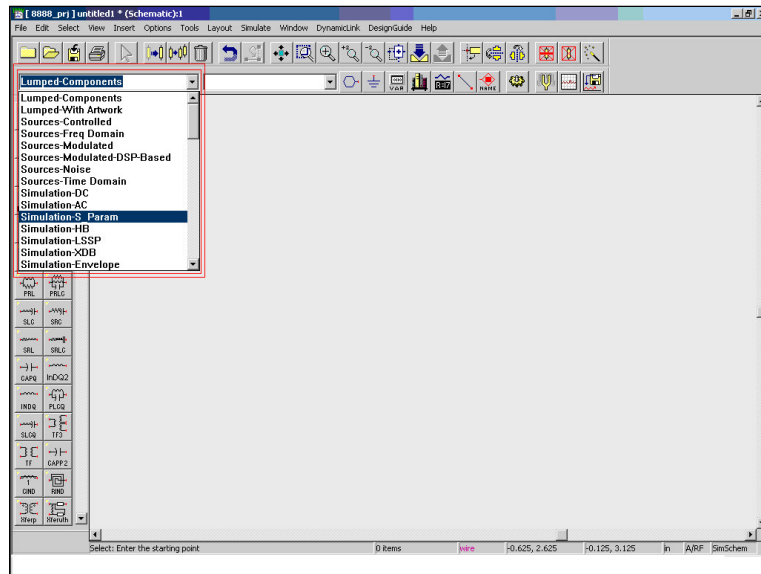


Figure 5 - Components categories.

2. EXPERIMENT PROCEDURE

2.1 Required Equipment

ADS software installed on a PC computer.

2.2 T Attenuator Simulation

1. Double click on the ADS icon. The main window will open. Open a new project, by pressing **File - New Project**, as shown in Figure 1.

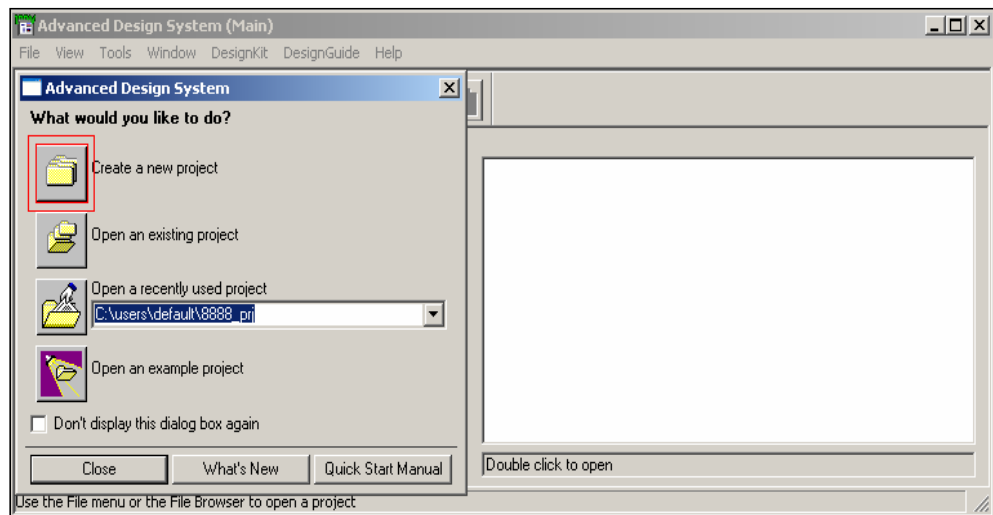


Figure 1 - Creating a new window.

Fill in a project name and change the units for the project to mm, as shown in Figure 2.

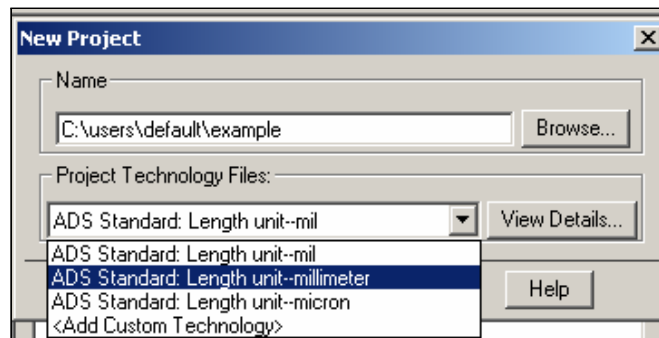


Figure 2 - New project.

2. New schematic window will be open. Draw a T attenuator, as shown in Figure 3.

3. A 10 dB T-Attenuator, which is also a matching network between 50Ω system and 75Ω system. All the components are taken from different categories in the left side of the window (see table-1).

4. Set the frequencies range for $1MHz - 500MHz$, by double clicking on the **S-PARAMETERS** icon. Set the parameters of the substrate (FR4 in our case) by double clicking the **MSub** icon.

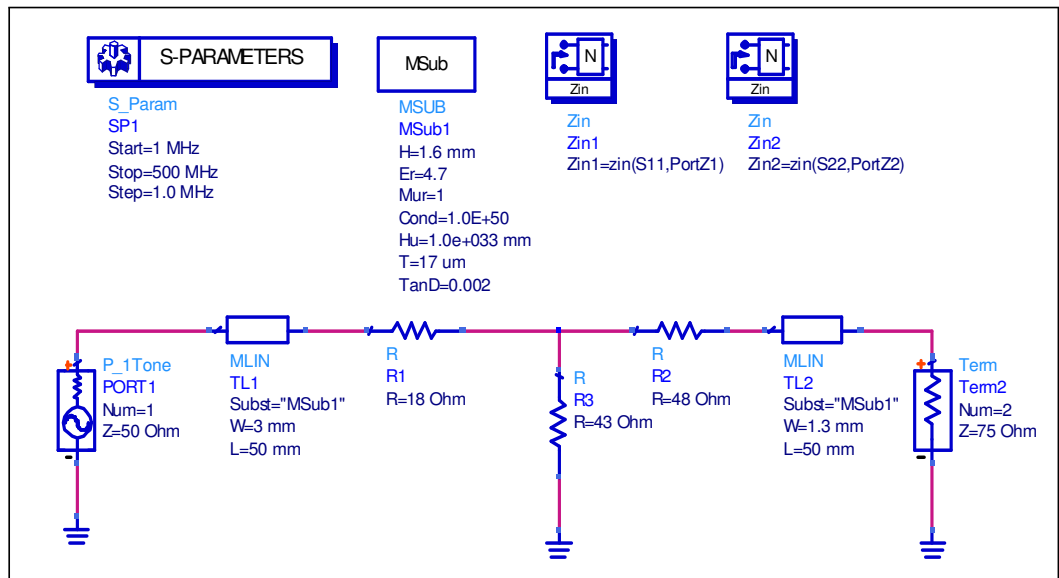


Figure 3 - T - Attenuator simulation

Component Name	Description	Categories List
Msub	Microstrip substrate	TLines-Microstrip
Terminaton	Term	Simulation S_param
MLIN	Microstrip Line	TLines-Microstrip
Zin	Port Input Impedance	Simulation S_Param
R	Resistor	Lumped Components
S-Parameters	S-parameter Simulation	Simulation S_Param
P_1Tone	Power Source	Sources-Freq Domain

Table-1, list for Figure 3

5. Simulate the circuit by clicking on the yellow tooth wheel icon. A debug window will be opened, as shown in Figure 4. Pay attention that there is not any error in the Summary section.

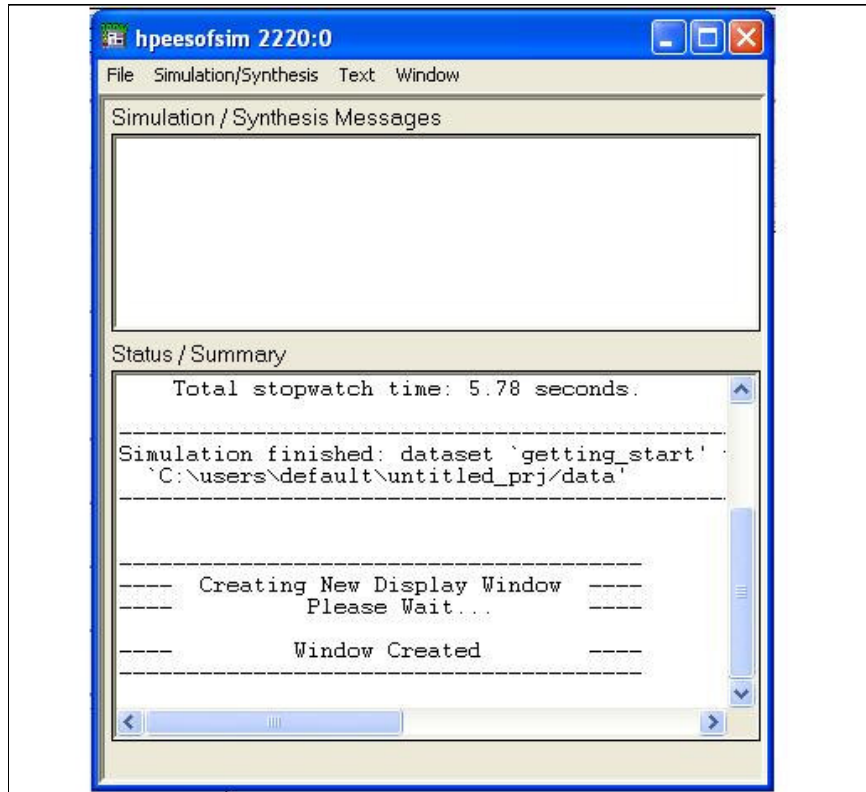


Figure 4 - Debug window.

6. After performing the simulation the results window open, choose the 'Rectangular Plot' button, as shown in Figure 5.

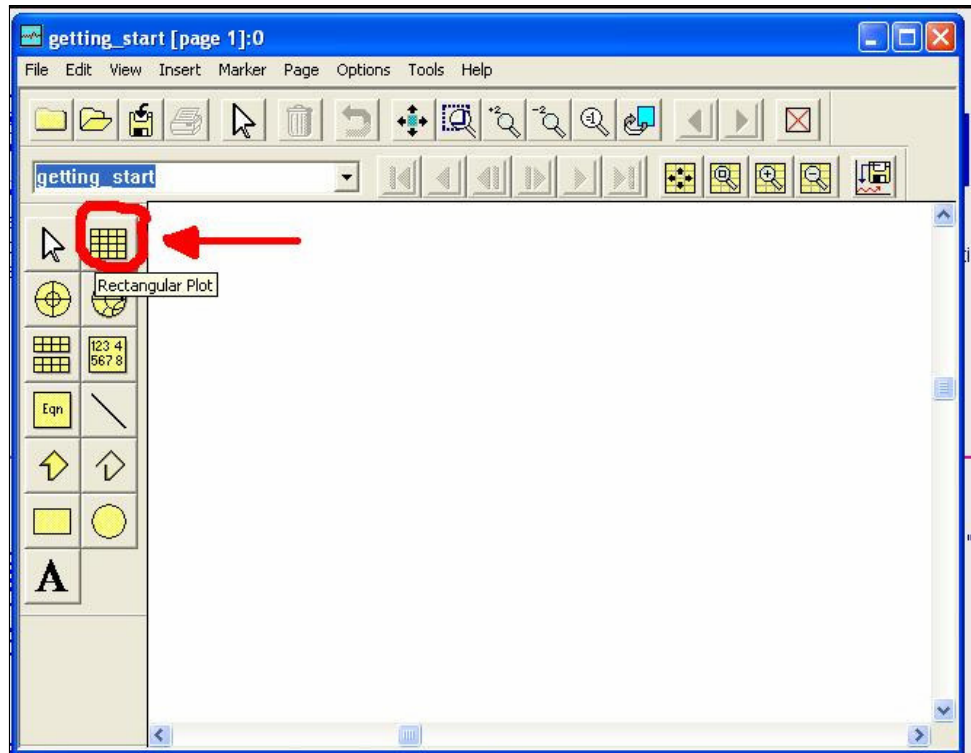


Figure 5 - 'Rectangular Plot' button.

An options window will be open, choose a parameter (e.g. $S(2,1)$), click on the button 'Add Vs...', choose the data units (in this case, dB), select the independent variable (in this case, freq) and press 'OK'. You should get, in the results window, a graph, as shown in Figure 6.

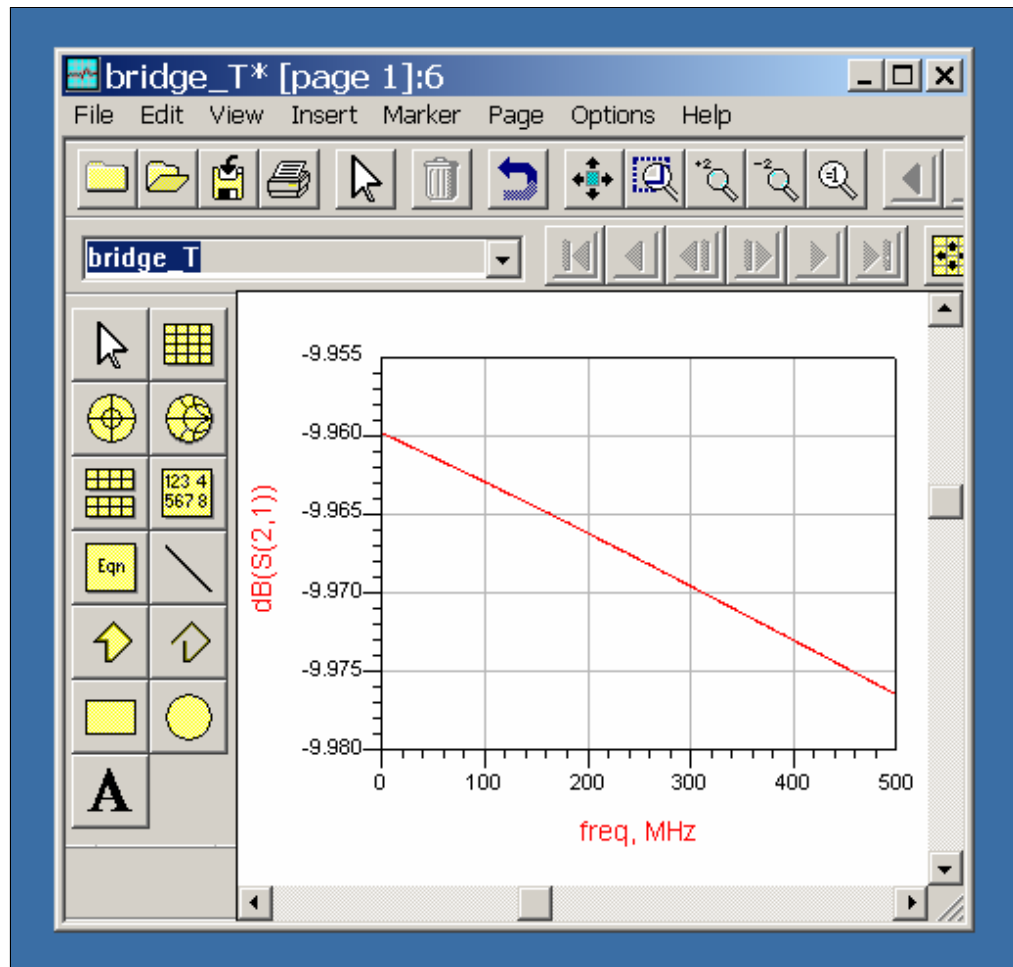


Figure 6 - Results window.

7. Draw the following graphs : $S_{21}(dB)$ (the attenuation) versus frequency, $S_{11}(dB)$ (input return loss) and $S_{22}(dB)$ (output return loss) versus frequency, $|Z_{in}|$ and $|Z_{out}|$ (magnitude of the input and output impedances) versus frequency.

2.3 Transmission Line Simulation

1. Draw a 50Ω transmission line terminated by 50Ω load, as shown in Figure 7.

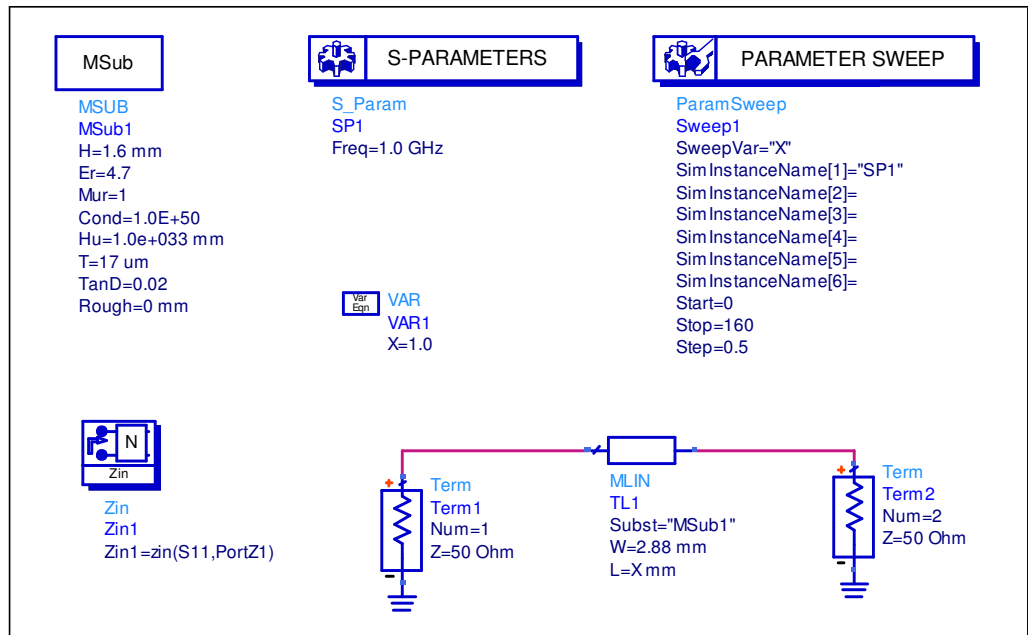


Figure 7 - 50 ohm transmission line.

Component Name	Description	Categories List
Msub	Microstrip substrate	TLines-Microstrip
Terminaton	Term	Simulation S_param
MLIN	Microstrip Line	TL-Microstrip
Zin	Port Input Impedance	Simulation S_Param
S-Parameters	S-parameter Simulation	Simulation S_Param
P_1Tone	Power Source	Sources-Freq Domain
Parameter Sweep	Swept parameter simulation	Simulation S_Param

Table-2, list for Figure 7

2. Draw the following graphs : $S_{21}(dB)$ (attenuation) versus the length X, Z_{in} (input impedance) versus the length X and $S_{21}(phase)$ (phase response) versus the length X, as shown in Figures 8 and 9.

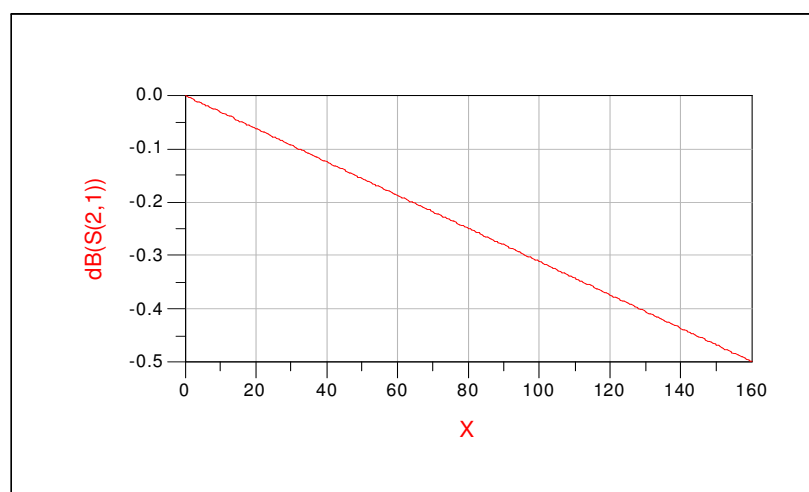


Figure 8 - Attenuation versus the length X.

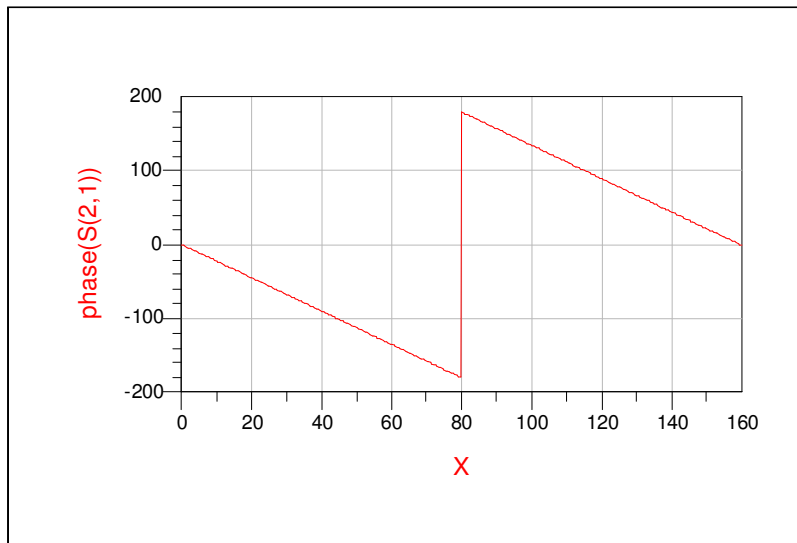


Figure 9 - Phase response versus the length X.

3. Draw a 50Ω transmission line terminated by a short, as shown in Figure 10.

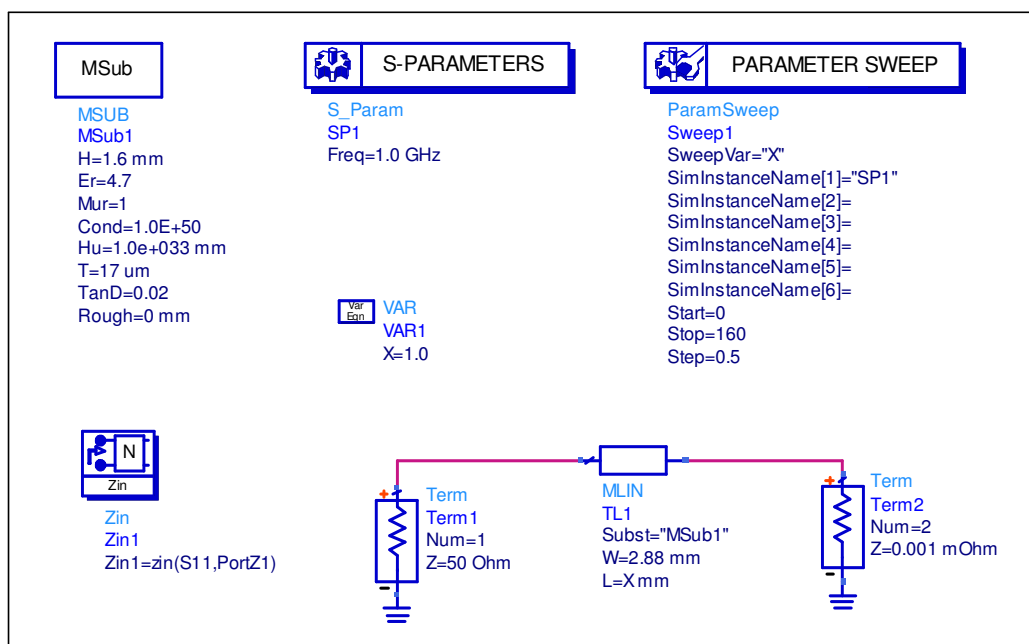


Figure 10 - 50 ohm transmission line terminated by a short.

4. Draw the following graphs: the attenuation, $S_{21}(dB)$, versus the length X, the input impedance, Z_{in} , versus the length X and the phase response, $S_{21}(phase)$, versus the length X.