Familiarization with ADS

1) Objective:

To provide an overview of the ADS software, and deliver stepwise guidance to familiarize the student about the basic features of ADS

2) Theory:

Advanced Design System (ADS) is the leading electronic design automation software for RF, microwave, and signal integrity applications produced by Keysight Technologies. ADS can be used to design transmission-line based circuits, passive circuits, active circuits, and system-level components. It can also incorporate linear and nonlinear device models and supports schematic capture, layout design rule checking, frequency-domain and time-domain circuit simulation, and electromagnetic field simulation. It is the self-proclaimed "premier RF & microwave design platform", which allows full characterization and optimization of an RF design.

3) Lab Exercises:

(a) Begin ADS by double-clicking the **Advanced Design System** icon on the desktop. Alternatively, the program may be launched from the start menu.

(b) Once the **Getting Started with ADS** window opens, click on **Create a New Project**. Otherwise, go to the **Advanced Design System** window and choose **File → Create a New Project**.

(c) Name the new project "LAB1". Specify an appropriate file location. Change the standard length unit to millimeters and click OK.

(d) ADS will then create a project named "LAB1_prj" in the specified location. This folder will serve as the directory for all files associated with your "LAB1" ADS project.

(e) After creating the "LAB1" project directory, an untitled schematic window will open along with the **Schematic Wizard**.

(f) Using the **Schematic Wizard** perform the following actions:

- (i) Select **Simulation** and then click **Next**
- (ii) Select **Linear Circuit**, **2-port**, and then click **Next**
- (iii) Select **I will design my own circuit**, and then click **Next**
- (iv) Select **Linear Frequency Sweep**
- (v) The description field will list the components that will be automatically inserted into the schematic window
- (vi) Click **Finish**

(g) The schematic window will 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 category from the dropdown box directly above it.

(h) Scattering parameter (S-parameter) measurements are taken from port terminations. The ADS name for these components is **Term**. The schematic wizard has already added two Term components to the design. 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.

(i) The **Schematic Wizard** has also placed a **Display Template** in the design workspace. A **Display Template** can be used to store data display items such as pre-configured plots and other graphical items that may be used for future simulations.

Familiarization with ADS

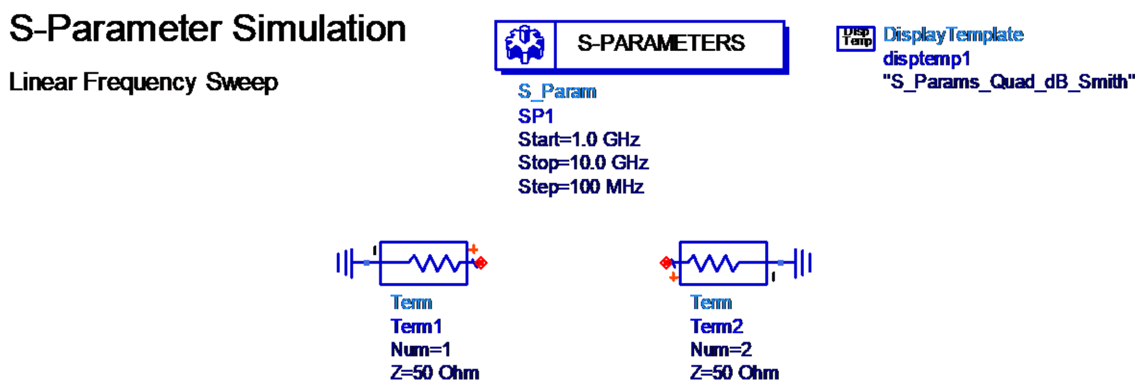

Figure 1: Original Design Workspace

(j) Starting from the original design workspace, add additional components and replicate the schematic design as shown in figure 2

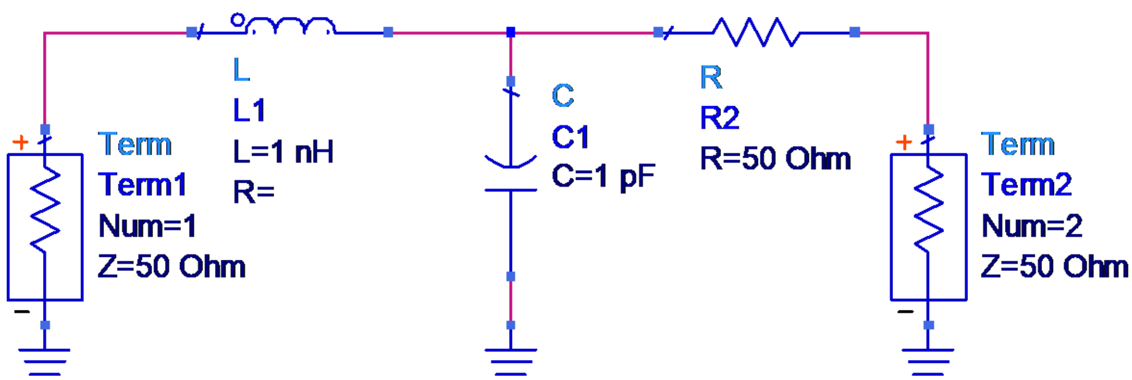

Figure 2: RLC Circuit Schematic

(i) To place the resistors and capacitors, select the **Lumped-Components** category from the drop-down menu, and then select the required components from the resulting palette
(ii) Use the **Insert Wire** and **Insert GROUND** buttons (along the top of the screen) as needed
(iii) Pressing **Ctrl+R** will rotate a selected component
(iv) Double click the **S-Parameter Simulation Controller** to adjust the start, stop and step size frequencies

(k) Once the circuit of figure 2 has been built, save the design. ADS designs are given the file extension ".dsn" and are stored in the networks folder of the project directory

(l) Simulate the design by selecting **Simulate → Simulate** from the top menu, or by pressing **F7** on the keyboard

(m) 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 **Data Display Window** will open. The data display window allows you to view and analyze a dataset using a variety of graphical and numerical methods

(n) If the **Data Display Window** contains some pre-existing plots, select all of the plots and delete them

Familiarization with ADS

(o) To create and place new plots, choose **Insert → Plot**. Move the cursor over the blank page and click to insert the new plot. The **Plot Traces & Attributes Window** will open from which the plot type (**Rectangular**, **Polar**, **Smith Chart**, **List**) can be selected and the variables to be plotted can be chosen as shown in figure 3

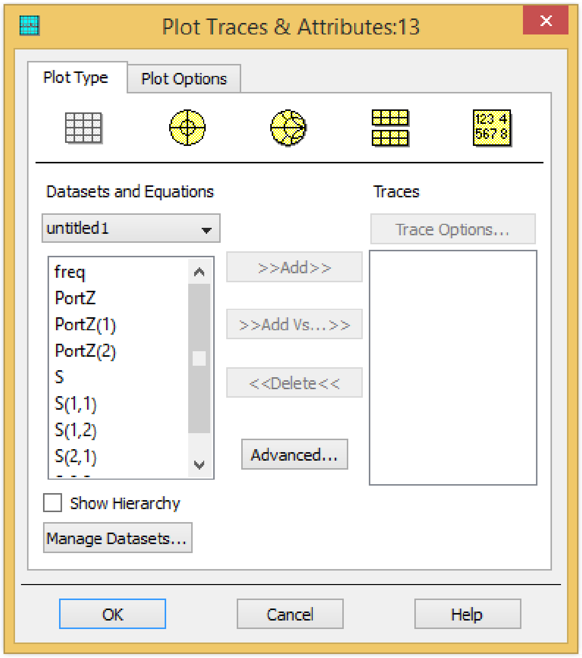

Figure 3: Plot Traces & Attributes Window

(p) To create and insert equations, choose **Insert → Equation**. Move the cursor over the blank page and click to insert the new equation. The **Equation Editor Window** will open through which the desired equation can be entered as shown in figure 4

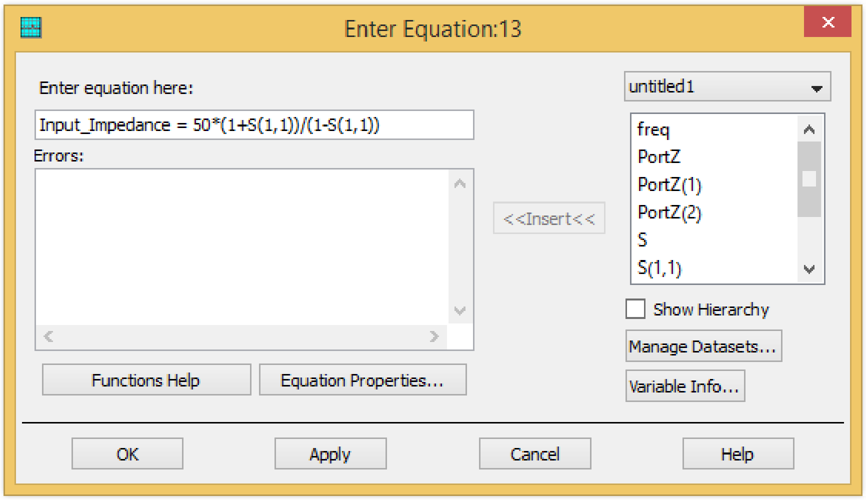

Figure 4: Equation Editor Window

Familiarization with ADS

(q) Use the **Rectangular Plot** option to generate the magnitude (dB) versus frequency and phase versus frequency of the S-parameters **S_{11}** and **S_{12}** as shown in figures 5, 6, 7 and 8. Use the **Smith Chart Plot** option to represent the S-parameters **S_{11}** and **S_{12}** as shown in figures 9 and 10.

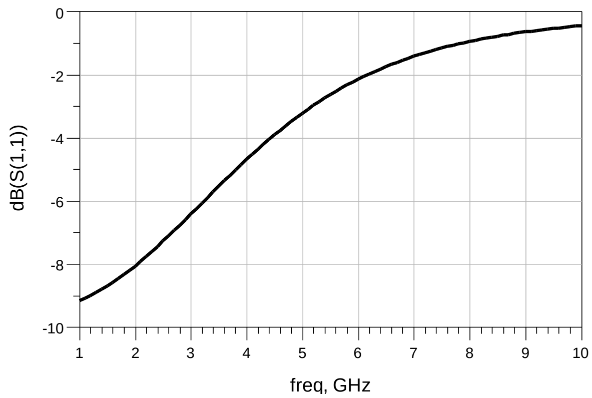

Figure 5: Magnitude (dB) of S_{11} versus Frequency

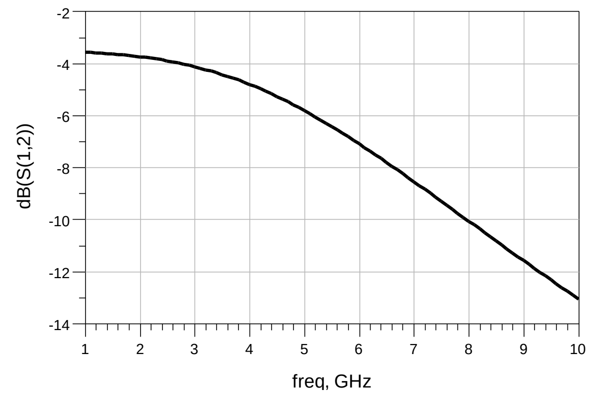

Figure 6: Magnitude (dB) of S_{12} versus Frequency

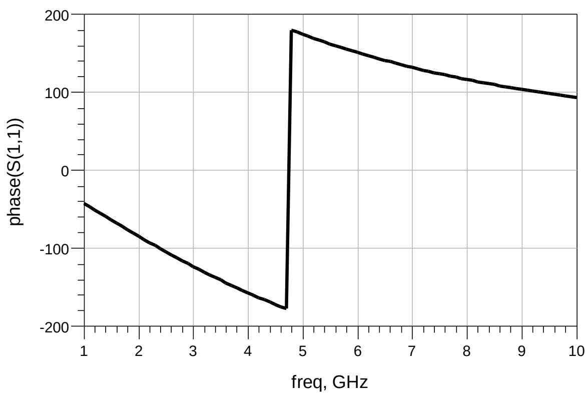

Figure 7: Phase of S_{11} versus Frequency

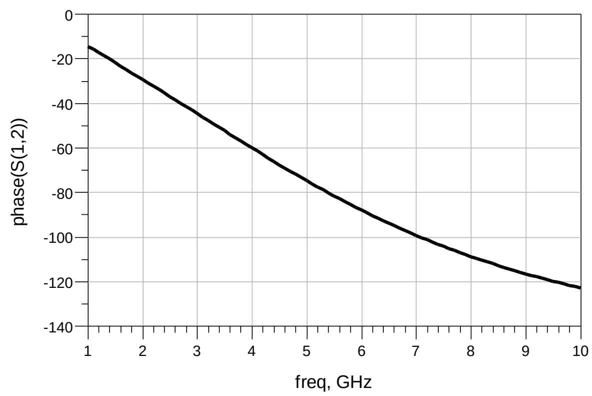

Figure 8: Phase of S_{12} versus Frequency

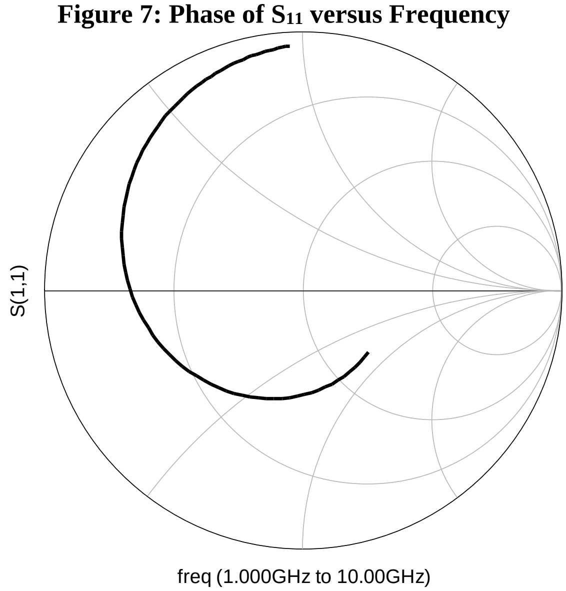

Figure 9: Smith Chart Plot of S_{11} versus Frequency

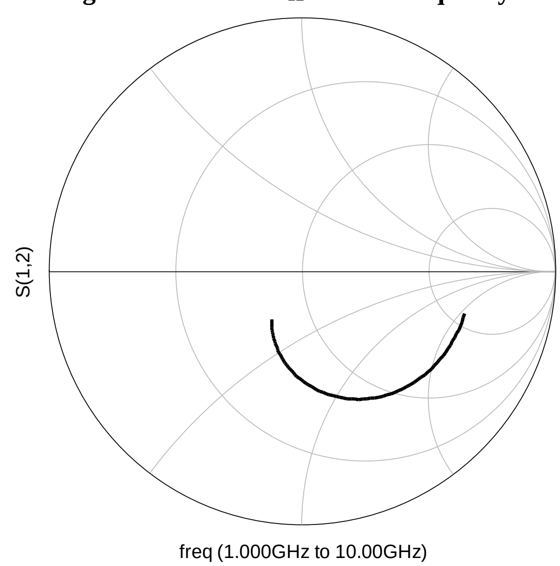

Figure 10: Smith Chart Plot of S_{12} versus Frequency

Familiarization with ADS

(r) Use the **List Plot** option to generate the table of S-parameters **S_{11}, S_{12}, S_{21},** and **S_{22}** versus frequency as shown in figure 11.

freq	S(1,1)	S(1,2)	S(2,1)	S(2,2)
1.000 GHz	0.348 / -43.097	0.663 / -14.473	0.663 / -14.473	0.334 / -7.311
1.100 GHz	0.351 / -47.375	0.662 / -15.937	0.662 / -15.937	0.334 / -8.067
1.200 GHz	0.354 / -51.643	0.661 / -17.405	0.661 / -17.405	0.334 / -8.829
1.300 GHz	0.358 / -55.896	0.660 / -18.877	0.660 / -18.877	0.334 / -9.599
1.400 GHz	0.362 / -60.134	0.659 / -20.355	0.659 / -20.355	0.335 / -10.377
1.500 GHz	0.367 / -64.354	0.658 / -21.837	0.658 / -21.837	0.335 / -11.162
1.600 GHz	0.372 / -68.553	0.656 / -23.324	0.656 / -23.324	0.335 / -11.956
1.700 GHz	0.377 / -72.728	0.655 / -24.817	0.655 / -24.817	0.335 / -12.758
1.800 GHz	0.383 / -76.878	0.653 / -26.315	0.653 / -26.315	0.335 / -13.570
1.900 GHz	0.389 / -80.999	0.652 / -27.819	0.652 / -27.819	0.335 / -14.390
2.000 GHz	0.395 / -85.090	0.650 / -29.327	0.650 / -29.327	0.335 / -15.219

Figure 11: List Plot of S_{11}, S_{12}, S_{21}, and S_{22} versus Frequency

(s) Insert an equation to calculate the impedance from the reflection coefficient (**S_{11}**) as shown in figure 12. Assume the characteristic impedance is 50 Ω. Use the **Rectangular Plot** to generate the magnitude versus frequency plot as shown in figure 12, and insert a **Marker** (m1) in the plot and obtain its reading.

Eqn Input_Impedance = 50*(1+S(1,1))/(1-S(1,1))

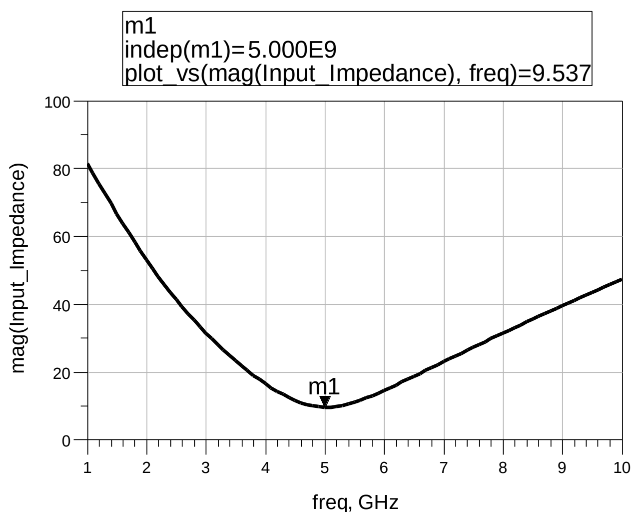

Figure 12: Impedance Equation and Rectangular Plot of Impedance versus Frequency

(t) Repeat the above steps for the circuit shown in figure 13. Simulation start frequency should be 1Hz, stop frequency should be 10 GHz and frequency step size should be 10 MHz.

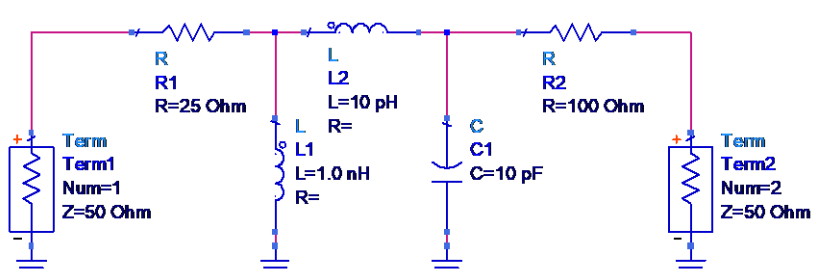

Figure 13: Practice Circuit