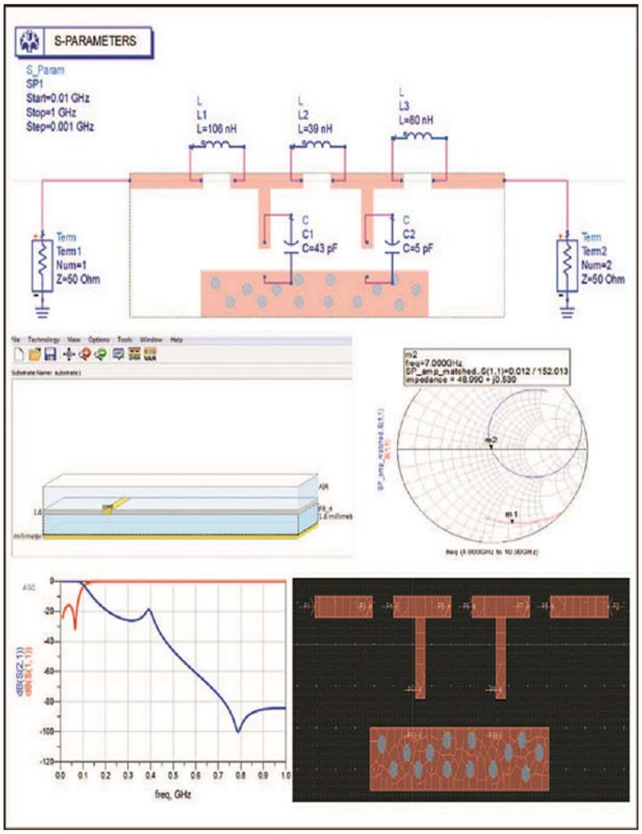
APPLICATION NOTE

Keysight EEsof EDA

RF Design Software Learning Kit



Step-By-Step Examples on Using ADS Software for an Introductory RF/ Microwave Course

KEYSIGHT TECHNOLOGIES

Contents

Preface

The intent of this book is to explore the ways in which Keysight EEsof EDA software can add value to the already challenging RF/microwave curriculum. Many texts used for the introductory RF/microwave course contain examples using CAD simulation software, but they do not explain how to set up those simulations. This text provides step-by-step examples for the course, highlighting the theory and application of the curriculum within the environment of the Keysight EDA Advanced Design System Electronic Design Automation (EDA) software, or ADS.

ADS is used by RF engineers in a range of industries. This text aims to provide the reader with the basic tools necessary to succeed when entering the workforce. Therefore, the intended audience is a student enrolled in an introductory microwave course, and the material is presented in the familiar homework style format. The topics covered range from basic transmission line theory to passive filters, and include three design projects intended to be used in the laboratory setting. The structure of the homework questions are designed to teach the user to apply the theory, expect a solution, and validate the hypothesis. Often, the problem with using a CAD tool is that the user does not know how to set up a correct simulation, and the software will only simulate what it is instructed to do. These examples are designed to show the capabilities of the software, while building an understanding of how it works and how to set up correct simulations. Although the material is presented for a classroom setting, the emphasis on fundamental theory opens the demographic to anyone interested in learning basic microwave theory and how to use ADS software.

Because this is a software learning tool, the introductory problem is the longest in length. The detail of each step within the ADS environment becomes less complex as the material continues. The text is designed to develop the user's skillset and the proficiency level increases with each chapter. To ease a user into the ADS environment, please refer to the Getting Started with ADS video.

Special thanks to Natalie Killeen for her work in creating this material, and Tim Wang Lee for his guidance.

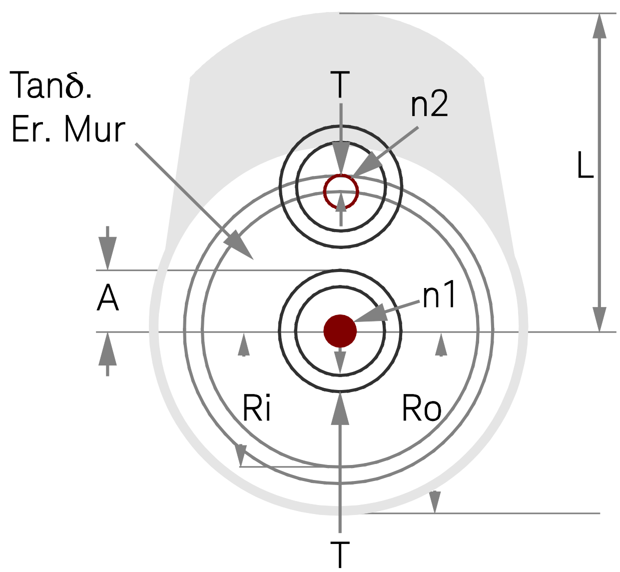
Chapter 1 – Transmission Line Theory

1.1 Field Analysis of Transmission Lines

Problem 1: Extracting the Lumped-Element Model from a Coax Line

Problem Statement

Determine the per unit length lumped-element equivalent of the following RG58 coax line graphically using ADS. Compare this to the derived values from the characteristic equations for the transmission line parameters (Figure 1-1).



A = 0.4191 mm	T = 0.02 mm
Ri = 1.4732 mm	Cond = $1e^{50}$
Ro = 2.4732 mm	Er = 2.25
Tanδ = 0.0002	Mur = 1.0

Figure 1-1. Transmission line parameters

Simulate using AC analysis from 1 MHz to 1 GHz.

Solution

Getting started within ADS

- Open ADS and choose Create a new workspace from the initial menu. Name the workspace, save it to the desired directory, and click Next.

- Choose the Analog/RF ADS library and click Next to reach the layout resolution section. For this example set, choose the Standard ADS Layers, 0.0001-mm layout resolution, click Next and then Finish.

- The Main widow will automatically appear. In the File menu, choose New and Schematic to open a New Schematic window. Here, the schematic can be given a more descriptive name than the workspace in the Cell text field. The workspace-schematic relationship is analogous to the parent-child relationship. It is important to name the schematic a descriptive, yet short name. This will become more evident as we proceed.

- The Schematic Wizard will then appear, which can be exited out of to reveal the schematic window.

Strategy

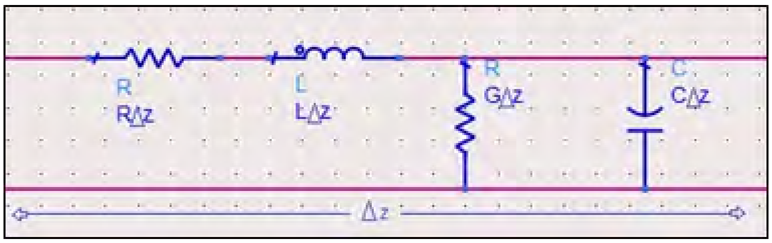


Figure 1-2. The coax line is a transmission line that can be modeled by the lumped-element circuit.

By strategically analyzing the coaxial line, the equivalent lumped-element values can be extracted (Figure 1-2). Terminating the lumped-element circuit in a short will cause only the first two elements to be seen: the resistor and inductor. Terminating the circuit with an open circuit will provide the other two elements. Separating the real and imaginary parts, and plotting the impedance of the coax line yields the lumped-element values.

The fundamental transmission line parameters can be calculated in terms of coax parameters from the following relationships[1]:

$$R = \frac{R_s}{2\pi}\left(\frac{1}{a}+\frac{1}{b}\right) \text{ Ohm/m} \qquad \text{Equation 1.1}$$

$$L = \frac{\mu}{2\pi}\ln {}^{b}/_{a} \text{ H/m} \qquad \text{Equation 1.2}$$

$$C = \frac{2\pi\epsilon'}{\ln {}^{b}/_{a}} \text{ F/m} \qquad \text{Equation 1.3}$$

$$G = \frac{2\pi\omega\epsilon''}{\ln {}^{b}/_{a}} \text{ S/m} \qquad \text{Equation 1.4}$$

Here, a is the radius of the inner conductor of the coax, while b is the radius of the outer conductor of the coax.

[1] *D. Pozar, Microwave Engineering 4th Ed., John Wiley & Sons, Danvers, MA, 2012.*

What to expect

Equations 1.1 - 1.4 for the transmission line parameters are valid only for ideal coax lines. Given the low frequency range and short length of coax line, it is expected that transmission line effects will not be experienced, and the simulated and calculated values will agree for the real components, R and G because they are not frequency dependent. The simulated reactive components, L and C, are expected to deviate from the calculated values given a frequency-dependent dielectric loss model for the coax line. For this reason, the coax line will be simulated using a frequency-independent dielectric loss model, and all parameters will be constant over the frequency sweep.

Execution

- Place the COAX_MDS element onto the schematic++ either by finding it within the TLines-Ideal Component palette or by searching for it in the text box above the palette (Figure 1-3).

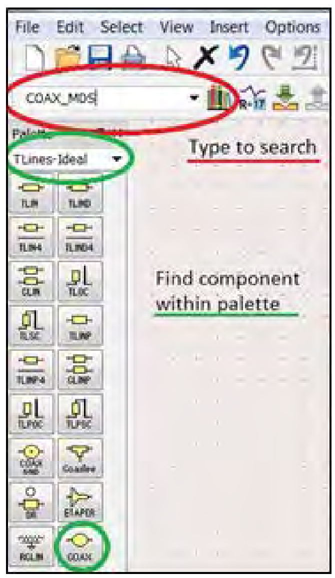


Figure 1-3. Find the element in the Component palette.

The component will come with default parameters that we will need to change to reflect the problem statement. It is a better practice to define each parameter as a variable, and to refer to the variable, instead of hard keying in each parameter. This makes it easier to duplicate components, and avoid errors. The variable component is found in the Toolbar located just above the schematic field, and is denoted by the following symbol 

.

- Add the VAR symbol to the schematic and double click to open the Edit Instance Parameters window. Here, the coax line parameter variables can be added all at once.

To avoid any unit errors, keep everything in the standard meter unit. The schematic thus far should look similar to Figure 1-4.

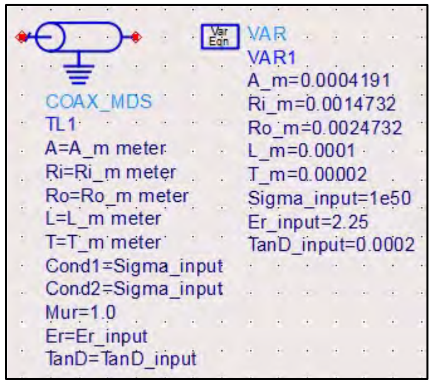


Figure 1-4. Schematic with coax line parameter variables added.

As stated in the "What to expect" section, the coax model uses the frequency-dependent dielectric constant model, by default the Svensson/Djordjevic model, which will cause the simulated values to disagree with the parameter equations for the inductor and capacitor. To change this, double click on the coax component and change the Dielectric Loss Model to Frequency Independent (Figure 1-5).

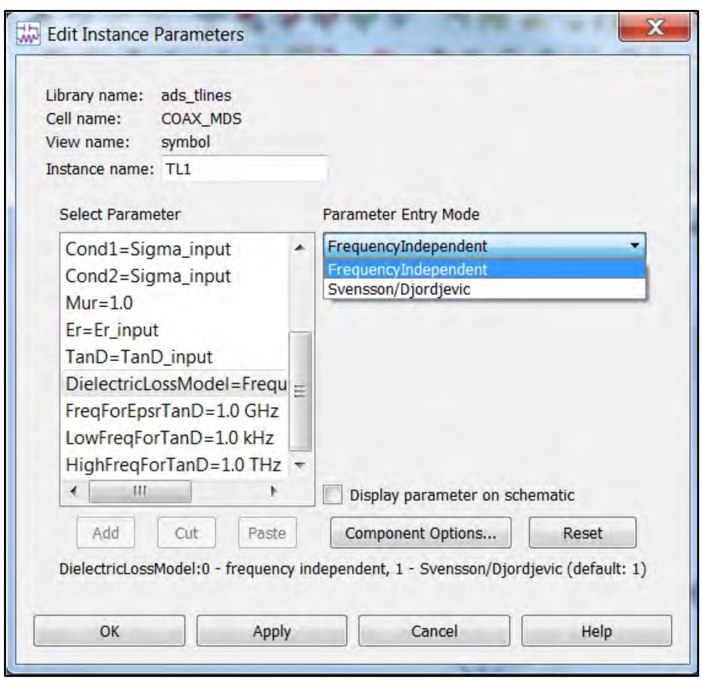
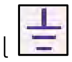
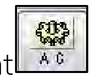


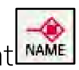
Figure 1-5. Changing the coax component to the Frequency-Independent model.

Short Circuit Analysis

First, we will use short circuit analysis to determine the equivalent per unit length resistance and inductance values on the front end of the lumped-element circuit. AC analysis will be used, requiring the AC current source I_AC to be connected to ground, denoted by the symbol , and a connecting wire or Ctrl+w. The AC current source can be found in the Sources - Frequency Domain palette. It may even be helpful to rotate the component by using Ctrl+r. This is a simulation, so 1A is fine for the input.

- Add the AC simulation component

 from the Component palette Simulation-AC and specify the frequency range for the problem. Double clicking the component will open the AC Small Signal Simulation window. Here, specify the Sweep Type as Log, and input the number of Pts./decade as 800.

- In AC simulation, the desired output needs to be specified by either a probe component, named connection or measured equation. Since the desired output is impedance, we will name the input wire as Z_SC for short circuit impedance using the name component

. Pressing the button will initiate the Wire/Pin Label window. Type in the name and click on the wire you wish to name, otherwise the name will not stick. Naming a node will automatically create a variable with that same name for the voltage measured at the node.

- Because the input current is 1A, the impedance will be the same as the voltage in this case. Then, similar to the VAR component, there is a measured equation component 

. The measured equation component can be found in any simulation palette, or by typing MeasEqn into the search bar. Here, we will use the simulation values to compute the resistive and inductive values per unit length. The equivalent per unit length resistor is the real part of Z_SC, while the equivalent per unit length inductance is the imaginary part divided by the frequency (Z_SC = R + jwL).

The schematic should now look like Figure 1-6.

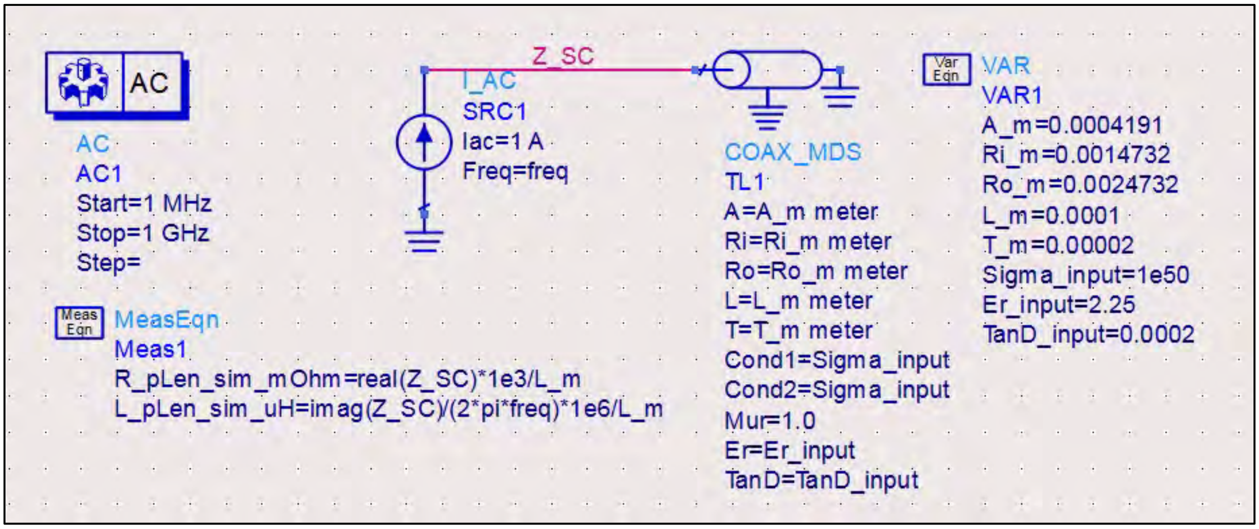


Figure 1-6. The schematic with the measured equation component added.

- In the Tool Menu above the schematic field, click the Simulate icon  to initiate simulation. Two windows will pop up: the hpeesofsim and the Data Display. If an error occurs during simulation, it will show in the hpeesofsim  window. The Data Display  window allows us to plot the simulation data in a variety of different ways. First, we will plot the resistive value from the measured equation versus frequency.
- In the Data Display window, choose the Rectangular Plot  from the palette on the left hand side and add it to the display field.
- The Plot Traces & Attributes window will automatically open (Figure 1-7). Choose the measured equation for the resistor and add it to the plot. The plot with its default settings will appear in the Data Display.

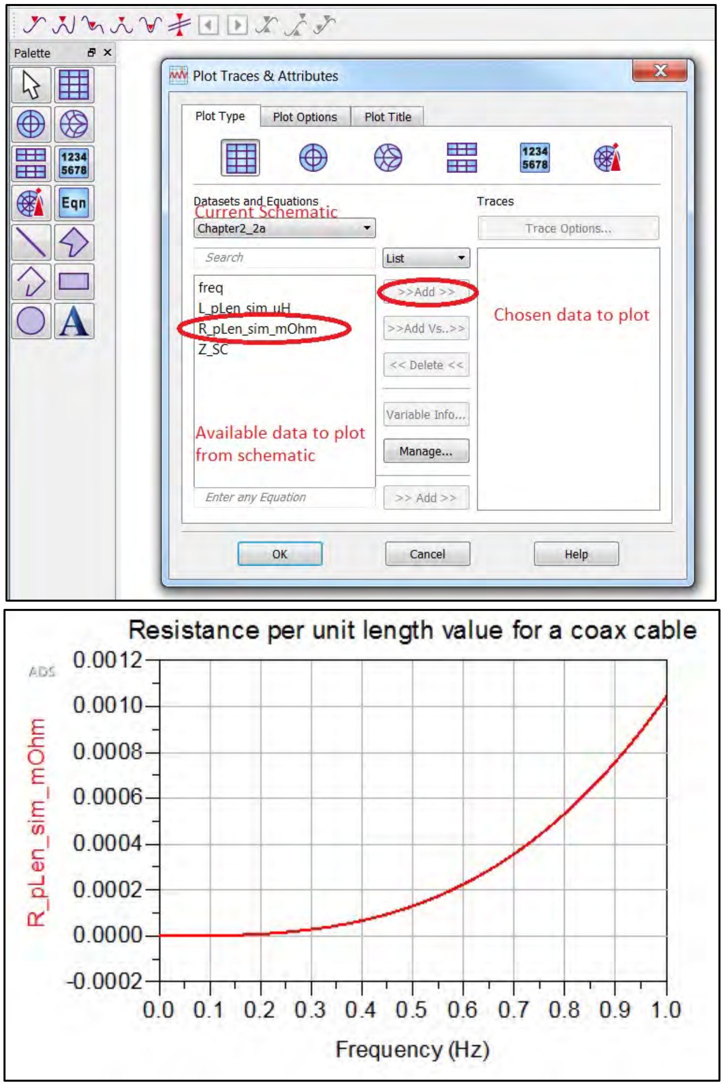


Figure 1-7. Plot of the resistive value from the measured equation versus frequency.

- Double click on the plot area to bring back up the Plot Traces & Attributes window.

- Use the Plot Options tab and change the x-axis to log scale (Figure 1-8). Also, change the y-axis min and max values to be more reasonable (e.g., -1 to 2). The axis and plot labels, as well as fonts can also be changed. Double clicking or right clicking on the trace line will enable manipulation of the line characteristics (Figure 1-9).

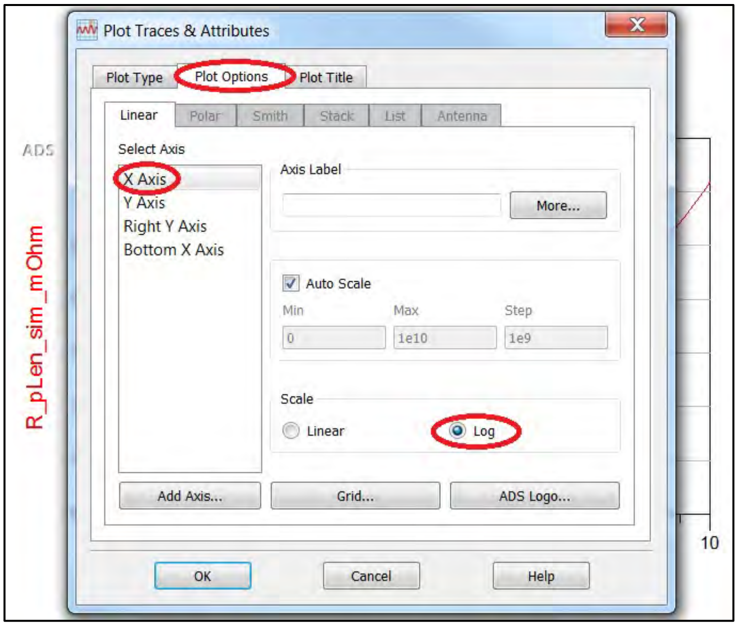


Figure 1-8. Plot Traces & Attributes window.

The plot is now more presentable in the form shown in Figure 1-9.

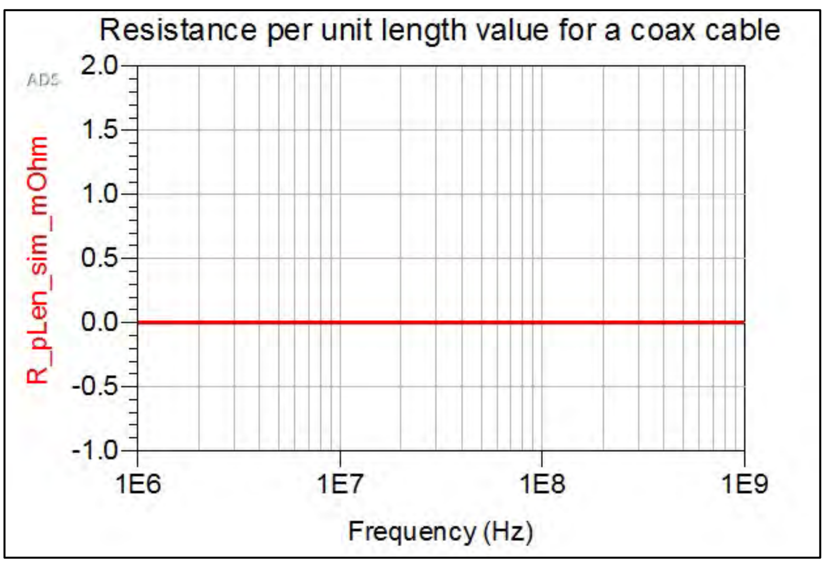
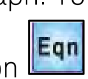


Figure 1-9. Plot of resistance per unit length value for a coax cable.

- For comparison to the equivalent transmission line parameter equations, the simulated values and the equations are plotted on the same graph. To write an equation within the Data Display window, click on the Equation button 

 in the palette, which will automatically bring up the Enter Equation window (Figure 1-10).

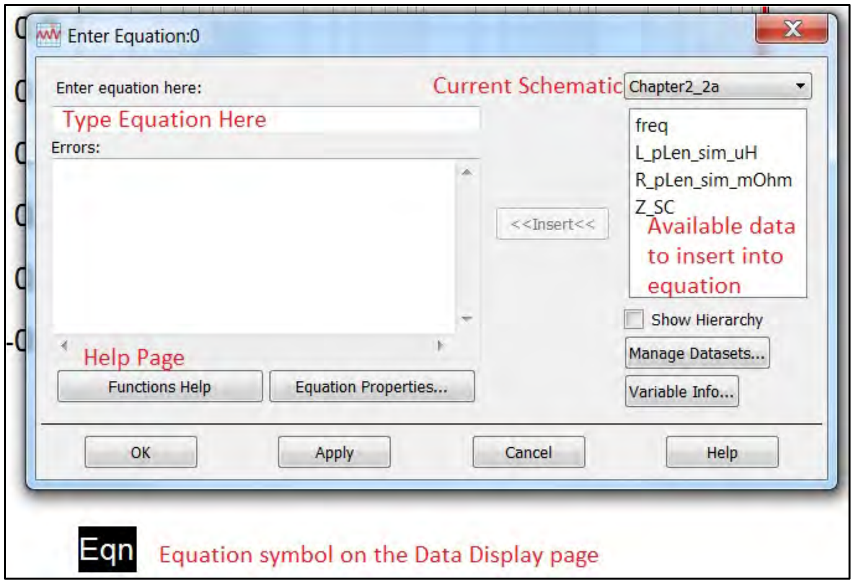


Figure 1-10. ADS Enter Equation window.

The equation for resistance per unit length in the text is given by:

$$R = \frac{R_s}{2\pi}\left(\frac{1}{a} + \frac{1}{b}\right)\ \mathrm{Ohm/m} \qquad \text{Equation 1.5}$$

Where R is the surface resistivity, a is the inner conductor radius and b is the outer conductor radius.

Using the Equation function, the variables are established and the equation for resistance, Equation 1.5, is given. If an invalid equation is entered, the equation will turn red and information to troubleshoot the error will be provided in the Enter Equation window. Once the equation is placed on the Data Display window, it can be copied and edited directly on the Display page. The set of equations should look like that shown in Figure 1-11.

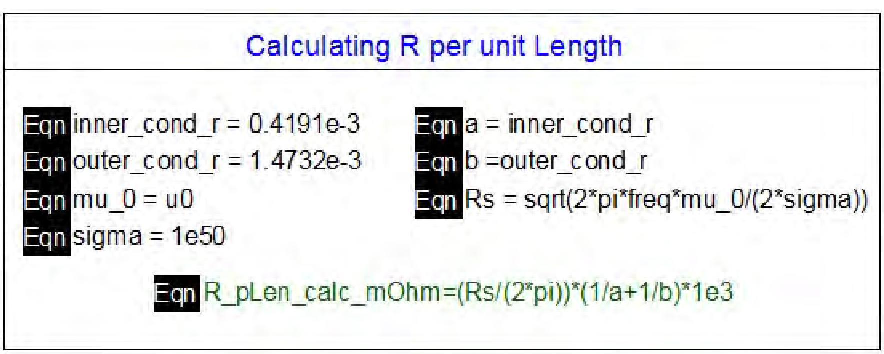


Figure 1-11. The equations for resistance.

The new calculated value is added to the plot to compare the simulated and calculated values. This is done by double clicking the plot and adding the name of the equation in the text field (Figure 1-12).

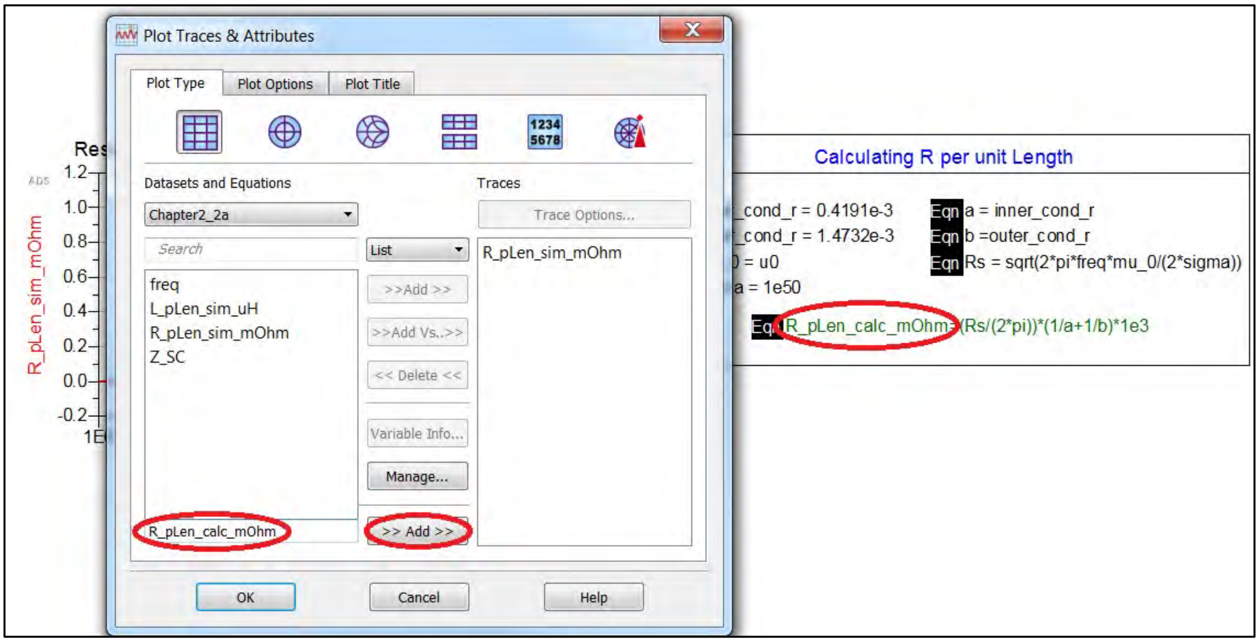


Figure 1-12. Comparing the simulated and calculated values.

With multiple traces on one graph, a marker would make reading the plot easier. To place a marker, use the Marker Menu at the top of the page. Multiple types of markers are available, but we will use the New Line marker. Click on any trace to set the marker. This will read out the values for both traces at the same time, while sliding the marker. The plot should now look like Figure 1-13.

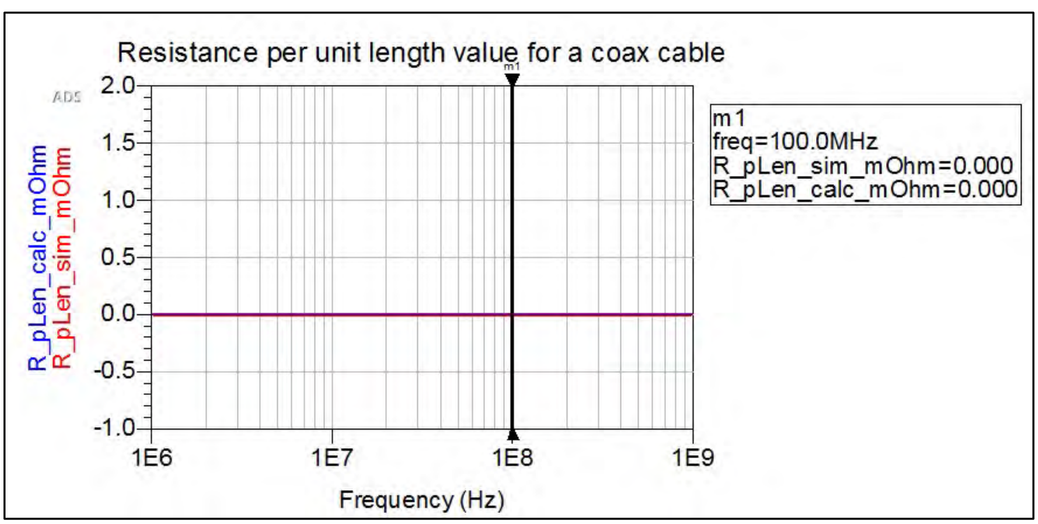


Figure 1-13. Shown here is a plot with a marker.

The plot shows that the calculated and simulated values agree, which is what was expected because the lumped-element model only applies to relatively simple lines.

Next, the simulated and calculated inductance per unit length will be compared. The equation for inductance per unit length in the text is given by:

$$L = \frac{\mu}{2\pi} \ln {}^{b}/_{a} \text{ H/m} \qquad \text{Equation 1.6}$$

Save the resistive Data Display page and move to another area of the Data Display. This will allow the re-use of some variables already set up in the previous part. It is easier to copy and paste the resistive plot and equations, and edit within the Data Display page as needed.

Because the simulated inductance value uses the independent variable “freq,” the calculated value also needs to incorporate this variable. The resistive value already did, so an error was not encountered, but now we must add the variable to the equation for the plot to accept the calculated value. The resulting equation and plot should look like that shown in Figure 1-14.

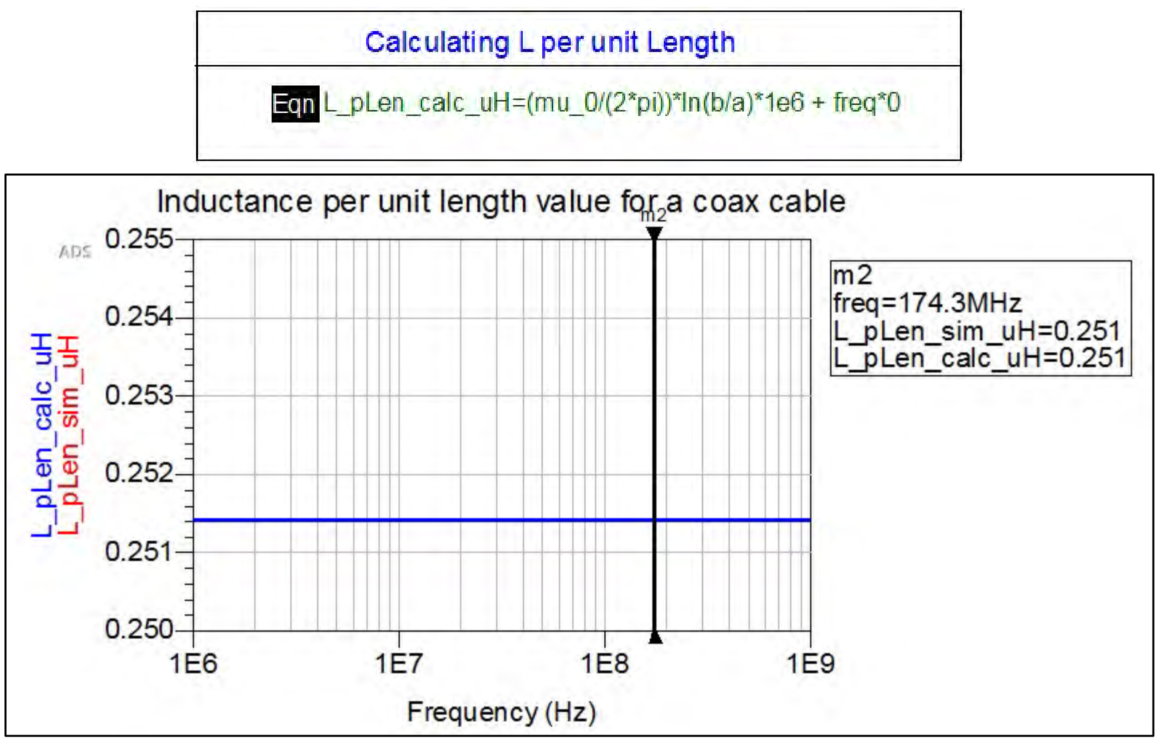


Figure 1-14. The equation and plot with the variable added to the equation.

Again, it is shown that the calculated and simulated values agree with each other. This is because the coax model was changed to a frequency-independent dielectric constant model. If curious, the coax model can be changed back and the effect observed.

Open Circuit Analysis

First, we will use the open circuit analysis to determine the equivalent per unit length capacitance and conductance values on the back end of the lumped-element circuit. Using the same schematic as the short circuit, copy and paste the new circuit to begin editing. The schematic should now look like that shown in Figure 1-15.

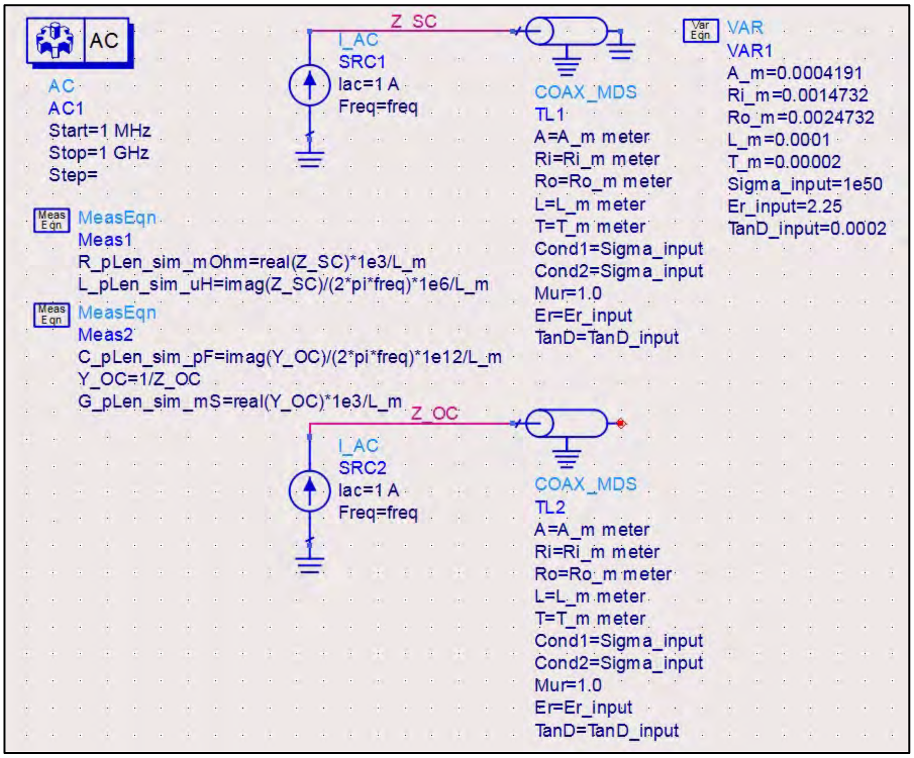


Figure 1-15. Updated schematic.

The equations for capacitance and conductance per unit length are given by:

$$C = \frac{2\pi\epsilon'}{\ln{}^{b}/_{a}}\ \text{F/m} \qquad \text{Equation 1.7}$$

$$G = \frac{2\pi\omega\epsilon''}{\ln{}^{b}/_{a}}\ \text{S/m} \qquad \text{Equation 1.8}$$

The following shows the calculated vs. simulated value plots for the open circuit analysis (Figure 1-16).

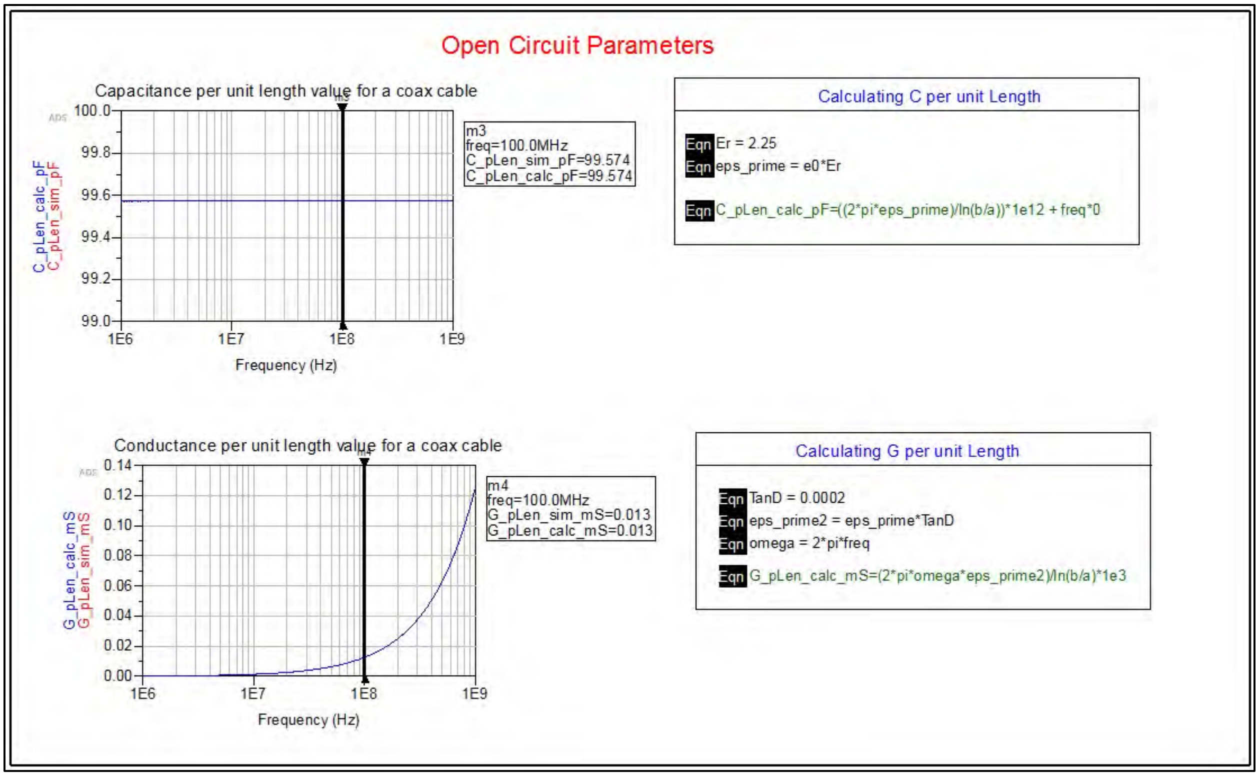


Figure 1-16. Shown here is the calculated vs. simulated value plots for the open circuit analysis.

Again, the simulated and calculated values agree, as expected. It is interesting to note that G is related to the dielectric loss of the substrate, which is linearly dependent on frequency, and the coax line becomes more conductive with higher frequencies. This is mathematically shown in the omega term in the characteristic equation for the equivalent conductance per unit length.

Conclusion

The equivalent per unit length lumped-element section of transmission line for the RG58 coax line at 100 MHz is given in Figure 1-17.

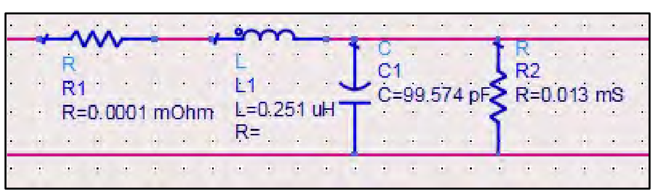


Figure 1-17. The per unit length lumped-element section of the transmission line.

1.2 The Terminated Lossless Transmission Line

Problem 2: The Short Circuit Terminated Ideal Transmission Line

Problem Statement

Show graphically in ADS the changes in reactance, magnitude voltage and magnitude current with respect to line length, for an ideal transmission line that is terminated in a short circuit.

Solution

Strategy

Use the ideal transmission model within ADS and connect to a 2-volt voltage source with 50-Ohm generator impedance. Terminate the line in a short circuit, and use node voltages to find the changes in reactance, voltage and current along the transmission line with respect to electric length.

What to expect

The impedance mismatch between the 50-Ohm transmission line and short circuit termination will cause a full reflection. The resulting impedance seen looking into the transmission line will vary with its length. Using the characteristic equation for input impedance for a transmission line, the results can be examined before simulation.

$$Z_{in} = Z_o \frac{Z_L + jZ_o \tan \beta l}{Z_o + jZ_L \tan \beta l} \qquad \text{Equation 1.9}$$

For the short circuited transmission line, $Z_L = 0$, and the equation reduces to $Z_{in} = jZ_o \tan \beta l$. At the frequency where the line is an integer multiple of λ, $\beta l = \frac{2\pi}{\lambda} \cdot n\lambda = 2n\pi$, making $\tan \beta l = 0$. As a result, we expect to see the short circuit termination only, $Z_{in} = 0$. The voltage and current will measure 0 V and 40 mA (2 volts/50 Ohm), and the reactance will be 0 Ohms. At the frequency where the line is an odd multiple of a quarter of a wavelength, $\beta l = \frac{2\pi}{\lambda} \cdot n\frac{\lambda}{4} = n\frac{\pi}{2}; n = \text{odd}$, making $\tan \beta l \to \infty$. As a result, the reactance will be infinite, making the line look like an open circuit. The incident signal from the source and reflected signal from the termination add constructively, resulting in a peak in the standing wave. The voltage and current will measure 2 volts (1 volt + 1 volt) and 0 mA (40 mA + -40mA). When the line length is 0.5 λ, the transmission line looks like a short again, $Z_{in} = 0$. There will be a phase change, but because we are plotting the magnitude voltage and current, the values will be the same as the 0-λ line length.

Execution

Set up the schematic for the short circuit transmission line with an AC voltage source and voltage probes between the generator and transmission line (Figure 1-18). The voltage probes are found in the Probe Components palette. Rename the probes something more descriptive than V_Probe1/2, like V_source and V_inc. The node can also be named to designate the voltage source, but for this analysis, the probe provides a better physical realization of the measurement.

The voltage source should send out 2 volts to keep the voltage division simple between a 50-Ohm generator and 50-Ohm line.

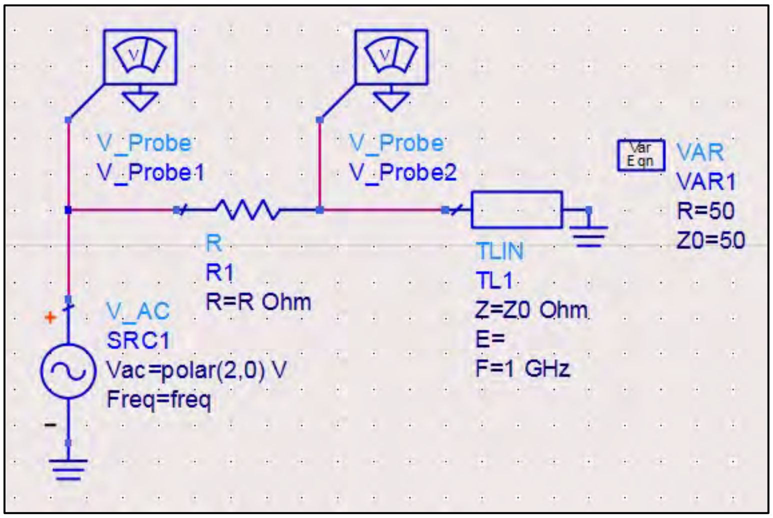


Figure 1-18. Schematic for the short circuit transmission line.

In order to obtain a plot of reactance or voltage versus electrical length, a parameter sweep will be done for the value of theta, in degrees. Create a variable for theta and place it in the electrical length spot for the ideal transmission line. The Parameter Sweep and AC analysis components are found in the Simulation-AC palette. Since the electrical length variation is the goal in this problem, the frequency range for simulation can be a single frequency point. The parameter sweep should reference the variable theta, the simulation plan, and the sweep range of 0 to 360 degrees (Figure 1-19).

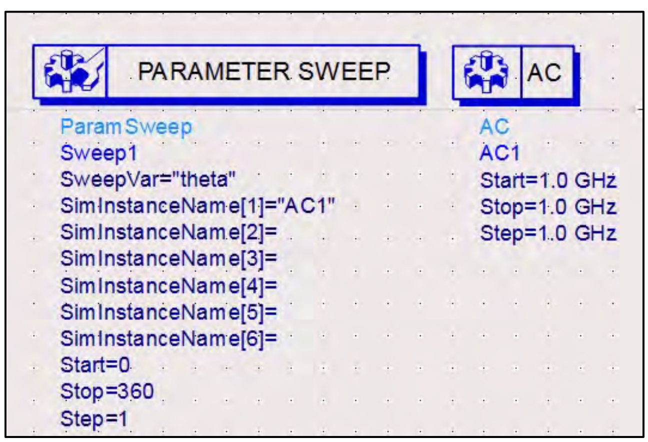


Figure 1-19. The parameter sweep referencing the variable theta,
the simulation plan, and the sweep range.

After adding the output variables as measured equations for Z0 and R, the final resulting schematic— ready for simulation—will look like that shown in Figure 1-20.

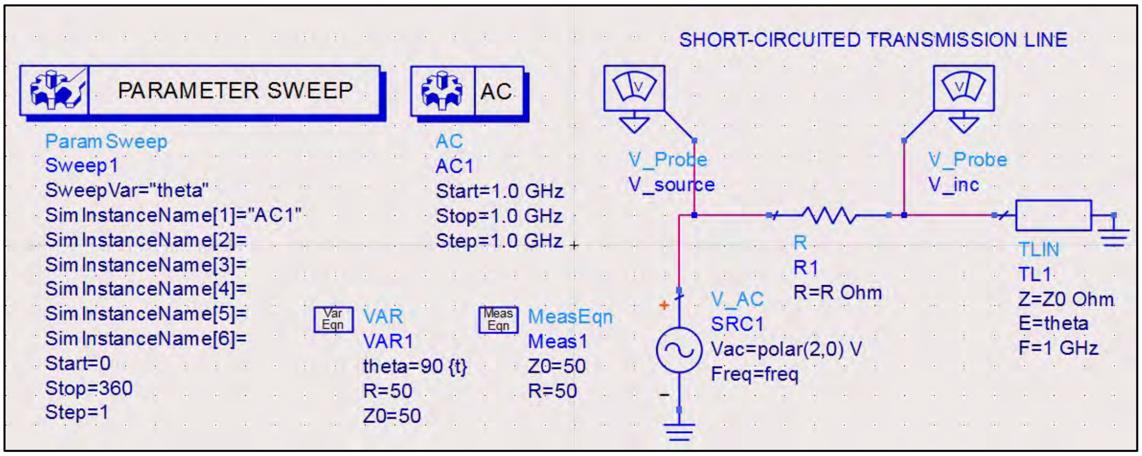


Figure 1-20. The final schematic ready for simulation.

In the Data Display, the following equations are set up to provide the variables necessary to generate the plots for reactance, voltage and current (Figure 1-21). Using probes requires two steps. It is more efficient and cleaner to use named voltage nodes. This chapter will keep the use of probes for a more physical interpretation, but subsequent chapters will name voltage nodes as the skill level of the user progresses.

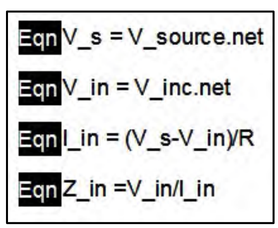


Figure 1-21. The following equations provide the variables necessary to generate reactance, voltage and current plots.

The first rectangular graph for reactance is created by plotting the imaginary part of Z_in with respect to theta. This can be done by manually typing the desired code into the Plot Traces & Attributes window.

Type plot_vs(imag(Z_in),(theta)) into the text box and click OK (Figure 1-22).

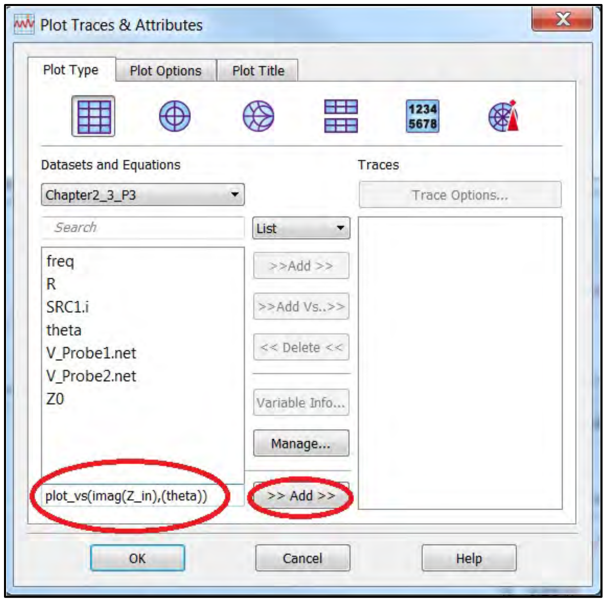


Figure 1-22. Plot Traces & Attributes window.

The plot axes will need to be adjusted for optimal viewing (Figure 1-23). The y-axis can be changed to ±600, and the x-axis from 0 to 360 degrees in 45 degree intervals.

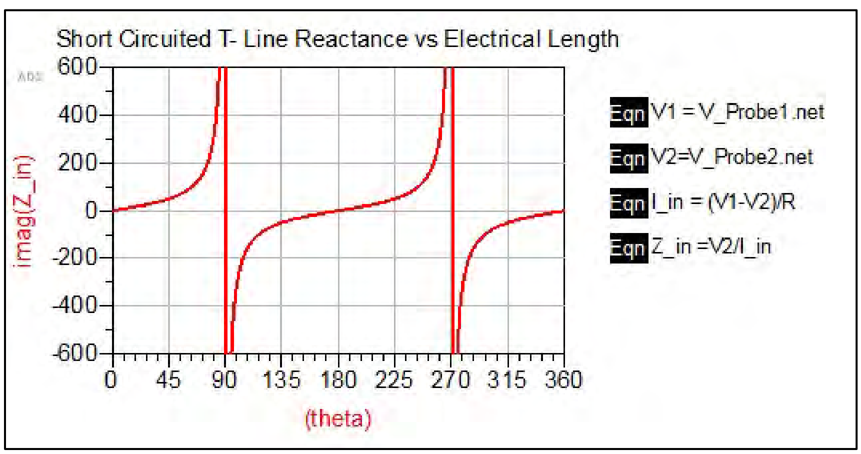


Figure 1-23. Adjustment of the plot axes for optimal viewing.

From the plot, it can be seen that as the electrical length varies, the reactance of the transmission line alternates between inductive and capacitive impedance. While this would be sufficient, it is better to look at the change in terms of lambda, **λ**. Double click on the trace line and manually change the variable from theta to theta/360 (Figure 1-24).

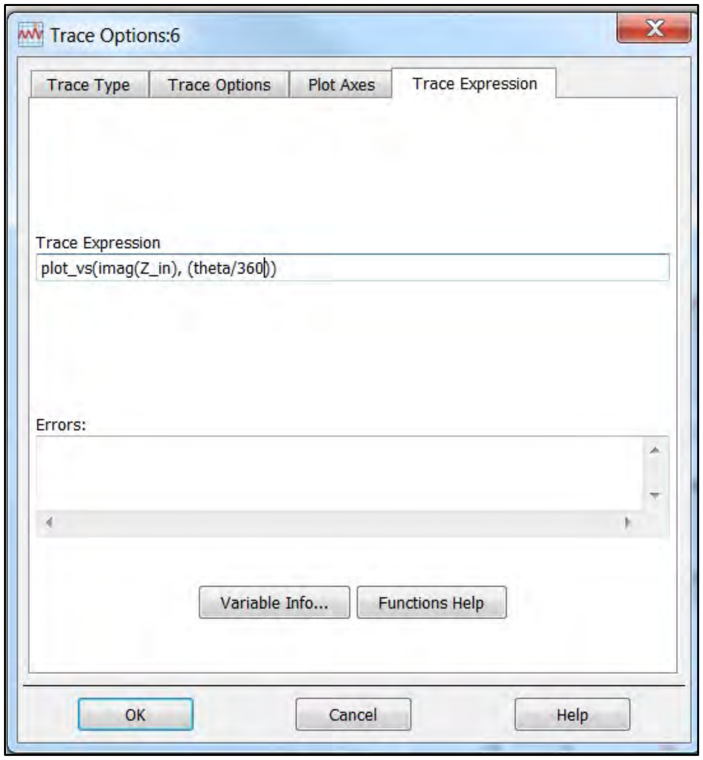


Figure 1-24. Trace Options window.

Making some cosmetic corrections to the axis labels, the reactance versus lambda graph is now complete (Figure 1-25).

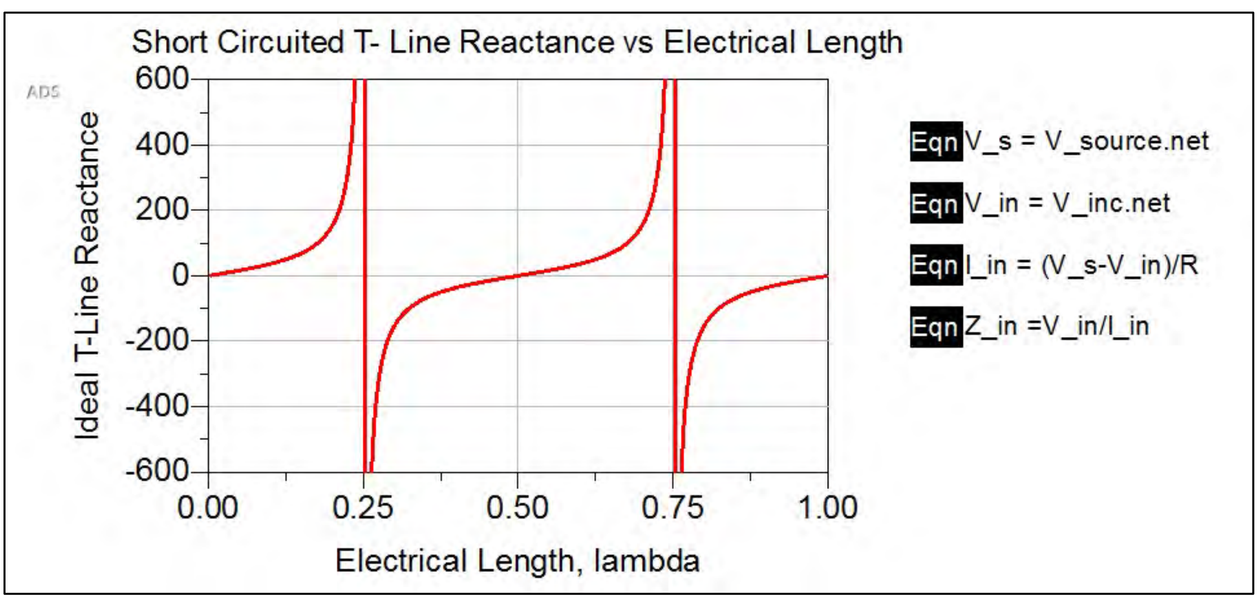


Figure 1-25. The completed reactance versus lambda graph.

The plot for the voltage, plot_vs(mag (V_in), (theta/360)), and current, plot_vs(mag(I_in_mA), (theta/360)), are also completed to show their respective changes versus lambda (Figure 1-26).

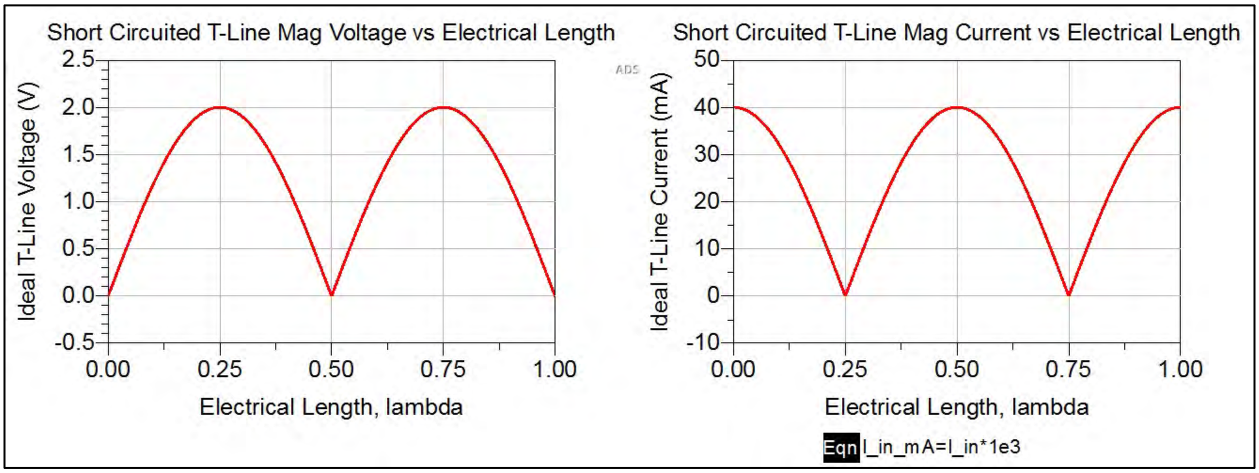


Figure 1-26. Plots for voltage (left) and current (right).

Conclusion

Unlike lumped elements, the voltage and current change with respect to distance on a transmission line. In addition, the current lags the voltage by a quarter wavelength, or 90 degrees, as expected. The short circuited line sees an infinite reactance at a quarter wavelength, and zero reactance at a half wavelength.

Problem 3: The Open Circuit Terminated Ideal Transmission Line

Problem Statement

Show graphically in ADS, the changes in reactance, magnitude voltage, and magnitude current with respect to line length for an ideal transmission line that is terminated in an open circuit. Compare the results for the open circuit to that of the short circuit.

Solution

Strategy

Copy the schematic for the short circuit and replace the short termination with a large resistor to simulate an open circuit.

What to expect

The impedance mismatch between the 50-Ohm source and open circuit termination will again cause reflections on either side of the transmission line, similar to the short circuit. The characteristic equation for input impedance for a transmission line is given by Equation 1-10:

$$Z_{in} = Z_o \frac{Z_L + jZ_o \tan \beta l}{Z_o + jZ_L \tan \beta l} \qquad \text{Equation 1-10}$$

Equation 1-10 can be reduced to $Z_{in} = -jZ_o \cot \beta l$, for the open circuit case. When the line length is 0 λ in length, we expect to see the open circuit termination only, $Z_{in} = \infty$. The voltage and current will measure 2 V and 0 mA, and the reactance will be ∞ Ohms. When the line length is a 0.25 λ, the input impedance will look like a short, making the reactance zero, the voltage 0 V and current 40 mA. When the line length is 0.5 λ, the transmission line will look like an open circuit again. There will be a phase change, but because we are plotting the magnitude voltage and current, the values will be the same as the 0-λ line length.

Execution

Following the same procedure as Problem 2, the final schematic for the open circuit terminated ideal transmission line is given in Figure 1-27.

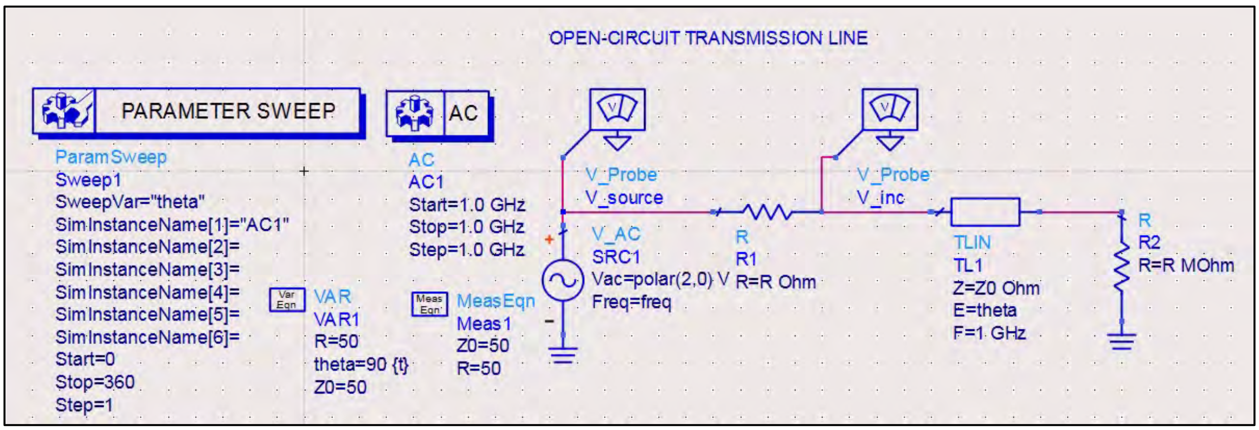


Figure 1-27. Open circuit, terminated ideal transmission line schematic.

In the Data Display window, the following plots for the open circuit reactance, voltage and current are plotted against lambda, similar to Problem 2 (Figure 1-28).

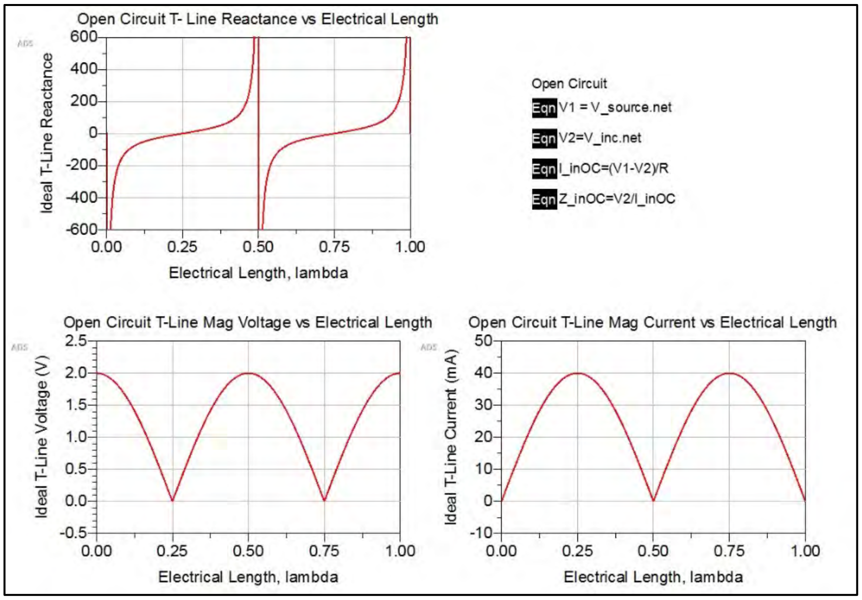


Figure 1-28. Shown here are the plots for the open circuit reactance (top), voltage (bottom left) and current (bottom right) plotted against lambda.

In order to graphically compare the short circuit and open circuit simulations, a second set of equations are created for the short circuit parameters. To grab data from a different simulation and plot it in the current simulation, open the Enter Equation window (Figure 1-29). The drop down menu on the top right is the simulated data selector. Select the Problem 2 data set, then the specific variable, and insert it into the equation text field.

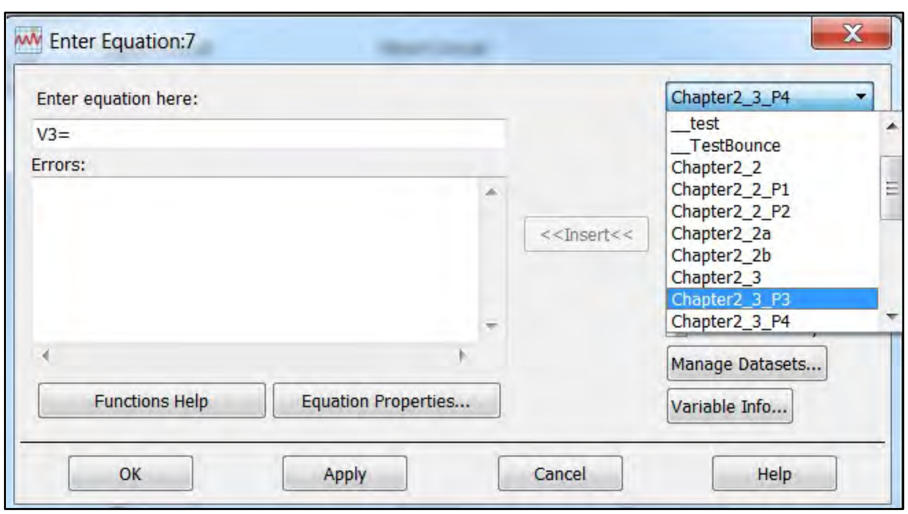


Figure 1-29. Enter Equation window.

The resulting set of equations are created for both the open and short circuit simulations (Figure 1-30). Even though the value of the generator impedance, R, is the same for both simulations, it is better to keep the data items separate and not reuse the R value for the open circuit simulation in the short circuit input current equation.

Open Circuit	Short Circuit
Eqn V1 = V_source.net	Eqn V3=Chapter2_3_P3..V_source.net
Eqn V2=V_inc.net	Eqn V4 = Chapter2_3_P3..V_inc.net
Eqn I_inOC=(V1-V2)/R	Eqn I_inSC=(V3-V4)/Chapter2_3_P3..R
Eqn Z_inOC=V2/I_inOC	Eqn Z_inSC=V4/I_inSC

Figure 1-30. Resulting equations for the open and short circuit simulations.

To plot both reactance traces on the same plot, simply type both equations into the Plot Traces & Attributes window (Figure 1-31).

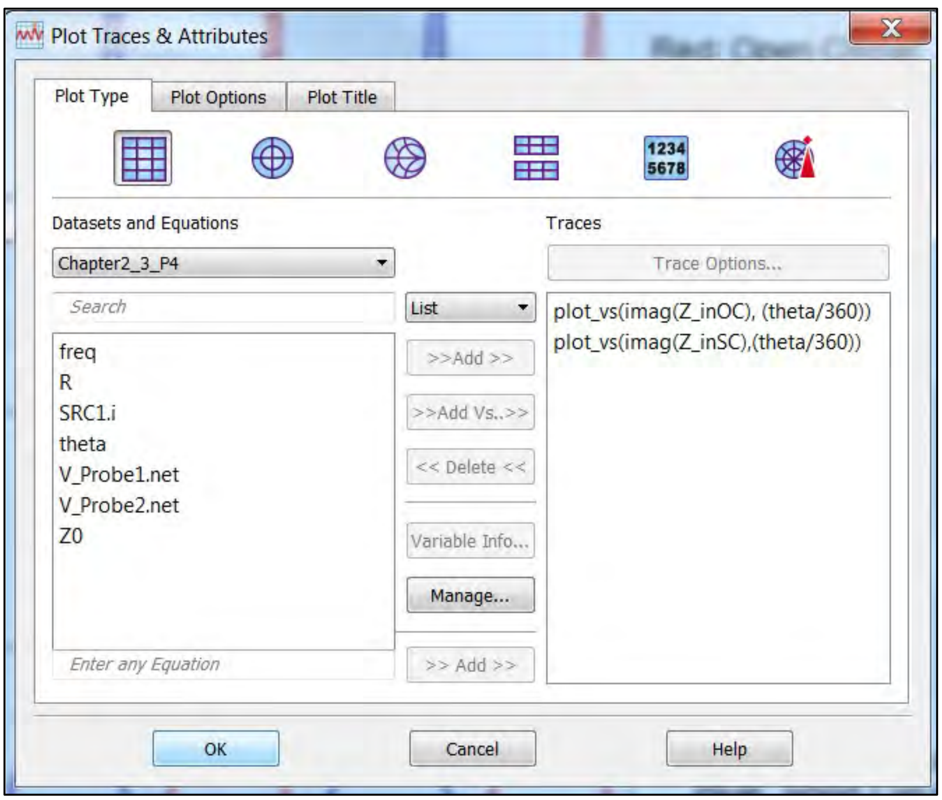


Figure 1-31. Plot Traces & Attributes window.

The three final comparison plots are provided in Figure 1-32.

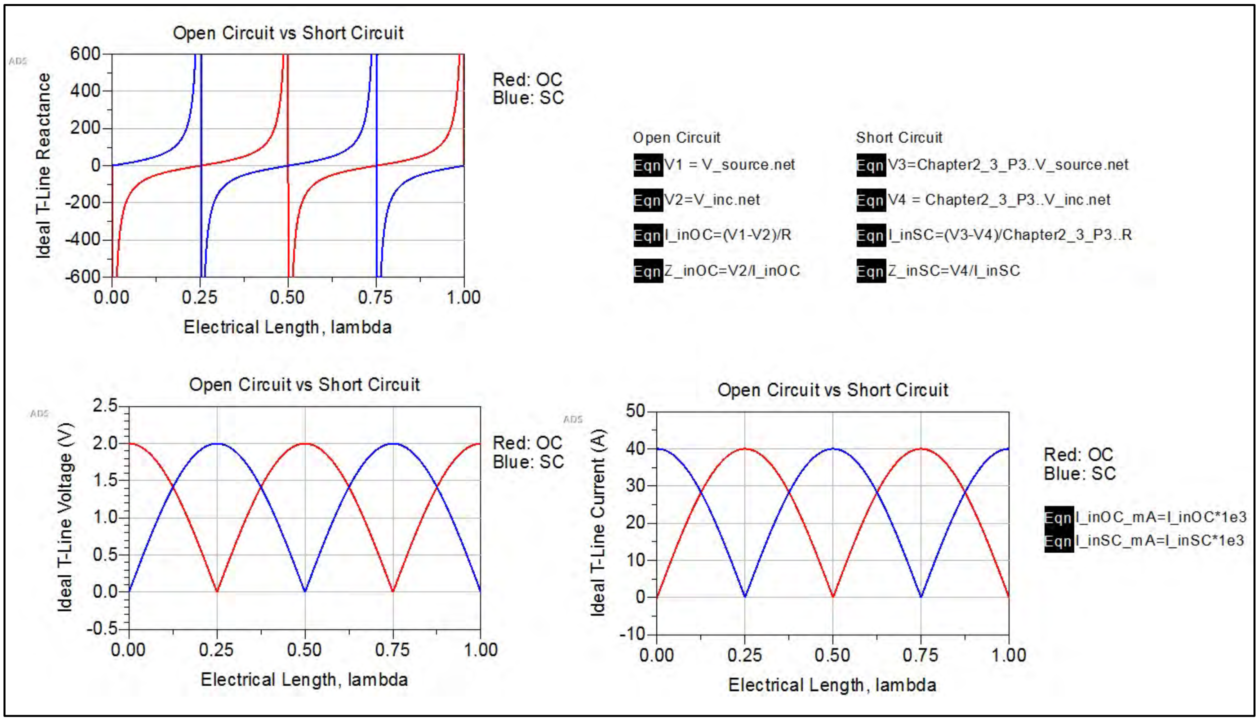


Figure 1-32. Open circuit versus short circuit comparison plots.

Conclusion

The comparison plots reveal that the open circuit and short circuit terminations behave as expected. The open circuit lags behind the short circuit in both current and voltage by 90 degrees. In addition, the open circuit exhibits infinite reactance at a half wavelength as opposed to the short circuit quarter wavelength. This change in impedance is expected when analyzing the fundamental equation for input impedance as shown in Equation 1-10.

For short circuit termination, the equation reduces to $Z_{in} = jZ_o \tan \beta l$, where $Z_{in} = 0$ for $l = 0$, and $Z_{in} = \infty$ for $l = \lambda/4$. For open circuit termination, the equation reduces to $Z_{in} = -jZ_o \cot \beta l$, where $Z_{in} = 0$ for $l = \lambda/4$, and $Z_{in} = \infty$ for $l = \lambda/2$.

1.3 The Quarter-Wave Transformer

Problem 4: Quarter-Wave Matching Transformer Design

Problem Statement

Design quarter-wave matching transformers to match the specified loads to a 50-Ohm transmission line of electrical length 60 degrees at operating frequency 1 GHz, as shown in Figure 1-33:

a) 25 Ohms b) 50 Ohms c) 75 Ohms d) 100 Ohms e) 125 Ohms f) 150 Ohms

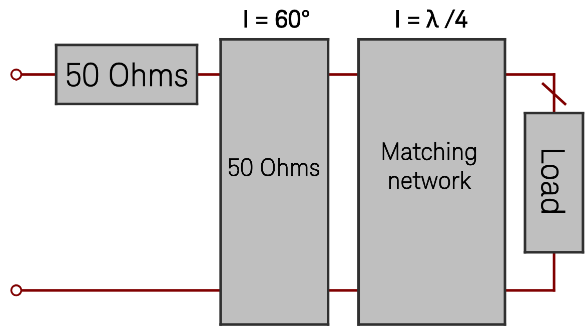


Figure 1-33. Schematic for the design of a quarter-wave matching transformers to match a specified load to a 50-Ohm transmission line.

For each design:

a) Plot the voltage standing wave ratio (VSWR) versus frequency for the circuit, both with and without the quarter-wave transform over the frequency range of 1-10 GHz.
b) Calculate and tabulate the magnitude input impedance for the circuit, both with and without the quarter-wave transform terminated in a 100-Ohm load for the frequency range of 1-5 GHz with step size 1 GHz.
c) Calculate and tabulate the characteristic impedance for each variable load quarter-wave transform.

Solution

Strategy

Use the Batch simulation function in ADS to run all iterations simultaneously. Then, use the slider tool in the Data Display window to separate each run.

What to expect

The circuit without the quarter-wave transform will produce a constant VSWR with respect to frequency. This is because there is only one gamma between the transmission line and load, causing reflections. When the quarter-wave transform is inserted, the VSWR is expected to vary between fully matched and mismatched with respect to frequency. This is due to the reflections between the two terminations, because the quarter-wave matching network is designed to be well matched at the designed frequency. The waves at different harmonics add constructively and destructively, depending on the reflection coefficient at the junction. When the incident and reflected waves add destructively to appear like nothing has been reflected back to the course, the circuit is considered matched.

For the quarter-wave matching circuit, the input impedance is expected to be 50 Ohms for the odd harmonics. The even harmonic values will inversely change with the load value.

The characteristic impedance of each matching section is expected to equal $Z_0 = \sqrt{Z_{in}Z_L}$, following the relationship for the input impedance of a transmission line (Equation 1-10) and solving with l = λ /4.

Execution

a) Open a new schematic and two circuits: the standalone mismatched circuit and a duplicate circuit with the addition of a quarter-wave transform. The two circuits are shown in Figure 1-34.

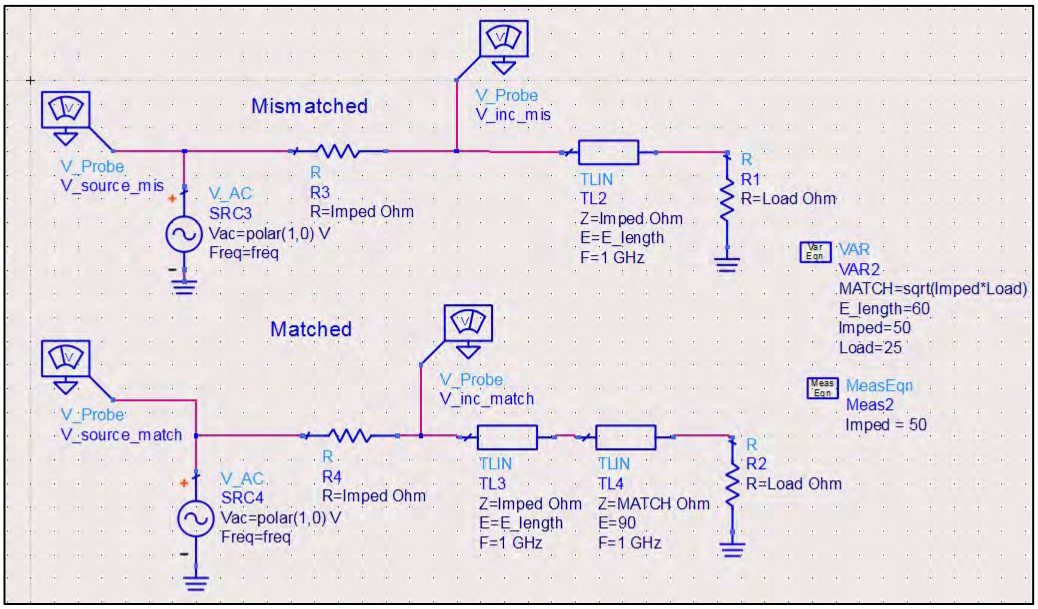


Figure 1-34. Shown here are a mismatched and matched schematic circuit.

The final action is to add the AC-Simulation and Batch Simulation (BatchSimController) components. The simulation frequency range will change between part a, b and c. For part a, the step size needs to be small enough to generate enough data points for a nice, smooth VSWR curve. If a 1-GHz step size is used, the traces will come out in sharp angles. Generally, a 0.01-GHz step size is sufficient.

The Batch Simulation will generate multiple simulations of the schematic circuits with varying loads from 25 to 150 Ohms (Figure 1-35). Due to the problem specifications, a 25-Ohm step size will be used.

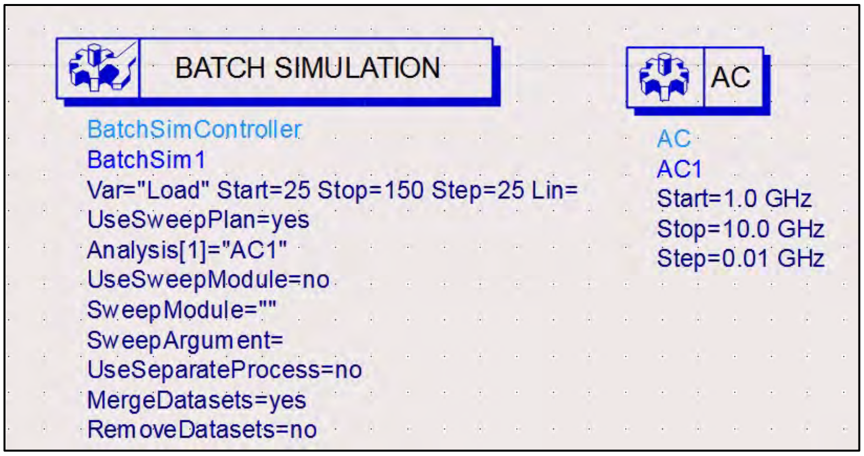


Figure 1-35. With Batch Simulation, multiple simulations of the schematic circuits with varying loads are generated.

After simulating the schematic and opening the Data Display, open a rectangular plot and calculate the VSWR for both circuits using the Equation Writer (Figure1-36).

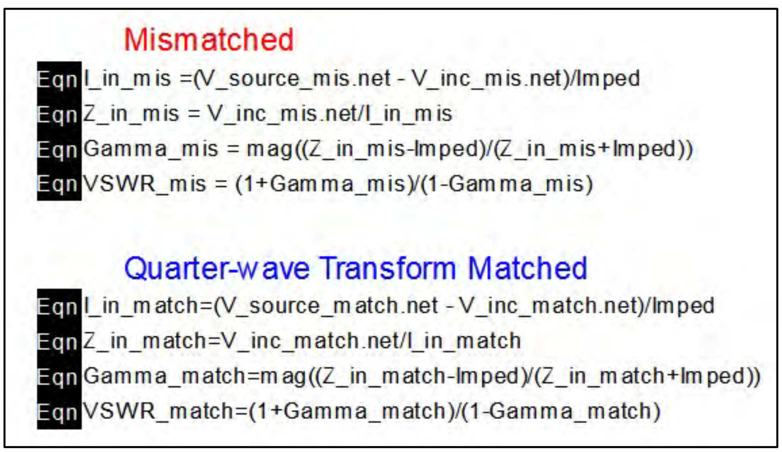


Figure 1-36. Calculation of the VSWR for both the mismatched and quarter-wave transform matched circuit.

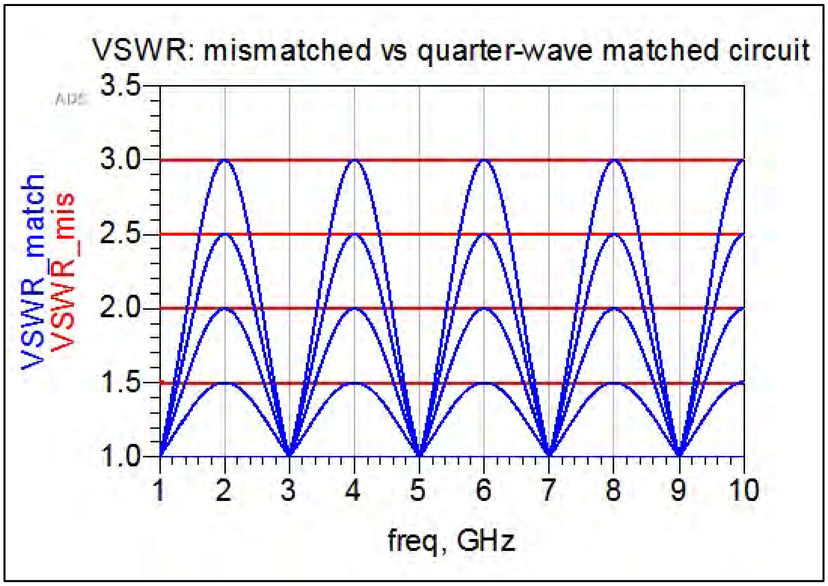


Figure 1-37. Resulting VSWR plot.

The VSWR plot shows each batch simulation for load impedances 25 to 150 Ohms (Figure 1-37). In order to separate each one out, we need to tell the plot command to reference only one element in the array of load terminations. The easiest way to do this is to use a slider tool, located in the Insert menu on the Data Display page. Add the tool onto the display page and a Slider Plot Trace & Attributes window will automatically appear (Figure 1-38). Choose the load variable and add it to the trace section.

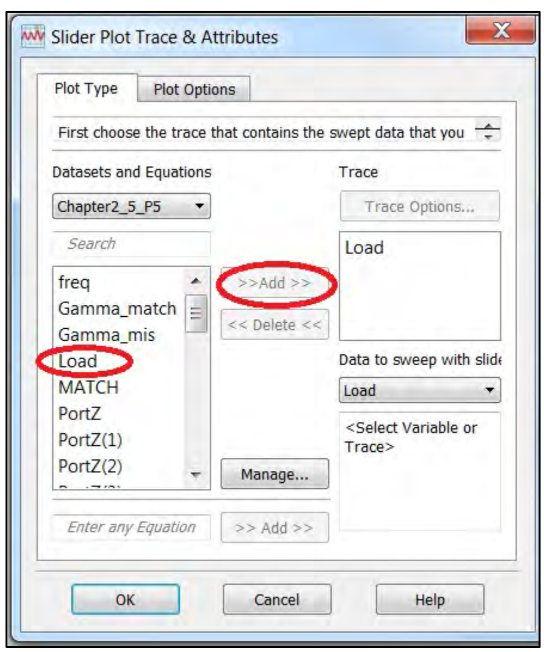


Figure 1-38. The Slider Plot Trace & Attributes window.

The resulting slider appears with default settings (Figure 1-39). Change the value range to 25-150, with 25-Ohm step sizes, to fit the problem specifications.

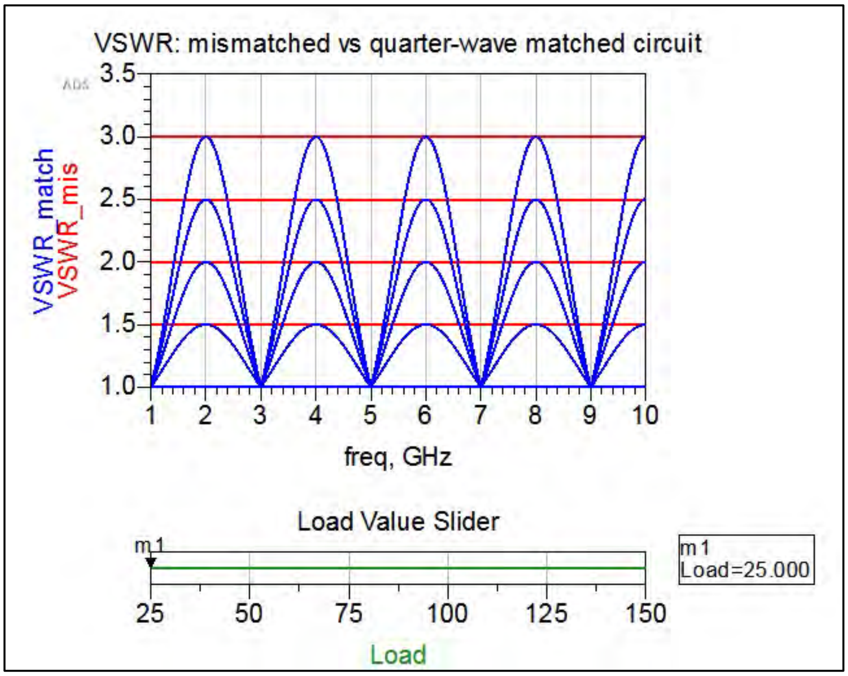


Figure 1-39. Plot of mismatched versus quarter-wave matched circuit with load slider.

The next step is to manually write the index call into the equation. Rather than opening up the Plot Trace & Attributes window again for the plot, the trace can be altered by clicking on the axis label. The string for calling the slider value into the plot is given by [marker#_variablename_index, datapoint]. Because it is desired to have all the data points for each index value of the marker to create a smooth plot instead of one point, the symbol "::" is used to specify all of the data points (Figure 1-40). The final equations for the plot are:

VSWR_mis[m1_Load_index,::]
VSWR_match[m1_Load_index,::]

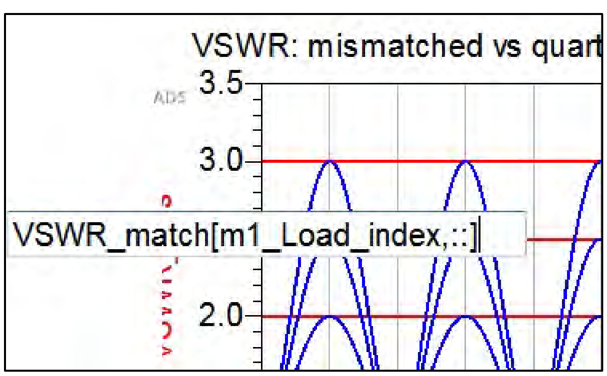


Figure 1-40. Creating a smooth plot rather than just one point.

The resulting plot for a 25-Ohm load is given in Figure 1-41. Changing the value of the marker will simultaneously change the plot.

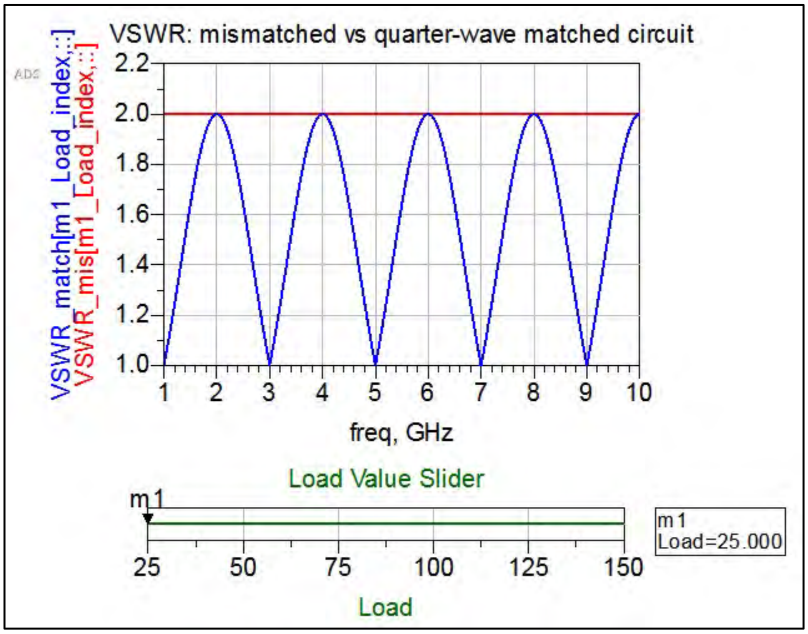
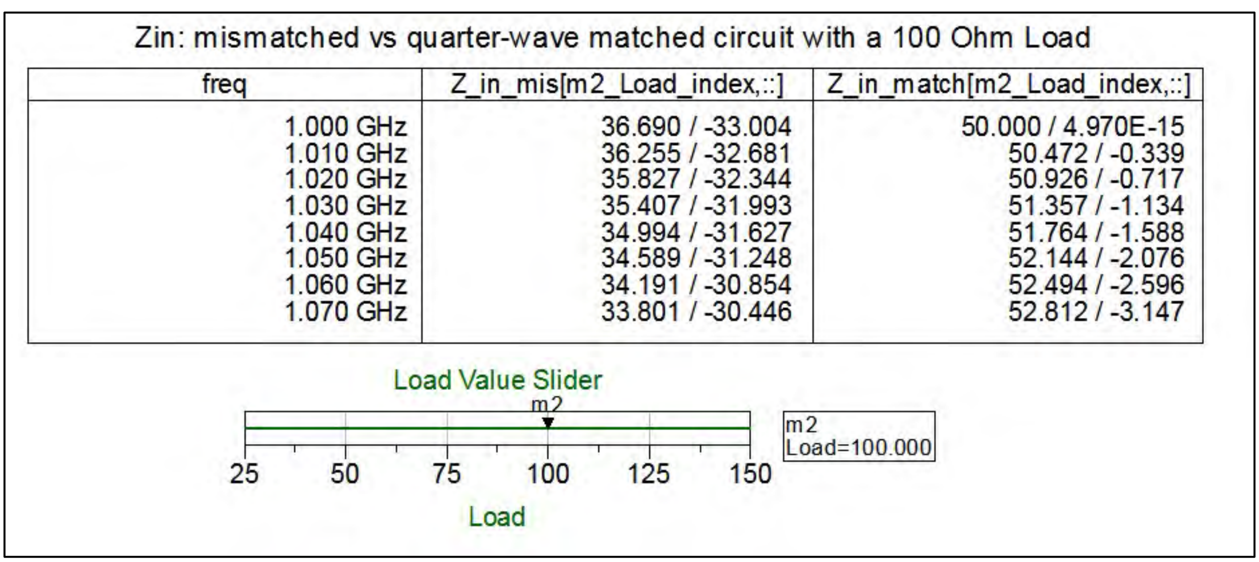


Figure 1-41. The resulting plot for a 25-Ohm load.

b) Add a table component  to the Data Display page. Adding Z_in_mis and Z_in_match to the table will yield impedances at each frequency at each load value. The load in the schematic could manually be changed to 100 Ohms, but since a variable load is needed in part c, another slider tool is used to target the desired data (Figure 1-42).


Zin: mismatched vs quarter-wave matched circuit with a 100 Ohm Load

freq	Z_in_mis[m2_Load_index,::]	Z_in_match[m2_Load_index,::]
1.000 GHz	36.690 / -33.004	50.000 / 4.970E-15
1.010 GHz	36.255 / -32.681	50.472 / -0.339
1.020 GHz	35.827 / -32.344	50.926 / -0.717
1.030 GHz	35.407 / -31.993	51.357 / -1.134
1.040 GHz	34.994 / -31.627	51.764 / -1.588
1.050 GHz	34.589 / -31.248	52.144 / -2.076
1.060 GHz	34.191 / -30.854	52.494 / -2.596
1.070 GHz	33.801 / -30.446	52.812 / -3.147

Figure 1-42. Data for the two circuits with the load changed to 100 Ohms.

Notice that the frequency step is still at 0.01 GHz. Change the step size in the AC simulation component on the schematic to 1.0 GHz, and the frequency range to 1-5 GHz, per the problem specifications, and re-simulate the schematic. The table will update with the new step size. The table data is automatically presented in magnitude and degrees. Double click on the trace expressions to open the Trace Options window and change the Complex Data Format to real/imaginary (Figure 1-43).

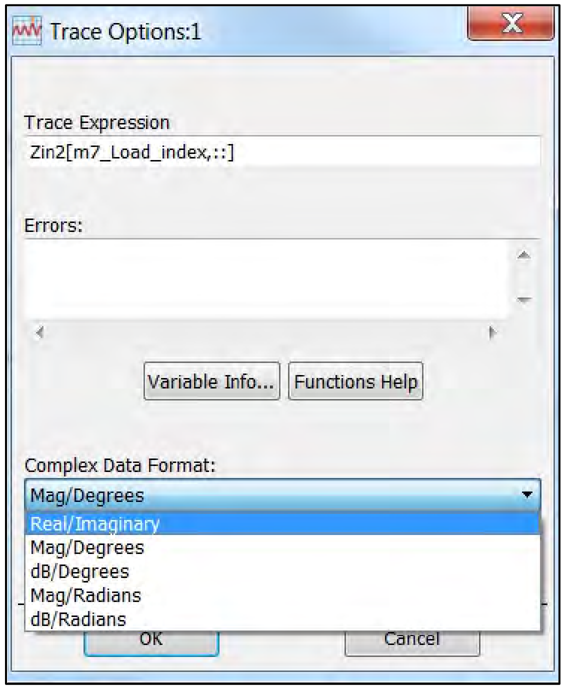


Figure 1-43. The Trace Options window.

c) The MATCH variable on the schematic page already calculates the characteristic impedance for the quarter-wave matching segment. It is frequency independent due to the use of ideal transmission lines. Add a MeasEqn for Match = sqrt(Imped*Load) and simulate the schematic. Add a table to the Data Display window with the variable Match. Since only one variable is tabulated, a slider is not needed.

Conclusion

There are multiple equations for the problem presented here. First, the final plots for VSWR versus frequency for the mismatched circuit and the quarter-wave transform circuit, with varying loads of 25 to 150 Ohms, are provided in Figure 1.44. The mismatched circuit experiences a constant VSWR, while the matching circuit experiences a sinusoidal VSWR that oscillates between perfectly matched and mismatched. This is caused by the multiple reflections that occur during the impedance transformation. As the impedance difference between the load termination and the transmission line increases, the VSWR range also increases, as expected. In this case, all the plots are generated on the same plot and color coded to match for load values. This can be done by double clicking on the trace and selecting Line Color Sequence First.

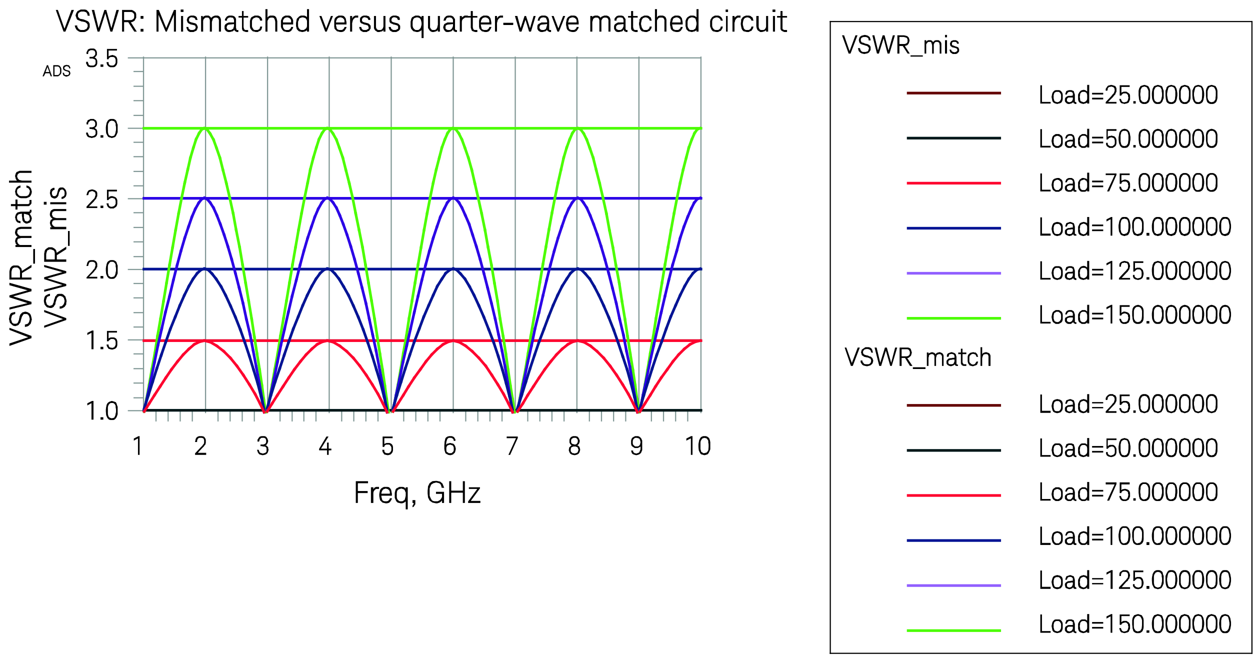


Figure 1.44. Shown here are the final plots for VSWR vs. frequency for the mismatched circuit and the quarter-wave transform circuit with varying loads of 25 to 150 Ohms.

Secondarily, the final table for Zin versus frequency for the mismatched circuit and the quarter-wave transform circuit with a 100-Ohm load is shown in Figure 1-45.

Zin: mismatched vs quarter-wave matched circuit with a 100 Ohm Load

freq	Z_in_mis[m2_Load_index,::]	Z_in_match[m2_Load_index,::]
1.000 GHz	30.769 - j19.985	50.000 + j4.337E-15
2.000 GHz	30.769 + j19.985	30.769 + j19.985
3.000 GHz	100.000 + j0.000	50.000 + j0.000
4.000 GHz	30.769 - j19.985	30.769 - j19.985
5.000 GHz	30.769 + j19.985	50.000 + j4.337E-15

Figure 1-45. The final table for Zin vs. frequency for the mismatched circuit and the quarter-wave transform circuit with a 100-Ohm load.

Finally, the final table for the characteristic impedance of the quarter-wave matching segment is given in Figure 1-46. The quarter-wave matching segment follows the nonlinear relationship of $Z_0 = \sqrt{Z_{in}Z_L}$ and increases concurrently with the load.

Quarter-wave Matching Impedance

Load	Match
25.000	35.355
50.000	50.000
75.000	61.237
100.000	70.711
125.000	79.057
150.000	86.603

Figure 1-46. Final table for the characteristic impedance of the quarter-wave matching segment.

1.4 Generator and Load Mismatches

Problem 5: Conjugate Matching for Maximum Power Transfer

Problem Statement

Create a lumped-element matching network to connect a 50 - j40 Ohm load to a generator with impedance 25 + j30 Ohm to achieve maximum power transfer at frequency 2 GHz (Figure 1-46).

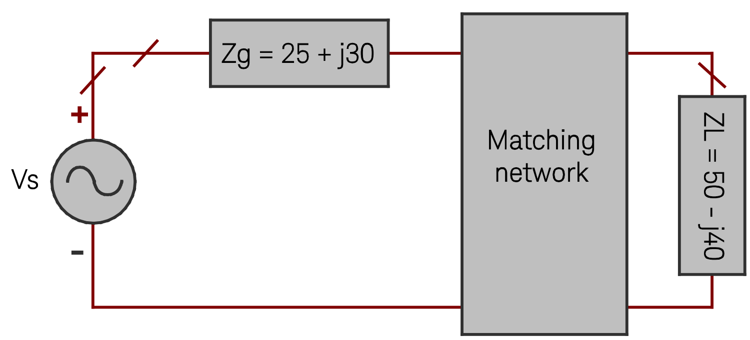


Figure 1-46. Shown here is a schematic for a lumped-element matching network to connect a 50 - j40 Ohm load to a generator with an impedance of 25 + j30 Ohm.

This problem involves two steps:

a) Match the generator using its complex conjugate. Graph the power delivered to the load.
b) Match the generator using a simple two-element network using the Smith chart. Compare the result to part a.

Solution

Strategy

Match the generator using its complex conjugate. This can be accomplished using a reverse L network with reactive components (Figure 1-47).

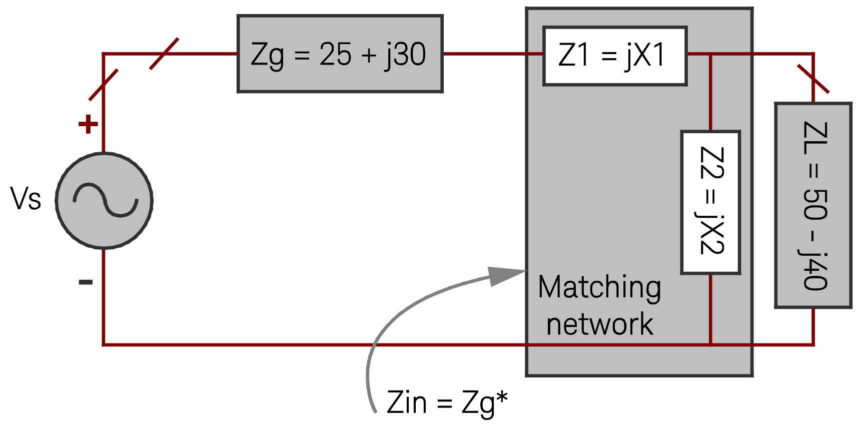


Figure 1-47. Matching the generator using a reverse L network with reactive components.

Then, use the ADS Smith Chart tool to match using lumped elements and compare this result to the hand calculation. Use AC simulation with an arbitrary input voltage of 5 volts to calculate the power transfer as given by Equation 1-11:

$$P = \frac{1}{2}|V_{in}|^2 Re\left\{\frac{1}{Z_{in}}\right\}. \quad \text{Equation 1-11}$$

What to expect

The conjugate matching method is expected to give the maximum power to the load at the designed frequency. The power should then roll off at either side of the center frequency. This is due to the relationship for power, as defined by Equation 1-11. Expanding this Equation results in Equation 1-12.

$$P = \frac{1}{2}\left|V_g\right|^2 \frac{R_{in}}{\left(R_{in}+R_g\right)^2+\left(X_{in}+X_g\right)^2} \quad \text{Equation 1-12}$$

Maximizing power with respect to the input impedance results in:

$$\frac{\partial P}{\partial R_{in}} = 0 \rightarrow R_{in} = R_g \quad \text{Equation 1-13}$$

$$\frac{\partial P}{\partial X_{in}} = 0 \rightarrow X_{in} = -X_g \quad \text{Equation 1-14}$$

Equations 1-13 and 1-14 reveal that for maximum power delivery to the load, $Z_{in} = Z_g^*$.

The simple two-element matching method is expected to drive a lower amount of power to the load. The two-element, L-matching network is chosen randomly based on ease of the Smith Chart utility. Note that this may not necessarily provide the optimal L-matching network for the given source and load impedances.

Execution

a) The generator and load impedance lumped-element equivalents at 2 GHz are:
Generator: 25 + j30 = 25-Ohm resistor and 2.38732-nH inductor in series.
Load: 50 – j40 = 50-Ohm resistor and 1.98944 pF-capacitor in series.

Solving for Z_{in}:

$$Z_{in} = Z_g^* = Z_1 + \frac{Z_2 Z_L}{Z_2 + Z_L}$$
$$25 - j30 = \frac{-X_1 X_2 + (X_1 + X_2)(40 + j50)}{50 + j(X_2 - 40)}$$

Simplify and equate real and imaginary parts:

$$X_2 = -100 - 2X_1$$
$$X_1 = -30 \pm 37.7492$$

Choose solution X_1 = -67.7492, X_2 = 35.4984

Solve for components at 2 GHz.

$$Z_1 = -j67.7492 = \frac{-j}{wC}$$
$$C = 1.17459\ pF$$
$$Z_2 = +j35.4984 = jwL$$
$$L = 2.82487\ nH$$

Open a new schematic and create the lumped-element network with a 5-volt source (Figure 1-48).

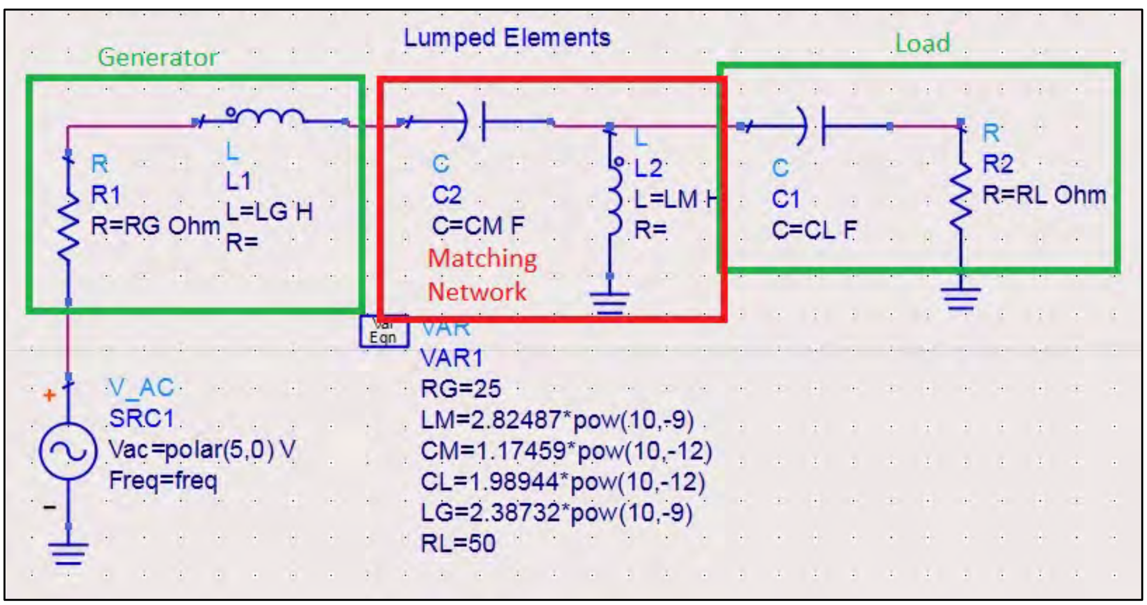


Figure 1-48. Schematic of a lumped-element network with a 5-volt source.

Adding the voltage probes, simulation components and measured equations yields the following final schematic for the lumped-element circuit using the complex conjugate matching network (Figure 1-49).

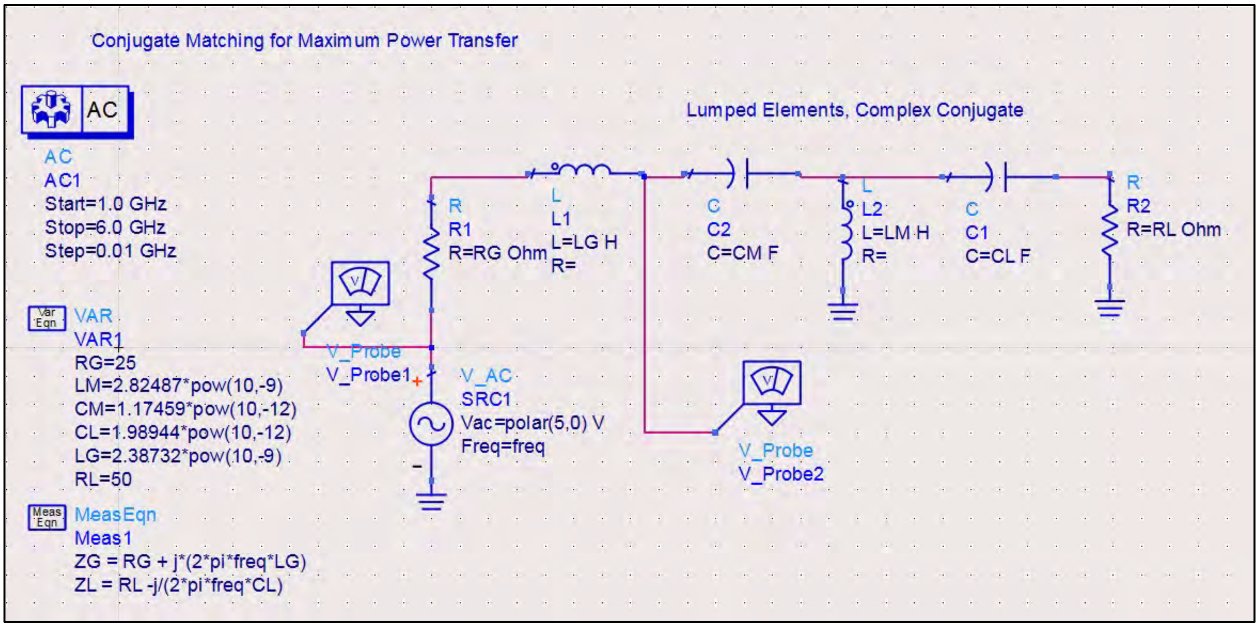


Figure 1-49. The final schematic for the lumped-element circuit using the complex conjugate matching network.

The resulting plot on the Data Display window shows that at 2 GHz, the conjugate matching network produces the maximum power transfer (Figure 1-50).

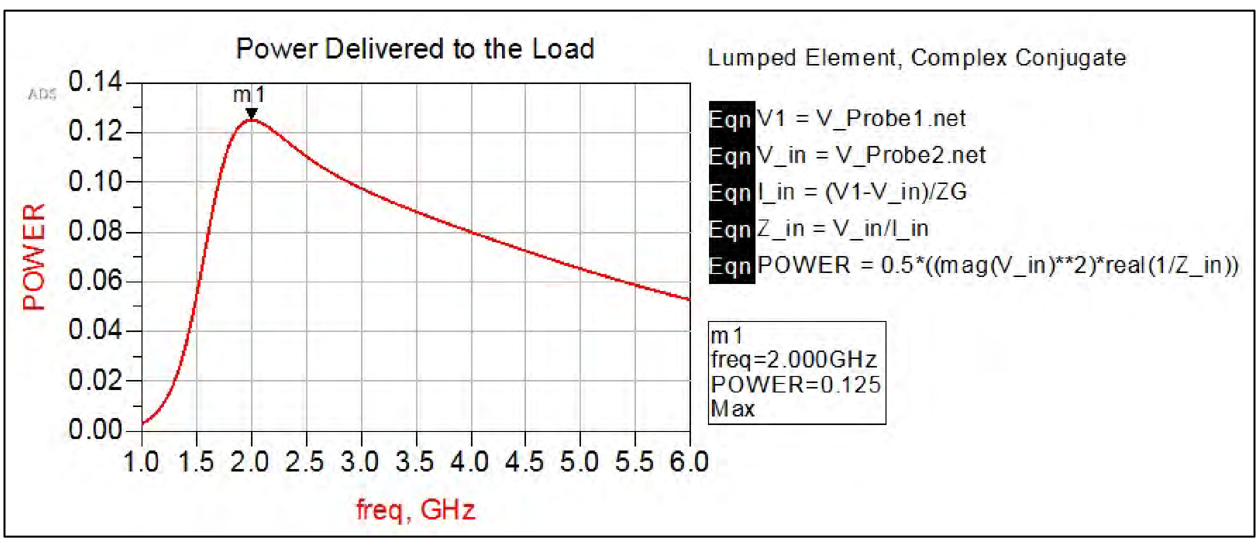


Figure 1-50. The resulting plot as shown on the Data Display window.

b) First Smith chart option: Open the Smith Chart tool in the Tool menu. The default impedance settings for the source and load are 50 Ohms. The default frequency is 1 GHz. Change the frequency to 2 GHz; also change the source and load impedances. Keep the normalization value at 50 Ohms. Repeat for the load impedance, Z_L (Figure 1-51).

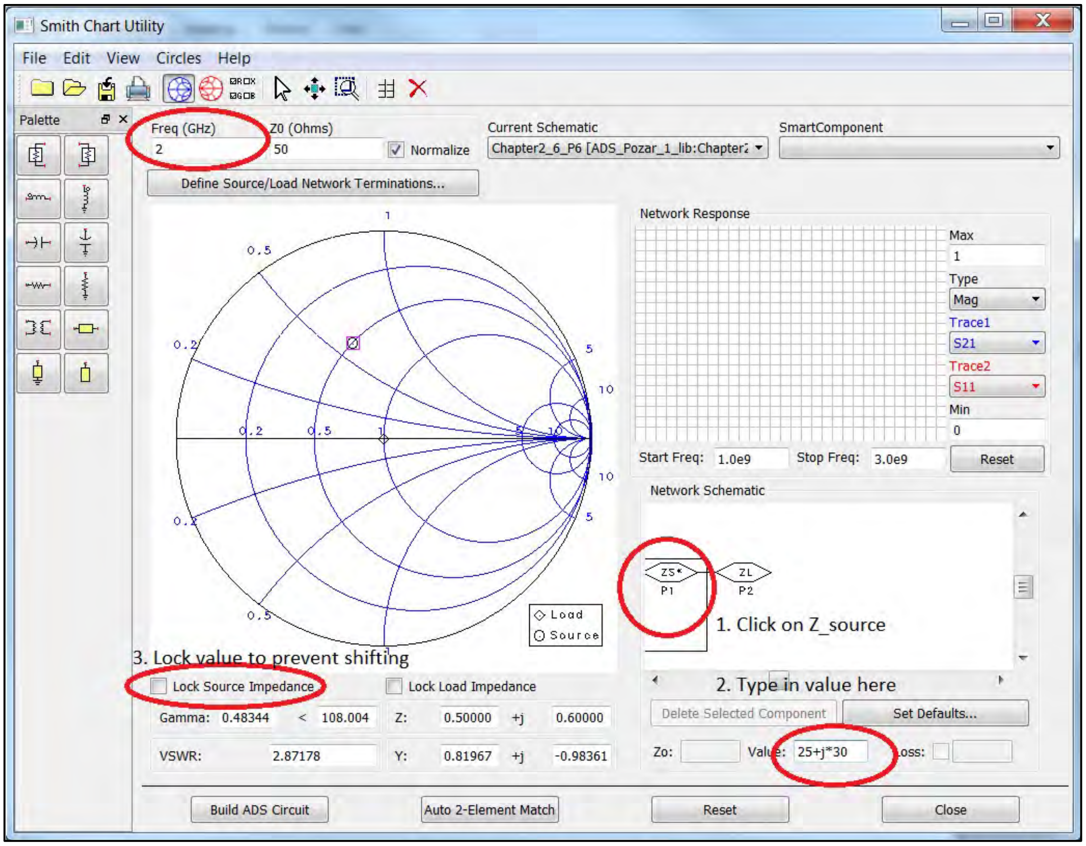


Figure 1-51. The Smith Chart tool in the ADS Tool menu.

The Smith chart is made up of constant resistance (r) and conductance (g) circles. For lumped elements, Figure 1-52 displays their movement along the Smith chart.

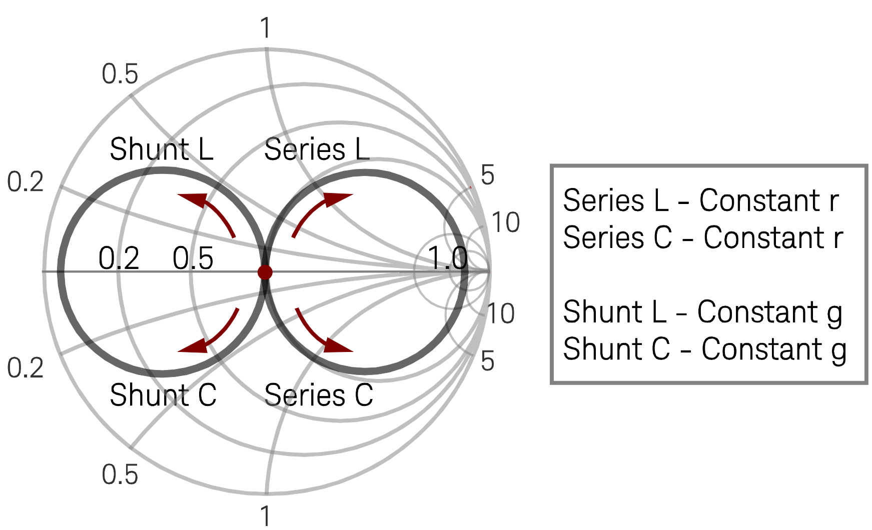


Figure 1-52. The movement of r and g for lumped elements on a Smith chart.

To begin matching, click on the source Z_S port and add a shunt inductor to reach a constant r = 0.5 circle (Figure 1-53). Although difficult to reach exactly, once the marker is close enough, the value can be typed into the impedance text field right beneath the Lock Load Impedance check box. It may take a couple of iterations, but the value will eventually reach 0.50000 + j0.75498.

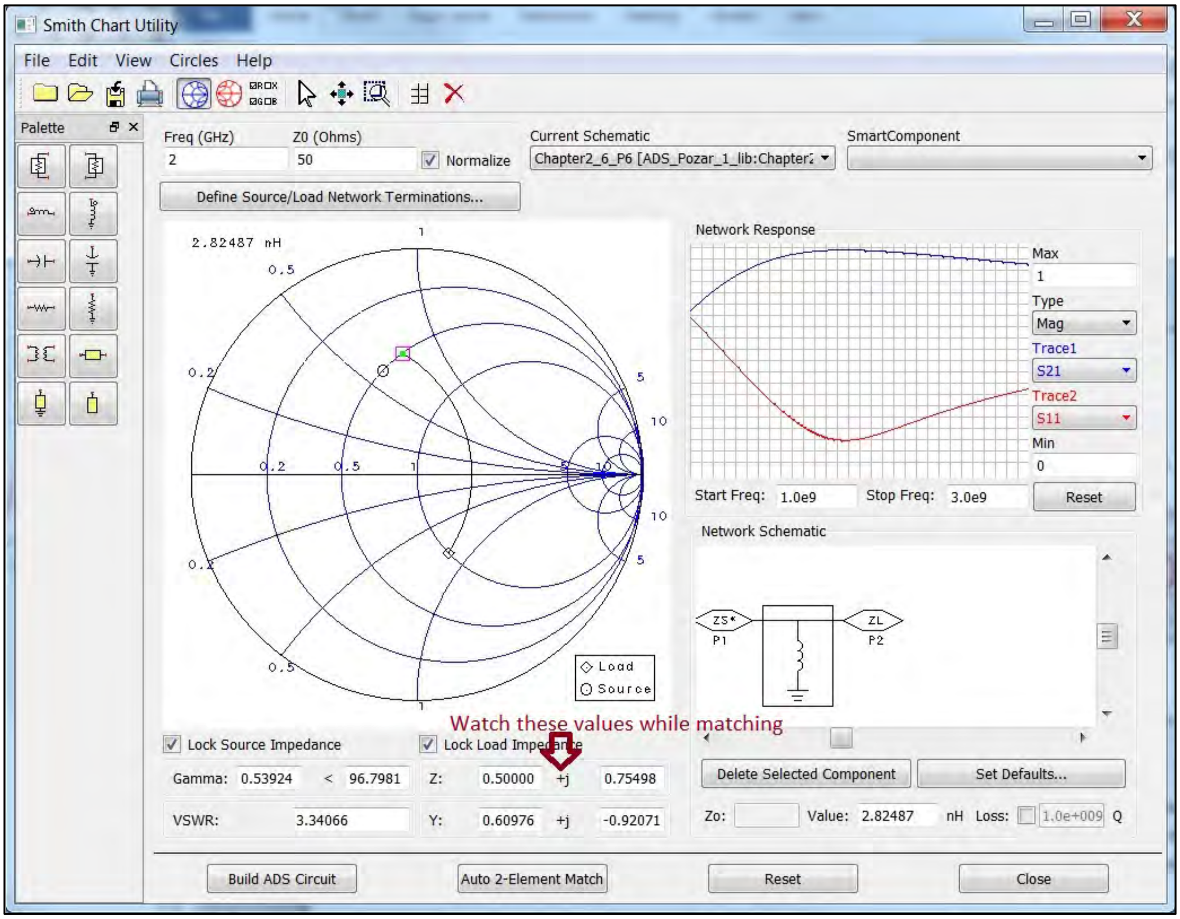


Figure 1-53. Reaching a constant r = 0.5 circle on a Smith chart.

With the shunt inductor highlighted in the network schematic field, add a series capacitor until the point Z = 0.50000 + j0.60000 is reached, the impedance of the source (Figure 1-54). Be sure that the capacitor is highlighted in the network schematic before typing into the reactance text field. If the inductor is still highlighted, the inductor value will shift.

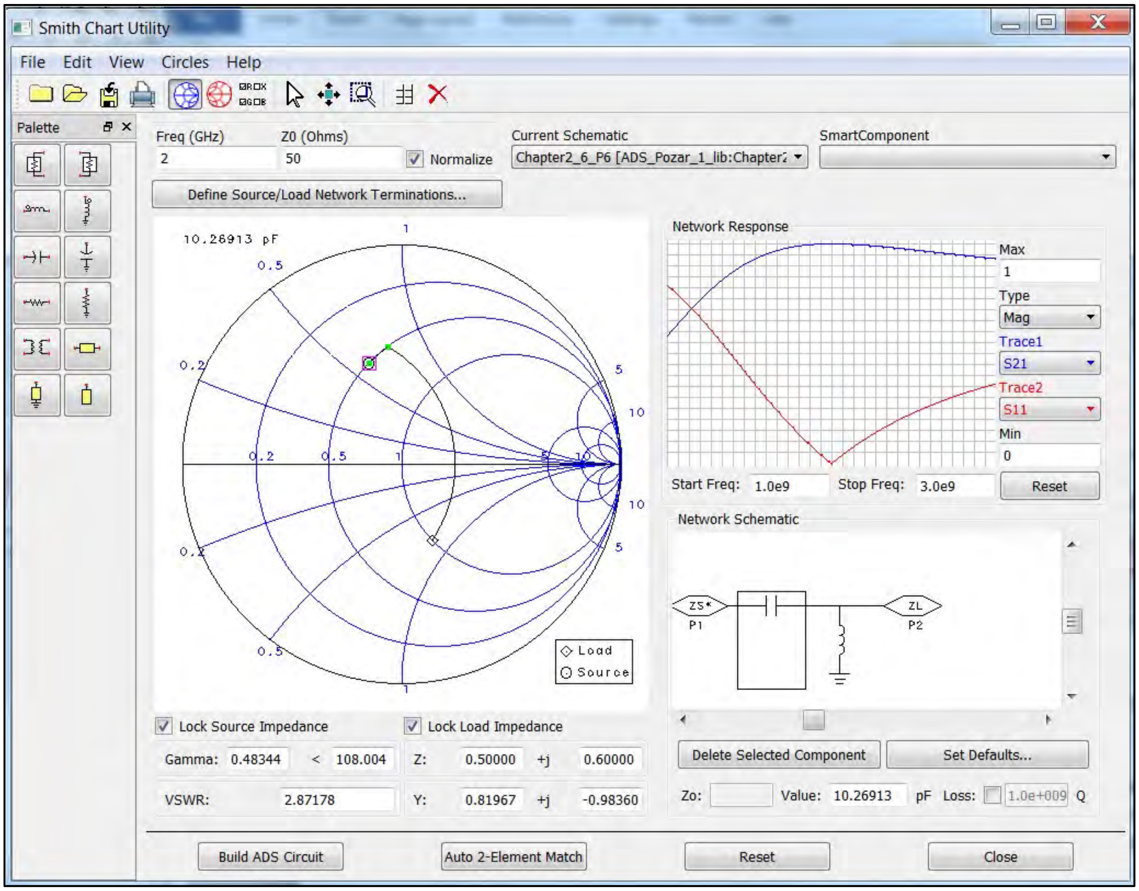


Figure 1-54. Here, a series capacitor is added to the shunt inductor until the point Z = 0.50000 + j0.60000 is reached.

Record the inductor and capacitor values and place them into a duplicate lumped-element circuit (Figure 1-55).

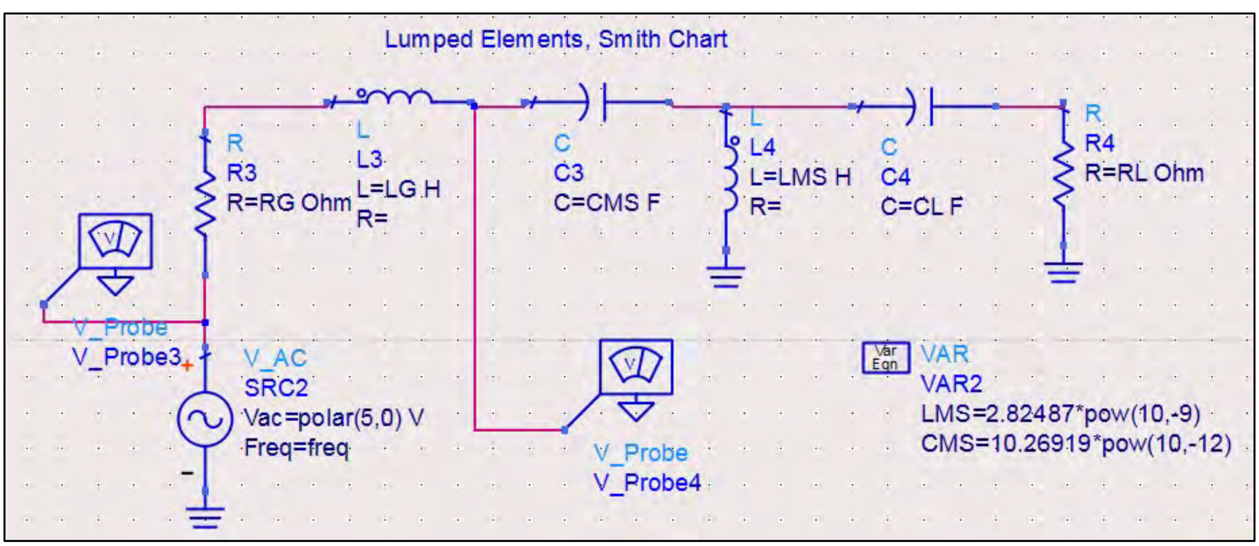


Figure 1-55. A duplicate lumped-element circuit.

The power transfer for the Smith chart matching network is overlaid onto the plot for part a. While both versions are considered matched, the blue line is not conjugate matched, which is why the peak at the design frequency of 2 GHz is not seen (Figure 1-56). The red trace is conjugate matched and the maximum power transfer to the load is achieved.

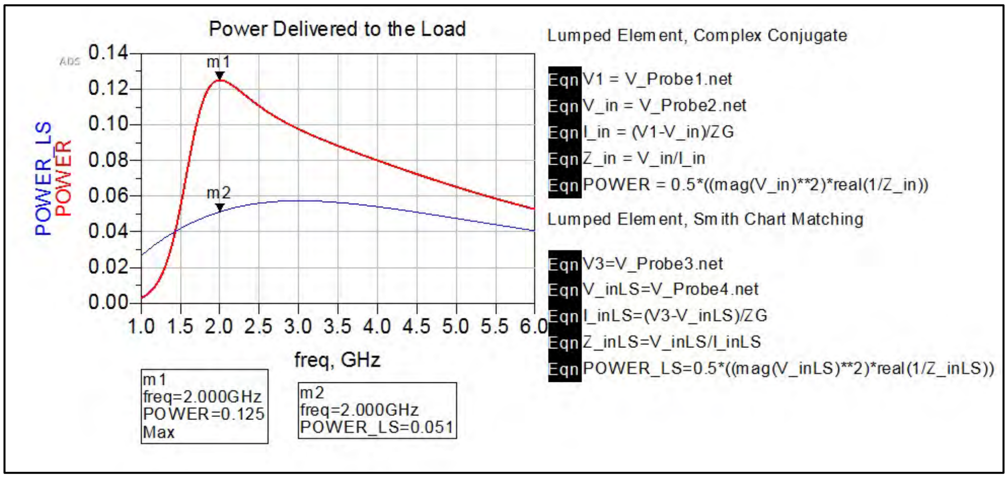


Figure 1-56. In this graph, the blue line is not conjugate matched, while the red trace is conjugate matched.

Second Smith chart option: Another feature of ADS is the Smith Chart Smart Component utility, which automatically creates a basic two-element matching network. Duplicate the lumped-element network, remove the matching elements and insert a Smith Chart Matching Network component  between the generator and load. The component can be found in the Smith Chart Matching palette (Figure 1-57).

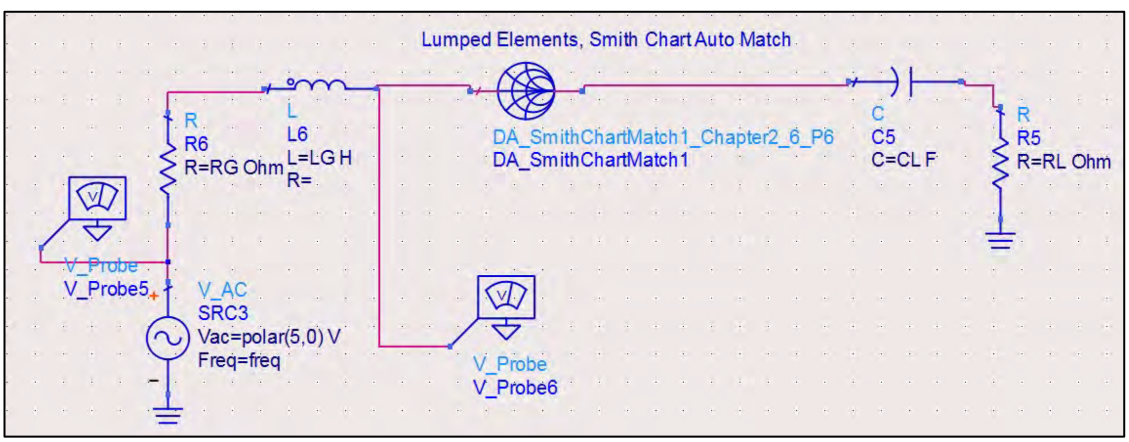


Figure 1-57. Using the ADS Smith Chart Smart Component Utility option to create a basic two-element matching network automatically.

Open the Smith Chart tool and select Update Smart Component from the Smith Chart Utility in the pop-up menu. Make sure the Smart Component is selected in the Smith Chart tool. Update the frequency, source and load impedances as before. Click Auto 2-Element Match at the bottom of the tool and a Network Selector window will appear (Figure 1-58). For comparison reasons, select the two-inductor element network.

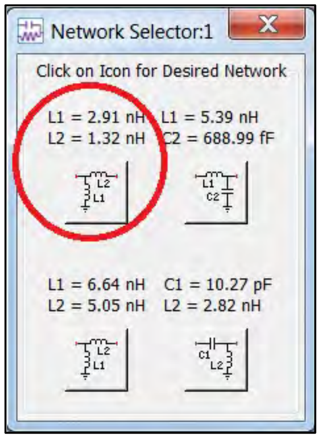


Figure 1-58. When the Network Selector window appears, select the two-inductor element network.

Note that the values for each element are auto-generated for each network type within the Network Selector window, and the CL network used previously has values that are almost exact to the Smith chart method used previously. Add the two-element CL network to the Smith Chart tool and select Build ADS Circuit located at the bottom of the window. Save and simulate the schematic. Plot the power delivered to the load and compare that to the previous Smith chart matching method. A plot of the two methods reveals that they yield different frequency responses that converge at the designed frequency of 2 GHz (Figure 1-59).

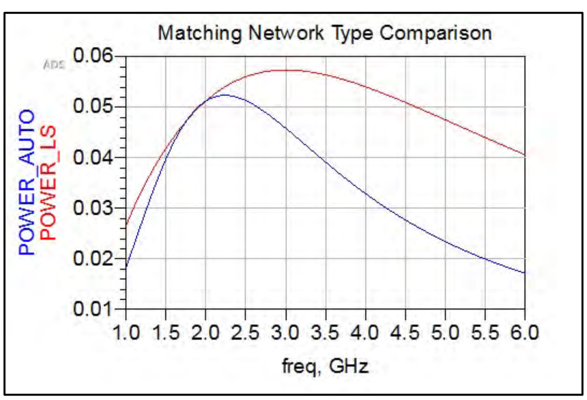


Figure 1-59. Plotting the two methods yields different frequency responses converging at 2 GHz.

Conclusion

For this problem, the complex conjugate-matching network solution provided the maximum power transfer. The simplest Smith chart method does provide a working matching network; however, there are an infinite number of possibilities with only one providing the optimal network. The overlapping plots show that the Smith chart method not only provided less power to the load because it is not conjugate matched, but also the maximum power occurred around 3 GHz, and not the specified 2 GHz. In addition, while using different matching L networks will provide the same power at the designed frequency, this is not true for neighboring frequency bands. Because of this, it is important to choose the correct matching network dependent on the design specifications.

1.5 Transients on Transmission Lines

Problem 6: Transient Propagation – Bounce Diagram

Problem Statement

The circuit shown in Figure 1-60 is excited by a rectangular pulse of duration **τ** = 0.5 µs. The transmission line is 100 m long with a phase velocity of 1×10^8 m/s. Plot the net voltage with respect to time for a duration of 7 µs.

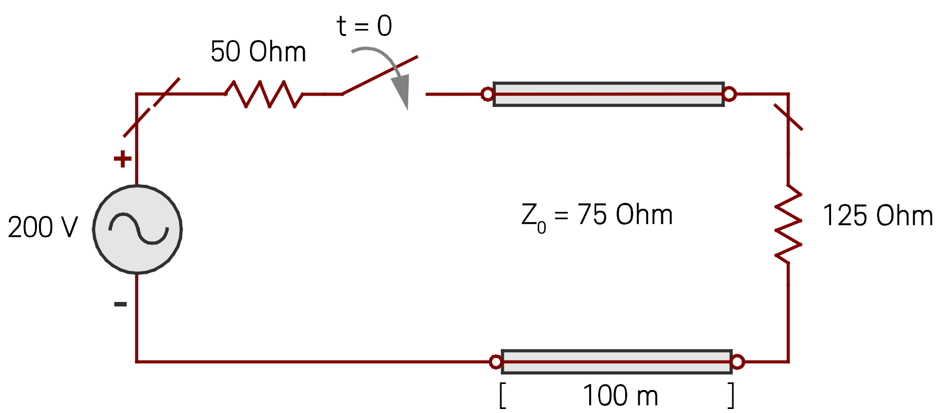


Figure 1-60. Shown here is a circuit excited by a rectangular pulse of duration **τ** = 0.5 µs.

Solution

Strategy

Split the rectangular pulse into two step functions; $V(t) = V_1(t) + V_2(t) = 200\mu(t) - 200\mu(t - 0.5\ \mu s)$ Perform Time Domain Reflectometry (TDR) analysis on both functions and superimpose them to find the net voltage response with respect to time.

What to expect

The propagating signal voltage will not be the initial t = 0 value. Instead, it will steadily increase as reflections due to mismatch that is incorporated. As $t \rightarrow \infty$, the reflections will die out and reach a steady-state response. Transient analysis will graphically show the reflected waves values with respect to time at the viewpoint of the input side. With the pulse duration of 0.5 µs, and a simulation time of 7 µs, one would expect to see a negligible reflection at the end of the simulation, thereby reaching the steady-state value for the input signal.

The steady-state value for the propagating signal, also denoted by $V_1(t)$, is calculated as:

$$V_\infty = \frac{V_g Z_L}{Z_g + Z_L} = \frac{(200\,V)(125\,Ohm)}{50\,Ohm + 125\,Ohm} = 142.857\,V \qquad \text{Equation 1-15}$$

Execution

To address this problem, there are a number of necessary calculations that must be performed, including the period of time for line travel, as given by:

$$T = \frac{l}{v_p} = \frac{100\,m}{1\,x\,10^8\,m/s} = 1\mu s \qquad \text{Equation 1-16}$$

The schematic for the active part of the rectangular pulse, $V_1(t)$ is shown in Figure 1-61. The delay for the transmission line is the variable T, and the step function has a high of 200 volts and a low of 0 volts.

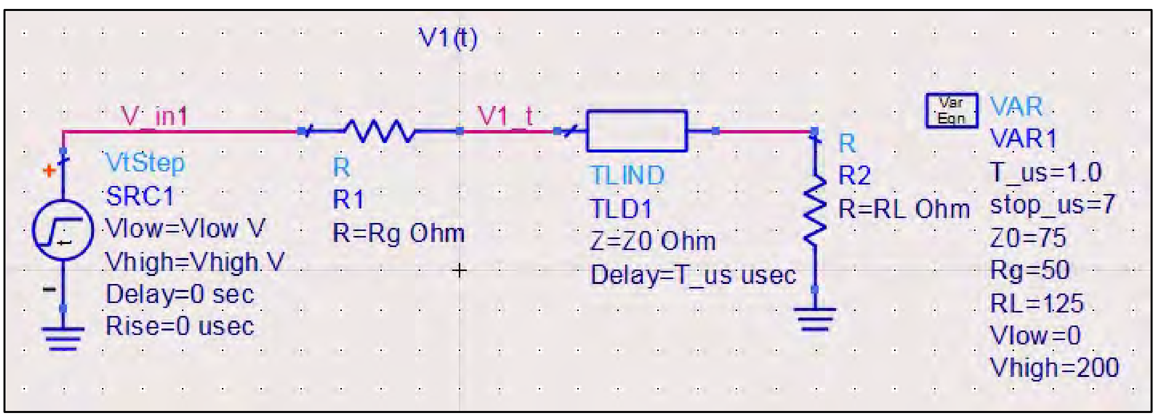


Figure 1-61. The schematic for the active part of the rectangular pulse, $V_1(t)$.

The diagram for $V_1(t)$ is duplicated for $V_2(t)$ in Figure 1-62; however, in this case the high voltage is now 200 volts and the signal is delayed by **τ** = 0.5 µs.

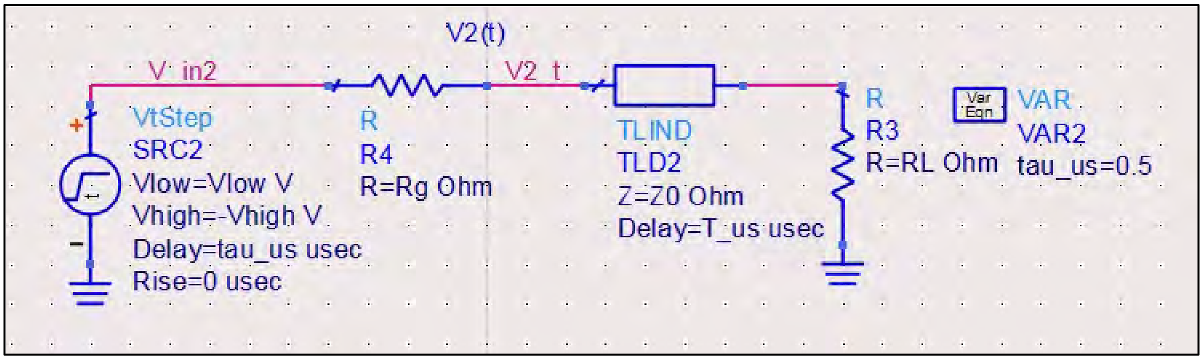


Figure 1-62. The schematic for the active part of the rectangular pulse, $V_2(t)$.

Next, the transient analysis simulation component, found in the Simulation-Transient palette, is added along with the total voltage measured equation component. The final schematic is shown in Figure 1-63.

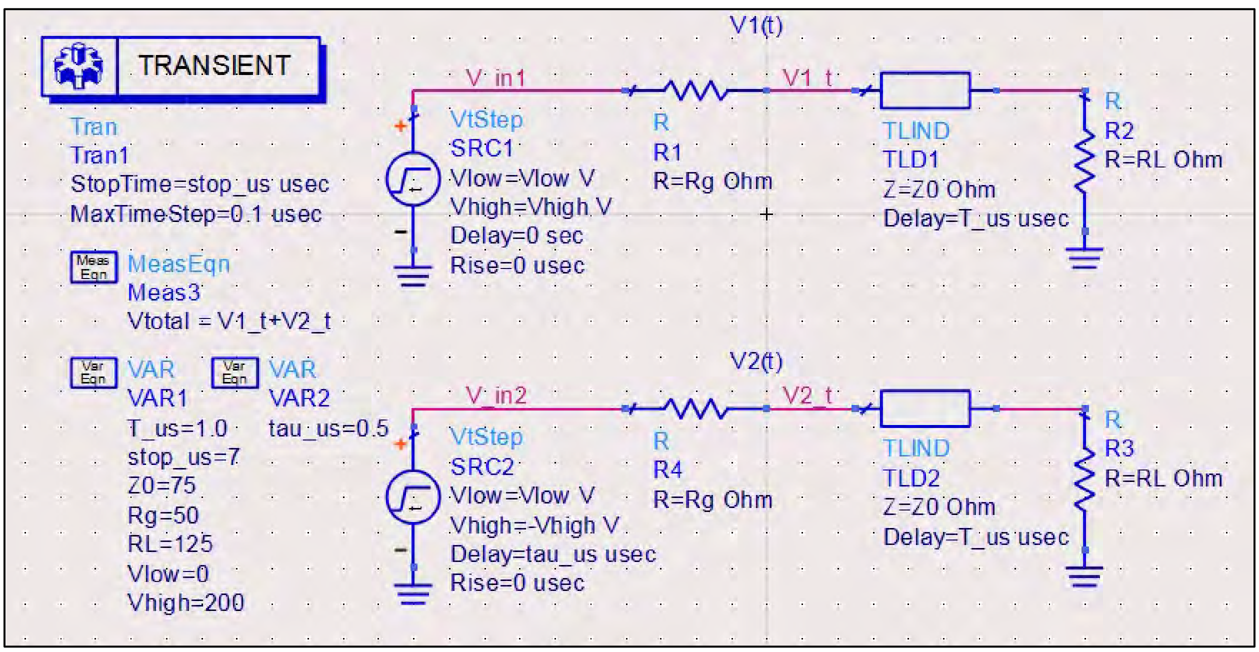
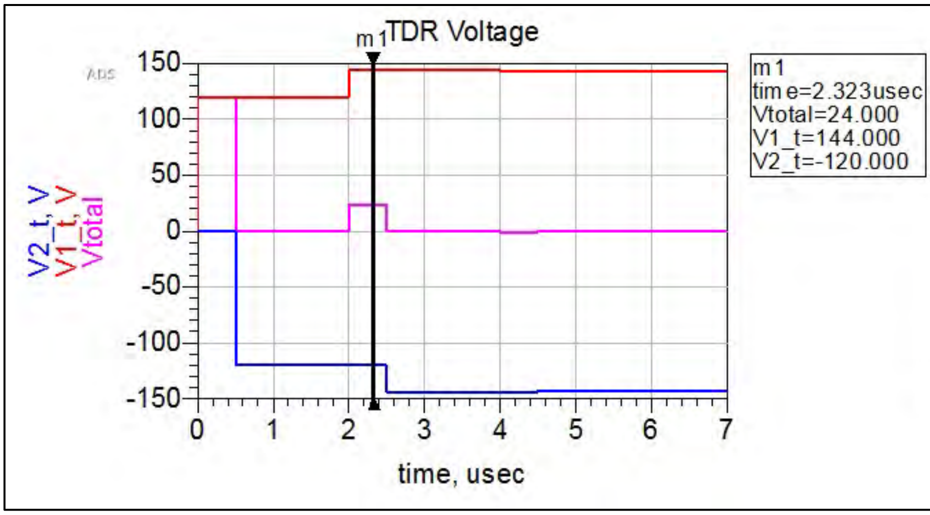


Figure 1-63. The final schematic for a circuit excited by a rectangular pulse of duration $\tau = 0.5$ μs.

Both step functions and the net voltage are graphed on a rectangular plot (Figure 1-64). The change in net voltage with time is tabulated for convenience.



Time (us)	Vtotal (V)
0-0.5	120.0
0.5-2.0	0.0
2.0-2.5	24.0
2.5-4.0	0.0
4.0-4.5	-1.2
4.5-6.0	0.0
6.0-6.5	0.06
6.5-7.0	0.0

Figure 1-64. A rectangular plot of both step functions and the net voltage.

Conclusion

The net voltage trace demonstrates the transient response of the reflections, which die out over time as expected. At τ= 7.0 µs the propagating signal reaches a value of 142.86 volts, which is almost the steady-state value of 142.857 volts. The conventional method for demonstrating TDR analysis is to use the bounce diagram. The conventional bounce diagram relating to this problem is given in Figure 1-65 for the rectangular pulse.

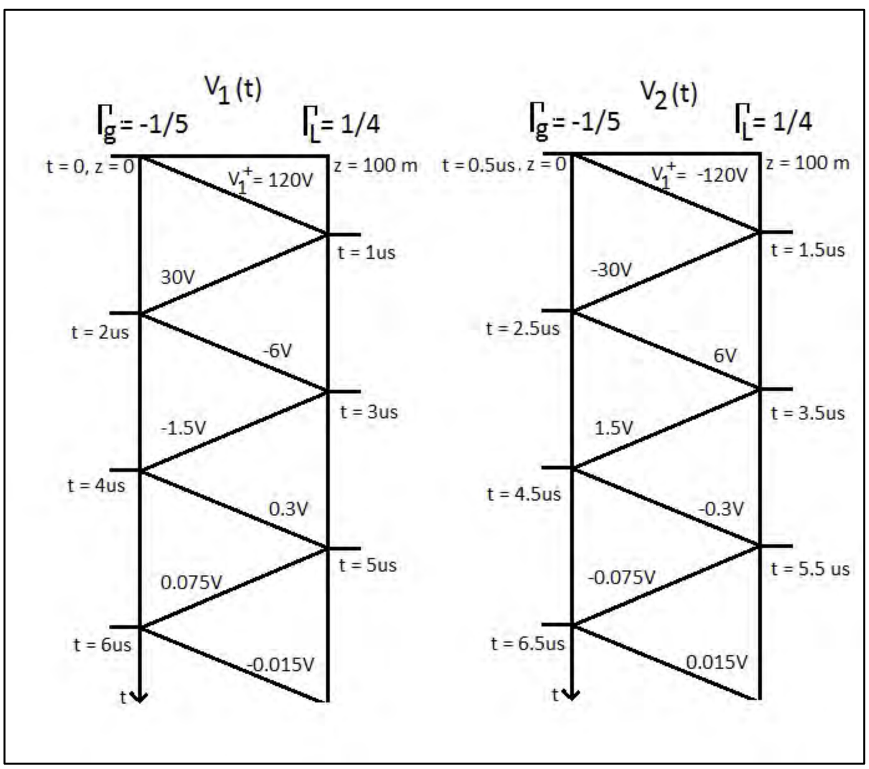


Figure 1-65. The conventional bounce diagram relating to this problem.

Their respective voltage plots are given in Figure 1-66, which agree with the ADS result.

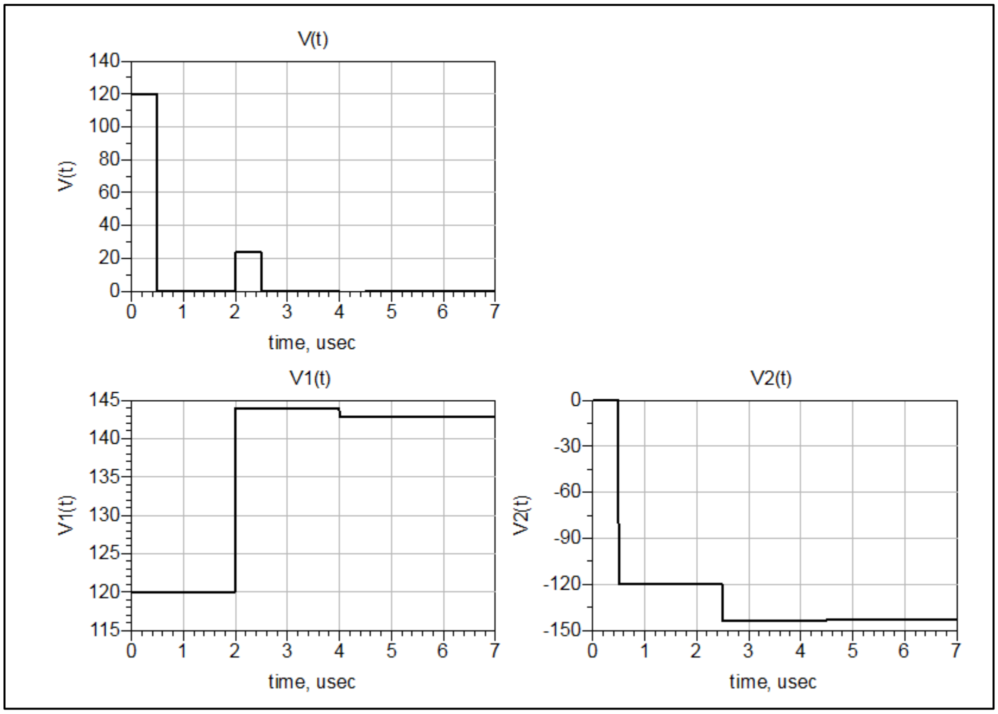


Figure 1-66. Voltage plots for V(t), V_1(t) and V_2(t).

Chapter 2 – Microstrip Line

Problem 1: Frequency Dependence in a Microstrip Line

Problem Statement

Show graphically how the characteristic impedance and effective dielectric constant of a microstrip line change with frequency.

Solution

Strategy

Create a realistic model using a multi-layer component rather than an ideal microstrip line because the multi-layer substrate uses a 2D field solver and frequency-dependent model for the dielectric. Using a basic two-port network, use S_{11} as the reflection coefficient to find the characteristic impedance, Z0, of the line. S-parameters will be covered in more depth in Chapter 3, but we are using them here specifically so we may isolate the effective dielectric constant. Strategically utilizing a quarter-wavelength line will provide an easier calculation of Z0. Plotting changes in the effective dielectric constant, Dk_eff, will be found by unwrapping the phase of the S_{21} parameter and manipulating it using the following equations:

$$S_{21} = Ae^{-j\theta} \qquad \text{Equation 1-17}$$

Where

$$\theta = \beta l = \omega \frac{l}{v} = \omega \frac{l\sqrt{\varepsilon_{eff}}}{c} \qquad \text{Equation 1-18}$$

What to expect

At higher frequencies, the electric dipoles will not be able to react as quickly since the electric field is changing. This causes the permittivity of the substrate to decrease, which in turn causes the characteristic impedance of the line to increase with frequency, as they are inversely related. Therefore, we expect the effective dielectric constant to decrease with frequency, and the characteristic impedance to increase.

Execution

Open a new schematic within the current workspace. In the TLines-Multilayer palette, place a MLSUBSTRATE2 onto the schematic. Place a VAR component next to it with the variables for dielectric constant (Dk), phase velocity (v_prop_mPsec), lambda (lambda_m) and length (length_m). The default values for the substrate will be kept, except for the dielectric constant and loss tangent.

Next, add a ML1CTL_C component from the same TLines- Multilayer palette onto the schematic and terminate both ports with a TERMG. We will be using S-parameter analysis since we are using S-parameters to extract both Z0 and Dk_eff. S- parameters require each port to be terminated with a TERM. Change the length of the component to length_m and the width W to 20.0 mil. Add the S_Param simulation component from the Simulation-S_Param palette and set the range from

10 MHz to 50 GHz. Double click on the component to set the Frequency Sweep Type to Log with 1000 Pts/decade. The schematic should now appear the same as Figure 2-1.

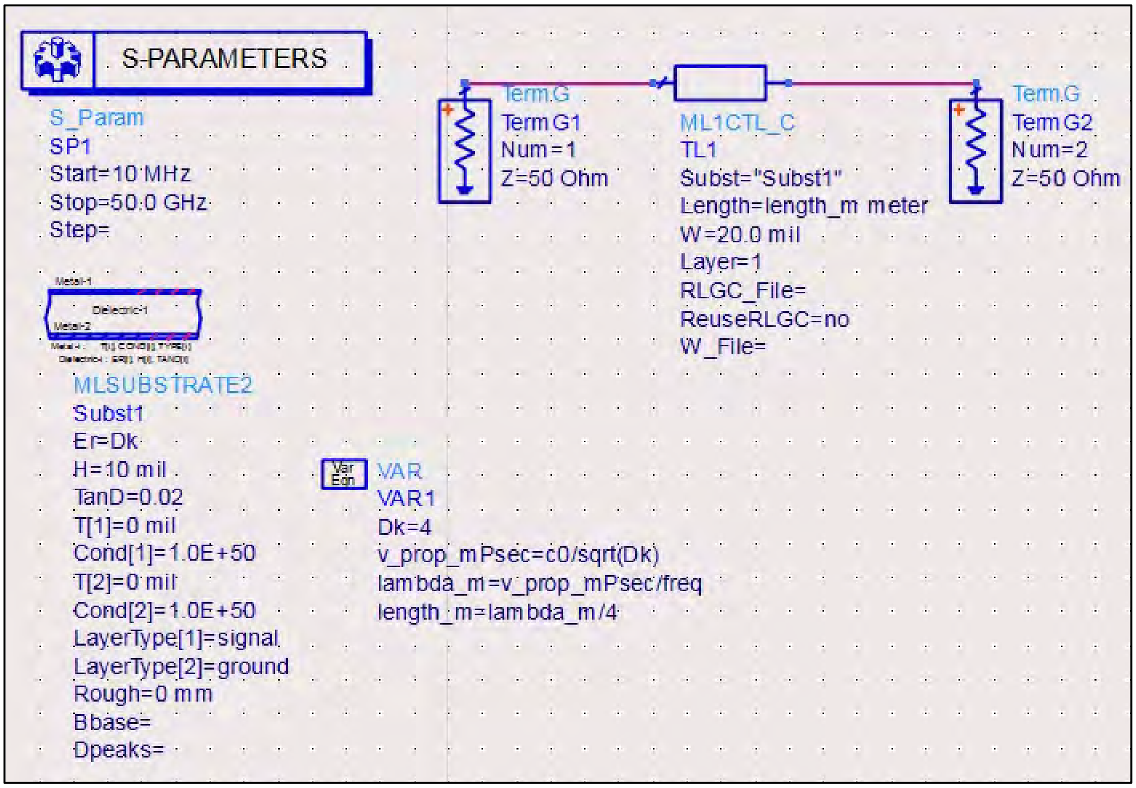


Figure 2-1. The new schematic within the current workspace.

The multilayer microstrip line is ready for simulation, and now all that remains is to create the measured equations for plotting Z0 and Dk_eff. Enter in the equations as shown in Figure 2-2.

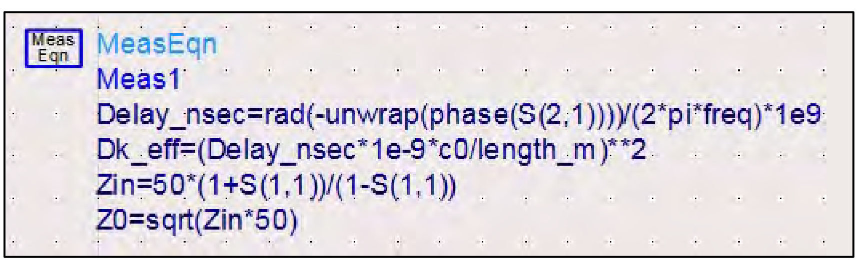
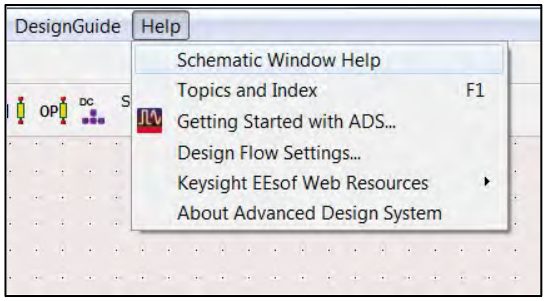


Figure 2-2. Shown here are the measured equations for plotting Z0 and Dk_eff.

Documentation on the functions and components used in ADS can be found using the Schematic Window Help in the Help menu. Type the name of the function or component into the search field to reveal more details (Figure 2-3).



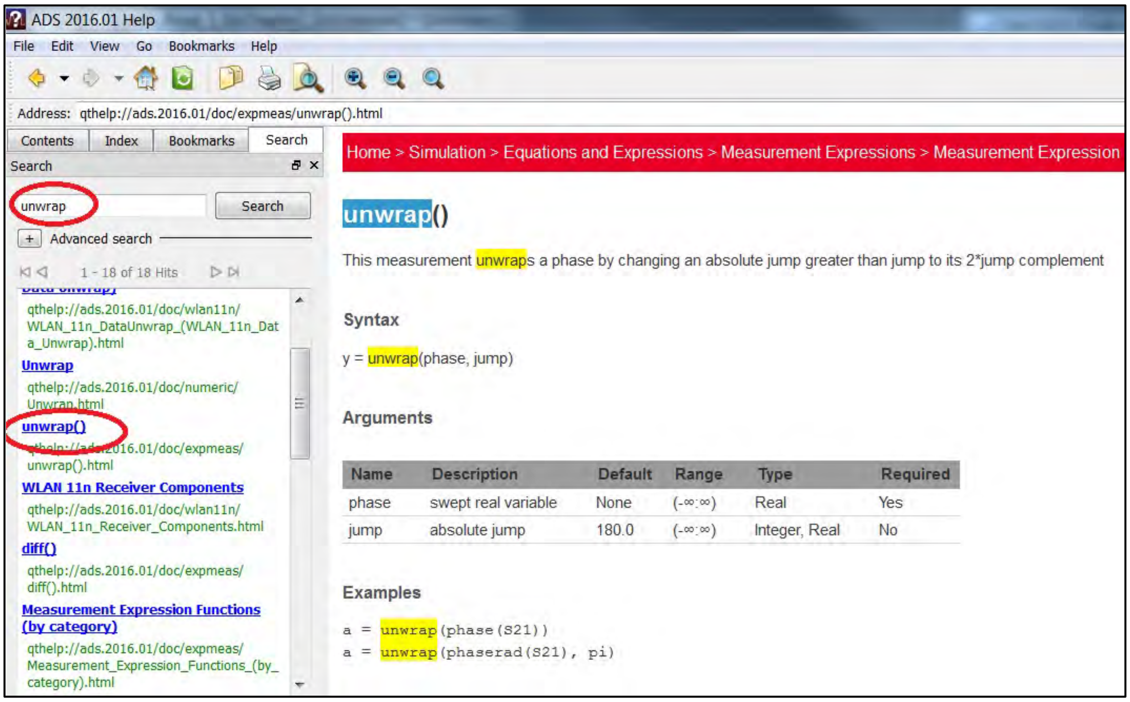


Figure 2-3. Shown here is the Schematic Window Help in the Help menu.

Simulate the schematic using the Simulation symbol  to open the Data Display window. Because Z0 = R0 + jX0, plot both the real and imaginary parts of Z0 versus frequency. On a separate rectangular plot, graph Dk_eff versus frequency.

Conclusion

The plots showing the frequency dependence of the characteristic impedance and effective dielectric constant are shown in Figure 2-4. As expected, the characteristic impedance increases with frequency until eventually it plateaus, similar to Dk_eff, which decreases with frequency.

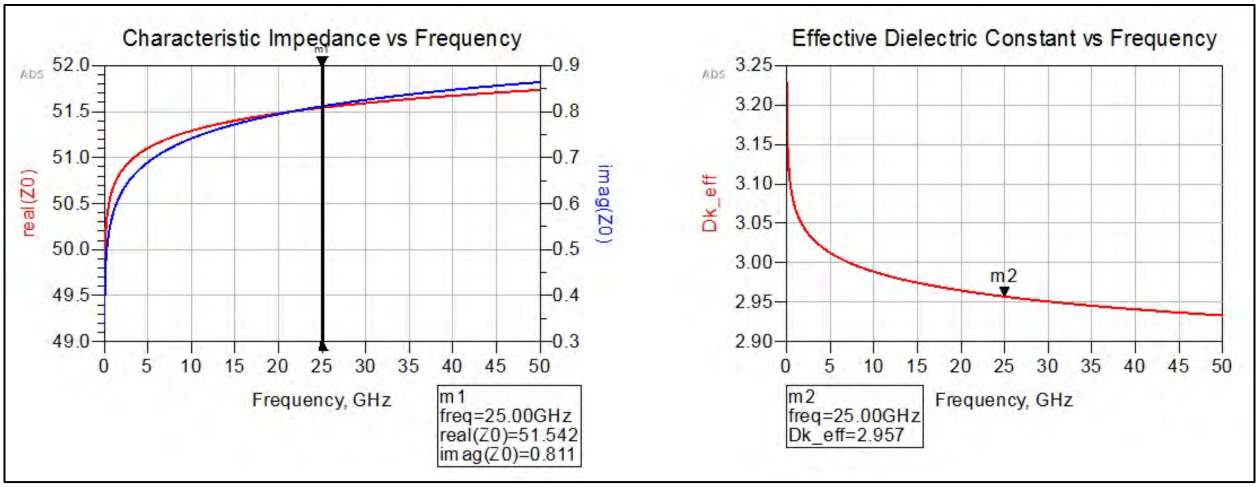


Figure 2-4. Plots showing the frequency dependence of the characteristic impedance and effective dielectric constant.

Problem 2: Microstrip Line Design

Problem Statement

Design a 50-Ohm lossless microstrip line with the following specifications:

Circuit board metal:	Copper
Circuit board metal thickness:	1.4 mil
Circuit board dielectric thickness:	20 mil
Dielectric constant:	2.1
Operating frequency:	10.0 GHz
Phase delay:	145°

a) Design the microstrip line both by hand using two different models: The I. J. Bahl and D. K. Trivedi Microstrip Line model and the E. Hammerstad and Ø. Jensen Microstrip Line model. Plot the insertion loss of the microstrip lines over the frequency band 10 MHz – 30 GHz.

b) Assume reasonable dielectric losses. Compare the result to one of the lossless responses in part a.

c) Create an ideal transmission line of the same electrical length, operating frequency and input impedance. Compare it to the responses in part a and b.

d) Show dispersion within the lossless microstrip line. Compare this result to the ideal transmission line.

Solution

Strategy

For part a, use the given parameters to calculate the needed width and length of the microstrip line by hand using the Bahl and Trivedi model. The ADS LineCalc tool uses the Hammerstad and Jensen model. Use this tool for the second model comparison. Use S-parameter analysis to plot the insertion loss, S_{21} parameter, for the frequency response in dB. For part b, compare the lossless and lossy circuits with the assumption that tan**δ** = 0.0002 is a reasonable dielectric loss tangent. For part c, use an ideal transmission line component and compare that result to the ADS lossless case. For part d, show dispersion by plotting the phase velocity versus frequency. If the phase velocity changes with respect to frequency, the line experienced dispersion.

What to expect

The frequency response is expected to have less insertion loss at lower frequencies with a steady increase in frequency. The insertion loss can be attributed to the dielectric and conductor loss of the microstrip line. This concept will be fully realized when comparing the microstrip line to the ideal transmission line, which is expected to be lossless. The microstrip line is expected to be dispersive in correlation with the changes in effective dielectric constant. The phase velocity of the microstrip line is expected to gradually increase until it plateaus, while the phase velocity of the ideal transmission line is expected to be a constant value of c.

Execution

To design a microstrip line to parameter specifications using Hand Calculation: I. J. Bahl and D. K. Trivedi Model[2], assume W/d > 2.

$$B = \frac{377\pi}{2Z_0\sqrt{\epsilon_r}} \qquad \text{Equation 2-1}$$

$$B = \frac{377\pi}{2 * 50\sqrt{2.1}}$$

$$B = 8.173$$

$$W/d = \frac{2}{\pi}\left[B - 1 - ln(2B - 1) + \frac{\epsilon_r - 1}{2\epsilon_r}\left\{ln(B - 1) + 0.39 - \frac{0.61}{\epsilon_r}\right\}\right] \qquad \text{Equation 2-2}$$

$$W/d = \frac{2}{\pi}\left[8.173 - 1 - ln(2 * 8.173 - 1) + \frac{2.1 - 1}{2 * 2.1}\left\{ln(8.173 - 1) + 0.39 - \frac{0.61}{2.1}\right\}\right]$$

$$W/d = 3.17307$$

$$W = 3.17307d = 3.17307(20mil) = 1.612\ mm$$

$$\epsilon_{eff} = \frac{\epsilon_r + 1}{2} + \frac{\epsilon_r - 1}{2}\frac{1}{\sqrt{1 + 12d/W}} \qquad \text{Equation 2-3}$$

$$\epsilon_{eff} = \frac{2.1 + 1}{2} + \frac{2.1 - 1}{2}\frac{1}{\sqrt{1 + 12(\frac{1}{3.17307})}}$$

$$\epsilon_{eff} = 1.80152$$

[2] *I. J. Bahl and D. K. Trivedi, "A Designer's Gide to Microstrip Line," Microwaves, May 1977, pp. 174-182.*

$$\varphi = \beta l = \omega l \frac{c}{\sqrt{\epsilon_{eff}}} = 145° \qquad \text{Equation 2-4}$$

$$l = 145° \frac{\sqrt{\epsilon_{eff}}}{c\omega} \qquad \text{Equation 2-5}$$

$$l = 145° \frac{\pi}{180°} \frac{\sqrt{1.80152}}{(3x10^8)(2\pi x10^{10})}$$

$$l = 9.003\ mm$$

To design a microstrip line to parameter specifications using the ADS LineCalc Tool calculation: E. Hammerstad and Ø. Jensen Model[3], proceed as follows. The ADS LineCalc Tool creates microstrip line models using the Hammerstad/Jensen model. This information can be found by placing a regular microstrip line component onto the schematic, opening up the Edit Instance Parameters window and clicking on the Help button. Under Notes/Equation, Hammerstad and Jensen are listed as the frequency-dependent model. To use LineCalc, go into the Tools menu, open up the LineCalc Tool and Start LineCalc. With the component type in MLIN for the microstrip line, enter in the design parameters and click the up arrow Synthesize button (Figure 2-5). This will populate the width and length values.

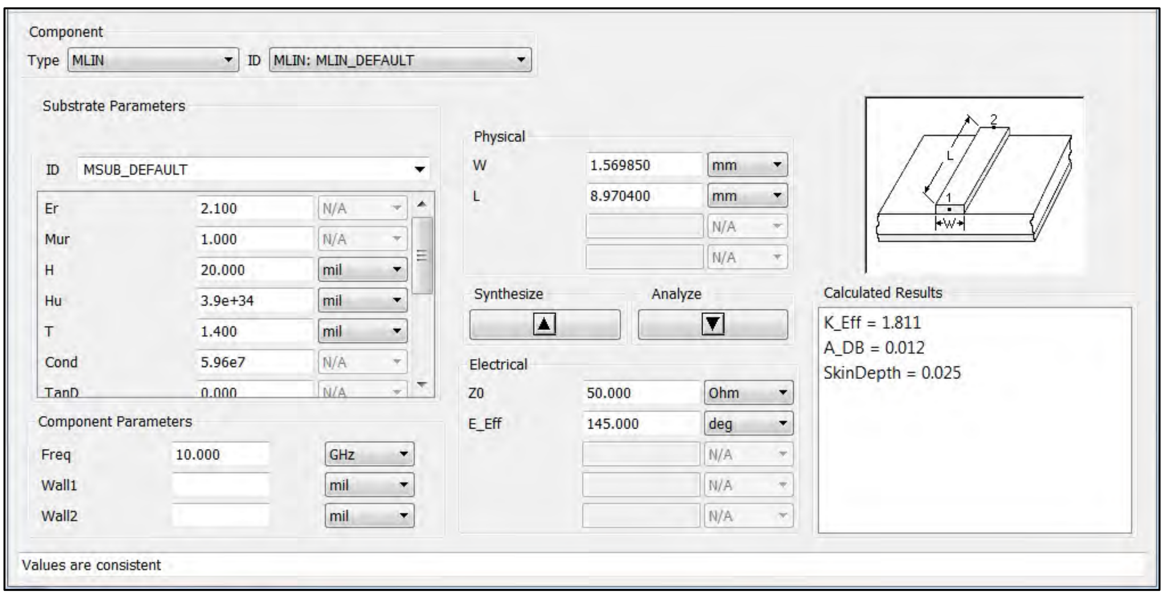


Figure 2-5. Using the LineCalc Tool to design a microstrip line.

a) Create the circuits for both the hand calculation and ADS LineCalc generated microstrip lines.
Use the microstrip line component called ML1CTL_C; the same one used in the previous problem. In addition to the S-parameter simulation component, add the MLSUBSTRATE2 component and define the microstrip line parameters (Figure 2-6).

---

[3] *E. Hammerstad and Ø. Jensen, "Accurate Models for Microstrip Computer-aided Design," MTT Symposium Digest, 1980.*

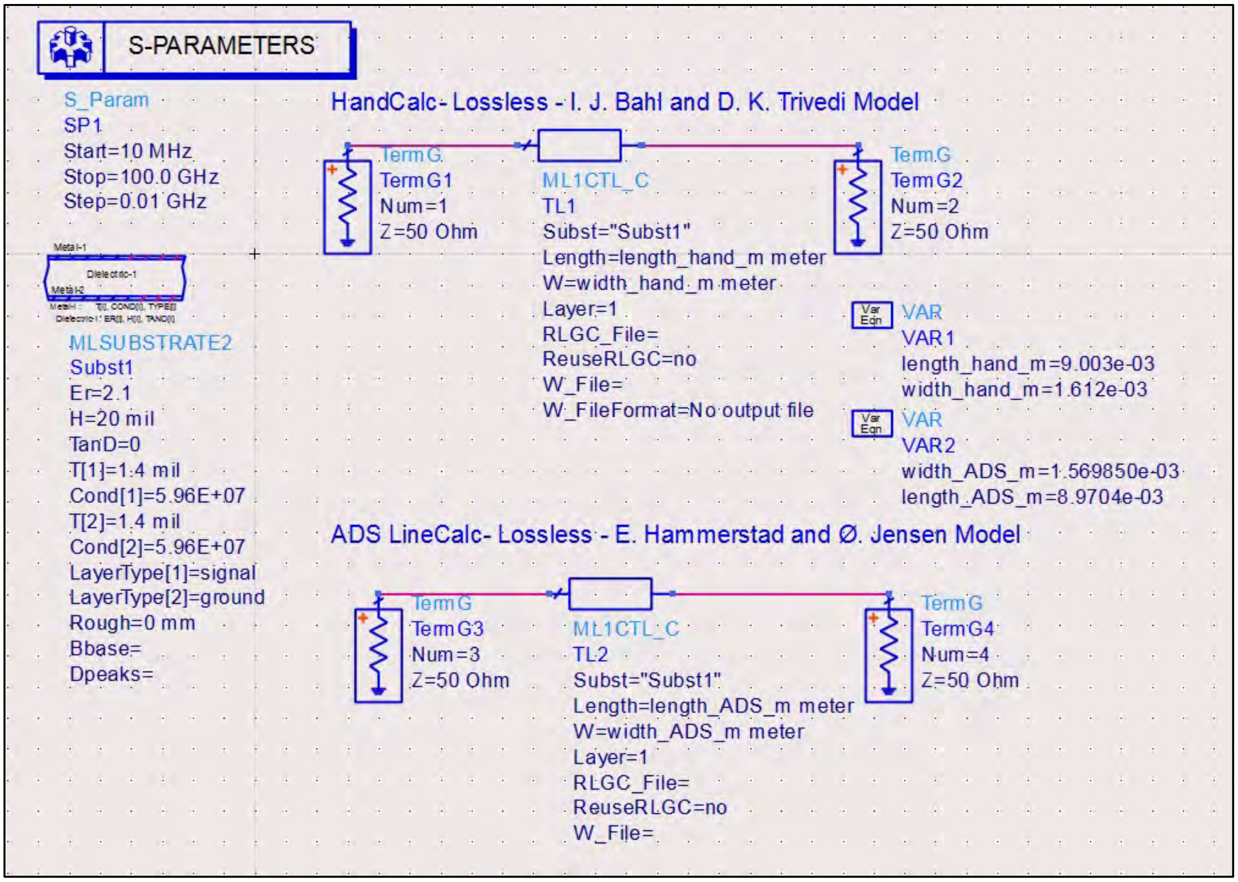


Figure 2-6. Creating the circuits for the hand calculated and ADS LineCalc generated microstrip lines.

A plot of the S_{21} parameter in dB for both circuits provides the insertion loss for both models versus frequency (Figure 2-7).

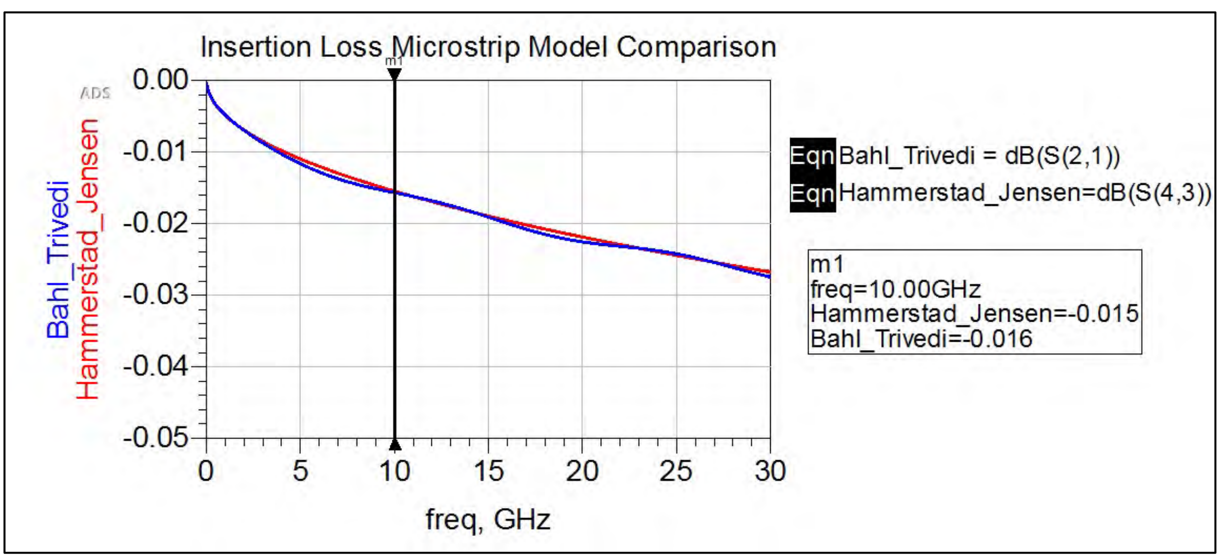


Figure 2-7. A plot of the S_{21} parameter in dB for both circuits.

Both models follow one another very closely, but it is important to realize the number of available models, and determine which one is appropriate for the specific design. These models are created from experimental data, which is why they differ slightly.

a) Moving forward with only the Hammerstad and Jensen model, a duplicate circuit is created with a separate MSUB for the lossy microstrip line by incorporating conductor loss. In the MSUB, Tanδ will equal 0.0002 (Figure 2-8).

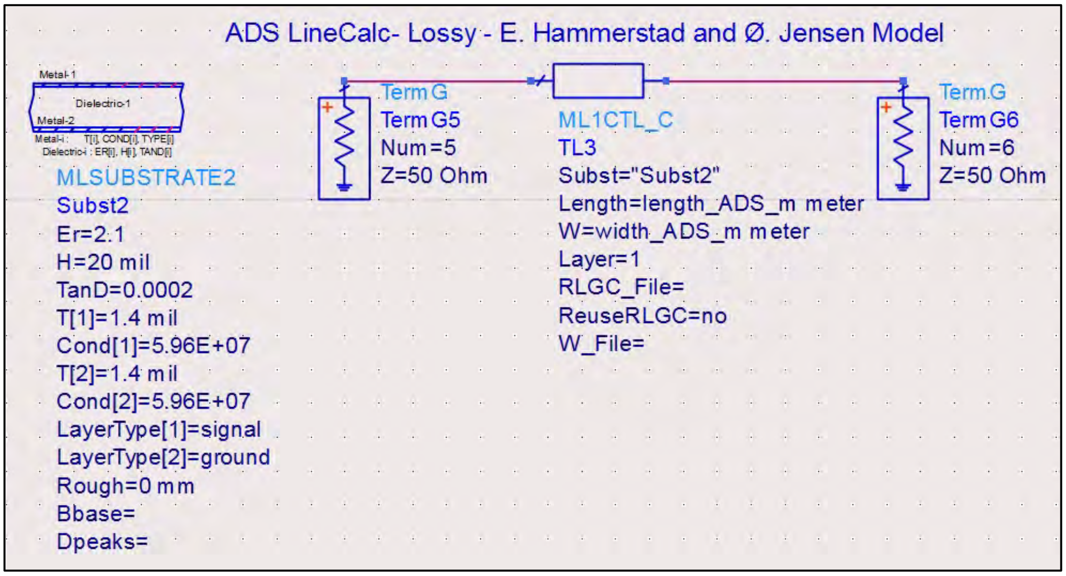


Figure 2-8. Shown here is a duplicate circuit with a separate MSUB for the lossy microstrip line.

Simulate and plot the lossy versus lossless microstrip lines as shown in Figure 2-9.

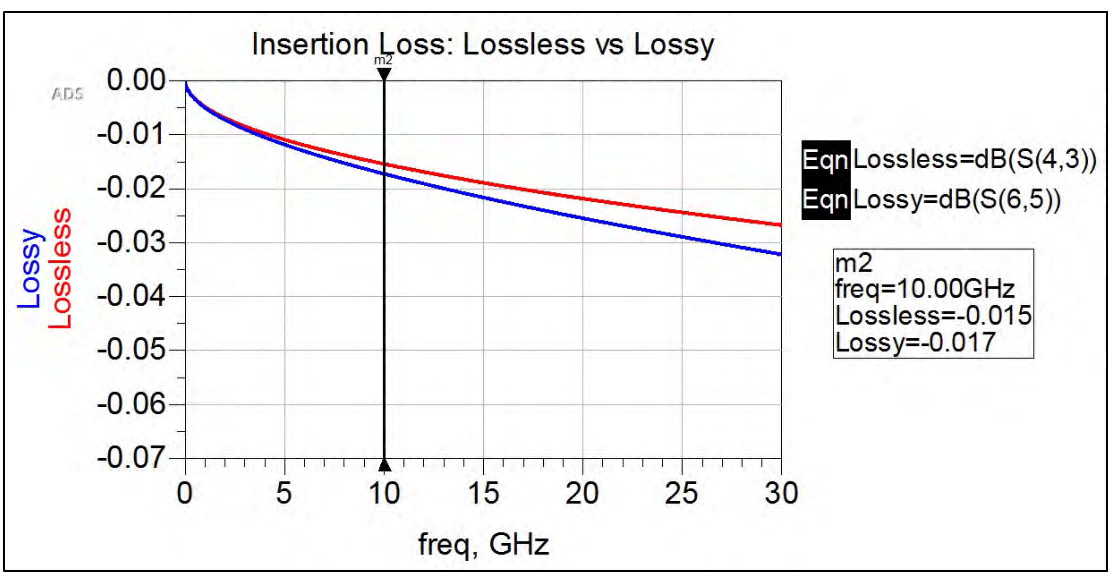


Figure 2-9. A plot the lossy vs. lossless microstrip lines.

b) The ideal transmission line circuit is created within the schematic to compare it to part b.

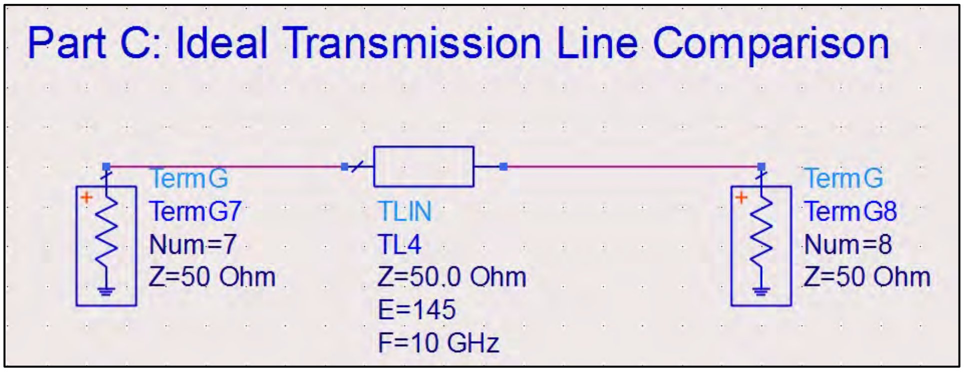


Figure 2-10. The ideal transmission line.

Simulate and plot the insertion loss for the lossy and lossless microstrip lines against the ideal transmission line (Figure 2-11).

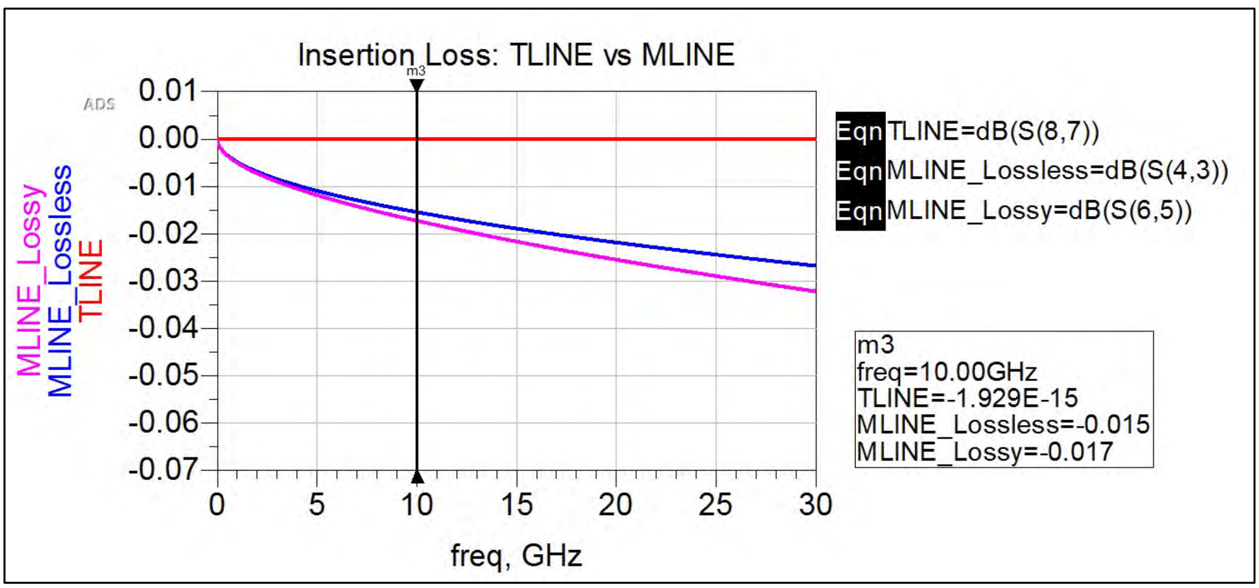


Figure 2-11. A plot of the insertion loss for the lossy and lossless microstrip lines against the ideal transmission line.

The ideal transmission line incurs no loss over all frequencies, as expected.

c) The phase velocity can be calculated by:

$$v_p = \frac{\omega}{\beta} = \frac{\omega l}{\theta} \qquad \text{Equation 2-6}$$

To calculate this, we will extract theta out of the S_{21} phase, just as was done in Problem 1.

The schematic setup for part d is given in Figure 2-12.

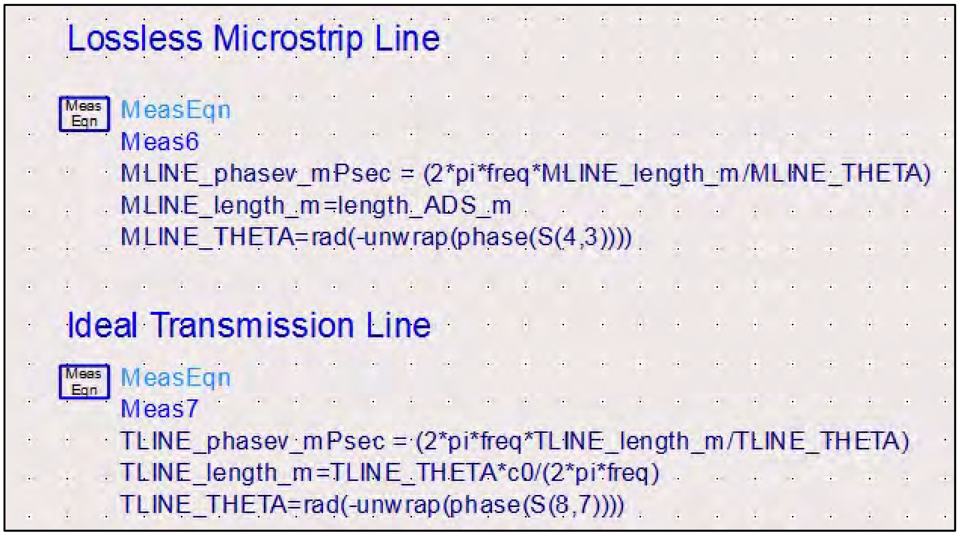


Figure 2-12. The schematic setup for part d of this problem.

The resulting plot of phase velocities is given in Figure 2-13. While very slight, phase velocity for the ideal transmission line does not change with respect to frequency, although it does for the microstrip line.

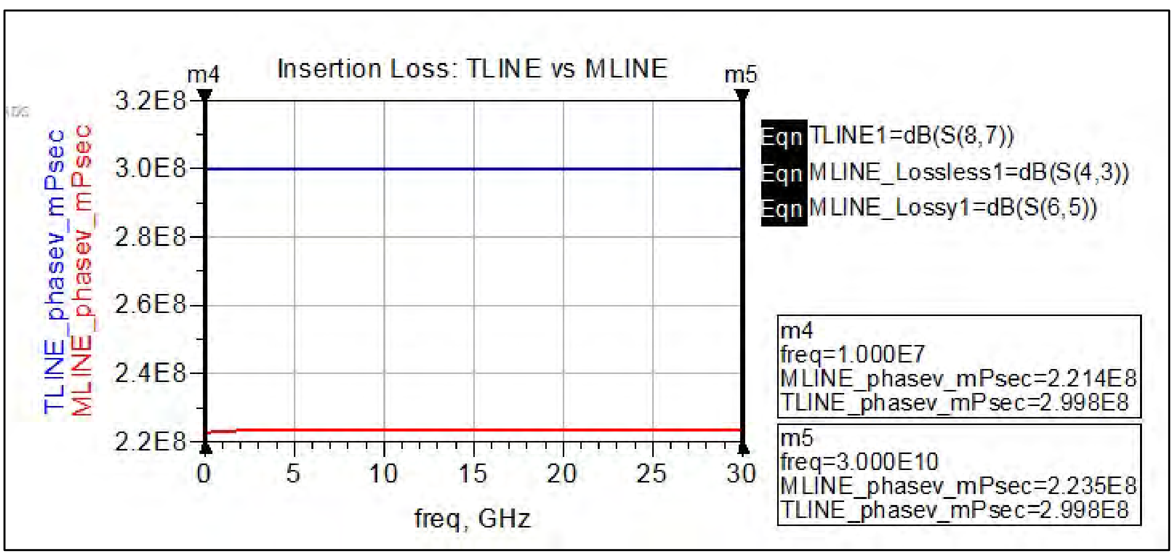


Figure 2-13. The resulting plot of phase velocities.

Conclusion

There are multiple microstrip line models derived from experimental data. This exercise compared the Hammerstad/Jensen model with the Bahl/Trivedi model, and yielded similar insertion loss responses over the 10 MHz–30 GHz frequency band. The microstrip line is lossy compared to the ideal transmission line. It is also very dispersive compared to the ideal transmission line. This means that signal distortion occurs when the waves with faster phase velocities lead the slower waves in phase. This dispersive behavior is caused by the change in effective dielectric constant within the microstrip line with respect to frequency. As the change in the dielectric constant plateaus, so too does the change in phase velocity.

Problem 3: 3-D Solver Momentum Example

Problem Statement

Show the current distribution for a section of 25-Ohm microstrip line over the frequency range 100 MHz to 10 GHz (Figure 2-14). Assume a standard 50-Ohm generator and load impedance, operating frequency of 5 GHz, dielectric constant of 4, substrate height of 20 mil, and copper conductor of thickness 1.4 mil.

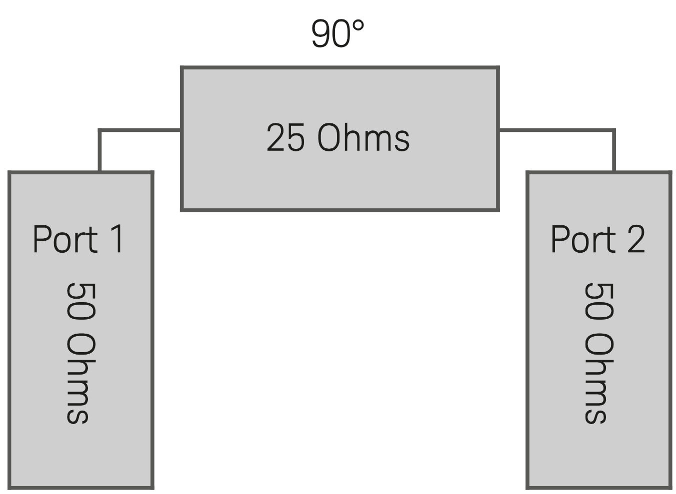


Figure 2-14. Shown here is the block diagram for Problem 3.

Solution

Strategy

Create the schematic in ADS and simulate with the 3-D solver Momentum to show the current distribution.

What to expect

For a section of 25-Ohm microstrip line connected to 50-Ohm terminals, one expects to see constructive and destructive interference during propagation of the signal due to reflections. Because the gamma at Port 1 and Port 2 are both 1/3, the first cycle is expected to experience constructive interference, meaning that the return loss will be at its maximum at a quarter wavelength. Therefore, the frequency at which the maximum return loss occurs can be easily estimated.

$$length = \frac{\lambda}{4} \qquad \text{Equation 2-7}$$

$$\lambda = 4length = \frac{v}{f} = \frac{c}{f\sqrt{\epsilon_r}}$$

$$f = \frac{c}{4length\sqrt{\epsilon_r}} = \frac{3\,x\,10^8\,(\frac{m}{s})}{4(length\,(m))(2)} = \frac{0.375\,x\,10^8(\frac{m}{s})}{length(m)} \qquad \text{Equation 2-8}$$

Execution

Using LineCalc, the schematic in Figure 2-15 was produced for the section of a 25-Ohm microstrip line.

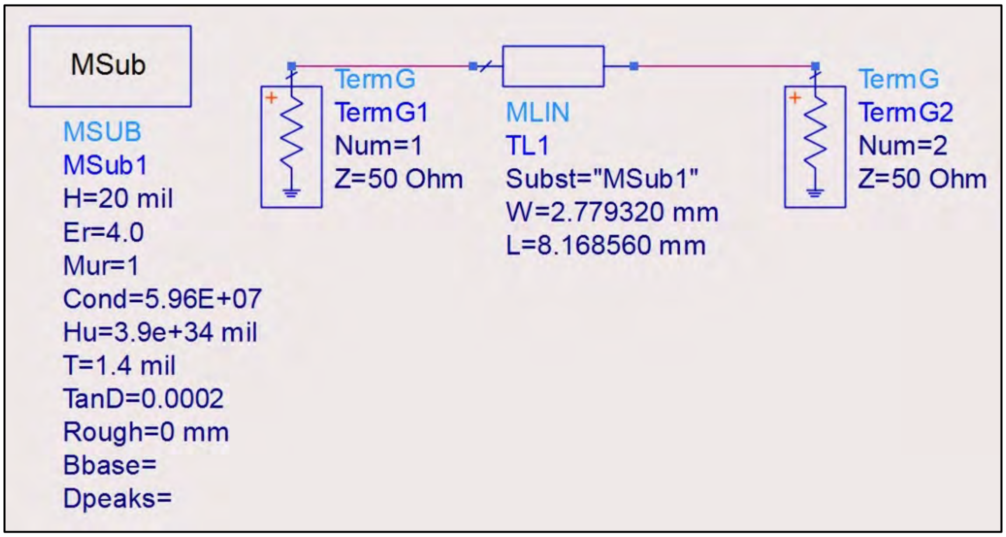


Figure 2-15. The schematic for the section of a 25-Ohm microstrip line.

In the top menu, go to Layout -> Generate/Update Layout. Figure 2-16 will appear.

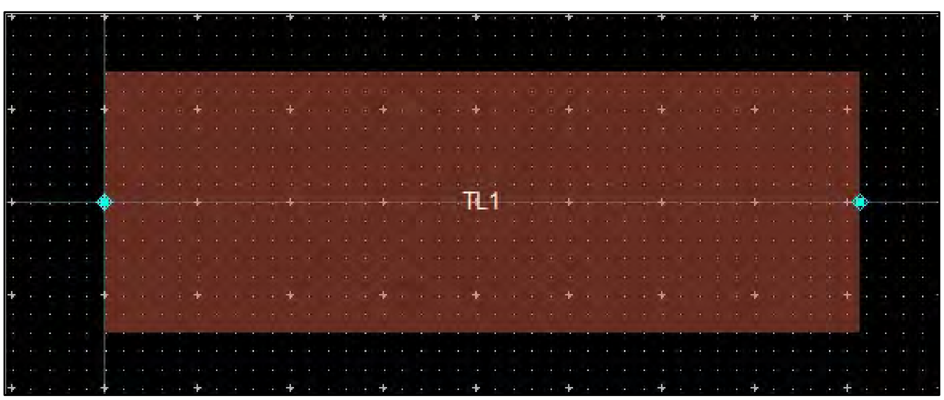


Figure 2-16. This figure appears when the layout is updated.

To add the ports, click on the Port/Pin button  and drag it to the middle left edge of TL1. Add Port 2 to the middle right edge of TL1. Save your design.

The substrate and conductor components need to be defined. Click on the substrate editor and click on the dielectric substrate on the microstrip image (Figure 2-17).

a) Change the thickness to 20 mil.

b) Change the substrate layer name to FR4. This is done by selecting the "..." button on the same row that says Material. Select the Dielectrics tab and then select Add Dielectric. Change the default material name to FR4.

c) Change the real part of the permittivity to 4.0 and the loss tangent to 0.0002.

d) Click on Apply, then OK.

e) Select FR4 in the Material pull down menu and then right click on any empty space on the microstrip image in the substrate editor.

f) Select the conductor layer on the microstrip line image ("cond") and change the material to copper by clicking on the "..." button next to the Material drop down list. On the Conductors tab, click Add from database and choose the copper conductor.

g) Click on Apply, then OK. Make sure both conducting planes are 1.4 mil thickness.

h) Right click on the substrate layer in the drawing and select Map Conductor Via.

i) Select the pcvia 1 Layer from the pull-down list and the material as Copper.

j) Click on the Check for Errors button. If errors are found, check the substrate and conductor values to ensure all steps were followed properly. If no errors are found, save your design.

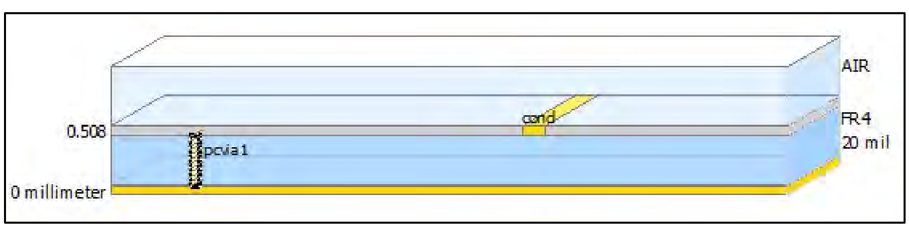


Figure 2-17. Defining the substrate and conductor components.

The next few steps take place in the EM Simulation Setup window.

a) Click on the Simulation Setup button

b) Click on the Ports tab and confirm that the ports have the following values:
Ref impedance: 50 Ohms; Calibration: TML

c) Click on the Substrate tab and select the substrate definition saved earlier.

Solve the Substrate

a) Click on the Frequency Plan tab. Enter a minimum frequency of 0.1 GHz and a maximum frequency of 10 GHz. Keep the sweep type as Adaptive.

b) Click on the Generate pull-down list at the bottom of the EM Simulation setup window and select Substrate. Press Go. Save your design.

Solve the Mesh

a) In the same window, click the Options tab and select Mesh -> Global.

b) Set the mesh frequency to the highest simulated frequency, 10 GHz.

c) The Number of Cells per Wavelength (CpW) indicates the number of pieces into which the layout is fragmented as a function of electrical size; the more cells per wavelength, the denser the resulting mesh. Typically, 20-30 cells per wavelength are required in order to obtain accurate results. For this example, specify 25 CpW.

d) Select the Edge Mesh option and specify the edge width as 0 mm. By using Edge Mesh, Momentum will create special cells along the edges of the conductors in order to calculate the current distribution more accurately. Note that on a microstrip line, the current is concentrated at the edges, and having cells along the edge improves the simulation results.

e) Click on the Generate pull down at the bottom of this window and select Mesh, then select Go. Save your design.

Your layout window should now look like Figure 2-18.

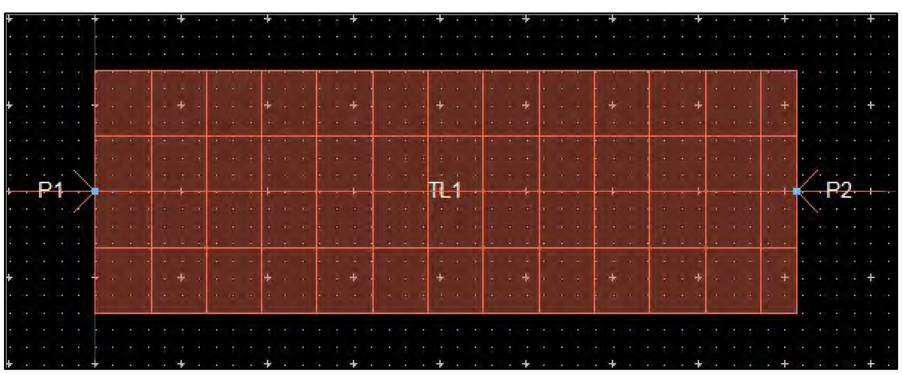


Figure 2-18. The resulting layout window when solving the mesh.

Analyze the Circuit

a) In the Simulation Setup window, click on the Frequency Plan tab. Change the number of sample points to 40.

b) Click on the Generate pull-down list. Select S-parameters and Simulate.

c) A Data Display window will automatically appear with a plot of each of the S-parameters.

d) The name of the data set that is created will be the same as the layout filename, with a "_a" extension. This file will automatically be saved to the data subdirectory of your project. If desired, it can be referenced from an S*P data element (where * indicates the number of ports) in an ADS schematic by selecting the file type as dataset.

e) Figure 2-19 was created for return loss of the stepped impedance microstrip line using Momentum generated S-parameters.

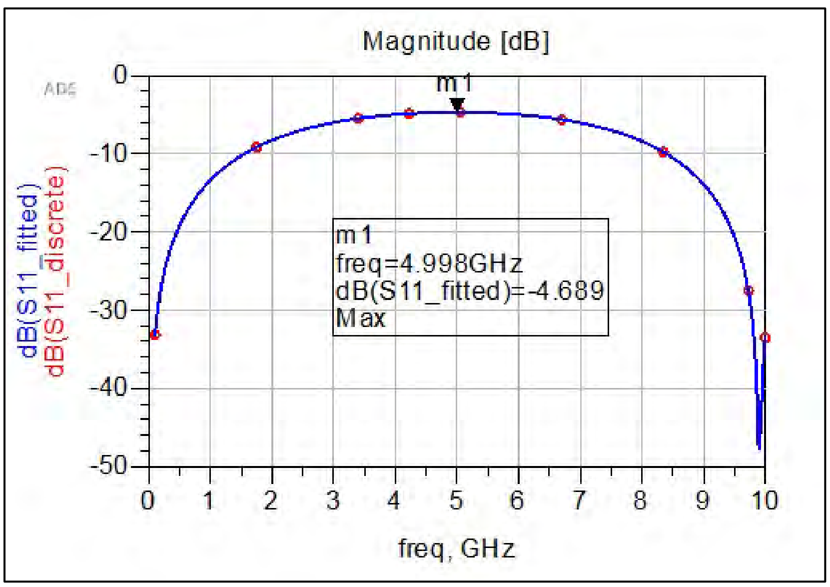


Figure 2-19. A plot created for the return loss of the stepped impedance microstrip line using Momentum generated S-parameters.

With a microstrip line length of 8.13693 mm, the expected frequency to experience the maximum return loss (S_{11}) is:

$$f = \frac{0.375\ x\ 10^8}{length} = \frac{0.375\ x\ 10^8}{8.168560\ x\ 10^{-3}} = 4.591\ GHz \qquad \text{Equation 2-9}$$

This estimation is very close to the 3D solved frequency of 4.981 GHz; about an 8.5% difference.

Generating Plots of Current Distribution

a) Click EM -> Post Processing -> Visualization in the main layout window.

b) The Momentum Visualization window will automatically pop up and generate a multicolored circuit, specifying the current density in each area.

c) Click on the Solution Setup tab and specify a Single Port setup. Click through the frequencies and see the current distribution change, along with areas of concentration.

Conclusion

Figure 2-20 shows the current distribution around 5 GHz. As expected, constructive interference happens and a maximum return loss is seen at Port 1. An interesting option is to view the Plot Properties tab and click Animate. The signal propagation will play like a video, and the peaks and valleys of the propagating signal can be viewed as red and blue, respectively.

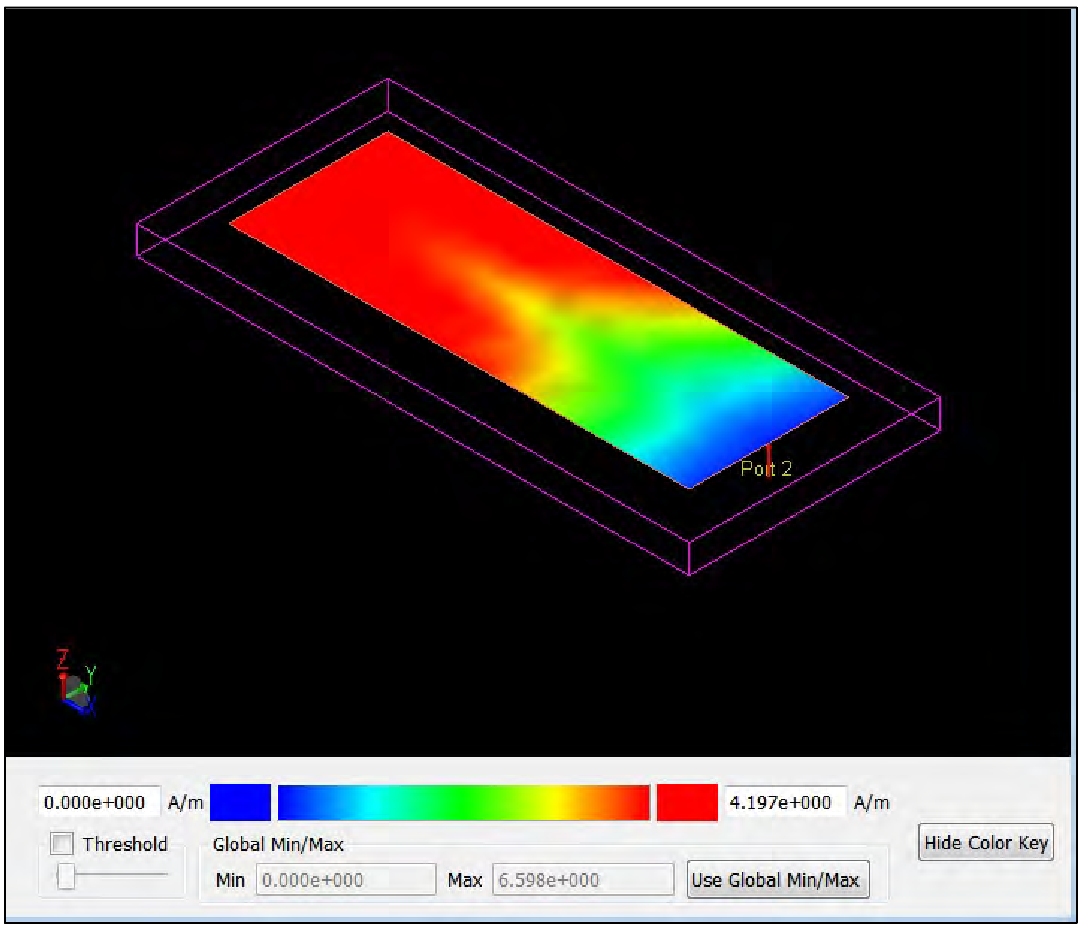


Figure 2-20. The current distribution around 5 GHz with the peaks and valleys of the propagating signal viewed as red and blue, respectively.

Chapter 3 – Network Analysis

This chapter on microwave network analysis develops the parameters necessary to interconnect components and analyze the characteristic behavior of the system overall. In a low-frequency circuit design course, one might become familiar with Z-, Y- and H-parameters to characterize the impedance, admittance and amplifier gain characteristics, respectively. Open and short circuits are utilized to determine these parameters in terms of voltage and current in an n-port network. For higher frequencies (above 100 MHz); however, it is difficult to achieve an ideal open circuit and it becomes more useful to look at normalized power waves[4] using matched loads. This characterization is called the scattering parameter matrix, or [S] matrix.

For an RF/microwave engineering course, the focus is mainly on S-parameters. This chapter will review the Z- and Y-parameters for characterization of a two-port network and lead into S-parameters. Most of the analysis in the following chapters will use S-parameters, and therefore any ADS simulation will be done using S-parameter simulation. S-parameters were briefly introduced in earlier chapters to develop familiarity, but it will now become the dominant simulation type. It is possible; however, to use AC simulation and characterize the network using voltage and current probes at lower frequencies, if desired. All three types of parameters may be easily converted to one another, but a more useful conversion is into the transmission [ABCD] matrix. Using Z-, Y- and S-parameters, any n-port microwave network can be characterized. It is convenient to break a system into many two-port networks, characterize them using the appropriate parameter, and then cascade their connections using the [ABCD] matrix. This conversion process can become cumbersome depending on the number of networks. When using ADS, the n-port networks can be cascaded in their original parameters and ADS will process the conversion in the background.

Problem 1: Two-Port Network Z-Parameters

Problem Statement

a) Find the Z-parameters and Z_{in} at Port 1 for the circuit in Figure 3-1, by hand, for the frequency 2 GHz.

---

[4] *K. Kurokawa, "Power Waves and the Scattering Matrix," IEEE Transactions on Microwave Theory and Techniques, vol. MTT-13, pp. 194-202, March 1965.*

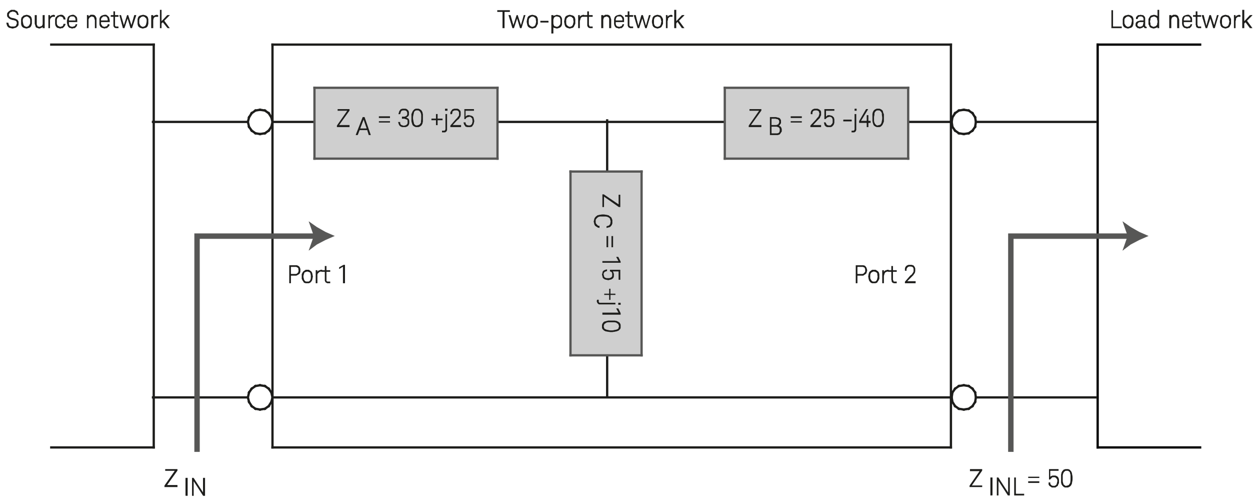


Figure 3-1. The circuit for the Problem 1.

b) Solve for the Z-parameters and Z_{in} in ADS for the frequency range 1 to 10 GHz.

c) Determine if the network is reciprocal, lossless, or both?

Solution

Strategy

Compute the [Z] matrix and Z_{in} for the two-port network by hand. Convert the circuit reactance to the appropriate inductors and capacitors. Use S-parameter simulation in ADS and use the ADS built-in equation stoz() to calculate the [Z] matrix since ADS is quite skilled at S-parameter calculation. Use the ADS built-in Z_{in} function to determine the Z_{in} of the circuit.

What to expect

The hand and ADS calculation of the Z-parameters for 2 GHz are expected to agree. If a load is taken into account for the hand analysis for the Z_{in} calculation, they will also agree. If there is no load attached, Z_{in} is expected to be just Z_{11}. This is because in order to calculate the Z-parameters from the S-parameter simulation, a load termination must be placed onto the schematic and factored into the Z_{in} calculation. It is also expected that the input impedance will increase with frequency due to the reactive components of the circuit. Eventually, the input impedance will look like an open circuit. The two-port network is expected to be a reciprocal network because it is purely passive and made up of lumped elements. The resistive part of the network; however, prevents it from being lossless.

Execution

a) The impedance matrix [Z] for the two-port network, as shown in Figure 3-2, is defined a

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad \text{Equation 3-1}$$

Providing the following relationships:

$$V_1 = Z_{11}I_1 + Z_{12}I_2 \qquad \text{Equation 3-2}$$
$$V_2 = Z_{21}I_1 + Z_{22}I_2 \qquad \text{Equation 3-3}$$

Find Z_{11}:

$$Z_{11} = \left.\frac{V_1}{I_1}\right|_{I_2=0} \qquad \text{Equation 3-4}$$

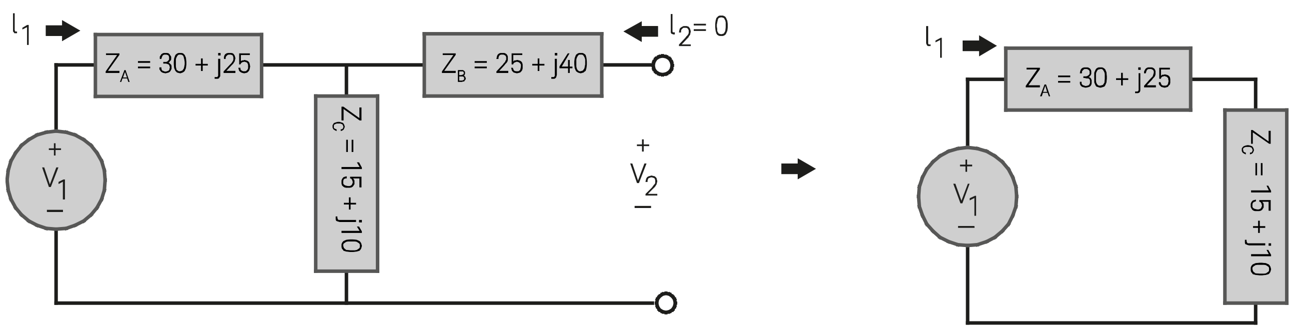


Figure 3-2. Finding the impedance matrix [Z] for the two-port network.

Performing KVL around the loop yields:

$$V_1 = I_1Z_A + I_1Z_C$$
$$\frac{V_1}{I_1} = Z_A + Z_C$$
$$Z_{11} = Z_A + Z_C = 45 + j35$$

Find Z_{12} (Figure 3-3):

$$Z_{12} = \left.\frac{V_1}{I_2}\right|_{I_1=0} \qquad \text{Equation 3-5}$$

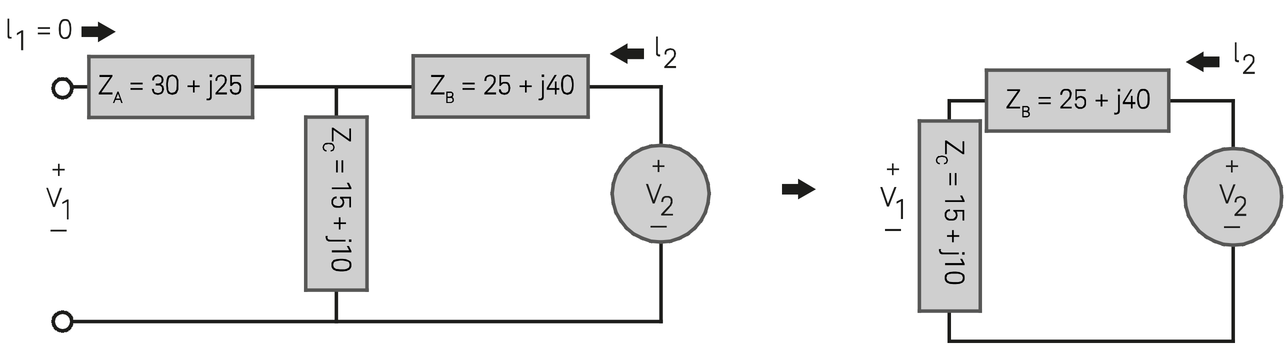


Figure 3-3. Finding Z_{12}.

Performing KVL around the loop yields:

$$V_2 = I_2Z_B + I_2Z_C \qquad \text{Equation 3-6}$$

A simple voltage divider provides the relationship between V_1 and V_2:

$$V_1 = \frac{Z_C}{Z_B+Z_C}V_2 \qquad \text{Equation 3-7}$$

Combining the two equations and solving for Z_{12}

$$V_1 = \frac{Z_C}{Z_B+Z_C}(I_2Z_B + I_2Z_C) \qquad \text{Equation 3-8}$$

$$V_1 = I_2\frac{Z_C(Z_B + Z_C)}{Z_B + Z_C}$$

$$Z_{12} = \frac{V_1}{I_2} = Z_C = 15 + j10$$

Following similar procedures for Z_{21} and Z_{22} provide the following [Z] matrix for the two-port network (Figure 3-4):

$$[Z] = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \begin{bmatrix} 45+j35 & 15+j10 \\ 15+j10 & 40-j30 \end{bmatrix} \qquad \text{Equation 3-9}$$

Find Z_{in}:

$$Z_{IN} = \frac{V_{IN}}{I_{IN}} = \frac{V_1}{I_1} \qquad \text{Equation 3-10}$$

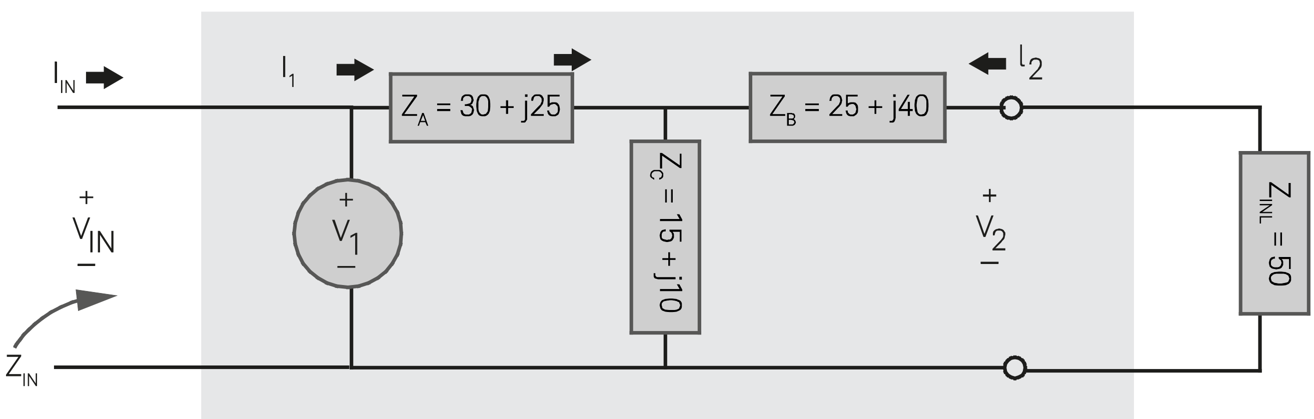


Figure 3-4. Finding Z_{in} for a two-port network.

From the Z-parameter definition, V_1 can be rewritten in terms of I_1 and I_2.

$$Z_{IN} = \frac{V_1}{I_1} = \frac{Z_{11}I_1 + Z_{12}I_2}{I_1} \qquad \text{Equation 3-11}$$

$$Z_{IN} = Z_{11} + Z_{12}\frac{I_2}{I_1}$$

From the diagram, V_2 can be defined as:

$$V_2 = -I_2Z_{INL} \qquad \text{Equation 3-12}$$

From the Z-parameter definition, V_2 is defined as:

$$V_2 = Z_{21}I_1 + Z_{22}I_2 \qquad \text{Equation 3-13}$$

Equating and solving for I_2/I_1, yields:

$$\frac{I_2}{I_1} = \frac{-Z_{21}}{Z_{22} + Z_{INL}}$$

Plugging this value into Z_{in} provides the final answer:

$$Z_{IN} = Z_{11} - \frac{Z_{12}Z_{21}}{Z_{22} + Z_{INL}}$$

$$Z_{IN} = 44.75 + j31.583$$

b) The impedance blocks have a reactive component and need to be converted into their lumped-element components for analysis over a frequency band. The converted circuit in ADS is given in Figure 3-5.

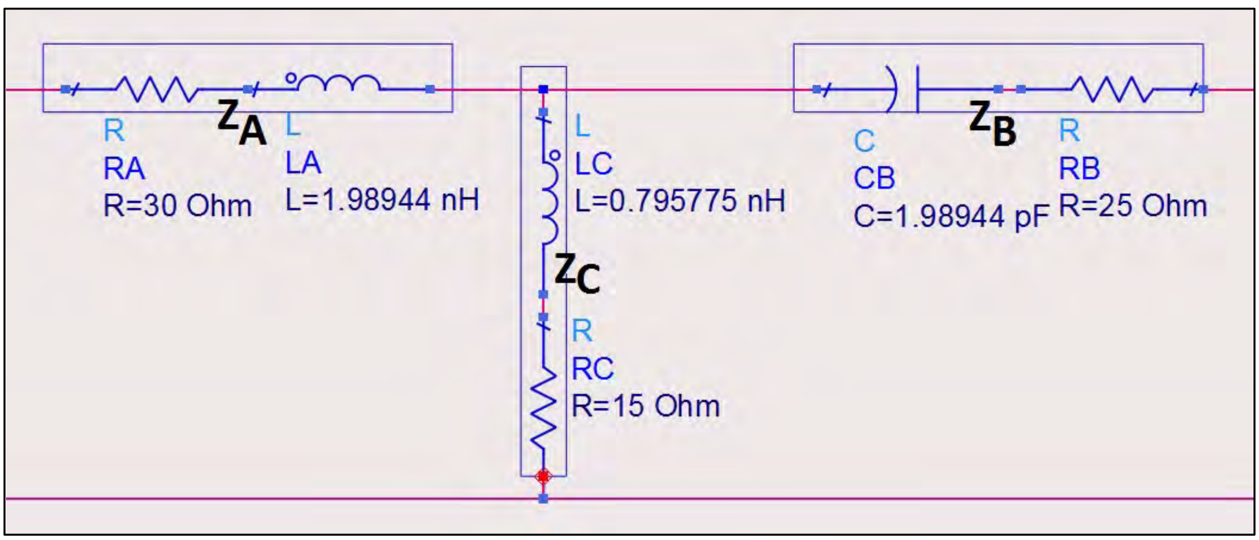


Figure 3-5. The converted circuit in ADS.

Add the S-parameter simulation component, two 50-Ohm terminations and the Z_{in} component from the Simulation-S_Param component palette. Simulate the schematic and open the Data Display page.

ADS calculates S-parameters, and is set up for mostly S-parameter analysis. It also has built-in functions to convert S-parameters to other parameters like Z and Y. In the Data Display page, create the following equation:

Z_param = stoz(S, 50) Equation 3-14

Where S references the scattering matrix of the two-port network and 50 is the reference impedance, or the termination values. This is why it is important to keep both terminations the same value in ADS for S-parameter simulation. The scattering matrix is calculated assuming equal reference impedances.

The equation provides the Z-parameter values In Table 3-1. Note that the 2-GHz values are equal to the hand-calculated values, verifying the accuracy of ADS. It is also important to note that the real part of the Z-parameters never changes with frequency, as expected, but the imaginary part does increase with frequency.

Z Parameters of the Two-Port Network

Eqn Z_param = stoz(S, 50)

freq	Z_param			
	Z_param(1,1)	Z_param(1,2)	Z_param(2,1)	Z_param(2,2)
1.000 GHz	45.000 + j17.500	15.000 + j5.000	15.000 + j5.000	40.000 - j75.000
2.000 GHz	45.000 + j35.000	15.000 + j10.000	15.000 + j10.000	40.000 - j30.000
3.000 GHz	45.000 + j52.500	15.000 + j15.000	15.000 + j15.000	40.000 - j11.667
4.000 GHz	45.000 + j70.000	15.000 + j20.000	15.000 + j20.000	40.000 + j3.944...
5.000 GHz	45.000 + j87.500	15.000 + j25.000	15.000 + j25.000	40.000 + j9.000
6.000 GHz	45.000 + j105.000	15.000 + j30.000	15.000 + j30.000	40.000 + j16.667
7.000 GHz	45.000 + j122.500	15.000 + j35.000	15.000 + j35.000	40.000 + j23.571
8.000 GHz	45.000 + j140.000	15.000 + j40.000	15.000 + j40.000	40.000 + j30.000
9.000 GHz	45.000 + j157.500	15.000 + j45.000	15.000 + j45.000	40.000 + j36.111
10.00 GHz	45.000 + j175.000	15.000 + j50.000	15.000 + j50.000	40.000 + j42.000

Table 3-1. A table of Z-parameter values assuming the use of Equation 3-12.

The Z_{in} calculation with respect to frequency is also tabulated and verified (Table 3-2).

Zin of the Two-Port Network

freq	Zin1
1.000 GHz	44.508 + j15.424
2.000 GHz	44.750 + j31.583
3.000 GHz	45.637 + j47.583
4.000 GHz	46.944 + j63.333
5.000 GHz	48.575 + j78.809
6.000 GHz	50.461 + j93.989
7.000 GHz	52.538 + j108.859
8.000 GHz	54.750 + j123.417
9.000 GHz	57.043 + j137.668

Table 3-2. A table of Z_{in} values for the two-port network.

c) The network is reciprocal because it does not contain any active devices, and is verified by the [Z] matrix over the frequency band. $Z_{12} = Z_{21}$ for all frequencies. The network is not lossless though since the Re $\{Z_{12}, Z_{21}\} \neq 0$.

Conclusion

The manual method works well for one frequency point and simple two-port networks. If the network is more complex and analysis is needed over a frequency band, it is easier and faster to use ADS. In addition, a passive network composed of lumped elements will be a reciprocal network, but not necessarily a lossless network.

Problem 2: Y-Parameters

Problem Statement

a) The lossless two-port network in Figure 3-5 has a Y_{11} parameter of 0+j0.20. Solve for the missing component at frequency 2 GHz.

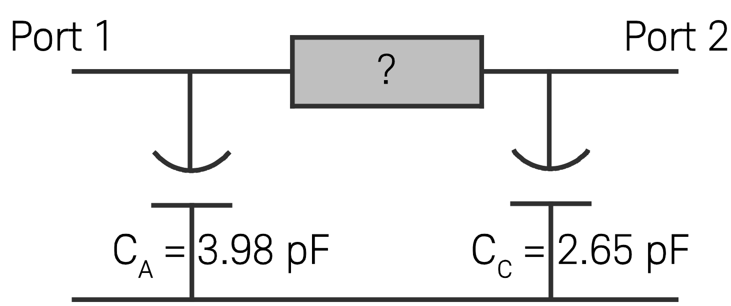


Figure 3-5. The schematic for the problem stated above.

b) Verify your answer in part a) using ADS.

Solution

Strategy

Solve for the Y-parameters in terms of Y_A, Y_B, and Y_C first, and then use their relationship to determine the missing component given the provided Y_{11} parameter. Once Y_B is determined, solve for the missing component at 2 GHz.

What to expect

From the Y-parameter definition, $I_1 = Y_{11}V_1 + Y_{12}V_2$, we expect Y_{11} to be a combination of Y_A and Y_B, similar to the Z_{11} parameter found in Problem 1. Additionally, the network is lossless, which requires all elements of the admittance matrix to be purely imaginary, and the net power delivered to the network must be zero. Therefore, the missing component must be either a capacitor or inductor. After hand analysis, it is expected that ADS will validate the found [Y] matrix.

Execution

a) The admittance matrix [Y] for the two-port network is defined as:

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \quad \text{Equation 3-15}$$

Providing the following relationships:

$$I_1 = Y_{11}V_1 + Y_{12}V_2 \qquad \text{Equation 3-16}$$

$$I_2 = Y_{21}V_1 + Y_{22}V_2 \qquad \text{Equation 3-17}$$

Find Y_{11} (Figure 3-6):

$$Y_{11} = \left.\frac{I_1}{V_1}\right|_{V_2=0} \qquad \text{Equation 3-18}$$

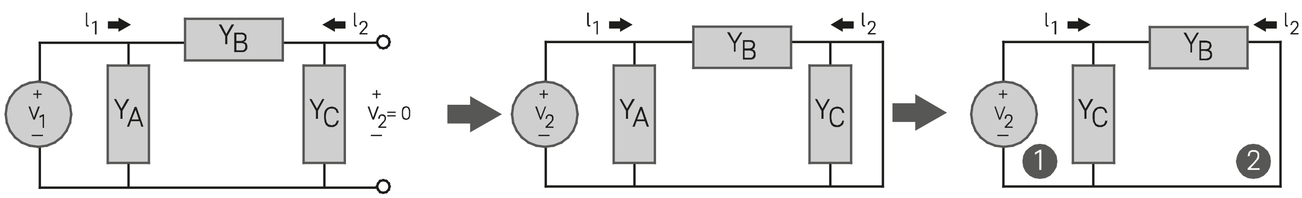


Figure 3-6. The schematic view of how to find Y_{11}.

Performing KVL around loop 1 yields:

$$V_1 = \frac{(I_1+I_2)}{Y_A} \qquad \text{Equation 3-19}$$

Performing KVL around loop 2 yields:

$$0 = \frac{I_2}{Y_B} + \frac{(I_2 + I_1)}{Y_A}$$

$$I_2 = \frac{-I_1Y_B}{Y_A + Y_B}$$

Substituting loop 2 equation into the loop 1 equation:

$$V_1 = \frac{I_1}{Y_A} - \frac{I_1Y_B}{Y_A(Y_A + Y_B)}$$

$$V_1 = \frac{I_1}{Y_A}\left(1 - \frac{Y_B}{Y_A + Y_B}\right)$$

Find Y_{11}:

$$Y_{11} = \frac{I_1}{V_1} = \frac{Y_A}{\left(1 - \frac{Y_B}{Y_A + Y_B}\right)} = Y_A + Y_B$$

Find Y_{12} (Figure 3-7):

$$Y_{12} = \left.\frac{I_1}{V_2}\right|_{V_1=0}$$

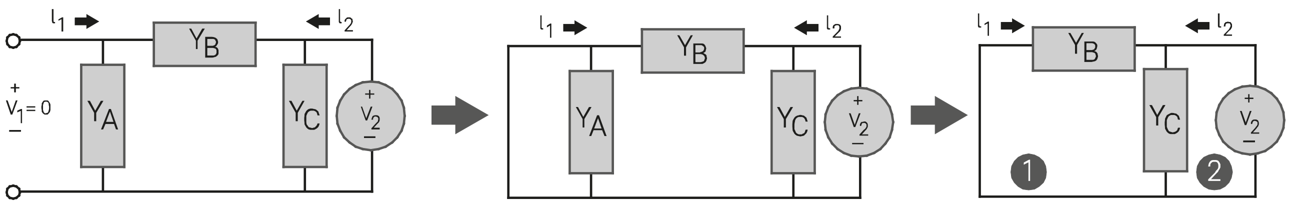


Figure 3-7. The schematic view of how to find Y_{12}.

Performing KVL around loop 1 yields:

$$0 = \frac{I_I}{Y_B} + \frac{(I_2 + I_1)}{Y_C}$$
$$I_2 = \frac{-I_1(Y_B + Y_C)}{Y_B}$$

Performing KVL around loop 2 yields:

$$V_2 = \frac{(I_1 + I_2)}{Y_C} \quad \text{Equation 3-20}$$

Substituting loop 1 equation into the loop 2 equation:

$$V_2 = \frac{I_1}{Y_C} - \frac{I_1(Y_B + Y_C)}{Y_C Y_B}$$
$$V_2 = \frac{I_1}{Y_C}\left(1 - \frac{Y_B + Y_C}{Y_B}\right)$$

Find Y_{12}:

$$Y_{12} = \frac{I_1}{V_2} = -Y_B$$

Following similar procedures for Y_{21} and Y_{22} provides the following [Y] matrix for the two-port network:

$$[Y] = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \begin{bmatrix} Y_A + Y_B & -Y_B \\ -Y_B & Y_B + Y_C \end{bmatrix} \quad \text{Equation 3-21}$$

Solve for the Y-admittance values:

$$Y_A = j\omega C_A = j2\pi(2x10^9)(3.98x10^{-12}) = +j0.05$$
$$Y_{11} = Y_A + Y_B = +j0.20 \rightarrow Y_B = +j0.15$$
$$Y_C = j\omega C_C = j2\pi(2x10^9)(2.65x10^{-12}) = +j0.033$$

Solve for the missing component:

$$Y_B = +j0.15 = j\omega C_B$$
$$C_B = \frac{+j0.15}{2\pi(2x10^9)} = 11.94\,pF$$

b) The completed [Y] matrix is

$$[Y] = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \begin{bmatrix} +j0.20 & -j0.15 \\ -j0.15 & +j0.183 \end{bmatrix}$$

To verify this in ADS, the lumped-element two-port network was simulated, as shown in Figure 3-8.

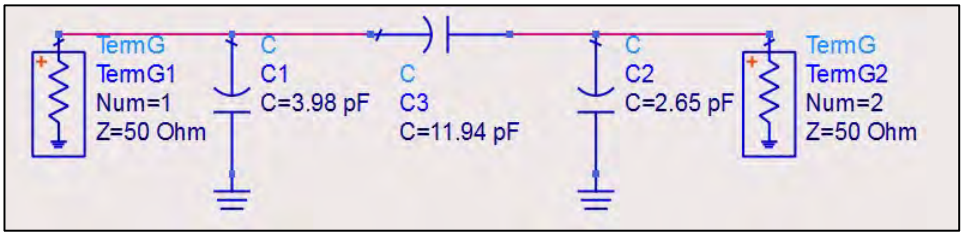


Figure 3-8. A simulation of the lumped-element two-port network.

Similar to Problem 1, the equation stoy(S, 50) can be used. An alternate method of extracting Y-parameters is to double click on the S-parameter simulation component and change the output parameter from S to Y (Figure 3-9).

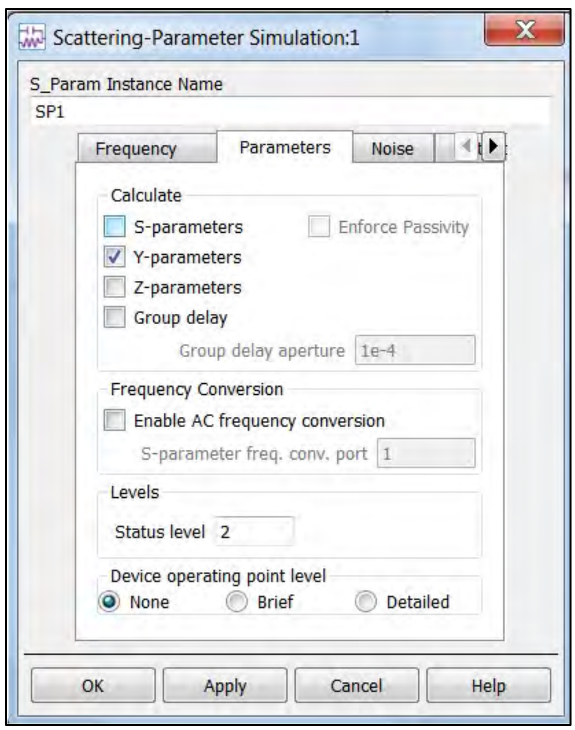


Figure 3-9. Changing the output parameter from S to Y.

The resulting Y-parameters are shown in Table 3-3, and agree with the hand analysis [Y] matrix. Rounding errors for the capacitors give a small real part, but the value is negligible and can be considered zero.

Y- Parameters, 2GHz

freq	Y			
	Y(1,1)	Y(1,2)	Y(2,1)	Y(2,2)
2.000 GHz	-8.739E-18 + j0.200	1.748E-17 - j0.150	-2.544E-18 - j0.150	-5.089E-18 + j0.183

Table 3-3. Using the S-parameter simulation component to extract Y-parameters.

Conclusion

Once again, ADS verifies the hand analysis calculation. To use ADS to find the missing component, the tuning tool could be used, but it is best to do this by hand. However, ADS is useful for checking the answer, and finding the Y-parameters of a more complex network where all the component values are known.

Problem 3: S-Parameters

Problem Statement

a) Find the [S] matrix of the general two-port network in Figure 3-10.

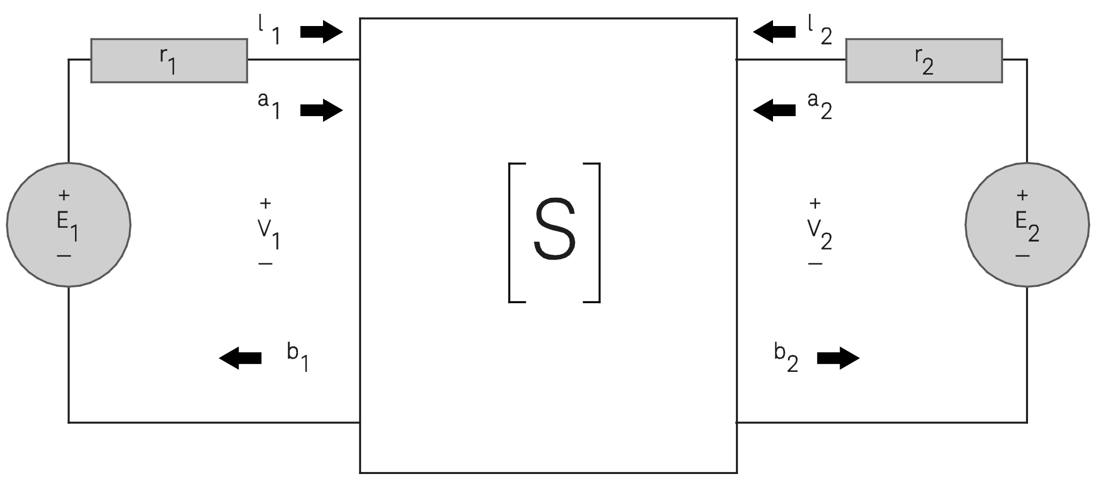


Figure 3-10. Finding the [S] matrix of a general two-port network, such as shown here.

b) Determine the expression for the reflection coefficient at Port 1 in terms of S-parameters.

c) Derive the [S] matrix expression for a lossless two-port network.

d) Use ADS to determine the S-parameters for the network in Figure 3-11 over 1 to 3 GHz using 0.5-GHz steps. Plot the insertion and return loss. What is this network? Is it lossless?

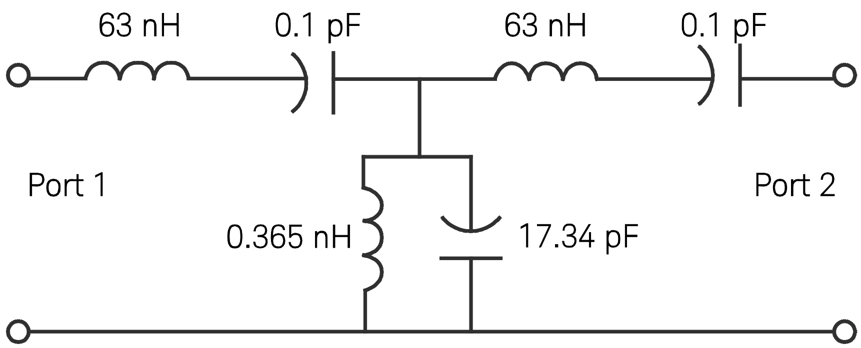


Figure 3-11. ADS can be used to find the S-parameters in this network.

Solution

Strategy

Derive the expressions for the S-parameters in terms of incident and reflected waves, similar to the Z- and Y-parameters. Utilize the expression $P_{avg} = Re\{V_i I_i^*\}$ and the fact that a lossless network has $P_{avg} = 0.$ For part d, the circuit will be simulated in ADS and plotted to review its S-parameters.

What to expect

It is expected that the S-parameters will yield ratios of incident and reflected waves. Therefore, the S-parameter focusing on one port, such as S_{11}, will describe the reflection coefficient at that port.

The provided circuit in part d has a similar structure to an LC filter, and contains resonating segments. It is expected that the circuit will be either a bandpass or band-stop filter. Because filters prevent transmission at certain frequencies, it is expected that the circuit will be a lossy network.

Execution

Determine characteristic relationships for S-parameters. The S-parameters are found using the relationship of traveling waves in the network shown in Figure 3-12.

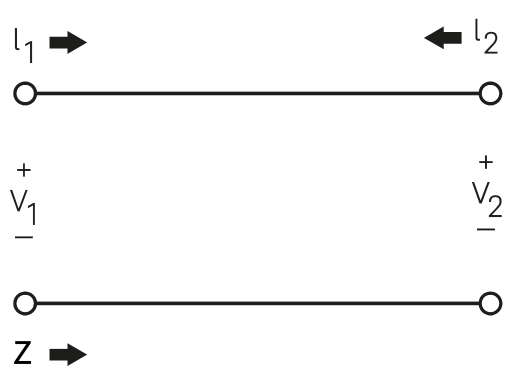


Figure 3-12. Shown here is the relationship of traveling waves in the network.

$$V(z) = V^{+}e^{-j\beta z} + V^{-}e^{j\beta z} \qquad \text{Equation 3-22}$$

When the voltages are measured on analyzers in real life, the RMS values are recorded as:

$$V(z) = \frac{V^{+}}{\sqrt{2}}e^{-j\beta z} + \frac{V^{-}}{\sqrt{2}}e^{j\beta z} \qquad \text{Equation 3-23}$$
$$V(z) = V_a e^{-j\beta z} + V_b e^{j\beta z}$$
$$V(z) = a + b$$

The right and left traveling waves are represented by a and b, respectively. Therefore,

$$I(z) = \frac{1}{Z_o}\Delta V(z) \qquad \text{Equation 3-24}$$
$$I(z) = \frac{1}{Z_o}(a - b)$$

The traveling waves in terms of normalized current and voltage may be solved for as follows:

$$a = \frac{1}{2}(V + Z_o I) \qquad \text{Equation 3-25}$$
$$b = \frac{1}{2}(V - Z_o I) \qquad \text{Equation 3-26}$$

De-normalizing the impedance for a one-port network provides:

$$a = \frac{1}{2\sqrt{z_g}}(V + z_g I) \qquad \text{Equation 3-27}$$
$$b = \frac{1}{2\sqrt{z_g}}(V - z_g I) \qquad \text{Equation 3-28}$$

The average power delivered is found by:

$$P_{avg} = Re\{VI^{*}\} \qquad \text{Equation 3-29}$$
$$P_{avg} = |a|^2 - |b|^2$$

To maintain this property when z_g is complex, the final definition of a and b are found as follows:

$$a = \frac{1}{2\sqrt{r_g}}(V + z_g I) \qquad \text{Equation 3-30}$$
$$b = \frac{1}{2\sqrt{r_g}}(V - z_g^{*} I) \qquad \text{Equation 3-31}$$

Or, more generalized for an n-port network,

$$a_i = \frac{1}{2\sqrt{r_i}}(V + z_i I)$$
$$b_i = \frac{1}{2\sqrt{r_i}}(V - z_i^{*} I)$$

The network may now be described by its incident and reflected waves:

$$b = sa$$ Equation 3-32

The scattering matrix [S] for the two-port network is defined as

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$ Equation 3-33

which provides the following relationships:

$$b_1 = S_{11}a_1 + S_{12}a_2$$
$$b_2 = S_{21}a_1 + S_{22}a_2$$

a) Find the [S] matrix of the general two-port network.

Find S_{11}:

$$S_{11} = \left.\frac{b_1}{a_1}\right|_{a_2=0}$$ Equation 3-34

$$\frac{b_1}{a_1} = \frac{\frac{1}{2\sqrt{r_1}}(V_1 - z_1^* I_1)}{\frac{1}{2\sqrt{r_1}}(V_1 + z_1 I_1)} = \frac{\frac{V_1}{I_1} - z_1^*}{\frac{V_1}{I_1} + z_1}$$

$$S_{11} = \frac{b_1}{a_1} = \frac{Z_{in1} - r_1}{Z_{in1} + r_1}$$

Find S_{12}:

$$S_{12} = \left.\frac{b_1}{a_2}\right|_{a_1=0}$$ Equation 3-35

When $a_1 = 0$, $\frac{1}{2\sqrt{r_1}}(V_1 + z_1 I_1) = 0$

$$V_1 = -z_1 I_1 = -r_1 I_1$$

$$b_1 = \frac{1}{2\sqrt{r_1}}(V_1 - z_1^* I_1) = \frac{1}{2\sqrt{r_1}}(V_1 - r_1 I_1) = \frac{V_1}{\sqrt{r_1}}$$

$$a_2 = \frac{1}{2\sqrt{r_2}}(V_2 + z_2 I_2) = \frac{E_2}{2\sqrt{r_2}}$$

$$S_{12} = \frac{b_1}{a_2} = \frac{\frac{V_1}{\sqrt{r_1}}}{\frac{E_2}{2\sqrt{r_2}}} = \sqrt{\frac{r_2}{r_1}}\frac{2V_1}{E_2}$$

Similarly,

$$S_{21} = \left.\frac{b_2}{a_1}\right|_{a_2=0} = \sqrt{\frac{r_1}{r_2}}\frac{2V_2}{E_1}$$

$$S_{22} = \left.\frac{b_2}{a_2}\right|_{a_1=0} = \frac{Z_{in2} - r_2}{Z_{in2} + r_2}$$

b) Determine the expression for the reflection coefficient at Port 1 in terms of S-parameters.

$$\Gamma_{in1} = \frac{reflected\ wave}{transmitted\ wave} \qquad \text{Equation 3-36}$$

$$\Gamma_{in1} = \frac{V^-(z)}{V^+(z)} = \frac{V_b}{V_a}$$

$$\Gamma_{in1} = \frac{b_1}{a_1} = S_{11}$$

c) Derive the [S] matrix expression for a lossless two-port network.

To be a lossless network, the total power absorbed by the network is equal to zero.

$$P_{total} = P_{port1} + P_{port2} = 0 \qquad \text{Equation 3-37}$$

The power absorbed at a port is given by:

$$P_{port,i} = Re\{V_i I_i^*\} \qquad \text{Equation 3-38}$$

Using the definitions for a_i and b_i, the voltage and current in terms of a and b are found as follows:

$$a_i = \frac{1}{2\sqrt{r_i}}(V_i + z_i I_i) \qquad \text{Equation 3-39}$$

$$b_i = \frac{1}{2\sqrt{r_i}}(V_i - z_i^* I_i) \qquad \text{Equation 3-40}$$

$$I_i = \frac{2a_i\sqrt{r_i} - V_i}{z_i} = \frac{V_i - 2b_i\sqrt{r_i}}{z_i^*} \qquad \text{Equation 3-41}$$

$$V_i = \frac{1}{\sqrt{r_i}}(a_i z_i^* + b_i z_i) \qquad \text{Equation 3-42}$$

$$V_i = 2a_i\sqrt{r_i} - z_i I_i = 2b_i\sqrt{r_i} + z_i^* I_i$$

$$I_i = \frac{1}{\sqrt{r_i}}(a_i - b_i)$$

Allowing for computation of the power absorbed:

$$P_{port,i} = Re\{V_i I_i^*\}$$
$$P_{port,i} = Re\left\{\frac{1}{\sqrt{r_i}}(a_i z_i^* + b_i z_i)\frac{1}{\sqrt{r_i}}(a_i - b_i)\right\}$$

$$P_{port,i} = \frac{1}{r_i} Re\{(a_i z_i^* + b_i z_i)(a_i - b_i)\}$$

$$P_{port,i} = \frac{1}{r_i} Re\{a_i a_i z_i^* + b_i a_i z_i - a_i b_i z_i^* - b_i b_i z_i\}$$

$$P_{port,i} = \frac{1}{r_i} Re\{(a_i^2 r_i - j a_i^2 x_i) + (b_i a_i r_i + j b_i a_i x_i) - (a_i b_i r_i - j a_i b_i x_i) - (b_i^2 r_i + j b_i^2 x_i)\}$$

$$P_{port,i} = \frac{1}{r_i} Re\{(a_i^2 r_i + \ b_i a_i r_i - a_i b_i r_i - b_i^2 r_i) + (-j a_i^2 x_i + j b_i a_i x_i + j a_i b_i x_i - j b_i^2 x_i)\}$$

$$P_{port,i} = \frac{1}{r_i}\{|a_i^2 r_i| - |b_i^2 r_i|\}$$

$$P_{port,i} = |a_i|^2 - |b_i|^2$$

The following relationship is determined:

$$P_{total} = \ P_{port1} + \ P_{port2} \qquad \text{Equation 3-43}$$
$$P_{total} = \ (|a_1|^2 - |b_1|^2) + \ (|a_2|^2 - |b_2|^2)$$

Expanding out, this becomes:

$$P_{total} = \ |a_1|^2 + \ |a_2|^2 - |\mathrm{b}_1|^2 - \ |b_2|^2$$

$$P_{total} = \ a_1^* a_1 + \ a_2^* a_2 - b_1^* b_1 - b_2^* b_2$$

$$P_{total} = \ [a^*]^T[a] - [b^*]^T[b]$$

$$P_{total} = \ [a^*]^T[a] - [Sa^*]^T[Sa]$$

$$P_{total} = \ [a^*]^T[a] - [a^*]^T[S^*]^T[S][a]$$

$$P_{total} = \ [a^*]^T[a - S^{*T}Sa]$$

$$P_{total} = \ [a^*]^T[I - S^{*T}S][a] = 0$$

Therefore,

$$[I - S^{*T}S] = 0$$

Or, the matrix expression for a lossless two-port network is given by

$$\begin{bmatrix}1 & 0\\0 & 1\end{bmatrix} - \begin{bmatrix}s_{11}^{*} & s_{21}^{*}\\s_{12}^{*} & s_{22}^{*}\end{bmatrix}\begin{bmatrix}s_{11} & s_{12}\\s_{21} & s_{22}\end{bmatrix} = \begin{bmatrix}0 & 0\\0 & 0\end{bmatrix}$$

d) The schematic in Figure 3-13 was created in ADS and simulated.

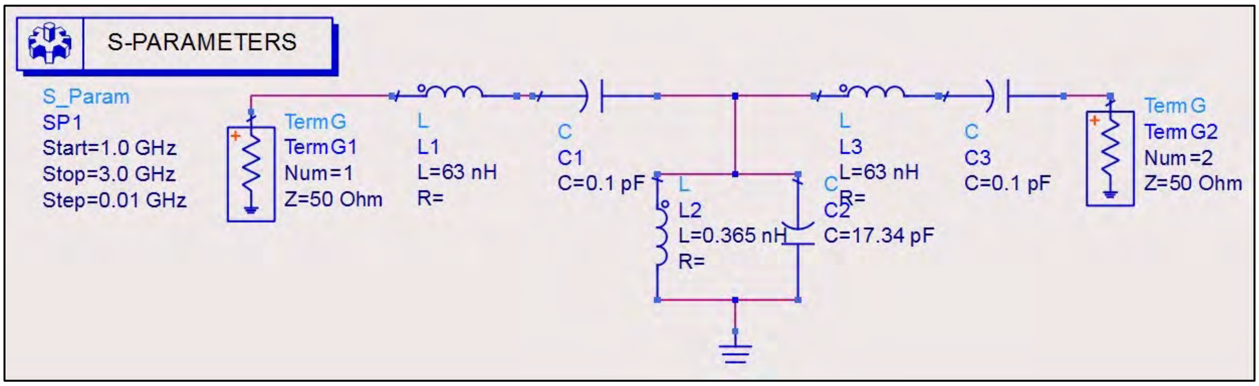


Figure 3-13. Shown here is a schematic created and simulated in ADS.

The simulated circuit provides the plot in Figure 3-14 for insertion vs. return loss. The S_{21} parameter, or insertion loss characteristic, reveals that the response at Port 2 due to a signal from Port 1 is almost non-existent, except for a small band centered around 2 GHz. The S_{11} parameter, or return loss characteristic, explains that the response at Port 1 due to a signal from Port 1 is inversely related to the S_{21} parameter. Most of the signal is reflected back outside of the frequency band 1.9 to 2.1 GHz. Once within the band, most of the signal from Port 1 is transmitted through to Port 2. The circuit is determined to be a bandpass filter.

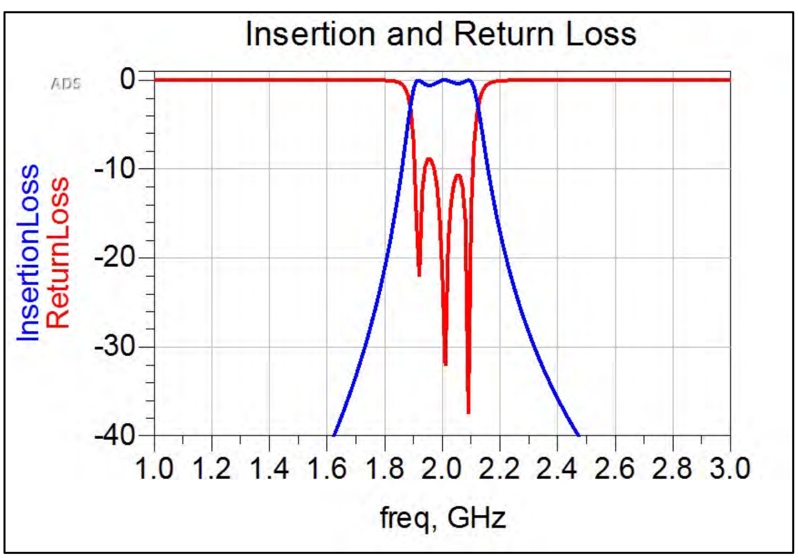


Figure 3-14. Plot of insertion versus return loss.

The [S] matrix for the circuit is given in Table 3-4.

freq	var("S")			
	(1,1)	(1,2)	(2,1)	(2,2)
1.000 GHz	0.996 - j0.084	-1.796E-5 - j2.138E-4	-1.796E-5 - j2.138E-4	0.996 - j0.084
1.500 GHz	0.977 - j0.215	-7.919E-4 - j0.004	-7.919E-4 - j0.004	0.977 - j0.215
2.000 GHz	0.007 - j0.078	0.993 + j0.084	0.993 + j0.084	0.007 - j0.078
2.500 GHz	0.958 + j0.286	-0.002 + j0.008	-0.002 + j0.008	0.958 + j0.286
3.000 GHz	0.988 + j0.153	-1.969E-4 + j0.001	-1.969E-4 + j0.001	0.988 + j0.153

Table 3-4. The [S] matrix for the circuit in question.

From the lossless properties determined in part c, the following S-parameter expressions must be satisfied:

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} - \begin{bmatrix} s_{11}^* & s_{21}^* \\ s_{12}^* & s_{22}^* \end{bmatrix} \begin{bmatrix} s_{11} & s_{12} \\ s_{21} & s_{22} \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$$

$$|s_{11}|^2 + |s_{21}|^2 = 1 \quad (1)$$
$$|s_{12}|^2 + |s_{22}|^2 = 1 \quad (2)$$
$$s_{11}^* s_{12} + s_{21}^* s_{22} = 0 \quad (3)$$
$$s_{12}^* s_{11} + s_{22}^* s_{21} = 0 \quad (4)$$

Test for Losslessness

freq	Test1	Test2	Test3	Test4
1.000 GHz	1.000	1.000	6.776E-21 / 3.142	6.776E-21 / 3.142
1.500 GHz	1.000	1.000	6.942E-19 / -0.675	6.942E-19 / 0.675
2.000 GHz	1.000	1.000	3.737E-16 / 2.917	3.737E-16 / -2.917
2.500 GHz	1.000	1.000	1.788E-18 / 1.326	1.788E-18 / -1.326
3.000 GHz	1.000	1.000	8.132E-20 / 0.000	8.132E-20 / 0.000

Table 3-5. The data here proves that the bandpass filter is a lossless two-port network.

All four tests were validated and the bandpass filter is a lossless two-port network (table 3-5).

Conclusion

S-parameters are useful for characterizing networks at high frequencies. However, it is cumbersome to calculate these parameters by hand and most of the time a network will be characterized physically using equipment analyzers. ADS is a helpful tool to check both designs and in understanding the scattering properties of an n-port network.

Problem 4: ABCD-Parameters

Problem Statement

Simplify the network in Figure 3-15 into a two-port network. Verify your answer in ADS.

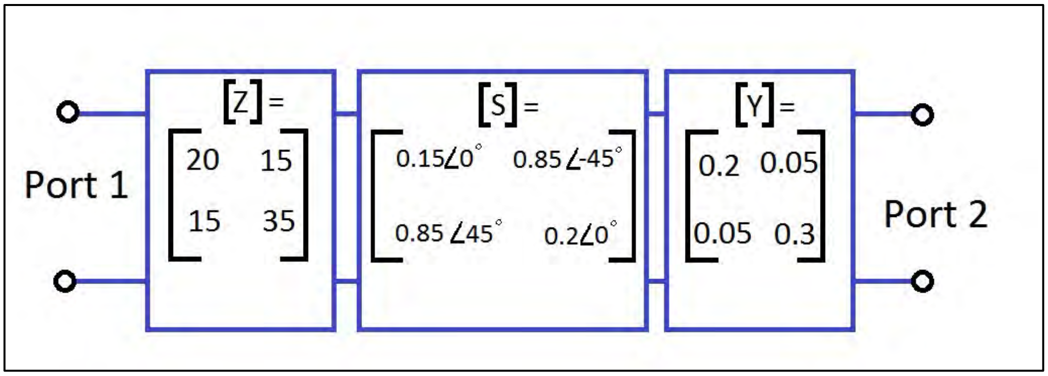


Figure 3-15. A schematic of the network for Problem 4.

Solution

Strategy

Convert each sub-network into ABCD parameters and multiply them together to find the overall two-port network characterization.

Execution

The ABCD matrix relationships are in terms of voltages and currents, as provided in Equation 3-44.

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} V_2 \\ I_2 \end{bmatrix} \qquad \text{Equation 3-44}$$

$$V_1 = AV_2 + BI_2$$
$$I_1 = CV_2 + DI_2$$

Convert the [Z] sub-network. This is accomplished using the [Z] parameter relationship following the ABCD sign convention.

$$V_1 = Z_{11}I_1 - Z_{12}I_2 \qquad \text{Equation 3-45}$$
$$V_2 = Z_{21}I_1 - Z_{22}I_2 \qquad \text{Equation 3-46}$$

$$A = \left.\frac{V_1}{V_2}\right|_{I2=0} \qquad \text{Equation 3-47}$$

$$A = \frac{Z_{11}I_1}{Z_{21}I_1} = \frac{20}{15} = 1.33$$

$$B = \left.\frac{V_1}{I_2}\right|_{V2=0} \qquad \text{Equation 3-48}$$

$$B = \frac{Z_{11}I_1 - Z_{12}I_2}{I_2} = \left.\frac{Z_{11}I_1}{I_2}\right|_{V2=0} - Z_{12}$$

$$B = Z_{11} \left.\frac{I_1}{\frac{I_1 Z_{21}}{Z_{22}}}\right|_{V2=0} - Z_{12}$$

$$B = \frac{Z_{11}Z_{22} - Z_{12}Z_{21}}{Z_{21}} = \frac{(700 - 225)}{15} = 31.67$$

$$C = \left.\frac{I_1}{V_2}\right|_{I2=0} \quad \text{Equation 3-49}$$

$$C = \frac{I_1}{Z_{21}I_1} = \frac{1}{15} = 0.067$$

$$D = \left.\frac{I_1}{I_2}\right|_{V2=0} \quad \text{Equation 3-50}$$

$$D = \frac{\frac{I_2 Z_{22}}{Z_{21}}}{I_2} = \frac{35}{15} = 2.33$$

The [ABCD] matrix for sub-network 1 is:

$$[ABCD]_1 = \begin{bmatrix} 1.33 & 31.67 \\ 0.067 & 2.33 \end{bmatrix}$$

Next, convert the [S] sub-network, using the pre-determined equations provided by Pozar[5], and assuming a 50-Ohm system:

$$A = \frac{(1 + S_{11})(1 - S_{22}) + S_{12}S_{21}}{2S_{21}} = 0.97\angle -45$$

$$B = Z_O \frac{(1 + S_{11})(1 + S_{22}) - S_{12}S_{21}}{2S_{21}} = 19.34\angle -45$$

$$C = \frac{1}{Z_O}\frac{(1 - S_{11})(1 - S_{22}) - S_{12}S_{21}}{2S_{21}} = 0.0005\angle 135$$

$$D = \frac{(1 - S_{11})(1 + S_{22}) + S_{12}S_{21}}{2S_{21}} = 1.02\angle -45$$

The [ABCD] matrix for sub-network 2 is:

$$[ABCD]_2 = \begin{bmatrix} 0.97\angle -45 & 19.34\angle -45 \\ 0.0005\angle 135 & 1.02\angle -45 \end{bmatrix}$$

Then, convert the [Y] sub-network using the pre-determined equations provided by Pozar:

$$A = \frac{-Y_{22}}{Y_{21}} = \frac{-0.3}{0.05} = -6$$

$$B = \frac{-1}{Y_{21}} = \frac{-1}{0.05} = -20$$

---

[5] *D. Pozar, Microwave Engineering 4th Ed., John Wiley& Sons, Danvers, MA, 2012, Chapter 4.*

$$C = \frac{-|Y|}{Y_{21}} = \frac{-(Y_{11}Y_{22} - Y_{12}Y_{21})}{Y_{21}} = -1.15$$

$$D = \frac{-Y_{11}}{Y_{21}} = \frac{-.2}{.05} = -4$$

The [ABCD] matrix for sub-network 3 is:

$$[ABCD]_3 = \begin{bmatrix} -6 & -20 \\ -1.15 & -4 \end{bmatrix}$$

Cascading the three networks together, gives us:

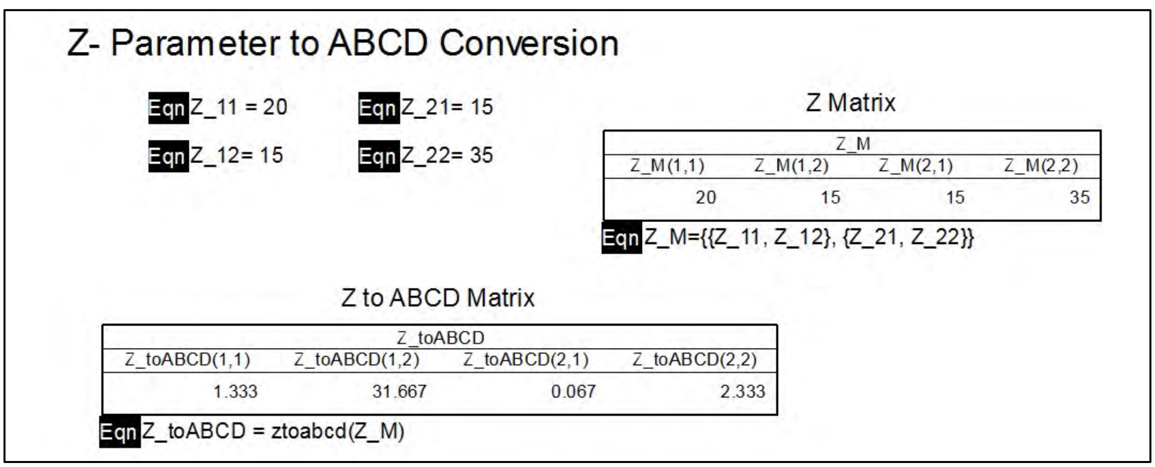
$$[ABCD]_{total} = [ABCD]_1[ABCD]_2[ABCD]_3 \qquad \text{Equation 3-51}$$

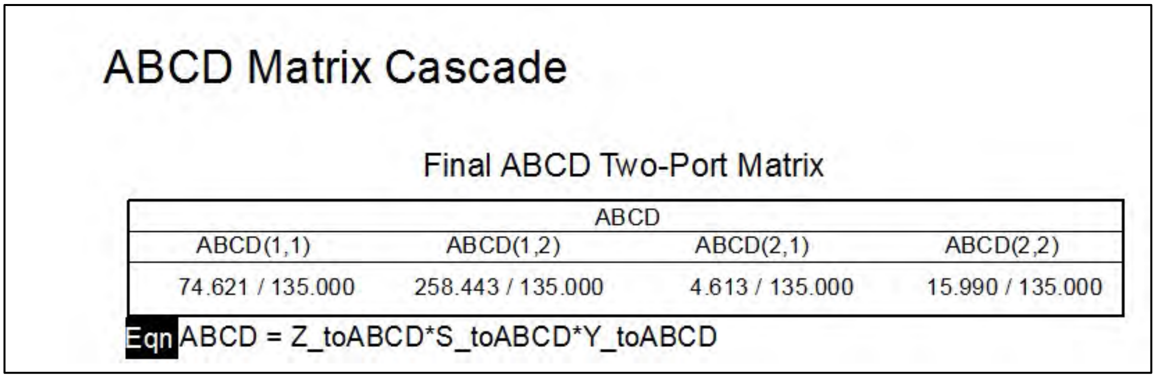
$$[ABCD]_{total} = \begin{bmatrix} 1.33 & 31.67 \\ 0.067 & 2.33 \end{bmatrix} \begin{bmatrix} 0.97\angle -45 & 19.34\angle -45 \\ 0.0005\angle 135 & 1.02\angle -45 \end{bmatrix} \begin{bmatrix} -6 & -20 \\ -1.15 & -4 \end{bmatrix}$$

$$[ABCD]_{total} = \begin{bmatrix} 74.621\angle 135 & 258.443\angle 135 \\ 4.613\angle 135 & 15.990\angle 135 \end{bmatrix}$$

Finally, check these results in ADS using the following steps:

1. Equation based in Data Display
   Each sub-network can be created as a matrix in the Data Display and then converted into ABCD parameters and cascaded together. The sample for the Z-network is shown in Figure 3-16, followed by the final ABCD matrix. Note, that the final ABCD matrix agrees with the hand calculation.



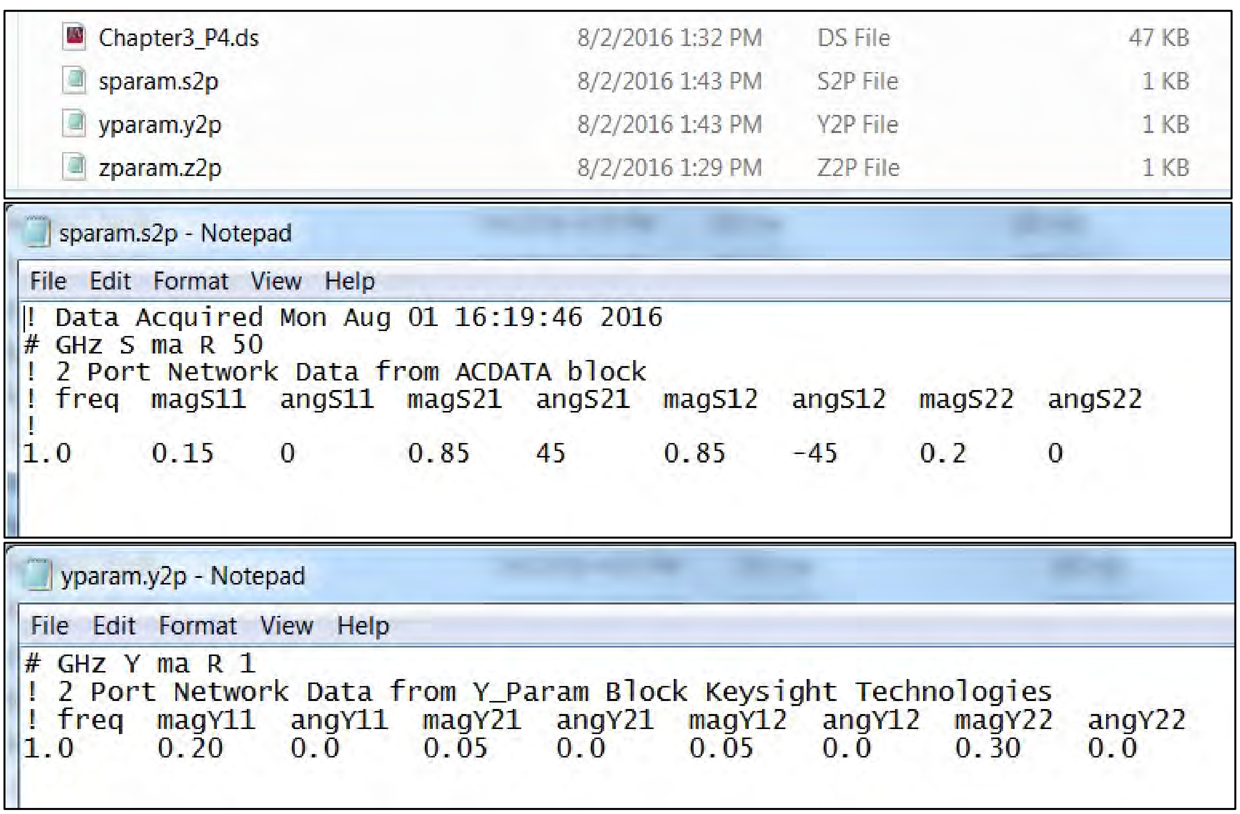


ABCD			
ABCD(1,1)	ABCD(1,2)	ABCD(2,1)	ABCD(2,2)
74.621 / 135.000	258.443 / 135.000	4.613 / 135.000	15.990 / 135.000

Figure 3-16. The top graphic shows how each sub-network converted into ABCD parameters and cascaded together. The final ABCD matrix is shown in the bottom graphic.

2. Using Touchstone data files
   An alternate method to checking the results in ADS is to create Touchstone data files for each sub-network. Any text editor can be used to create the file. Because the network analysis is frequency independent, only one line of parameters is needed per file. Each sub-network will have its own file, designated by the .s2p, .z2p and .y2p extensions. The ! denotes a comment, and the # symbol is a header required for each data file, with each item separated by a single space. The parameter information of magnitude and angle is separate by a tab.

   The touchstone text for each sub-network is shown in Figure 3-17.



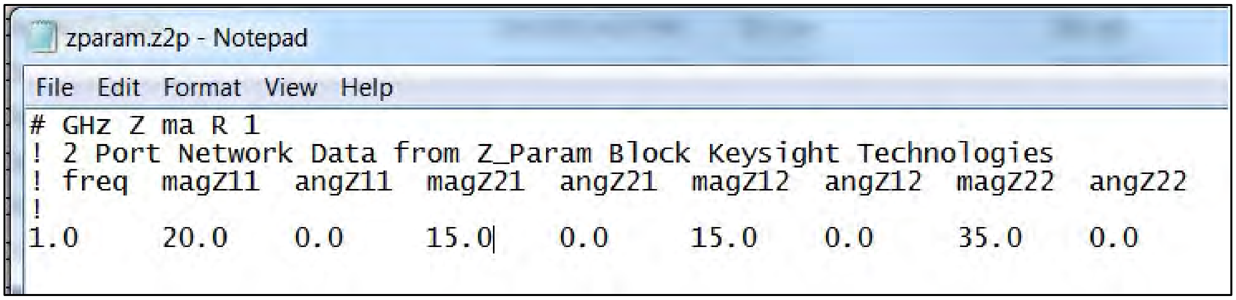


Figure 3-17. The Touchstone text for each sub-network.

The S-parameters have a 50-Ohm termination reference, but use a standard 1 Ohm for the Z- and Y-parameters. Using the Data Items palette, choose the SNP component and place it on a new schematic. Double click on the component and change the number of ports from 1 to 2 (Figure 3-18). Also in this window is where the data file can be specified.

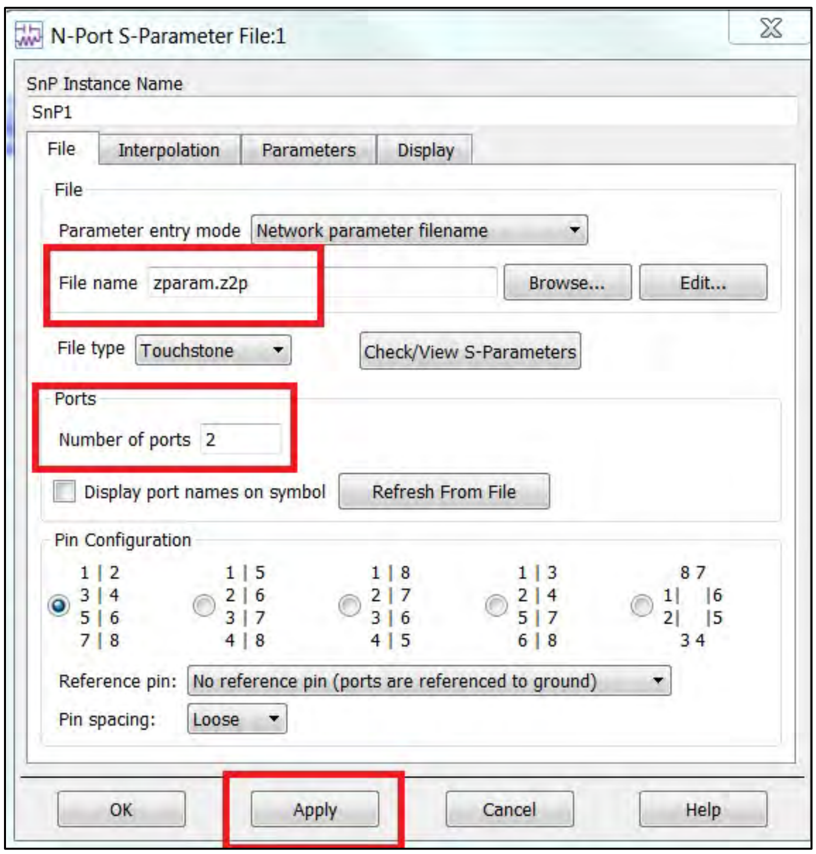


Figure 3-18. Changing the number of ports from 1 to 2..

Cascade the three networks together and simulate at a single frequency (Figure 3-19).

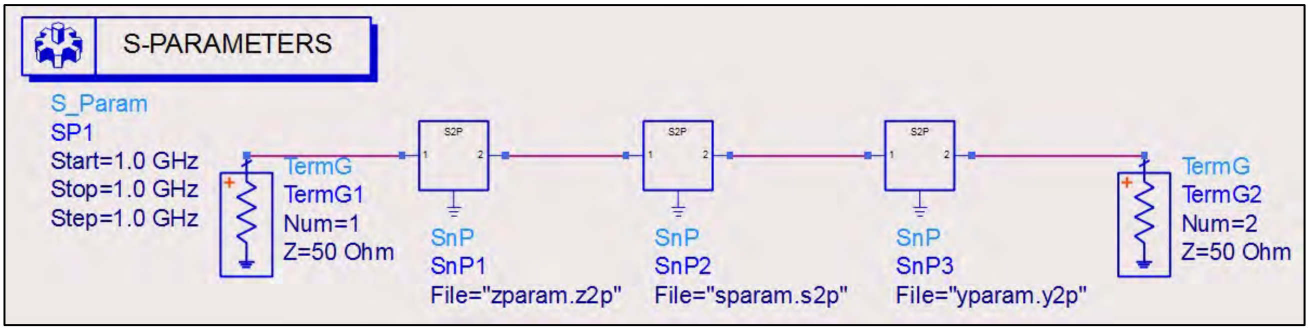


Figure 3-19. Here, the three networks are cascaded together and simulated at a single frequency.

ADS automatically calculates the S-parameters of the entire network. Next, create the Equation 'stoabcd(S)' in the Data Display (Figure 3-20). It's clear that the cascaded network agrees with both the hand calculation and the Data Display equation-based methods.

ABCD Matrix

freq	stoabcd(S)			
	(1,1)	(1,2)	(2,1)	(2,2)
1.000 GHz	74.613 / 135.000	258.419 / 135.000	4.612 / 135.000	15.988 / 135.000

Figure 3-20. The Equation 'stoabcd(S)' created in the Data Display.

Chapter 4 – Impedance Matching and Tuning, Distributed Elements

Problem 1: Single-Stub Tuning, Shunt Network

Problem Statement

Match a load with impedance Z_L= 75+j20 Ohms to a 50-Ohm line using a single-stub, open-circuit shunt tuning network at 2.4 GHz. Plot the reflection coefficient versus frequency.

Solution

Strategy

Convert the load impedance into a lumped-element network and match using a shunt piece of 50-Ohm transmission line using a Smith chart. Use the values found for the matching network to determine the electrical lengths needed for ADS simulation.

What to expect

The stub matching network will need to be composed of two lengths of line to match the load to both the resistive and reactive parts of the generator impedance. It is expected that the reflection coefficient, or S_{11} parameter, will display a perfect match at the designed frequency of 2.4 GHz and see a mismatch on either side frequency.

Execution

Convert load impedance into a lumped-element network:

$$Z_L = 75 + j20 = R + jX \qquad \text{Equation 4-1}$$
$$R = 75\ Ohms$$
$$jX = jwL = j20\ \rightarrow L = 1.32629\ nH$$

Normalize to 50 Ohms:

$$z_L = \frac{Z_L}{50} = 1.5 + j0.40$$

Plot z_L on a Smith chart and convert to load admittance y_L along a constant Standing Wave Ratio (SWR) circle (Figure 4-1). Because we are adding a shunt stub, we must convert to admittance terms instead of impedance.

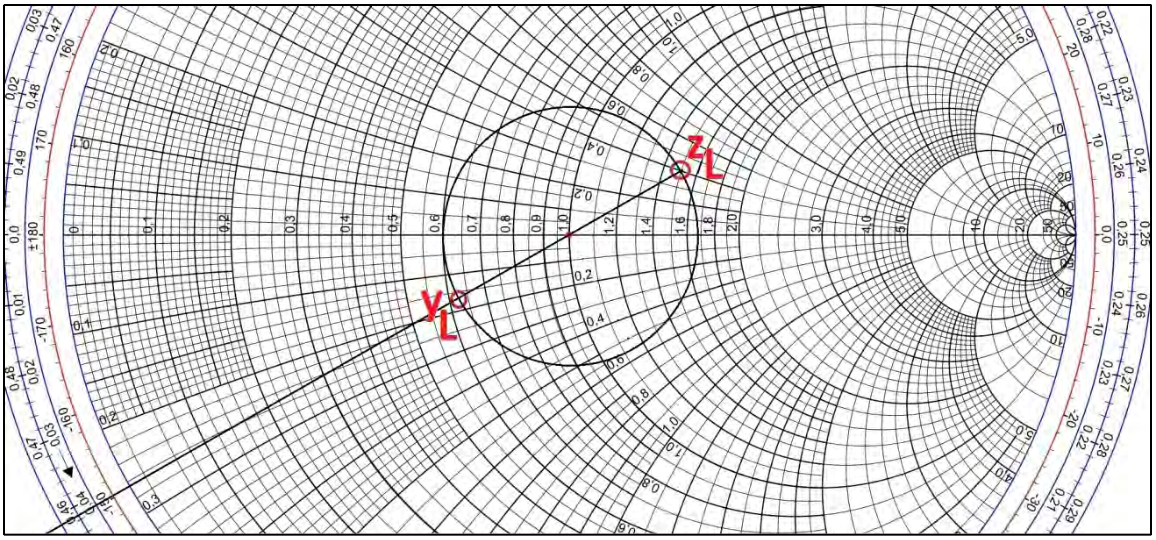


Figure 4-1. A plot z_L on a Smith chart converted to load admittance y_L.

Add a section of 50-Ohm transmission line toward the generator to reach the constant r=1 circle at Point A. The line length difference (0.187 λ, or 67.32°) between these two points is the length of transmission line needed to achieve a match for the real part of the transmission line. The line length on the Smith chart is in units of wavelength. Point A is located at 1+ j0.525.

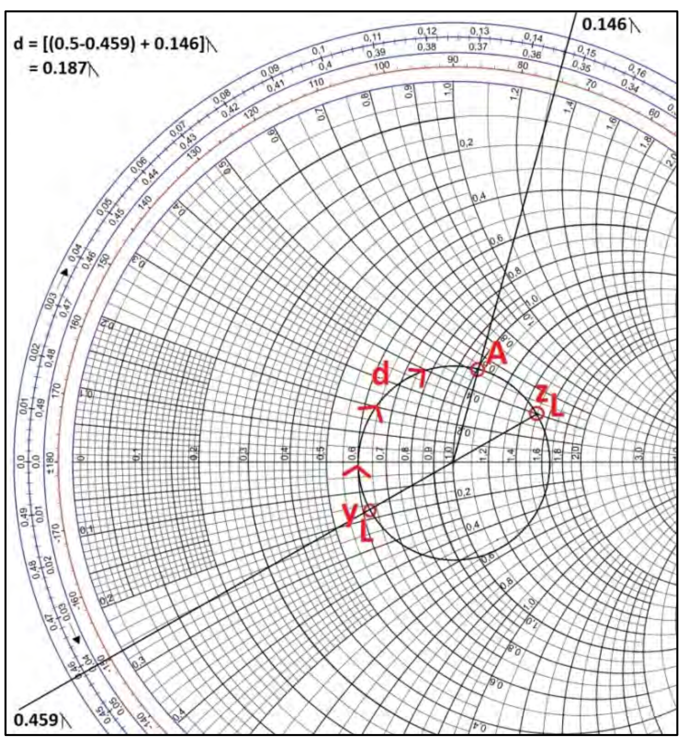


Figure 4-2. In this Smith chart, a section of 50-Ohm transmission line has been added toward the generator to reach the constant r=1 circle at Point A.

Now the imaginary part needs to be matched. To do this, add an open circuit shunt stub with a susceptance of -j0.525 (Point B) to match the load to the 50-Ohm line. Start at the open circuit point and move 90°, or 0.25 wavelengths, to account for the open circuit shunt. The wavelength difference between this point and point B will be the length of the open circuit shunt stub. Again, move toward the generator on the Smith Chart (Figure 4-3). The stub line length will be 0.423 **λ**, or 152.28°.

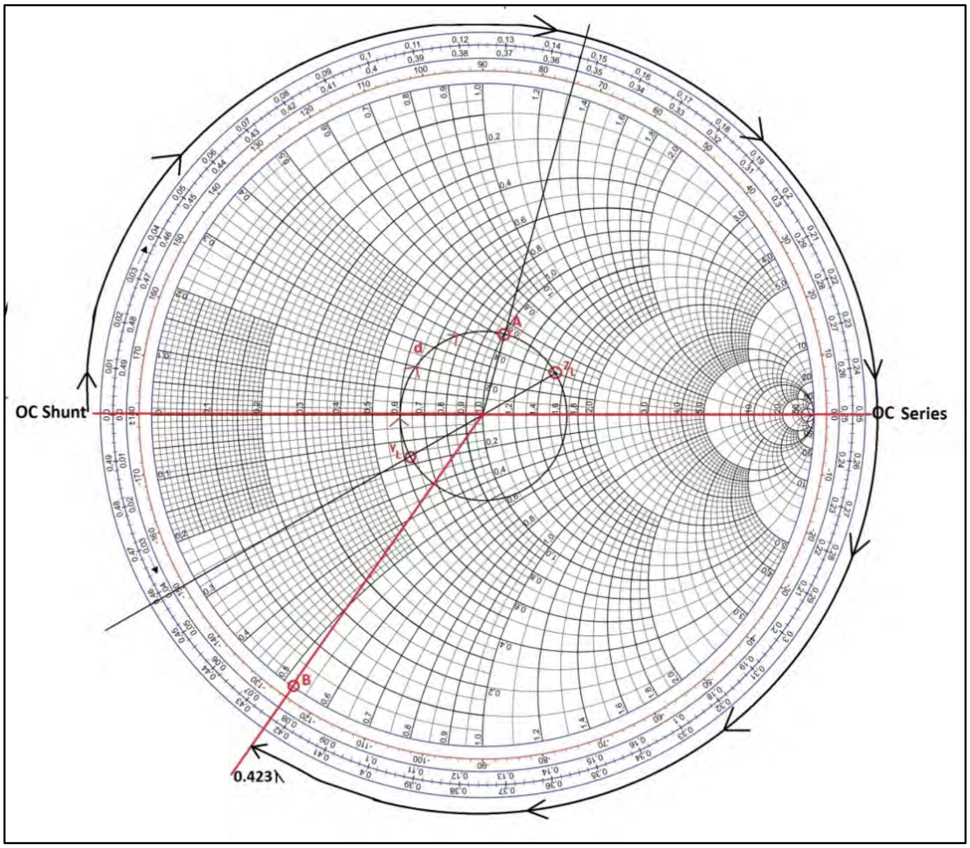


Figure 4-3. The new Smith chart with the imaginary part matched.

The final circuit in ADS is presented in Figure 4-4.

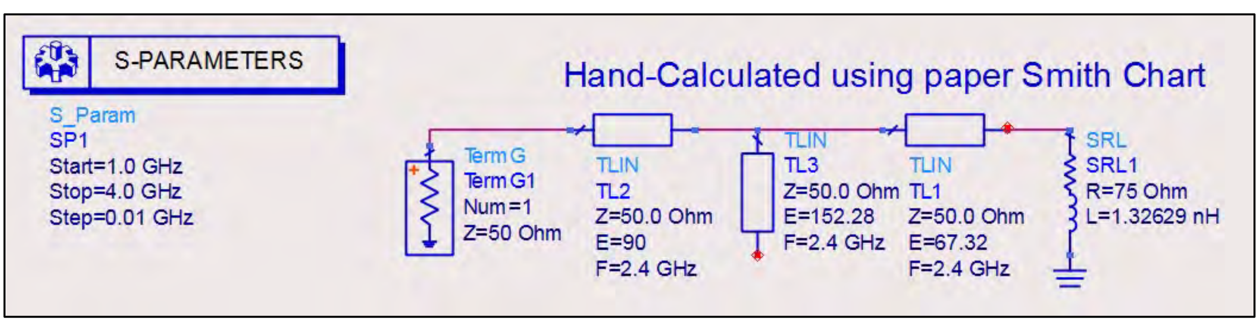


Figure 4-4. Shown here is the final circuit in ADS.

Plotting the reflection coefficient, S_{11}, validates that the minimum reflection occurs at the design frequency of 2.4 GHz (Figure 4-5).

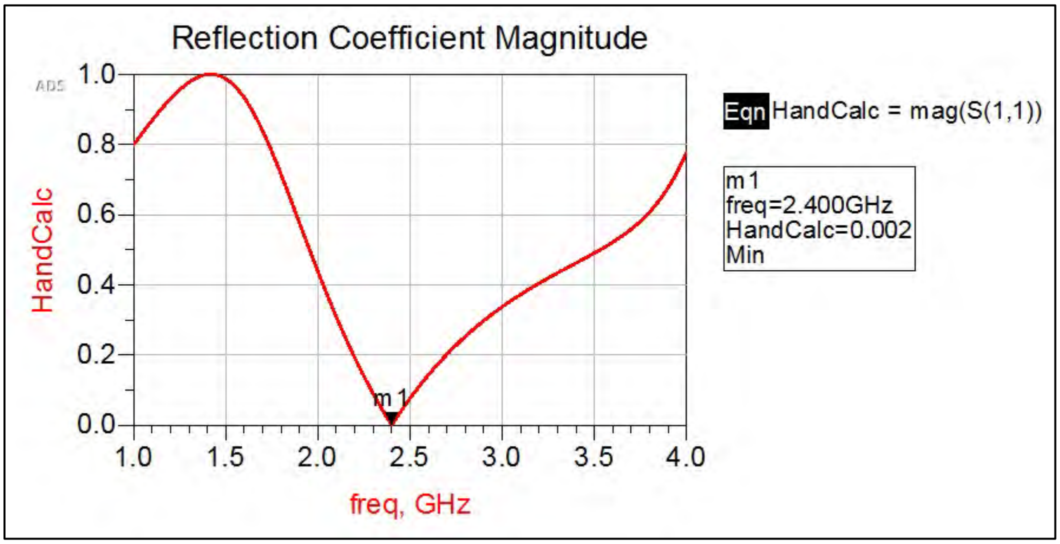


Figure 4-5. A plot of the reflection coefficient, S_{11}.

Alternate Method: ADS Smith Chart Tool

Open the ADS Smith Chart tool and input the frequency and load values for this exercise. Lock the load and source impedances. The shunt stub requires working with admittances, and by hand the impedance values were converted to the admittance values by traveling 180° on a constant VSWR circle. The Smith Chart tool allows the activation of the constant g-circles, which appear in red. Activate these circles to achieve the Smith chart shown in Figure 4-6.

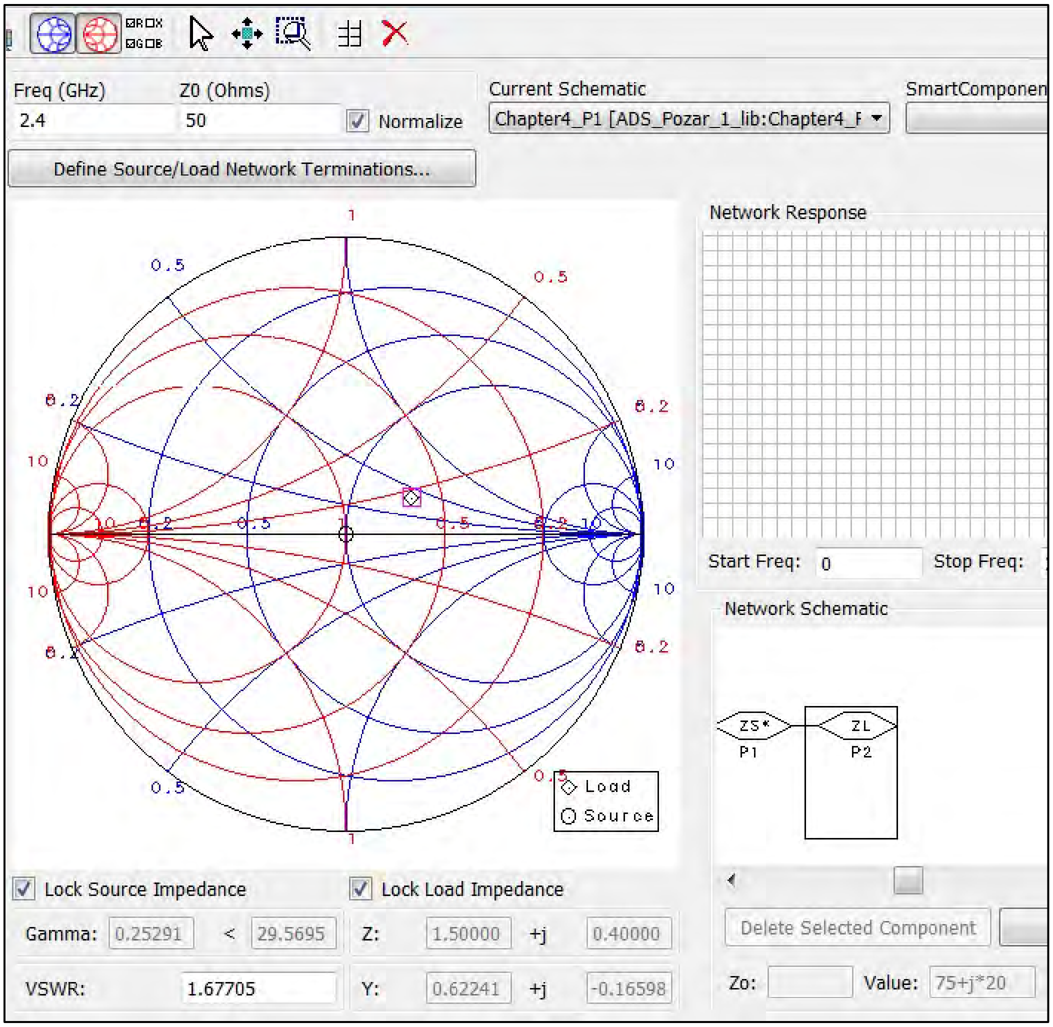


Figure 4-6. Shown here is the Smith chart achieved once the constant g-circles were activated.

Now, instead of converting the impedance and matching to a constant r=1 circle, we can match directly to a constant g=1 circle. Add a length of transmission line to reach the constant g=1 circle. The Smith chart shows that the line length needed is 67.11°, which is almost exactly the same as the line length calculated by hand (Figure 4-7).

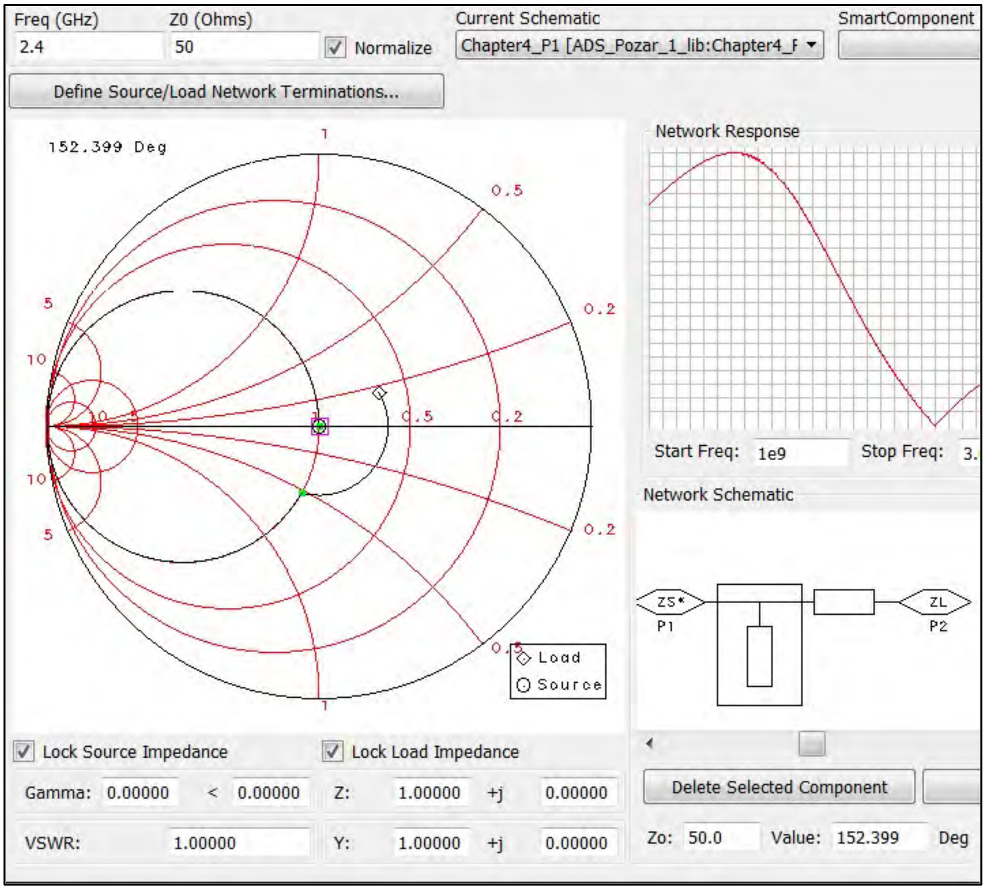


Figure 4-7. For a constant g=1 circle, the Smith chart shows that the line length needed is 67.11°.

Add an open-circuit shunt stub to reach the normalized 50-Ohm matching point in the center of the circle. The Smith Chart Tool reveals that a stub of line length 152.399° is needed. Again, this is also very similar to the hand calculated value. The frequency response of the circuit as compared to the hand calculated matching network is shown in Figure 4-8. While the traces appear to overlap perfectly, at the designed frequency, the precision of the ADS Smith Chart Tool versus estimation by eye can be measured.

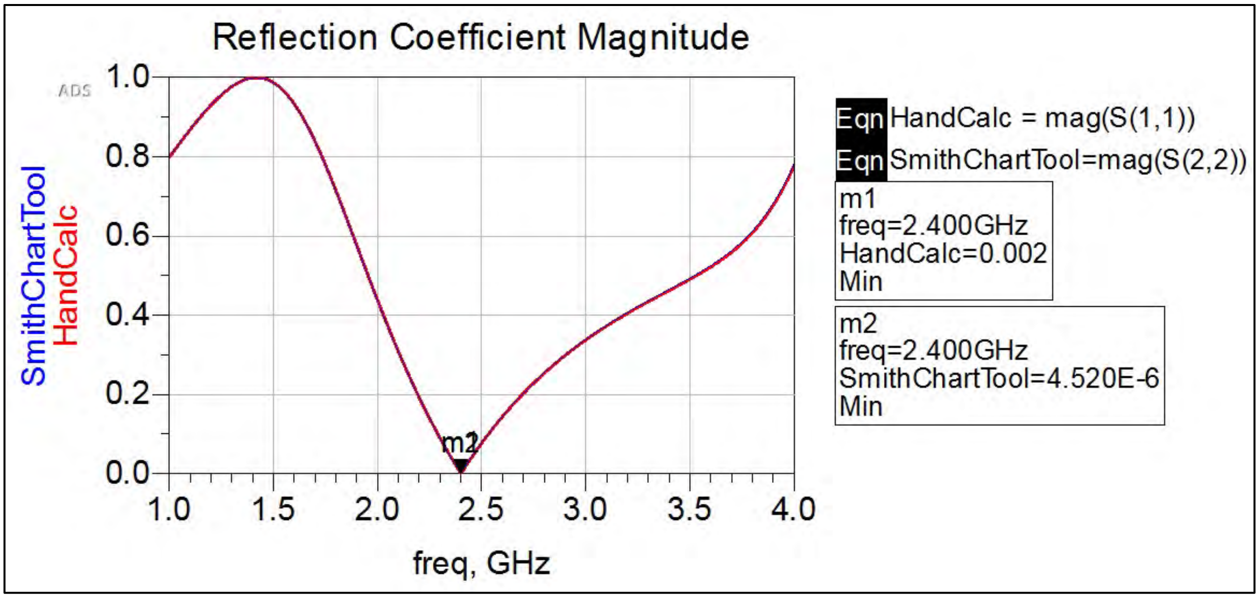


Figure 4-8. The graphic shows the frequency response of the circuit as compared to the hand calculated matching network.

Conclusion

The single-stub shunt open circuit provided a solution to the mismatched load impedance. Furthermore, it is easy to fabricate a shunt stub on a microstrip line. However, because the single-stub requires a length of line between the load and the stub, and this length changes with respect to the load impedance, it becomes problematic if different loads are attached. In addition, this exercise focused on the shortest stub match, but the number of possible solutions is infinite. Finally, the open circuit stub is favorable for fabrication of a microstrip line because a hole via the substrate to the ground conductor plane would need to be constructed for a short circuit stub. This is cumbersome and the extra fabrication step creates more room for error.

Problem 2: Single-Stub Tuning, Open Circuit

Problem Statement

Match the same load in Problem 1, Z_L= 75+j20 Ohms, to a 50-Ohm line using a single-stub open circuit series tuning network at 2.4 GHz. Plot the reflection coefficient to see the frequency response and compare the result to the open circuit shunt-matching network in Problem 1.

Solution

Strategy

Use the ADS Smith Chart Tool to match the circuit using a series open stub. Plot the shunt and series stubs against one another to determine which method is better for transmission lines.

What to expect

The use of an open-circuit shunt stub will result in a longer length of line for the tuning stub. This is expected to result in a narrower bandwidth for the frequency response.

Execution

The line length up to the stub will be exactly the same as Problem 2. Convert load impedance into a lumped-element network as in Figure 4-1:

$$Z_L = 75 + j20 = R + jX$$

$$R = 75\ Ohms$$
$$jX = jwL = j20\ \rightarrow L = 1.32629\ nH$$

Normalize to 50 Ohms:

$$z_L = \frac{Z_L}{50} = 1.5 + j0.40$$

Due to the nature of the transmission line, shunt-stub matching is preferred. Therefore, the ADS Smith Chart tool stub components are shunt stubs. While it is not possible to use the Smith Chart tool to directly match the series stub, we can still determine the line lengths needed. Match the load to the constant r = 1 circle and note the line length (Figure 4-9).

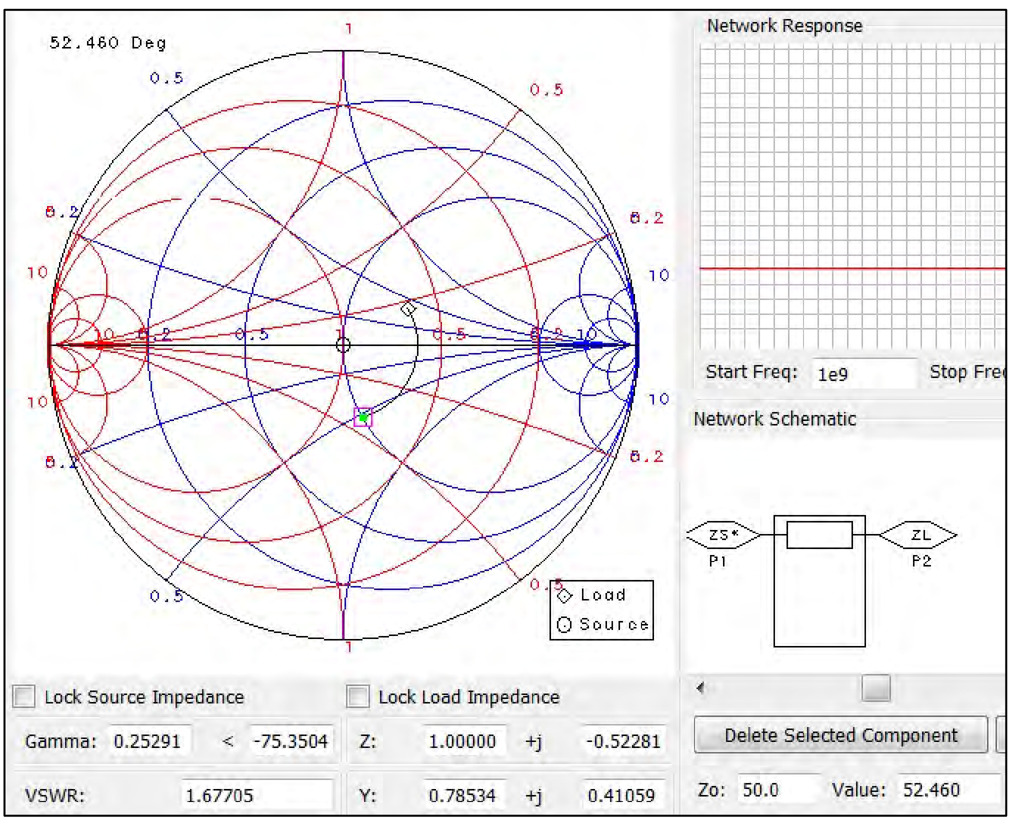


Figure 4-9. Here, the load is matched to the constant r = 1 circle.

Next, the length of line for an open circuit stub in series with the impedance equivalent of +j0.52281 is added. Remove the transmission line component and change the load impedance so that it is extremely large and resembles an open circuit. Add a length of line from the open-circuit node until the point (0.00,+j0.52281) is reached. For the series stub, the TLIN4 component will be used. This component specifies the incident and return waves. The bottom conductor must be explicitly grounded. Record the two line values into the ADS schematic and simulate (Figure 4-10).

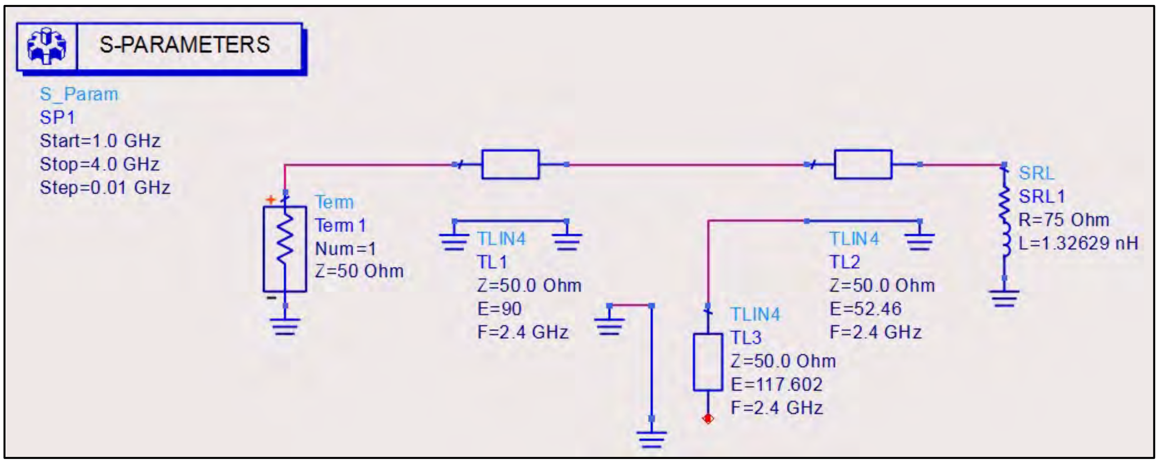


Figure 4-10. Here, the length of line for an open circuit stub in series with the impedance equivalent of +j0.52281 has been added.

The magnitude of the reflection coefficient for both matching methods is shown in Figure 4-11.

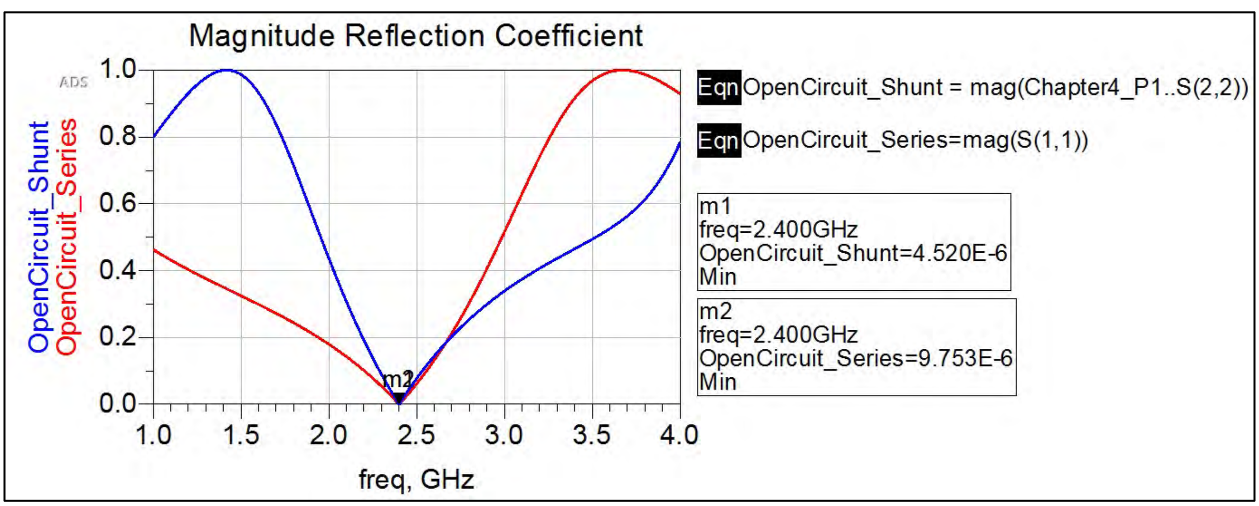


Figure 4-11. This graphic depicts the magnitude of the reflection coefficient for both matching methods.

The reflection coefficients are realistically equal at the design center frequency, but are different in the sidebands. To get a better look at the frequency response, the dB return loss is plotted (Figure 4-12).

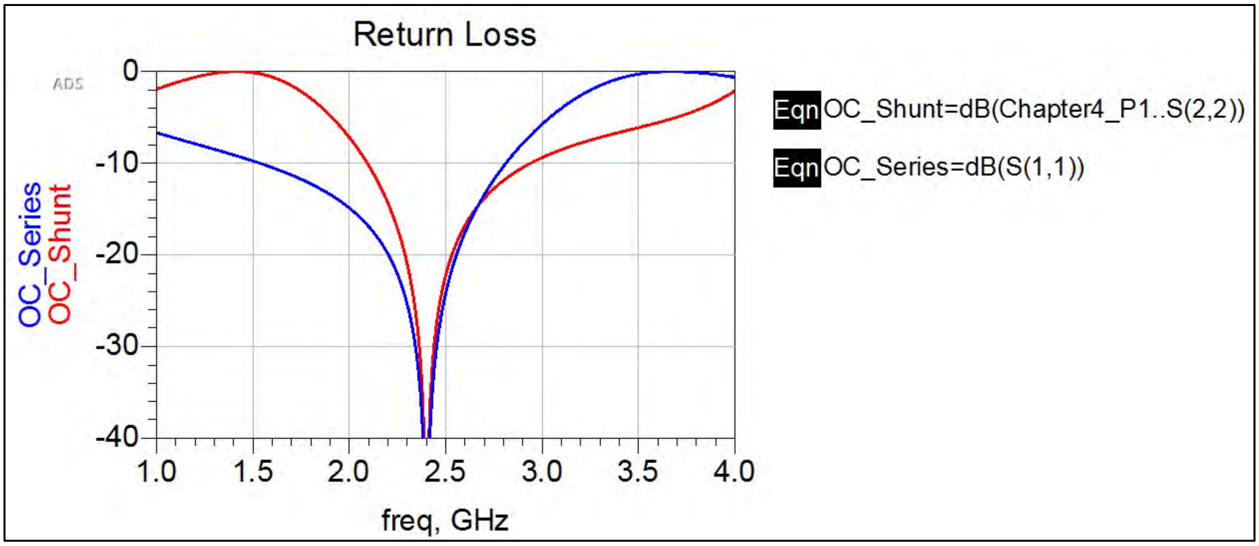


Figure 4-12. A plot of the dB return loss.

Assuming a 20-dB out-of-band rejection, the shunt tuning network provides a narrower passband. This could be favorable depending on what the design specifications require.

Conclusion

For narrower bandwidth, the shunt open-circuit stub is favored over the series open-circuit stub. The reader may also compare short circuit series and shunt tuning networks. For manufacturing using a microstrip line, open stub-shunt tuning networks are preferred due to narrow bandwidth and ease of fabrication. A final note is that the purpose of the stub tuner is to match the load to a generator through means of canceling out a reactance. The resulting fixed tuner provides a solution for only the frequency at which the tuner was designed.

Problem 3: Maximally Flat Quarter-Wave Transformer

Problem Statement

Design an N-section maximally flat quarter-wave transformer to match a 100-Ohm load to a 50-Ohm microstrip line. Design for values of N = 1, 3, 5. Design for a center frequency of 3 GHz and compare each design graphically using an out of bandwidth rejection VSWR value of 1.2.

The microstrip line specifications are as follows:

Dielectric thickness	20 mil
Dielectric constant	4.2
Dielectric loss	0.0002
Conductor thickness	0.5 mil
Conductor	Copper

Solution

Strategy

Use the maximally flat N-section transformer technique[6] to design quarter-wave transformers using 1, 3 and 5 sections. After determining the section impedances, ADS LineCalc will be used to realize the microstrip line transformer. The transformer will then be simulated in ADS for bandwidth comparison.

What to expect

It is expected that the higher N-sectioned transformer will produce a wider bandwidth. The fractional bandwidth is dependent on a constant A, which describes the reflection coefficient between the source and load impedances.

$$A = 2^{-N}\frac{Z_L - Z_g}{Z_L + Z_g} \qquad \text{Equation 4-2}$$

The relationship of the constant A and the fractional bandwidth is given by[1]:

$$\frac{\Delta f}{f_o} = 2 - \frac{4}{\pi}cos^{-1}\left[\frac{1}{2}\left(\frac{\Gamma_m}{|A|}\right)^{1/N}\right] \qquad \text{Equation 4-3}$$

Mathematically, as the number of sections N increases, the constant A decreases, causing an increased fractional bandwidth. Physically, a single piece of quarter-wavelength transmission line will only look like a quarter wavelength at one frequency. By cascading multiple sections and increasing the line length, more frequencies appear to have this quarter-wavelength matching. Thus, the higher number of sections increases the overall line length and allows a wider bandwidth of frequencies that appear matched to the load.

Execution

The multisection transformer can be visually represented by Figure 4-13.

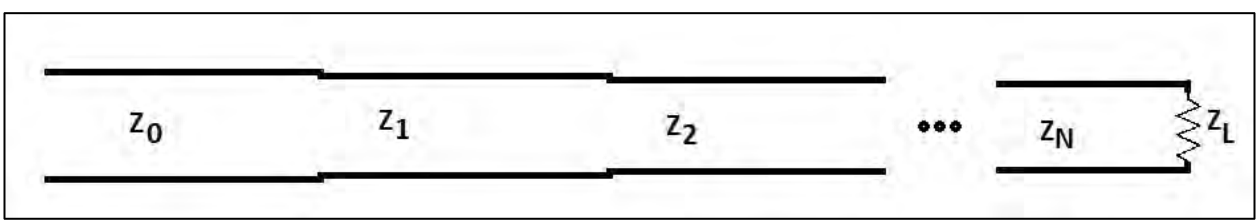


Figure 4-13. A visual representation of the multisection transformer.

The characteristic impedance of each section can be found by the following relation[7]:

$$ln\, Z_{n+1} = ln\, Z_n + 2^{-N}C_n^N\, ln\frac{Z_L}{Z_0} \qquad \text{Equation 4-4}$$

---

[6] *D. Pozar, Microwave Engineering, 4th edition, John Wiley and Sons, New Jersey, 2012.*
[7] *D. Pozar, Microwave Engineering, 4th edition, John Wiley and Sons, New Jersey, 2012.*

Where C_n^N is the binomial coefficient in the reflection coefficient magnitude binomial expansion:

$$C_n^N = \frac{N!}{(N-n)!n!} \qquad \text{Equation 4-5}$$

N = 1, Single Section Transformer
It is already known that a single section quarter-wavelength line has the characteristic impedance of:

$$Z_0 = \sqrt{Z_{in}Z_L} = \sqrt{(50)(100)} = 70.7107\ Ohms$$

Use this fact to double check the characteristic impedance equation provided above.

$$ln\, Z_{n+1} = ln\, Z_n + 2^{-N} C_n^N\, ln \frac{Z_L}{Z_0}$$

$$ln\, Z_1 = \ln Z_0 + 2^{-1} C_0^1\, ln \frac{Z_L}{Z_0}$$

$$ln\, Z_1 = ln\, 50 + 2^{-1} \frac{1!}{(1-0)!\,0!} ln \frac{100}{50}$$

$$ln\, Z_1 = 4.2586$$

$$Z_1 = e^{4.2586} = 70.7107\ Ohms \quad \surd$$

ADS LineCalc provides the schematic in Figure 4-14 for the N=1 quarter-wavelength transformer.

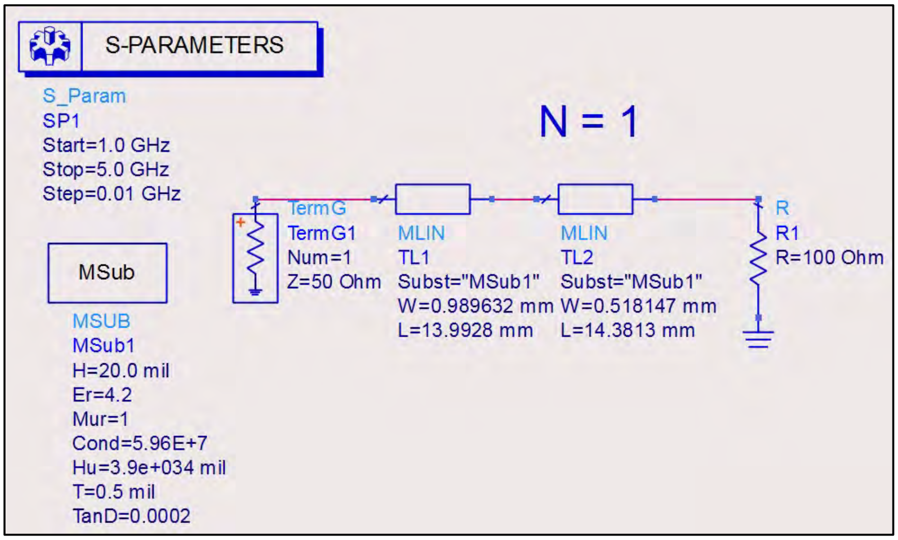


Figure 4-14. The schematic for N=1.

N = 3, Multi Section Transformer

$$ln\, Z_1 = ln\, Z_0 + 2^{-3} C_0^3\, ln \frac{Z_L}{Z_0}$$

$$Z_1 = e^{3.99867} = 54.5254\ Ohms$$

$$ln\, Z_2 = ln\, Z_1 + 2^{-3} C_1^3\, ln \frac{Z_L}{Z_0}$$

$$Z_2 = e^{4.2586} = 70.7107\ Ohms$$

$$ln\,Z_3 = ln\,Z_2 + 2^{-3}C_2^3\,ln\frac{Z_L}{Z_0}$$
$$Z_3 = e^{4.51853} = 91.7004\ Ohms$$

ADS LineCalc provides the schematic in Figure 4-15 for the N=3 quarter-wavelength transformer.

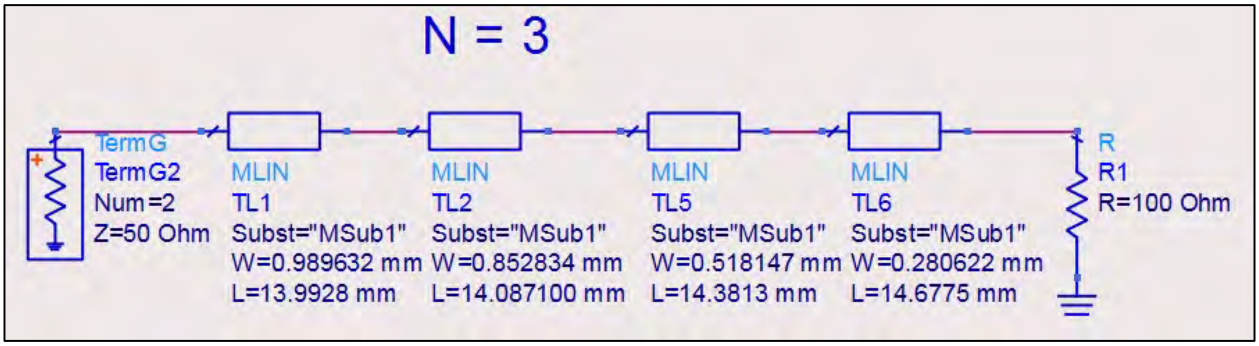


Figure 4-15. The schematic for N=3.

N = 5, Multi Section Transformer

$$ln\,Z_1 = ln\,Z_0 + 2^{-5}C_0^5\,ln\frac{Z_L}{Z_0}$$
$$Z_1 = e^{3.93368} = 51.0949\ Ohms$$

$$ln\,Z_2 = ln\,Z_1 + 2^{-5}C_1^5\,ln\frac{Z_L}{Z_0}$$
$$Z_2 = e^{4.04199} = 56.9394\ Ohms$$

$$ln\,Z_3 = ln\,Z_2 + 2^{-5}C_2^5\,ln\frac{Z_L}{Z_0}$$
$$Z_3 = e^{4.2586} = 70.7107\ Ohms$$

$$ln\,Z_4 = ln\,Z_3 + 2^{-5}C_3^5\,ln\frac{Z_L}{Z_0}$$
$$Z_3 = e^{4.4752} = 87.8126\ Ohms$$

$$ln\,Z_5 = ln\,Z_4 + 2^{-5}C_4^5\,ln\frac{Z_L}{Z_0}$$
$$Z_3 = e^{4.58351} = 97.8572\ Ohms$$

ADS LineCalc provides the schematic in Figure 4-16 for the N=5 quarter-wavelength transformer.

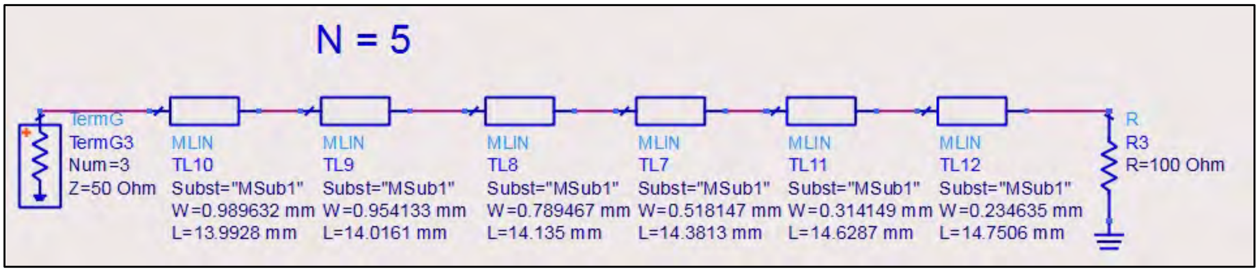


Figure 4-16. The schematic for N=5.

Simulate and plot the reflection coefficient, or magnitude S(1,1), for all three transformers on the same plot (Figure 4-17).

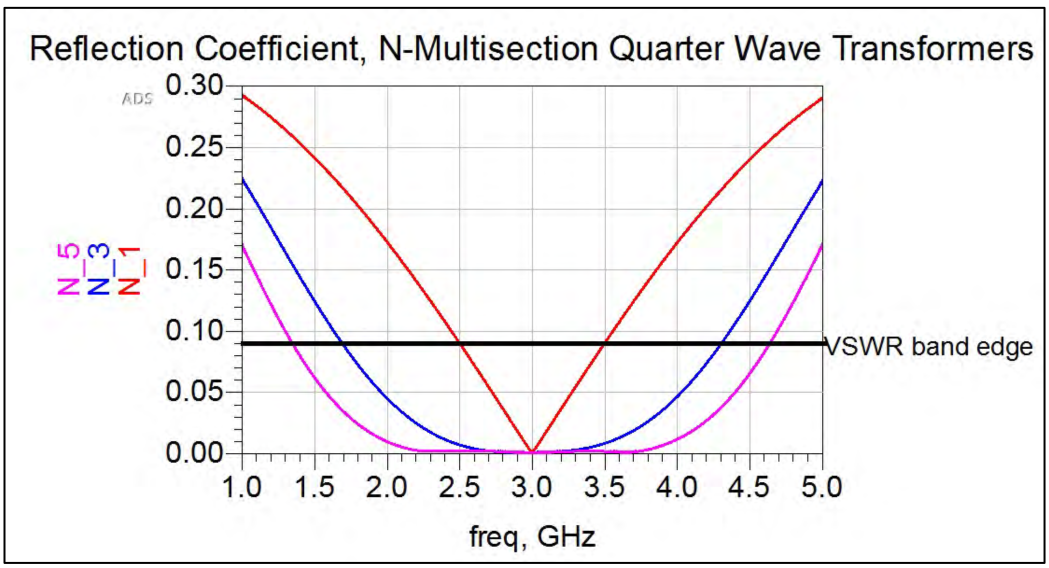


Figure 4-17. A plot of the reflection coefficient for all three transformers.

As the number of sections increase, the passband becomes wider and flatter as expected, due to the maximally flat design technique used. The measured bandwidths of each circuit is shown in Figure 4-18.

Bandwidth N=1: 2.5 - 3.5 = 1 GHz
Bandwidth N=3: 1.69 - 4.3 = 2.61 GHz
Bandwidth N=5: 1.34 - 4.64 = 3.3 GHz

Figure 4-18. Measured bandwidths of each circuit.

Conclusion

The quarter-wave transformer was introduced in Chapter 1 where a λ/4 section of transmission line was inserted between a load and source with a characteristic impedance of $Z_0 = \sqrt{Z_{in}Z_L}$. This characteristic impedance was chosen so that the superposition of the multiple reflections on the line add destructively at the input, resulting in minimum reflection at the input port and making the load well matched to the source. This single-section transformer produces a very narrowband impedance match because of the short line length. If a wider band is desired, the multisection quarter-wave transformer is used to increase the line length and the number of frequencies that are seen, as well as matched by the generator. This exercise predefined the number of sections for the transformers to drive the design, but the same process can likewise be used to find the number of sections required to meet a bandwidth specification, which is more realistic.

Problem 4: Chebyshev Transformer

Problem Statement

Design a 3-section Chebyshev quarter-wave transformer to match a 100-Ohm load to a 50-Ohm microstrip line. Design for a center frequency of 3 GHz and compare the results graphically to the maximally flat transformer designed in Problem 3 for N=3 and 5. Use an out-of-bandwidth rejection VSWR value of 1.2.

The microstrip line specifications are as follows:

Dielectric thickness	20 mil
Dielectric constant	4.2
Dielectric loss	0.0002
Conductor thickness	0.5 mil
Conductor	Copper

Solution

Strategy

Use the Chebyshev N-section transformer technique[8] to design the transformer. After determining the section impedances, ADS LineCalc will be used to realize the microstrip line transformer and then be simulated in ADS for bandwidth comparison.

What to expect

It is expected that the Chebyshev transformer will perform better than the maximally flat transformer in terms of bandwidth. The Chebyshev design uses polynomials to optimize bandwidth at the sacrifice of introducing an in-band ripple. This ripple will stay constant at the VSWR bandwidth rejection value, thus creating a wider passband with some frequencies better matched than others. This equal-ripple is a characterization behavior of the Chebyshev transformer and matches a wider band of frequencies than the maximally flat transformer, while still performing optimally at the center frequency.

---

[8] *D. Pozar, Microwave Engineering, 4th edition, John Wiley and Sons, New Jersey, 2012.*

Execution

The reflection coefficient as a function of frequency in terms of a Fourier cosine series for odd N is

$$\Gamma(\theta) = 2e^{-jN\theta}[\Gamma_0 \cos N\theta + \Gamma_1 \cos(N-2)\theta + \cdots + \Gamma_{(N-1)/2} \cos\theta] \qquad \text{Equation 4-6}$$

For N=3,

$$\Gamma(\theta) = 2e^{-j3\theta}[\Gamma_0 \cos 3\theta + \Gamma_1 \cos\theta]$$

The series is symmetrical, so

$$\Gamma_2 = \Gamma_0$$

which can be reduced in terms of the Chebyshev polynomials as

$$\Gamma(\theta) = Ae^{-j3\theta}T_3\,(\sec\theta_m \cos\theta) \qquad \text{Equation 4-7}$$

The next step is to find the constant A. Let the design specification for the VSWR band-edge cutoff be the maximum reflection coefficient, as given by Equation 4-8:

$$\Gamma_m = max[Ae^{-j3\theta}T_3\,(\sec\theta_m \cos\theta)] \qquad \text{Equation 4-8}$$
$$\Gamma_m = |A| max[e^{-j3\theta}T_3\,(\sec\theta_m \cos\theta)]$$
$$\Gamma_m = |A|(1)$$
$$\Gamma_m = |A|$$

Equate the Fourier series with the Chebyshev polynomials to determine Γ_0 and Γ_1:

$$2e^{-j3\theta}[\Gamma_0 \cos 3\theta + \Gamma_1 \cos\theta] = Ae^{-j3\theta}[\sec^3\theta_m\,(\cos 3\theta + 3\cos\theta) - 3\sec\theta_m \cos\theta]$$

For Γ_0:

$$2e^{-j3\theta}\Gamma_0 \cos 3\theta = Ae^{-j3\theta}\sec^3\theta_m\,(\cos 3\theta)$$
$$2\Gamma_0 = \Gamma_m \sec^3\theta_m$$
$$\Gamma_0 = \frac{\Gamma_m}{2}\sec^3\theta_m \qquad \text{Equation 4-9}$$

For Γ_1:

$$2e^{-j3\theta}\Gamma_1 \cos\theta = Ae^{-j3\theta}[3\sec^3\theta_m \cos\theta - 3\sec\theta_m \cos\theta]$$
$$2\Gamma_1 = 3\Gamma_m \sec\theta_m\,[\sec^2\theta_m - 1]$$
$$\Gamma_1 = \frac{3}{2}\Gamma_m \sec\theta_m\,[\sec^2\theta_m - 1] \qquad \text{Equation 4-10}$$

Find θ_m. Use the secant approximation in conjunction with the Chebyshev polynomials.

$$\sec\theta_m \approx \cosh\left[\frac{1}{N}\cosh^{-1}\left(\frac{\ln^{Z_L}/_{Z_0}}{2\Gamma_m}\right)\right] \qquad \text{Equation 4-11}$$
$$\theta_m = \sec^{-1}\left[\cosh\left[\frac{1}{N}\cosh^{-1}\left(\frac{\ln^{Z_L}/_{Z_0}}{2\Gamma_m}\right)\right]\right]$$
$$\theta_m = \sec^{-1}\left[\cosh\left[\frac{1}{3}\cosh^{-1}\left(\frac{\ln 2}{2(0.0909)}\right)\right]\right]$$

$$\theta_m = 0.62586 \frac{rad}{s}$$

Solve for Γ0, Γ1, Γ2. For Γ0 and Γ2:

$$\Gamma_0 = \frac{\Gamma_m}{2} sec^3\, \theta_m$$

$$\Gamma_0 = \frac{0.0909}{2} sec^3(0.62586)$$

$$\Gamma_0 = 0.085385 = \Gamma_2$$

For Γ1:

$$\Gamma_1 = \frac{3}{2}\Gamma_m\, sec\, \theta_m\, [sec^2\, \theta_m - 1]$$

$$\Gamma_1 = \frac{3}{2}(0.0909)\, sec(0.62586)\, [sec^2(0.62586) - 1]$$

$$\Gamma_1 = 0.087901$$

Find the impedances of each section. From the maximally flat design in Problem 3.

$$ln\, Z_{n+1} = ln\, Z_n + 2^{-N} C_n^N\, ln \frac{Z_L}{Z_0}$$

This can be approximated to:

$$ln\, Z_{n+1} = ln\, Z_n + 2\Gamma_n \qquad \text{Equation 4-12}$$

$$Z_1 = e^{ln\, Z_0 + 2\Gamma_0} = e^{ln\, 50 + 2(0.085385)}$$

$$Z_1 = 59.3109$$

$$Z_2 = e^{ln\, Z_1 + 2\Gamma_1} = e^{ln\, 59.3109 + 2(0.087901)}$$

$$Z_2 = 70.7107$$

$$Z_3 = e^{ln\, Z_2 + 2\Gamma_2} = e^{ln\, 70.7107 + 2(0.085385)}$$

$$Z_3 = 83.8783$$

Use ADS LineCalc to determine the microstrip line section widths and lengths. The schematic is shown in Figure 4-19.

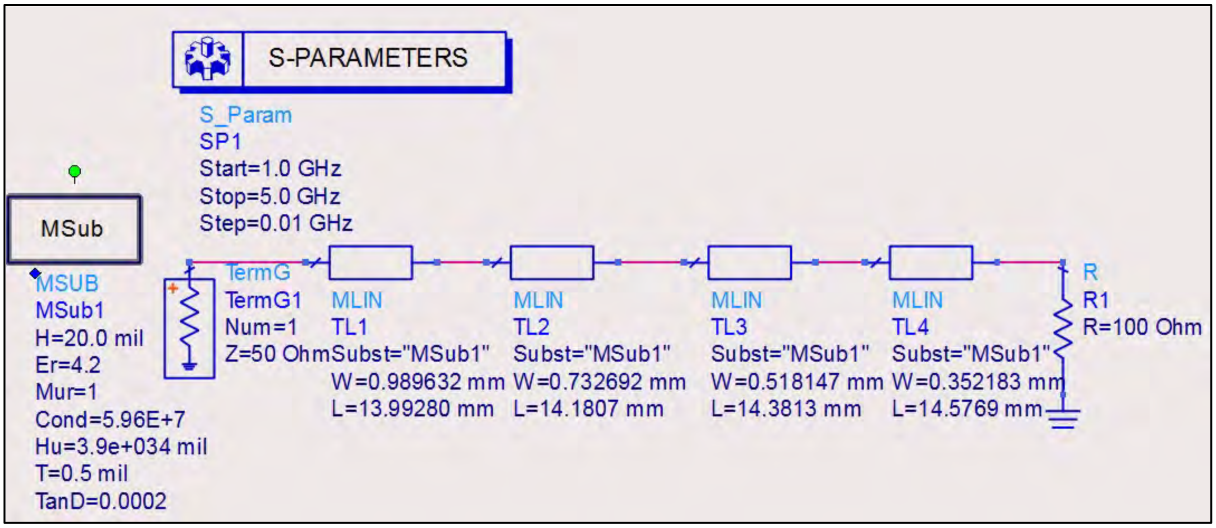


Figure 4-19. A schematic of the microstrip line section widths and lengths.

Simulate the transformer and compare to Problem 3: N=3, 5 sections (Figure 4-20).

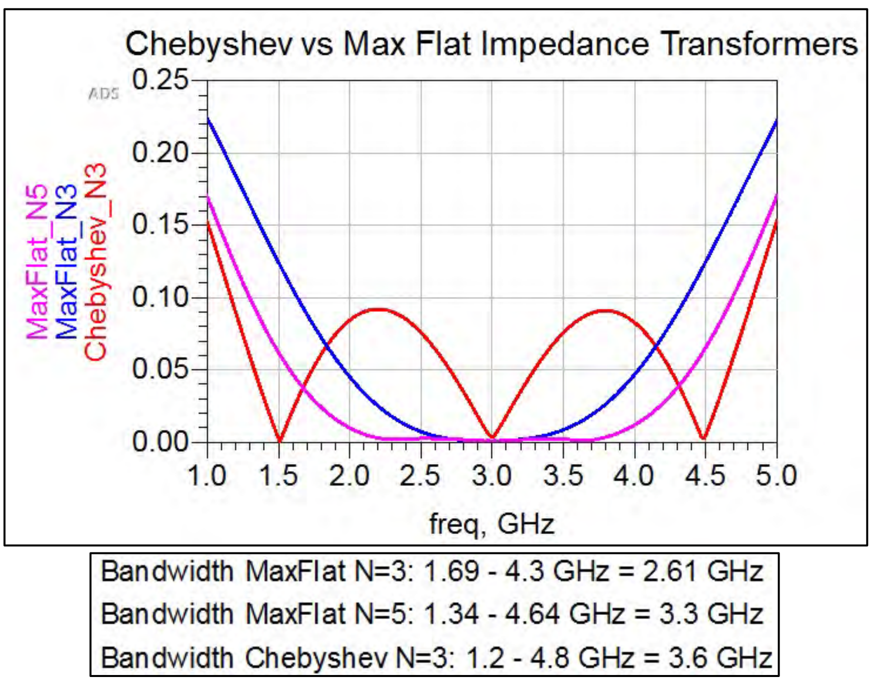


Figure 4-20. Comparison of the simulated transformer to Problem 3: N=3, 5 sections.

Conclusion

While the Chebyshev design does not have the smooth, flat in-band response, it does give a much wider bandwidth than the maximally flat transformer. The three-section Chebyshev even outperforms the five-section maximally flat transformer. The tradeoff of having an in-band ripple is the decreased number of sections of the transformer, thereby reducing the length of the line for fabrication. It is also important to note that overall, the length of the Chebyshev transformer is equal to the length of the maximally flat transformer. Recall that increasing the length of the transformer allows more frequencies to appear matched at a quarter-wavelength. Therefore, the widening of the band using the same overall length is a characterization of the Chebyshev. The design specifications for the transformer about center frequency will dictate which design to use.

Chapter 5 – Microwave Resonators

Problem 1: Lumped Element Resonators

Problem Statement

For the RLC resonating circuit given in Figure 5-1:

a) Graphically find the resonant frequency of the resonator and verify numerically.
b) Determine the unloaded Q factor, Q_0.
c) Determine the external Q factor, Q_e.
d) Use parts b) and c) to find the loaded Q factor, Q_L. Then, find Q_L using a 3-dB bandwidth and compare. Plot the S(1,1) parameter on a Smith chart.
e) Change R to 10 Ohms. What happens to the resonator?

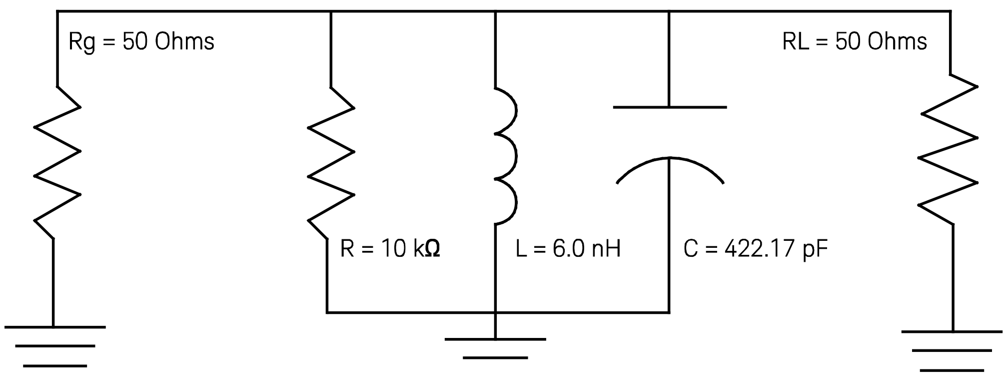


Figure 5-1. An RLC resonating circuit.

Solution

Strategy

Plot the S(2,1) parameter of the resonator to view the frequency response, because Q_L depends on the 3-dB bandwidth of the transmitted signal. Eventually the problem requires the use of a Smith chart, which describes exclusively the reflection coefficient. Here, it will be more intuitive to view the S(1,1) parameter rather than the S(2,1) parameter.

What to expect

Resonant frequency is the frequency when the stored magnetic and electric energies of the circuit are equal. The resonant frequency can be calculated by equating the energies for the inductor and capacitor.

$$W_m = W_e \quad \text{Equation 5-1}$$

$$\frac{1}{4}|I|^2 L = \frac{1}{4}|I|^2 \frac{1}{\omega^2 C} \quad \text{Equation 5-2}$$

$$\omega_o = \frac{1}{\sqrt{LC}} = 2\pi(100\ MHz) \quad \text{Equation 5-3}$$

It is expected that the S(2,1) plot will show that there is a resonant frequency at 100 MHz, but how will it look (Figure 5-2)?

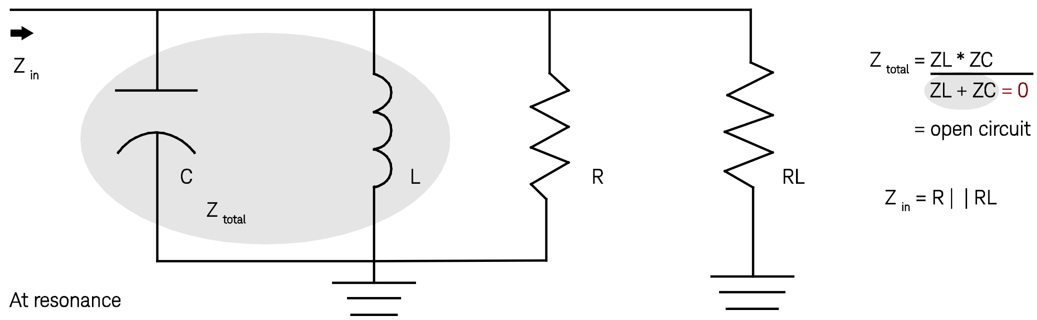


Figure 5-2. The S(2,1) plot.

For the parallel RLC resonator, at resonant frequency, the resonant LC pair act as an open circuit because the sum of the reactance is zero, thereby allowing all of the energy to transmit through to the resistors. Therefore, with a fixed load resistor, the reflection coefficient will depend on the value of R in the resonator circuit. In the specified problem, R is very large, so the S(2,1) parameter will show that the source is matched to the load because they are the same impedance of 50 Ohms.

$$R_{eq} = \left.\frac{RR_L}{R+R_L}\right|_{R \gg RL} = \frac{RR_L}{R} = R_L \qquad \text{Equation 5-4}$$

The S(2,1) plot is then expected to look like a bandpass filter. It will become more intuitive to look at how the impedance changes with frequency. The equivalent impedance of the resonator comes out to:

$$Z_{resonator} = \left[j\omega C + \frac{1}{j\omega L} + \frac{1}{R}\right]^{-1} \qquad \text{Equation 5-5}$$

$$= \frac{\omega RL}{jR(\omega^2 LC - 1) + \omega L}$$

$$= \frac{\omega RL}{jR\left(\left(\frac{\omega}{\omega_0}\right)^2 - 1\right) + \omega L}$$

Case 1: $w << w_o$,

$$Z_{resonator} = \frac{\omega RL}{\omega L - jR} = \frac{R(\omega L)^2 + jR^2(\omega L)}{(\omega L)^2 + R^2}$$

When w is very small, $(wL)^2$ is very small,

$$Z_{resonator} = \frac{R(\omega L)^2 + jR^2(\omega L)}{R^2}$$

$$Z_{resonator} = \frac{(\omega L)^2}{R} + j\omega L = j\omega L \qquad \text{Equation 5-6}$$

Case 2: $w=w_o$,

$$Z_{resonator} = \frac{\omega RL}{jR\left(\left(\frac{\omega}{\omega_0}\right)^2 - 1\right) + \omega L}$$

$$Z_{resonator} = \frac{\omega RL}{\omega L} = R \qquad \text{Equation 5-7}$$

Case 3: $w>>w_o$,

$$Z_{resonator} = \frac{\omega RL}{jR\left(\left(\frac{\omega}{\omega_0}\right)^2 - 1\right) + \omega L}$$

$$Z_{resonator} = \frac{\omega RL}{jR\left(\left(\frac{\omega}{\omega_0}\right)^2\right) + \omega L}$$

$$Z_{resonator} = \frac{\omega RL}{\omega L + jR\left(\left(\frac{\omega}{\omega_0}\right)^2\right)} \cdot \frac{\omega L - jR\left(\left(\frac{\omega}{\omega_0}\right)^2\right)}{\omega L - jR\left(\left(\frac{\omega}{\omega_0}\right)^2\right)}$$

$$Z_{resonator} = \frac{\omega^2 RL^2 - jR^2 L\frac{\omega^3}{\omega_o^2}}{\omega^2 L^2 + R^2\frac{\omega^4}{\omega_o^4}}$$

When w is very large, $\left(\frac{\omega}{\omega_0}\right)^4 >> (\omega L)^2$:

$$Z_{resonator} = \frac{\omega^2 RL^2}{R^2\frac{\omega^4}{\omega_o^4}} - j\frac{R^2 L\frac{\omega^3}{\omega_o^2}}{R^2\frac{\omega^4}{\omega_o^4}}$$

$$Z_{resonator} = \frac{L^2\omega_o^4}{R\omega^2} - j\frac{L\omega_o^2}{\omega}$$

$$Z_{resonator} = \frac{1}{\omega^2 RC^2} - j\frac{1}{\omega C} \qquad \text{Equation 5-8}$$

After analyzing all three cases, for frequencies away from the resonant frequency, there will be reactive components that will promote a mismatch between the generator and load, thus creating a larger reflection coefficient. At low frequencies, the generator will see an inductive reactance, and at high frequencies the generator will see a capacitive reactance. At resonance, the reactive components will cancel one another out and the equivalent impedance will be just the resistor. The large R value, in parallel with the source and load impedances, will cause the 50-Ohm generator to see the 50-Ohm load, resulting in a better resonator. At resonance, only the resistor is seen and a larger shunt resistor means lower energy loss, which is ideal for resonators.

If the value of R changes, so will the previously calculated R_{eq} value, which will change the Q values. This relationship holds for the parallel case, but different circuit topologies of the inductor and capacitor will cause the resonator to look like either an open or short circuit. The reader is encouraged to evaluate the series-series LC network and compare.

Execution

a) Graphically find the resonant frequency of the resonator, and verify numerically.

The schematic is placed into ADS for S-parameter simulation over a reasonable band of frequencies. Double clicking the S-Parameter Simulation component, and going to the Objects tab will allow an output of selected variables (Figure 5-3). This helps maintain a cleaner looking schematic.

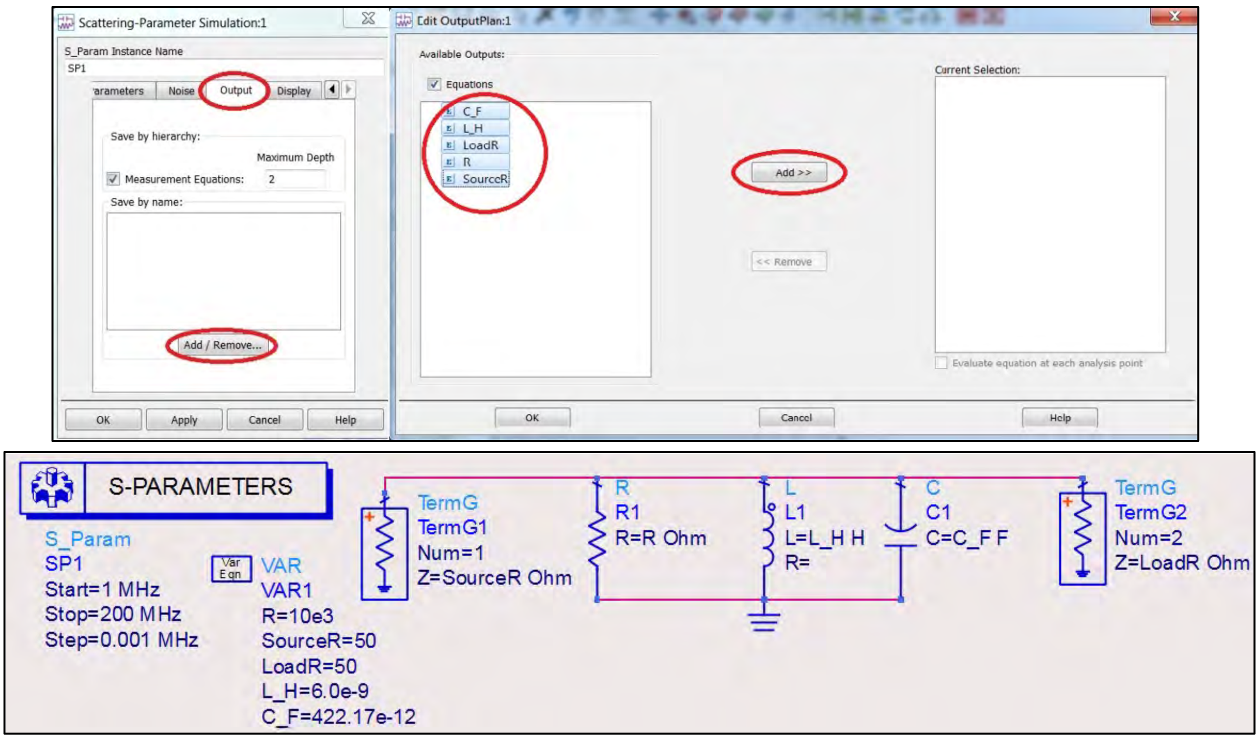


Figure 5-3. Placing the schematic into ADS for S-parameter simulation.

Simulating the circuit, and plotting the S(2,1) parameter yields the frequency response and makes it possible to determine the resonant frequency. The shape and center frequency agree with expectations. At center frequency, the source is seeing just the termination, so the reflection coefficient is at a minimum (Figure 5-4).

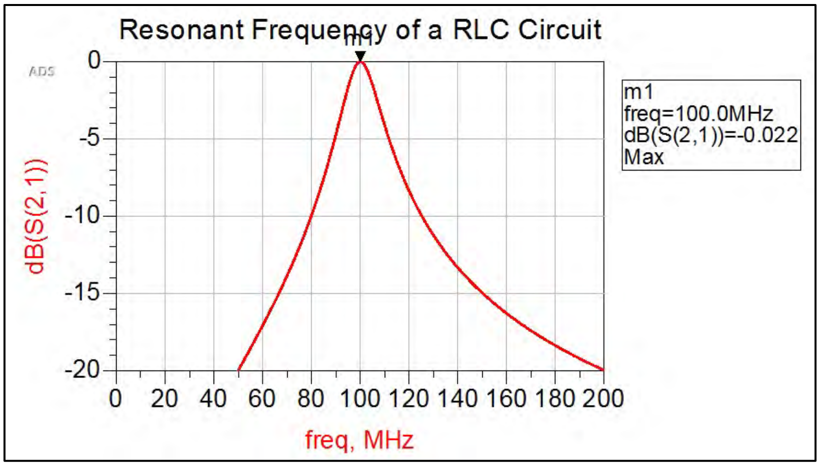


Figure 5-4. At center frequency, the reflection coefficient is at a minimum.

b) Determine the unloaded Q factor, Q_0.

The unloaded Q factor describes the energy ratio of only the resonator itself, and can be found numerically by:

$$Q_0 = \omega_0 RC \qquad \text{Equation 5-9}$$

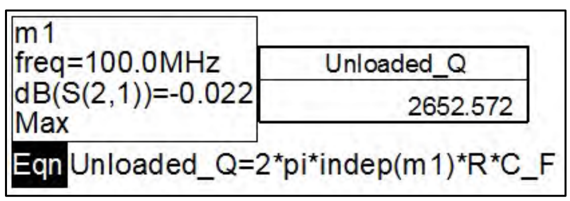


Figure 5-5. Determining the unloaded Q factor.

The resonant frequency, f_o, is extracted from the marker m1 on the plot by calling the independent variable of the m1 data point. The result, as shown in Figure 5-5, is $Q_0 = 2{,}653$.

c) Determine the external Q factor, Q_e.

The external Q factor describes only the energy ratio relating to any external impedance and can be found numerically using Equation 5-10.

$$Q_e = \frac{R_L}{\omega_0 L} \qquad \text{Equation 5-10}$$

The load resistance will include not only the load resistor R_L, but also any other external loads, such as any resistance from the source. Therefore, the equivalent effective resistance, R_{Leq} is the source, resonator and load resistances in parallel (Figure 5-6).

$$Q_e = \frac{R_{Leq}}{\omega_0 L} = \frac{R_g || R_L}{\omega_0 L}$$

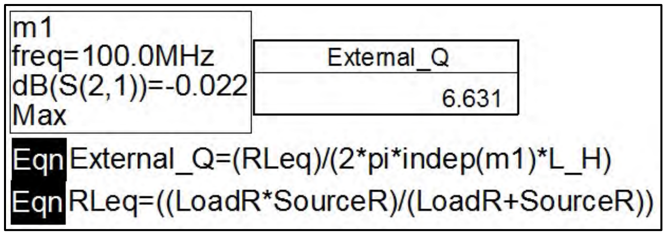


Figure 5-6. Determining the Q_e.

d) Use parts b) and c) to find the loaded Q factor, Q_L. Then, find Q_L using the 3-dB bandwidth and compare.

The loaded Q factor can be found using the following relationship with the other Q factors:

$$\frac{1}{Q_L} = \frac{1}{Q_0} + \frac{1}{Q_e} \qquad \text{Equation 5-11}$$

$$Q_L = 6.61$$

Conversely, Q_L can be found using the 3-dB bandwidth.

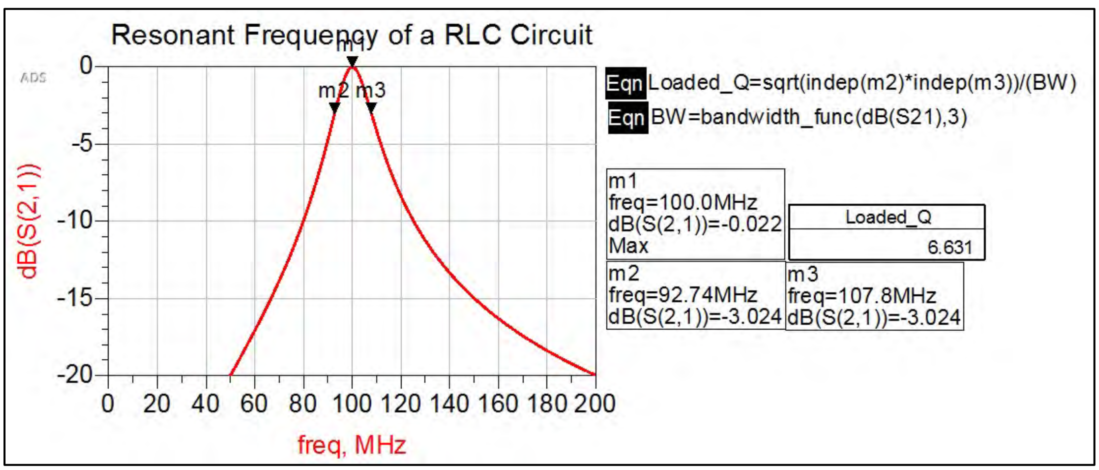


Figure 5-7. Finding Q_L using the 3-dB bandwidth.

The graphical and numerical values for the loaded Q can be considered identical. The geometric mean of the bandwidth is used to determine the actual center frequency of the 3-dB band, instead of using the resonate frequency, even though they are almost identical in this case (Figure 5-7).

The S(1,1) parameter is then plotted on the Smith chart, which shows that at resonant frequency, the generator is well matched to the load, as expected. This is because the input impedance is the large resistor in parallel with 50 Ohms (Figure 5-8). Typing the text box “m4 impedance is @Z_smith_imp” lets the text box value change with the marker. At resonant frequency, the input impedance is almost 50 Ohms, the value of the load resistor, as expected.

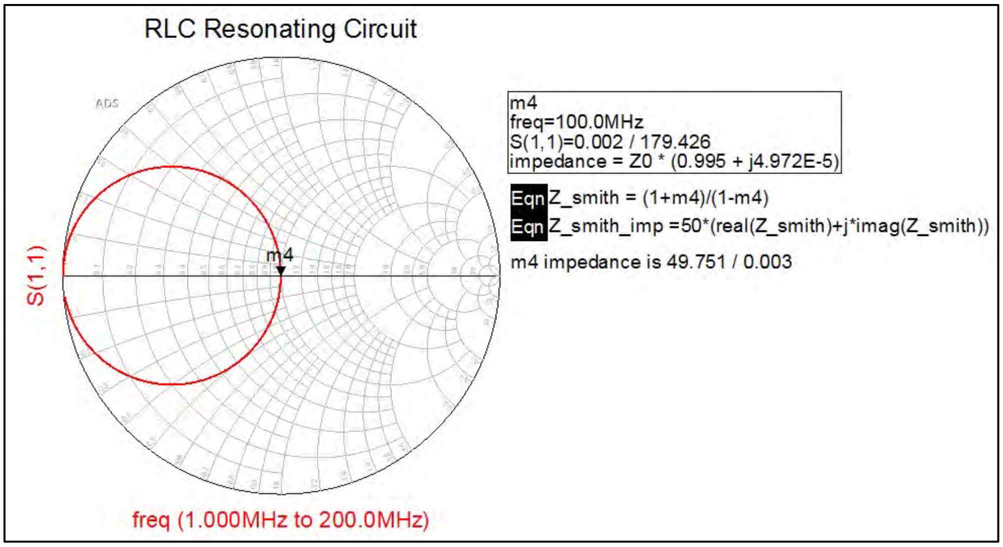


Figure 5-8. Plotting the S(1,1) parameter on a Smith chart.

e) Change R to 10 Ohms. What happens to the resonator?

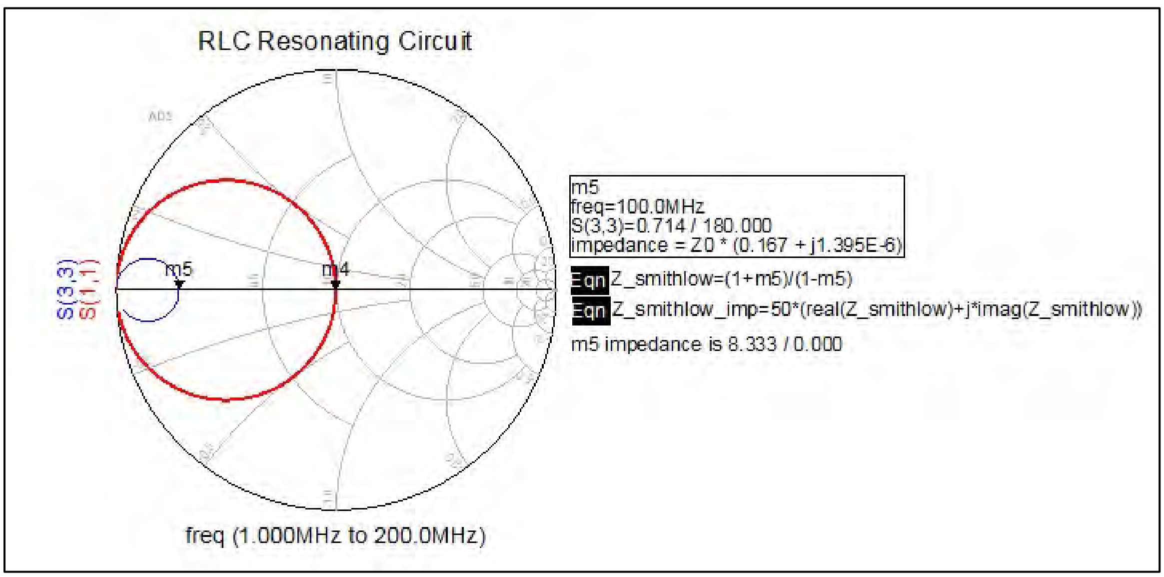


Figure 5-9. Here, R is changed to 10 Ohms.

When the resonator resistor decreases, the unloaded Q factor decreases, causing the loaded Q value to decrease. It is interesting to note that the lower Q value corresponds to a higher reflection coefficient, and the generator no longer sees the 50-Ohm load. Instead, the generator sees an input impedance of 8.3 Ohms (Figure 5-9).

While double checking that the resonant frequency stayed the same, as it should, because only R was changed, the effect of the change on the bandwidth can be seen. The maximum is now more than 10 dB lower and the 3-dB bandwidth is larger. The new loaded Q factor is calculated to be 1.9, significantly lower than the original circuit with R = 10 kOhms (Figure 5-10).

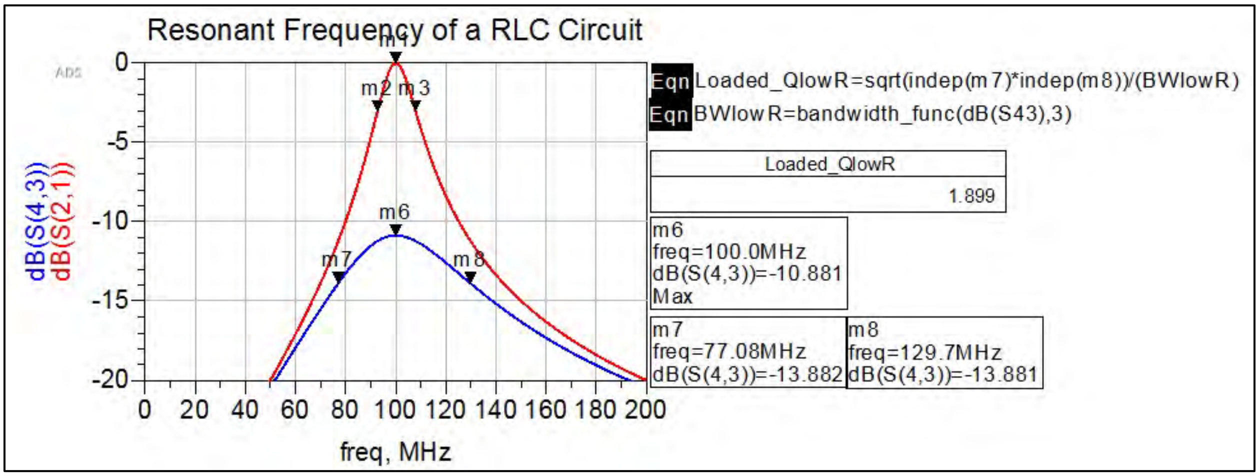


Figure 5-10. Here, the new loaded Q factor is calculated to be 1.9

Conclusion

As expected, the resonant frequency depends on the resonating LC network; however, the resistive component plays a part in the Q value. Keeping all other variables constant, a high internal resonator R value produced a Q value of 6.6, while a low internal R value produced a Q value of 1.9, which is drastically lower. The resonator R value changed the unloaded Q, but the loaded Q value is also related to the external Q. It is important to note that the external Q values will change with respect to the external impedances as well.

This problem computed the three different Q values separately, but the impedance of most real-world circuits will not be as easy to compute as the basic RLC resonator covered in this problem. Sometimes, the resonator may look like a black box. Consequently, it was important to compute the loaded Q using 3-dB bandwidth in ADS.

Problem 2: Transmission Line Resonators

Problem Statement

a) For the lumped-element resonator in Figure 5-11, design the equivalent open-circuited microstrip line resonator. The substrate is FR4 (ϵ_r=4.0, tanδ = 0.0002), with a dielectric thickness of 20 mil, and copper conductor thickness of 1.4 mil.

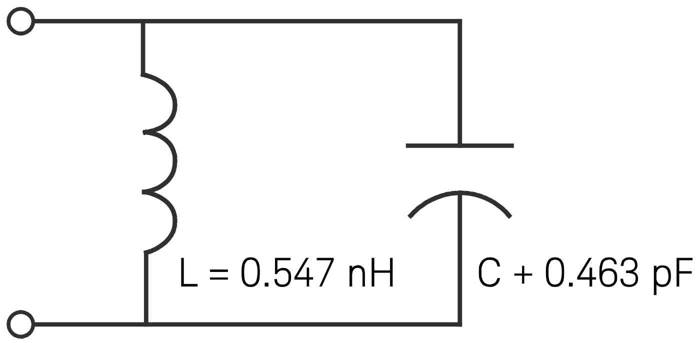


Figure 5-11. The lumped-element resonator for Problem 2.

b) Plot the frequency response of both circuits and compare their Q values.
c) Find the unloaded Q value of the microstrip line resonator.

Solution

Strategy

The strategy will be to convert the lumped-element network into the distributed network by equating the input impedances for a standard open-circuit line and a shunt RLC network. Because most of the physical properties of the microstrip line are given, the variables left to find are the design frequency, characteristic impedance, and length and width. The loaded Q value will be found using the S(2,1) bandwidth, while the unloaded Q value will be determined using a relationship between loaded Q and S(2,1).

What to expect

It is expected that the frequency response of the lumped and distributed circuits will be similar around the resonant frequency, but they will start to diverge away from f_o. The reason distributed-element resonators are used in high frequency rather than lumped elements is because ideal lumped elements are not attainable at microwave frequencies. Therefore, the Q factor for the microstrip line is expected to be better than the lumped-element resonator.

Execution

a) Design the equivalent open-circuited microstrip line resonator.

1. The resonant frequency of the lumped network needs to be found to determine at what frequency the microstrip line should be designed.

$$f_o = \frac{1}{2\pi\sqrt{LC}} = 10\ GHz \qquad \text{Equation 5-12}$$

2. The characteristic impedance of the line needs to be determined. This will be done by equating the input impedance of a parallel RLC resonant circuit with the input impedance of an open circuited transmission line. Because the lumped-element network is a parallel

circuit, the characteristic impedance needs to be in terms of capacitance. A series network would require it in terms of inductance.

Z_{in} of the RLC network:

$$Z_{in,RLC} = \left[j\omega C + \frac{1}{j\omega L} + \frac{1}{R}\right]^{-1} \quad \text{Equation 5-13}$$

Let w = w_o + Δw, where Δw is very small

$$Z_{in,RLC} = \left[j(\omega_o + \Delta\omega)C + \frac{1}{j(\omega_o + \Delta\omega)L} + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[j\omega_o C + j\Delta\omega C + \frac{1}{j\left(1+\frac{\Delta\omega}{\omega_o}\right)\omega_o L} + \frac{1}{R}\right]^{-1} \quad \text{Equation 5-14}$$

Near resonance, the series expansion $\frac{1}{1+x} \approx 1 - x$ can be applied[9],

$$Z_{in,RLC} = \left[j\omega_o C + j\Delta\omega C + \frac{1}{j\omega_o L}\left(1 - \frac{\Delta\omega}{\omega_o}\right) + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[\left(j\omega_o C + \frac{1}{j\omega_o L}\right) + j\Delta\omega C - \frac{1}{j\omega_o L}\left(\frac{\Delta\omega}{\omega_o}\right) + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[\left(\frac{j\omega_o}{\omega_o^2 L} + \frac{1}{j\omega_o L}\right) + j\Delta\omega C - \frac{1}{j\omega_o L}\left(\frac{\Delta\omega}{\omega_o}\right) + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[\left(-\frac{1}{j\omega_o L} + \frac{1}{j\omega_o L}\right) + j\Delta\omega C - \frac{1}{j\omega_o L}\left(\frac{\Delta\omega}{\omega_o}\right) + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[j\Delta\omega C + j\frac{\Delta\omega}{{\omega_o}^2 L} + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[j\Delta\omega C + j\Delta\omega C + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \left[2j\Delta\omega C + \frac{1}{R}\right]^{-1}$$

$$Z_{in,RLC} = \frac{R}{1+2j\Delta\omega RC} \quad \text{Equation 5-15}$$

Z_{in} of an open circuited, lossy transmission line:

$$Z_{in,dist} = Z_0\, coth^{-1}(\alpha + j\beta)l \quad \text{Equation 5-16}$$

$$Z_{in,dist} = Z_0 \frac{1 + j\tan\beta l \tanh\alpha l}{\tanh\alpha l + j\tan\beta l}$$

---

9 *D. Pozar, Microwave Engineering, 4th edition, John Wiley and Sons, New Jersey, 2012.*

$$Z_{in,dist} = Z_0 \frac{1 + j\,tan\left(\frac{\omega}{v_p}\frac{\lambda}{2}\right) tanh\,\alpha l}{tanh\,\alpha l + \ j\,tan\left(\frac{\omega}{v_p}\frac{\lambda}{2}\right)}$$

Again, let w = w_o + Δw, where Δw is very small,

$$Z_{in,dist} = Z_0 \frac{1 + j\,tan\left(\frac{\omega_o + \Delta\omega}{v_p}\frac{\pi v_p}{\omega_o}\right) tanh\,\alpha l}{tanh\,\alpha l + \ j\,tan\left(\frac{\omega_o + \Delta\omega}{v_p}\frac{\pi v_p}{\omega_o}\right)}$$

Also, assume αl is very small, so $tanh\,\alpha l \approx \alpha l$,

$$Z_{in,dist} = Z_0 \frac{1 + j\,tan\left(\frac{\pi\Delta\omega}{\omega_o} + \pi\right) \alpha l}{\alpha l + \ j\,tan\left(\frac{\pi\Delta\omega}{\omega_o} + \pi\right)}$$

$$Z_{in,dist} = Z_0 \frac{1 + j\,tan\left(\frac{\pi\Delta\omega}{\omega_o}\right) \alpha l}{\alpha l + \ j\,tan\left(\frac{\pi\Delta\omega}{\omega_o}\right)}$$

With the assumption that αl and $\frac{\pi\Delta\omega}{\omega_o}$ are very small, $1 \gg j\,tan\left(\frac{\pi\Delta\omega}{\omega_o}\right)\alpha l$ and $tan\left(\frac{\pi\Delta\omega}{\omega_o}\right) = \frac{\pi\Delta\omega}{\omega_o}$

$$Z_{in,dist} = \frac{Z_0}{\alpha l + j\frac{\pi\Delta\omega}{\omega_o}}$$ Equation 5-17

Compare $Z_{in,RLC}$ with $Z_{in,dist}$

$$\frac{R}{1 + 2j\Delta\omega RC} = \frac{Z_0}{\alpha l + j\frac{\pi\Delta\omega}{\omega_o}}$$

$$\frac{R}{1 + 2j\Delta\omega RC} = \frac{Z_0/\alpha l}{1 + \ j\frac{\pi\Delta\omega}{\omega_o \alpha l}}$$

The conclusions made are:

$$R = \ Z_0/\alpha l$$

and

$$2j\Delta\omega RC = \ j\frac{\pi\Delta\omega}{\omega_o \alpha l}$$

$$C \ = \ \frac{\pi}{2Z_0\omega_o}$$

Therefore, the characteristic impedance of the microstrip line will be:

$$Z_0 = \frac{\pi}{2C\omega_o} = 54\ Ohms \qquad \text{Equation 5-18}$$

3. The width and length of the microstrip line need to be calculated. This can be done either by hand, or using the ADS LineCalc tool. LineCalc provides the following dimensions:
   L = 8.601940 mm; W = 0.882385 mm

The final equivalent open-circuited microstrip line resonator is given in Figure 5-12.

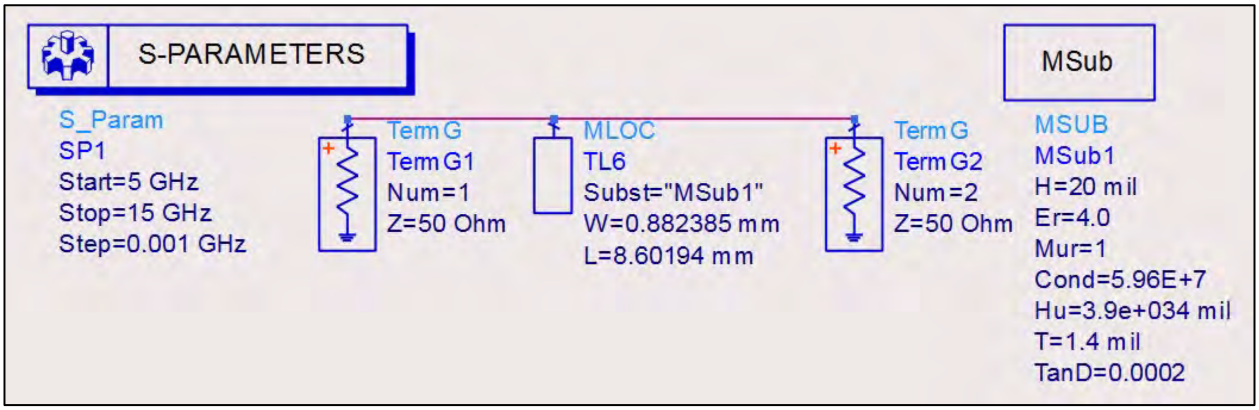


Figure 5-12. The final open-circuited microstrip line resonator calculated using ADS LineCalc.

b) Plot the frequency response of both the circuits and compare their Q values.

The original lumped-element schematic is also created for simulation. The microstrip does have some conductor and substrate resistance that should be modeled by a resistor. However, following computation, the equivalent resistor is almost 13 kOhms, making it negligible in a shunt resonator with a 50-Ohm load. The equation for calculating this resistance is given by:

$$R = \frac{Z_o}{(\alpha_c+\alpha_d)l} \qquad \text{Equation 5-19}$$

Where:

$$\alpha_c = \frac{R_s}{Z_o W}$$

$$\alpha_d = \frac{k_0\varepsilon_r(\varepsilon_{eff}-1)\tan\delta}{2\sqrt{\varepsilon_{eff}}(\varepsilon_r-1)}$$

Because the problem was to compare the given LC circuit with its microstrip equivalent, the schematic in Figure 5-13 is entered into ADS. The user can add the 13-kOhm shunt resistor and determine that the same frequency response is generated, if desired.

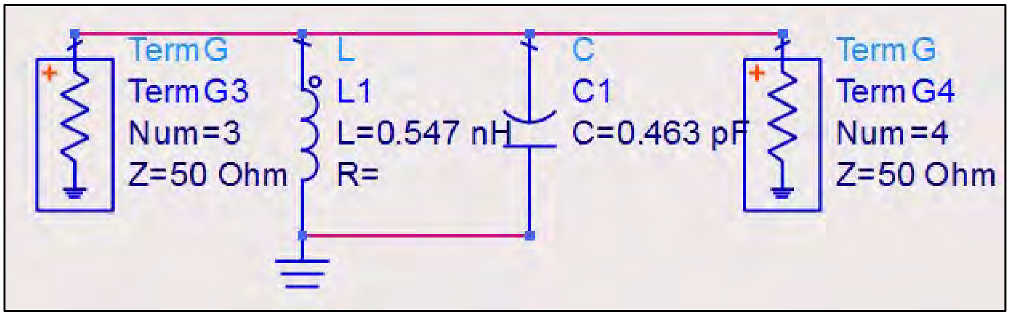


Figure 5-13. A schematic comparing the LC circuit to its microstrip equivalent.

The S(2,1) responses in dB are plotted In Figure 5-14 for comparison.

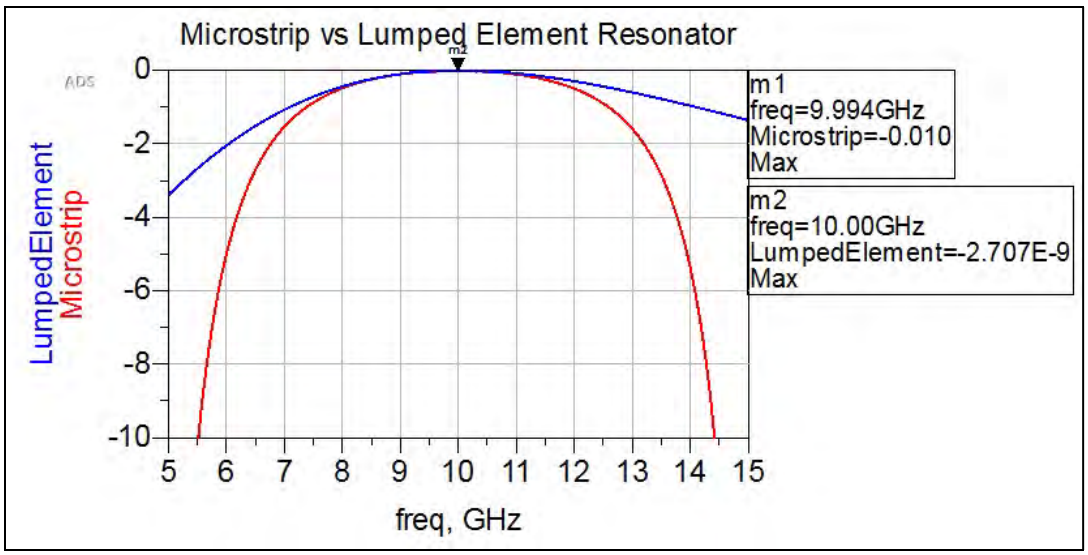


Figure 5-14. A plot of the S(2,1) responses in dB.

While both circuits perform similarly right around the center frequency, the lumped-element frequency response begins to lose its band shape, especially in the higher frequencies. This verifies that distributed elements are more desirable for higher frequencies. Calculating the Q values for each using the 3-dB bandwidth method provides the same notion that distributed elements are better at this resonant frequency. The microstrip line resonator has a Q value almost twice as large as the lumped element (Figure 5-15).

Loaded_Qmicro	Loaded_Qlump
1.306	0.729

Figure 5-15. Table of calculated Q values.

d) Find the unloaded Q value of the microstrip line resonator.

To do this, unloaded Q must be described in terms of S-parameters and loaded Q so that it can be determined. The lumped-element characteristic equations do not apply at higher frequencies.

1. Take the characteristic equation for the Q values and find Q_o in terms of Q_L.

$$\frac{1}{Q_L} = \frac{1}{Q_0} + \frac{1}{Q_e}$$

$$\frac{1}{Q_L} = \frac{1}{Q_0}\left(1 + \frac{Q_0}{Q_e}\right) = \frac{1}{Q_0}(1 + g)$$

$$Q_0 = Q_L(1 + g) \qquad \text{Equation 5-20}$$

2. Determine the value of g using the lumped-element Q values

$$g = \frac{Q_0}{Q_e} = \frac{R/\omega_o L}{R_L/\omega_o L}$$

$$g = \frac{R}{R_L} = \frac{R}{Z_o/2}$$

$$g = 2R/Z_o \qquad \text{Equation 5-21}$$

3. Choose the S(2,1) parameter to be consistent. Find S(2,1) in terms of g.

For a shunt network, the ABCD matrix is given to describe the network (Figure 5-16).

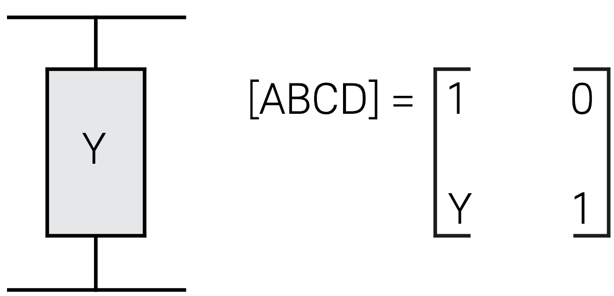


Figure 5-16. The ABCD matrix for a shunt network.

In this case, Y is the equivalent admittance of the line at resonance, which would be 1/R.

The S(2,1) parameter in terms of ABCD parameters is given by:

$$S_{21} = \frac{2}{A + {}^{B}/_{Z_o} + Z_o C + D}$$

$$S_{21} = \frac{2}{1 + 0 + Z_o/R + 1}$$

$$S_{21} = \frac{2}{2 + Z_o/R}$$

$$S_{21} = \frac{{}^{2R}/_{Z_o}}{{}^{2R}/_{Z_o} + 1}$$

$$S_{21} = \frac{g}{1+g} \qquad \text{Equation 5-22}$$

4. Find Q_0 in terms of S(2,1) and Q_L.

$$Q_0 = Q_L(1 + g)$$

$$Q_0 = Q_L\left(1 + \frac{S_{21}}{1-S_{21}}\right) \qquad \text{Equation 5-23}$$

5. Solve for Q_0.

Using the magnitude value for S(2,1), the unloaded Q value was found easily in the Data Display of ADS (Figure 5-17).

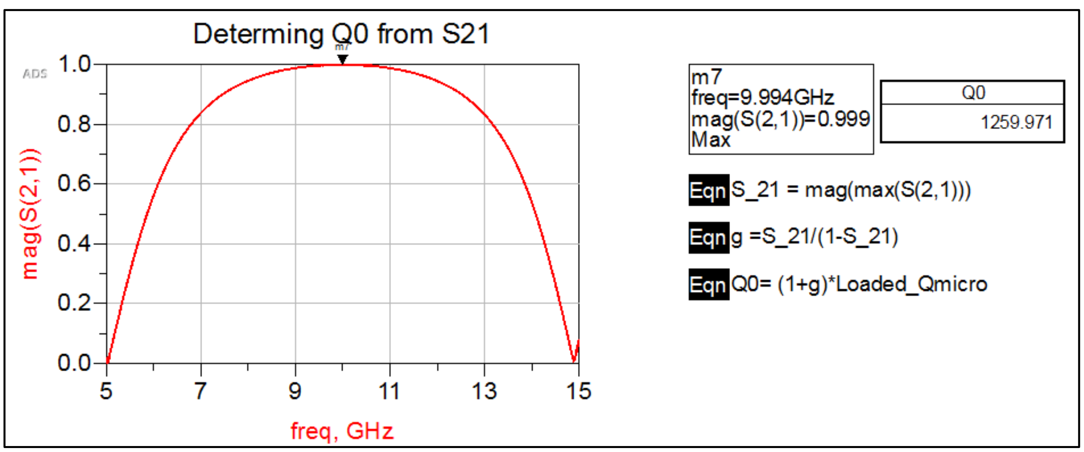


Figure 5-17. The ADS Data Display of the unloaded Q value.

This result makes sense, because the actual resonator is an ideal LC circuit, so the unloaded Q should be large. The external Q must then be closer to the loaded Q value, which also makes sense when taking into account the input impedance surrounding the resonant frequency.

Conclusion

Microstrip line resonators are used in high frequency rather than lumped elements because lumped-element inductors or capacitors suffer from parasitic effects at high frequencies. This was made apparent when comparing the Q factor between the lumped and distributed resonators. Microstrip resonating circuits can be open or a shorted section of line. Similar to the open-circuit stubs, the open-circuit resonator is used for fabrication convenience.

This exercise focused on finding the Q values of a resonator using the S(2,1) parameter, which is important and useful for microwave resonators. Microwave resonators are used in not only filters, as was shown in this chapter, but also in amplifiers and oscillators. The oscillator is an important circuit because it generates a signal with a specific frequency that can be used for signal modulation. The quality factor of these oscillators determines the length of time an oscillation lasts. The lower the Q value, the quicker the oscillation is dampened and eventually dies down.

Chapter 6 – Power Dividers and Directional Couplers

Problem 1: T- Junction Power Divider

Problem Statement

A T-junction power divider has a Port 1 characteristic impedance of 50 Ohms.

a) Determine the impedances of Port 2 and 3 if they have a power ratio of 3:1, respectively.
b) What are the reflection coefficients looking into each port?
c) Simulate the T-junction power divider using an ideal transmission line and measure the power seen at Ports 2 and 3, given an operating frequency of 1 GHz.
d) Simulate the power divider using a microstrip line with reasonable losses and compare it to part c.

Solution

Strategy

To determine the impedances at each port, the equations for power going into each port will be used. Once the impedances are known, the reflection coefficients can be easily calculated by looking at the input impedance at the junction. For example, looking into the junction at Port 2 yields an input impedance of $Z_1||Z_3$.

The T-junction power divider is best analyzed using S-parameters, specifically the S(2,1) parameter. The S(2,1) parameter in dB provides the power ratio for the amount of power at Port 2 due to a signal at Port 1. To properly compute the S-parameters, the termination references must be identical, which creates unintended reflections due to an impedance mismatch. Therefore, quarter-wave matching sections will be utilized to remove the unintended reflections at terminations. To keep the exercise simple, quarter-wave sections will be used for Ports 1, 2 and 3. The ADS LineCalc tool will be used to determine the length and width of the microstrip line sections.

What to expect

For the ideal calculations in parts a-c, it is expected that the results will show the 3:1 power ratio. Port 2 will have 75% of the power from Port 1, while Port 3 will have 25%. Due to the relationship between power and impedance:

$$P = \frac{1}{2}\frac{V^2}{Z} \qquad \text{Equation 6-1}$$

Port 2 is expected to have lower impedance than Port 3, so more power will divert into Port 2.

The simulated ideal transmission line power divider is expected to yield the expected results of a 3:1 power divider at the designed frequency. Because the distributed power divider is frequency dependent, the power divider will behave differently outside the operating frequency due to reflections at the junction.

Execution

a) Determine the impedances of Port 2 and 3, assuming they have a power ratio of 3:1, respectively.

The T-junction power divider is represented by Figure 6-1.

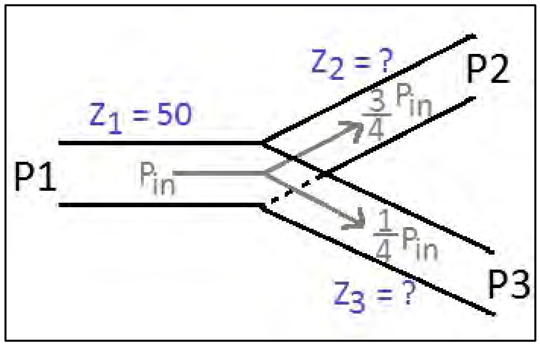


Figure 6-1. The schematic for the T-junction power divider in Problem 1.

To determine the impedance of each section, the ratio of power at each port needs to be analyzed. The input power sent from Port 1 is defined as:

$$P_{in} = \frac{1}{2}\frac{V_1^2}{Z_1} \quad \text{Equation 6-2}$$

The output power seen at Port 2 will then be:

$$P_{out,2} = \frac{1}{2}\frac{V_1^2}{Z_2} = \frac{3}{4}P_{in} \quad \text{Equation 6-3}$$

Solving for Z_2 at Port 2:

$$\frac{1}{2}\frac{V_1^2}{Z_2} = \frac{3}{4}P_{in} = \frac{3}{4}\frac{1}{2}\frac{V_1^2}{Z_1}$$

$$Z_2 = \frac{4}{3}Z_1 = 66.67\ \Omega$$

The output power seen at Port 3 is:

$$P_{out,3} = \frac{1}{2}\frac{V_1^2}{Z_3} = \frac{1}{4}P_{in} \quad \text{Equation 6-4}$$

Solving for Z_3 at Port 3:

$$\frac{1}{2}\frac{V_1^2}{Z_3} = \frac{1}{4}P_{in} = \frac{1}{4}\frac{1}{2}\frac{V_1^2}{Z_1}$$

$$Z_3 = 4Z_1 = 200\ \Omega$$

b) What are the reflection coefficients looking into each port?

At Port 1 looking into the junction, the impedances of Port 2 and 3 are seen in parallel.

$$Z_{in,P1} = Z_2 || Z_3 \qquad \text{Equation 6-5}$$

$$Z_{in,P1} = \left(\frac{1}{66.67} + \frac{1}{200}\right)^{-1} = 50\ \Omega$$

Port 1 is matched to the Port 2 and Port 3 junction. The reflection coefficient $\Gamma_1 = 0$.
At Port 2 looking into the junction, the impedances of Port 1 and 3 are seen in parallel.

$$Z_{in,P2} = Z_1 || Z_3 \qquad \text{Equation 6-6}$$

$$Z_{in,P2} = \left(\frac{1}{50} + \frac{1}{200}\right)^{-1} = 40\ \Omega$$

Port 2 is not matched to the junction. The reflection coefficient is:

$$\Gamma_2 = \frac{40 - 66.67}{40 + 66.67} = -0.25$$

At Port 3 looking into the junction, the impedances of Port 1 and 2 are seen in parallel.

$$Z_{in,P3} = Z_1 || Z_2 \qquad \text{Equation 6-7}$$

$$Z_{in,P3} = \left(\frac{1}{50} + \frac{1}{66.67}\right)^{-1} = 28.57\ \Omega$$

Port 3 is also not matched to the junction. The reflection coefficient is:

$$\Gamma_3 = \frac{28.57 - 200}{28.57 + 200} = -0.75$$

c) Simulate the T-junction power divider using an ideal transmission line and measure the power seen at Ports 2 and 3.

To simulate the power divider, three sections of ideal transmission line are used to represent each port. For power measurement, S-parameter simulation is used for easy analysis at each port. However, this requires identical terminations at each port, which causes an unintended reflection of power at Ports 2 and 3, and produces an incorrect measurement. In order to accurately measure the power received at Ports 2 and 3, the respective transmission lines need to be matched to the termination loads. This is done using a simple quarter-wave matching section. The finished ADS schematic for the ideal transmission line is provided in Figure 6-2.

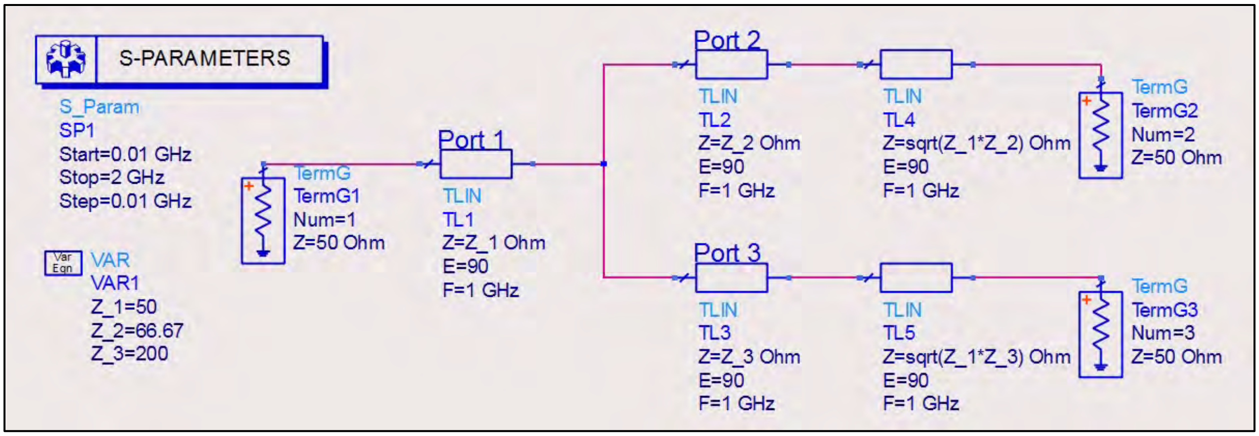


Figure 6-2. The finished ADS schematic for the ideal transmission line.

The power at each port is plotted in the Data Display using the dB S(2,1) for Port 2, and the dB S(3,1) parameter for Port 3 (Figure 6-3).

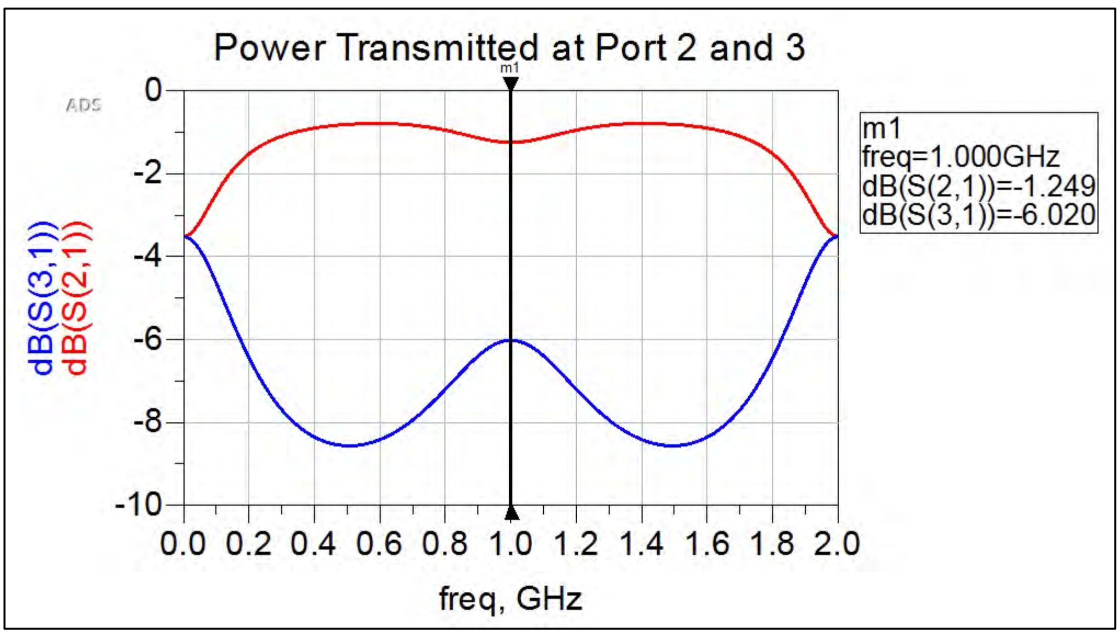


Figure 6-3. The power at each port plotted in the ADS Data Display.

The plot, while it does not provide the expected answer, demonstrates the frequency dependence of the transmission line. In a design at 1 GHz, the quarter-wave matching section allows the T-junction power divider to behave ideally. Outside of this operating frequency, the reflections occurring at the junction cause a loss in power received at the ports. It is most noticeable at Port 3, which has a characteristic impedance of four times Port 1 and a reflection coefficient of -0.75. The plot shows the power gain at each port with respect to Port 1. This is normally how the power is plotted, but we are interested in the ratio of power split between Port 2 and Port 3. To back out the power ratio, the following equations are used in the Data Display. Power ratios are expressed as:

$$10\, log_{10}(^{b}/_{a}) \qquad \text{Equation 6-8}$$

Voltage ratios are expressed as:

$$20\, log_{10}(^{b}/_{a}) \qquad \text{Equation 6-9}$$

The power split between Port 2 and 3 are the expected values of ¾ and ¼ respectively (Figure 6-4).

powerP2_ratio	powerP3_ratio
0.750	0.250

Eqn powerP2_ratio = 10**(-1.249/10)

Eqn powerP3_ratio=10**(-6.021/10)

Figure 6-4. This table shows the power split between Port 2 and 3.

It will also be interesting to see how the reflection coefficient changes with respect to frequency. Plotting $\Gamma_1, \Gamma_2, and\ \Gamma_3$ on a Smith chart provides the plot shown in Figure 6-5. The values calculated in part b agree with the simulated reflection coefficients. They are also mostly resistive with little reactive components.

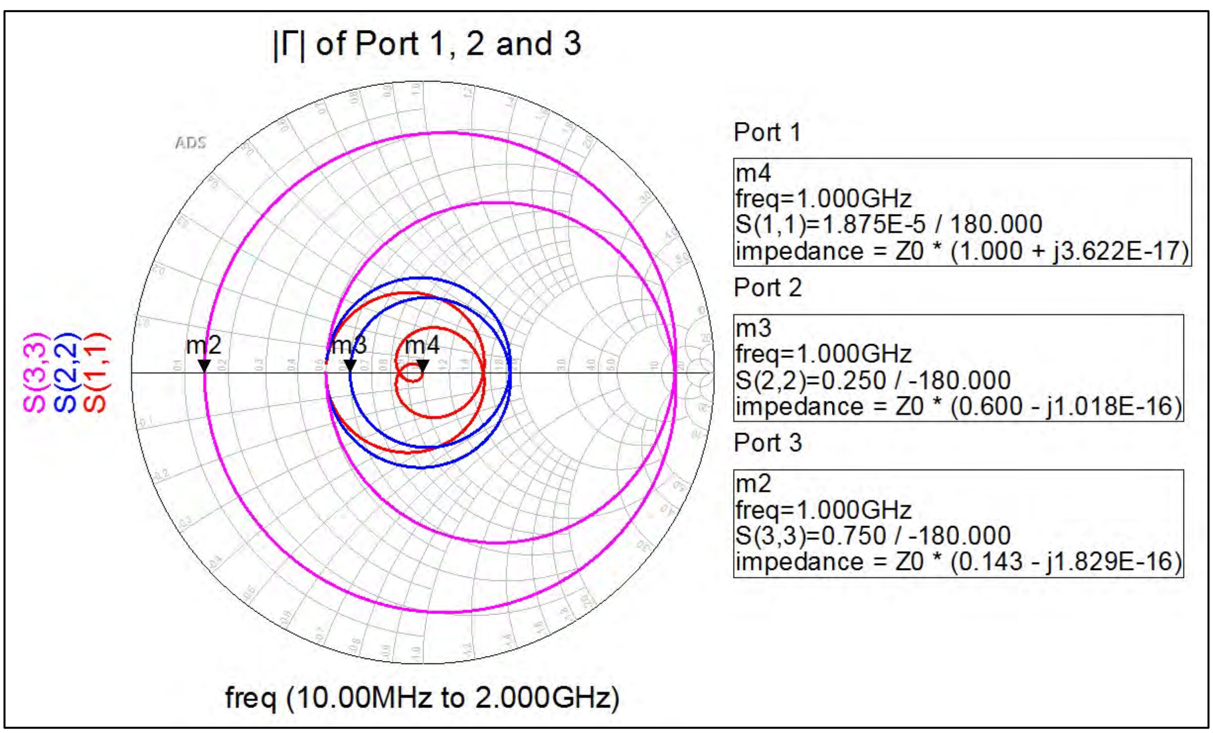


Figure 6-5. A plot of $\Gamma_1, \Gamma_2, and\ \Gamma_3$ on a Smith chart.

d) Simulate the power divider using a microstrip line with reasonable losses and compare it to part c.

The same type of microstrip line used in previous problems will be used here. For reasonable losses, the conductor used is copper and the dielectric loss, tanδ, will be 0.0002. Assume 90 degree length lines, as with the transmission line simulation. The width and length for each port is found using the ADS LineCalc Tool. Again, quarter-wave transforms are used to connect the ports to the S-parameter terminations (Figure 6-6).

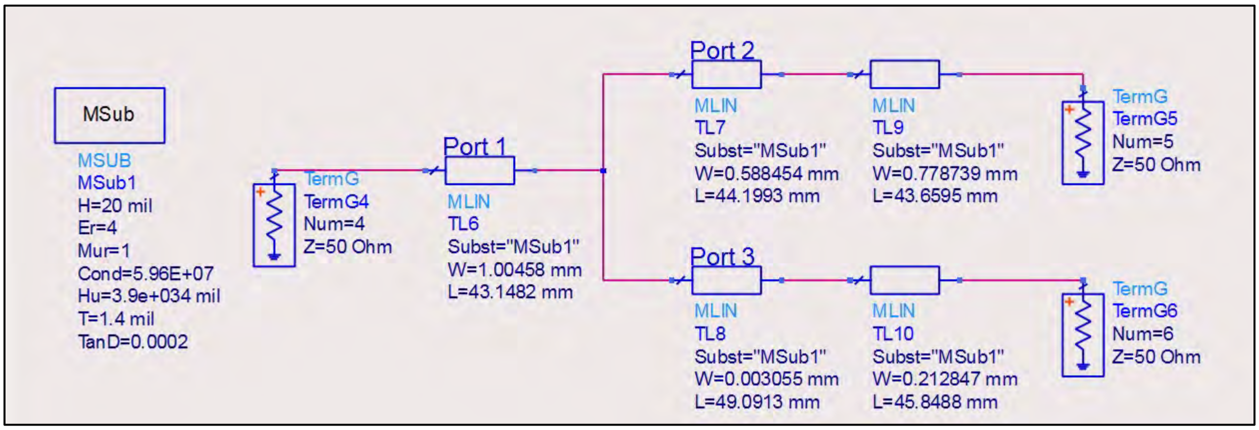


Figure 6-6. Shown here is a schematic of the microstrip line used for this example, with quarter-wave transforms used to connect the ports to the S-parameter terminations.

The power at each port is plotted in the Data Display using the dB S(5,4) for Port 2, and the dB S(6,4) parameter for Port 3 (Figure 6-7). The power ratio between the two ports is also extracted, similar to part c.

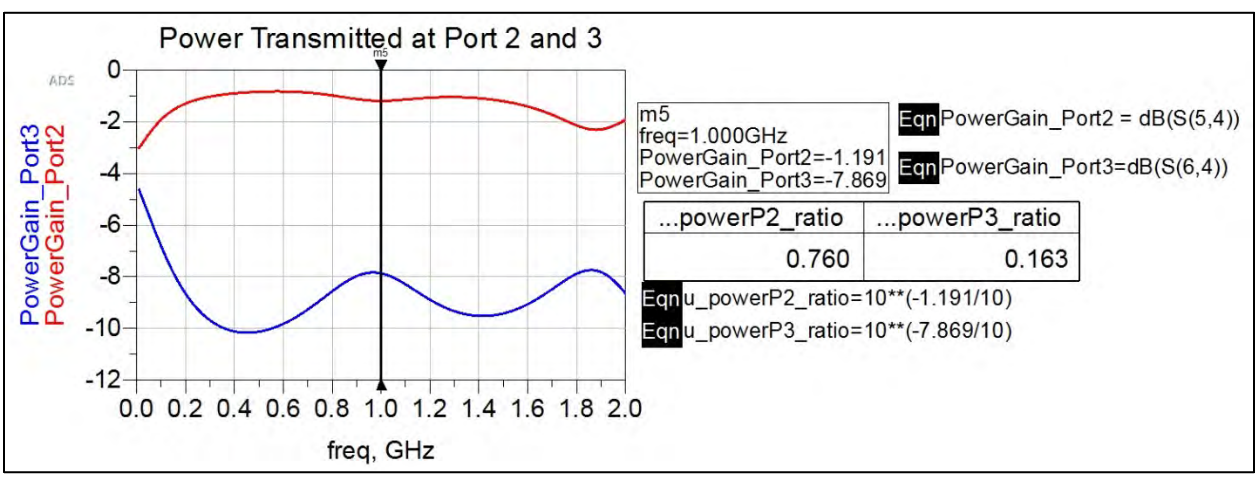


Figure 6-7. Plotted here is the power transmitted at Port 2 and Port 3.

This time, the losses at the design frequency, as well as over the simulated frequency range, are noticeable. Port 2 actually receives a bit more power than designed, mostly because of the large impedance at Port 3. The power received went from an ideal 25% to 16.3%. Why so much loss compared to the ideal transmission line? A plot of the reflection coefficients may help determine the cause (Figure 6-8).

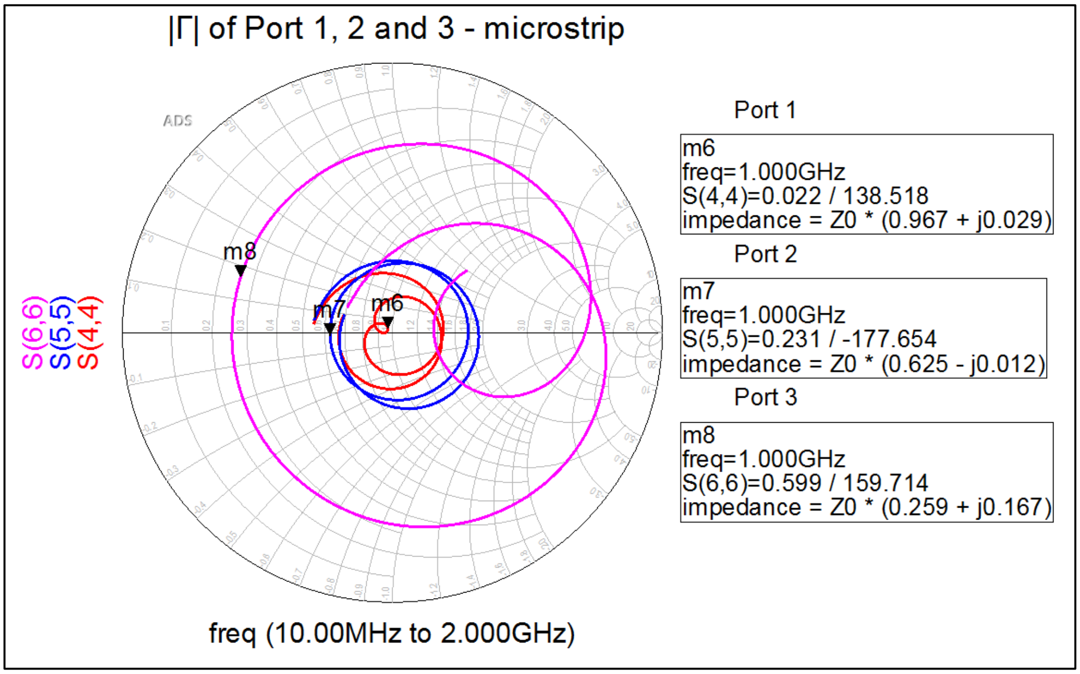


Figure 6-8. A plot of the reflection coefficients.

The reflection coefficient for Port 3 now has a large reactive part. The physical setup of the T-junction power divider causes multiple discontinuities relating to currents, surface waves, radiation, and scattering parameters at the junctions[10]. To properly account for this behavior, a higher level electromagnetics course using the Method of Moments must be taken. The punch line is that the bends and turns within a distributed structure affect its characteristic behavior, unlike lumped-element networks. The ideal transmission line ignores the losses associated with these bends and turns, which is why the microstrip line model provides a more realistic T-junction power divider.

Conclusion

While the ideal models of the T-junction divider provide favorable results, the realistic microstrip line model reveals the lossy nature of the physical T-junction structure. The abrupt bends in the structure create discontinuities that prevent the ideal split of power to occur. Extra pieces of line to smooth out the transition between the ports will be needed to minimize the discontinuities. Also, the large impedance differences between the ports cause more power to divide into the

[10] S. Wu, H. Yang, N.G. Alexopoulos and I. Wolff, "A Rigorous Dispersive Characterization of Microstrip Cross and T Junctions," IEEE Trans. Microwave Theory and Techniques, vol. 38, No. 12, Dec. 1990.

lower impedance port. Therefore, if a T-junction power divider is to be used, it may be best to have an equal split power divider.

Problem 2: Wilkinson Power Divider

Problem Statement

Design a 3 port Wilkinson power divider with a 3:1 power ratio, similar to the T-junction power divider in Problem 1. Simulate the design using both an ideal transmission and microstrip line, and compare the result to the T-junction power divider simulations. Assume an operating frequency of 1 GHz, and microstrip line parameters identical to Problem 1.

Solution

Strategy

Most texts focus on the equal-split Wilkinson power divider, but often they do not expand to any arbitrary power division design. Using the power ratio design technique presented in the paper by Li and Wang[11] , the 3:1 Wilkinson power divider will be designed and simulated. Using ADS, the power divider designed with both an ideal transmission line and microstrip line can be realized and compared to the T-junction divider.

What to expect

The Wilkinson divider is similar to the T-junction divider, with the addition of a shunt resistor between Ports 2 and 3. This shunt resistor allows the network to appear lossless when the output ports are matched, and the reflected power is dissipated into the resistor. The general diagram for the Wilkinson power divider is given in Figure 6-9.

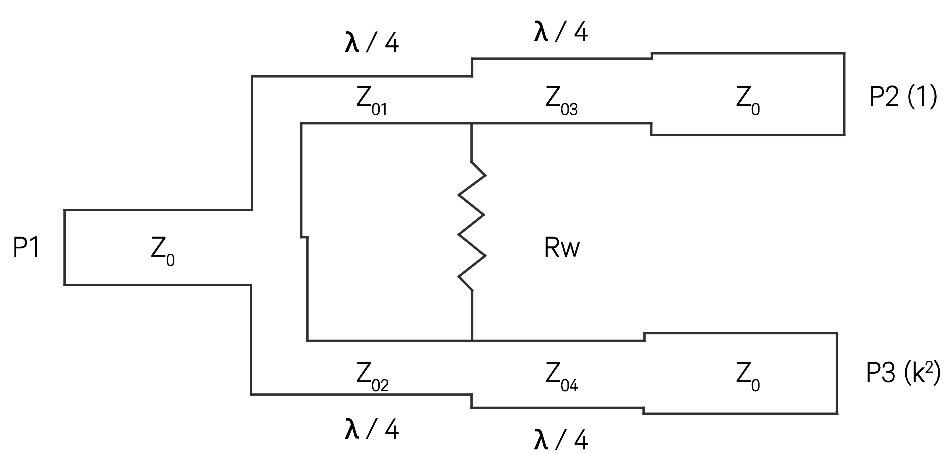


Figure 6-9. A general diagram for the Wilkinson power divider.

[11] *Jia-Lin Li and Bing-Zhong Wang, "Novel Design of Wilkinson Power Dividers with Arbitrary Power Division Ratios," IEEE Trans. Industrial Electronics, vol. 58, No. 6, June 2011.*

It is expected that the dissipation of the reflected power will provide better results for the realistic microstrip line power divider than the T-junction divider.

Execution

The technique for solving for the impedances and shunt resistor for the arbitrary power divider with a power division ratio of 1:k^2 are as follows:

$$Z_{01} = Z_0\sqrt{k(1+k^2)} \qquad \text{Equation 6-10}$$

$$Z_{02} = Z_0\sqrt{\frac{1+k^2}{k^3}} \qquad \text{Equation 6-11}$$

$$Z_{03} = Z_0\sqrt{k} \qquad \text{Equation 6-12}$$

$$Z_{04} = \frac{Z_0}{\sqrt{k}} \qquad \text{Equation 6-13}$$

$$R_W = Z_0\frac{1+k^2}{k} \qquad \text{Equation 6-14}$$

For our 1:3 ratio, this yields a k value of √3. The line impedance values are:

$$Z_{01} = 50\sqrt{\sqrt{3}(1+3)} = 131.607$$

$$Z_{02} = 50\sqrt{\frac{1+3}{3\sqrt{3}}} = 43.8691$$

$$Z_{03} = 50\sqrt[2]{3} = 65.8037$$

$$Z_{04} = \frac{50}{\sqrt[2]{3}} = 37.9918$$

$$R_W = 50\frac{1+3}{\sqrt{3}} = 115.47$$

The final ADS schematic for the ideal transmission line is shown in Figure 6-10.

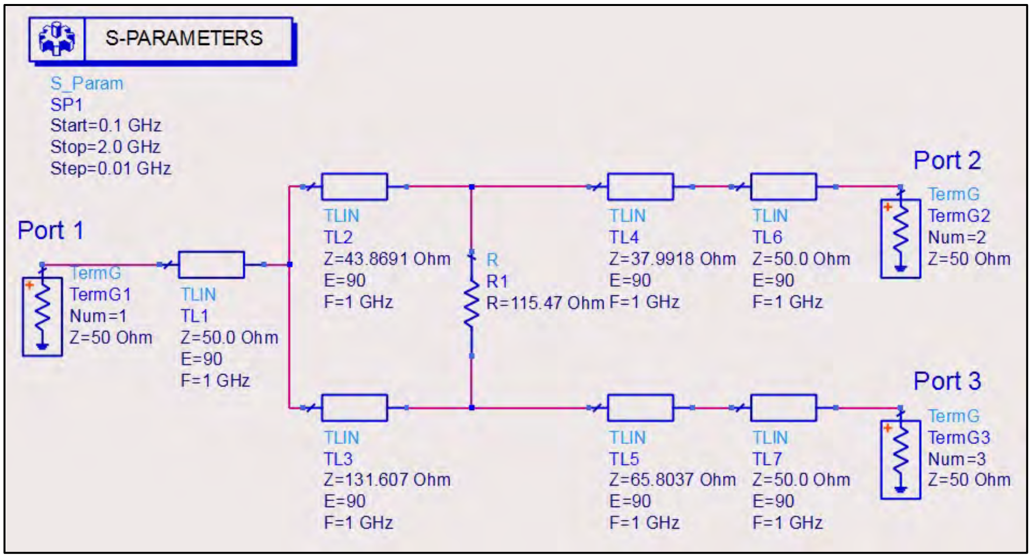
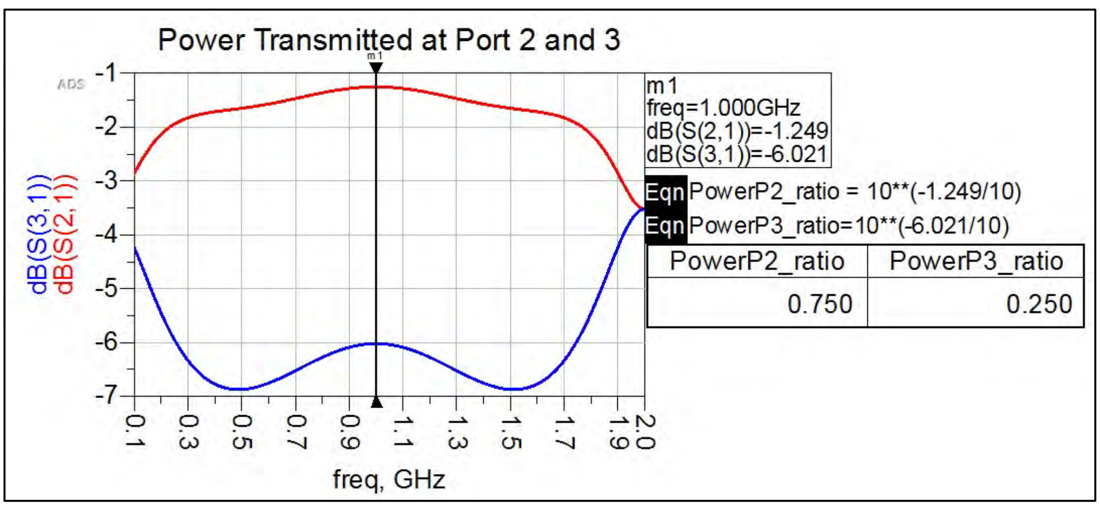


Figure 6-10. Shown here is the final ADS schematic for the ideal transmission line.

In the Data Display, the power division at both ports can be analyzed to make sure the ideal power division was accomplished (Figure 6-11).



PowerP2_ratio	PowerP3_ratio
0.750	0.250

Figure 6-11. Here, the power division at both ports is analyzed.

At the operating frequency of 1 GHz, the power division is perfect, as expected, since this is an ideal transmission line. The results can be compared to that of the T-junction power divider (Figure 6-12).

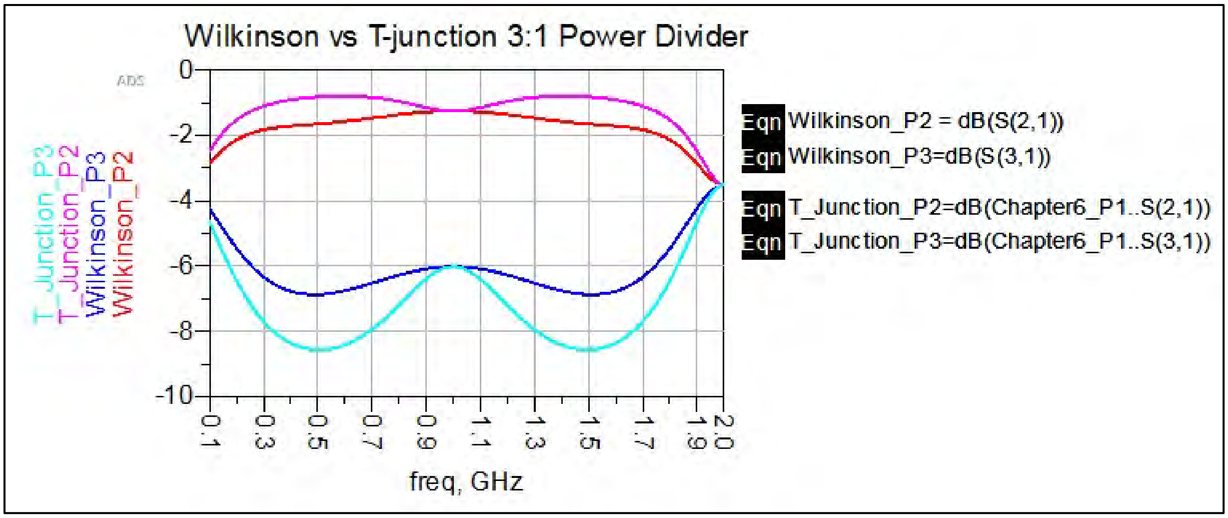


Figure 6-12. Shown here is a comparison of a Wilkinson versus a T-junction 3:1 power divider.

At the operating frequency of 1 GHz, the two power dividers behave identically. Outside 1 GHz is where the effects of the shunt resistor can be seen. The Wilkinson power divider stays closer to the desired 3:1 ratio than the T-junction power divider because the reflected power is dissipated into the shunt resistor.

Ideally, the Wilkinson performs better, but what about realistically with a microstrip line? The Wilkinson divider in the microstrip line realization is given in Figure 6-13. The physical dimensions of the lines were found using the ADS LineCalc Tool.

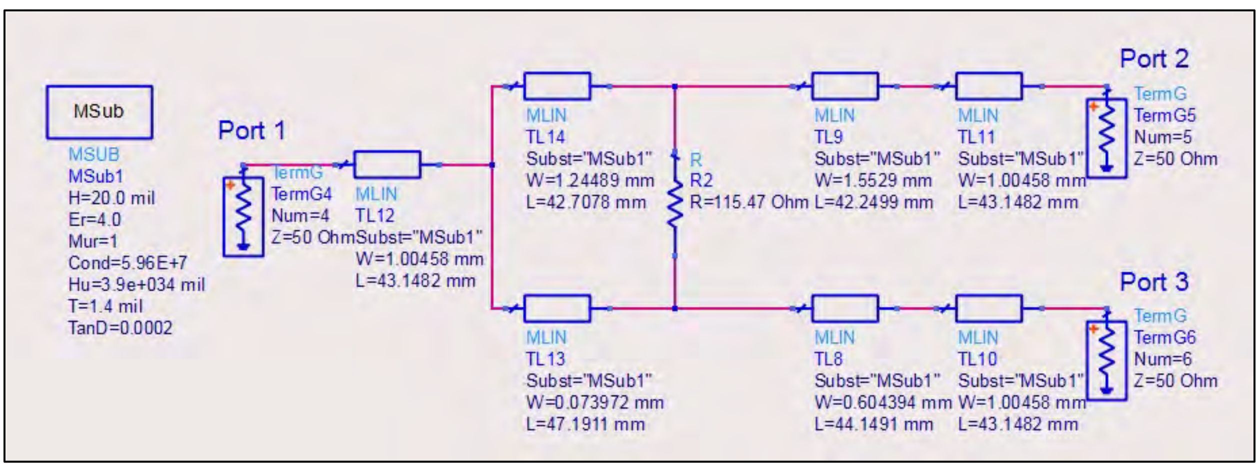


Figure 6-13. Shown here is the Wilkinson divider in the microstrip line realization.

In the Data Display, the power division at both ports can be compared to that of the T-junction power divider (Figure 6-14).

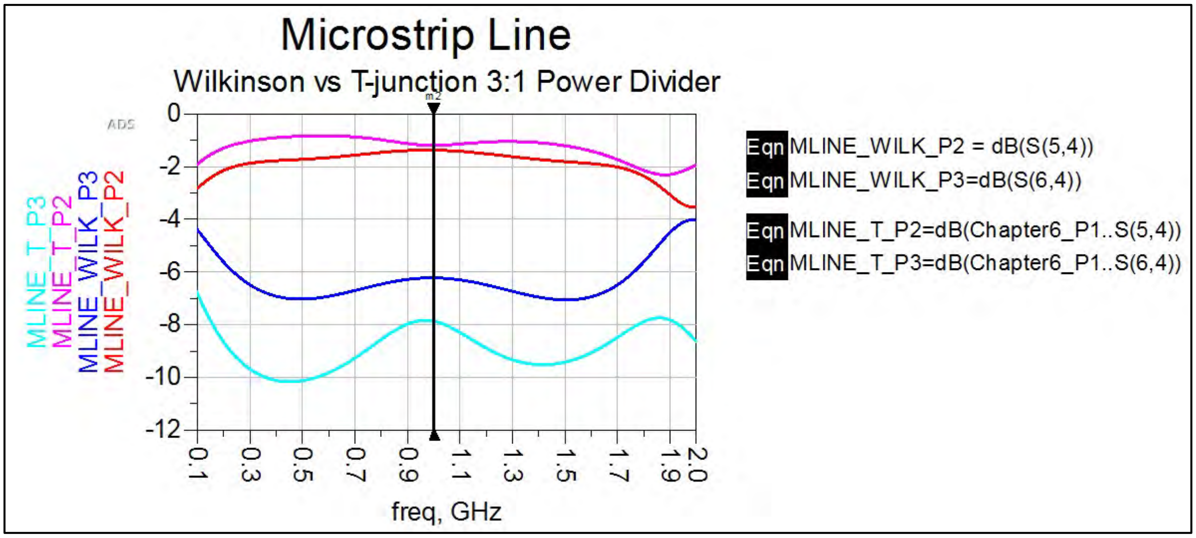


Figure 6-14. Here, the power division at both ports can be compared to that of the T-junction power divider.

Again, the Wilkinson divider seems to hold the 3:1 ratio better over a larger frequency range. The power division for the Wilkinson and T-junction power divider at the operating frequency is compared in the Data Display by backing out the power ratio from the S-parameters. The Wilkinson divider proves to be the most favorable design for the microstrip realization. It has a 3.056:1 ratio, while the T-junction has a 4.654:1 ratio (Figure 6-15).

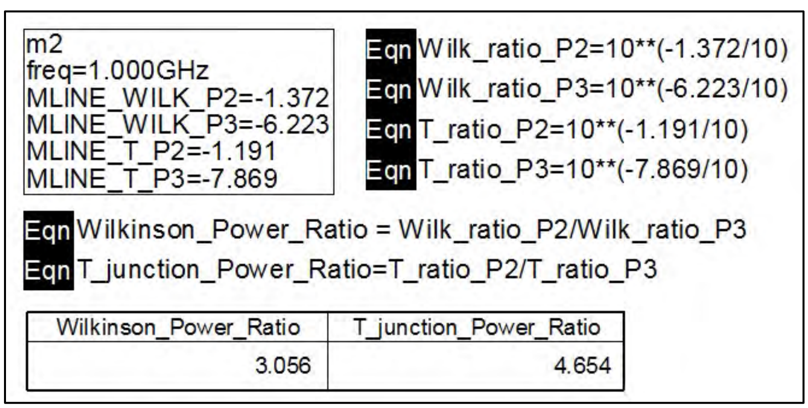


Figure 6-15. Given the ratios, the Wilkinson divider proves to be the most favorable design for the microstrip realization.

Conclusion

The shunt resistor in the Wilkinson power divider helps dissipate the reflected power that occurs from the bends and junctions in the microstrip line. Therefore, it performs better than the T-junction power divider for the operating frequency of 1 GHz. So why does the T-junction power divider exist if it is outperformed by the Wilkinson? It really depends on the design specifications. It may be more desirable to have the maximum power transfer only at the center frequency. Also, the Wilkinson has an extra stage of line after the resistor that takes up more space. However, designs are now being pushed to perform faster at higher frequencies in smaller packages, so while the Wilkinson divider is the champion in this example, it may not always be, depending on the design specifications.

Problem 3: Directional Coupler

Problem Statement

Design a single-section, coupled-line 15-dB directional coupler connected to 50-Ohm ports. Compare the ideal transmission line and microstrip line models. Use a quarter-wavelength coupling section with a center frequency of 2.4 GHz.

a) Plot the insertion loss.
b) Plot the coupling.
c) Plot the isolation.
d) Plot the directivity.

Solution

Strategy

Use the coupling specification of the coupler to determine the even and odd line impedances of the coupled-line section. The line impedances are calculated using the following equations:

$$Z_{0e} = Z_0\sqrt{\frac{1+C}{1-C}} \quad \text{Equation 6-15}$$

$$Z_{0o} = Z_0\sqrt{\frac{1-C}{1+C}} \quad \text{Equation 6-16}$$

where C is the coupling factor in decimal form.

Using ADS and the LineCalc Tool, the ideal transmission and microstrip line realizations will be simulated and analyzed.

What to expect

The T-junction and Wilkinson power dividers for this problem are three-port circuits. The directional coupler is a four-port circuit that is a passive, reciprocal network. Each port has a special part in the directional coupler. The general diagram for the directional coupler is shown in Figure 6-16. Port 1 is the input port, Port 2 is the coupled port, Port 3 is the isolated port, and Port 4 is the through port. The numbering convention changes between texts, but the relationship between the physical ports will always hold.

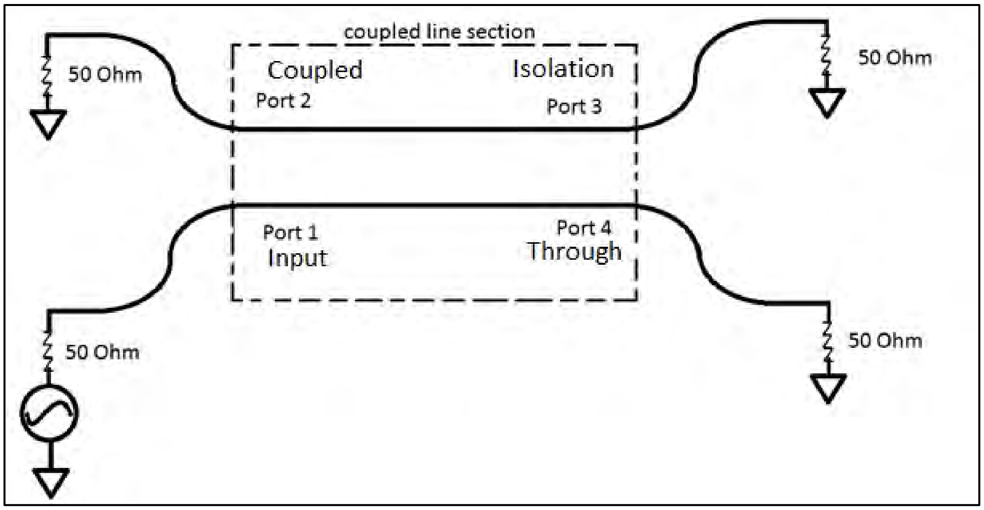


Figure 6-16. The general diagram for the directional coupler.

The characteristics used to describe the behavior of the directional coupler are Insertion Loss (IL), Coupling (C), Isolation (I), and Directivity (D). The ideal transmission line is expected to have high isolation and directivity, and a coupling characteristic of 15 dB. The microstrip line is expected to incur some losses, but will still have relatively high isolation and directivity. The coupling is also expected to be near 15 dB, but have slightly less coupling due to the microstrip losses.

The coupling from the input line with Ports 1 and 4 over to the adjacent line with Ports 2 and 3 occurs through edge coupling. Edge coupling occurs when two transmission lines are close enough in proximity for energy to pass from one line to another. For a coupled-line directional coupler to work, two properties must be satisfied.

First, the coupled section is a quarter wavelength at the operating frequency. Second, the even and odd mode impedances of the coupled line must hold the following relationship:

$$Z_0 = \sqrt{Z_{0e} Z_{0o}} \qquad \text{Equation 6-17}$$

What are the even and odd mode impedances? For the coupled-line section, there are two modes of current flow: the primary current flowing down the conductor from Port 1 to Port 4, and the displacement current coupling over into the line with Ports 2 and 3 that flows in the opposite direction. Similar to the differential pair circuit, these two current modes make up the common or differential mode of the coupled line, also known as the even and odd modes, respectively. These modes have associated impedances, termed Z_{0e} for the even mode and Z_{0o} for the odd mode.

The coupling between the two lines creates parasitic capacitances and inductance, which influence the even and odd impedances. For tighter coupling, the capacitance decreases and the inductance increases causing $Z_{0e} > Z_{0o}$. This exercise calls for 15-dB coupling, which can be considered medium coupling, and it is expected that $Z_{0e} > Z_{0o}$. An example of tight coupling is a

3-dB coupler, which couples half of the energy over to the adjacent line. Weaker coupling, or the closer the impedances are to one another, such as 30 dB+, will require spacing between the lines that prevents a significant amount of energy to transfer over to the adjacent line. This will result in a circuit that appears to have only one conductor. The even and odd impedance can be calculated using the coupling factor and put into the ADS LineCalc Tool to determine the physical dimensions of the microstrip line.

Execution

The coupling factor C for a 15-dB coupler is:

$$C = 10^{-15/20} = 0.1778$$

This yields the following impedances for the even and odd mode characteristic impedances:

$$Z_{0e} = 50\sqrt{\frac{1+0.1778}{1-0.1778}} = 59.8435$$

$$Z_{0o} = 50\sqrt{\frac{1-0.1778}{1+0.1778}} = 41.7756$$

The ADS schematic for the directional coupler in both the ideal transmission and microstrip lines is shown in Figure 6-17. The coupled transmission line component CLIN can be found in the TLines-Ideal palette. Likewise, the coupled microstrip line component MCLIN can be found in the TLines-Microstrip palette. In the LineCalc Tool, specify the type of line to be MCLIN instead of the default MLIN.

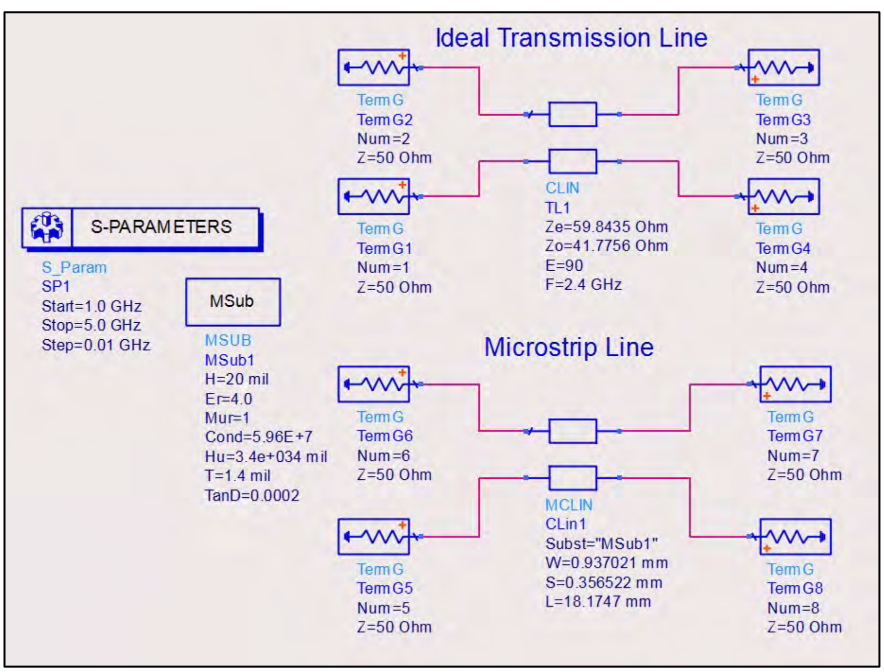


Figure 6-17. The ADS schematic for the directional coupler.

a) Plot the insertion loss.

The insertion loss for the coupler is described by the S(4,1) parameter. The directional coupler characteristics are supposed to be presented in positive dB, making the equation for insertion loss: $IL = -20\,log_{10}(S_{41})$. The dB() function in ADS is by default $20\,log_{10}(x)$, so the insertion loss can be found by using the equation in the Data Display in Figure 6-18.

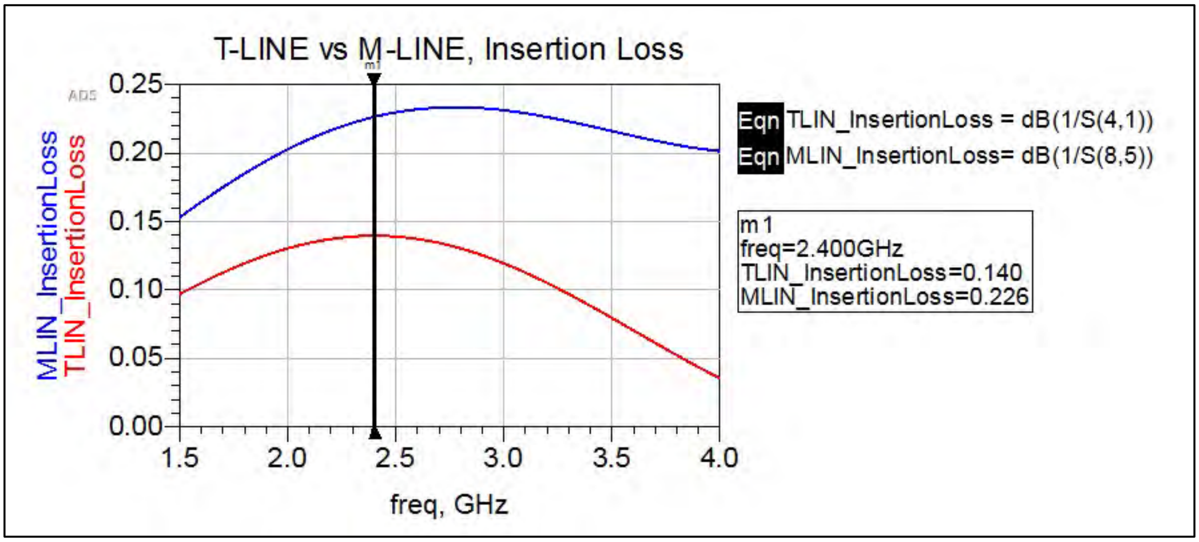


Figure 6-18. Finding the insertion loss using the equation in the ADS Data Display.

There is some insertion loss, which is to be expected because a portion of the energy is coupled over into the adjacent line. The tighter the coupling, the greater the insertion loss.

b) Plot the coupling.

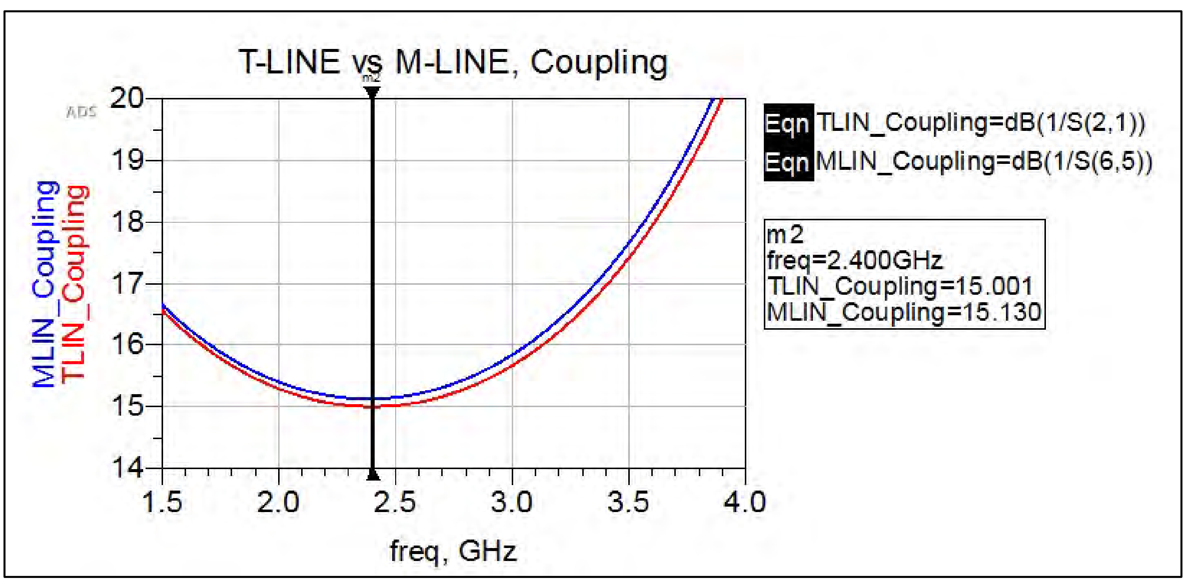


Figure 6-19. A plot of the coupling.

The coupling for the ideal transmission line is almost exactly 15 dB and the microstrip line has coupling very close to 15 dB, as expected (Figure 6-19). It is also noticeable, and important to note, that the coupling increases at surrounding frequencies.

c) Plot the isolation.

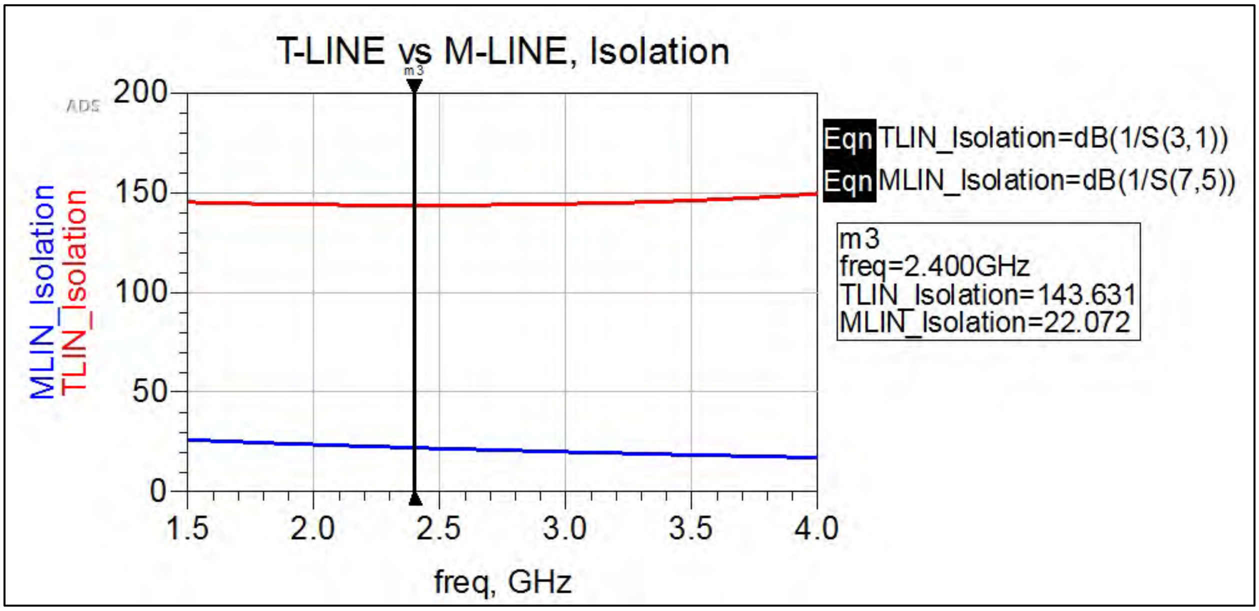


Figure 6-20. Plotting the isolation.

The isolation for the ideal transmission line is extremely high as expected; almost no energy is at Port 3 (Figure 6-20). The microstrip line; however, sees less isolation at a value to 22 dB. While this is not ideal, it is still less than the 15 dB coupled over into port 2, so the directional coupler is still functional. Sometimes to improve the isolation, the port is terminated internally or externally with a matched load.

d) Plot the directivity.

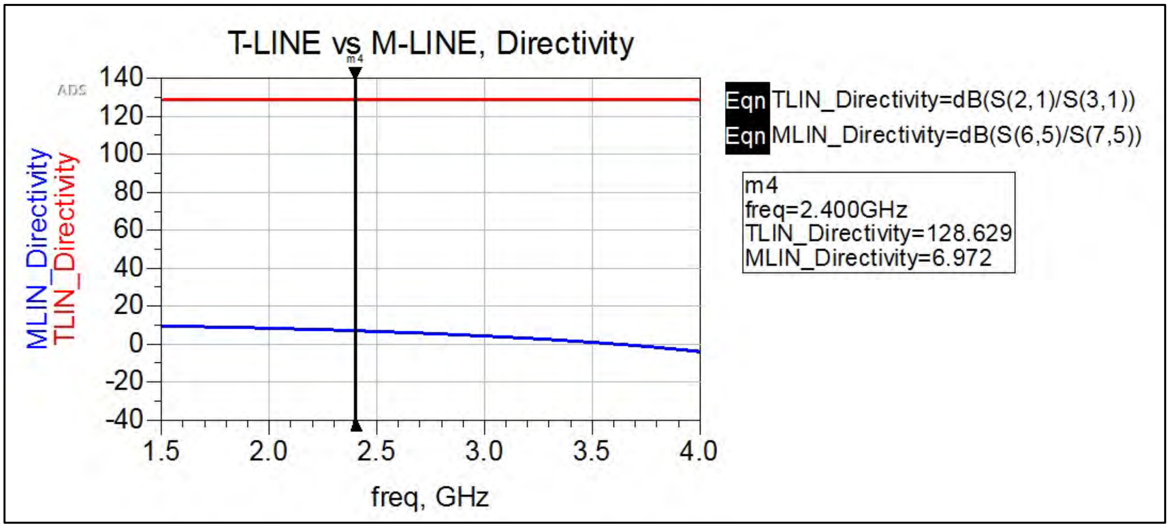


Figure 6-21. Plotting the directivity.

The directivity should be as high as possible, as shown by the ideal transmission line in Figure 6-21. Because directivity is a function of isolation, improving the isolation is key to improving the directivity.

Conclusion

The coupled-line directional coupler is an interesting tool, but why would it be used? The coupling characteristic is important mostly for power monitoring. A small portion of the power going through a circuit can be coupled off and analyzed to determine whether the level of power is enough, too little or too much. This is especially important for circuits containing transistors that require a certain level of power for optimum performance. Another application for the directional coupler is to utilize its isolation characteristic to combine signals. Inputs placed at Ports 3 and 4 can be combined to flow out of Port 1. The isolated Port 2 will dissipate losses from Ports 3 and 4. Finally, the directional coupler is a power divider. Is it effective at high frequencies, easy to fabricate and takes up little room due to the proximity of the coupled lines. A common example of a directional coupler is that it is used in network analyzers to sample both the incident and reflected waves without damaging the equipment.

Chapter 7 – Design Projects

Prologue

This chapter contains design projects that realize the concepts covered previously in an instructional lab environment. It is important to learn not only the theory of microwave engineering, but also the practical application. The design projects covered are presented in three stages: design, fabrication and design validation. While the actual fabrication and validation stages are not possible to show in the textbook structure, the processes will be described in detail. The Appendices to this chapter will cover the fabrication process in order to validate the circuit designs.

The labs will be presented using the design parameters of a Rogers RT/duroid ® 5880 laminate at the center frequency of 2.4 GHz. The Rogers PCB is chosen for its combination of a low dielectric constant, mechanical stability and low cost. Users; however, are encouraged to alter the designs based on the materials available at hand, and if there is a desired center frequency. 2.4 GHz is chosen because it is a standard Wi-Fi frequency that is unlicensed and provides a physical circuit size easy for beginners to use.

Project 1: Stepped Impedance Low-Pass Filter

Design Specifications

Filter type	Maximally flat
3-dB cutoff frequency	2.4 GHz
Out-of-band rejection	at least 20 dB at 4.0 GHz
Generator/load impedance	50 Ohms
Dielectric thickness	20 mil
Dielectric constant	2.2
Dielectric loss	0.0004
Conductor thickness	0.5 mil
Conductor	Copper
Maximum line impedance	100 Ohms
Minimum line impedance	20 Ohms

Design

Step 1: Determine the number of sections

The attenuation is found with respect to the normalized out-of-band rejection frequency.

$$\frac{\omega}{\omega_c} - 1 = \frac{2\pi\,(4.0\ x\ 10^9)}{2\pi\ (2.4\ x\ 10^9)} - 1 = 0.67 \qquad \text{Equation 7-1}$$

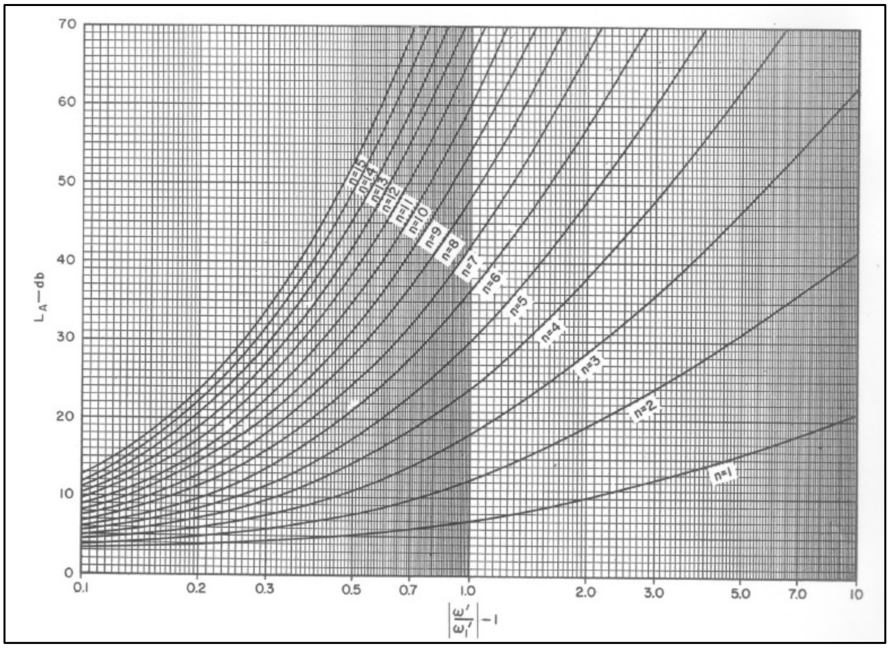


Figure 7-1. Attenuation verses normalized frequency. Reproduced by permission from G. Matthaei, E.M.T. Jones, and L. Young, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, Norwood, MA: Artech House, Inc., 1980. © 1980 by Artech House, Inc.

In Figure 7-1, use the attenuation versus normalized frequency plot by Matthaei, Young and Jones at the specified out-of-band rejection point. 20 dB of attenuation at the normalized frequency of 0.67 yields n = 5 sections. To be safe and expect reasonable losses during fabrication, overshoot the goal and use n = 6 sections.

The equations for generating the plot are given by Equation 7-2 and 7-3 if a hand calculation is desired[12].

$$L_A(\omega) = 10\, log_{10}\left[1 + \epsilon\left(\frac{\omega}{\omega_c}\right)^{2n}\right] \qquad \text{Equation 7-2}$$

$$\epsilon = \left[antilog_{10}\left(\frac{L_A}{10}\right)\right] - 1 \qquad \text{Equation 7-3}$$

Step 2: Pick a lumped-element circuit prototype

There are two prototypes available: the T-network and the π-network. Either one will work. The π-network is chosen (Figure 7-2).

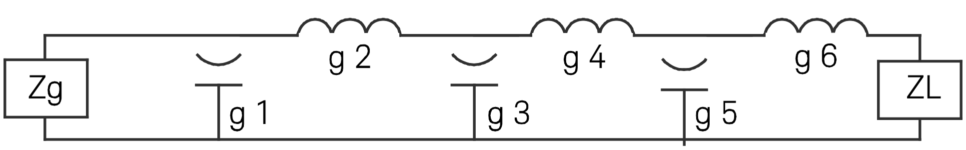


Figure 7-2. A generic prototype.

[12] *G. Matthaei, E.M.T. Jones, and L. Young, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, Norwood, MA: Artech House, Inc., 1980. © 1980 by Artech House, Inc.*

Step 3: Find the g-value of each section

The element values for a maximally flat low-pass filter are found by the following relationship[11]:

$$g_0 = 1$$
$$g_k = 2\,sin\left[\frac{(2^k-1)\pi}{2^n}\right]\;;\; k = 1, 2, \ldots n \qquad \text{Equation 7-4}$$
$$g_{n+1} = 1$$

For this design, the following values are calculated:

g0 = Zg = 1
g1 = C1 = 0.5176
g2 = L1 = 1.4142
g3 = C2 = 1.9318
g4 = L2 = 1.9318
g5 = C3 = 1.4142
g6 = L3 = 0.5176
g7 = ZL = 1

Step 4: Impedance and frequency scaling

Each element now needs to be scaled for impedance and frequency. In the π-network, the source and load impedances are normalized to a value of 1. Therefore, by multiplying the source and load impedances by a value of R_o, their original values are returned. In this design, R_o is the characteristic impedance, 50 Ohms. The shunt capacitor elements will be scaled by the admittance $1/R_o$ and the series inductors are scaled by R_o similar to the source and load impedances.

The elements also need to be frequency scaled by changing the cutoff frequency from w = 1 rad/s to $\omega = \frac{\omega}{\omega_c}$. The shunt capacitor susceptance transforms from $j\omega C_k \; to \; j\frac{w}{w_c}C_k'$. The same applies to the series inductor reactance. The final impedance and frequency scaled element values are given below.

$$C_k' = \frac{C_k}{R_o\omega_c}\;;\; L_k' = \frac{R_oL_k}{\omega_c} \qquad \text{Equation 7-5}$$

$$C_1' = \frac{C_1}{R_o\omega_c} = \frac{0.5176}{50(2\pi\,(2.4\,x\,10^9))} = 0.686488\,pF$$
$$L_1' = \frac{R_oL_1}{\omega_c} = \frac{50(1.4142)}{2\pi\,(2.4\,x\,10^9)} = 4.6891nH$$
$$C_2' = 2.56213\,pF$$
$$L_2' = 6.40532\,nH$$
$$C_3' = 1.87564\,pF$$
$$L_3' = 1.71622nH$$

Step 5: Check the lumped-element frequency response
The scaled lumped-element circuit is simulated in ADS to double check that the design specifications are satisfied (Figure 7.3).

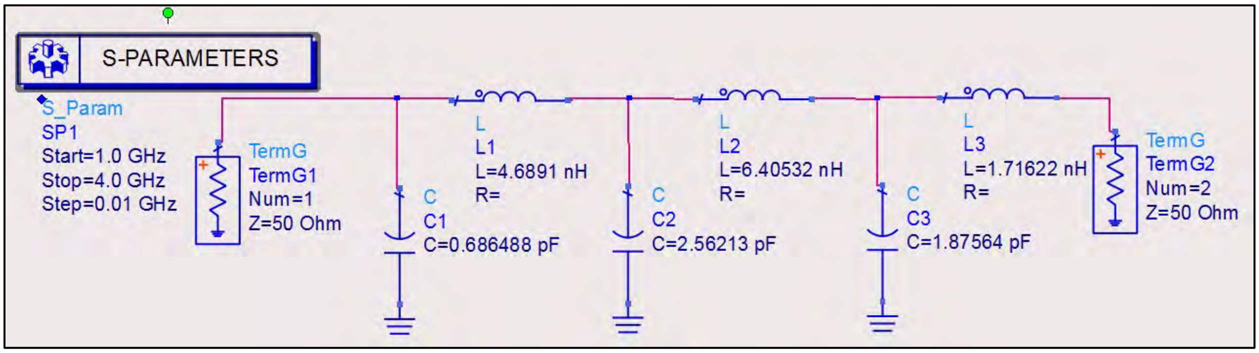


Figure 7-3. Simulation of the scaled lumped element circuit in ADS.

The frequency response at the cutoff frequency and out-of-band rejection frequency is observed for design verification. The 3-dB cutoff and minimum 20-dB out-of-band rejection specifications are validated (Figure 7-4). The circuit can be transformed into its microstrip line equivalent for fabrication.

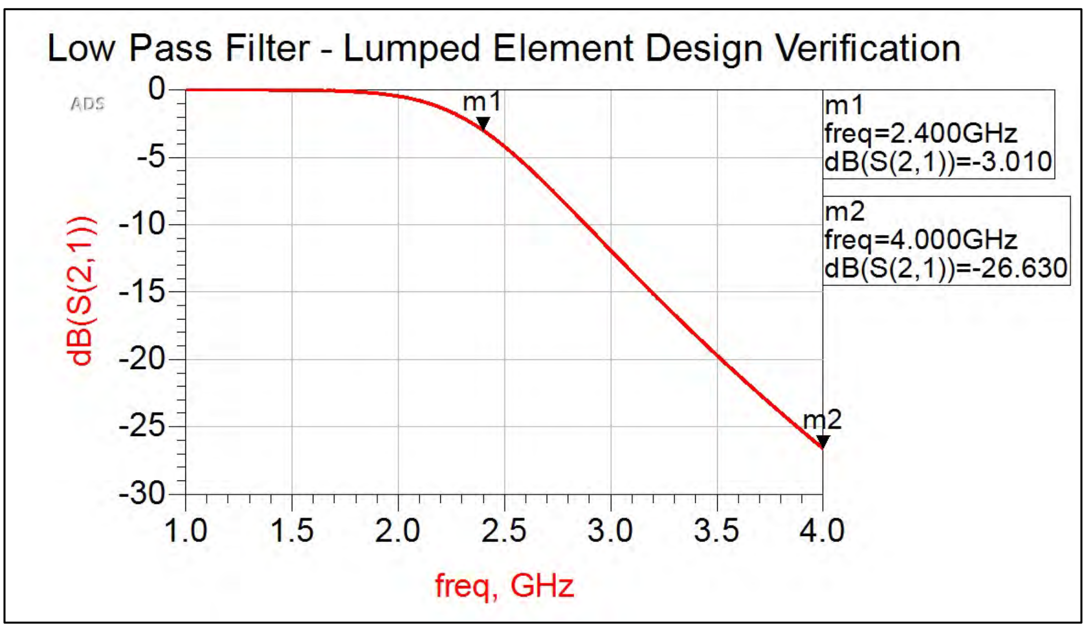


Figure 7-4. Verification of the design.

Step 6: Transform into distributed elements
The electrical length of each section is found by:

$$\beta l_k = \frac{C_k Z_{low}}{R_o} \text{ for the capacitors} \qquad \text{Equation 7-6}$$

$$\beta l_k = \frac{L_k R_o}{Z_{high}} \text{ for the inductors} \qquad \text{Equation 7-7}$$

$$\beta l_{C1} = \frac{C_1 Z_{low}}{R_o} = \frac{0.5176(20)}{50}\left(\frac{360°}{2\pi}\right) = 11.8488°$$

$$\beta l_{L1} = \frac{L_1 R_o}{Z_{high}} = \frac{1.4142(50)}{100}\left(\frac{360°}{2\pi}\right) = 40.51385°$$

$$\beta l_{C2} = 44.2736°$$

$$\beta l_{L2} = 55.34199°$$

$$\beta l_{C3} = 32.4111°$$

$$\beta l_{L3} = 14.82815°$$

Step 7: Use ADS LineCalc to find the width and length of each section (Figure 7-5):

Element	Width (mm)	Length (mm)
1. C1	5.1976	2.89294
2. L1	0.4345	10.6303
3. C2	5.1976	10.8096
4. L2	0.4345	14.5209
5. C3	5.1976	7.91332
6. L3	0.4345	3.89069
50 Ohm Line	1.5462	22.8006

Figure 7-5. The length and width of each section are shown in this table.

Step 8: Check the distributed-element frequency response

The microstrip line realization is simulated in ADS to verify that the design is still valid (Figure 7-6). This simulation will also provide a basis for understanding any errors incurred during fabrication.

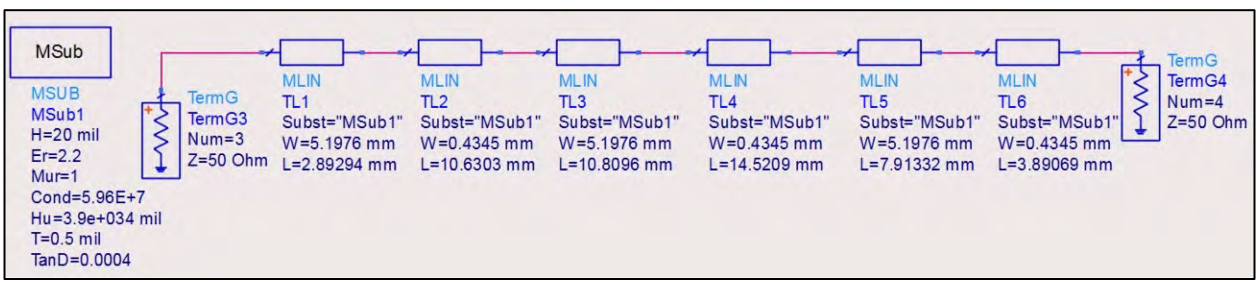


Figure 7-6. The microstrip line realization is simulated in ADS to verify it is still valid.

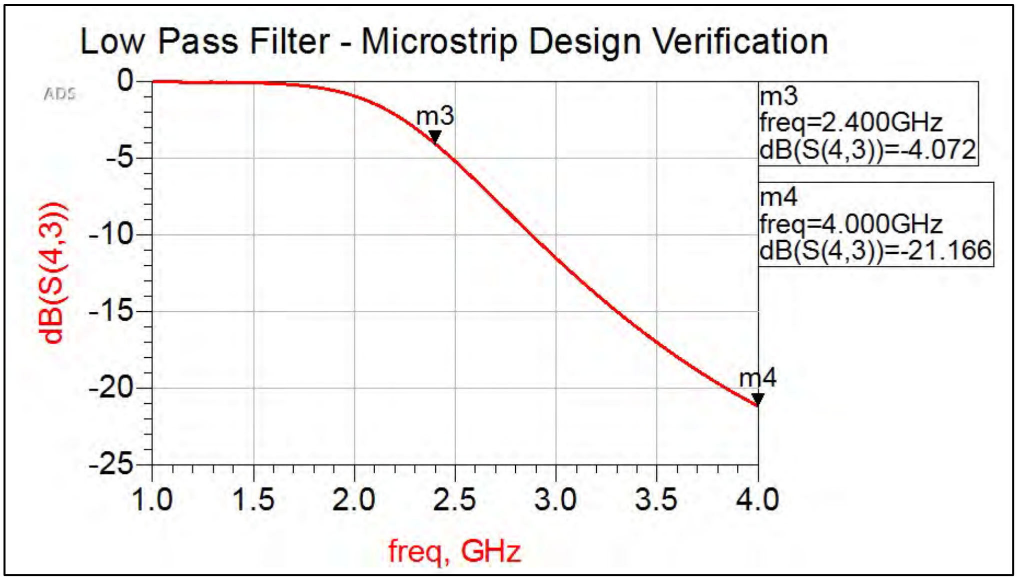


Figure 7-7. Here the cut-off frequency has decreased to 4 dB.

The cutoff frequency response has decreased to 4 dB, but the out-of-band rejection is still at least 20 dB (Figure 7-7). This realization is acceptable enough to continue fabrication because some tuning will be done to achieve the 3-dB cutoff frequency.

Fabrication

Prepare the schematic for fabrication.

Step 1: Add the 50-Ohm connecting lines

Open a new schematic and copy the microstrip line circuit over onto it. Place a 50-Ohm piece of line at each port. A quarter-wavelength is a safe length of line, and it can be cut down during fabrication for space, if needed. It is better to have too much line, rather than not enough.

Step 2: Add an MSTEP component between each impedance section

The junctions where the step impedances meet involve large, abrupt changes in line width. These step junctions create parasitics and discontinuities[13] that the simulation does not address unless it is explicitly instructed to do so by the user. By inserting a MSTEP component, these discontinuities are better accounted for at a cost of lengthening the microstrip line and subsequently, the frequency response (Figure 7-8).

---

[13] *Norbert H. L. Koster and Rolf. H. Jansen, "The Microstrip Step Discontinuity: A Revised Description," IEEE Transactions on Microwave Theory and Techniques, Vol. MTT-34, No. 2, February 1986.*

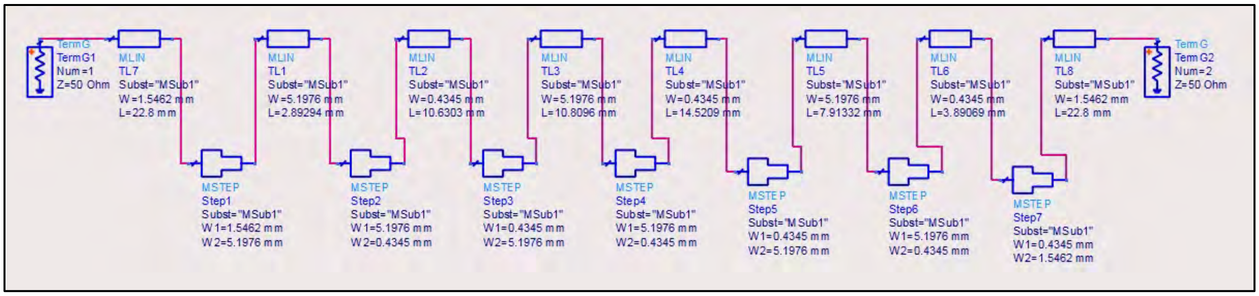


Figure 7-8. Adding an MSTEP component between each impedance section.

Step 3: Tune the new circuit

The MSTEP component added line length that needs to be compensated for elsewhere. Use the Tuning tool  in ADS to fine tune the design to achieve the 3-dB cutoff frequency response at 2.4 GHz (Figure 7-9). No element should be changed more than 20% of its length to maintain the design integrity. It is best to tune symmetrically, so the first and sixth element section will be tuned together. Once the Tune Parameters window appears, it will prompt the user to click on the parameters to tune. Click directly onto the length parameter for the first and sixth elements. The default step size may be too large initially. Feel free to change the step size as needed for tuning. If the 20% limit is hit on any element pair, move to the next pair to tune. Some pairs will tune the frequency response better than others.

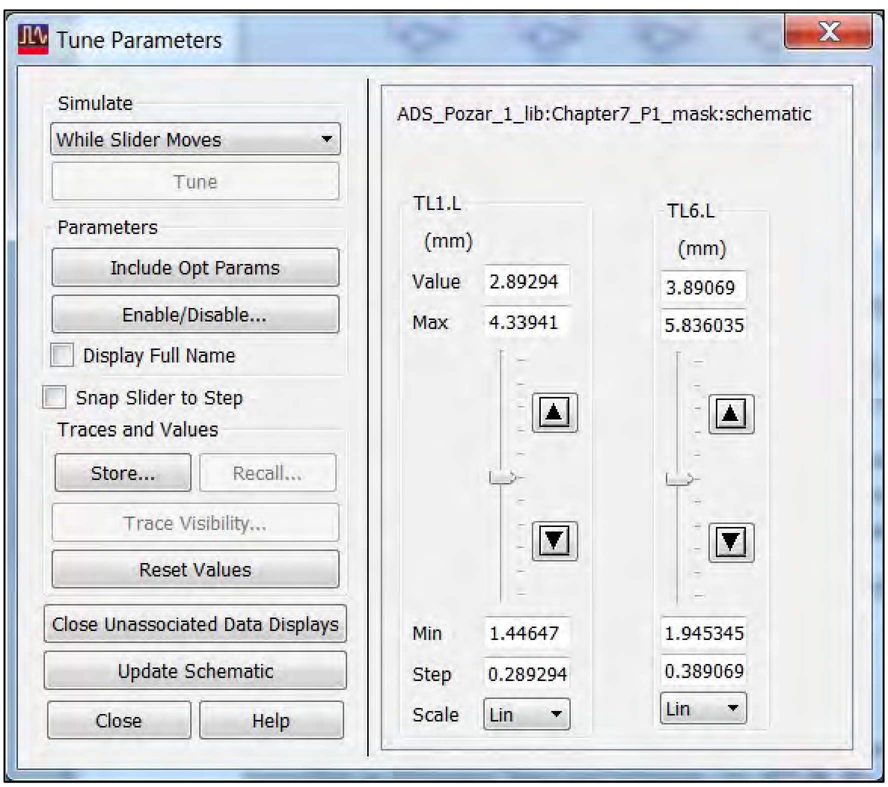


Figure 7-9. The ADS Tuning tool can be used to tune the circuit.

For the best tuning technique, first plot the S(2,1) response in the Data Display. As the element values are tuned, the frequency response will change automatically. Place markers at the two design frequency specifications: 2.4 GHz and 4.0 GHz for guidance on which way to tune the circuit. The initial, pre-tuned circuit is shown in Figure 7-10 with the tuning setup.

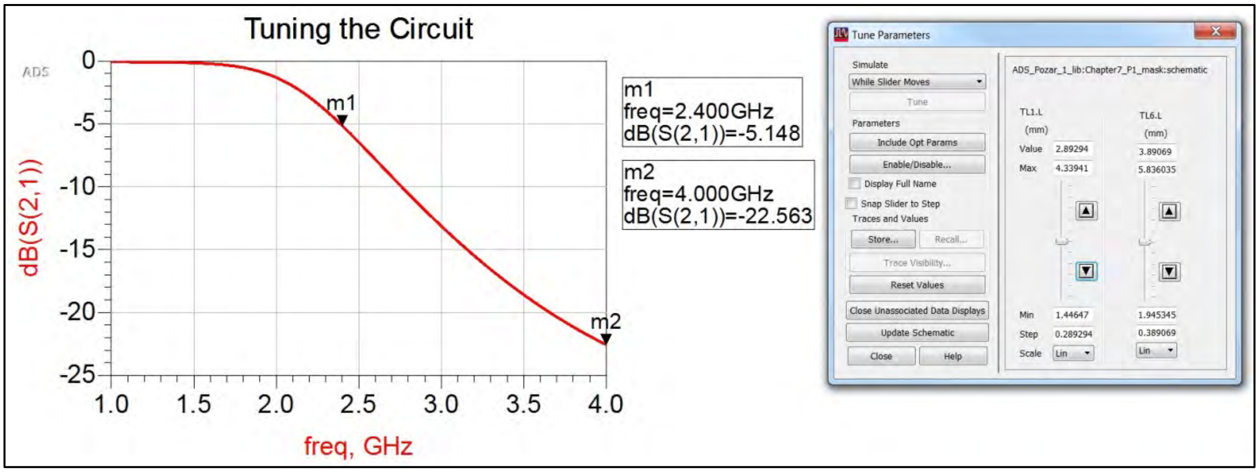


Figure 7-10. The initial, pre-tuned circuit and tuning setup are shown here.

After tuning impedance sections 1, 2, 5, and 6, the final frequency response is achieved. Both the 3-dB cutoff and out-of-band rejection design specifications are met (Figure 7-11).

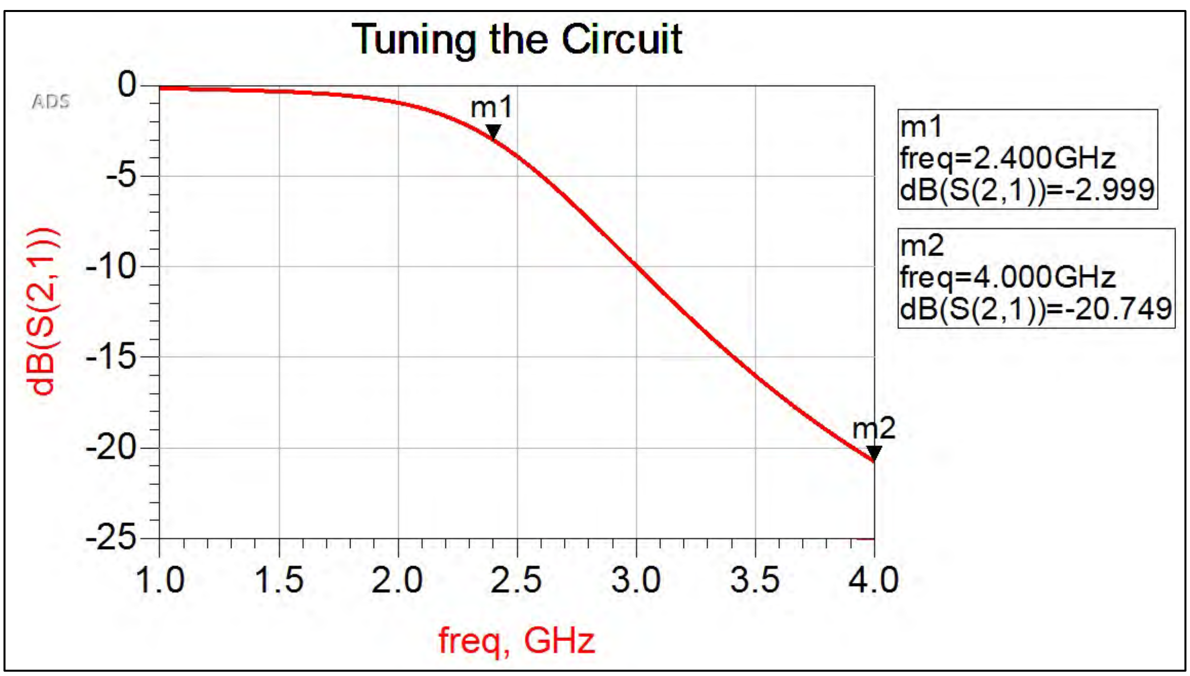


Figure 7-11. Tuning the circuit.

In the Tuning window, an option to update the schematic is available. This will push the tuned values into the schematic and automatically update the lengths. For the mask, the MSUB component is needed, but the S-parameter simulation and the terminations are not needed. Remove those components entirely. The final circuit schematic is shown in Figure 7-12.

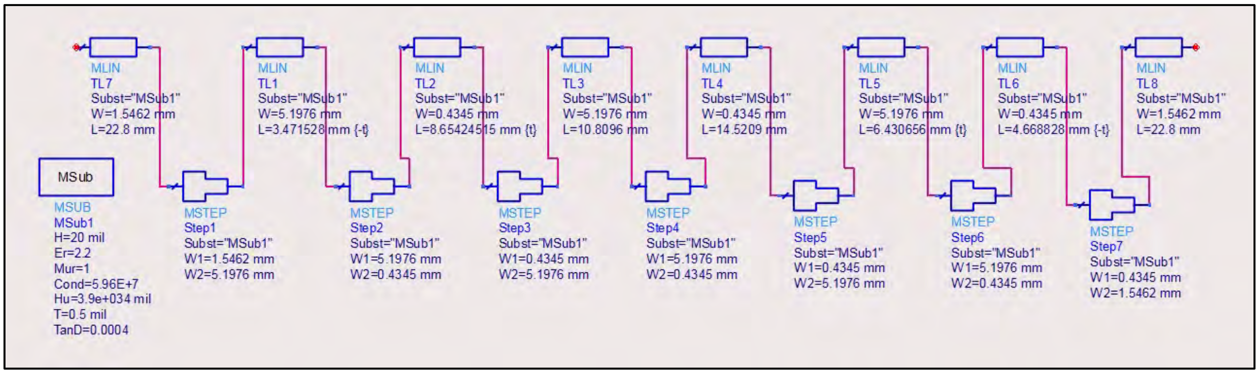


Figure 7-12. The final circuit schematic.

Step 4: Generate the layout mask
Follow the ADS Mask Layout procedure in Appendix A.

Step 5: Fabricate the circuit
Fabricate the circuit using the Circuit Fabrication Technique in Appendix B.

Validation

With the network analyzer properly calibrated for this lab, use it to measure the S-parameters of the circuit. Using a Connection Manager, use ADS to read data from the network analyzer into .S2P files. Recalling from Chapter 3, a Data File component can be used to simulate the behavior of the fabricated circuit in ADS.

Compare the experimental results to the design in ADS and determine the success of the fabricated circuit.

Project 2: Coupled-Line Bandpass Filter

Design Specification

Filter type	Equal-ripple
Center frequency	2.4 GHz
3-dB Fractional bandwidth	10%
Number of resonators	3
In-band equal-ripple	0.5 dB
Generator/load impedance	50 Ohms
Dielectric thickness	20 mil
Dielectric constant	2.2
Dielectric loss	0.0004
Conductor thickness	0.5 mil
Conductor	Copper

Design

Step 1: Choose a circuit prototype

There are two prototypes available: the T-network and the π-network. Either one will work. The π-network is chosen (Figure 7-13).

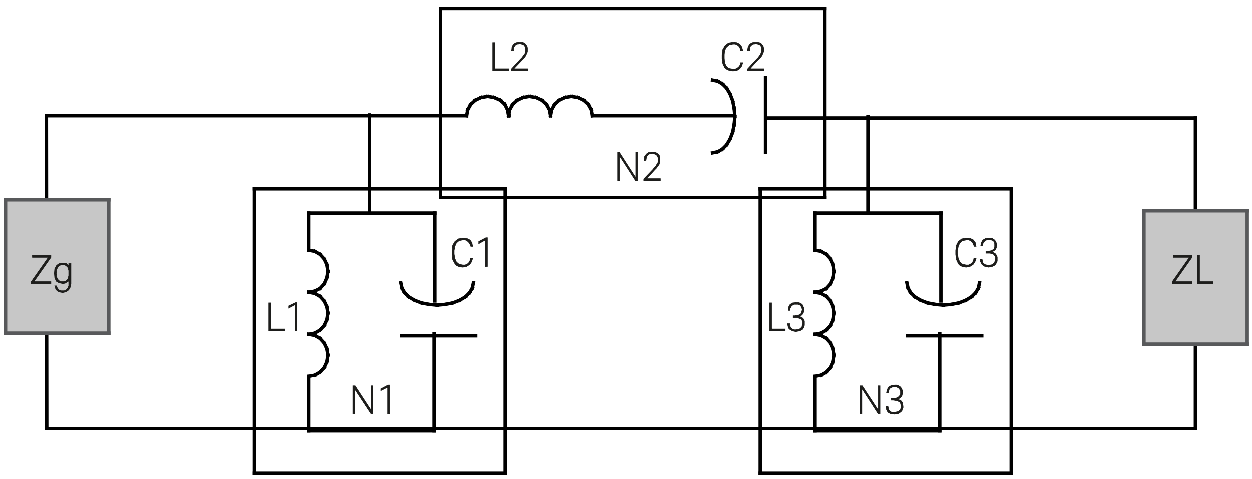


Figure 7-13. The generic prototype to be used in Project 2.

Step 2: Find the g-value of each section

The element values for an equal-ripple band-pass filter are found by the following relationship[14]:

$$\beta = ln\left(coth\frac{L_{Ar}}{17.37}\right) ; L_{Ar} = in - band\ ripple\ in\ dB \qquad \text{Equation 7-8}$$

$$\gamma = sinh\left(\frac{\beta}{2n}\right) \qquad \text{Equation 7-9}$$

$$a_k = sin\left[\frac{(2k-1)\pi}{2n}\right] ; k = 1, 2, \ldots n \qquad \text{Equation 7-10}$$

$$b_k = \gamma^2 + sin^2\left(\frac{k\pi}{n}\right) ; k = 1, 2, \ldots n \qquad \text{Equation 7-11}$$

$$g_0 = 1$$

$$g_1 = \frac{2a_1}{\gamma}$$

$$g_k = \frac{4a_{k-1}a_k}{b_{k-1}g_{k-1}} ; k = 2, 3, \ldots n$$

$$g_{n+1} = 1 ; for\ n\ odd$$

$$= coth^2\left(\frac{\beta}{4}\right) ; for\ n\ even$$

[14] *G. Matthaei, E.M.T. Jones, and L. Young, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, Norwood, MA: Artech House, Inc., 1980. © 1980 by Artech House, Inc.*

For n = 3,

g0 = Zg = 1.00
g1 = L1,C1 = 1.5963
g2 = L2,C2 = 1.0967
g3 = L3,C3 = 1.5963
g4 = ZL = 1.00

Step 3: Impedance and frequency scaling
For the shunt sections of the π-network in Step 1, or sections n = 1, 3, the following scaling transformation will be made:

$$L'_k = \frac{\omega_2 - \omega_1}{\omega_0}\frac{R_0}{\omega_0 C_k}$$
$$C'_k = \frac{C_k}{R_0(\omega_2 - \omega_1)}$$

For the series sections of the π-network in Step 1, or section n = 2, the following scaling transformation will be made:

$$C'_k = \frac{\omega_2 - \omega_1}{\omega_0}\frac{1}{\omega_0 R_0 L_k} \quad \text{Equation 7-12}$$
$$L'_k = \frac{R_0 L_k}{\omega_2 - \omega_1} \quad \text{Equation 7-13}$$

Per the design specifications, the center frequency is 2.4 GHz and there is a 10% fractional bandwidth.

Let the center frequency be the geometric mean of its band edges.

$$\omega_0 = \sqrt{\omega_1 \omega_2} \quad \text{Equation 7-14}$$

Using the 10% fractional bandwidth:

$$0.1\,\omega_0 = \omega_2 - \omega_1$$

The quadratic equation for one of the band edges can be found.

$$\omega_1^2 + 0.1\omega_0\omega_1 - \omega_0^2 = 0$$

$$\omega_1 = 1.434\, x10^{10}\, rad/s$$

$$f_1 = 2.282\, GHz$$

$$f_2 = 2.524\, GHz$$

$$C'_1 = 21.178\, pF$$
$$L'_1 = 0.2077\, nH$$
$$C'_2 = 0.1209\, pF$$

$$L_2' = 36.364\,nH$$
$$C_3' = 21.178\,pF$$
$$L_3' = 0.2077\,nH$$

Step 4: Check the lumped-element frequency response
The scaled, lumped-element circuit is simulated in ADS to double check that the design specifications are satisfied (Figure 7-14).

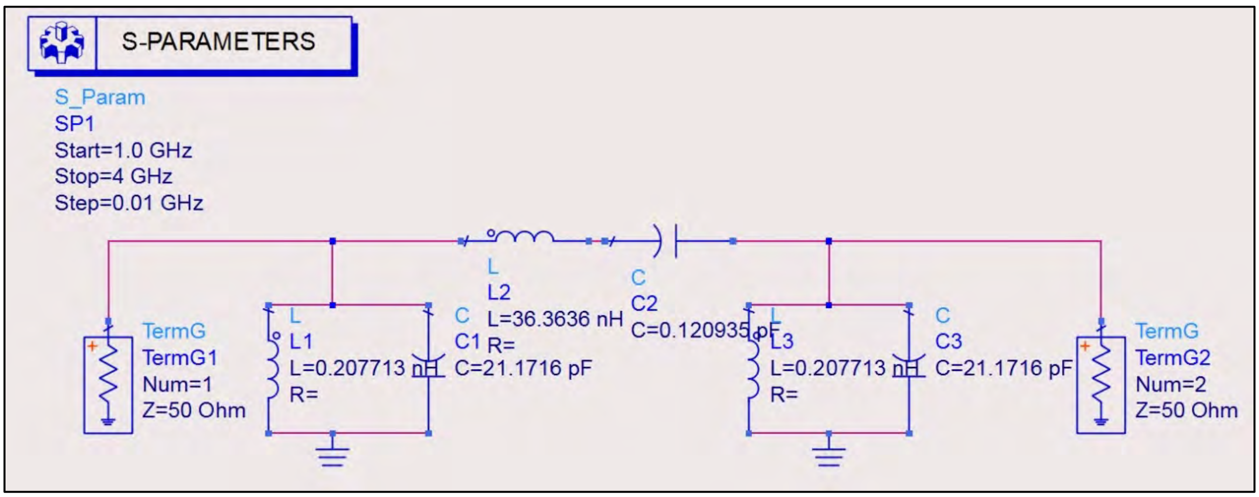


Figure 7-14. Simulation of the scaled, lumped-element circuit to ensure its design specifications are satisfied.

The frequency response at the center frequency and in-band ripple is observed for design verification (Figure 7-15). The circuit can be transformed into its microstrip line equivalent for fabrication.

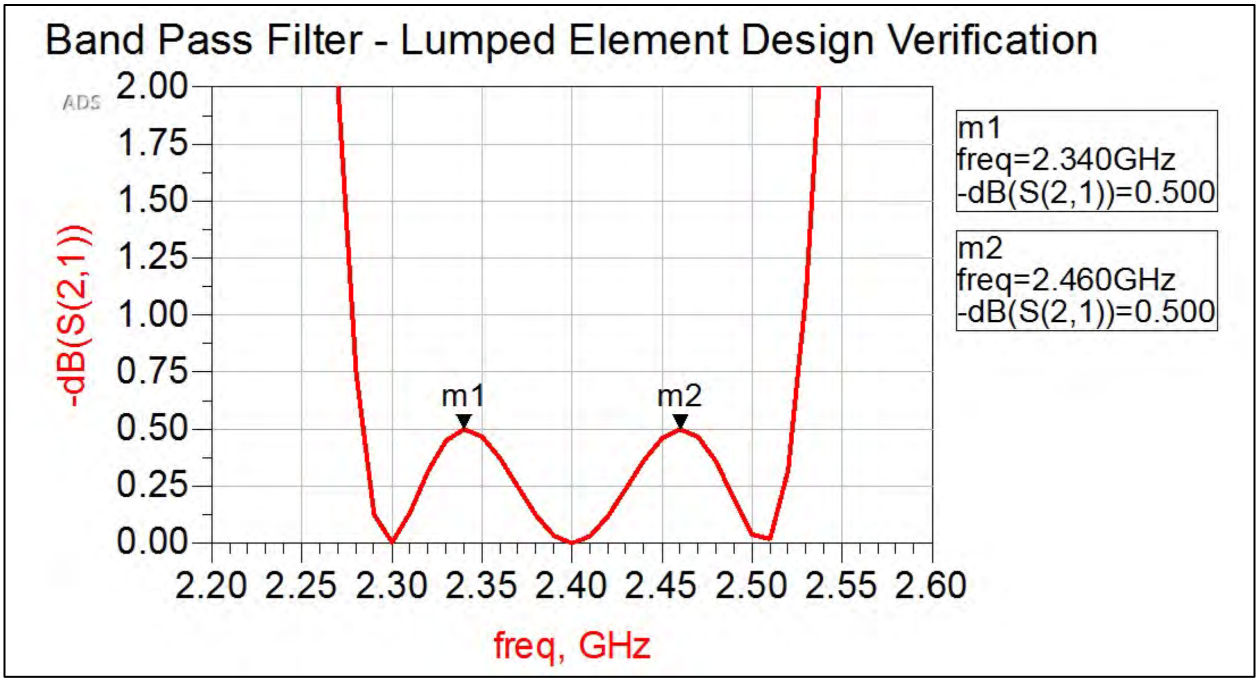


Figure 7-15. Observe the frequency response at the center frequency and in-band ripple to verify the design.

Step 5: Transform into distributed elements

The use of admittance inverters is one possible way to transform the lumped-element model into a fully distributed model. They can be formed using a quarter-wave transformer of specific characteristic impedance. There will be a total of n+1 admittance inverters for the bandpass filter.

$$Z_o J_1 = \sqrt{\frac{\pi\Delta}{2g_1}} = \sqrt{\frac{\pi(10\%)}{2(1.5963)}} = 0.313691$$

$$Z_o J_2 = \frac{\pi\Delta}{2\sqrt{g_1 g_2}} = \frac{\pi(10\%)}{2\sqrt{(1.5936)(1.0967)}} = 0.118719$$

$$Z_o J_3 = \frac{\pi\Delta}{2\sqrt{g_2 g_3}} = \frac{\pi(10\%)}{2\sqrt{(1.0967)(1.5936)}} = 0.118719$$

$$Z_o J_4 = \sqrt{\frac{\pi\Delta}{2g_3 g_4}} = \sqrt{\frac{\pi(10\%)}{2(1.5963)(1.0)}} = 0.313691$$

Because this is a coupled-line bandpass filter, the even and odd modes of each section are calculated.

$$Z_{oe} = Z_o[1 + JZ_o + (JZ_o)^2] \qquad \text{Equation 7-15}$$

$$Z_{oo} = Z_o[1 - JZ_o + (JZ_o)^2] \qquad \text{Equation 7-16}$$

$$Z_{oe1} = Z_o[1 + J_1 Z_o + (JZ_o)^2] = 50[1 + 0.313691 + (0.313691)^2] = 70.6047\ Ohms$$

$$Z_{oo1} = Z_o[1 - J_1 Z_o + (JZ_o)^2] = \ 50[1 - 0.313691 + (0.313691)^2] = 39.2356\ Ohms$$

$$Z_{oe2} = 56.6407\ Ohms$$

$$Z_{oo2} = 44.7688\ Ohms$$

$$Z_{oe3} = 56.6407\ Ohms$$

$$Z_{oo3} = 44.7688\ Ohms$$

$$Z_{oe4} = 70.6047\ Ohms$$

$$Z_{oo4} = 39.2356 Ohms$$

Step 6: Use ADS LineCalc to find the width and length of each section (Figure 7-16):

Element	Width (mm)	Length (mm)	Spacing (mm)
1. CLIN1	1.21767	23.3003	0.108993
2. CLIN2	1.49136	22.9172	0.533083
3. CLIN3	1.49136	22.9172	0.533083
4. CLIN4	1.21767	23.3003	0.108993
50-Ohm Line	1.5462	22.8006	N/A

Figure 7-16. A table containing the width and length of each section.

Step 7: Check the distributed-element frequency response
The microstrip line realization is simulated in ADS to verify the design is still valid (Figure 7-17). This simulation will also provide a basis for understanding any errors incurred during fabrication.

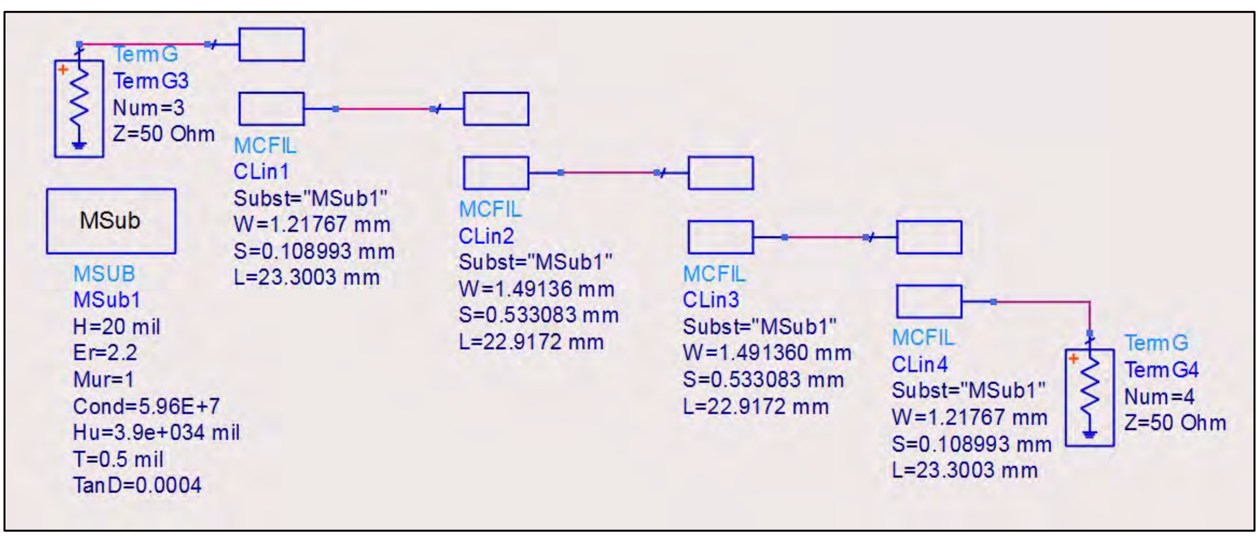


Figure 7-17. Simulation of the microstrip line realization in ADS to verify the design is still valid.

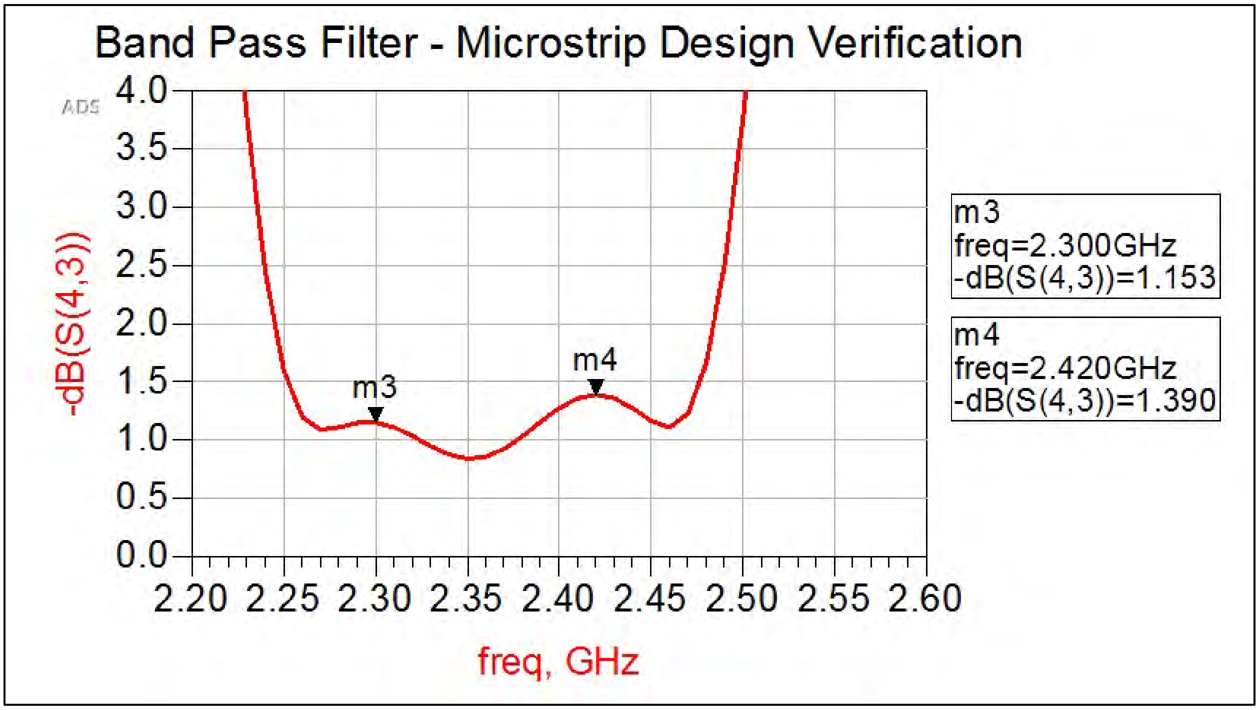


Figure 7-18. The frequency response of the microstrip design.

The frequency response is not ideal, but it is still a similar response to the lumped-element design (Figure 7-18). Similar to the low-pass filter, some tuning will be needed for fabrication.

Fabrication

Prepare the schematic for fabrication.

Step 1: Add the 50-Ohm connecting lines

Open a new schematic and copy the microstrip line circuit over onto it. Place a 50-Ohm piece of line at each port. A quarter wavelength is a safe length of line, and it can be cut down during fabrication for space, if needed. It is better to have too much line instead of not enough.

Step 2: Tune the circuit

The current frequency response of the microstrip circuit gives a 3-dB bandwidth of about 11.5% around a center frequency of 2.35 GHz. The fractional bandwidth is very close, but the circuit needs to be tuned so that the center frequency is 2.4 GHz. Again, the coupled-line section pairs should be tuned together at no more than a 20% difference in its original length. The original microstrip line frequency response is shown in Figure 7-19 as a reminder.

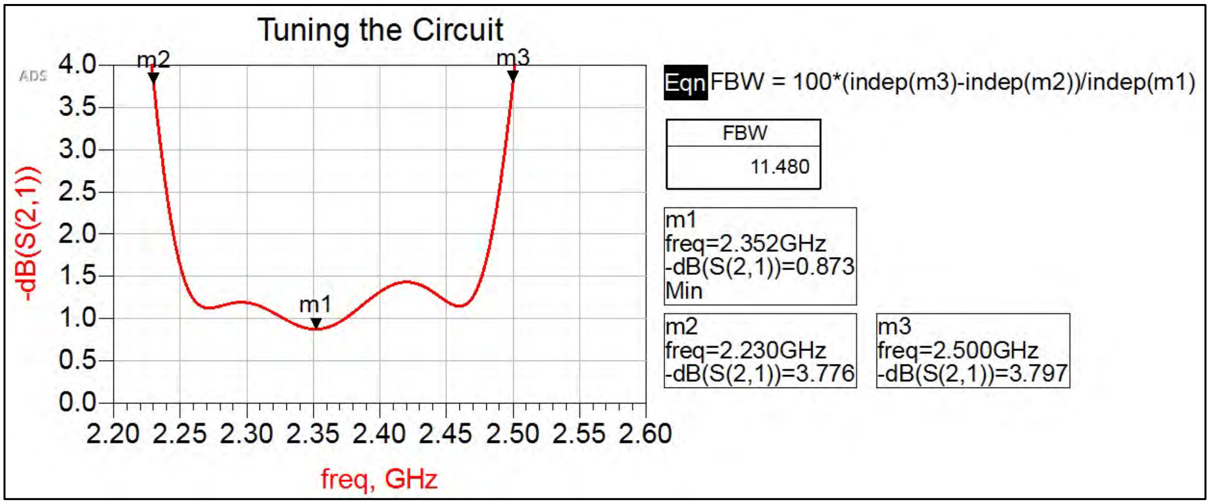


Figure 7-19. The original microstrip line frequency response.

The final tuned circuit and frequency response are given in Figures 7-20 and 7-21.

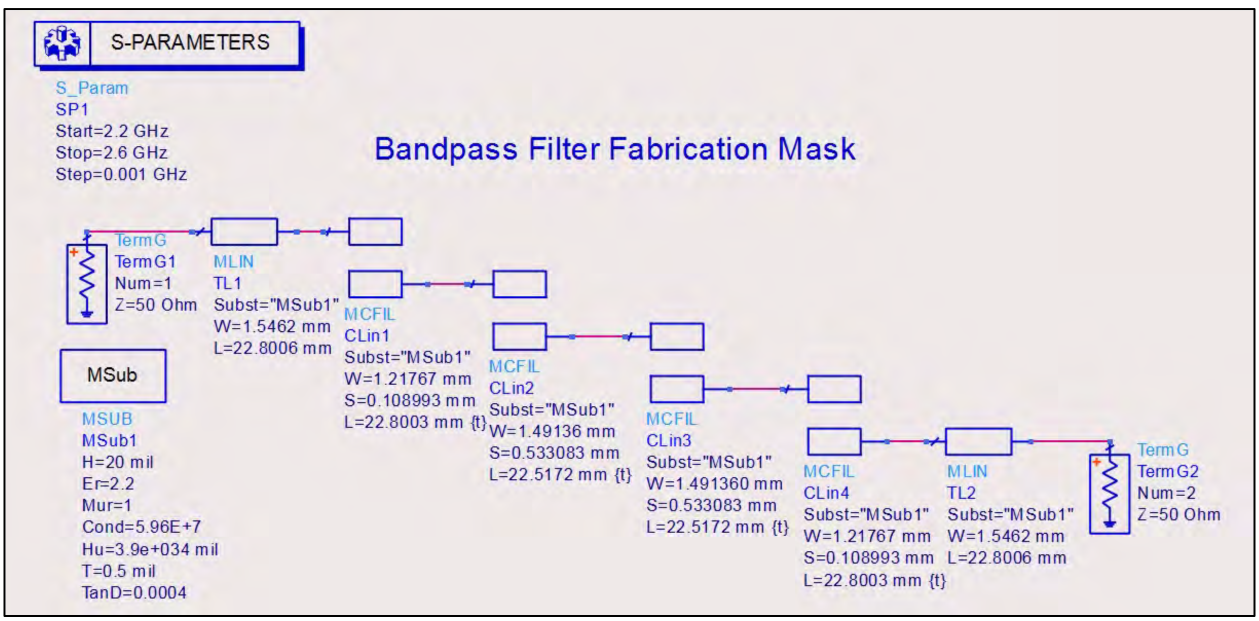


Figure 7-20. The final tuned circuit.

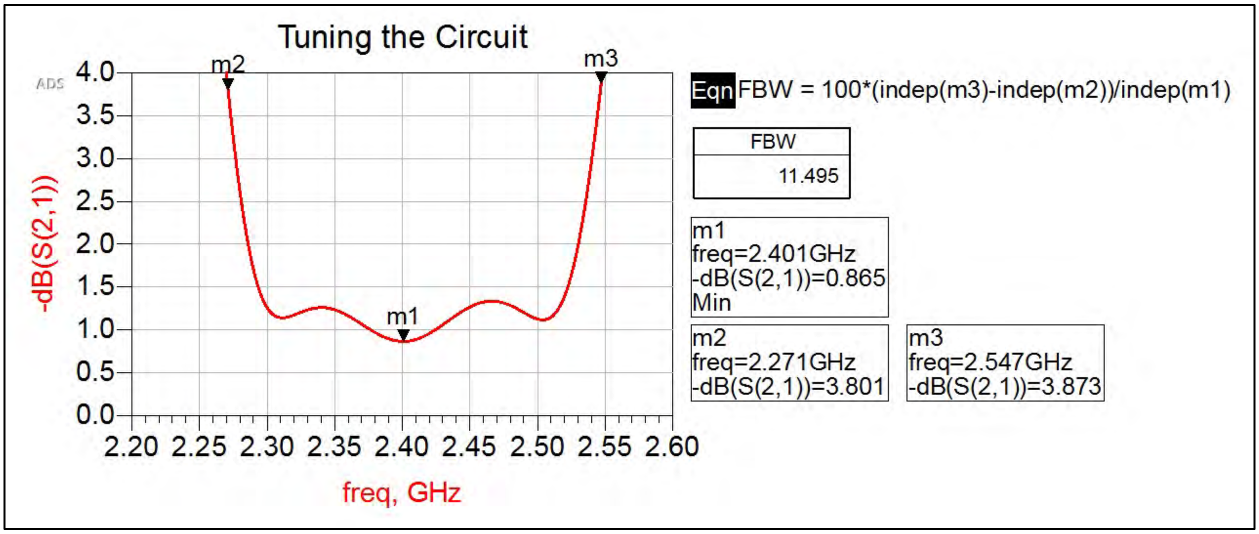


Figure 7-21. The final frequency response of the tuned circuit.

Step 3: Generate the layout mask

Follow the ADS Mask Layout procedure in Appendix A. Make sure the steps are connected perfectly at each coupled section. The mask for this filter will look similar to Figure 7-22.

Figure 7-22. The mask for the filter.

Step 4: Fabricate the circuit

Fabricate the circuit using the Circuit Fabrication Technique in Appendix B. With the short spacing between coupling sections, it may be required to use a scalpel to verify the separation of the conducting layer. The coupling between the lines will not work correctly if there is a connection.

Validation

With the network analyzer properly calibrated for this lab, use it to measure the S-parameters of the circuit. Using a Connection Manager, use ADS to read data from the network analyzer into .S2P files. Recalling from Chapter 3, a Data File component can be used to simulate the behavior of the fabricated circuit in ADS.

Compare the experimental results to the design in ADS and determine the success of the fabricated circuit. It may be required to tune the circuit even further after fabrication to ensure the correct center frequency. If this is needed, use ADS to shift the center frequency over some Δf. For example, if the measured center frequency is 2.3 GHz, the ADS circuit will need to be shifted 2.4 GHz + 0.1 GHz to reach 2.5 GHz in simulation. Determine the amount of length that needs to be added or taken away from the coupled sections. It is best to start with half the amount and test.

This way the user can slowly reach the center frequency rather than accidently tune too much. If removing line length, a scalpel will suffice. If line needs to be added, copper conducting tape can be used. Create a bond of solder between the board and tape to ensure a smooth, conductive flow.

Project 3: 20-dB Directional Coupler

Design Specifications

Center frequency	2.4 GHz
Coupling	20 dB
Generator/load impedance	50 Ohms
Dielectric thickness	20 mil
Dielectric constant	2.2
Dielectric loss	0.0004
Conductor thickness	0.5 mil
Conductor	Copper

Design

Step 1: Determine the impedance of the coupled section
The coupling factor is 20 dB.

$$C = 20\ dB = 10^{-20/20} = 0.1$$

The even and odd impedances of the coupling section are then[15]:

$$Z_{oe} = Z_o\sqrt{\frac{1+C}{1-C}} = 50\sqrt{\frac{1+0.1}{1-0.1}} = 55.2771$$

$$Z_{oo} = Z_o\sqrt{\frac{1-C}{1+C}} = 50\sqrt{\frac{1-0.1}{1+0.1}} = 45.2267$$

Step 2: Build the microstrip line circuit in ADS
Using LineCalc in ADS, the coupled-line dimensions are calculated for the quarter-wavelength, coupled-line section. The 50-Ohm termination ports are added in order, and the final schematic is shown in Figure 7-23.

[15] *G. Matthaei, E.M.T. Jones, and L. Young, Microwave Filters, Impedance-Matching Networks, and Coupling Structures, Norwood, MA: Artech House, Inc., 1980. © 1980 by Artech House, Inc.*

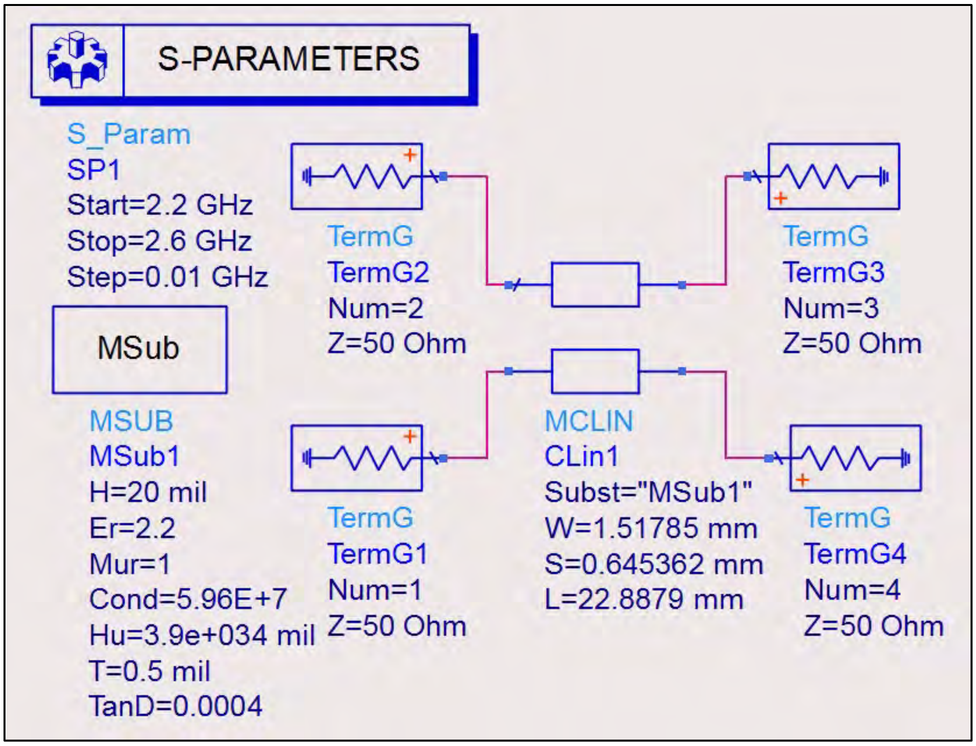


Figure 7-23. The final schematic of the microstrip line circuit in ADS.

Step 3: Verify the frequency response

The frequency response is simulated to verify the 20-dB coupling (Figure 7-24).

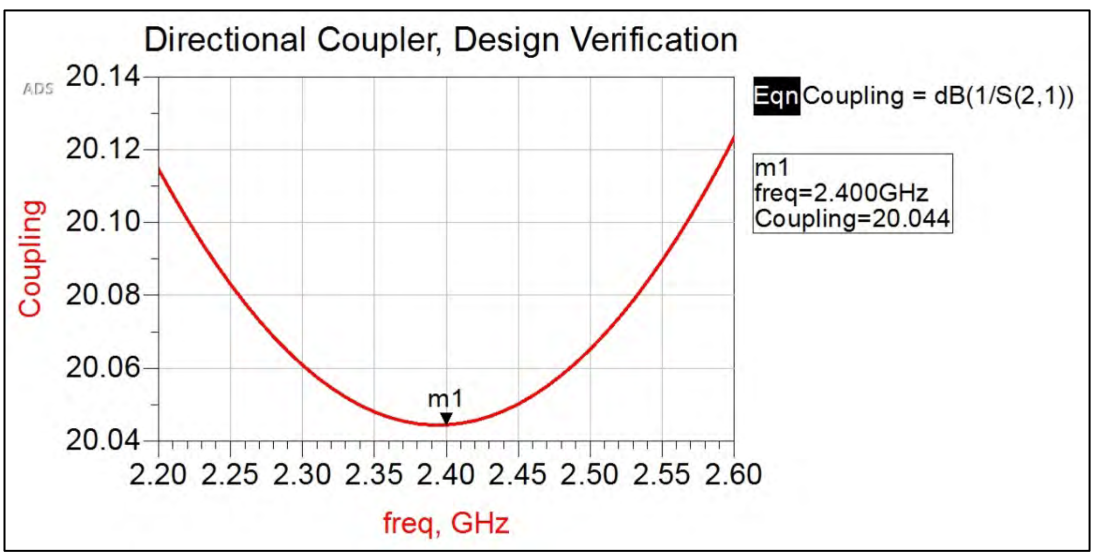


Figure 7-24. Simulation of the design’s frequency response.

At 2.4 GHz, the coupling factor is 20.044 dB, which is almost perfect. The design is acceptable for fabrication. There should not be too many issues with such a simple coupling section. If the design is not ideal, there was an error somewhere in the previous steps.

Fabrication
Prepare the schematic for fabrication.

Step 1: Add the 50-Ohm connecting lines
Open a new schematic and copy the microstrip line circuit onto it. Place a 50-Ohm piece of line at each port. A quarter wavelength is a safe length of line, and it can be cut down during fabrication for space, if needed. The four-port circuit poses a physical problem. The coupled ports will be too close to one another and it will be impossible to mount the connectors. Use ideal bends, MSABND_MDS, for maximum power transfer in ADS to turn Ports 2 and 4 toward the adjacent side of the board. The new schematic will look like Figure 7-25.

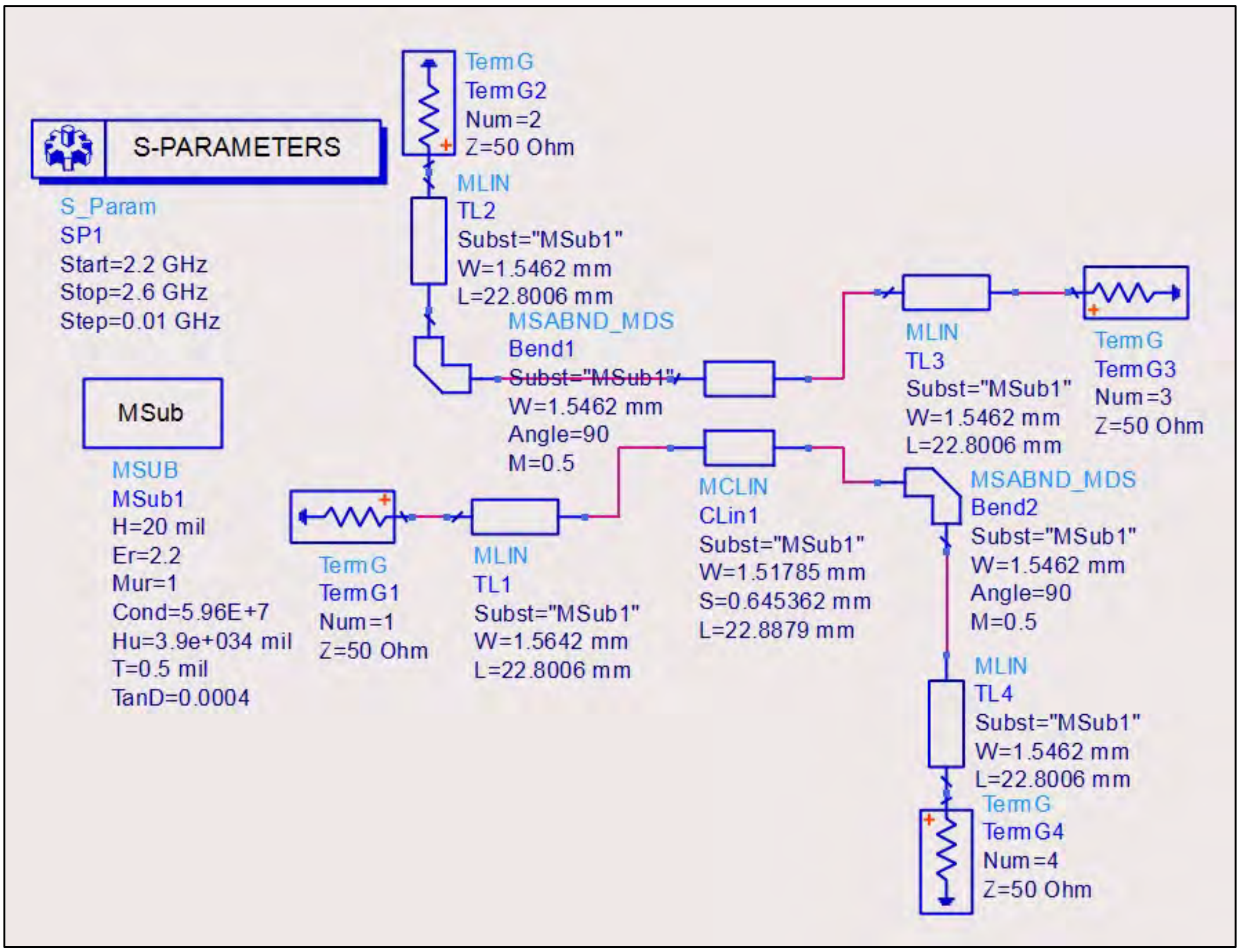


Figure 7-25. The new circuit schematic.

Step 2: Tune the circuit
With every addition of new line, the frequency response should be checked to verify the design still holds.

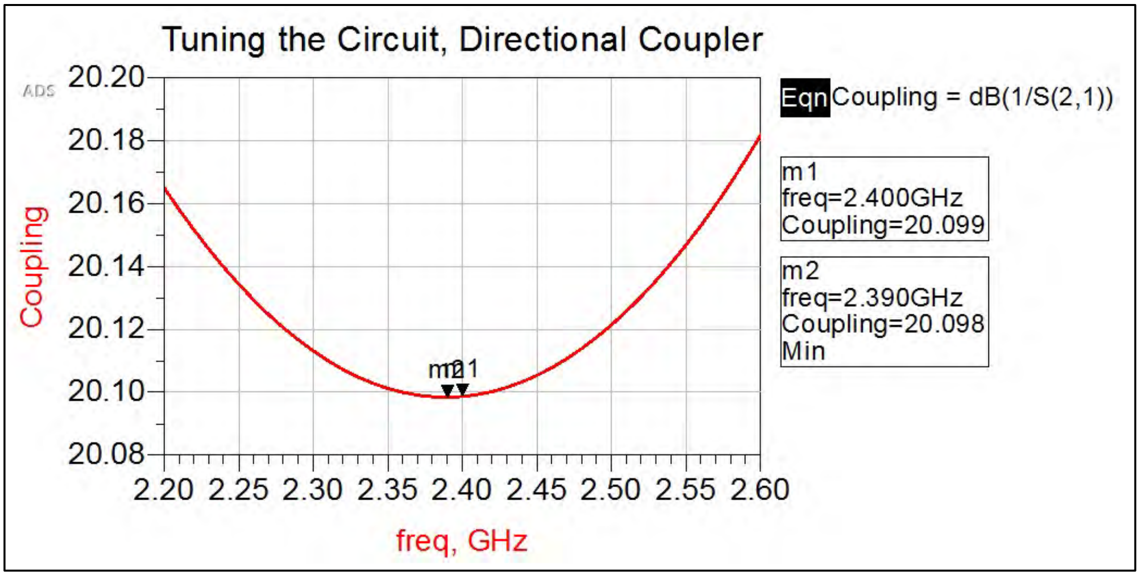


Figure 7-26. The frequency response of the microstrip circuit.

The current frequency response of the microstrip circuit gives a 20.099-dB coupling response at 2.4 GHz (Figure 7-26). The center frequency is almost exactly as designed, and the coupling factor error is negligible. If desired, the circuit can be tuned to shift the frequency response over 10 MHz, but the error is not large.

Step 3: Generate the layout mask

Follow the ADS Mask Layout procedure in Appendix A. Make sure the steps and bends are connected perfectly. The mask for the coupler will look similar to Figure 7-27.

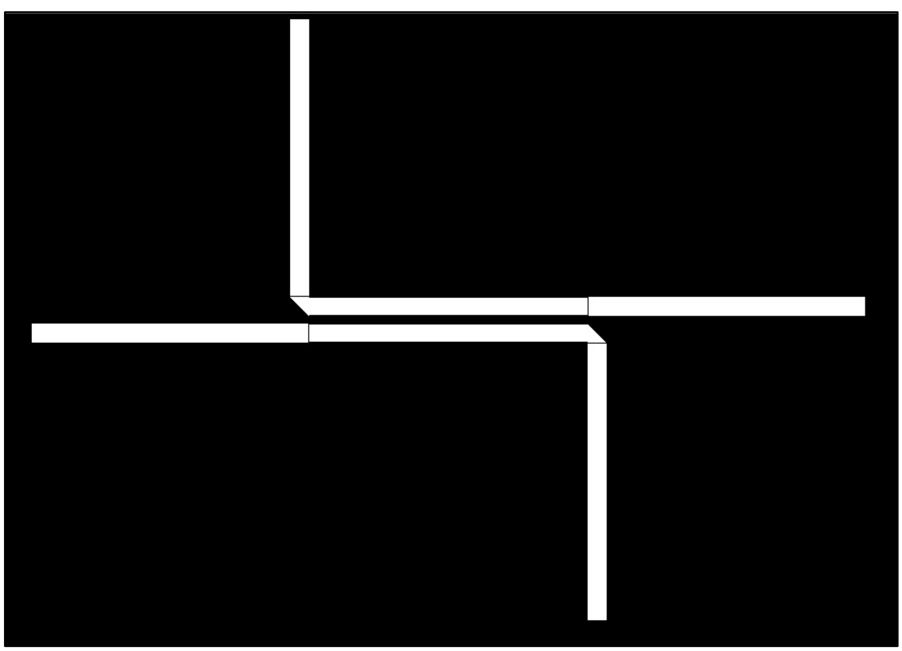


Figure 7-27. The mask of the coupler.

Step 4: Fabricate the circuit
Fabricate the circuit using the Circuit Fabrication Technique in Appendix B. With the short spacing between coupling sections, it may be required to use a scalpel to verify the separation of the conducting layer. The coupling between the lines will not work correctly if there is a connection.

Validation

With the network analyzer properly calibrated for this lab, use it to measure the S-parameters of the circuit. Using a Connection Manager, use ADS to read data from the network analyzer into .S2P files. Recalling from Chapter 3, a Data File component can be used to simulate the behavior of the fabricated circuit in ADS.

Compare the experimental results to the design in ADS and determine the success of the fabricated circuit.

Appendix A: ADS Mask Layout

Step 1: Generate the layout

Once the schematic has been prepped, go to the Layout menu and select Generate/Update Layout.

A Layout window will appear, select OK. The Layout window should appear. The stepped-impedance, low-pass filter layout is provided in Figure A-1 as an example.

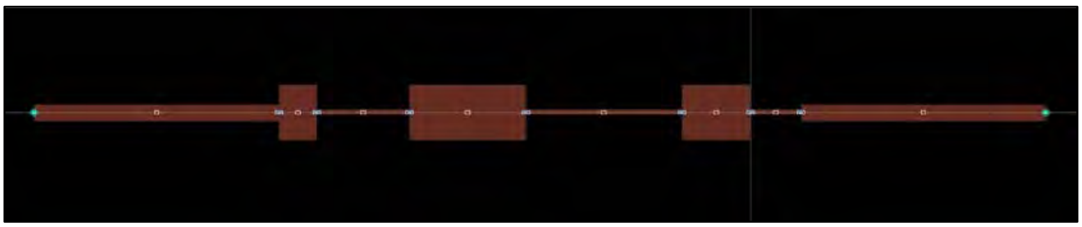


Figure A-1. Shown here is the layout for a stepped-impedance, low-pass filter.

Step 2: Verify connections

Sometimes when the layout is generated, the components are not completely connected, or they may overlap. Zoom in on each junction and confirm the sections are connected as desired. If a section needs to be moved, the snap enabled preference—which can be selected or removed through the Options menu—may be helpful.

Step 3: Draw the boundaries of the board

Make sure the v,s default: drawing preference is chosen and draw a rectangle around the circuit that represents the boundaries of the circuit board (Figure A-2). Keep the board dimensions tight to prevent wasting materials, but also leave some room for ease of handling during fabrication.

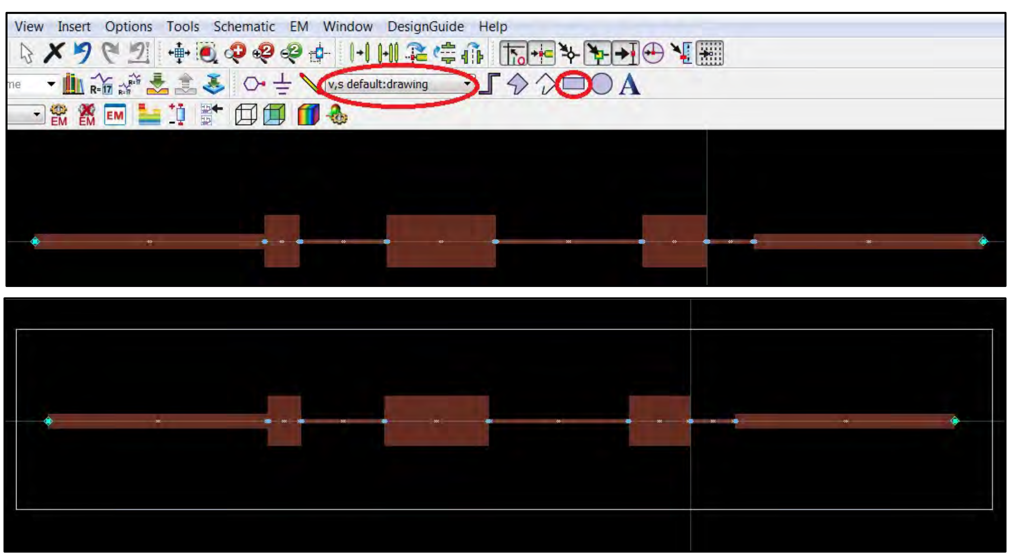


Figure A-2. In this layout, the “v,s default: drawing” preference is chosen and a rectangle is drawn around the circuit.

Step 4: Export the layout to a Gerber file

The layout is currently a brown and black color that does not help transfer the mask to the board for fabrication. The colors will be changed so that it can be printed into a transparency and used for fabrication.

Highlight the entire circuit and surrounding rectangle. Go to the File menu and select Export with a "Gerber/Drill" file type. Choose the default destination or change to what is most convenient.

Step 5: Open with a Gerber Viewer and edit

Gerbv is an easily downloadable open source Geber File Viewer and works well for this step of the layout. In Gerbv, go to the File menu and select Open layer(s). Go to the directory where the Gerber files were saved, and open both the conductor and default layers at the same time (Figure A-3).

Figure A-3. Open both the conductor and default layers at the same time.

The mask will open in the Gerber Viewer and appear similar to Figure A-4.

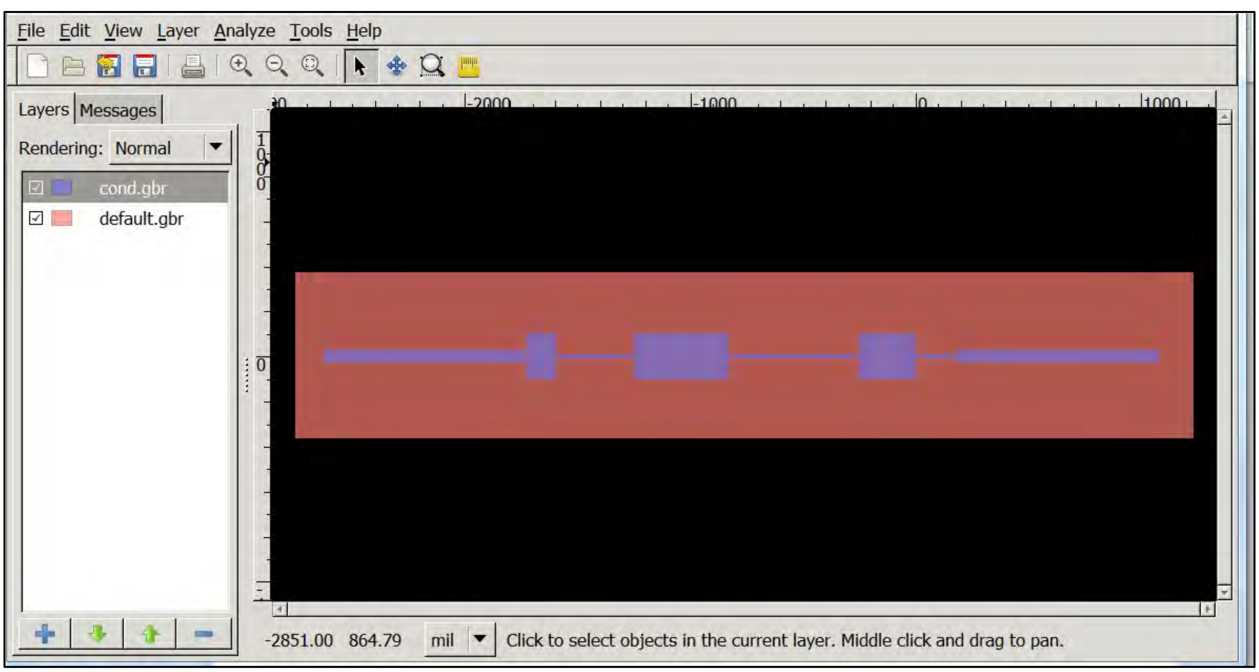


Figure A-4. The mask opened in the Gerber Viewer.

Step 6: Change the dielectric to black and the conductor to white

Right click on default.gbr and select Change Color. A window will appear that will allow the layer to be altered (Figure A-5). Drag the slider within the triangle all the way to black and maximize the Opacity.

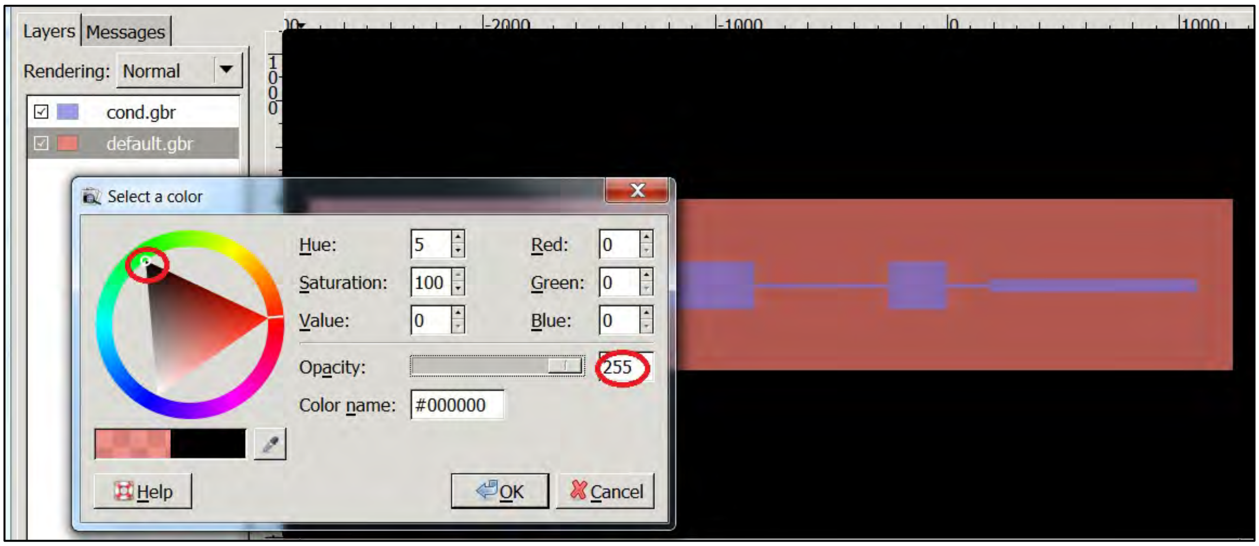


Figure A-5. Alter the layer by dragging the slider within the triangle to black and maximizing the opacity.

Right click on cond.gbr and select Change Color. Drag the slider within the triangle all the way to white and maximize the Opacity. The final mask will look similar to the Figure in A-6.

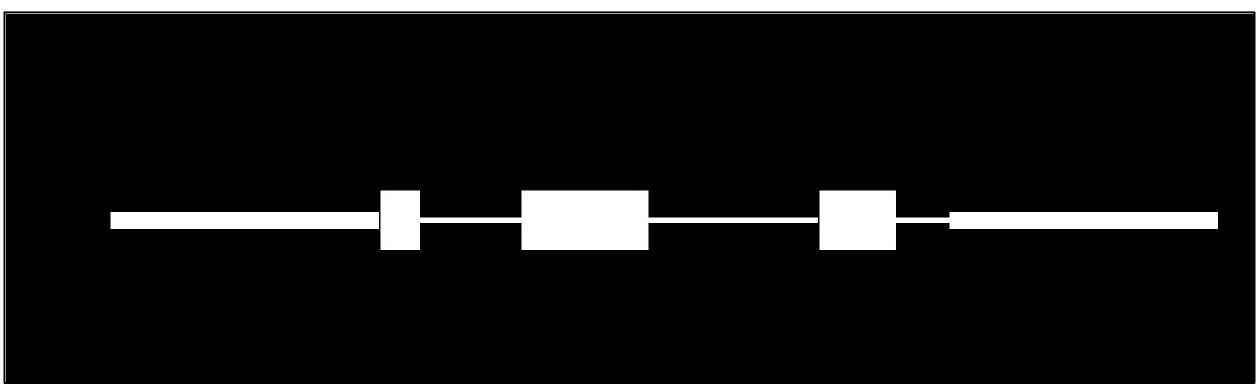


Figure A-6. The final mask.

Step 7: Print the mask

Test run the mask by printing it on a regular piece of paper. Go to File/Print and verify the circuit is without any irregularities. This test run will also aid in cutting the correct amount of laminate board for fabrication. Once the mask is verified, print it on a transparency.

Appendix B: Circuit Fabrication Technique

The fabrication procedure involves using chemicals that are light sensitive. A dark room containing only yellow light is required and the standard chemical safety procedures should be followed. This procedure is adapted from the microwave instructional program run by Professor G. R. Branner at the University of California, Davis.

Step 1: Cut the circuit board

From the test printed mask obtained in Appendix A, cut out the minimal required piece of circuit board from the larger laminate Rogers RT/duroid® 5880 board.

Step 2: Prepare the board for fabrication

Use a fine steel wool pad to gently buff and rub out any impurities on one side of the board. This will be the top conducting layer and needs to be very clean for fabrication. Handle only the bottom conductor and edges to prevent any unwanted dust, oil or other debris.

Move to the darkroom. Put on gloves to handle the board from this point forward. Rinse the buffed side of the board with propanol to further clean the surface.

Step 3: Add the photoresist layer

Use a dropper to add a uniform layer of photoresist onto the buffed side of the circuit board. If a spinner is available, it can provide the most even layer and is the recommended procedure. If not, hold one corner of the board and let the photoresist drip down evenly over the surface. Avoid adding more than one layer of photoresist, as this will cause uneven etching. If the layer must be redone, go back to Step 2.

Step 4: Bake the circuit

Place the circuit into a convection oven for 60 minutes. Bake at a temperature that is best for the type of photoresist and laminate board used, but typically around 120°C is acceptable. Leave the oven door cracked slightly to allow air flow and adjust for any errors in the temperature setting. Once the circuit is baked, remove it and let it cool.

Step 5: Expose to UV light

Take the transparency mask and place it directly onto the surface of the circuit board with the photoresist layer. Place the circuit board face down onto a UV Light source and expose it for 3 minutes. The exposure time is dependent on the thickness of the photoresist layer and the intensity of the UV light source. Generally, 2.5 to 3 minutes is acceptable. This step is vital in transferring your circuit design to the board. The exposed photoresist will become polymerized and make it more difficult for the material to dissolve in Step 6. Once it is complete, you will be able to see a faint outline of your circuit when looking at an angle. If you do not see any change in the surface of the circuit board, an error has occurred and you will need to go back to Step 2.

Step 6: Develop the photoresist

Place the entire circuit into a photoresist developer solution inside of a fume hood. The exposed photoresist will remain on the circuit board and the outer photoresist will be removed in the solution. The board should remain in the developer solution between 3 to 3.5 minutes.

Step 7: Prepare the board for etching

Rinse the developed board under running water to remove the developer solution. Pat dry the bottom of the board and let the top air dry. Do not touch the top of the circuit. Rather, handle it by the edges. Once the board is clean and dry, tape the bottom so that the conducting layer is completely covered. Make sure the tape is on firmly, with as little air bubbles as possible. If this step is not followed, etchant will seep between the tape and the board, and your bottom conductor will dissolve in Step 8.

Step 8: Etch the board

There are multiple options for etching solution, but this procedure uses ammonium peroxydisulfate powder, or $(NH_4)_2S_2O_8$. Mix 220 g of powder into 1 Liter of hot water to create the etching solution. Pour some solution into a heat safe container to a level that will just cover the entire circuit. Keep the etchant solution warm inside of the fume hood by placing the container on a hot plate over low heat. The etching process should take 25 to 40 minutes. The time is dependent on the strength of the etchant solution (e.g., how many times it has been used) and the size of the board. Multiple boards can be etched together, but it will take longer. Gently agitating the etchant container will help the top conductor layer etch away.

Once the unwanted conducting layer is etched away, rinse the board under running water and remove the bottom layer of tape. The board is now fabricated and can be exposed to regular light.

Step 9: Prepare the board for testing

The board will be tested using a network analyzer and will need to be connected to the instrument via probes. Cut of any excess board between the edge of the board and the 50-Ohm line ports. The 50-Ohm line should be flush up against the edge. Using the mounting blocks as a guideline, drill holes at each port to set the mounting blocks for the microwave connectors. Finally, using a soldering iron, solder the connector tabs down to the board. The board is now ready for testing.

Download the ADS example book in PDF format

- Getting started with Advanced Design System
- Tuning and optimization
- Harmonic balance (HB) simulation
- Planar electronmagnetic (EM) simulation in ADS
- RF system design
- Microwave discrete and microstrip filter design
- Discrete and Microstrip coupler design
- Microstrip and CPW power divider design
- Microwave amplifier design and Smith chart utility for Z matching network

www.keysight.com/find/eesof-ads-rfmw-examples

Connect with Us in Keysight EEsof EDA Community Blog:

https://community.keysight.com/community/keysight-blogs/eesof-eda

Start a 30-day free trial of Keysight ADS:

www.keysight.com/find/eesof-ads-evaluation

“How-To” Video Series

Download the accompanying example workspace to get a jumpstart on your design.

http://www.keysight.com/find/eesof-how-to-videos

ADS Basics Playlist

In addition to the basics, you can learn more about RF/Microwave and layout design.

www.keysight.com/find/eesof-ads-tutorial-videos

Learn more at: www.keysight.com

For more information on Keysight Technologies’ products, applications, or services, please contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus