School of Engineering & Information Technology

UNSW Canberra

ZEIT 3220 Engineering Electromagnetics

Lab #1: Transmission Lines and Matching

Dr Andrey S. Alenin and Prof J. Scott Tyo

Session 1, 2021

Aim

These laboratory activities guide you through an investigation of transmission line circuit behaviour. This is achieved through a comparison of theoretical performance predictions for various transmission line circuits with the simulation and measurement results for these circuits. Transmission lines are studied from both the time and frequency domain perspectives. A design activity is included to illustrate one of the practical applications of transmission lines. These lab activities are designed to help you consolidate your understanding of the behaviour and properties of transmission line structures, and to demonstrate their application to real world problems.

Overview

We live in a world where technology has stimulated enormous change in many aspects of our lives. People are connected to each other and to sources of information to an unprecedented extent, and this had led to rapid changes in social, political, business and security on a global scale. Underpinning this technological revolution are high-speed electronic circuits, where the term high-speed implies the rate of change of the voltages and currents in these circuits occurs on time scales comparable to the propagation delay of these electrical disturbances from one part of the circuit to another. Such circuits are considered as distributed structures, and design processes and procedures need to be adapted to take the distributed nature of the circuit into account. Transmission lines are an example of such a distributed circuit that has been engineered to maximise the efficiency of transport of an electrical signal from input to output, with a predictable frequency dependent behaviour.

ADS Training

ADS is a commercial CAD tool produced by Agilent Technologies (now Keysight Technologies). ADS is used extensively in this series of lab exercises. To assist you in learning how to use ADS, there is an optional ADS training activity that you can complete in Week 1 (22 and 25 Feb) using the computers in the assigned Rooms 114/116, Bld 16.. The ADS Tutorial is available on Moodle, along with the ADS design file. The lab itself starts in Week 2 (1 and 4 March).

Lab 1A — Time Domain and Frequency Domain Simulation

In this lab activity you will use the ADS CAD tool to observe the voltages at the input and output of a lossless coaxial transmission line when a step voltage source is used to excite the transmission line, and then compare the simulated values to your calculated values for the input and output voltages. The specific learning outcomes associated with this lab are:

1. gain proficiency with ADS for schematic capture, transient simulation and data interpretation;
2. calculate the reflection coefficients at the input and output planes;
3. calculate the time dependent input and output voltages in response to a step excitation;
4. gain proficiency in exporting ADS data for processing with a Matlab script;
5. gain proficiency with ADS S-parameter simulations and data interpretation.

You will make use of the following resources:

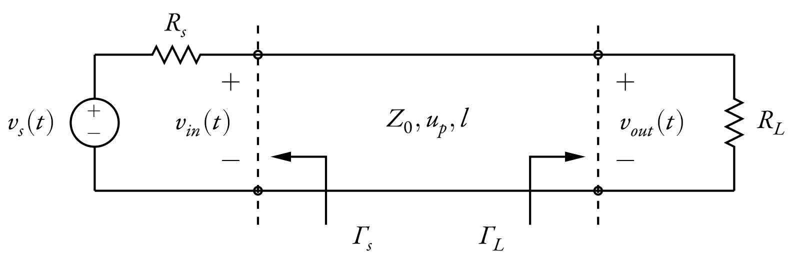
- ADS CAD Software (Keysight Technologies), as installed on the PCs in the 114/116 labs;
- Matlab.

Pre-Lab

Refresh your knowledge of the bounce diagrams from Unit 1 Lecture 1.

Narrative

Figure 1 illustrates a transmission line connected to a voltage source with source resistance R_s and load resistance R_L. For the case where these resistance values are different to the value of the transmission line characteristic impedance, Z_0, non-zero reflection coefficients Γ_s and Γ_L will exist at the input and output ports, respectively. In general, the input and output voltages $v_{in}(t)$ and $v_{out}(t)$ will be determined by the summation of incident and reflected waves travelling in either direction along the transmission line.



$$
\begin{aligned}
Z_0 &= 50\ \Omega \\
u_p &= 0.66c \\
l &= 198\text{ m} \\
R_s &= 20\ \Omega \\
R_L &= 200\ \Omega \\
v_s(t) &= 1.0\ V \text{ for } t \geq 0; \\
&= 0 \text{ otherwise}
\end{aligned}
$$

Figure 1: Lossless transmission line of length l excited by a source $v_s(t)$ with source impedance R_s, terminated in load resistance R_L. Z_0 and u_p are the transmission line characteristic impedance and phase velocity respectively.

Figure 2 illustrates the schematic for the ADS simulation circuit used for the time domain part of the lab activity. The transmission line is modeled using the ADS *COAX_MDS* simulation component, available on the *TLines-Ideal* palette. The dimensions a and b, and the relative dielectric permittivity $\varepsilon_r = 2.3$ produce a $Z_0 = 50\ \Omega$ transmission line with a phase velocity of $c/\sqrt{\varepsilon_r} = 0.66c$, where c is light speed. The length of the line is chosen such that the delay from input to output is 1.0μs. A step voltage source *Vtstep* (*Sources-Time Domain* palette) is used to excite the transmission line, with ideal resistors $R1$ and $R2$ used to simulate the source and load resistances. Note that the voltage source is configured for a 1.0μs delay relative to the simulation start time ($t = 0$). The parameter definition *TxLine_var* contains the various model parameter values such as the transmission line model parameters $a, b, tbraid, len, er$ and resistances *Rg* and *RL*. The simulation is controlled by the transient simulation controller *Tran1*, and runs from time $t = 0$ to $t = 10.0\mu$s.

Now, use an S-parameter simulation to simulate the transmission line structure. Refer to Figure 4 for the setup. Note the S-parameter simulation controller, *S_Param*, available on the *Simulation-S_Param* palette. A frequency range from 40MHz to 60MHz is selected

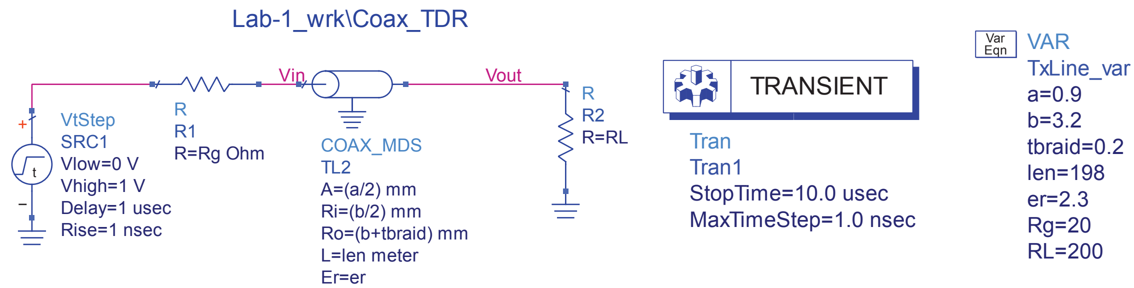


Figure 2: *Left:* ADS transient simulation schematic for an ideal transmission line excited by a step voltage source with source resistance R_g, terminated in a resistance R_L; *Right:* transient simulation controller and variables definition.

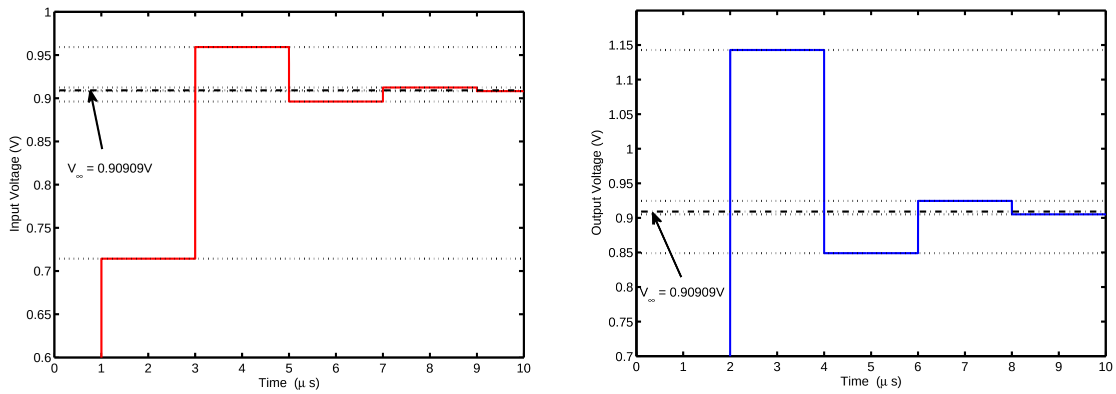


Figure 3: *Left:* Input and *Right:* output voltages produced by an ADS transient simulation of the schematic in Fig. 2. Configuration: 1 V step after a 1 μs delay, $Z_0 = 50\ \Omega$, $R_g = 20\ \Omega$ and $R_L = 200\ \Omega$.

for these simulations. Rather than explicitly using a source component, the S-parameter simulations use *Term* components to terminate the ports of a circuit, thereby automatically providing a 50 Ω load impedance and implied excitation. Simulation results appear in the data display as $S(i, j)$ variables where i, j are the port numbers (signal flow from j to i). Note that in this simulation only one port is required, hence the simulation output is a frequency dependent array of complex $S(1, 1)$ values over the simulated frequency range. Figure 5 illustrates these simulation results, plotted on a polar graph with Smith chart coordinates.

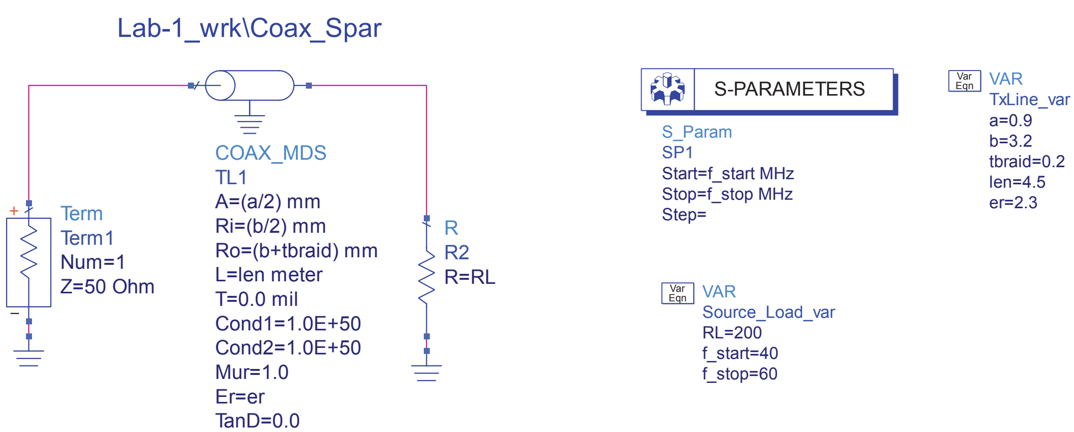


Figure 4: ADS S-parameter simulation schematic for a coaxial transmission line with inner and outer conductor diameters $a = 0.90$ mm and $b = 3.20$ mm and insulation permittivity of 2.30, terminated in a load resistance of 200 Ω resistor.

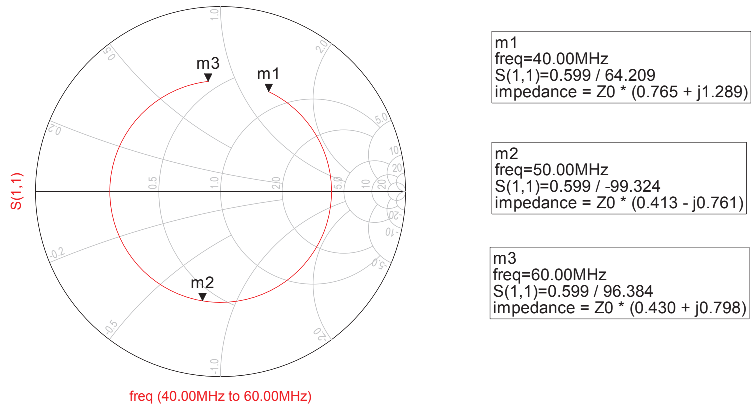


Figure 5: ADS S-parameter simulation results for the circuit in Fig. 4 for a frequency sweep from 40 MHz to 60 MHz. Markers indicate the S_{11} values at 40 MHz, 50 MHz and 60 MHz. [Length = 4.5 m, $R_L = 200\ \Omega$]

Lab Activity

1. construct the simulation circuit shown in Figure 2;
2. run the simulation, create separate plots of the input and output voltages in the data display, then place markers on the various voltage steps to accurately determine the voltage step values;
3. from the Data Display export the results to a text file (*File > Export > Write selected item to tab-delimited ASCII* so that you are able to plot the results using Matlab (see Figure 3).
4. change the simulation of the 4.5 m long transmission line circuit to that of Figure 4;
5. run the simulation, then in the Data Display create separate plots of S_{11} in polar and Smith chart formats; place markers on each trace and observe the values as a function of frequency;

Exercise 1

Determine the source and load reflection coefficients. Calculate $v_{in}(t)$ and $v_{out}(t)$ after two bounces.

Exercise 2

Study the features of the input and output voltages (ie. step values and transition times) to firstly explain to yourself the following questions, then discuss with a Lab Demonstrator:

- why are the voltage steps offset in time?
- why are the steps alternating in sign?
- how long will it take for v_{in} and v_{out} to reach equilibrium?

Exercise 3

Plot the simulated S_{11} values for the 4.5 m cable from 40 MHz to 60 MHz.

Lab 1B — Frequency Domain + Dielectric Coefficient

In this lab activity you will examine and compare the theoretical, simulated and measured frequency dependent behaviour of a short length of transmission line. The specific learning outcomes associated with this lab are:

1. gain proficiency with Network Analyser measurements of a transmission line structures;
2. gain insight into real world transmission line behaviour through comparison of the theoretical, simulation and measured results;
3. develop proficiency with incorporation of the material properties of transmission lines;
4. gain proficiency in estimating the dielectric coefficient of a coaxial cable.

In this lab activity you will make use of the following resources and equipment:

- ADS CAD Software (Keysight Technologies), as installed on the PCs in the 114/116 labs;
- Matlab.
- uVNA Vector Network Analyser
- Three selected lengths of 50 Ω RG58 cables.

Pre-Lab

Write a piece of code that can calculate values for the reflection coefficient Γ_{in} at the input of an arbitrary length cable (default of 4.5 m long), 50 Ω lossless transmission line having an arbitrary dielectric coefficient (default of 2.3), when terminated in a 200 Ω load impedance. Plot the sweep from 40 MHz to 60 MHz and denote 40 MHz, 50 MHz and 60 MHz on the Smith chart.

Narrative

Figure 6 illustrates a generic transmission line circuit, where a sinusiodal voltage source with source impedance Z_s excites a transmission line terminated in an impedance Z_L. The transmission is described by its characteristic impedance Z_0, phase velocity u_p and physical length l. For a lossless transmission line Z_0 and u_p are constant values, ie. independent of frequency.

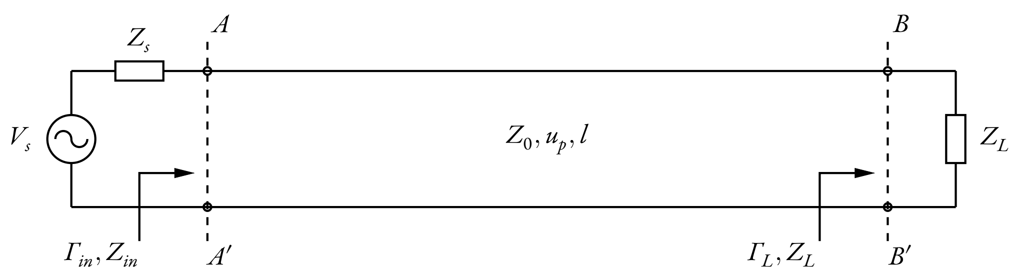


Figure 6: Transmission line of length l excited by a source V_s with source impedance Z_s, terminated in load Z_L. Z_0 and u_p are the transmission line characteristic impedance and phase velocity respectively. The load Z_L is located at position B, B' while the source connects at the input position A, A'. The load impedance is transformed to input impedance Z_{in} by the transmission line.

For measurements you will use the uVNA Network Analyser (see Appendix B). Your Lab Demonstrator will guide you in the setup and use of this equipment. Note that as the School has only five of these units, you will need to share access with other students. While you are waiting for access to one of the uVNAs, you can focus on the ADS simulations.

Lab Activity

1. measure the physical length of your cable #1 (carefully note its serial number), then use the uVNA to measure S_{11} for the cable when terminated in the 200 Ω load; save these results to a .csv file;
2. edit the ADS schematic to change the length of the cable from 4.5 m to your measured value (ie. the *len* variable)), then rerun the simulation;
3. from the Data Display export the results of the polar plot to a text file (*File > Export > Write selected item to tab-delimited ASCII*;
4. write a Matlab script to (a) read the ADS and measured data files, (b) calculate $\Gamma_{in}(\omega)$, (c) compare all three on polar and cartesian plots (eg. see Figure 7); How closely to they compare?
5. use the uVNA to measure S_{11} for cables # 2 and #3 when terminated in the 200 Ω load; save these results to a .csv file;
6. calculate ε_r for cables #2 and #3

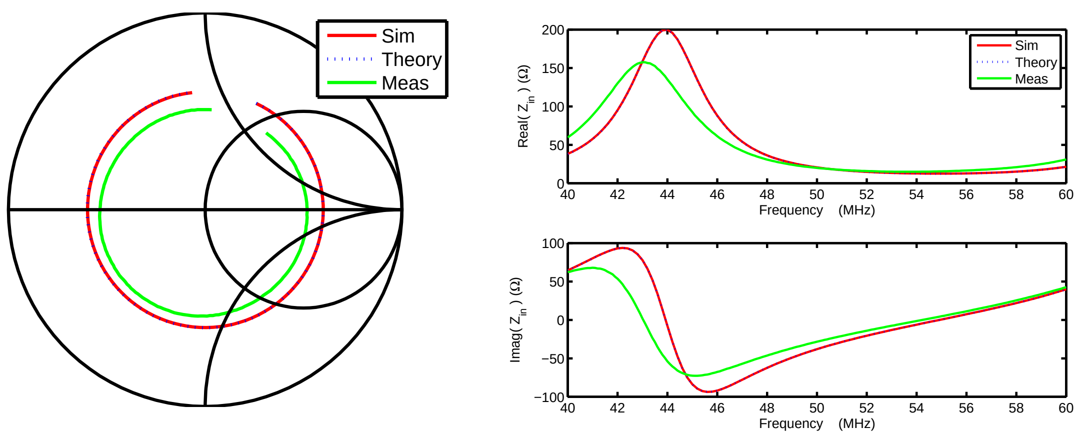


Figure 7: *Left:* Comparison of (i) theoretical Γ_{in}, (ii) ADS simulated S_{11} and (iii) measured S_{11} for the transmission line circuit of Fig. 4; *Right:* comparison of corresponding input impedances Z_{in}. [Length = 4.5 m, R_L = 200 Ω]

Exercise 4

Observe the measured and simulated S_{11} values for your particular cable, write down the values at 40 MHz, 50 MHz and 60 MHz, develop explanations for any differences you observe, then discuss with a Lab Demonstrator.

Exercise 5

Observe the measured S_{11} values for the two new cables. Using the values at 40 MHz, 50 MHz and 60 MHz, estimate the value of ε_r. Compare your results for the three cables. Develop explanations for any differences you might see in the results, then discuss with a Lab Demonstrator.

Exercise 6

Develop explanations as to why it might be advantageous to go through the process of estimating the value of ε_r. What is it that you are verifying?

Lab 1C — Matching Networks

In this lab activity you will design stub and quarter-wave transformer matching circuits to match a measured load impedance at a specified frequency (98 MHz), then compare how well these solutions perform over the frequency range of interest (88 MHz to 108 MHz, ie. the FM broadcast band). Due to time and cost constraints, you are not required to actually fabricate these matching circuits, rather just simulate their performance. The specific learning outcomes associated with this lab are:

1. gain expertise with design of stub matching networks;
2. gain expertise with the design of quarter wave transformer matching networks;
3. simulate stub and quarter wave transformer matching network circuits using ADS;
4. use theory to calculate the frequency response of stub and quarter wave transformer matching networks;
5. consolidate your expertise with transmission line theory, ADS simulations and post-processing (eg. using Matlab) of measured and simulation results to identify and articulate experimental outcomes.

In this lab activity you will make use of the following resources and equipment:

- Matlab;
- uVNA Vector Network Analyser;
- lumped element RLC networks mounted on BNC connector.

Pre-Lab

Reflect on your results for Labs 1B. Start developing explanations as to why the measured S_{11} results might differ from ADS simulations and theoretical predictions. Think about how these might affect the matching network procedure in Lab 1C.

Narrative

Figure 8 illustrates a transmission connected to a load Z_L through a matching network. Since the source impedance is the same as the transmission line characteristic impedance (ie. Z_0), maximum power will be transferred from the source to the load when the matching network presents an impedance of Z_0 to the transmission line. The role of the matching network can then be interpreted as transforming the load impedance Z_L to Z_0.

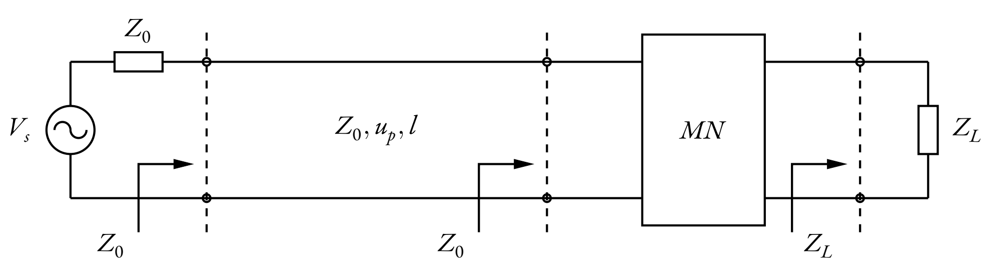


Figure 8: Matching network MN that transforms load impedance Z_L to Z_0, thereby presenting a matched load to the transmission line and source.

Lab Activity

1. select one of the supplied load impedances, and note it's serial number;
2. use the uVNA to measure the reflection coefficient of your load impedance; save these results to a .csv file for subsequent processing;

3. from your measurements identify the reflection coefficient value at a frequency of 98 MHz, then calculate the corresponding load impedance;
4. apply the stub matching procedure to find the lengths of the series and stub lines to match the 98 MHz load impedance to 50 Ω; use ADS to verify the performance of your stub matching solution;
5. repeat Step 4 to find the solution for a quarter wave transformer match;
6. write a Matlab script to calculate the input impedance Z_{in} as a function of frequency for both the stub and quarter wave transformer matching networks (eg. see Figure 7); how closely to they compare?

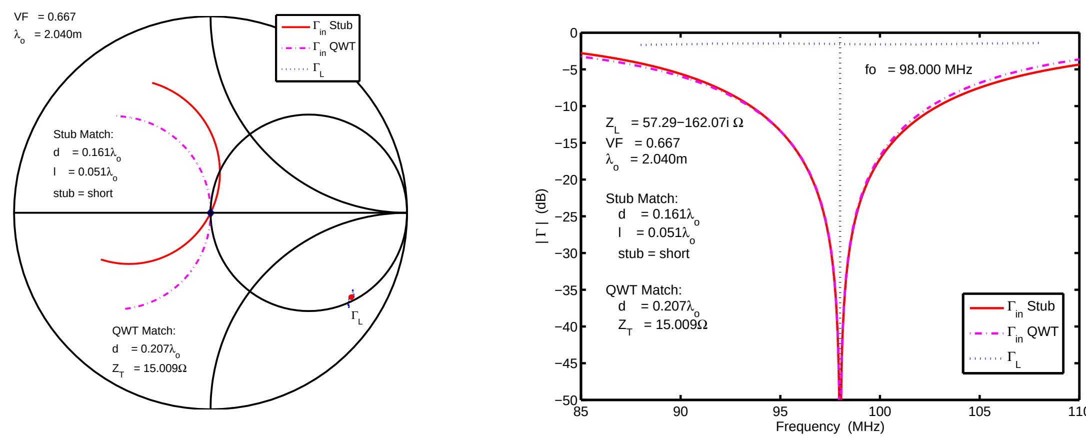


Figure 9: Stub matching network solution: measured load reflection coefficient (dashed blue traces); matching network solution (solid red traces) for the matching solution at the mid-frequency position (red asterisks); *Left:* complex valued Γ_L, Γ_{in}, *Right:* magnitude responses.

Exercise 7

Determine the load impedance of the unknown load.

Exercise 8

Determine the parameters for the shunt stub matching network. Verify its performance in ADS by simulating the proposed MN.

Exercise 9

Determine the parameters for the quarter wave matching network. Verify its performance in ADS by simulating the proposed MN.

Exercise 10

The frequency range where the magnitude of the reflection coefficient is such that the VSWR is less than 2 is often used to quantify impedance bandwidth. Observe your simulated frequency response results for the reflection coefficient at the input of the stub and quarter wave transformer matching networks. Which matching network achieves the wider impedance bandwidth? What are the 2:1 impedance bandwidths for each matching network? [*hint:* calculate $|\Gamma| = |S_{11}|$ for $VSWR = 2$.] Develop an explanation as to why a particular matching network provides the better frequency response.

A The Vector Network Analyser

Low frequency measurements of electronic circuits are typically carried out using an oscilloscope and function generator. This approach cannot be used at high frequencies, due to transmission line effects and device stability considerations. Instead, circuit components are grouped together into functional blocks (devices) and characterised in terms of the voltage/current/power relationships at the input and output ports. For most applications, 1 or 2 ports are sufficient, although 3 or more port devices are available. The electrical performance of the $n-$port device is measured by applying a signal to one port and measuring the response at this and all of the remaining $n-1$ ports.

At high frequencies S-parameters are used to characterise an n-port device, rather than z, y, h or other port parameters. A Vector Network Analyser is used to make S-parameter measurements on multi-port devices. This instrument consists of a signal generator, couplers, receiver and display device, usually in one piece of equipment. Magnitude and phase are measured, allowing complete characterisation of the device. Note that 'vector' refers to the measurement of signal vectors (ie. amplitude and phase of the S Parameters), rather than spatial vectors. Internally the instrument consists of a signal generator, signal routing circuitry, a receiver/detector, and a processor/display section. Fig. 10 illustrates a 2-Port VNA architecture. The source operates over a wide range of frequencies. The

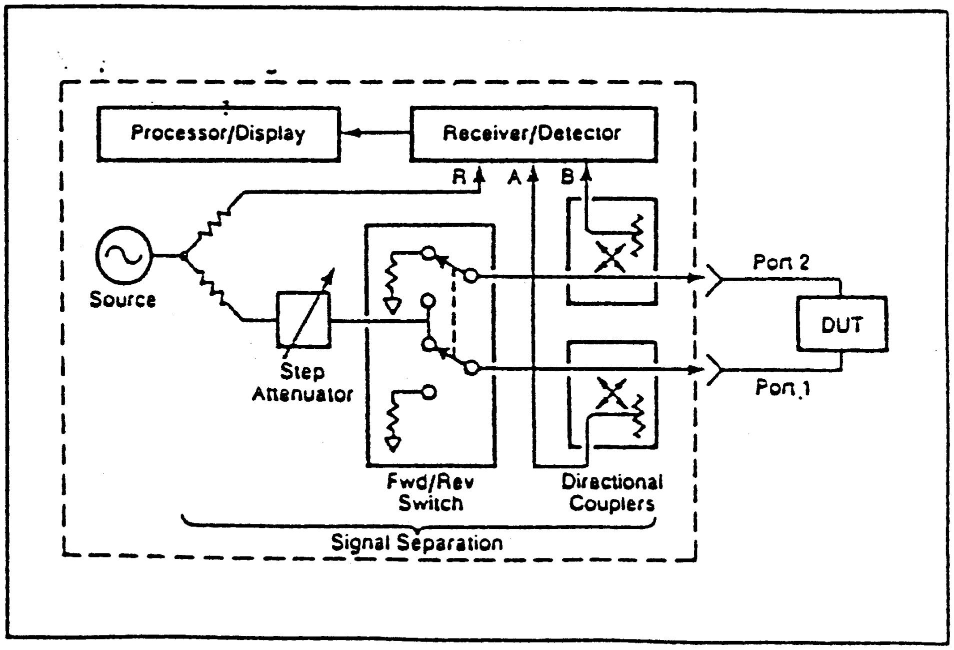


Fig A1.1 Network Analyser Architecture

Figure 10: Vector Network Analyser Architecture (ref: HP 8719A Documentation)

signal routing circuitry sets the output signal level, switches the output to appear at Port 1 or Port 2 of the VNA, and provides coupling of reflected signals from Port 1 and 2 to the receiver. The Receiver/Detector has 3 input channels (A, B and R where R is the reference channel) and forms the ratios A/R and B/R. The magnitude and phase of these ratios represent the raw measurement of the transmitted or reflected signal at Port 1 or 2, and are fed to the Processor/Display for error correction and display manipulation.

Error correction is necessary to remove the errors induced in the measurement due to coupler frequency response, finite coupler directivity and source mismatch. For a 1 Port measurement 3 error correction terms are required, whereas a 2 Port measurement requires 12 error correction terms. These error corrections are applied automatically by the instrument. A calibration procedure to determine these error correction terms is carried out before measurements are made. The error correction terms are stored in the instruments internal storage registers along with the instrument configuration.

B The Htech uVNA Vector Network Analyser

For this lab activity, a Htech uVNA Vector Network Analyser (VNA) will be used. This instrument instrument consists of a hardware unit (Figure 11) and virtual instrument software for user interface and instrument operation. The uVNA can display measured S_{11} or S_{21} data over a frequency range from 500 kHz to 120 MHz.

Figure 11 illustrates the physical appearance of the uVNA device, and a screen shot of the uVNA interface.

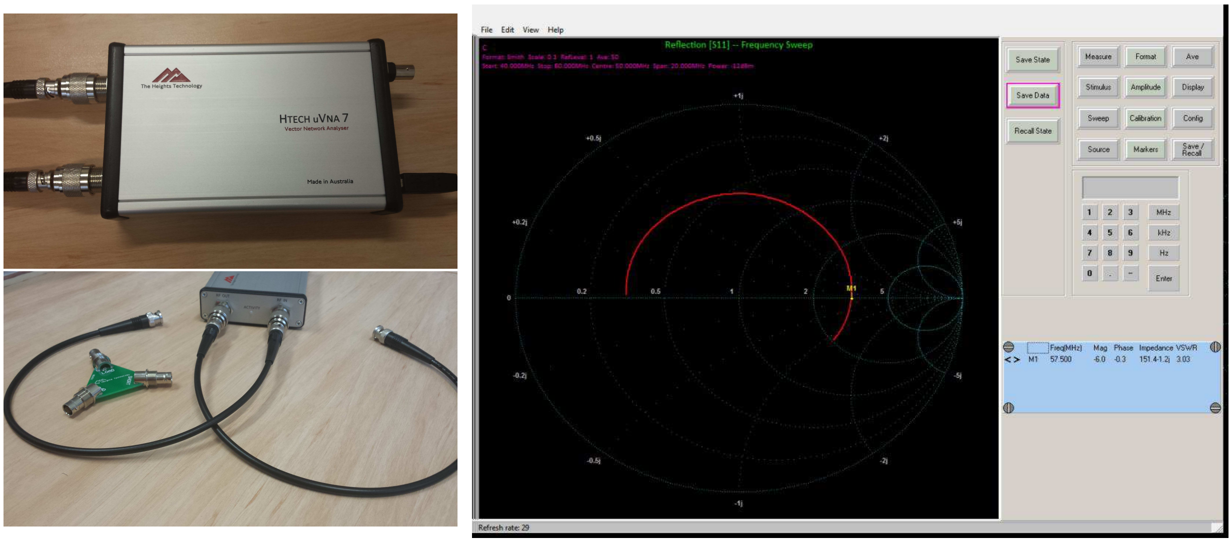


Figure 11: *Left:* Views of the uVNA hardware device, illustrating the two connection ports and calibration tool; *Right:* Screenshot of the uVNA virtual instrument display, configured for a reflection measurement (S_{11}) with a Smith chart format to display the measured data.

The uVNA is operated as follows:

- connect the N-BNC adapters to Port 1 and Port 2, then connect the two 0.5 m BNC cables;
- connect the device to the PC using the USB cable;
- launch the uVNA application; verify operation by observing a flashing indicator on the uVNA device;
- select the measurement mode (Reflection or Transmission), then select Calibration to calibrate for the selected operation mode (follow the on-screen prompts);
- connect to the device under test and adjust the amplitude scale and reference level to achieve the desired display.

C S Parameters and the Smith Chart

High frequency devices are often characterised by their Scattering or S Parameters. Suppose a device has two ports, 1 and 2. The device might be a transmission line, and amplifier or a filter. The VNA is designed for devices with input and output transmission lines impedances of 50 Ω.

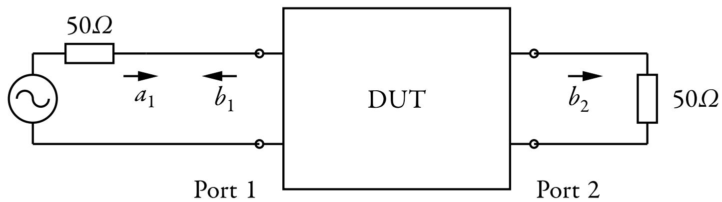


Figure 12: Two Port Device Measurement

Consider a 2 Port device as shown in Fig. 12. Port 2 is matched (ie. connected to a 50 Ω load, and the voltage source at Port 1 has an internal impedance of 50 Ω). A signal a_1 flows into the device under test (a_1 may be a voltage or current). A signal b_1 is reflected and a signal b_2 is transmitted and appears at the output. We can write:

$$S_{11}(\omega) = \frac{b_1}{a_1} \tag{1a}$$

$$S_{21}(\omega) = \frac{b_2}{a_1} \tag{1b}$$

In general the S Parameters vary with frequency and are shown in these equations to be functions of ω. An obvious interpretation is that S_{11} is the reflection coefficient and S_{21} is a transmission coefficient or transfer function. Instead of the arrangement of Fig. 12, we might have connected the matched 50 Ω load to Port 1 and the signal generator to Port 2. In this case a signal of amplitude a_2 would flow into Port 2 and signal b_2 would be reflected off Port 2, with signal b_1 being transmitted in the reverse direction out of Port 1. The most general case is where signal a_1 and a_2 are simultaneously applied to both ports, each contributing to outwardly flowing signals b_1 and b_2. We can express the outputs b_1 and b_2 in terms of the inputs a_1 and a_2 and the four S Parameters as:

$$b_1 = S_{11}a_1 + S_{12}a_2 \tag{2a}$$

$$b_2 = S_{21}a_1 + S_{22}a_2 \tag{2b}$$

If a two port device is passive (ie. no internal power source) and satisfies reciprocity, then the S Parameters are equal to their corresponding transposes:

$$S_{11} = S_{22} \tag{3a}$$

$$S_{21} = S_{12} \tag{3b}$$

Further, if the two port is lossless and matched at the input and output, conservation of energy requires that the input power equal the sum of the output power from all ports, that is:

$$|S_{11}|^2 + |S_{21}|^2 = 1 \tag{4a}$$

$$|S_{22}|^2 + |S_{12}|^2 = 1 \tag{4b}$$

The phase velocity of an electromagnetic wave along a coaxial transmission line (coaxial cable) is determined by the relative permittivity ε_r of the dielectric material between the inner and outer conductors:

$$v_p = \frac{c}{\sqrt{\varepsilon_r}} \tag{5}$$

So for a coaxial cable of length L the time taken for the wave to propagate the length of the cable is $\Delta t = L/v_p = L\sqrt{\varepsilon_r}/c$.

From transmission line theory we can associate a reflection coefficient Γ_L of the load at the end of the transmission line with the impedance Z_L of this load:

$$\Gamma_L = \frac{Z_L - Z_0}{Z_L + Z_0} \tag{6}$$

where Z_0 is the characteristic impedance of the transmission line. If we plot the reflection coefficient on a polar diagram (ie. magnitude and phase, or alternatively imaginary vs real) we find that for passive devices all possible values of $\mathit{\Gamma}_L$ lie within the unit circle. Since a particular $\mathit{\Gamma}_L$ corresponds to a unique Z_L, we can use this polar diagram to plot the load impedance. All impedances $Z = R + jX$ where $0 \leq R \leq \infty$ and $-\infty \leq X \leq \infty$ lie within the same unit magnitude circle. It can be shown that the locus of constant resistance or constant reactance are circles on this polar diagram, having circle centers and radii given by:

$$C_r = \frac{r}{1+r} \qquad R_r = \frac{1}{1+r} \qquad \text{resistance circle} \tag{7a}$$

$$C_x = 1 + j\frac{1}{x} \qquad R_x = \frac{1}{|x|} \qquad \text{reactance circle} \tag{7b}$$

$$r = \frac{R}{Z_0} \qquad x = \frac{X}{Z_0} \qquad \text{normalised resistance and reactance} \tag{7c}$$

where C_r and C_x are the circle centers for the resistance and reactance circles respectively, and R_r and R_x are the circle radii. Such a polar diagram with a number of resistance and reactance circles is called a Smith Chart, see Fig. 13.

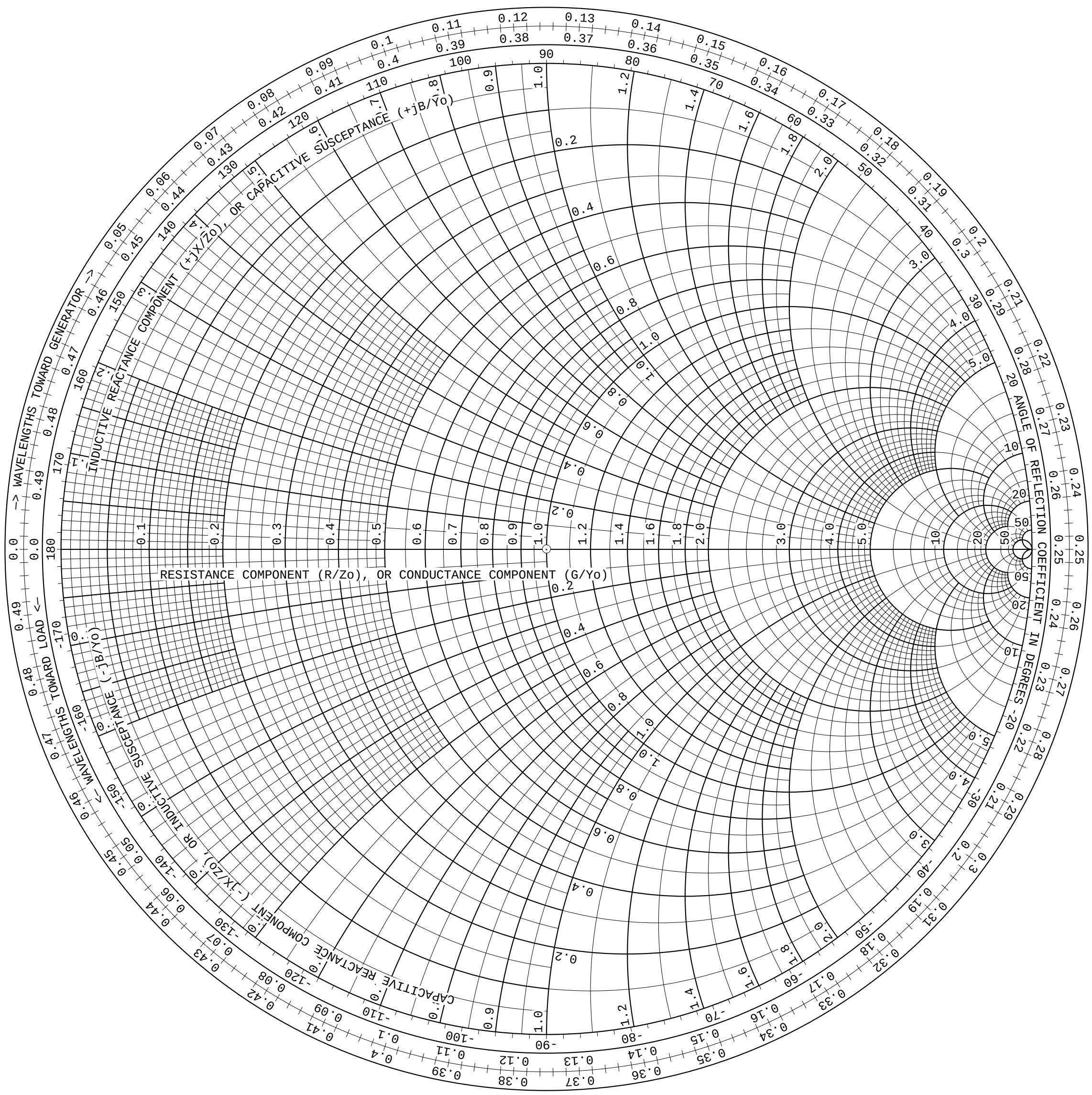


Figure 13: Smith Chart