Lab 1:

Microwave Circuit Principles and Design

EEEN60171
&
EEEN40171

Keysight Advanced Design System

Part I: ARTIFICIAL TRANSMISSION LINE

Part II: MATCHING CIRUITS

MSc (CaSP)

&

4th Year Module Option

1st SEMESTER

November 2023

MECD

Part I: ARTIFICIAL TRANSMISSION LINE

ABOUT KEYSIGHT'S ADS

Keysight's Advanced Design System (ADS) is a comprehensive communications design package. It can analyse both digital and analogue circuits. Here we will use it to build up a circuit schematic and to simulate the microwave performance of that circuit. Once satisfied, you can then provide a layout of the circuit from the schematic if necessary. An ADS circuit or system can be represented in one of three ways either as a schematic, that is as a series of network elements, ports, wires and ground points; or as a pure layout, which is a graphical picture of the circuit as laid out on a substrate or circuit board; or finally as a netlist which is a text file used by the simulator containing network node descriptions. Within ADS there are a series of powerful packages enabling linear and non-linear circuit simulation, transmission line analysis, analogue and digital filter design, digital circuit design, simulation of both digital and analogue networks and systems and finally a 2.5D planar circuit analyser.

GETTING STARTED WITH ADS

From the PC desktop window select **Start → All Programs** and then find **Keysight ADS**. Run the 64 bit simulation version if the option is given. The ADS title screen will now appear and a box titled **'Getting Started with ADS'.** Ignore the offer to **'Introduce Me to ADS 2016'** if it appears; this is primarily aimed at people that are already familiar with ADS. By all means towards the end of the course you can avail yourself of this feature but for now don't bother but click on **'Create a new workspace'**. A new workspace wizard will open, so click **'Next'** and on the next screen use a suitable workspace name such as *artlin1* and note where this workspace directory is saved, in C:\users. If you are happy with this then continue so long as you know where this is being saved – it may be under your user name. Then click next past the **Add Libraries** you don't need to worry about these just yet and then click '**Finish.**'

You should now see ADS Main Window and the workspace directory you have just created. It should look a bit like this:

Schematic window button
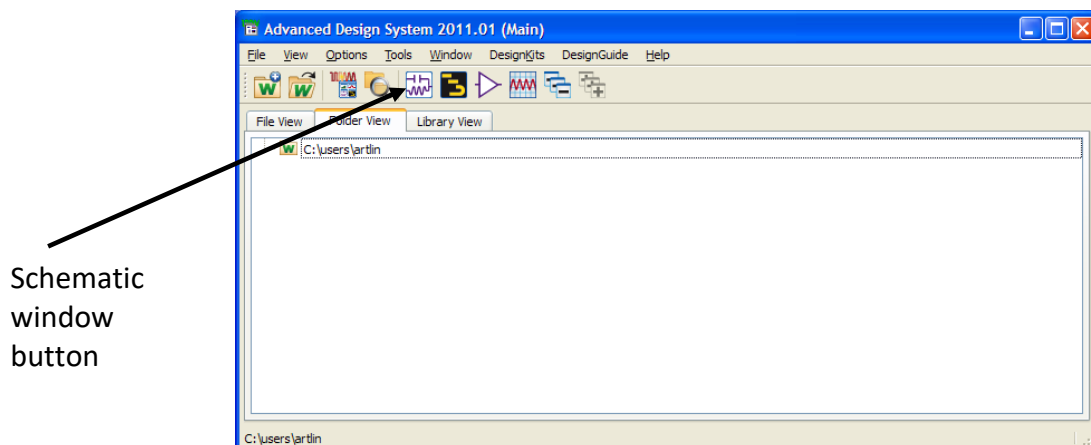
Starting a Design.

Note, when setting the dimensions for a project it's better to use millimetre or mm which should not be confused with mil which is thousandths of an inch. Now you need to bring up a schematic window. It is this window that you will work for the positioning of the components. Click on the corresponding button in the workspace window as shown above. A new window will pop up and leave it at the default entries and **cell_1** and press **ok.** Now a schematic window and wizard will pop up. Cancel the wizard window.

You should now have something like this:
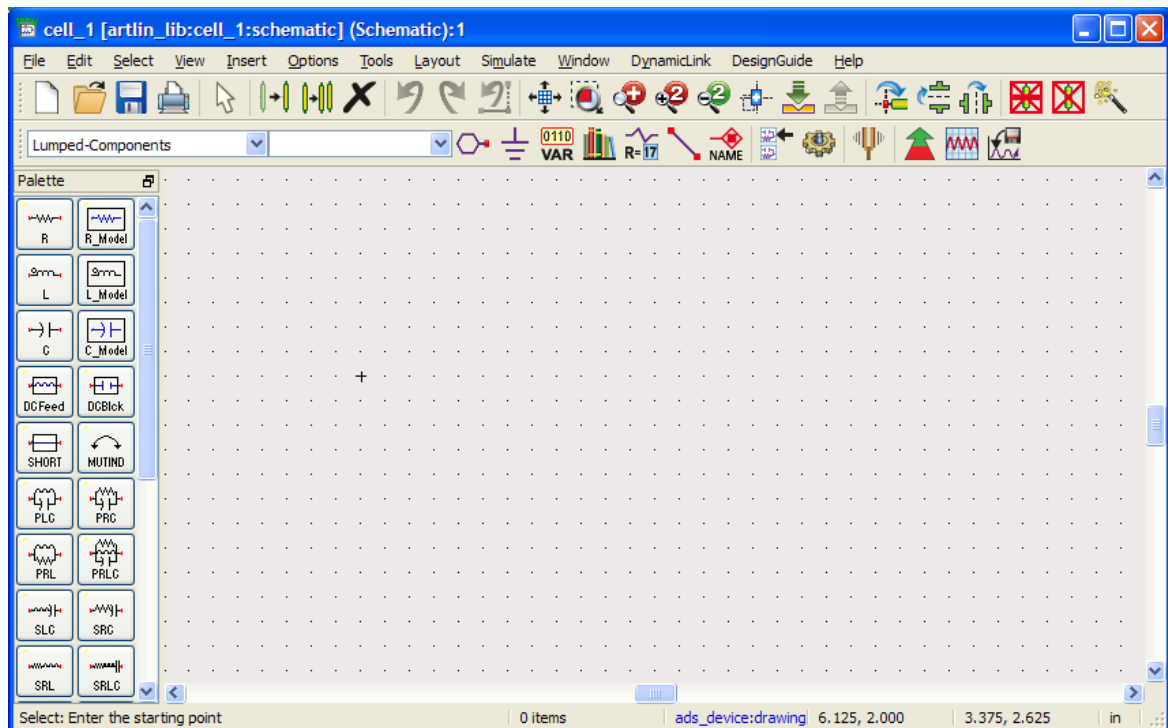
You are now all set to build up circuits and simulate them.

Aim of this Laboratory

To design and then study and confirm the behaviour of an artificial line and finally compare its performance against the structure it is attempting to model. You will perform a number of tests on the line in the time domain firstly with a steady state, ac source (simulating an oscillator source) and the second time with a step source. Each time a test will be done for varying load impedance and voltages monitored along the artificial transmission line. The line you are to model is 3,600m long and the signal of operation will be at 40kHz.

Comparisons can be made between the two methods of line analysis.

Artificial Transmission Line Theory

The line you will construct will simulate a length of transmission line 3600m long. The inductive elements are of high Q type. These would traditionally be wound on a ferrite ring and provide the series element of the transmission line and provide a typical value of 300 micro Henry.

Artificial lines are a discrete, lumped element representation of a distributed constant line. If it were possibly to split the 3600m line into an infinite number of sections then the model would be truly distributed. For a finite number of **Π** or **T** sections the line is only representative of a transmission line below a high frequency limit which you will calculate and demonstrate through simulation. By comparison with the transmission line equations, which give the impedance of any line a true representation, can be obtained at any frequency.

Part 1: AC Simulation of Discrete Transmission Line

Now once the schematic wizard pops up, **Cancel** it. You will setup the schematic test bench manually. When the schematic window pops up then save this design as **art_freq1**, or something similar. Before constructing the artificial line in the simulator you need to calculate suitable values for the series inductance L and parallel capacitance C components. So if the inductance per section is 300 micro Henries and the line impedance is to be 250Ω what is the value of parallel capacitance given that the **Π**-section structure is to be implemented?

Series Inductance per section L_s = 300µH,

Parallel capacitance per section, C_s =   nF

The electrical length per section can be calculated from the equation,

$$E_\ell = \omega\sqrt{L_s C_s}\,\text{radians}$$

Where, ω is the radial frequency of the signal (rad/s). Assuming the transmission line is air filled then the phase velocity of the applied signal is equal to the speed of light, c=3x10^8 m/s. Then the equivalent physical length of the transmission line is,

$$\ell = c\sqrt{L_s C_s}\,\text{meters}$$

Complete the box below for your calculation,

Equivalent physical length of line per section, $\ell =$ meters

Now L_s and C_s correspond to the total L and C **per section** which for the **Π**-section configuration the capacitance is split equally between two capacitors, one either side of the inductance. We will call these components L_{ser} and C_{par}. Here, L_{ser} = L_s but C_{par} is half the above calculated capacitance.

$$\therefore C_{par} = \frac{C_s}{2} = \quad \text{nF}$$

So a single **Π**-section corresponds to ℓ meters of line and is shown below. To begin with build up a single **Π**-section as shown below in Figure 1.

The capacitors, inductors and resistors (forming the loads) can be found in the Lumped Components palette and the AC simulation block in Simulation-AC and the voltage source in Sources- Freq Domain. Use the Insert Wire command on the top tool bar to wire or connect the

components.

Note you can move text associated with components around using the F5 key.

To begin with label the input voltage, **Vin**, and the output voltage across the load **Vout**, as in Figure 1 using the **Insert Wire/Pin Label** button on the tool bar. If this is not clear ask one of the demonstrators to show you what to do. Next click on the **VAR** button on the tool bar and place this in place on your schematic.
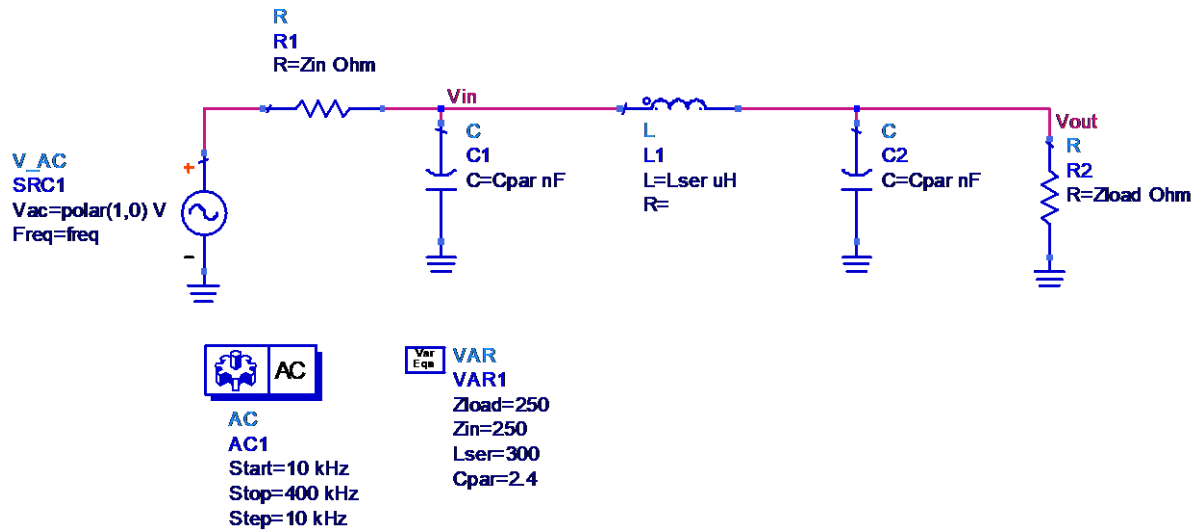

Figure 1: Single Π-section AC Simulation

The load and source impedances are set up to be the same as the transmission line characteristic impedance. Take care editing the **VAR** and component blocks make sure that the correct units of nF and μH are used for the reactive components. The generator or source is an AC source set to a frequency sweep between 10 kHz and 400 kHz. Now simulate and you should get a response something like the one below in Figure 2.
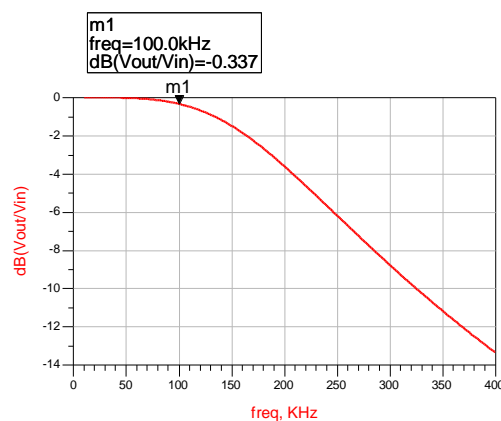

Figure 2: Frequency response of the single Π-section.

You can see from this plot that if we are to simulate a real transmission line of around 3,600m for an operating frequency of 40kHz that the response is still well below the knee due to the circuit behaving as a low-pass filter.

Now we wish to model a transmission line of length 3,600m. So how many sections will you need to

simulate the whole line and what would the corresponding wavelength at 40kHz be?

> Number of sections required to model 3,600m =
>
> Wavelength in air at 40kHz =   m

To put multiple section in your schematic copy your single section and step and repeat copy in the schematic. It should look like Figure 3 below.
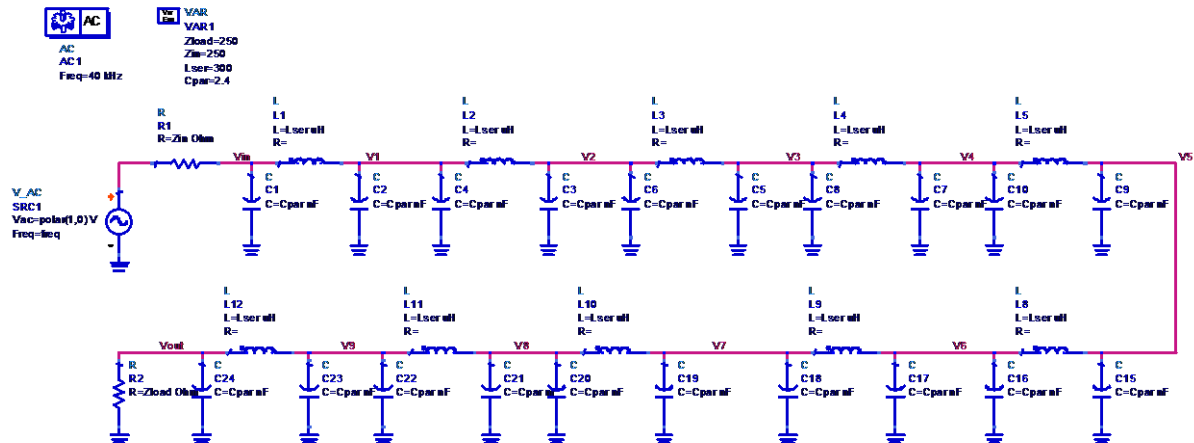

Figure 3: Multiple Π-section AC Simulation of an Artificial Transmission Line

Then label each of the shunt capacitance nodes V1,.., V9, these you will monitor to examine the standing wave behaviour of the line. Initially start with a matched load of Zload = 250 Ω and simulate at the single frequency of 40kHz. Now fill out the table below for a load of 250 Ohm.

$\lvert V_{in}\rvert$ (V)	$\lvert V_1\rvert$ (V)	$\lvert V_2\rvert$ (V)	$\lvert V_3\rvert$ (V)	$\lvert V_4\rvert$ (V)	$\lvert V_5\rvert$ (V)	$\lvert V_6\rvert$ (V)	$\lvert V_7\rvert$ (V)	$\lvert V_8\rvert$ (V)	$\lvert V_9\rvert$ (V)	$\lvert V_{Out}\rvert$ (V)

Table 1: Standing Wave Voltages at Each Node for Zload = 250 Ω

Then repeat this procedure for load impedances of 0 Ω and 10000 Ω (representing an open circuit).

$\lvert V_{in}\rvert$ (V)	$\lvert V_1\rvert$ (V)	$\lvert V_2\rvert$ (V)	$\lvert V_3\rvert$ (V)	$\lvert V_4\rvert$ (V)	$\lvert V_5\rvert$ (V)	$\lvert V_6\rvert$ (V)	$\lvert V_7\rvert$ (V)	$\lvert V_8\rvert$ (V)	$\lvert V_9\rvert$ (V)	$\lvert V_{Out}\rvert$ (V)

Table 2: Standing Wave Voltages at Each Node for Zload = 0 Ω

$\lvert V_{in}\rvert$ (V)	$\lvert V_1\rvert$ (V)	$\lvert V_2\rvert$ (V)	$\lvert V_3\rvert$ (V)	$\lvert V_4\rvert$ (V)	$\lvert V_5\rvert$ (V)	$\lvert V_6\rvert$ (V)	$\lvert V_7\rvert$ (V)	$\lvert V_8\rvert$ (V)	$\lvert V_9\rvert$ (V)	$\lvert V_{Out}\rvert$ (V)

Table 3: Standing Wave Voltages at Each Node for Zload = ∞ Ω

You should use these results to draw out the standing wave as a function of position or node on the line. Bearing in mind the wavelength of at 40 kHz do your results look reasonable?

From your knowledge of the standing wave and by changing the simulation frequency you can determine the equivalent line length. From transmission line theory when the line is quarter wavelength long and open circuit load will appear as a short circuit $\lambda/4$ away. Determine the frequency at which the artificial line is $\lambda/4$ long. Use this to determine the equivalent total line length.

Part 2: Time Domain Simulation of Discrete Transmission Line

For the second part of this CAD laboratory you will simulate the artificial line in the time domain using the time domain reflectometry (TDR) abilities of ADS. So first save your current schematic and then rename the circuit as **art_time1**, or something similar. Next build the circuit below in Figure 4.

  a. Replace the AC voltage source in Figure 3 with a voltage step, **Vtstep**, from the **Sources-Time domain** palette. Set the rise time to 1V to be 1 usec.
  b. Replace the AC simulator with the Transient simulator (working in the time domain again). Ensure the **Start time** is 0nsec; the **Stop time** is 100 usec and the maximum time step, **MaxTimeStep**, is 100 nsec.
  c. Double click on the Transient simulator and then click on the Integration tab, then select the Fixed Time Step Control Method.
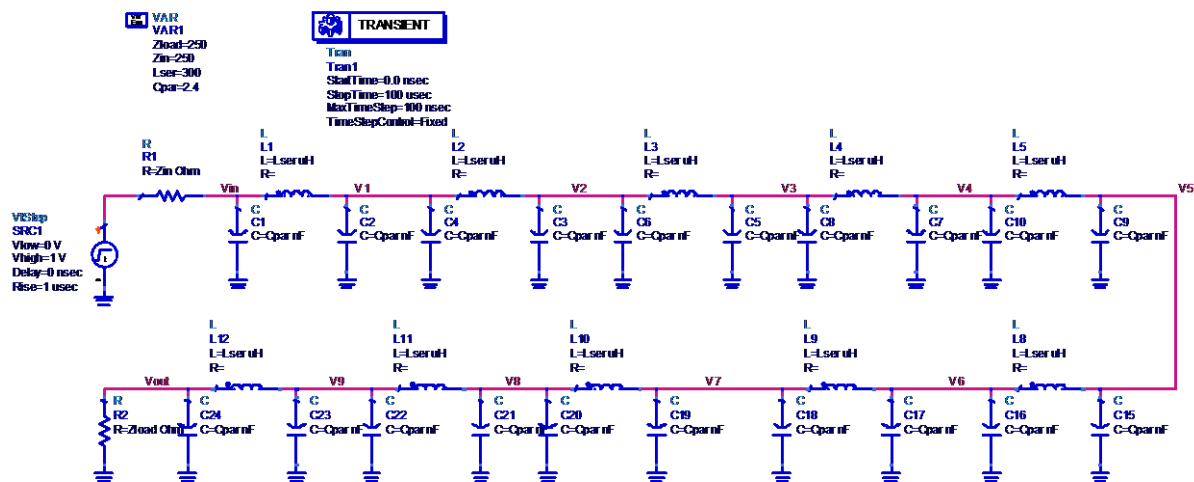

Figure 4: Multiple Π-section Transient Simulation of an Artificial Transmission Line

Now simulate and add the plots for Vin and Vout to the plot. You will need to double click on the rectangular plot to open the dialogue box and click on Plot Options and select x-axis and switch off the auto scale and change the maximum time value to 1e-4 or 100 microseconds as you specified in the transient simulation. You should get a plot something like Figure 5. Using markers you can determine how long the pulse took to travel along the transmission line and assuming a phase velocity of c, you should be able to calculate the equivalent length of transmission line. How does this compare with the answer you derived in Part 1? Comment upon this in your report.
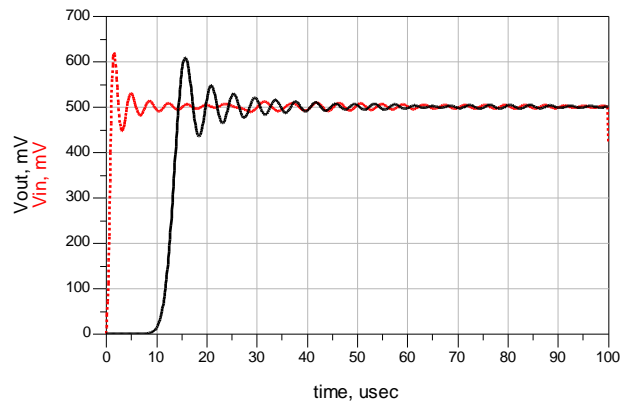
Figure 5: Time Step response of Artificial Line

Finally to examine pulse propagation along the line change the load to open and then short circuit loads for each load condition monitor the input and load voltages as a function of time and one point some way down the line, such as, V_5. For each condition run the simulation and in your report explain what is happening to the voltage in terms of forward and reflected waves.

IMPORTANT: Now make sure you save your work plus any figures you have generated from the current working directory, **C:\work** to your pen drive. This can be done using the **Copy Project…** command in the main ADS project window.

The work that you have done today will form the first part of your report. Next week's lab will form the second part etc..

Part II: MATCHING CIRUITS

GETTING STARTED WITH ADS

Just as in last week, from the PC desktop window, find ADS 2016; it should be in the **Start** list. Click on the program to get it started on your local machine.

Starting a Design: Reminder

First **Clear** any release note that appears on screen by ticking 'Don't display this dialog box again' → **OK**. Then at the prompt of 'What would you like to do?' click on **'Create a new workspace'**. Use a suitable workspace name such as **CAD_lab_matching_wrk** created in the first instance in C:\users directory. The new workspace directory will be created in this local work directory on the local machine. Click **Next** several times and **Finish**.

Note, millimetre or mm should not be confused with mil which is thousandths of an inch.

Now just a word here – should you get stuck anywhere in this or even in subsequent labs then ask a demonstrator or member of staff to help. It's what we're here for! Now back to the project…

Now open a new Schematic Wizard window in the Main ADS window. Dismiss the Wizard window by selecting **Cancel.** The circuit you are going to simulate is a commonly used one based upon the artificial line you are going to design a quarter wave transformer.

Example 1: Quarter-Wave Transformer Made from an Artificial Line

$$Z_L = \frac{j\omega L_s}{2} \qquad Z_L = \frac{j\omega L_s}{2}$$
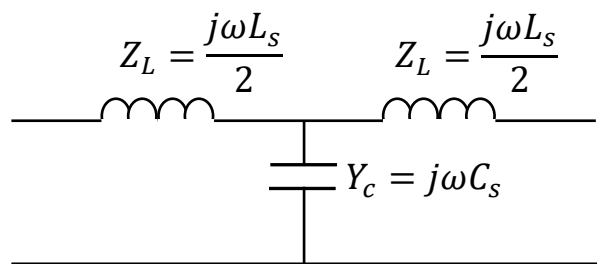
$$Y_c = j\omega C_s$$

Figure 1: T – section representation of a transmission line

A discrete transmission line is to be made comprising 6 T-sections as shown in Figure 1 using lumped capacitors and inductors. The line is to operate as a quarter-wave transformer at 400MHz is to match a load of $Z_L = 650\Omega$ to a source impedance of $Z_g = 50\Omega$. Find values for C_s and $L_s/2$.

- So firstly decide how long electrically each section is to be.
- Then from your notes on the quarter-wave transformer if you know the load impedance and the generator impedance, what impedance does the quarter-wave transformer need to be?
- Then you should have 2 unknowns and 2 equations → solve for L_s and C_s. Now build this up in ADS using the S-parameter simulator, something like that shown below in Figure 2. Remember

your calculated value for L_s is now split between the two inductors. You will need to set the load impedance at the terminal to be 650Ω. Record your values for your report.
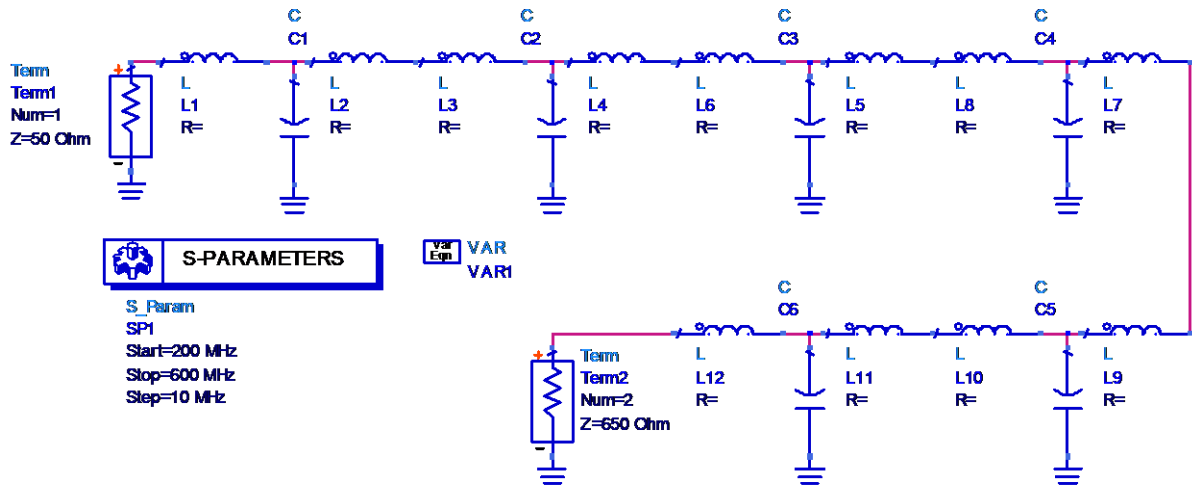

Figure 2: 6 T-section artificial line representing a quarter-wave transformer at 400MHz

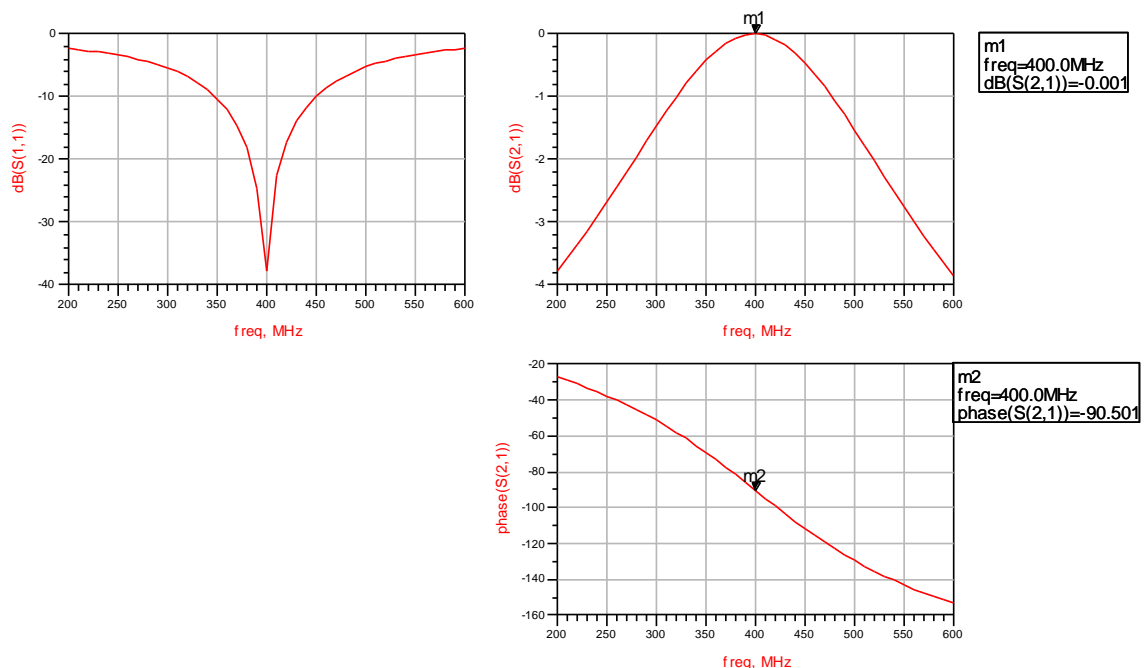The S-parameter response should be like that shown below in Figure 3.

Figure 3: S-parameter response at 400MHz of the artificial line

Notice that at 400MHz the line is matched (ie. S_{11} <-30dB and the phase delay of S_{21} is 90 degrees.

Example 2: S-parameters of Series and Shunt Resistors

    a) Find the S-parameters for a single series resistor, R.
    b) Find the S-parameters for a shunt resistance R.

You should be able to calculate the S-parameters of a resistor based upon the theory we have covered in lectures. Test your S-parameter calculations in ADS by putting values into your equations

for series and shunt resistors. The frequency of simulation doesn't matter since these are resistive values and are independent of frequency so a single frequency point of say 1GHz will yield the S-parameters.

Finally for this CAD laboratory you are going to use a Smith Chart tool to design some matching networks.

Example 3: Matching Networks Using the Smith Chart Tool

In this section you are going to match using lumped elements to a variety of loads. This is exactly the procedure you would use to match transistors for amplification and for filter networks. First of all in the Schematic window click on **Tools → Smith Chart**.

To begin with assume the source is a fixed 50 Ohms and the load to begin with is 50 + j50 Ohms. Apply this to the chart by dragging the load symbol to 1+j1 (normalised to 50 Ohm) or by entering the value directly in the load box. Set the frequency to be 2 GHz. By selecting the components in the palette you can see the effect of adding them. Notice ADS here has no problem with negative values! So match the normalised load, 1+j1, back to the source. What value of series capacitor is needed? You should get a value of around 1.59pF. Notice you can also toggle between admittance and impedance Smith Charts.

So now complete the table below for matching the loads back to the source impedance using the networks described. You may wish to work some of these out on the Smith Chart attached to this script.
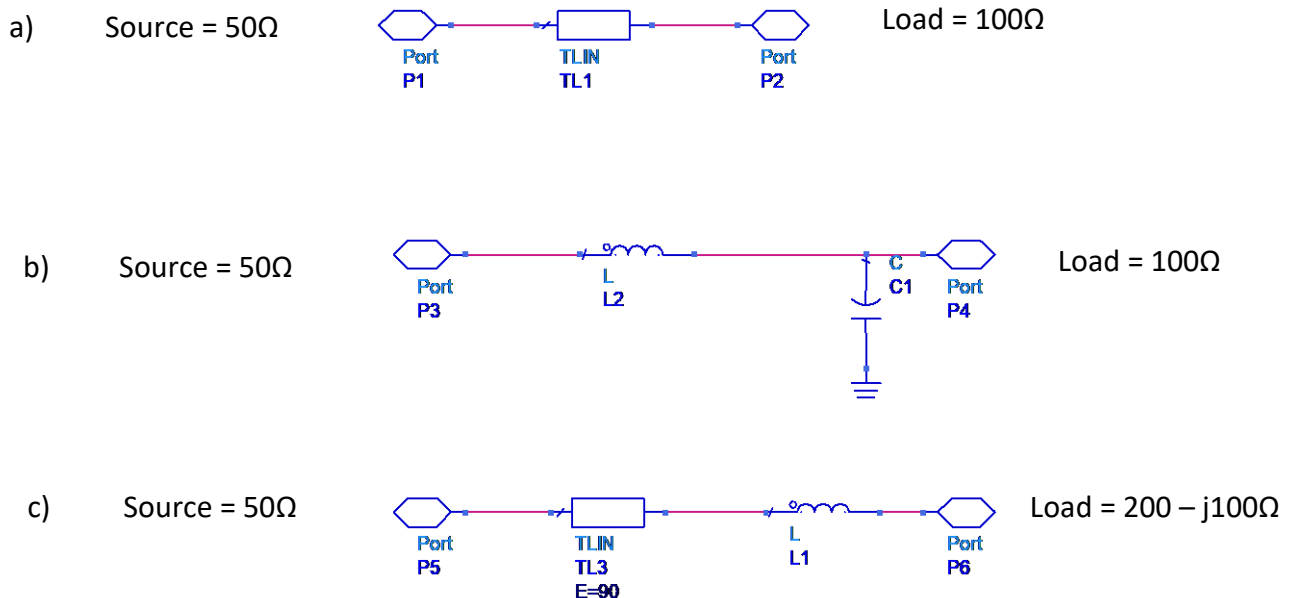
a)   Source = 50Ω

Load = 100Ω

b)   Source = 50Ω

Load = 100Ω

c)   Source = 50Ω

Load = 200 – j100Ω

Figure 4: Matching networks

Complete this table ready for filling in your exercise sheet, below.

	Value of Components
a) TL1: Electrical Length	Deg.
Characteristic Impedance	Ω
b) C1	fF
L2	nH
c) TL3: Characteristic Impedance	Ω
	nH
L1	

Example 4: Single Shorted-Stub Matching

Figure 5 shows a 50 Ω transmission line connected to an antenna with a load impedance $Z_L = (25 - j50)\Omega$. Find the position and length of the short-circuited stub required to match the line.
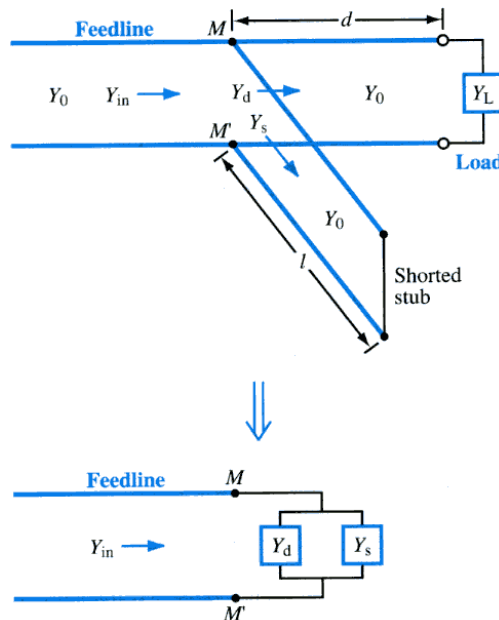

Figure 5: Shorted-stub matching network

Firstly we need to generate this load impedance in ADS. Assuming we are working at 2GHz then what component will give a reactance of –j50Ω? A series capacitance should do it but how about a shunt capacitance across a pure resistance?

The question is what values are R1 and C1 and R2 and C2 at 2GHz to give a load impedance of $Z_L = (25 - j50)\Omega$? Figure 6 shows the two possible networks one with a shunt C and one with a series

C. Using the Smith Chart calculate the values. Once calculated, choose one of these to act as the load for the matching problem used in the rest of the lab.

Record the values you get below:

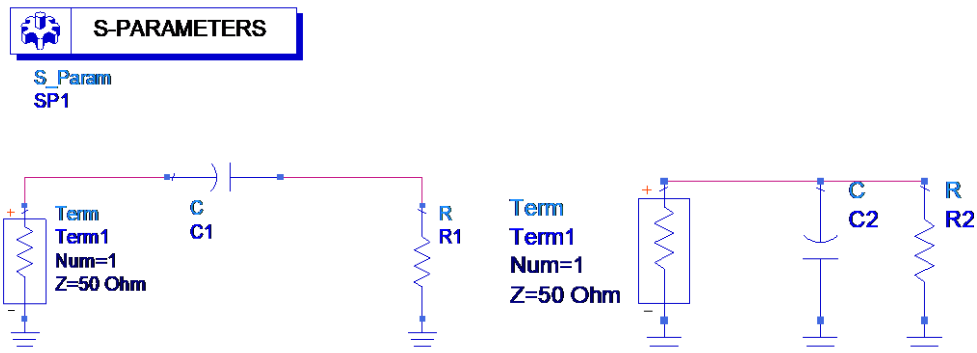| R1 Ω	C1 pF	R2 Ω	C2 pF

Figure 6: Series and shunt representations of load impedance $Z_L = (25 - j50)\Omega$

Next we will series match at 2GHz with a series inductance. Find the values of length ℓ_1 and inductance L_1.

The circuit configuration is given below. Use an ideal transmission line of characteristic impedance $Z_0 = 50\Omega$. You will need a Smith Chart to help with the calculations. You will need to enter the line length of the ideal line as an electrical length at a reference frequency. Here use 2GHz as your reference frequency. Figure 7 shows the schematic. Firstly simulate the circuit without the inductor to make sure you are getting the right answer for the line length and then include the inductor to provide the match.
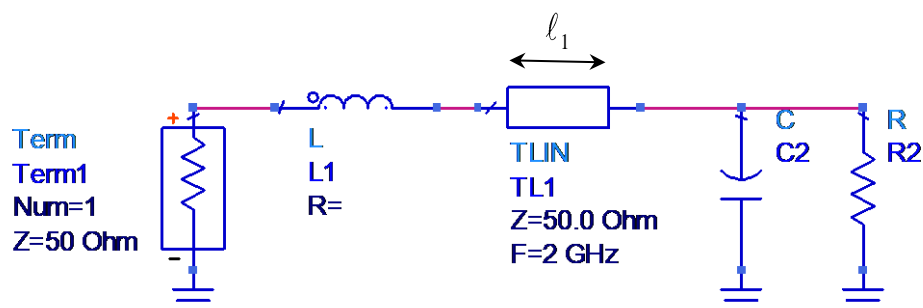

Figure 7: Series shunt inductance match

Record your values in the table below:

Line length, ℓ_1 (degrees)	Series inductance, L_1 (nH)

Now do the same for a shunt capacitance as in Figure 8: you will need a Smith Chart again to calculate this and the line length ℓ_2 will be different from the series inductance. Record you values in the table below.
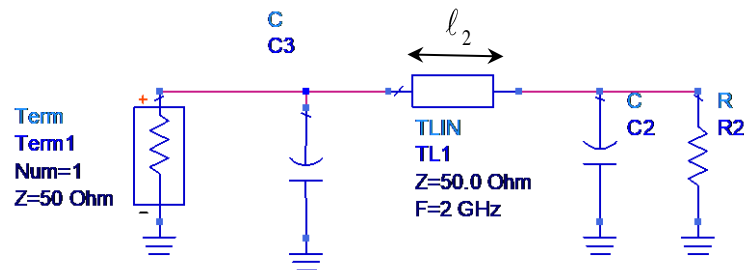

Figure 8: Shunt capacitance matching network

Line length, ℓ_2 (degrees)	Shunt capacitance, C_3 (pF)

When you have a suitable answer then check what the bandwidth of the system is by winding out the S-parameter calculations in the frequency sweep from 1 to 3GHz. You should get a S_{11} plot something like that of Figure 9, unless you chose the series load equivalence.
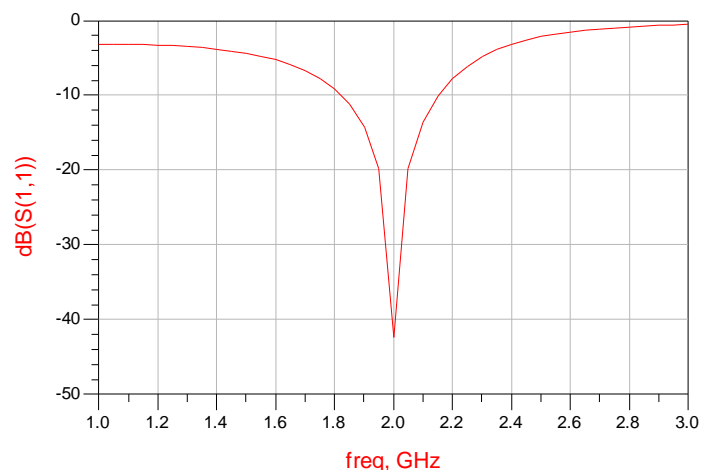

Figure 9: Input reflection coefficient response for the shunt capacitor matched line

Now replace this capacitor with an open circuit stub manufactured in and ideal transmission line TLIN, as in Figure 6. Note the line length ℓ_2 is unchanged because the open circuit stub is acting like

a capacitor. Question is how long should the open circuit stub be?

Term
Term1
Num=1
Z=50 Ohm

Ref

TLOC
TL2
Z=50.0 Ohm
F=2 GHz

TLIN
TL1
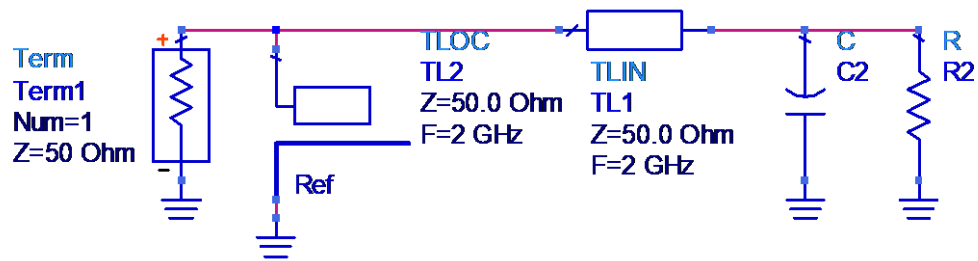Z=50.0 Ohm
F=2 GHz

C
C2

R
R2

Figure 10: Open circuit stub matching

Now if you have time do Example 4 from the notes and confirm the values shown. Include the plot of S_{11} versus frequency and confirm that you get a similar answer.

IMPORTANT: As with the first laboratory, make sure you save your work plus any figures you have generated from the current working directory, **C:\work** to your pen drive. This can be done using the **Copy Project…** command in the main ADS project window.