High Speed Circuit Design
Microwave Office Notes

James E Green

August 2010

Contents

Chapter 1

Introduction

1.1 Microwave Office

Microwave Office is a software product designed by Advanced Wave Research. It is one of the 'big four' software packages used in the design and development of microwave systems and circuits. The other major software packages are *Agilent ADS*, *Eagleware Genesis* and *Ansoft Designer*. Their companies' websites can be found by searching the Internet.

Microwave Office is provided to the Department of Electronic and Electrical Engineering at Sheffield, free of charge. It is intended for the use of students and researchers in a non-profit capacity.

There are twenty licenses available at any time. Licenses are used only when the software is running on a computer. Please close the software when you are not using it, so that licenses can be distributed to other users.

1.2 The Notes

During the 'High speed circuit design' course (EEE403), the lecturer presents several examples of design problems which can be solved by the use of a Smith Chart. This set of notes is a guide to implementing the solutions to those examples using Microwave Office. The objective of these notes is principally to give the student a grounding in the use of one of the major microwave design software packages prior to entering the engineering profession, and secondly to reinforce the concepts taught in the High Speed Circuit Design course.

The notes are presented as one whole unit in a single PDF file. This chapter introduces the notes and their purpose. The next chapter is an introduction to the Microwave Office environment. All subsequent chapters are examples from the High Speed Circuit Design lectures. These chapters begin with the first Smith Chart which is introduced in EEE403 and continue (with some omissions, such as signal flow diagrams) untill the last Smith Chart in the course. The notes are *not* designed to make you think about electronics problems. You will be guided through the electronics problems. The notes are firstly to teach you the basics of Microwave Office, and secondly to reinforce the high speed circuit design course. The notes are complete and contain numerous ‘screen shots’ which will guide you. The screen shots are presented after the text in each chapter. It is not expected that any student will find the material in these notes difficult.

If you run into problems, you may contact me by e-mail at j.e.green@shef.ac.uk.

I hope you will find the use of AWR’s Microwave Office an enjoyable experience. The principle of an interactive Smith chart, which can be updated in a second, without the necessity of a compass and a pencil eraser, would have appealed to me when I was studying similar material for the first time. Finally, when it comes to exam time, you will not have the benefit of Microwave Office in the examination hall. Since there are no credits available for the completion of the tasks documented in these notes, I strongly encourage you to make certain that you have a good working knowledge of your High Speed Circuit Design course and are capable of performing the examples with a pencil and compass. Notwithstanding that, I firmly believe that the use of these notes can cement the concepts in the High Speed Circuits Design Course, and I encourage you to use them to that end.

James E Green (August 2010)

Chapter 2

The Microwave Office Environment

2.1 Environment Overview

The Microwave Office environment is divided into three main areas. The ‘browser’, the ‘window’ and the ‘menus and toolbars’. A typical screen shot is shown in figure 2.1. Some areas of interest have been numbered.

- The ‘browser’, on the left of the screen is further divided by three tabs (number 1 in figure 2.1). These are ‘Project’, ‘Elements’ and ‘Layout’. The ‘Project Browser’ is presently displayed. The ‘Elements Browser’ is shown in figure 2.2.
- Number 2 shows the list of circuit schematics in the present project. For example ‘DC Bias’ and ‘Two Tone Amp’ are both schematics. The items under the schematics in the tree of schematics are a: circuit annotations and b: subcircuits. Circuit annotations are used to display DC currents and voltages on schematics. Subcircuits are used to build hierarchical designs in a similar way to many CAD and EDA programs.
- Number 3 shows the list of graphs in the present project. For example ‘IP3’ and ‘IV BJT’ are rectangular plots. ‘Input Reflection’ is a Smith chart. The items under the graphs are ‘measurements’. Measurements are one of the methods by which useful data is extracted from the simulation of circuits (and layouts) and plotted on graphs.
- Number 4 is a Smith chart.
- Number 5 is a rectangular plot.
- Number 6 is part of the toolbar.
- Number 7 is the menu system.
- Number 8 is a schematic. This schematic also appears as a subcircuit in number 9.

2.2 Toolbars

The toolbars allow quick access to commonly used functions. The toolbars are shown in number six of figure 2.1. An enlarged, numbered, version is shown in figure 2.3. With reference to figure 2.3:

- Number 1 is the 'Analyse' button. This runs simulations.
- Number 2 is the ground element which you will need for creating schematics.
- Number 3 is the linear port element which is distinct from the 'port' tree in the elements browser (figure 2.2). The ports listed in the elements browser are for use in non-linear simulations such as harmonic balance and load pull. The lecture course that we are following does not touch on non linear methods, but a good harmonic balance and load pull tutorial can be found in the 'getting started' guide. If you don't know what harmonic balance or load pull are then you should not attempt them until you are used to the Microwave Office environment *and* the linear simulations in these notes.
- Number 4 is the 'tune tool' a method of adjusting (linear) circuits in real time. We will make extensive use of it.
- Number 5 opens the 'tune' dialogue box which is the counterpart of the tune tool, and is used in conjunction with the tune tool.

There are many other toolbuttons on several other toolbars, but they need not concern us presently. Some are obvious such as open and save. Others not so, all have tool tips and are documented in the help system.

2.3 Menu System

The menu system contains all the available commands. The menus are dynamic however, you will have different menus available depending on the object which is presently selected. For example, it is not meaningful to place ports on a Smith chart, so this option is removed from the 'Draw' menu when a Smith chart is selected.

With reference to figure 2.4:

- Number 1 is the rotate command. There are several methods for rotating elements. Generally you can select an element by left clicking it (for example a resistor or a port) and select 'Edit' - 'Rotate'. Alternatively under most circumstances you can select the element and right click to produce a pop-up menu which will contain a rotate option. It is also possible to select the element and use Ctrl+R on the keyboard. There are several methods for producing most results in Microwave Office. In general the method presented is my preferred method. You may find other ways which you prefer to use. Provided our answers agree, it does not matter which you choose.
- Number 2 shows a set of commands which add common elements to schematics, the linear port and ground.
- Number 3 is a command to add a schematic to the project. Schematics can be 'new', 'imported' or 'linked'. In this document all of the schematics will be created from scratch. If there are options regarding how to create a schematic you should always choose 'new'.

- Number 4 contains commands to add a graph to the project, add a measurement to a graph and to set goals for the optimiser. The use of these tools will be introduced shortly.

- Number 5 is ‘Project Options’. The first thing you should do with a new project is to set up the ‘base’ units. This is done using the ‘Project Options’. You will be instructed on the settings to choose, dependant on the example you are following.

- Number 6 is the ‘Getting Started’ part of the help. Once you have finished these documents, if your workload permits, attempt the tutorials in the ‘Getting Started’ guide. The analyses you will be undertaking and the circuits and systems involved will not have been formally taught to you, so while you may be able to perform the analyses without difficulty, you may have to do some background reading on microwave engineering. The library should be your first port of call.

- Number 7 is a cluster of commands which are also available on the toolbar. The Analyse button has been introduced, as has the tune tool and the tune dialogue box. The Optimiser is a tool which will be introduced in the chart chapters. The ’Analyse’ option at the top of the simulation menu is distinct however from ‘Run/Stop System Simulators’ in the ‘Simulate’ menu. The ’Analyse’ command deals with circuits and electromagnetic structures such as circuit board layouts and some antennas (far field only). The ‘Run/Stop System Simulators’ runs the ‘Virtual System Simulator’ (VSS). The VSS is useful for implementing entire electronic systems for example a Bluetooth link or a QAM radio link, it is capable of producing, for example, BER estimations and IQ constellations. VSS will not appear again in these notes, and you will not need the ‘Run/Stop System Simulators’ command. For more information on VSS and other areas of Microwave Office which will not be discussed here see the ‘Getting Started’ guide in the ‘Help’ menu. The ‘Getting Started’ guide is a good tutorial on the use of Microwave Office. You can find a QAM example and a Bluetooth example by selecting ‘Open Example’ from the ‘File’ menu.

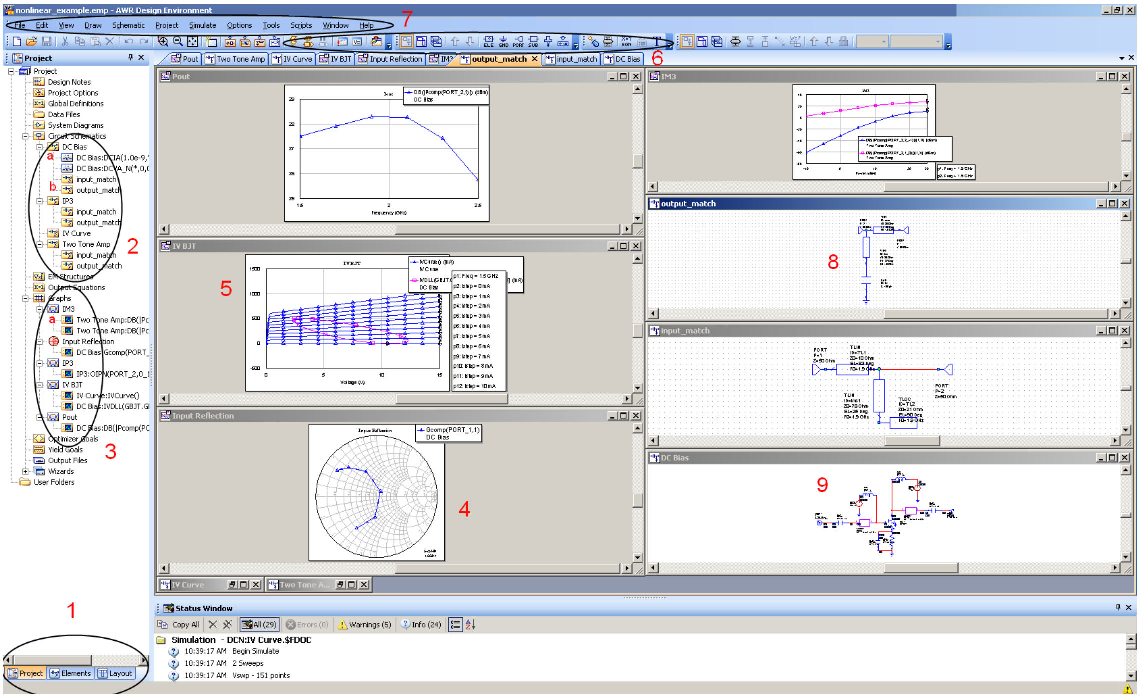


Figure 2.1: Microwave Office Environment

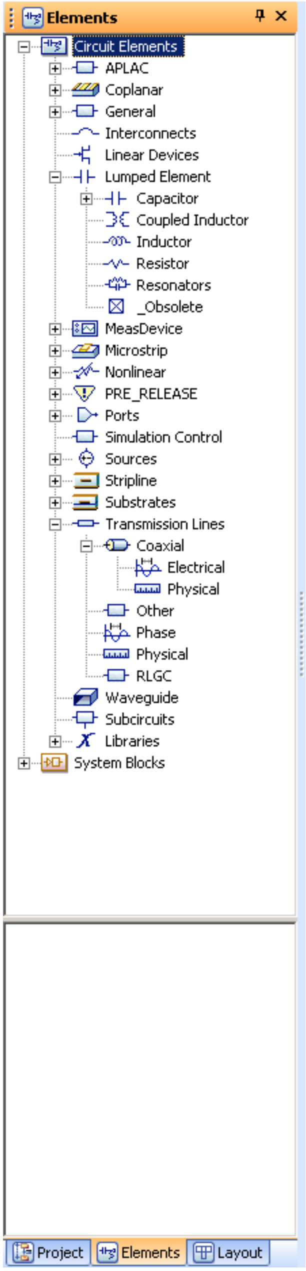


Figure 2.2: Elements Browser

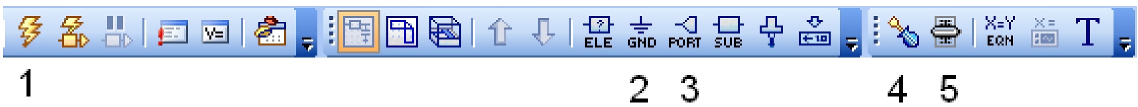


Figure 2.3: Partial Toolbar

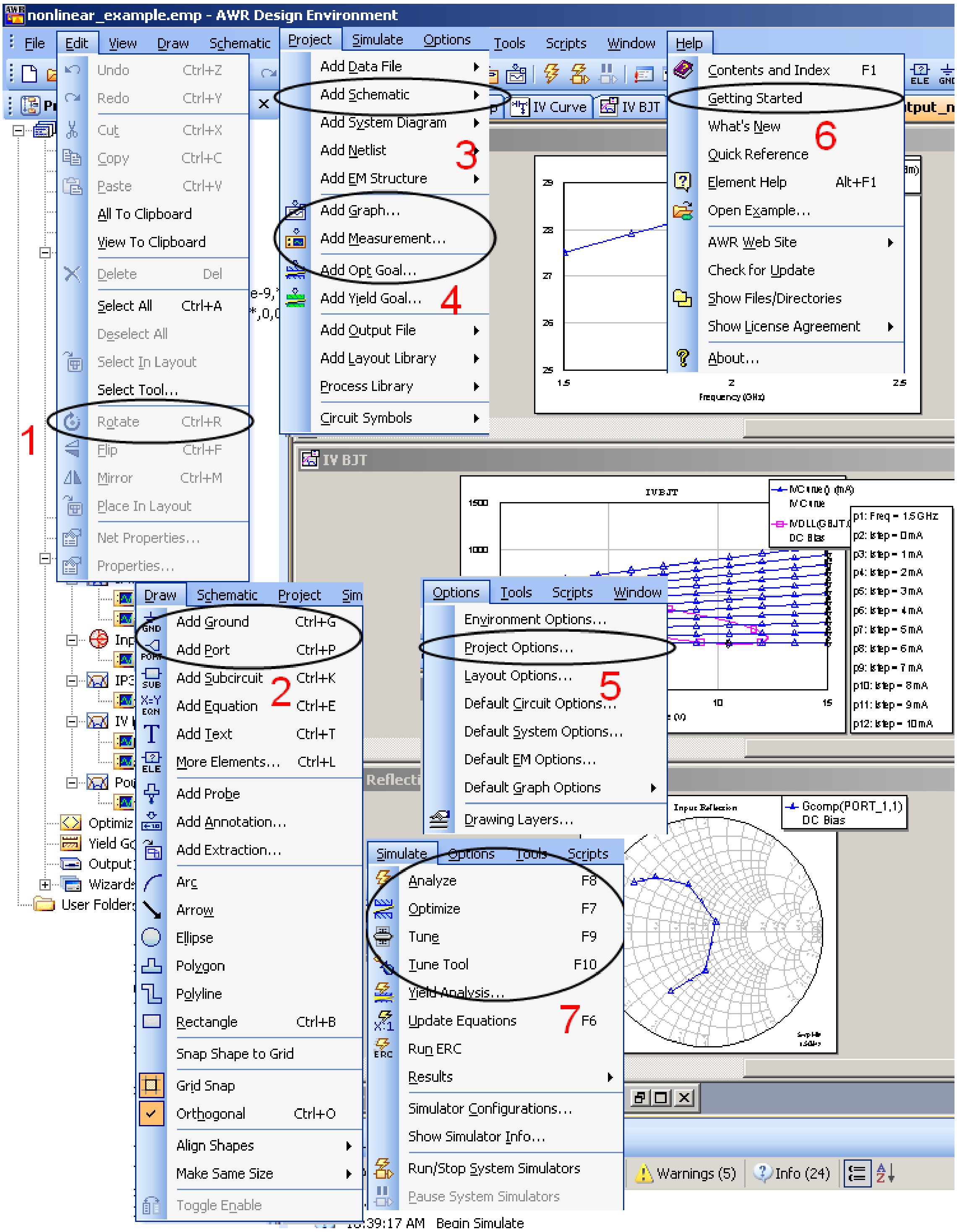


Figure 2.4: Menu System Shown for Schematic View

Chapter 3

Chart 1

3.1 Introduction

In this chapter the example in chart 1 is followed. A a transmission line terminated with an unknown load has a complex input impedance of $Z_{in} = (1 + j1)\Omega$. The line length is 135 electrical degrees or $3/8\lambda$. The objective is to find the load impedance.

Question
If $Z_{in} = (1 + j1)\Omega$ and the line length is $\frac{3}{8}\lambda$

Solution on a Manual Smith Chart

- Plot $Z_{in} = (1 + j1)\Omega$ at point 'A'.
- Construct a line from the center to the edge of the chart through point A.
- Read the 'wavelengths towards the load' scale at point A add $\frac{3}{8}\lambda$ to this value to find Point B.
- Move around the circumference of the chart from Point A to Point B (anti-clockwise - towards the load)
- Measure from the center of the chart to Z_{in}.
- Measure a similar distance from the center of the chart towards Point B.
- Point B, $Z_L = (2 - j1)\Omega$.

3.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart1'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

3.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 3.1.

Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

3.2.2 Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Load Impedance Example'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right click until the port has it's connecting pin facing to the right.
- Place the port with a left click.

Add a transmission line element.

- To add the TLIN element open the 'Elements Browser' select 'Transmission Lines' - 'Coaxial' - 'Electrical' - 'COAX2'.
- Move the mouse over the schematic
- Right click until the TLIN is horizontal.
- Place it (left click) level with the port.
- Edit the 'electrical length' (EL) property, make it 135 degrees.
- Edit the 'design frequency' (Fo) property, make it 5.4GHz

The load impedance is a series connection of a resistive element and an reactive element. To simplify matters, a complex impedance can be specified.

Add the impedance which will be transformed:

- To add the impedance, in the 'Elements browser' select 'General' - 'Passive' - 'Other' - 'IMPED'.
- Left click and drag the 'IMPED' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left mouse click.
- Double left click the 'R' parameter, change it to 0.
- Double left click the 'X' parameter, change it to 0.

If you have already placed the element and need to rotate it, left click it and press Ctrl + R, *or* right click it and select rotate from the pop-up menu, *or* choose 'Rotate' from the 'Edit' menu.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit. At this point your schematic should look like figure 3.2.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 5.4GHz.

- Open the 'Project Options' from the 'Options' menu.
- Ensure your 'frequencies' tab is as shown in figure 3.3.
- Click **'Apply'**, then 'ok'. If you *only* click 'ok' nothing will be changed!

3.2.3 Impedance Chart

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Impedance'.

To appropriately arrange the windows choose 'Tile Vertically' from the 'Window' menu.

To run a simulation and view the output in Microwave Office a process of 'Measurements' is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 3.4.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 3.5.

3.2.4 Load Impedance

To find the load impedance that will produce the input impedance in the question the Tune Tool can be used.

To set up the tuning:

- Click the schematic to make it the active window.
- Select 'tune tool' from the 'Simulate' menu, *or* press F10.
- Click the R and X parameters of the IMPED element. They should turn blue. See figure 3.6
- Select 'tune' from the 'Simulate' menu. A dialogue box should appear.
- In the tune dialogue box set the 'min' and 'max' values to -200 and +200 respectively for both the resistance and reactance. Set the 'step' parameter to 1 See figure 3.7
- Select 'Analyse' from the 'Simulate' menu.
- Move the sliders in the tune dialogue box to see the effect of increasing the load resistance and reactance on the Smith chart.

The load impedance required may be found by trial and error but is $(2 - j1)\Omega$.

The final chart should look like 3.8

Note that this is not the best method for solving this problem in Microwave Office, but it alows you to see how changing the resistive and reactive parts of a load affects the Smith chart.

3.3 Figures

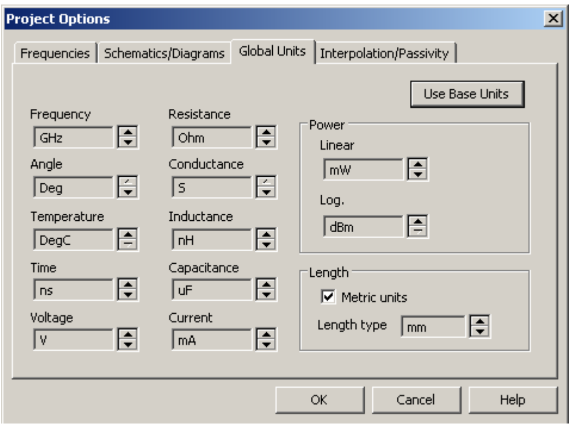


Figure 3.1: Chart 1 : Project Options Dialogue Box

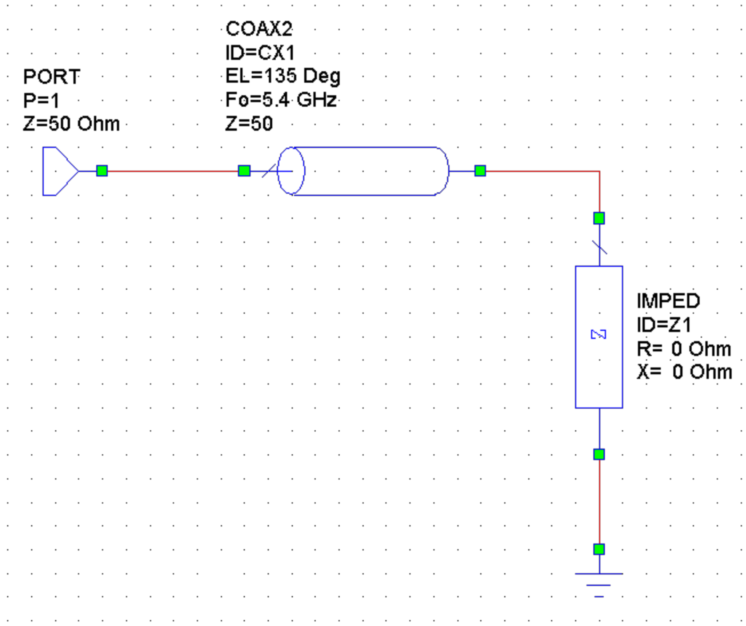


Figure 3.2: Chart 1 : Input Impedance Schematic

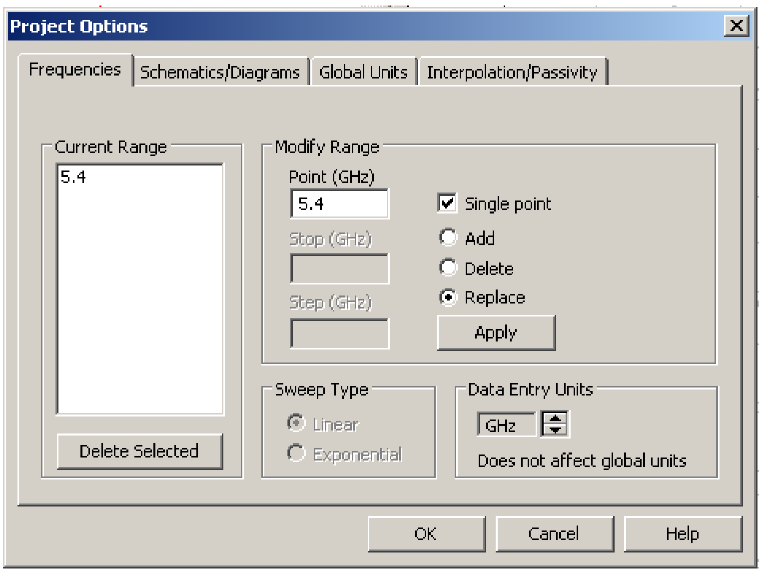


Figure 3.3: Chart 1 : Project Frequencies

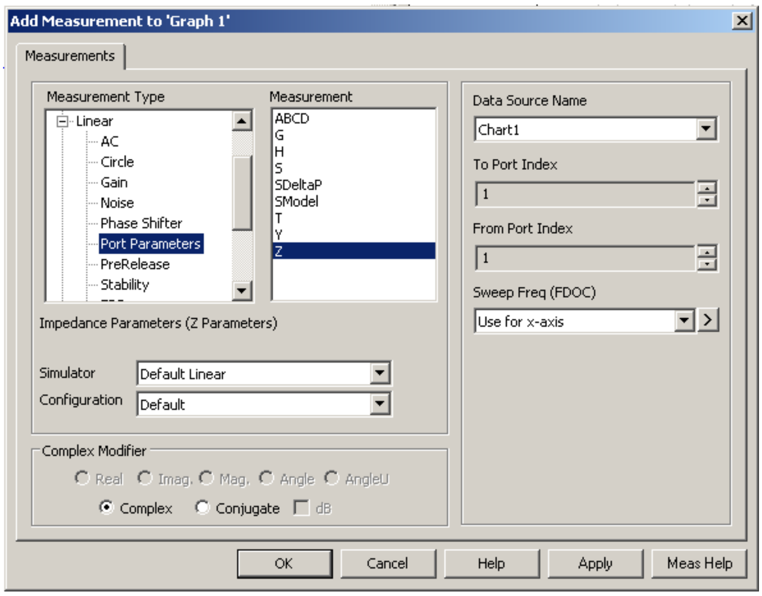


Figure 3.4: Chart 1 : Add Measurement Dialogue Box (Input Impedance)

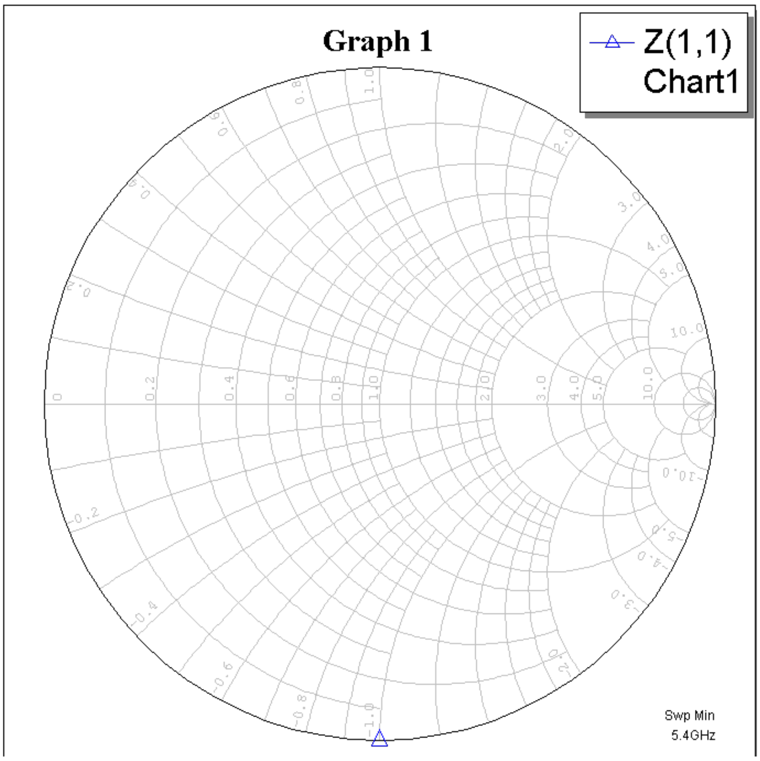


Figure 3.5: Chart 1 : Input Impedance Smith Chart

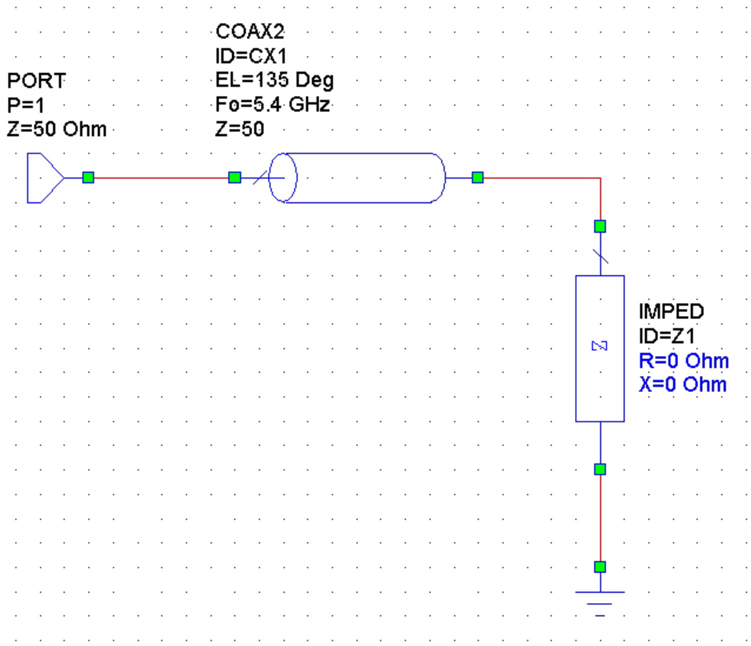


Figure 3.6: Chart 1 : Input Impedance Schematic showing Tune Tool selection of the electrical line length

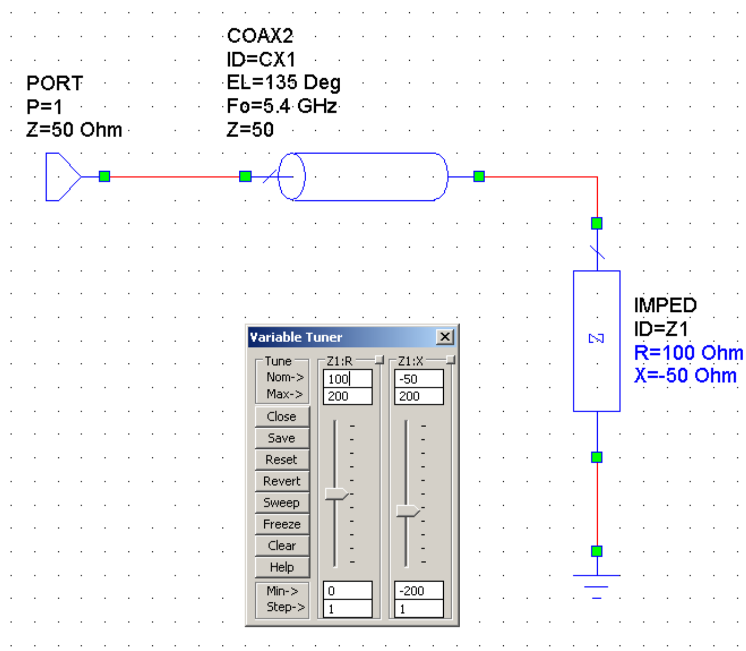


Figure 3.7: Chart 1 : Tune Tool Dialogue Box

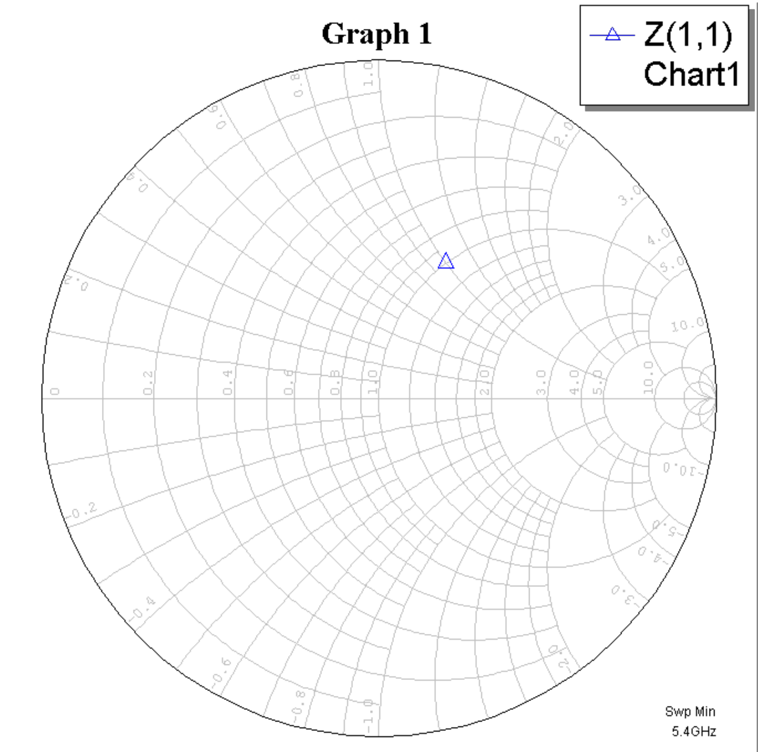


Figure 3.8: Chart 1 : Smith Chart Showing Input Impedance of $(1 + j1)\Omega$

Chapter 4

Chart 2

4.1 Introduction

In this chapter the example in chart 2 is followed. A complex load is specified $Z_L = (1 + j1)$ and connected to a transmission line. The line length is unknown, however the input impedance is given as $Z_{in} = (2 - j1)$ The objective is to find the length of the line.

Question
If $Z_{in} = (2 - j1)\Omega$ and $Z_L = (1 + j1)\Omega$ what is the line length?

Solution on a Manual Smith Chart

- Plot $Z_L = (1 + j1)\Omega$ at point 'A'.
- Construct a line from the center to the edge of the chart through point A.
- Plot $Z_{in} = (2 - j1)\Omega$ at point 'B'.
- Construct a line from the center to the edge of the chart through point B.
- Read the 'wavelengths towards the load' scale at point A.
- Move around the circumference of the chart from Point B to Point A (anti-clockwise - towards the load)
- Read the 'wavelengths towards the load' scale at point B.
- Subtract the reading for point A from point B to yield the line length.

4.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart2'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

4.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 4.1.

Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

4.2.2 Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Line Length Example'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right click until the port has it's connecting pin facing to the right.
- Place the port with a left click.

Add a transmission line element.

- To add the TLIN element open the 'Elements Browser' select 'Transmission Lines' - 'Coaxial' - 'Electrical' - 'COAX2'.
- Move the mouse over the schematic
- Right click until the TLIN is horizontal.
- Place it (left click) level with the port.
- Edit the 'electrical length' (EL) property, make it 0 degrees.
- Edit the 'design frequency' (Fo) property, make it 1800MHz

The load impedance, is a series connection of a resistive element and an inductive element. To simplify matters, a complex impedance can be specified.

Add the impedance which will be transformed:

- To add the impedance, in the 'Elements browser' select 'General' - 'Passive' - 'Other' - 'IMPED'.
- Left click and drag the 'IMPED' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left mouse click.
- Double left click the 'R' parameter, change it to 50.
- Double left click the 'X' parameter, change it to 50.

If you have already placed the element and need to rotate it, left click it and press Ctrl + R, *or* right click it and select rotate from the pop-up menu, *or* choose 'Rotate' from the 'Edit' menu.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit. At this point your schematic should look like figure 4.2.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 1.8GHz.

- Open the 'Project Options' from the 'Options' menu.
- Ensure your 'frequencies' tab is as shown in figure 4.3.
- Click **'Apply'**, then 'ok'. If you *only* click 'ok' nothing will be changed!

4.2.3 Impedance Chart

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Impedance'.

To appropriately arrange the windows choose 'Tile Vertically' from the 'Window' menu.

To run a simulation and view the output in Microwave Office a process of 'Measurements' is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 4.4.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 4.5.

4.2.4 Electrical Length of the Line

To find the line length that will produce the input impedance in the question the Tune Tool can be used.

To set up the tuning:

- Click the schematic to make it the active window.
- Select 'tune tool' from the 'Simulate' menu, *or* press F10.
- Click the electrical length parameter of the line TLIN element. It should turn blue. See figure 4.6
- Select 'tune' from the 'Simulate' menu. A dialogue box should appear.
- In the tune dialogue box set the 'min' and 'max' values to 0 and 180 respectively. Set the 'step' parameter to 1 See figure 4.7
- Select 'Analyse' from the 'Simulate' menu.
- Move the slider in the tune dialogue box to see the effect of increasing the line length on the Smith chart.

The line length that will produce an input impedance of $(2 - j1)$ is 0.125λ.

4.3 Figures

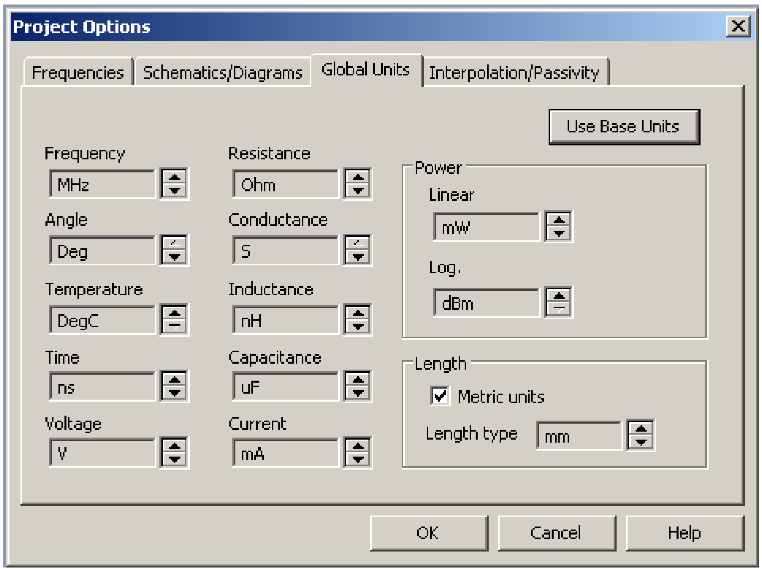


Figure 4.1: Chart 2 : Project Options Dialogue Box

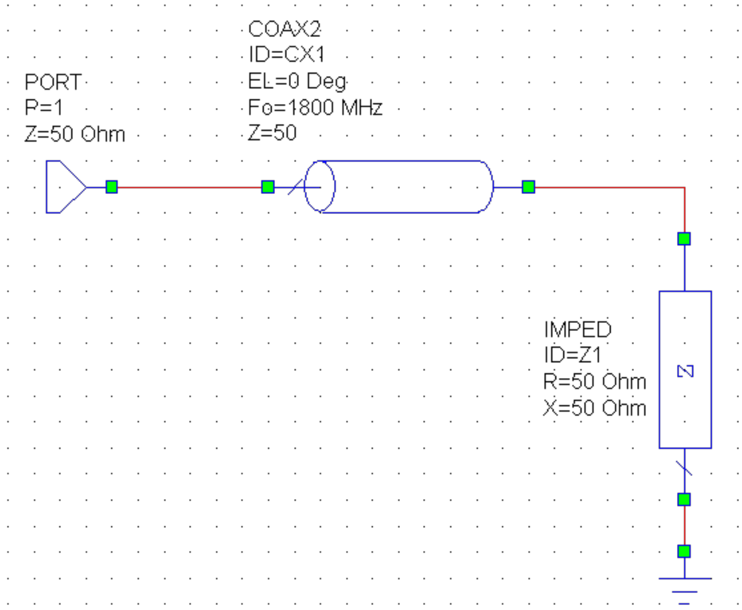


Figure 4.2: Chart 2 : Input Impedance Schematic

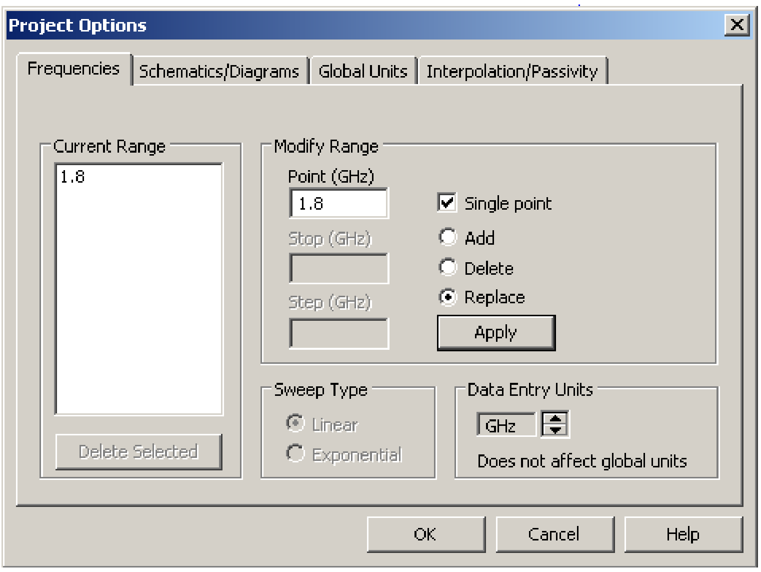


Figure 4.3: Chart 2 : Project Frequnecies

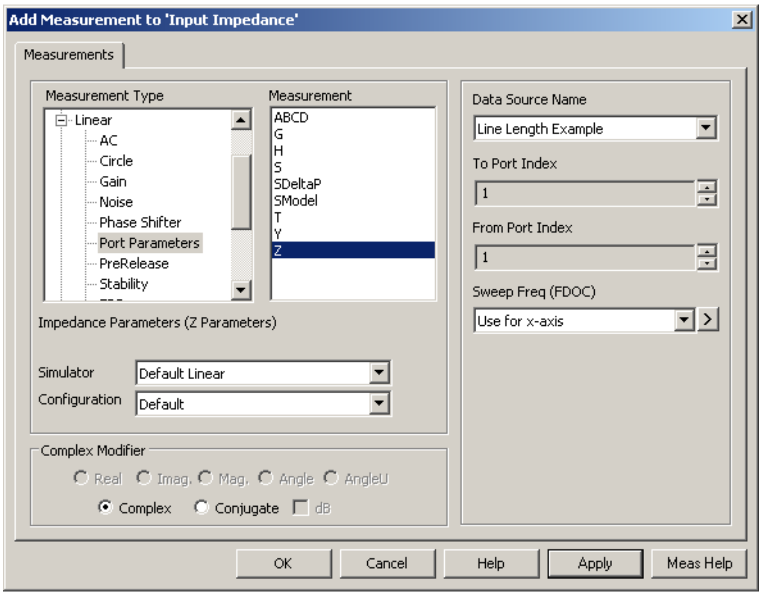


Figure 4.4: Chart 2 : Add Measurement Dialogue Box (Input Impedance)

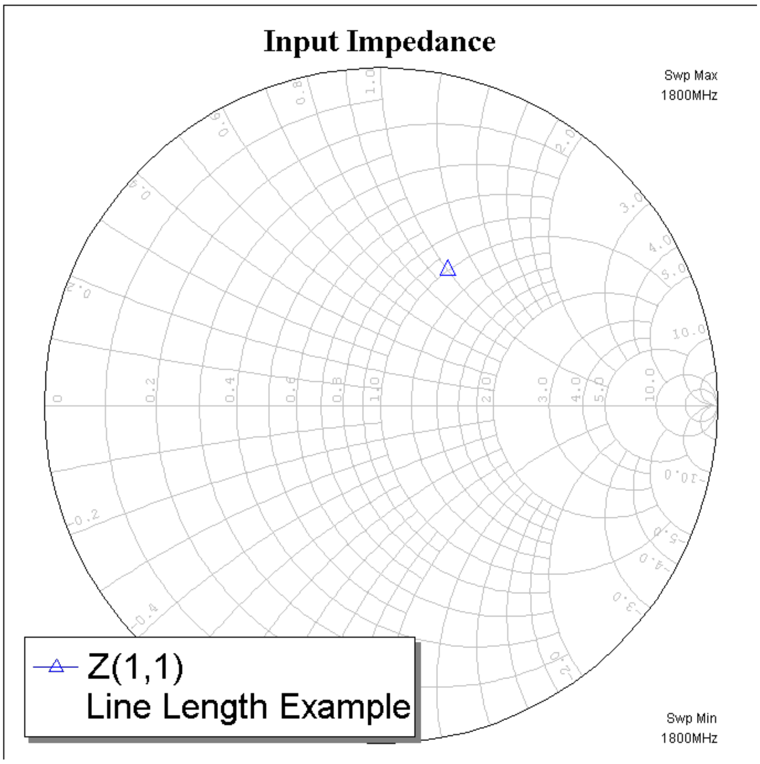


Figure 4.5: Chart 2 : Input Impedance Smith Chart

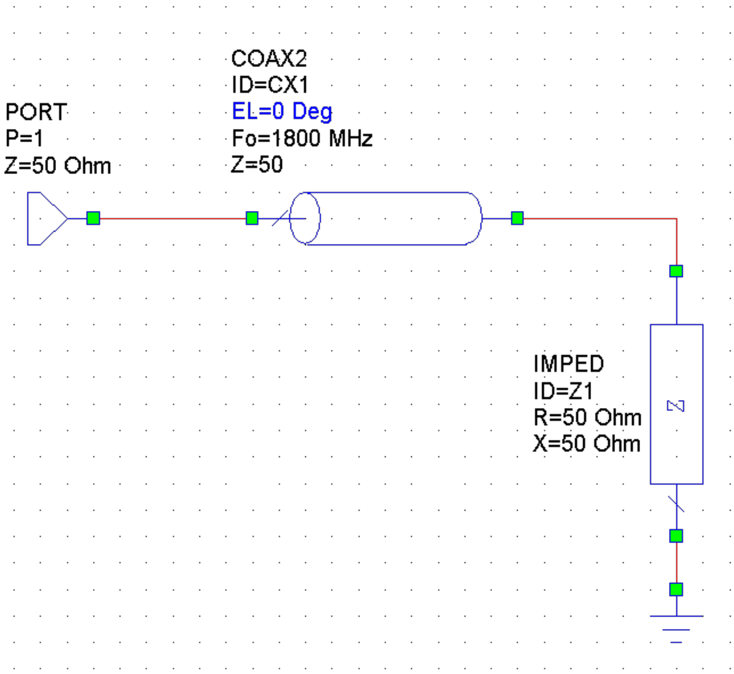


Figure 4.6: Chart 2 : Input Impedance Schematic showing Tune Tool selection of the electrical line length

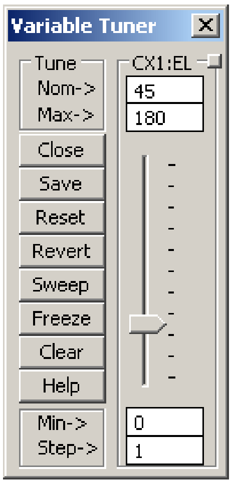


Figure 4.7: Chart 2 : Tune Tool Dialogue Box

Chapter 5

Chart 3

5.1 Introduction

In this chapter the example in chart 3 is followed. A complex load is specified and connected to a transmission line. The VSWR on the line must be found The exact question, and the solution given in the lecture notes are given below.

Question
Find the VSWR on a line with $Z_L = (1.1 + j1.2)$

Solution Find the input impedance of the line whose length is such that Z_{in} is purely resistive. This value of R_{in} has the same numerical value as the VSWR. The proof of this statement is given in the EEE403 lecture notes.

- Plot $Z_L = (1.1 + j1.2)\Omega$ at point 'A'
- Construct an arc of radius A in a clockwise direction until it crosses the real axis (horizontal) R_{in} is now 3 so the VSWR is 3 for the given load.

5.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart3'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

5.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 5.1. Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

5.2.2 Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Call your schematic 'VSWR Example'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right mouse click until the port has it's connecting pin facing to the right.
- Place the port with a left mouse click.

The load is a series connection of a resistive element and a capacitive element. To simplify matters, a complex load can be specified.

Also add the load:

- To add the load, in the 'Elements browser' select 'General' - 'Passive' - 'Other' - IMPED.
- Left click and drag the 'IMPED' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking
- Place the load with a left mouse click.

If you have already placed the element and need to rotate it, left click it and press Ctrl + R, *or* right click it and select rotate from the pop-up menu, *or* choose 'Rotate' from the 'Edit' menu.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit *and* change the port and load data to match figure 5.2.

5.2.3 Load Impedance

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Impedance'.

To run a simulation and view the output in Microwave Office a process of 'Measurements' is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.

- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 5.3.
- Click 'Apply' and 'Ok'.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 1000MHz.

- Open the 'Project Options' from the 'Options' menu.
- Ensure your 'frequencies' tab is as shown in figure 5.4.
- Click **'Apply'**, then 'ok'. If you only click 'ok' nothing will be changed!

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 5.5.

5.2.4 VSWR

The VSWR can be found on the smith chart, but a more accurate answer can be obtained on a rectangular plot.

Add a rectangular plot to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Rectangular' from the dialogue box which appears.
- name your chart 'VSWR'.

A measurement of the VSWR can be set-up as follows.

- Right click the VSWR plot icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 5.6.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. Read the VSWR from the rectangular plot. It is 2.9829424, approximate this to 3.

On a conventional Smith chart the VSWR would be found by constructing an arc from the load impedance toward the real impedance line (horizontal) and reading the value of the SWR from there. You could also measure with a ruler the distance from the point $(1 + j0)$ to the load and then mark this distance on the SWR bar under the 'Radially Scaled Parameters' section of a chart. In Microwave office a VSWR circle can be constructed on a Smith chart as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.

- In the pop-up menu select ‘Add Measurement’.
- Change your measurement dialogue box to match figure 5.7.
- Click ‘Apply’ and ‘Ok’.

Select ‘Analyse’ from the ‘Simulate’ menu. A SWR circle should appear and pass through the load impedance and the point $(3 + j0)$ as shown in figure 5.8.

5.3 Figures

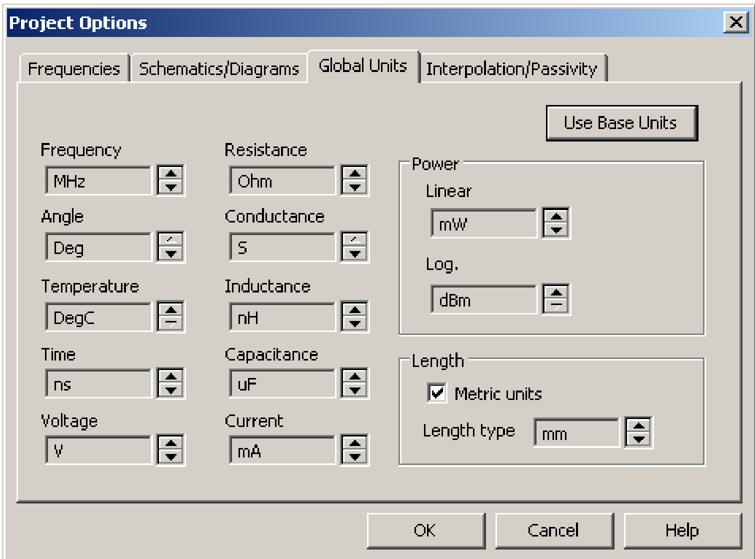


Figure 5.1: Chart 3 : Project Options Dialogue Box

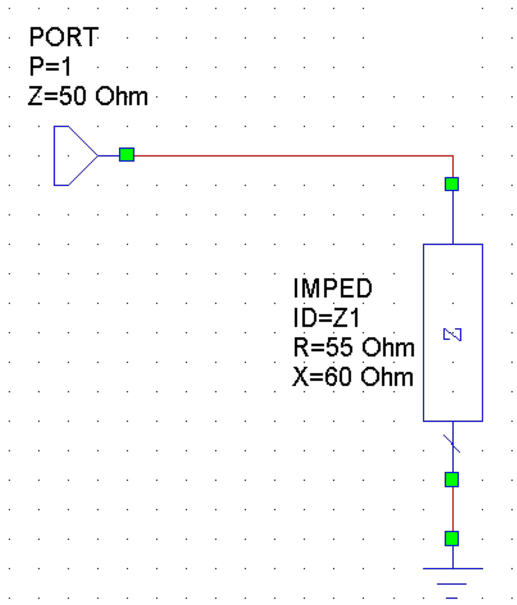


Figure 5.2: Chart 3 : VSWR Example Schematic

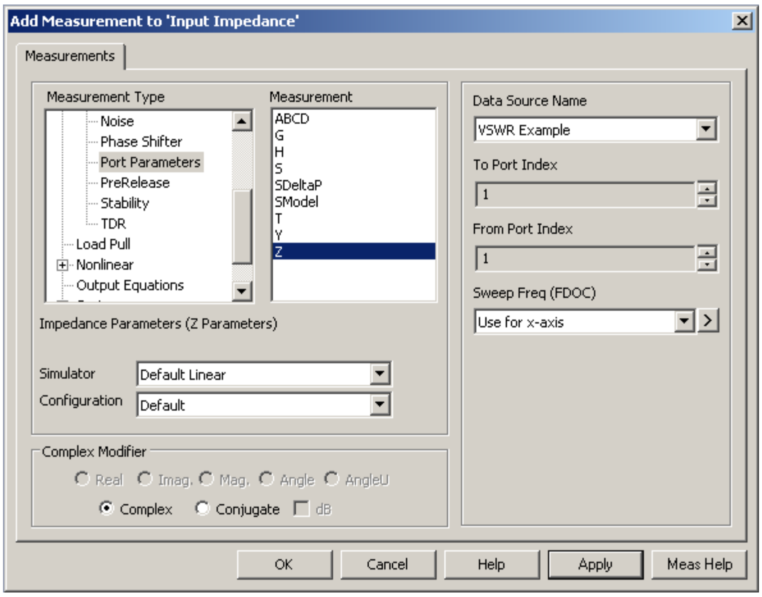


Figure 5.3: Chart 3 : Add Measurement Dialogue Box (Input Impedance)

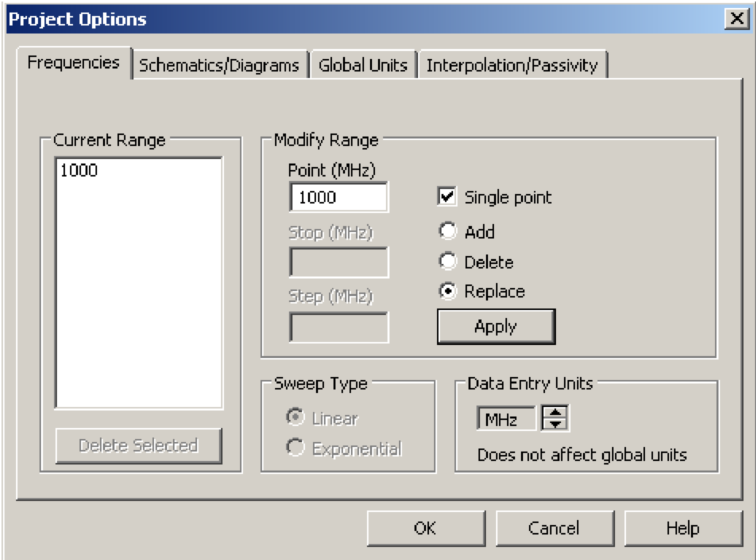


Figure 5.4: Chart 3 : Project Frequencies

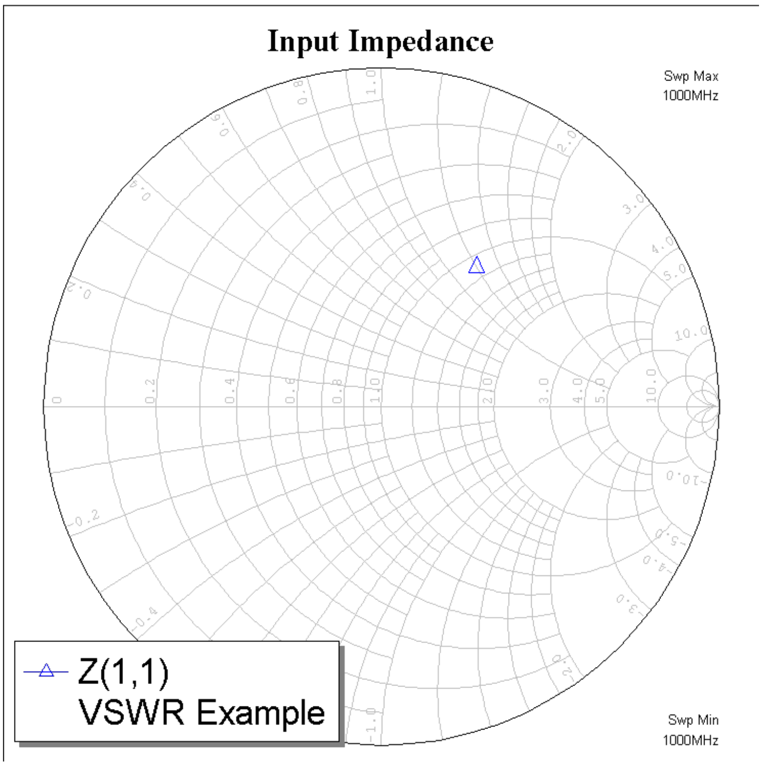


Figure 5.5: Chart 3 : Smith Chart with Load Impedance

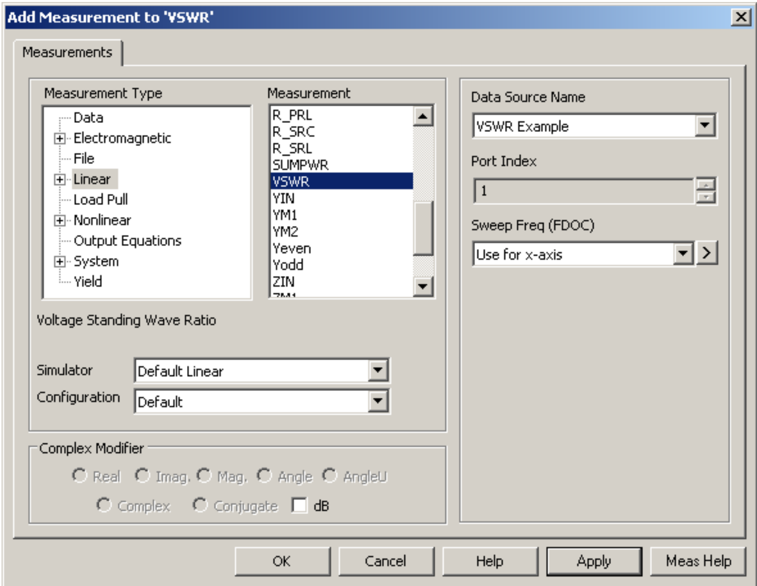


Figure 5.6: Chart 3 : Add Measurement Dialogue Box, VSWR (Rectangular Plot)

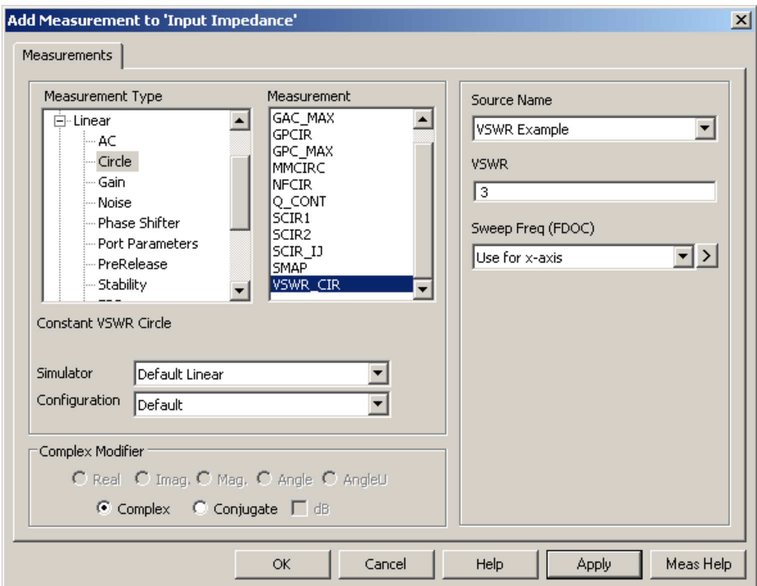


Figure 5.7: Chart 3 : Add Measurement Dialogue Box, VSWR Circle (Smith Chart)

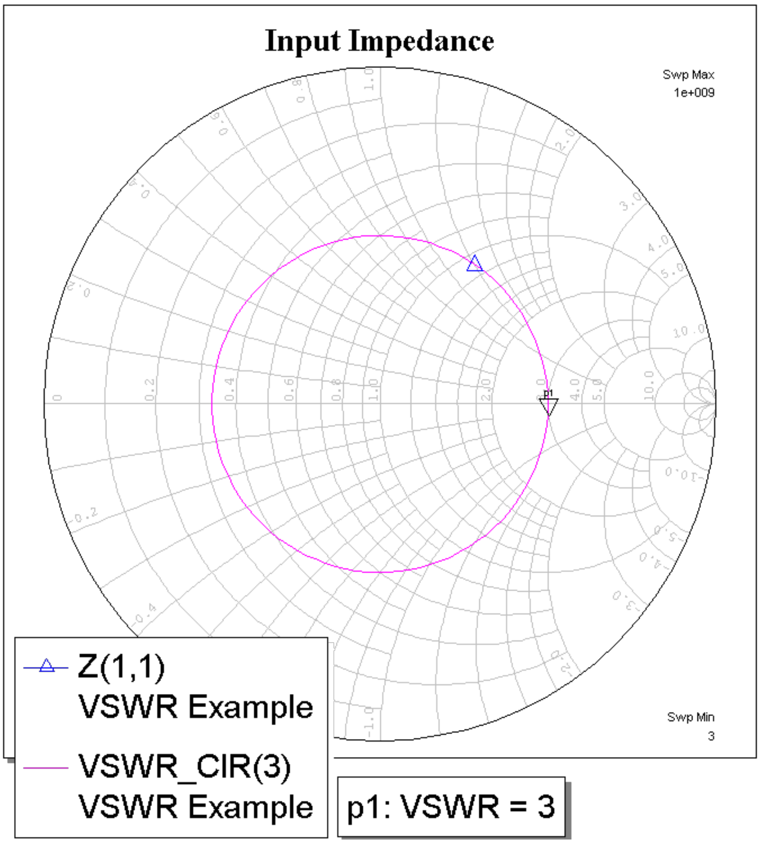


Figure 5.8: Chart 3 : Smith Chart with Load Impedance and VSWR Circle

Chapter 6

Chart 4

6.1 Introduction

In this chapter the example in chart 4 is followed. The question deals with transmission lines that have loss associated with them. All physically realisable transmission lines have some loss. It is sometimes necessary to account for the loss. Situations where the circuits are electrically large or the lines are unusually lossy are the most likely situations where loss will need special consideration. Most of the time at lower microwave frequencies loss can be ignored in small circuits. More on loss mechanisms in transmission lines can be found at `http://www.microwaves101.com/encyclopedia/transmission_loss.cfm`

Question

A lossy line with attenuation coefficient of 18.64dB/m at 300MHz is terminated with a 250 ohm load. Find the VSWR at 375mm from the load.

Solution

- The wavelength at 300MHz is given by $c = f\lambda$.
- $\lambda = 1$ m. $d = 0.375\lambda$. $\alpha d = 0.375 \times 18.64 = 6.99$dB.
- To convert dB/m into Nepers/m use Nepers/m = dB/m $\times$ 8.686.
- Therefore $\alpha d = \frac{6.99}{8.686} = 0.805$ Nepers. $2\alpha d = 1.61$ Nepers.

On the Chart

- Plot point 'A' at (5 +j0). This is 250Ω normalised to 50Ω.
- Move 0.375λ round the chart in the direction of the generator (clockwise). This is point 'B'.
- The radius at point 'B' must be reduced by a factor of $e^{-2\alpha d}$. In this case the reduction is 80%.
- The VSWR is 1.3.

6.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart4'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

6.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 6.1. Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

6.2.2 Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Lossy Line'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right click until the port has it's connecting pin facing to the right.
- Place the port with a left click.

The load is a series resistive element.

Add the load resistance:

- To add the resistance, in the 'Elements browser' select 'Lumped Elements' - 'Resistor' - 'RES'.
- Left click and drag the 'RES' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left mouse click.
- Double left click the 'R' parameter, change it to 250.

The lossy line is modelled by a TLINP element. This element is a coaxial transmission line with its shield grounded. It includes a loss model suitable for this example.

Add the lossy line resistance:

- To add the lossy line, in the 'Elements browser' select 'Transmission Lines' - 'Physical' - 'TLINP'.

- Left click and drag the 'TLINP' element on to the schematic.
- Place the load with a left mouse click.
- Change the TLINP parameters as follows $Z_O = 50\Omega$, $L = 0$mm, Eeff= 1, Loss= 18.64, $F_0 = 300$MHz.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit. At this point your schematic should look like figure 6.2.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 300MHz.

- Open the 'Project Options' from the 'Options' menu.
- Ensure your 'frequencies' tab is as shown in figure 7.3.
- Click **'Apply'**, then 'ok'. If you *only* click 'ok' nothing will be changed!

6.2.3 Impedance Chart

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Impedance'.

6.2.4 VSWR Chart

Add a Rectangular chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Rectangular' from the dialogue box which appears.
- Name your chart 'VSWR'.
- Having created the chart modify the axes by right clicking the chart ans selecting 'properties' from the pop-up menu.
- Change the y-axis limits such that the minimum is 0 and the maximum is 6.

- Change the x-axis limits such that the minimum is 100MHz and the maximum is 1000MHz, also select the 'log scale' option.
- The properties dialogue box should look like figure 6.4

To appropriately arrange the windows choose 'Tile Vertically' from the 'Window' menu.

To run a simulation and view the output in Microwave Office a process of 'Measurements' is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 6.5.
- Click 'Apply' and 'Ok'.

A measurement of the VSWR on the Rectangular Graph is added as follows

- Right click the VSWR Graph icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 6.6.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. At this point your Microwave Office window should look like figure 6.7.

Because the line presently has zero length, the VSWR is 5, and the impedance looking into the line is the same as the load impedance 250Ω.

6.2.5 Adjusting the Length of the Line

We could set the line length to 375mm now, and look at the result by simulating. However it will be instructive to look at the Smith chart while adjusting the line arbitrarily over the region 0mm to 1000mm (1 wavelength). This can be done in Microwave Office in real time by using the Tune Tool.

To set up the tuning:

- Click the schematic to make it the active window.
- Select 'tune tool' from the 'Simulate' menu, *or* press F10.
- Click the length parameter of the line TLINP element. It should turn blue. See figure 6.8
- Select 'tune' from the 'Simulate' menu. A dialogue box should appear.

- In the tune dialogue box set the ‘min’ and ‘max’ values to 0 and 1000 respectively. See figure 6.9

- Select ‘Analyse’ from the ‘Simulate’ menu.

- Move the slider in the tune dialogue box to see the effect of increasing the line length in the VSWR graph and Smith chart.

It should not be surprising that the marker spirals towards the centre of the chart as the line length increases. The effect of a lossy line is to make any load look matched from the point of view of the input, if the line is sufficiently long. The VSWR graph should show a reduction in VSWR from 5 when the line length is zero to 1.0184 when the line is 1000mm long. At 375mm the VSWR is 1.308.

6.3 Figures

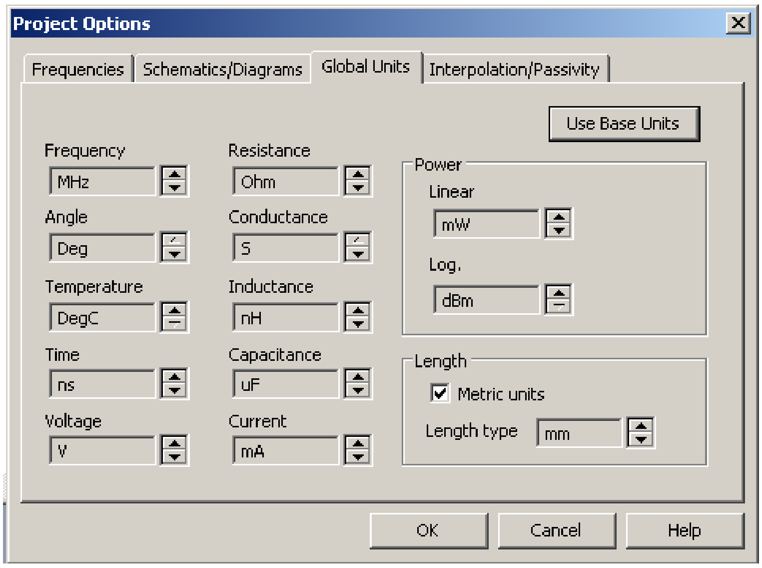


Figure 6.1: Chart 4 : Project Options Dialogue Box

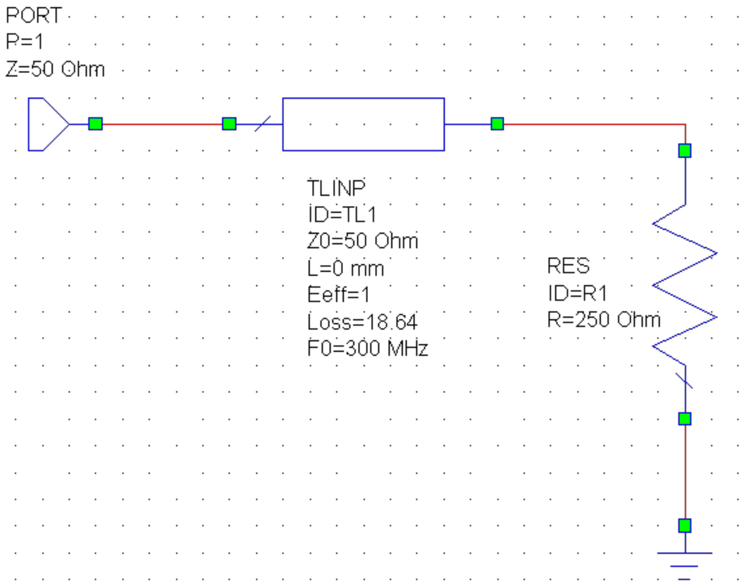


Figure 6.2: Chart 4 : Lossy Line Schematic

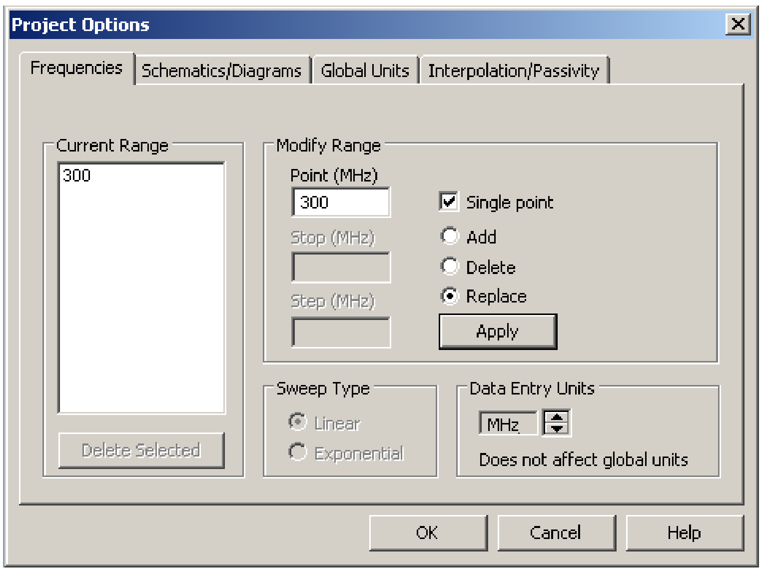


Figure 6.3: Chart 4 : Project Frequencies Dialogue Box

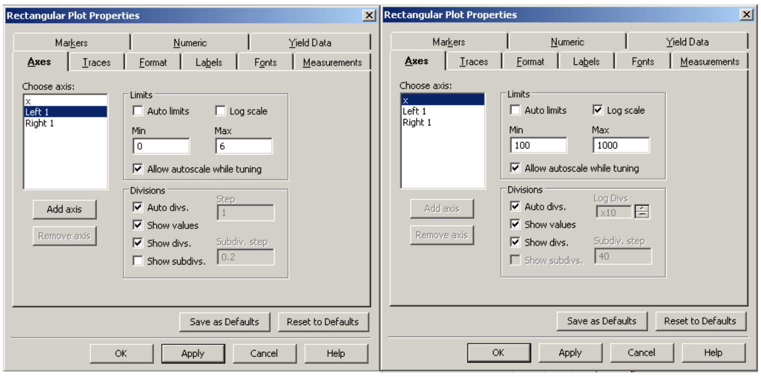


Figure 6.4: Chart 4 : VSWR Graph Properties Dialogue Box

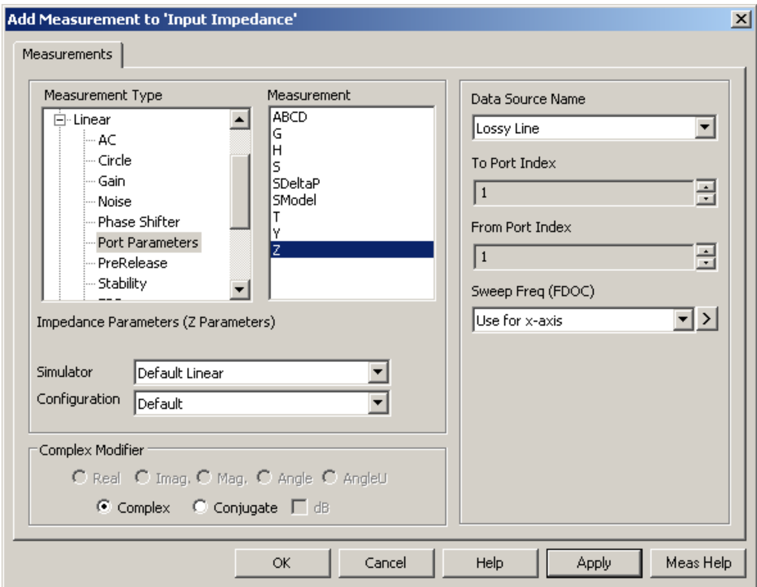


Figure 6.5: Chart 4 : Add Measurement Dialogue Box for Input Impedance (Smith Chart)

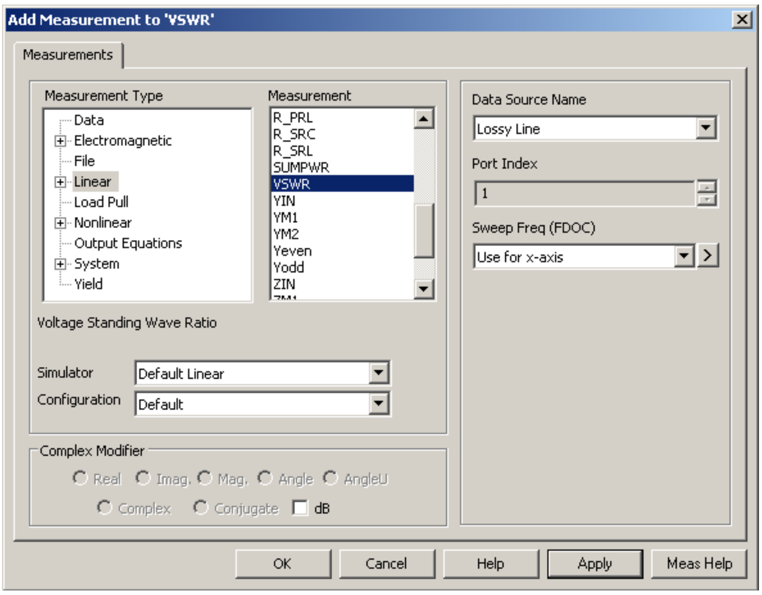


Figure 6.6: Chart 4 : Add Measurement Dialogue Box for VSWR (Rectangular Graph)

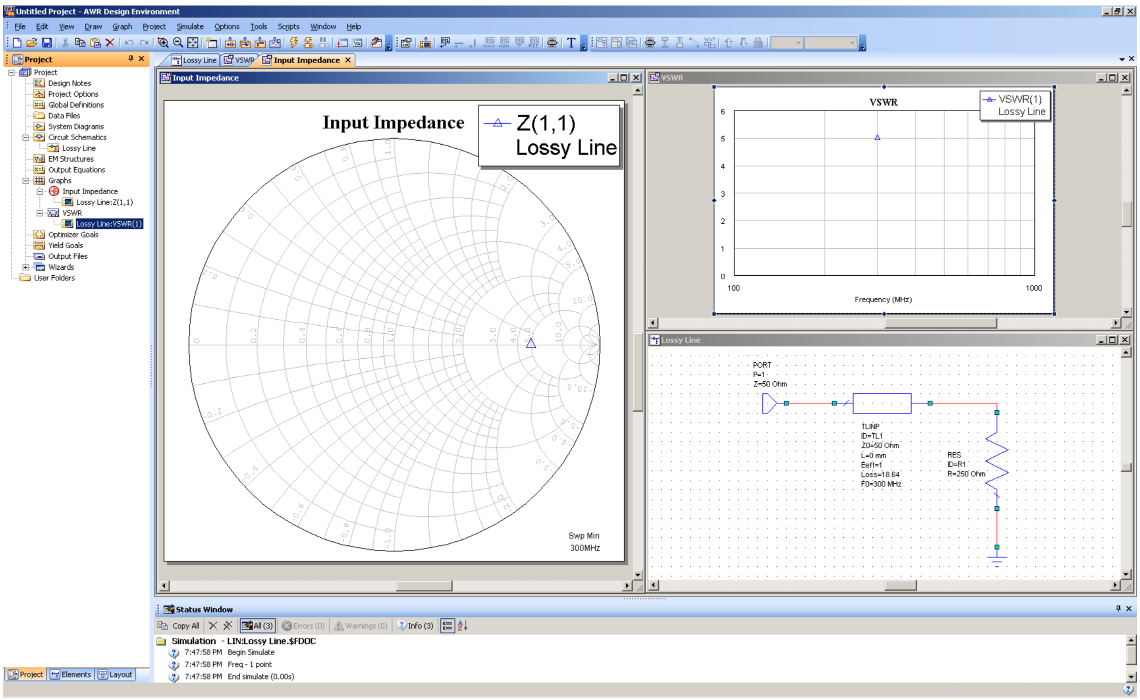


Figure 6.7: Chart 4 : Example of preferred MWO Window Layout

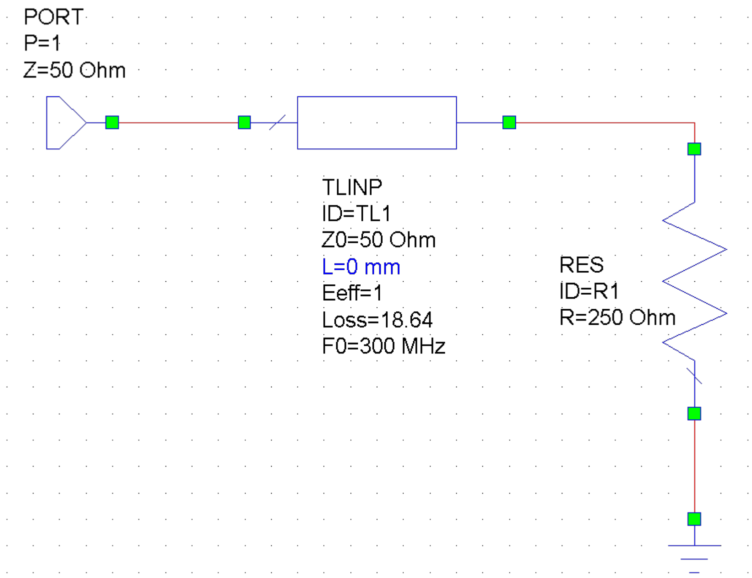


Figure 6.8: Chart 4 : Lossy Line Schematic with TLINP line length parameter selected using the Tune Tool

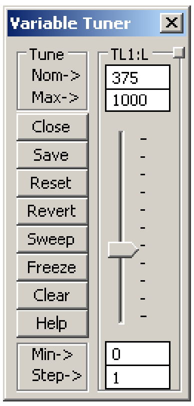


Figure 6.9: Chart 4 : Tune Dialogue Box with Minimum and Maximum set correctly. ‘Norm’ is the value of the marker.

Chapter 7

Chart 5

7.1 Introduction

In this chapter the example in chart 5 is followed. A complex impedance is specified $(0.25 + j0.5)$. This represents a series circuit composed of a (real) resistive element and an (imaginary) reactive element. The objective is to transform this series impedance representation into a parallel admittance representation. Admittance is composed of a (real) conductance and an imaginary (susceptance).

Impedance is represented on a Smith chart by

$$z = r + jx = \frac{1 + \Gamma(d)}{1 - \Gamma(d)}$$

This can be re-written as

$$y = \frac{1}{z} = \frac{1 - \Gamma(d)}{1 + \Gamma(d)} = \frac{1 - \Gamma(d)e^{-j\pi}}{1 + \Gamma(d)e^{-j\pi}}$$

Hence the Smith chart can be used to transform impedance to admittance and vice versa. This is achieved by rotating an arc with radius $|\Gamma(d)|$ through $180° \equiv \frac{\lambda}{4}$

Consider an Example
A point 'A' = $(0.25 + j0.5)$ on the chart represents a series circuit.
so

$$y_A = \frac{1}{(0.25 + j0.5)}$$

to perform the transformation, form an arc of constant radius over 180°to a point'B' at $(0.8 - j1.6)$. This represents a parallel circuit of the form $(G - jB)$ where G = 0.005 and B = 0.01 (denormalised).

7.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart5'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

7.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 7.1. Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

7.2.2 Impedance Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Original Impedance'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right click until the port has its connecting pin facing to the right.
- Place the port with a left click.

The impedance, which will be transformed, is a series connection of a resistive element and a capacitive element. To simplify matters, a complex impedance can be specified.

Add the impedance which will be transformed:

- To add the impedance, in the 'Elements browser' select 'General' - 'Passive' - 'Other' - 'IMPED'.
- Left click and drag the 'IMPED' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left mouse click.
- Double left click the 'R' parameter, change it to 12.5.
- Double left click the 'X' parameter, change it to 25.

If you have already placed the element and need to rotate it, left click it and press Ctrl + R, *or* right click it and select rotate from the pop-up menu, *or* choose 'Rotate' from the 'Edit' menu.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit.

At this point your schematic should look like figure 7.2.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 495MHz.

- Open the ‘Project Options’ from the ‘Options’ menu.
- Ensure your ‘frequencies’ tab is as shown in figure 7.3.
- Click **‘Apply’**, then ‘ok’. If you *only* click ‘ok’ nothing will be changed!

7.2.3 Impedance Chart

Add a Smith chart to the project:

- Right click ‘Graphs’ in the ‘Project Browser’
- Select ‘New Graph’ from the pop-up menu.
- Select ‘Smith Chart’ from the dialogue box which appears.
- Name your chart ‘Input Impedance’.

To appropriately arrange the windows choose ‘Tile Vertically’ from the ‘Window’ menu.

To run a simulation and view the output in Microwave Office a process of ‘Measurements’ is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the ‘Project Browser’.
- In the pop-up menu select ‘Add Measurement’.
- Change your measurement dialogue box to match figure 7.4.
- Click ‘Apply’ and ‘Ok’.

Select ‘Analyse’ from the ‘Simulate’ menu. At this point your Smith Chart should look like figure 7.5.

Now a VSWR circle should be added to the chart.

- Right click the Input Impedance Smith chart icon in the ‘Project Browser’.
- In the pop-up menu select ‘Add Measurement’.
- Change your measurement dialogue box to match figure 7.6.
- Click ‘Apply’ and ‘Ok’.

Select ‘Analyse’ from the ‘Simulate’ menu. At this point your Smith chart should look like figure 7.7

7.2.4 Transformed Impedance

We will now add a second schematic to draw a representation of the transformed impedance. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Transformed Impedance'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right mouse click until the port has its connecting pin facing to the right.
- Place the port with a left mouse click.

The transformed impedance is a series connection of a resistive element and a capacitive element, it is not an admittance yet. This plot shows the point around the VSWR circle that you should move the impedance to in order to produce the correct admittance. It is 180°around the chart.

Add the transformed impedance:

- To add the impedance, in the 'Elements browser' select 'Lumped Elements' - 'Resistor' - 'ZFREQ'.
- Left click and drag the 'ZFREQ' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left mouse click.
- Double left click the 'R' parameter, change it to 40.
- Double left click the 'X' parameter, change it to -80.

Now plot this on the Input Impedance Chart

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 7.8.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 7.9.

7.2.5 Admittance

The transformed impedance is now converted to an admittance. By any means you choose show that:

$$y = \frac{1}{(40 - j80)} = (0.005 + j0.01) \tag{7.1}$$

We will now add a schematic to draw a representation of the admittance circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'. Name your schematic 'Admittance'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right mouse click until the port has its connecting pin facing to the right.
- Place the port with a left click.

The admittance is a parallel connection of a conductance element and a susceptance element. To simplify matters, a complex admittance can be specified.

- To add the admittance, in the 'Elements browser' select 'General' - 'Passive' - 'Other' - 'ADMIT'.
- Left click and drag the 'ADMIT' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking.
- Place the load with a left click.
- Double left click the 'G' parameter, change it to 0.005.
- Double left click the 'B' parameter, change it to 0.01.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu. Connect the elements.

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Admittance'.
- Right click the chart
- Select Properties from the pop-up menu

- The 'Grid' tab should be set up according to fig 7.10

To appropriately arrange the windows, minimise all windows except the Admittance Schematic, and the Input Admittance Smith Chart, then choose 'Tile Vertically' from the 'Window' menu.

Now Plot the admittance on the Input Admittance chart

- Right click the Input Admittance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 7.11.
- Click 'Apply' and 'Ok'.

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 7.12.

It should be simple to confirm that $y = (0.25 - j0.5)$S on the 50 normalised Smith chart represents a denormalised value of (0.005 - j0.01)S. Hence the transformation via the Smith chart has provided the same answer as the complex algebra in Eqn. 7.1.

7.3 Figures

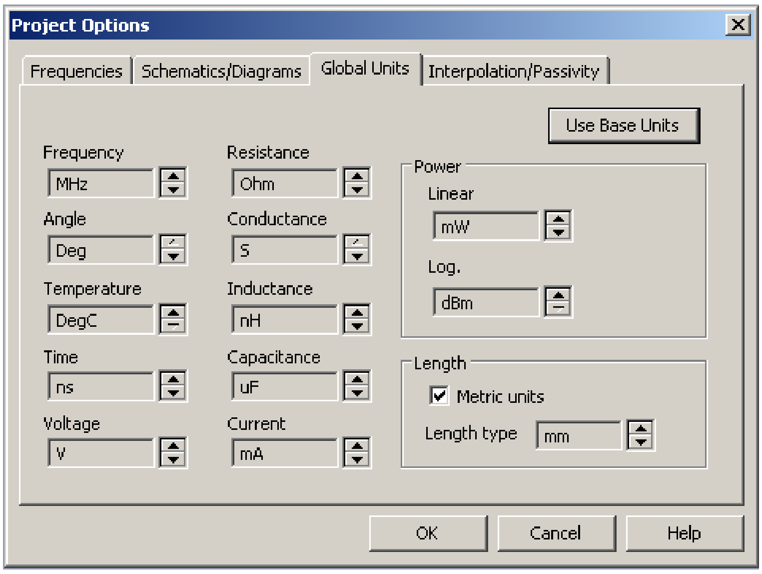


Figure 7.1: Chart 5 : Project Options

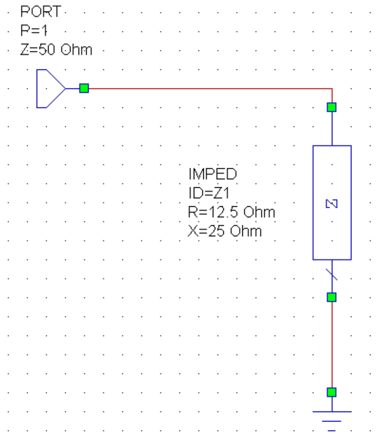


Figure 7.2: Chart 5 : Impedance Schematic

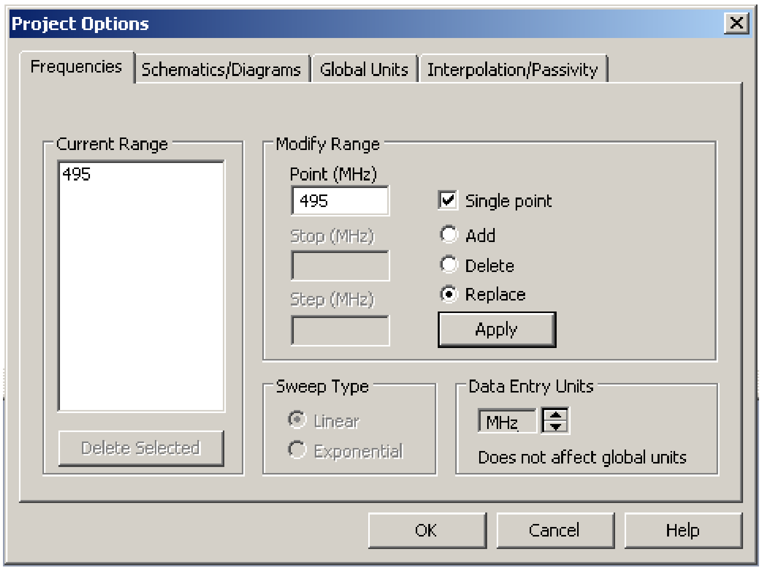


Figure 7.3: Chart 5 : Project Frequencies

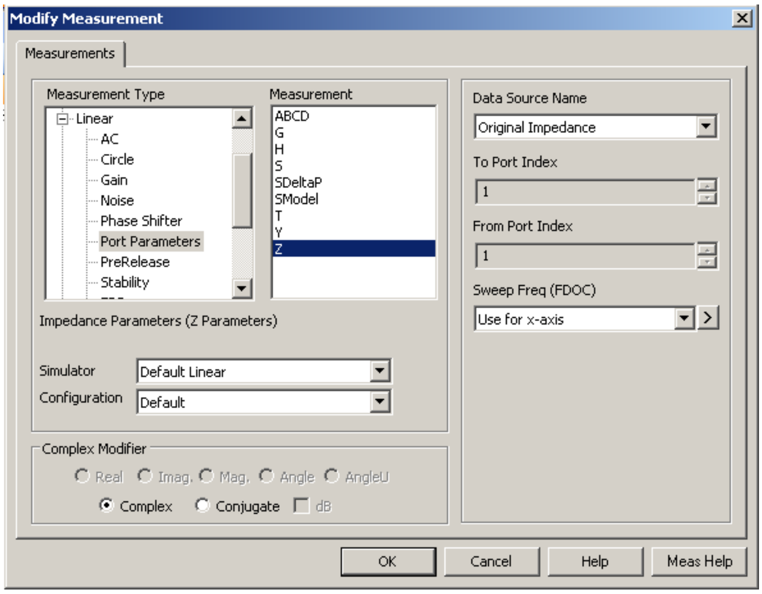


Figure 7.4: Chart 5 : Add Measurement Dialogue Box (For Input Impedance)

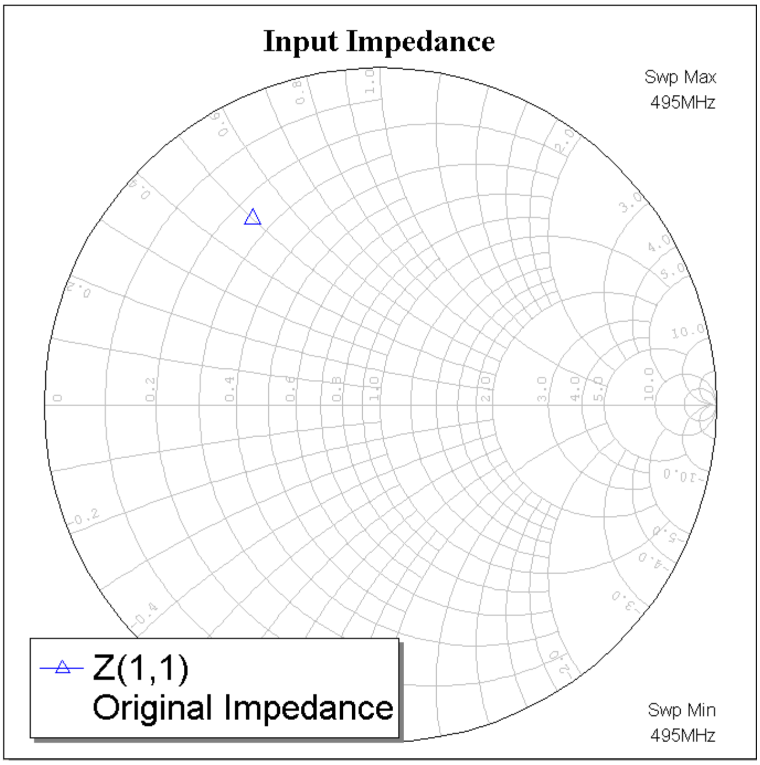


Figure 7.5: Chart 5 : Input Impedance Smith Chart

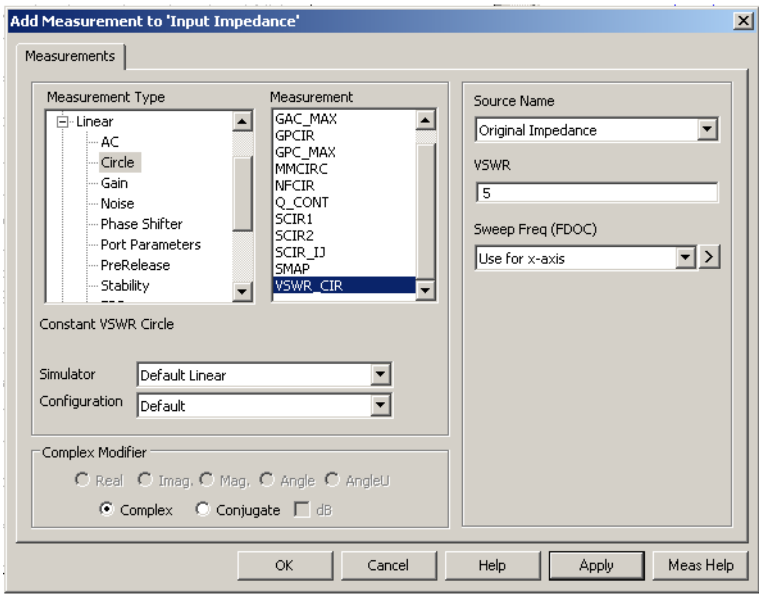


Figure 7.6: Chart 5 : Input Impedance VSWR Measurement

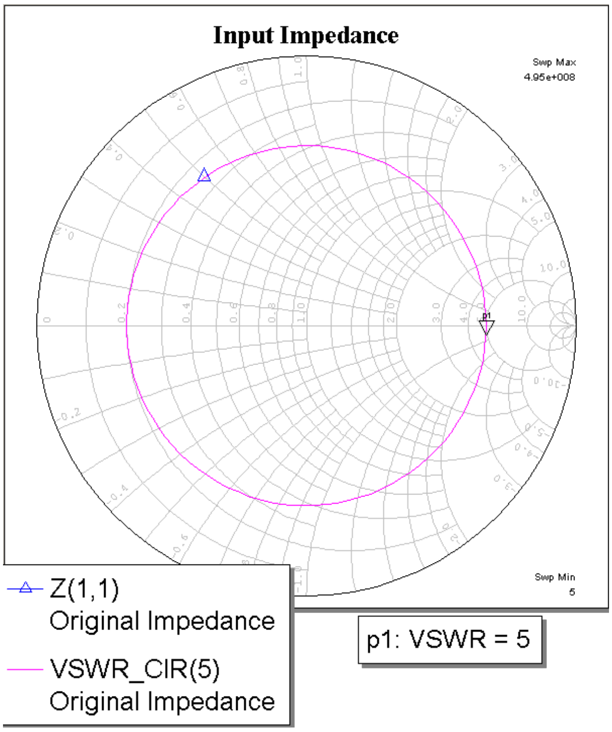


Figure 7.7: Chart 5 : Input Impedance Smith Chart, with VSWR Circle

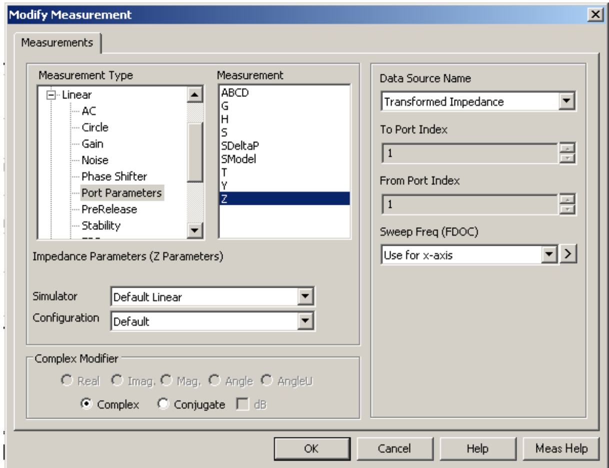


Figure 7.8: Chart 5 : Add Measurement Dialogue Box for 'Transformed' Impedance

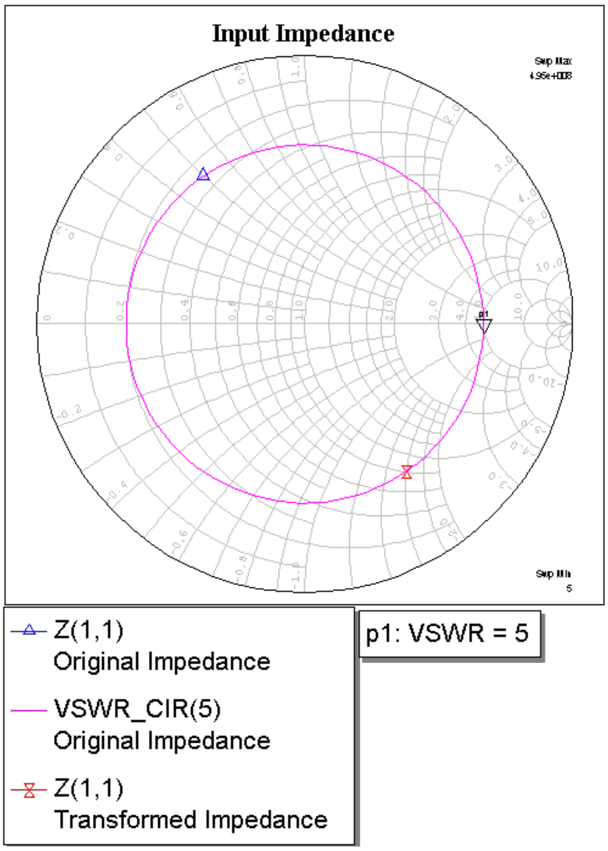


Figure 7.9: Chart 5 : Input Impedance Smith Chart, with 'Transformed' Impedance

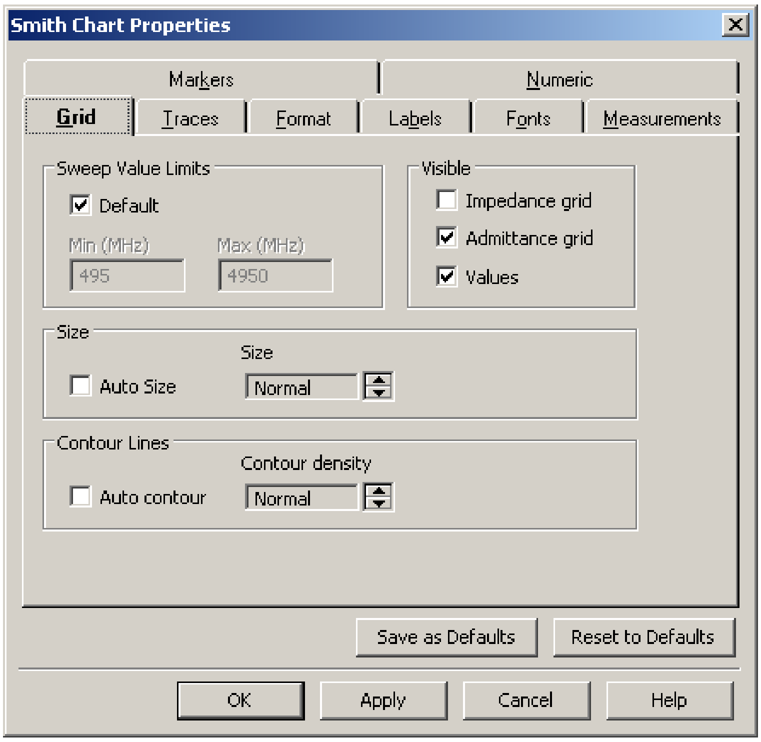


Figure 7.10: Chart 5 : Input Admittance Smith Chart Properties Dialogue Box

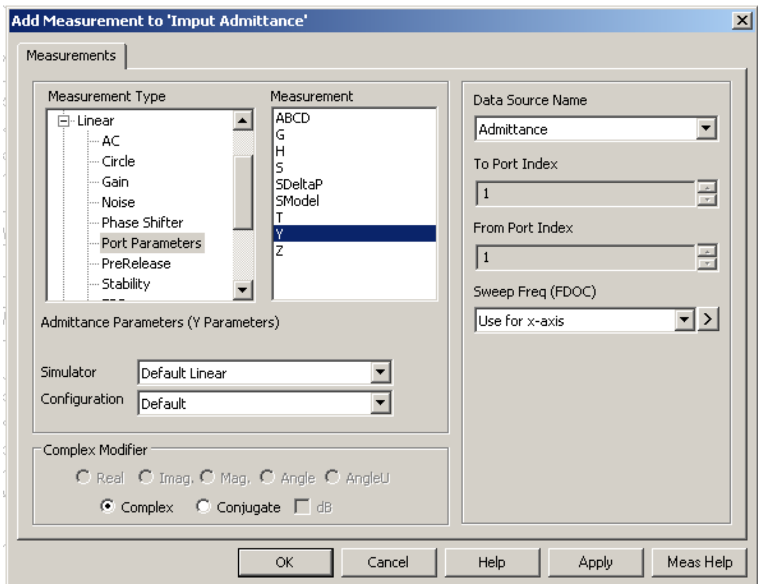


Figure 7.11: Chart 5 : Add Measurement Dialogue Box for Input Admittance

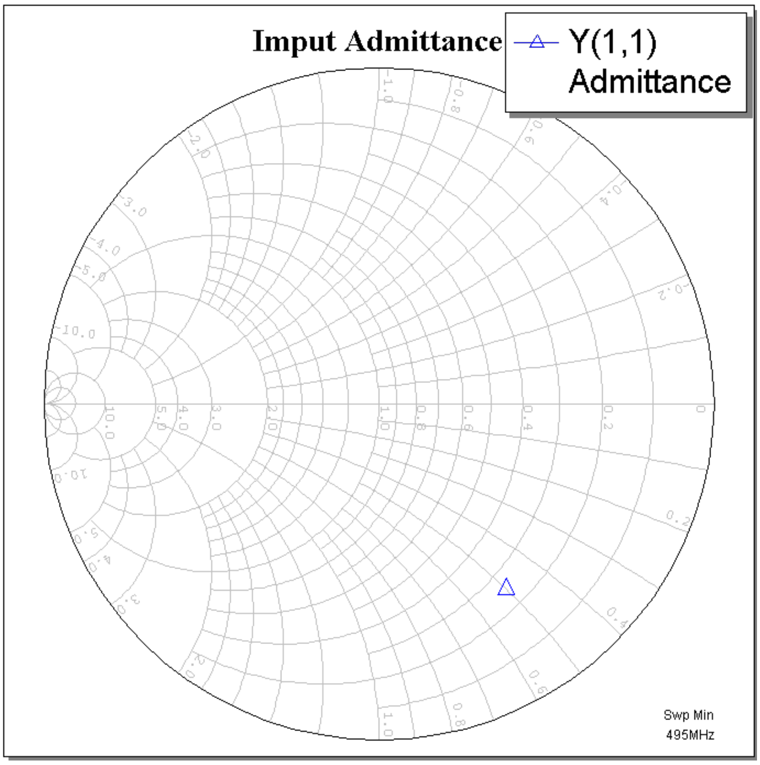


Figure 7.12: Chart 5 : Input Admittance Smith Chart

Chapter 8

Chart 6

8.1 Introduction

In this chapter the example in chart 6 is followed. A complex load is specified and connected to a transmission line. The transmission line is used to make the load appear real at the line input. Another transmission line is used as a quater wave transformer to impedance match the (now real) load impedance to a generator of specified source impedance. The exact question, and the solution given in the lecture notes are given below.

> Given a lossless transmission line with $Z_O = 75\Omega$ and Z_L=(150 - j112)Ω, design a matching section that employs a $\frac{1}{4}\lambda$ transformer to match this load to a generator with $Z_S = 75\Omega$ at a frequency of 920MHz. Note since Z_L is complex it must be made real first.

The solution given in EEE403 is as follows.

- Normalise Z_L to $Z_L = \frac{(150-j112)}{75} = (2 - j1.49)$.
- Plot the load (2 - j1.49) 'point A'.
- Construct a radial line from the centre through point A.
- Note the reading on the angular scale 'Towards Generator' (0.2912λ).
- Construct a circle with radius A. This is the VSWR circle. Its value can be read as 3.3.
- Move from point A clockwise to point B on the chart and note the reading on the 'Towards Generator' scale. (0.5λ)
- The required line length to make the complex load impedance appear real can be found by measuring the length from A to B. l = (0.5 - 0.292) = 0.208.
  The impedance at point B is 75 $\times$ (0.3 + j0) = (22.5 + j0)Ω.

- A $\frac{\lambda}{4}$ transformer will be used to match this impedance to $Z_S = 75\Omega$. From Equation 18 (in your course notes) $Z_O = \sqrt{75 \times 22.5} = 41.4\Omega$
- The frequency is 920MHz so

$$\lambda = \frac{Vp}{f} = \frac{3 \times 10^8}{920 \times 10^6} = 0.326m$$

The transmission line length is $0.21\lambda = 6.85$cm.
The transformer length is $0.25\lambda = 8.15$cm.

8.2 Microwave Office Implementation

First it is necessary to create a new project and save it as 'Chart6'. Choose 'File' - 'New', then 'File' - 'Save Project As'.

8.2.1 Setup

To make sure that your project has the same units as are used in these notes select 'Project Options' from the 'Options' menu. Ensure that your 'global units' tab matches figure 8.1. Microwave Office may complain that there will be rounding errors in the database. Don't worry about it. This error is only significant if the project will contain electromagnetic structures or schematic layouts. This example contains neither.

8.2.2 Schematic

We will now add a schematic to draw a representation of the circuit. In the 'Project Browser' right click 'Circuit Schematics' and in the pop-up menu that appears select 'New Schematic'.

Populate the schematic with a port.

- To add the port click the 'Port' icon in the tool bar *or* in the menus choose 'Draw' - 'Add Port' *or* press Ctrl + P on the keyboard.
- Move the mouse over the schematic
- Right click untill the port has it's connecting pin facing to the right.
- Place the port with a left click.

The load is a series connection of a resistive element and a capacitive element. To simplify matters, a complex load can be specified.

Also add the load:

- To add the load, in the 'Elements browser' select 'Lumped Components' - 'Resistor' - 'ZFREQ'.
- Left click and drag the 'ZFREQ' element on to the schematic.
- Rotate the element into a vertical orientation by right clicking
- Place the load with a left click.

If you have already placed the element and need to rotate it, left click it and press Ctrl + R, *or* right click it and select rotate from the pop-up menu, *or* choose 'Rotate' from the 'Edit' menu.

Add a ground by clicking the ground icon in the tool bar, *or* select 'Add Ground' from the 'Edit' menu.

To connect the elements move the mouse near to a component leg. A wire icon will appear when the mouse is near to a component leg. Left click to make connections. Connect the circuit *and* change the port and load data to match figure 8.2.

You may choose to produce a schematic similar to figure 8.3 which will accomplish an identical task. The capacitance is calculated with the following formula:

$$C = \frac{1}{2\ \pi\ 920 \times 10^6 \times 112} = 1.5445pF$$

Don't forget that $\frac{1}{j} = -j$. The resistor element is called 'res', the capacitor element is 'cap'. Both are under 'Lumped Elements' in the 'Elements Browser'.

8.2.3 Load Impedance

Add a Smith chart to the project:

- Right click 'Graphs' in the 'Project Browser'
- Select 'New Graph' from the pop-up menu.
- Select 'Smith Chart' from the dialogue box which appears.
- Name your chart 'Input Impedance'.

To appropriately arrange the windows choose 'Tile Vertically' from the 'Window' menu.

It is necessary to tell Microwave Office that we will be using $Z_O = 75\Omega$.

- Right click the Smith Chart and select 'Properties'.
- Ensure that your dialogue box looks like the one in figure 8.4.

To run a simulation and view the output in Microwave Office a process of 'Measurements' is used. A measurement of the impedance looking into the load can be set-up as follows.

- Right click the Input Impedance Smith chart icon in the 'Project Browser'.
- In the pop-up menu select 'Add Measurement'.
- Change your measurement dialogue box to match figure 8.6.
- Click 'Apply' and 'Ok'.

Prior to simulating it is necessary to specify the frequencies over which a result is required. In this case only one frequency is of interest, 920MHz.

- Open the 'Project Options' from the 'Options' menu.
- Ensure your 'frequencies' tab is as shown in figure 8.7.
- Click **'Apply'**, then 'ok'. If you only click 'ok' nothing will be changed!

Select 'Analyse' from the 'Simulate' menu. At this point your Smith Chart should look like figure 8.8.

8.2.4 Voltage Standing Wave Ratio

The next part of this question requires the VSWR. You will have noticed that the Smith charts provided in the EEE403 lectures have considerably more detail around the circumference. This is because the chart given out in the lectures is designed to be used manually, whereas the Microwave Office chart is not.

To display the VSWR,

- Add a 'Rectangular' Graph (right click the 'Graphs' tree in the 'Project Browser', select 'New Graph'.)
- Name it 'VSWR'.
- Add a measurement which conforms to figure 8.5. This is done by right clicking the VSWR graph in the 'Project Browser' and in the pop-up menu select 'Add Measurement'.
- Simulate, by choosing 'Analyse' from the 'Simulate' menu.

The resulting output should look similar to figure 8.9. You can tidy this plot up by right clicking it and editing its properties. The value of VSWR agrees well with the example we are following.

8.2.5 Transmission Line Length

Insert into your schematic a 'COAX2' element. This can be found in the 'Elements Browser' under 'Transmission Lines' - 'Coaxial' - 'Electrical'. Change the Z parameter to 75. Change F_O to 920 and change the ID to 'TransmissionLine'. The schematic should look similar to figure 8.10. Run the simulation again. You should find that the blue triangle in the Smith chart has moved.

In the lecture notes the length of the line required to make the load look real, from the point of view of the quarter wave transformer, is found by reading the scales around the edge of the Smith chart. As was said earlier there are no scales around Microwave Office Charts. The electrical length of the line required to make the load look purely resistive must be found. If this important point is neglected the quarter wave transformer will not work properly. Two methods will be introduced to find the electrical length required.

8.2.6 The Tune Tool

The Tune tool is a method by which many parameters of a circuit can be varied and the resulting change in the Smith chart viewed in real time. This is a very powerful method of checking 'what if?' scenarios. It can also be thought of as producing an interactive Smith chart, which is a good learning tool. It gives insight into how the Smith chart represents what is going on in the circuit.

We will tune the electrical length of the line.

- Begin by selecting the 'Tune Tool' from the 'Simulate' menu. The cursor should change into a screwdriver.
- Click the 'electrical length (EL)' parameter of the transmission line in the Schematic. It should turn blue.
- Select 'Tune' in the 'Simulate' menu. A dialogue box similar to figure 8.11 should appear.
- You may have to edit the maximum and minimum values to match the example.
- Arrange your windows such that the Smith chart and the schematic are next to each other and the VSWR graph is not visible. The easiest way to achieve this is by closing the VSWR graph and then selecting 'Tile Vertically' from the 'Window' menu.

Look at the Smith chart while moving the tuning slider. You will find that the blue marker moves in a circle around the centre of the chart. The present value of the electrical length can be read from the uppermost text box in the tune dialogue box. You should find that an electrical length of 75°produces an impedance with only a real part (from the point of view of the generator). A length of 165°also produces a real impedance. It should not surprise you to find that (165 - 90) = 75°given what you have learned from Chart 5. The lower number is the prefered answer.

We could go on to design the quarter wave transformer now using the answer of 75°, and the error would be small. However, Microwave Office has an optimisation tool. It can be used to obtain an answer accurate to 4 s.f. The optimiser will be introduced prior to discussion of the quarter wave transformer. To prepare to use the optimiser the tune tool must be closed.

- Close the 'Tune' dialogue box.
- Un-select the electrical length of the transmission line (left click it). The text should turn black again.
- Choose 'Tune Tool' from the 'Simulate' menu to return to an ordinary cursor.

8.2.7 Optimiser

The optimiser requires the definition of 'goals' and the selection of 'parameters'. The goals are the desired result. The parameters are things we are prepared to change in order to bring about the goals. In this case, the electrical length of the transmission line is the parameter. The goal is to obtain an impedance looking in to the transmission line that only has a real component. Since we have already used the tune tool to get an answer close to optimum we should give the optimiser a challenge. Set the electrical length of the transmission line to 125°. This should result in the marker, on the Smith chart, resting close to (0.65 + j0.95), far from the condition we require.

To set the goals:

- Right click the ‘Optimizer Goals’ tree in the ‘Project Browser’
- In the resulting pop-up menu select ‘Add Optimizer Goal’.
- A dialogue box similar to figure 8.12 will appear. If there are any goals already listed, make sure they are disabled by un-checking the ‘Enable Goal’ box.
- Click ‘New/Edit Meas...’. The new measurement dialogue box appears.
- Change your measurement box to look like figure 8.13.
- Dismiss the optimisation measurement dialogue box by clicking ‘Ok’.
- Change the ‘New Optimisation Goal’ dialogue box to look like figure 8.14. Carefully note each option, or your optimisation will provide an incorrect result.

Having set the goals the parameters must be defined.

- Press F7 *or* choose ‘Optimization’ from the ‘Simulate’ menu.
- Select the ‘Variables’ tab from the bottom of the dialogue box.
- Select the check box that corresponds to optimisation of the electrical length (EL) of the transmission line. When you have done this the dialogue box should look similar to 8.15.

Prior to setting up the main optimiser settings and running the simulation you should arrange the Microwave Office window so that the Smith chart and the schematic are next to eachother, and the VSWR graph is closed. The easiest way to achieve this is by closing the VSWR graph and then selecting ‘Tile Vertically’ from the ‘Window’ menu.

- Click the ‘Optimiser’ tab.
- Set the method to ‘Random (Local)’
- Set the number of iterations to ‘500’.
- Click ‘Start’ and watch the Smith chart. The result should look similar to figure 8.16.

Looking at the schematic the electrical length of the line has changed to a little over 75°. Observation of the Smith chart shows that the impedance has a very small imaginary part. Left click and hold the blue marker to show the impedance at this point. It is approximately 22Ω, which is consistent with the example.

8.2.8 Quarter Wave Transformer

The quarter wave transformer is quite simple to implement.

- Take another ‘COAX2’ element from the ‘Element Browser’ and place it in your schematic between the transmission line and the port.
- Set the electrical length to 90°and the F_O to 920MHz. Set the ID to ‘QuaterWave’.

- Connect the schematic to match 8.17.

You now have three options.

1. Use the formula given in the lecture notes (and at the beginning of this example) to find the correct Z_O to match the Transmission line to the Generator.
2. Use the optimiser to search for a change in the Z_O parameter of the quarter wave transformer to realise a goal of the input impedance having a real part equal to 75Ω.
3. Use the tune tool in place of the optimiser.

The processes required to use the tune tool and the optimiser are the same as shown for the electrical length of the transmission line.

Once you have entered the correct characteristic impedance for the quarter wave transformer the VSWR graph should look similar to 8.18. The axes have been modified to be more aesthetically pleasing. The Smith chart should be similar to 8.19

8.3 Figures

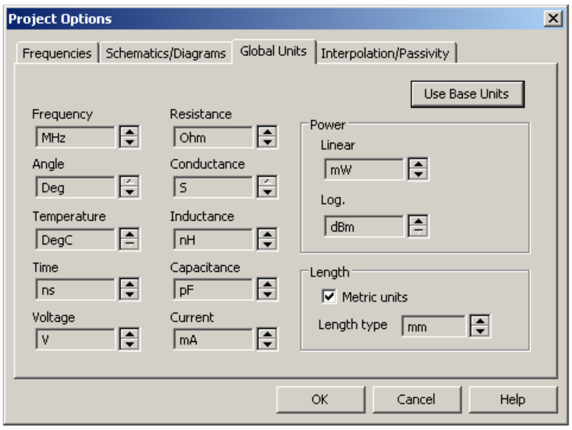


Figure 8.1: Chart 6 : Project Options

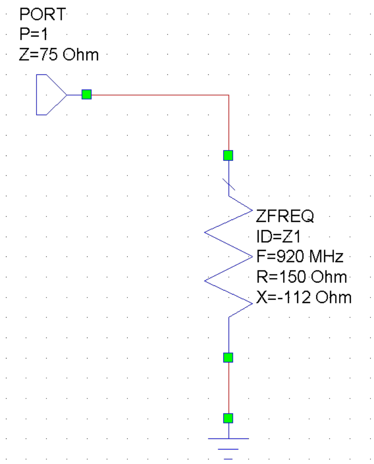


Figure 8.2: Chart 6 : Schematic 1

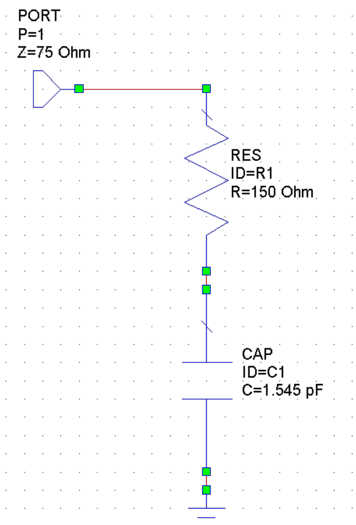


Figure 8.3: Chart 6 : Schematic 2

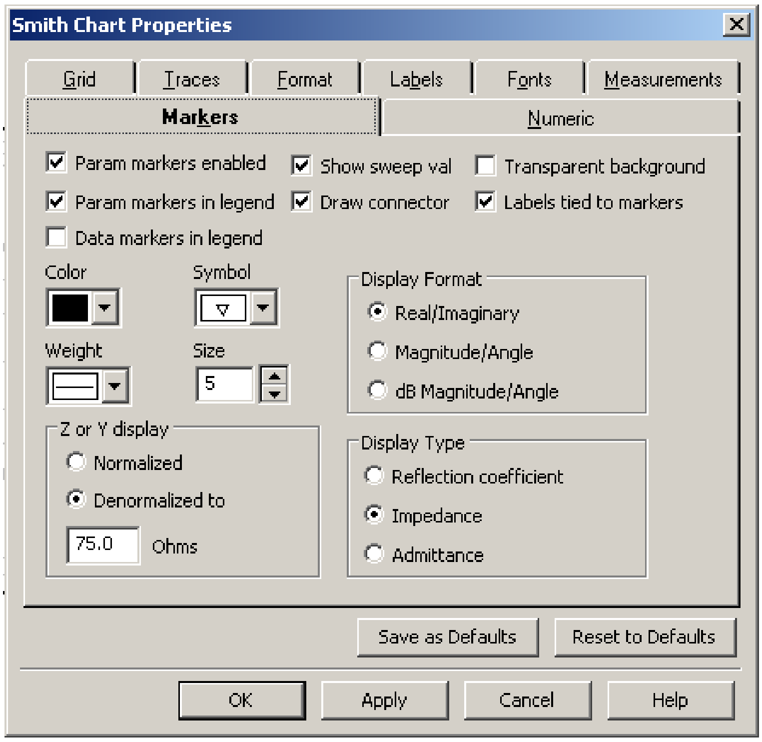


Figure 8.4: Chart 6 : Smith Chart Properties

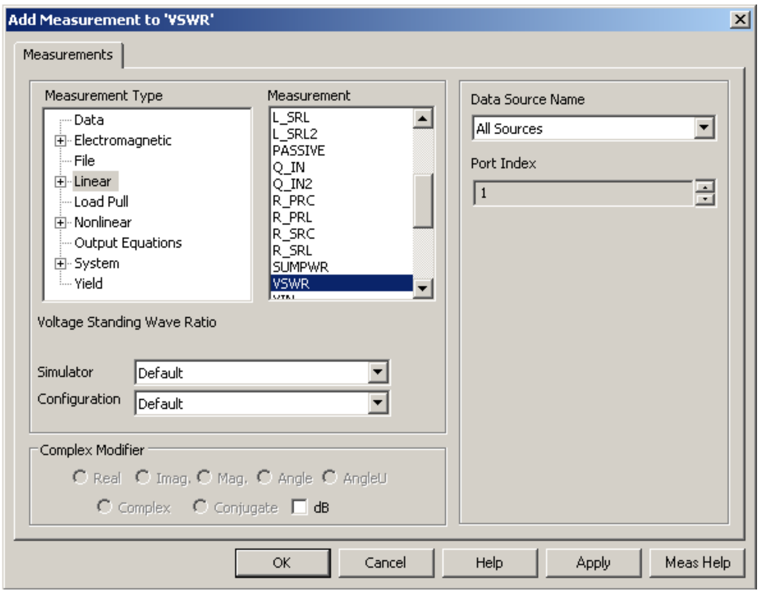


Figure 8.5: Chart 6 : VSWR Measurement Setup

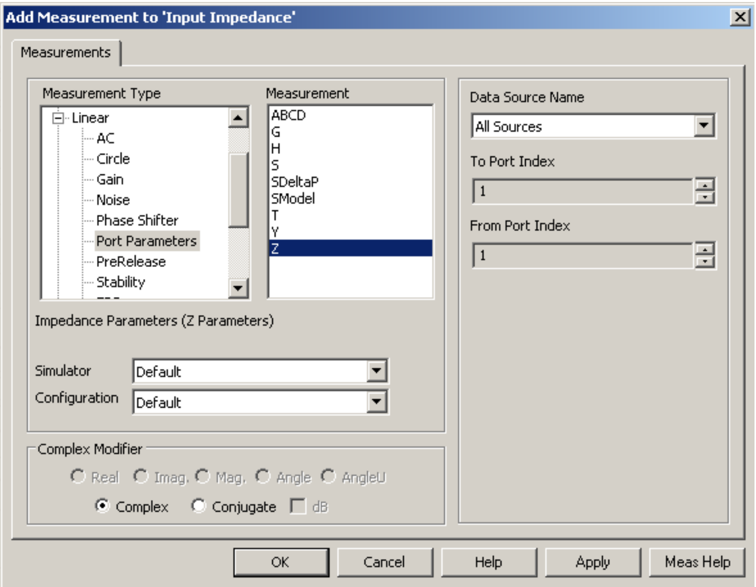


Figure 8.6: Chart 6 : Add Measurement

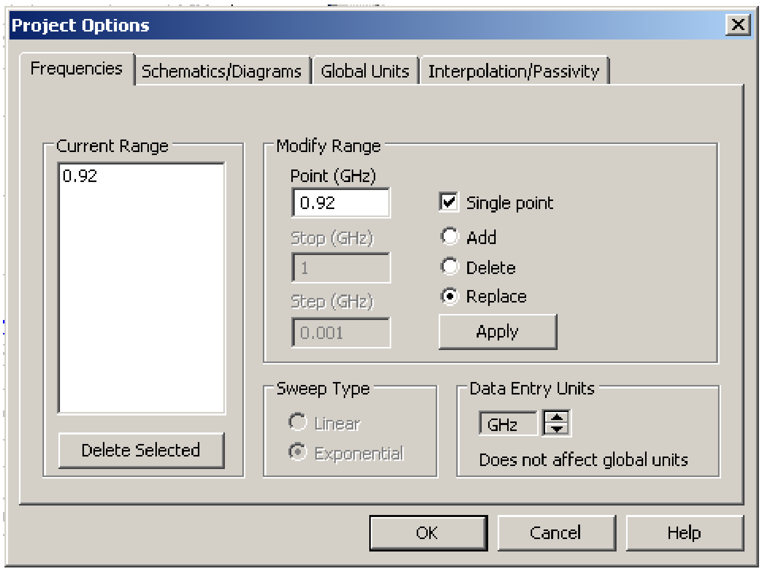


Figure 8.7: Chart 6 : Set Simulation Frequencies

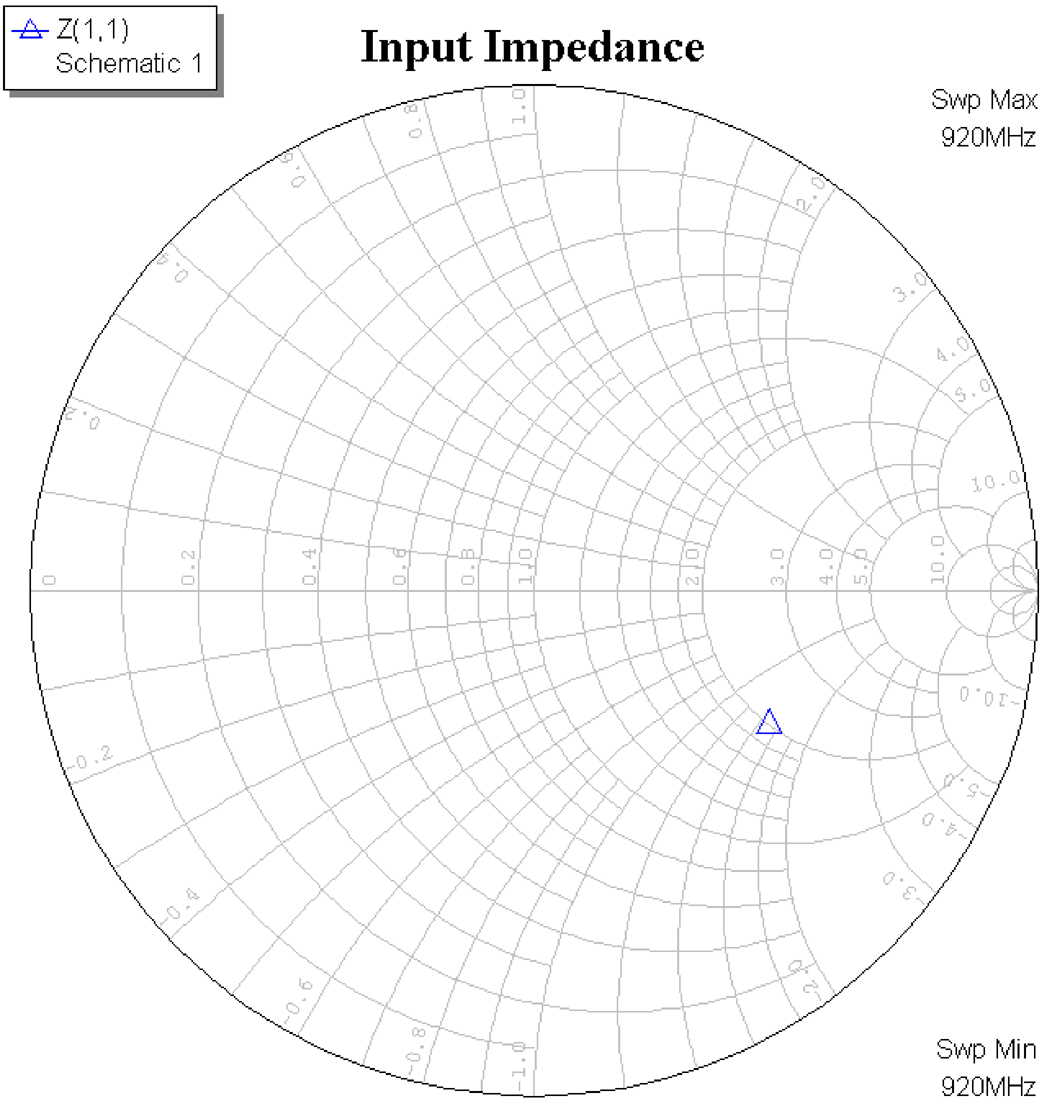


Figure 8.8: Chart 6 : Smith Chart 1 : Load Impedance

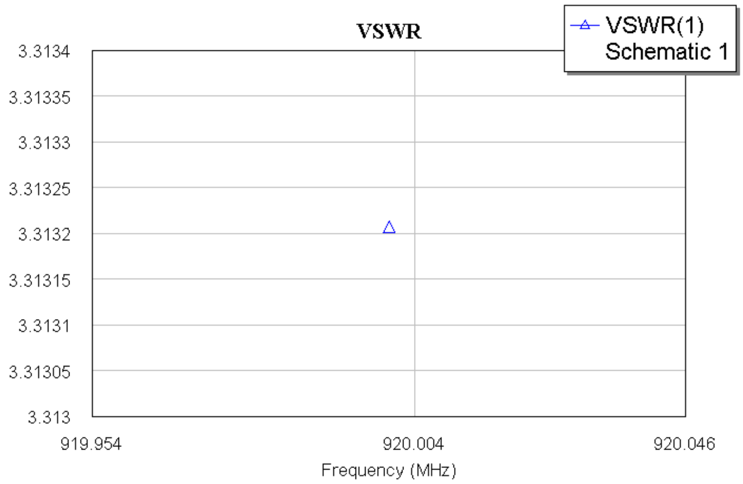


Figure 8.9: Chart 6 : VSWR Result

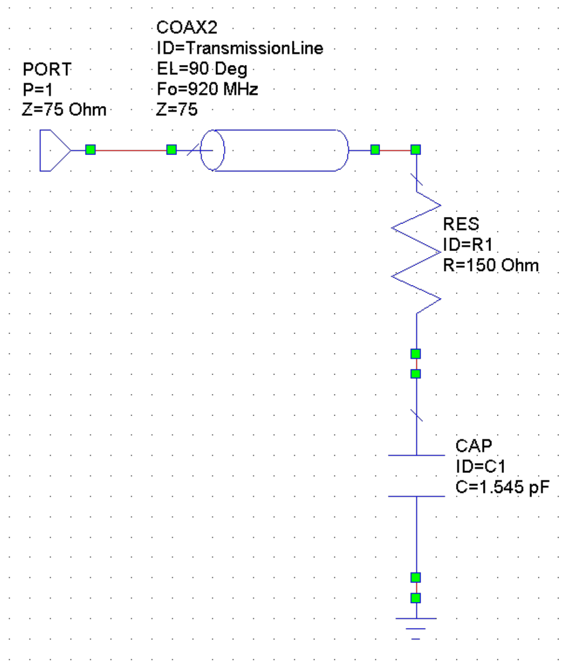


Figure 8.10: Chart 6 : Schematic 3

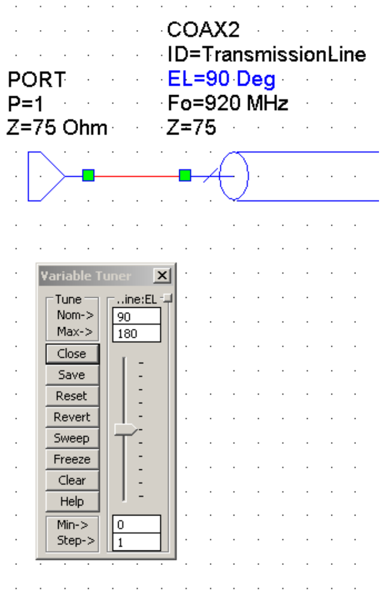


Figure 8.11: Chart 6 : Tuning the Circuit

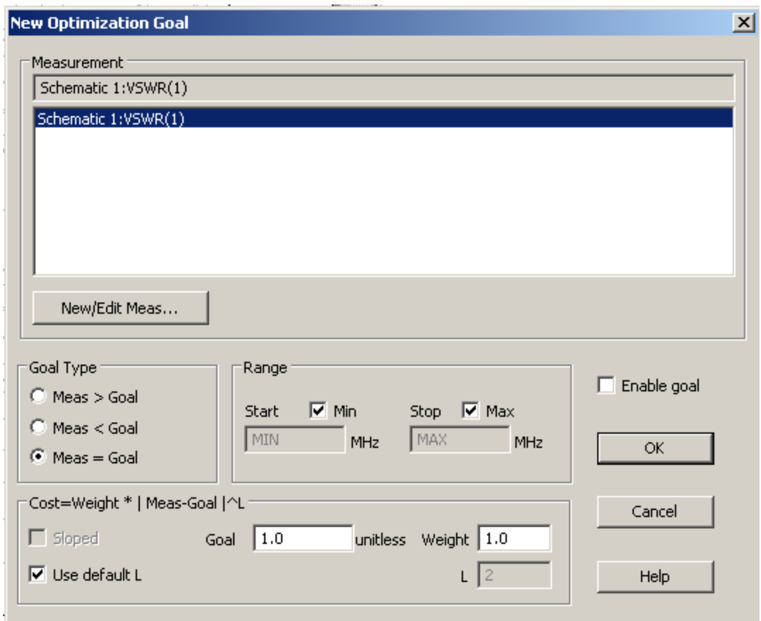


Figure 8.12: Chart 6 : Optimisation Goals Dialogue Box

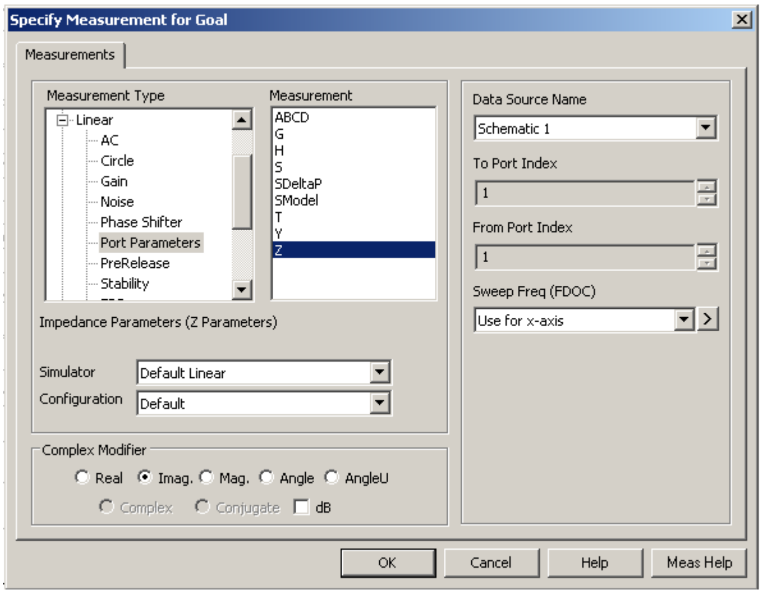


Figure 8.13: Chart 6 : Optimisation Measurement Dialogue Box

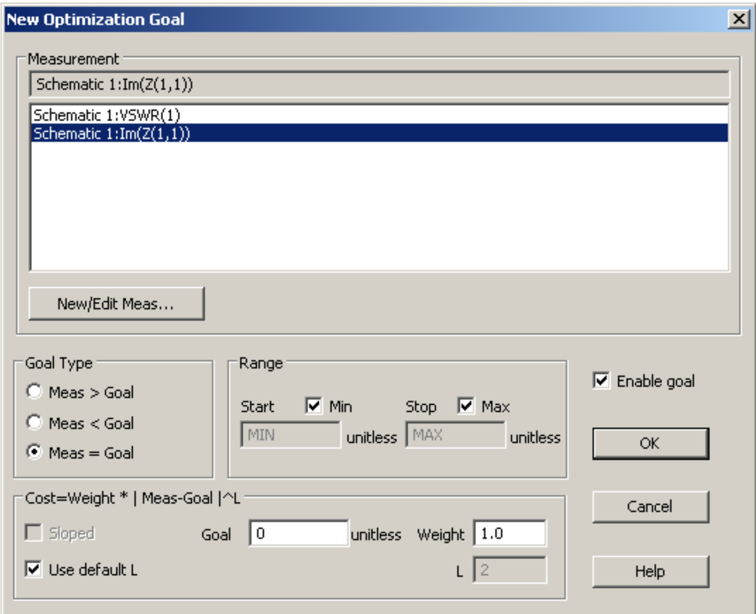


Figure 8.14: Chart 6 : Optimisation Goals Dialogue Box

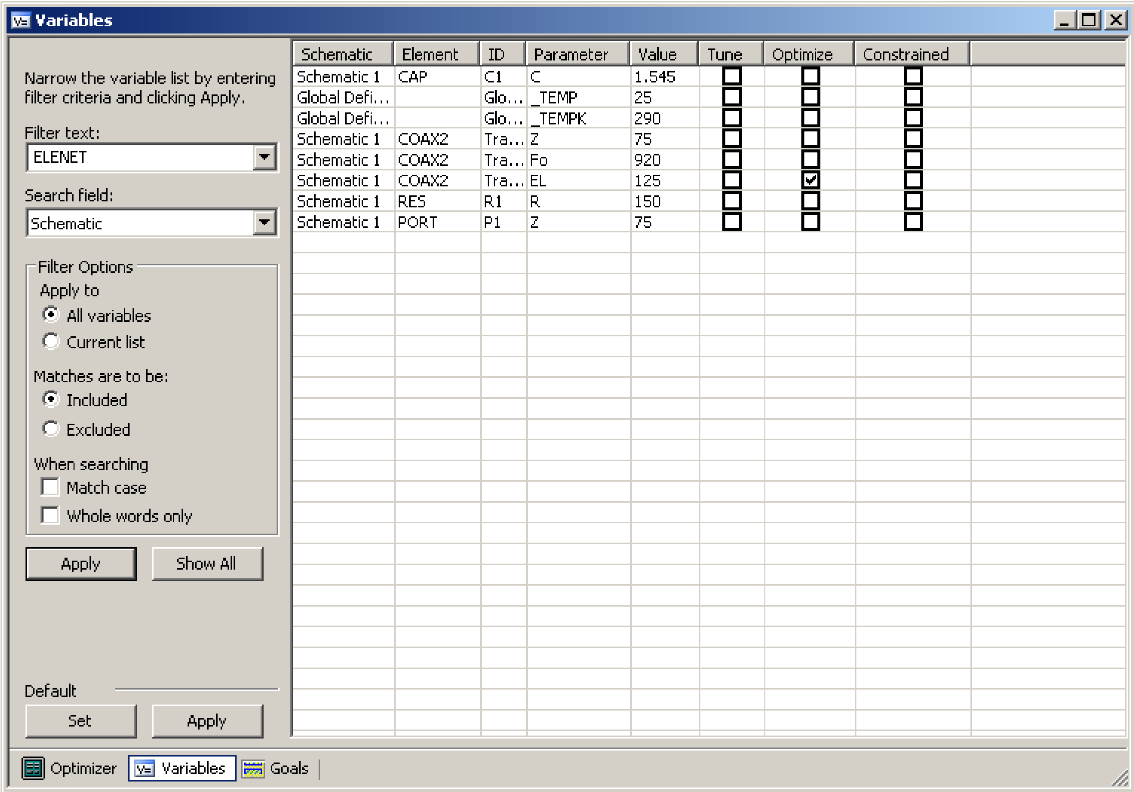


Figure 8.15: Chart 6 : Optimisation Dialogue Box

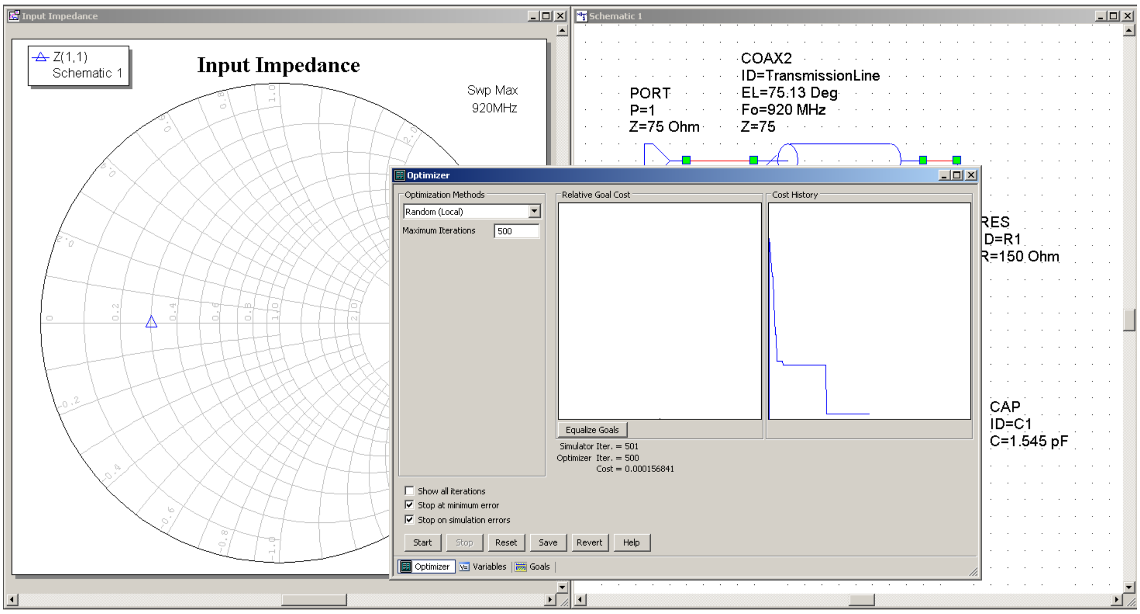


Figure 8.16: Chart 6 : Optimisation Result

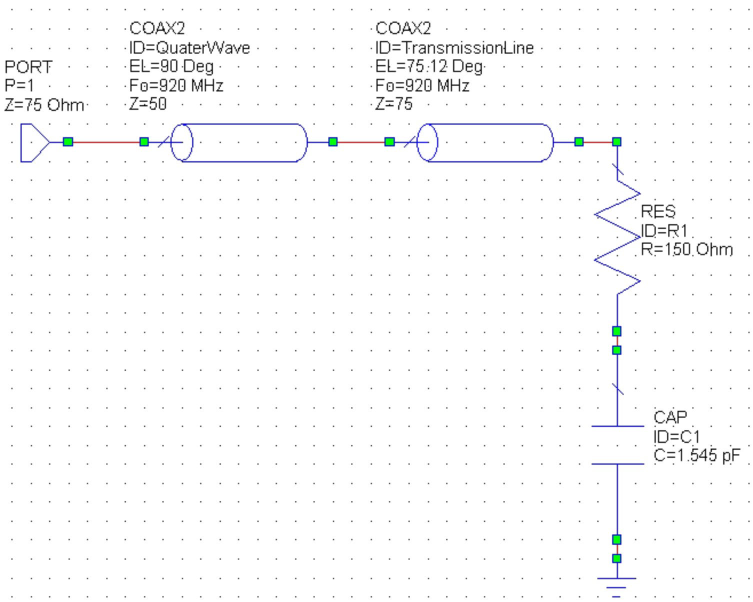


Figure 8.17: Chart 6 : Schematic 4

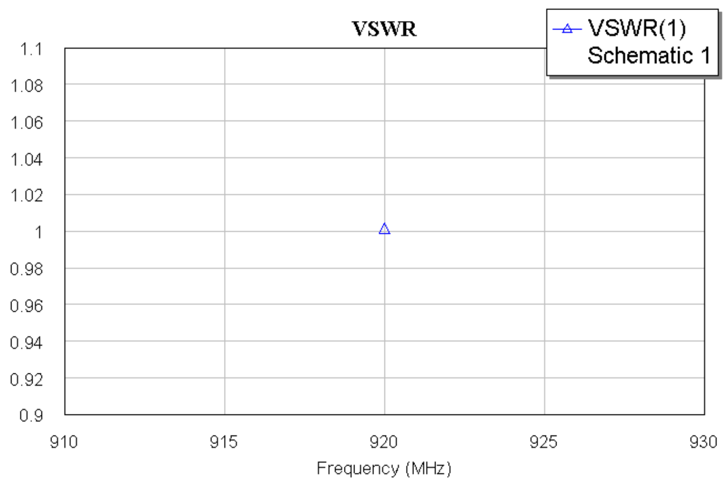


Figure 8.18: Chart 6 : VSWR Graph showing Matched Condition

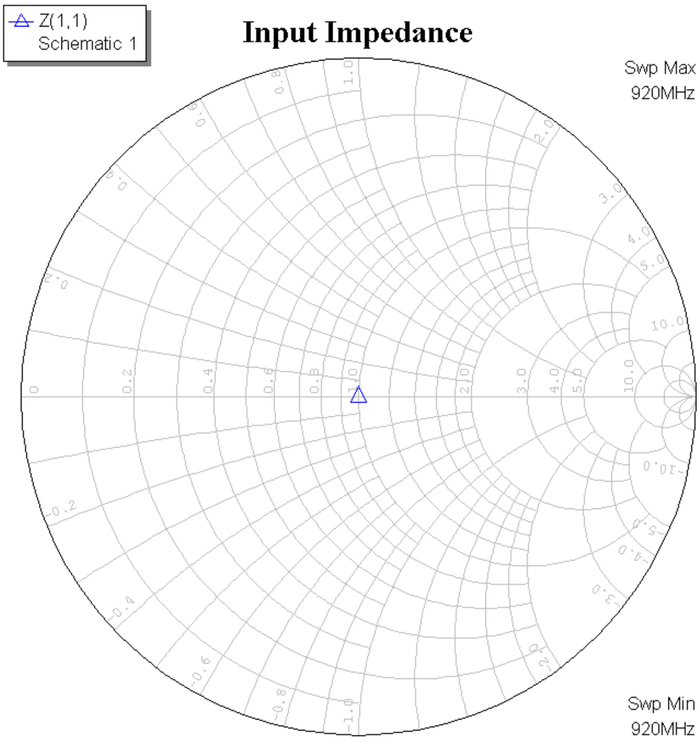


Figure 8.19: Chart 6 : Smith Chart showing the Matched Condition

Chapter 9

Chart 7

9.1 Introduction

The example given in chart 7 follows on from chart 6. The question requires the use of the chart 6 simulations to find out the effect of the impedance missmatch if the operating frequency is changed. If this problem were attempted on a manual Smith chart there would be several steps involved. These are documented below for completeness. In Microwave Office the computation of a solution over a range of frequencies is a simple matter requiring almost no extra work.

Question

For the $\frac{\lambda}{4}$ transfomrer designed in the previous example find the effect of impedance mismatching if the opperating frequency is changed to 1012MHz.

Solution

$f = 1012$MHz so

$$\lambda = \frac{V_p}{f} = \frac{3 \times 10^8}{1012 \times 10^6} = 0.2964m$$

The transmission line length which connects the load to the quater wave transformer is

$$6.85cm = \frac{0.0685}{0.2964} = 0.23\lambda$$

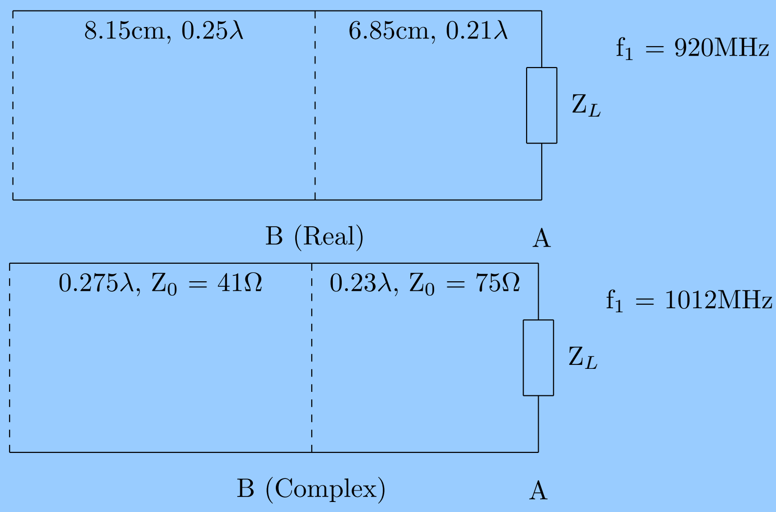
The length of the quater wave transformer section is

$$8.15cm = \frac{0.0815}{0.2964} = 0.275\lambda$$

Point 'A' on the chart is the normalised load impedance at 0.29λ Move towards the generator by a distance $d = 0.23\lambda$ and point 'B' is the end of the transmission line length.

$$0.23\lambda + 0.29l\lambda = 0.52l\lambda \equiv 0.02l\lambda$$

The normalised impdance at 'B' is $(0.3 + j0.12)$. Now we are at the input of the matching transformer.



Denormalise the impedances $75 \times (0.3 + j0.12) = (22.5 + j9)\Omega$

The transformer is no longer presented with a real impedance.

For the transformer section $Z_0 = 41\Omega$ it is different from the transmission line Z_0 which is 75Ω. It is necessary to normalise the impedance of Port B to the Z_0 of the transformer section. This leads to a normalised impedance of $(0.55 + j0.22)$ which is plotted as Point C on the Chart.

Move from C by the transformer's length (0.275λ) towards the generator to point D on the chart. The normalised impedance at D is $(1.25 - j0.7)$ which is equal to $41.1 \times (1.25 - j0.7) = 51.4 - j28.8)\Omega$. This represents the input impedance of the whole circuit.

To get back to the 75Ω system, normalise Point D to 75Ω. This gives Point E on the chart which represents $Z_{in} = 0.69 - j0.38\Omega$ giving VSWR of 1.8. Hence the properly matched circuit at a design frequency of 920MHz gives VSWR of 1. At 1012MHz gives VSWR of 1.8.

9.2 Microwave Office Implementation

This chart is a continuation of chart 6. If you have not completed chart 6 you should do that first.

The only modifications required are in specifying the range of frequencies to simulate over. Change the project options dialogue box to match 9.1. Select 'Analyse' from the 'Simulate' menu. At this point your Microwave Office window should look like figure 9.3. Your VSWR graph should look like figure 9.2.

9.3 Figures

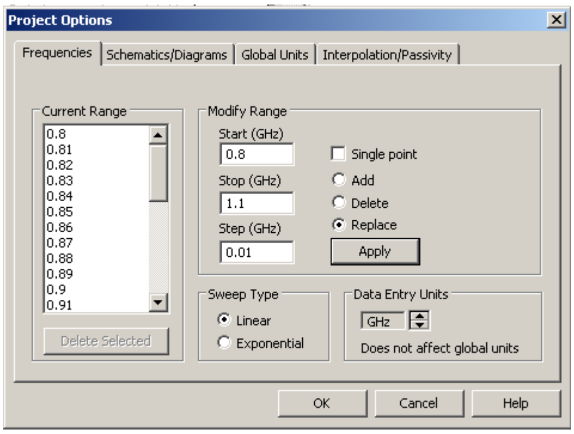


Figure 9.1: Chart 7 : Project Options

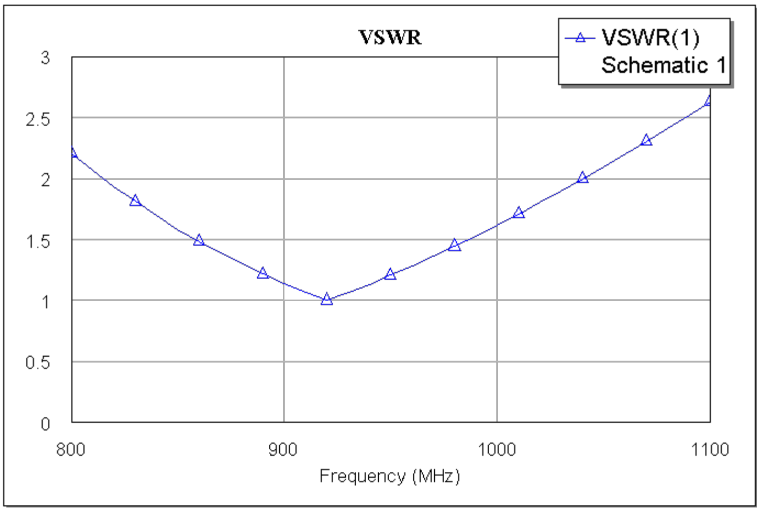


Figure 9.2: Chart 7 : VSWR

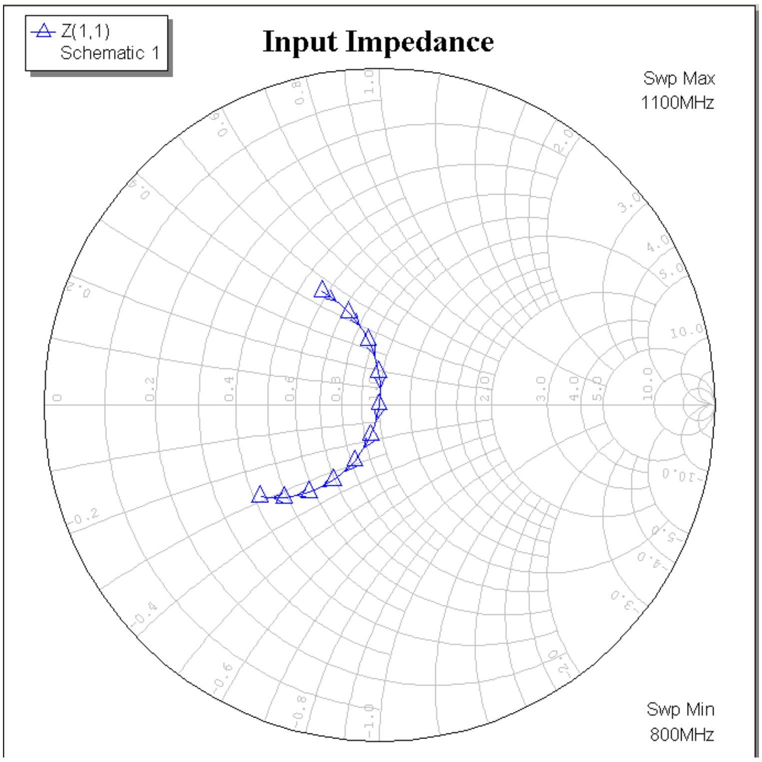


Figure 9.3: Chart 7 : Smith Chart

Chapter 10

Chart 8

10.1 Introduction

In this chapter the example in chart 8 is followed. The task is to design a single stub matching arrangement to match a load $Z_L = (100 + j100)\Omega$ to a transmission line with $Z_0 = 50\Omega$. The question, and the solution given in the lecture notes are given below.

Question
Design a single stub matching network to match a load $Z_L = (100 + j100)\Omega$ to a line of $Z_0 = 50\Omega$.
Solution

- Normalise Z_L to $Z_L = \frac{(100+j100)}{50} = (2 + j2)$.
- Plot the load $(2 + j2)$ (Point A).
- Convert the load impedance to an admittance. $\frac{1}{Z_L} = (0.25 + j0.25)S$ (Point B).
- d must be chosen such that the real part of the admittance is 1 $(1/50\Omega)$. This means we will move from point B towards the generator with a radius of OB to point C or point D on the chart:

  Point C: $Y = (1 - j1.56)$
  Point D: $Y = (1 + j1.56)$

- Hence we need a stub to cancel $(-j1.56)$ or $(+j1.56)$. The stub type can be open or short circuit. Its length should be as short as possible.
- The position of the stub from the load d is given by the length of the arc B→C or B→D both are measured towards the generator.

- **Case I:**

$$d = B \to C$$
$$B = 0.459\lambda$$
$$C = 0.322\lambda$$
$$d = 0.322\lambda + (0.5 - j0.459)\lambda = 0.363\lambda$$

- At point C $(1 - j1.56)$ and open circuit stub with a suitable length is needed to provide an admittance of $+j1.56$.
- From Eq^n 25: $y_{in} = j\tan(\beta d_1)$ i.e.

$$\text{When } d_1 \to 0\ ,\ y_{in}(\text{O/C}) \to 0$$
$$\text{When } d_1 \neq 0\ ,\ y_{in}(\text{O/C}) \to \text{positive}$$

- Point E on the chart represents a zero length open circuit stub.
- Point H corresponds to an open circuit stub with:

$$y_{in}(O/C) = +j1.56$$
$$d_1 = E \to H = 0.159\lambda$$

- **Case II:**

$$d = B \to D$$
$$B = 0.459\lambda$$
$$D = 0.179\lambda$$
$$d = 0.179\lambda + (0.5 - 0.459)\lambda = 0.220\lambda$$

- At Point D $(1 + j1.56)$ A short circuit stub with a suitable length is needed to provide an admittance of $-j1.56$.
- From Eq^n 24: $y_{in}(S/C) = -j\cot(\beta d_1)$ i.e.

$$\text{When } d_1 \to 0\ ,\ y_{in}(\text{S/C}) \to \infty.$$
$$\text{When } d_1 \neq 0\ ,\ y_{in}(\text{S/C}) \to \text{negative.}$$

- Point G on the chart represents a zero length short circuit stub.
- Point F corresponds to a short circuit stub with length $y_{in}(S/C) = -j1.56$
  $d_1 = G \to F = 0.091\lambda$

Hence the two possible designs are:

Line Length	Stub Length
$d = 0.363\lambda$	$d_1 = 0.159\lambda$
$d = 0.220\lambda$	$d_1 = 0.09\lambda$

10.2 Microwave Office Implementation

I assume that the methods of setting up a schematics, graphs and measurements are now sufficiently familiar that some instructions may be ommited for brevity. Rather than guide the reader, step by step, through the example given in the notes as was done in the prior chapters, it will be more beneficial to use the interactive nature of Microwave Office to examine how single stub matching works conceptually. Therefore, the point of each part of the circuit, and how the matching is achieved will be discussed through an example.

The first method is just for fun and will yield little information about matching neworks. But, it is nice to watch the simulator solve the problem for us in a mater of a second or two. The optimiser will solve the problem if given suitable goals:

- Create a new project, and save it as 'Chart 8'.
- Open the 'Project Options' from the Options menu.
- Open the 'Frequencies' tab arrange a single frequnency point at 6.3GHz. This frequency is the usual frequency chosen for the local oscillator in an X-band down-converter block.[1].
- Open the 'Global Units' tab, make sure the 'Frequencies' option is set to 'GHz'.
- Close the 'Project Options' dialogue box.

Add a schematic to the project and arrange the circuit shown in figure 10.1. All the elements should be familiar except the 'OPEN' element, which can be found in 'Circuit Elements' - 'Interconnects'.

It should be clear from looking at figure 10.1 that we will initially solve the problem using an open circuit stub. To solve with a short circuit use a ground element insted of 'OPEN'. Try both and select the solution which provides the shortest stub.

Complete the following:

- Add a Smith chart called 'Input Admittance'.
- Add a 'measurement' (to the chart) of the complex admittance looking into port one (Y(1,1))[2]
- Select 'Analyze' from the 'Simulate' menu. The result should be a marker at $(0 + j0)\Omega$.
- Add an 'Optimiser Goal' (right click 'Optimisation Goals' in the 'Project Browser') which forces the real part of the complex impedance looking into port one to 50Ω. See figure 10.3 and figur 10.2.

[1] All of the frequencies used in this tutorial are notable for some reason. It would be educational to try to find out what they all are.

[2] It is easier to work in admittance when analysing circuits which include parrallel (shunt) branches. If a series stub were being designed, it would be easier to work in impedance. If this is unclear consider two resistors both in parrallel with a battery, R_A and R_B. The total resistance $R_T = \frac{R_A \times R_B}{R_A + R_B}$. Converting the resistances to conductances $G_A = \frac{1}{R_A}$, $G_B = \frac{1}{R_B}$ and $G_T = \frac{1}{R_T}$ and then summing them $G_T = G_A + G_B$ will yield the reciprocal of the total resistance. This can be shown without difficulty, by inserting $\frac{1}{R_T}$, $\frac{1}{R_A}$ and $\frac{1}{R_B}$ into $R_T = \frac{R_A \times R_B}{R_A + R_B}$ and simplifying. Series stub circuits are not considered in EEE403.

- Add another ‘Optimiser Goal’ which forces the imaginary part of the complex impedance looking into port one to 0Ω. See figure 10.5 and figure 10.4. The two goals are defined seperately. Each has their own entry in the ‘Project Browser’. See figure 10.6.

Then:

- Open the Optimiser by pressing F7 or select ‘Optimiser’ from the ‘Simulate’ menu.
- Open the ‘Variables’ tab, select the electrical length (EL) of both lines under the ‘Optimize’ column. See 10.7
- Open the ‘Optimizer’, check the settings are reasonable (Random Local, 1000+ itterations) make sure you can see your Smith chart and click Start.

You should see the marker move to the middle of the chart. When you inspect the electrical length of the lines in the schematic they should agree with the example given for the open stub at the beginning at the chapter. Change the circuit to a shorted stub and run the optimiser again to produce the shorted stub example. While this is nice to watch, and doesn't take very long, it tells us little about the point of the two transmission lines and how they come to match the load to the line. This is what we will explore next.

10.3 How Single Stub Matching Works

The load must be made to look like 50Ω from the port's point of view. To achieve this move down the line to a point where the load impedance looks like it has a real part equal to 50Ω. That is, we use the properties of the transmission line to find a point where the real part of the impedance is matched to the line. In the Microwave Office simulation this line has the id ‘tline’. Then we must add a certain amount of capacitance or inductance at the point where the real part is matched, in order to make the imaginary part of the impedance equal to zero. It's as simple as that! The stub line adds the necessary capacitance or inductance based on its length and its termination. Open circuit stubs can be capacitive or inductive based on their length and the same is true of short circuit stubs. For example, all possible combinations of line lengths less than 180° to solve the problem are:

Solution	Stub Type	Line Length	Stub Length
1	Open	$d = 78.98°$	$d_1 = 122.5°$
2	Open	$d = 130.8°$	$d_1 = 57.5°$
3	Short	$d = 130.8°$	$d_1 = 147.5°$
4	Short	$d = 78.98°$	$d_1 = 32.4°$

It should not be suprising that solution number 1 has a stub length (d_1) which is 90° longer than solution 4. Equally Solution 3 and solution 2 share the same property. There is no advantage to using a longer stub than necessary so solutions 1 and 3 are not usualy used, but they do exist. d_1 may be made greater in steps of 90° provided the stub type is changed and may be made greater in steps of 180° if a longer stub of the same type is desired. There are some situations where having a very short stub may be difficult for practical reasons. Solutions 2 and 4 are the ones given in the EEE403 example.

We will now investigate the effect the two transmission lines have on the matching by using the tune tool. I assume that you are now familiar with the tune tool. It has been used in most of the prior chapters. If this is not the case you should become familiar with it by attempting some of the earlier charts.

- Right click on the Smith chart, select 'Properties', select the 'Grid' tab and *check* 'Admittance' and 'Values'. *Uncheck* 'Impedance'. Dismiss this dialogue box.
- On your schematic, delete the stub, leaving only the port, line, load and the ground which is connected to the load.
- Select the 'Tune Tool' (F10), then select the length of the transmission line.
- Open the 'Tune Dialogue Box' (F9). Set the maximum value to 180 and the minimum value to 0 and the step to 0.5.
- Move the tuning slider through the full range of values. Notice that there is a constant Γ for the marker (the radial distance from the center of the chart to the marker). Tune the marker position until it intersects the real unity circle. There are *two* solutions, make a note of both. See figure 10.8.
- Allow the marker to rest at on the real unity circle at about $(1 + j1.6)S$. Remember that this is an admittance chart, take care over the sign of the imaginary part; it is reversed compared to a standard impedance Smith chart. Positive imaginary numbers fall on the lower half of the chart which is inverted compared to the impedance Smith chart.

- Now add the stub line. Usa a COAX2 element and a ground to form a shorted stub. Don't forget to change the Fo of the new line to the correct value (6.3GHz).
- Select the 'Tune Tool' (F10), if it is not already selected. Unselect the electrical length of the main line, and select the electrical length of the stub line.
- Simulate your circuit.
- Tune the stub length between 0 and 180 degrees. The marker moves around the unity conductance circle with varing suseptance according to electrical length of the stub line. Zero suseptance (the matched condition) is given by an electrical length of approximately 32.5°. See figure 10.9

It is possible to choose the electrical length of the main line such that $Y = (1 - j1.6)S$ and then select a stub type and length that produce a matched condition also. It would be wise to try all four of the conditions given in table 10.3, and convince yourself that each stub type has an infinite number of solutions each 180° apart. Also convince yourself that both stub types have an infinite number of solutions 90° apart.

10.4 Figures

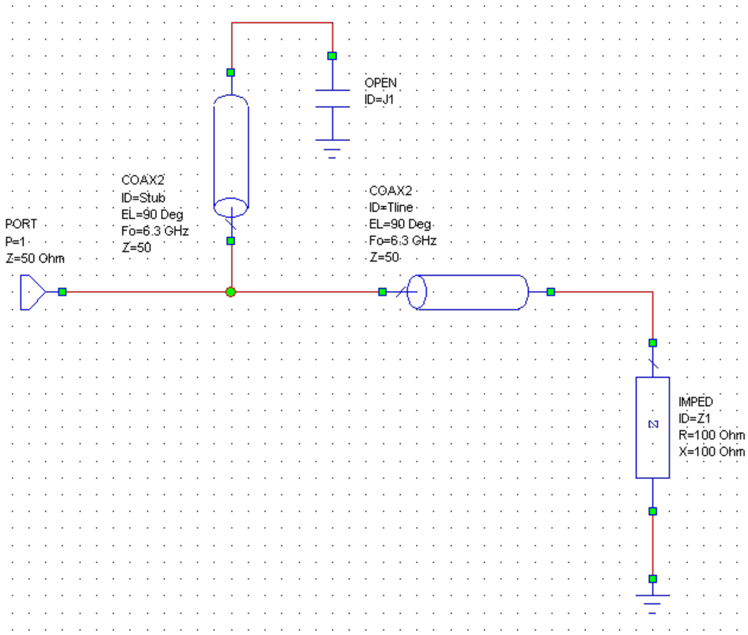


Figure 10.1: Chart 8 : Schematic

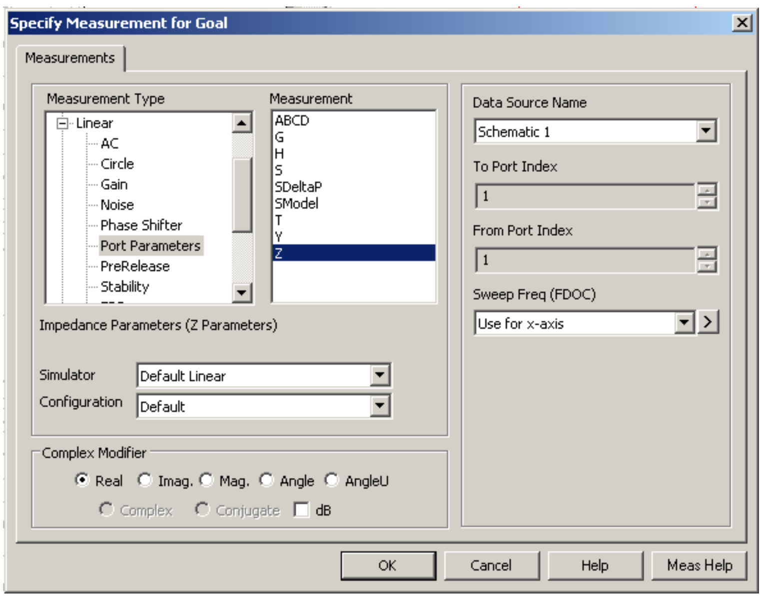


Figure 10.2: Chart 8 : Real Optimizer Goal Measurement

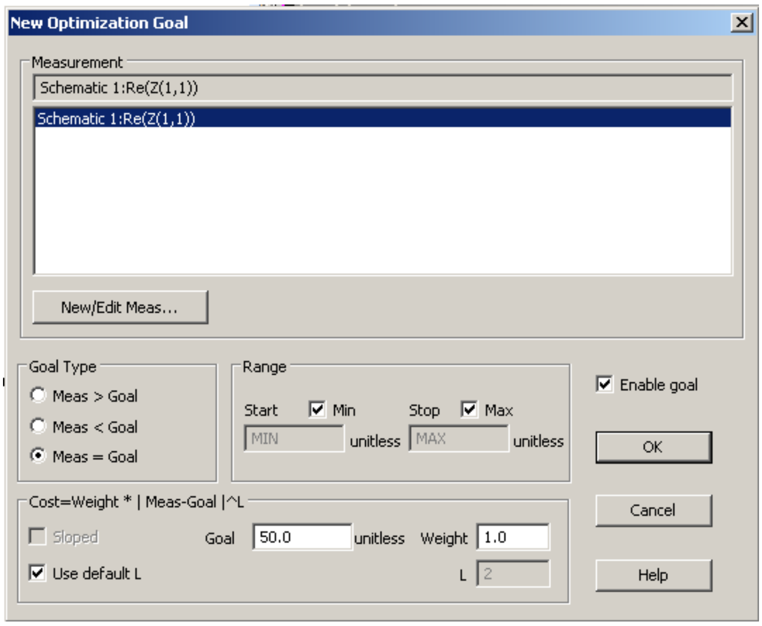


Figure 10.3: Chart 8 : Real Optimizer Goal

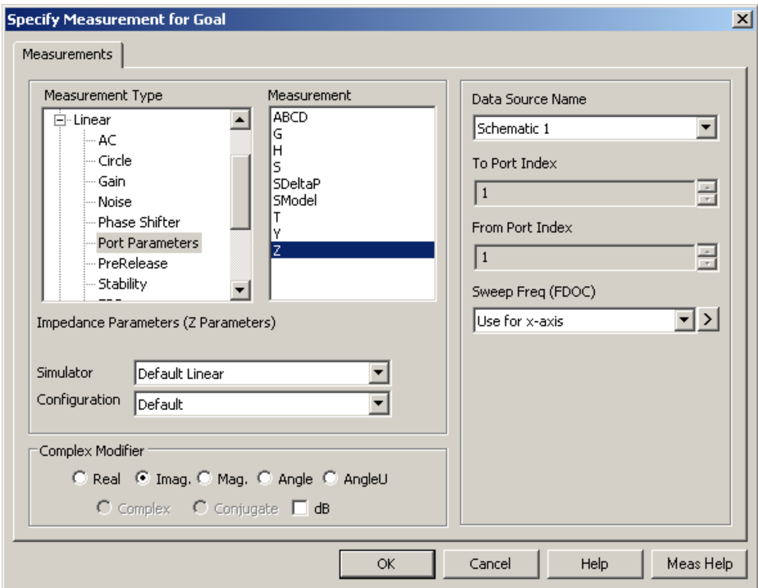


Figure 10.4: Chart 8 : Imaginary Optimizer Goal Measurement

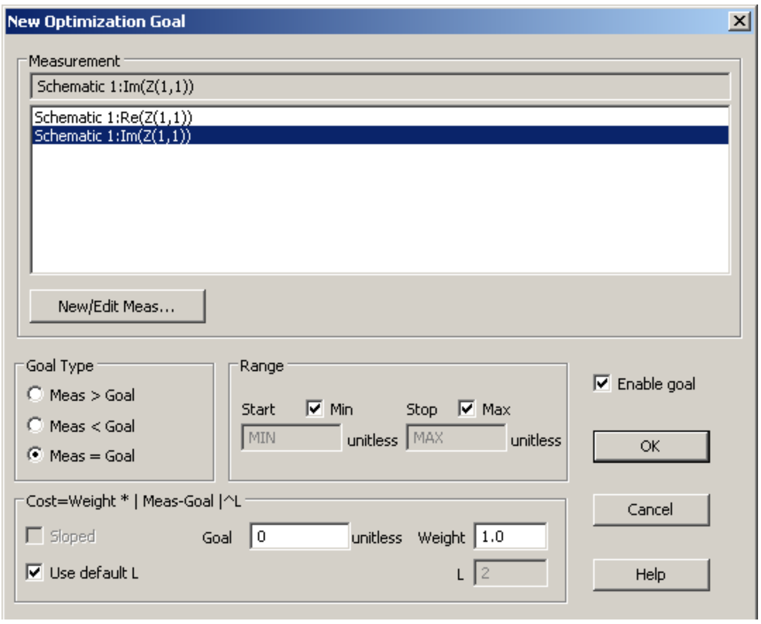


Figure 10.5: Chart 8 : Imaginary Optimizer Goal

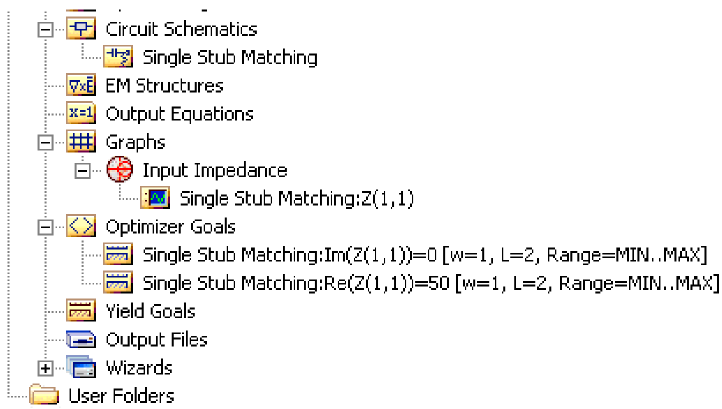


Figure 10.6: Chart 8 : Project Tree

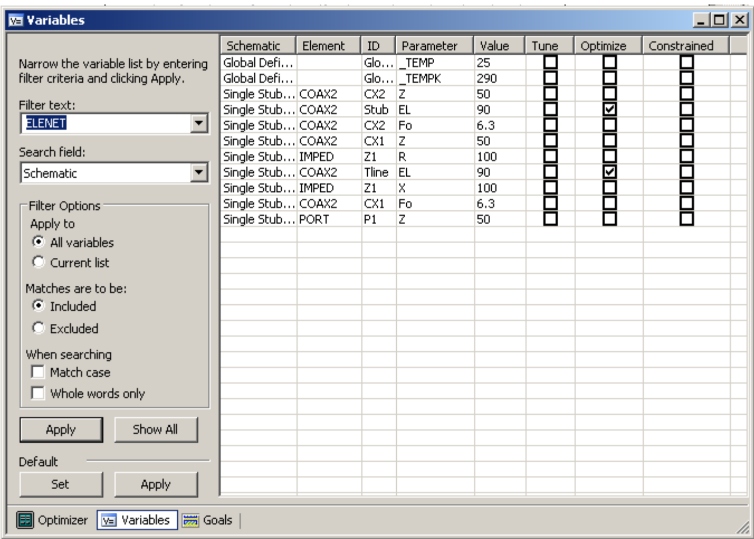


Figure 10.7: Chart 8 : Optimizer Variables

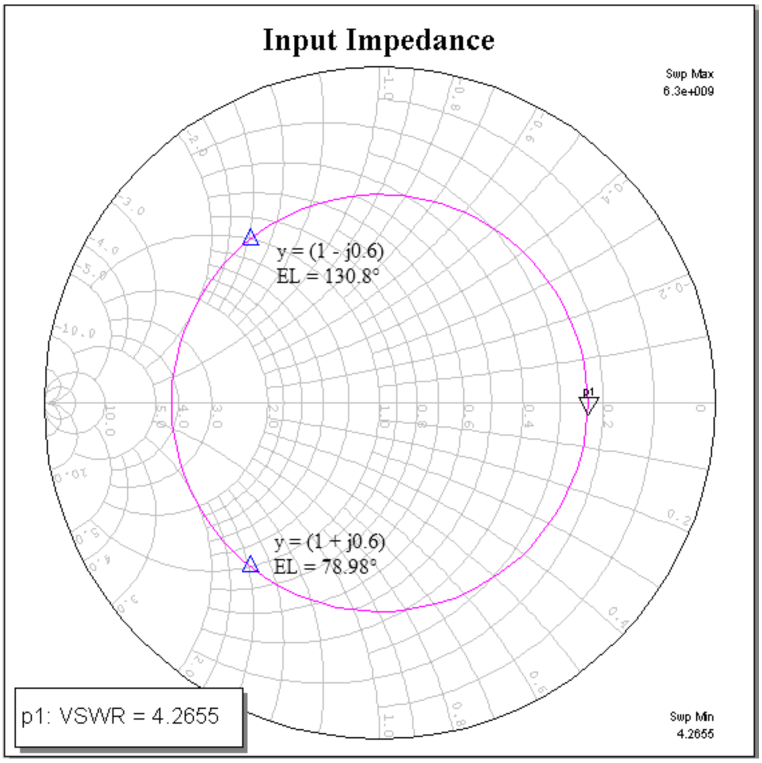


Figure 10.8: Chart 8 : Real Admitance Part

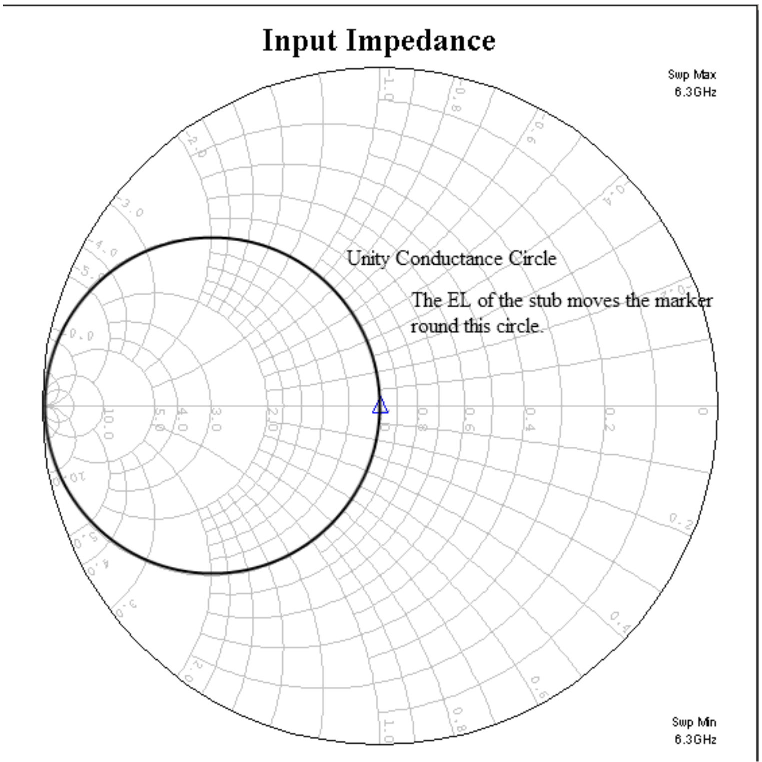


Figure 10.9: Chart 8 : Imaginary Admitance Part

Chapter 11

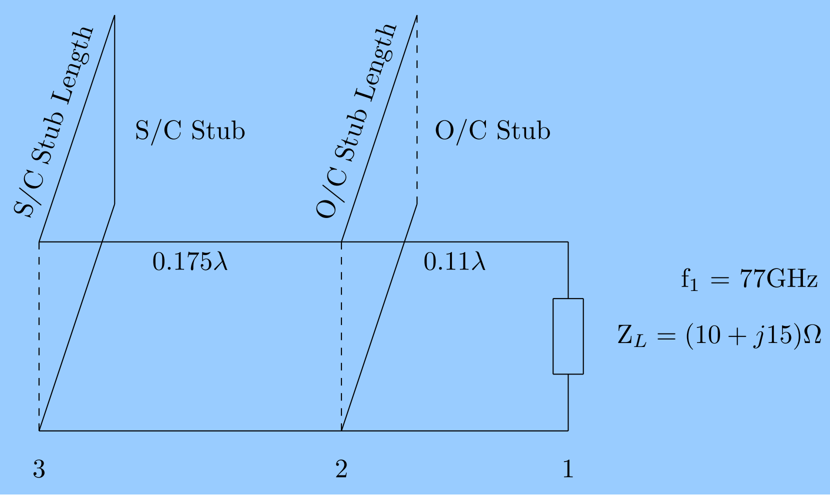
Chart 9

11.1 Introduction

This chart shows the design of a double stub matching circuit. Unfortunately the example given in the notes doesn't lead to a solution. This is purposly arranged so that the student will realise that not all impedances can be matched by a double stub system where one stub is at the load. Unfortunately this is not hepfull in terms of this Microwave Office tutorial. Therefore a different double stub matching example will be shown. The *method* of solution is similar for all double stub matching problems.

Question
Design a double stub matching circuit to match a load $Z_L = (10 + j15)\Omega$ to a line with $Z_0 = 50\Omega$. The first stub will be placed 0.11λ from the load. The second stub will be placed 0.175λ from the first. The matching frequency is 77GHz.

Solution
The solution is not substantially more complex than the single stub circuit of Chart 8. However the movement of the 'matching circle' or 'unity conductance circle' often causes considerable confusion. Refer to the Smith chart in the figures, and the diagram below:



To solve this problem on a manual Smith chart the following steps should be used:

- Plot the normalised load impedance $Z_L = (0.2 + j0.3)\Omega$. (Point A in figure 11.1, and Point 1 in digram.
- Refer the normalised load impedance through the center of the chart to yield the normalised load admitance $Y_L = (1.5 + j2.3)S$ (Point B).
- The first stub is 0.11λ from the load (in the direction of the generator). Move *towards the generator* by 0.11λ to arrive at point C. Note that moving along the main line can only result in a movement around a circle of constant radius. So B and C must be the same distance from the point $(1 + j0)$.

Having arrived at point C, a decision must be reached about the length of the O/C stub. In a single stub arrangement, the distance between the stub and the load is chosen such that when the stub is reached the real part of the conductance is unity. The stub length is chosen to negate any suseptance that exists at that point on the line. *A stub can add or subtract any suseptance but a stub can not change the conductance.* In shunt stub matching the conductance is changed by moving along the (main) transmisson line. Often the distance between the first stub and the load is fixed by the question or by the application.

In double stub matching the objective is to end up somewhere on the unity gain conductance circle when we are at point 3 on the diagram. Presently we are still at point 2. From our viewpoint, looking towards the generator from point 2 towards point 3, the unity conductance circle is 0.175λ away, in the direction of the generator.

We know that when we reach point 3 we must be on the unity conductance circle. We must move to some new suseptance value by adjusting the O/C stub such that, when we move down the line to point 3 we will find ourselves on the unit conductance circle.

So we are looking for a point that, when moved by a certain amount will result in us ending up on the unity conductance circle. Remember that traveling up or down the main line results in moving round a circle of constant radius.

If we move the unity conductance circle towards us, that is, *towards the load* (remember that our current position is at point 2 on the diagram) by 0.175λ, we can add a sufficient suseptance with the O/C stub to move onto to the rotated matching circle, safe in the knowledge that, when we move along the main line to point 3 (back *towards the generator*) the point we have chosen will be on the un-rotated unity conductance circle.

We can then add or subtract any suseptance needed with the S/C stub to move around the unity conductance circle in order to arrive at the matched condition. To implement all of this:

- Move the unity conductance circle from point 3 to point 2 on the diagram by moving it towards the load (round a circle of constant radius) by 0.175λ. The unity conductance circle is presently at 0.25λ and wavelengths towards the load are increasing so $0.25\lambda + 0.175\lambda = 0.425\lambda$. Draw a line from the center of the chart to 0.425λ on the *towards load* scale. Draw a circle with radius equal to the radius of the unity conductance circle such that the circumference of the circle passes through the middle of the chart, and the center of the circle is on the line between the center of the chart and 0.425λ on the towards load scale. Have a look at the Smith chart in figure 11.1 if this is not clear.

- Move from point C to the rotated unity conductance circle. (Point D)
- Moving from C to D has added $+j0.65$S suseptance at point 2 on the line.
- To find the length of stub required to add this suseptance mark $+j0.65$ on the outside of the chart, and draw a line through the middle of the chart to the other edge. Since we are using an O/C stub measure the distance from $(\infty + j\infty)$ clockwise round the chart to the line which has just been drawn. This yields the stub length, which is 0.09λ.
- We are still at point D. We must now travel down the main line in the direction of the generator by 0.175λ. We can be sure that this will result on us ending on the unity conductance circle. Point D is at 0.009λ on the *Towards Generator* scale moving towards the generator by 0.175λ leaves us at 0.184λ. (Point E)
- We have indeed arrived on the unity conductance circle at the point $(1 + j1.6)$S. We must now use the stub at point 3 on the diagram to negate the $+j1.6$S.
- Mark the edge of the chart at $-j1.6$ and draw a line through the center of the chart to the opposite edge. The towards generator scale reads 0.09λ. Since we are now finding the length of an S/C stub we measure its distance from $(0 + j0)$S, since this point is zero on the wavelengths towards generator scale, the stub length is 0.09λ.

11.3 Microwave Office Implementation

The implementation of this example is quite simple, especially when compared to the manual approach.

- Create a new project and save it with a suitable name.
- Create a schematic, and a Smith chart called 'input admittance'.
- Populate the schematic with a load ('IMPED', R = 10, X = 15).
- Also add a COAX2 element of length 0°.
- Add a port and a ground in the logical places.

Then:

- Add a measurement of Y(1,1) to the Smith chart.
- Convert the Smith chart from an impedance chart to an admittance chart (Right click the chart - Properties - Grid...)
- Analyse the schematic. Confirm that the admittance of the load is $Y_L = (1.5 - j2.3)$.
- Make the electrical length of the COAX2 element $0.11\lambda = 360 \times 0.11 = 39.6°$.
- Analyse the circuit again. Check that the admittance is now equal to that at point C in the manual chart.

This is where things diverge somewhat. It is not possible to construct a rotated unity conductance circle in Mircowave Office.

- Add a COAX2 element in series with the main line. make its electrical length $0.175\lambda = 63^{\circ}$.
- Add the open circuit stub. connect one end of a COAX2 element between the two transmission lines and connect an 'OPEN' element to the other end. See figure 11.2
- Use the tune tool to select the electrical length of the stub.
- Open the tune dialogue box and set the maximum and minimum and step to sensible values (You should be able to try all possible values of suseptance)
- Analyse the circuit.

Because there is no possibilty of a rotated unity conductance circle we can not easily look at point D and know that it is meaningful. However, it is possible to add suseptance with the O/C stub while viewing the effect from point 3 on the diagram. The desired result then is to arrive at the unity conductance circle, which *is* on the Microwave Office Smith chart.

- Move the tuner such that the electrical length of the O/C stub becomes 32°. The marker should appear on the unity conductance circle in the lower half of the chart, see figure 11.3.
- Add a S/C stub in parrallel with the input port. Use COAX2 + Ground, see figure 11.4.
- Add the S/C stub electrical length to the tuner, use suitable limits on this tuning variable also.
- Move the tuner over the full range you have chosen, observe that any suseptance is possible and the marker is always on the unity conductance circle.
- notice that and electrical length of 34° produces the matched condition.

11.2 Figures

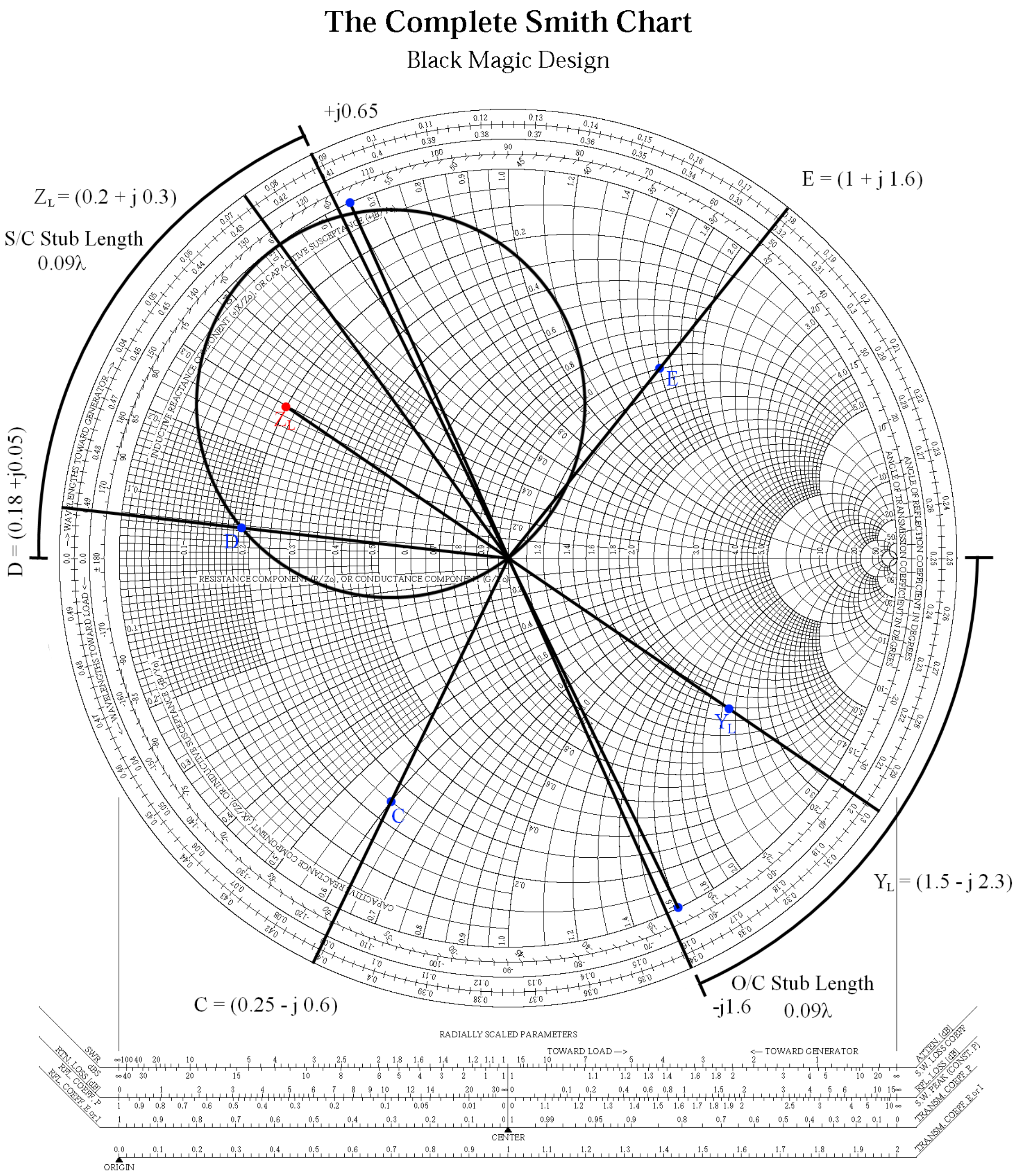


Figure 11.1: Chart 9 : Manual Chart Solution

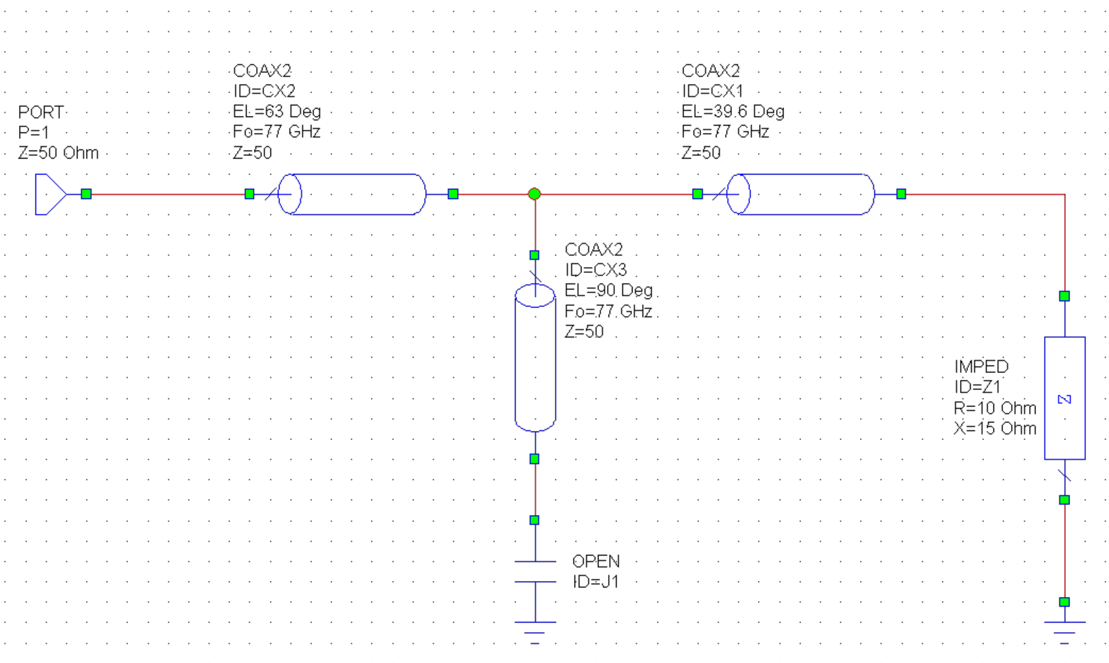


Figure 11.2: Chart 9 : O/C Stub Construction

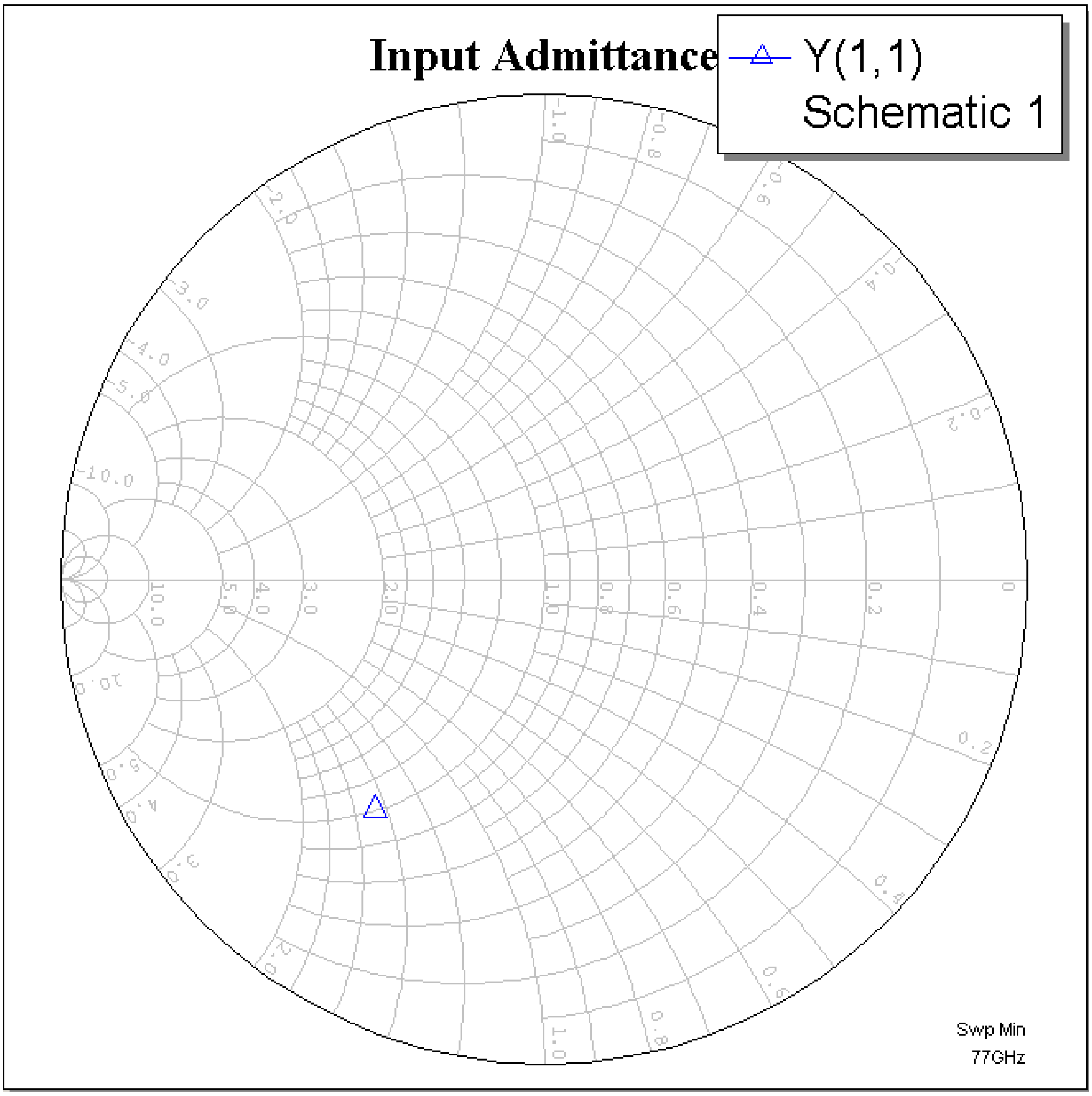


Figure 11.3: Chart 9 : O/C Stub Chart

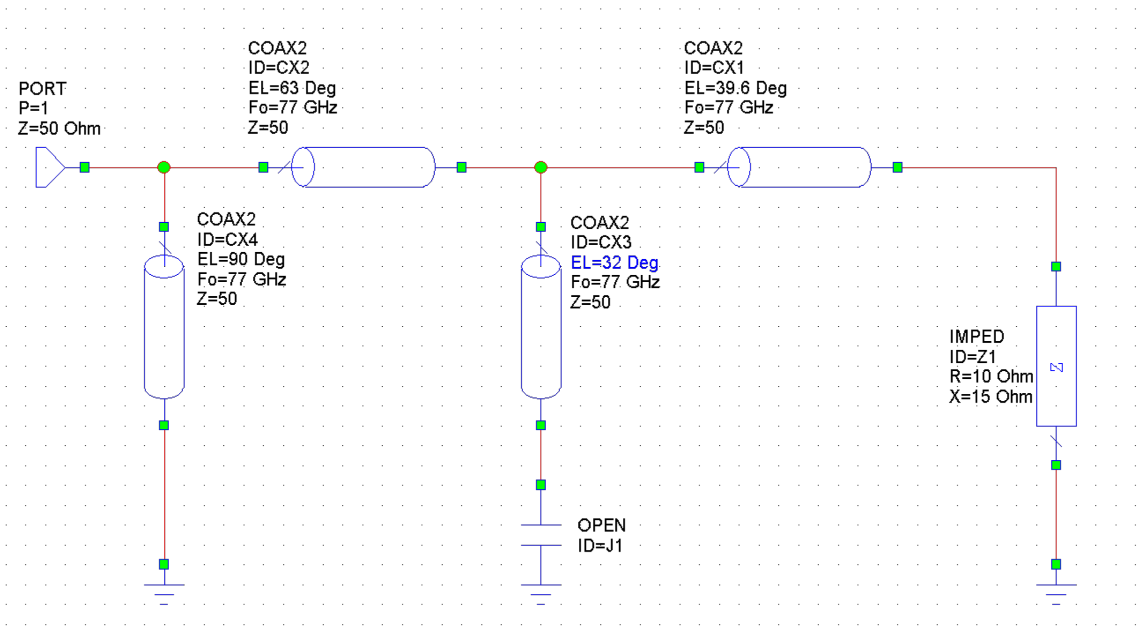


Figure 11.4: Chart 9 : Double Stub Circuit