MICROWAVE LABORATORY USING PUFF

by

Guanghong Wu
B.Eng., East China University of Science & Technology, 1993

PROJECT SUBMITTED IN PARTIAL FULFILLMENT OF
THE REQUIREMENTS FOR THE DEGREE OF

MASTER OF ENGINEERING

In the
School
of
Engineering Science

© Guanghong Wu 2006

SIMON FRASER UNIVERSITY

Fall 2006

All rights reserved. This work may not be reproduced in whole or in part, by photocopy or other means, without permission of the author.
APPROVAL

Name: Guanghong Wu
Degree: Master of Engineering
Title of Project: Microwave Laboratory Using PUFF

Supervisory Committee:

Chair: Dr. Faisal Beg
Assistant Professor of School of Engineering Science

Dr. Shawn Stapleton
Senior Supervisor
Professor of School of Engineering Science

Dr. Kyoung-Joon Cho
Supervisor
Research Associate of School of Engineering Science

Date Defended/Approved: Nov, 17, 2006
DECLARATION OF PARTIAL COPYRIGHT LICENCE

The author, whose copyright is declared on the title page of this work, has granted to Simon Fraser University the right to lend this thesis, project or extended essay to users of the Simon Fraser University Library, and to make partial or single copies only for such users or in response to a request from the library of any other university, or other educational institution, on its own behalf or for one of its users.

The author has further granted permission to Simon Fraser University to keep or make a digital copy for use in its circulating collection (currently available to the public at the “Institutional Repository” link of the SFU Library website <www.lib.sfu.ca> at: <http://ir.lib.sfu.ca/handle/1892/112>) and, without changing the content, to translate the thesis/project or extended essays, if technically possible, to any medium or format for the purpose of preservation of the digital work.

The author has further agreed that permission for multiple copying of this work for scholarly purposes may be granted by either the author or the Dean of Graduate Studies.

It is understood that copying or publication of this work for financial gain shall not be allowed without the author’s written permission.

Permission for public performance, or limited permission for private scholarly use, of any multimedia materials forming part of this work, may have been granted by the author. This information may be found on the separately catalogued multimedia material and in the signed Partial Copyright Licence.

The original Partial Copyright Licence attesting to these terms, and signed by this author, may be found in the original bound copy of this work, retained in the Simon Fraser University Archive.

Simon Fraser University Library
Burnaby, BC, Canada

Revised: Fall 2006
ABSTRACT

Lab experiments play a significant role in enhancing the understanding of material presented in course subjects. This project is to be used as the lab manual for ENSC 426, High Frequency Electronic Circuits. Four labs are presented in which students will use the theories learned in the course and the complimentary background information to design, analyze, fabricate, and test some basic radio frequency (RF) circuits. The first lab reviews concepts relating to transmission lines and scattering parameters, and it also works to introduce the operation of RF network analyzers. The second lab covers impedance matching networks and PUFF. PUFF is a computer-aided design (CAD) program used to design and analyze microwave and RF integrated circuits. Lab three reviews RF low pass filters and includes both insertion-loss and stepped-impedance design approaches. Lastly, lab four explains the basic characteristics of mixers and uses different RF equipment to perform measurements.

Keywords:

Microwave; Scattering parameters; Network analyzer; the Smith Chart; impedance matching; RF low pass filter; insertion-loss; stepped-impedance; passive Mixer.
ACKNOWLEDGEMENTS

I would like to take this opportunity to express my thanks to Dr. Shawn Stapleton for his supervision, inspiration, and advices during the process of this project.

In addition, I also want to deliver my appreciation to the supports and patience from my wife and our children. Needless to say, without their help, my education and this project would have never been completed.

Special thanks go to Dr. Kyoung-Joon Cho and Wan-Jong Kim for their supports throughout this project.
# TABLE OF CONTENTS

Approval ................................................................................................................................. ii
Abstract ................................................................................................................................... iii
Acknowledgements .............................................................................................................. iv
Table of Contents ................................................................................................................... v
List of Figures ........................................................................................................................ vii
List of Tables ........................................................................................................................... ix

## 1 LAB #1 S-PARAMETERS AND VECTOR NETWORK ANALYZER

1.1 Introduction .................................................................................................................. 1
1.2 Background .................................................................................................................. 1
1.2.1 Network Description ............................................................................................... 1
1.2.2 Transmission Line Concepts .................................................................................. 2
1.2.3 Scattering Parameters .......................................................................................... 10
1.2.4 Directional Couplers ............................................................................................ 12
1.2.5 VNA ....................................................................................................................... 14
1.3 Experiments ................................................................................................................ 16
1.3.1 Important Precautions .......................................................................................... 16
1.3.2 Lab Calibrations and Measurements ...................................................................... 17

## 2 LAB #2 IMPEDANCE MATCHING

2.1 Introduction ................................................................................................................ 20
2.2 Background ................................................................................................................ 20
2.2.1 Smith Chart .......................................................................................................... 20
2.2.2 Impedance Matching ............................................................................................ 25
2.3 Simulation .................................................................................................................... 32
2.3.1 Getting Started ...................................................................................................... 33
2.3.2 An Example Using PUFF .................................................................................... 40
2.4 Fabrication .................................................................................................................. 45
2.4.1 Preparing the Artwork .......................................................................................... 45
2.4.2 Exposing the Board ............................................................................................. 45
2.4.3 Developing the Exposed Board ......................................................................... 46
2.4.4 Etching the Developed Board .......................................................................... 46
2.5 Design of Maximum Gain for Small Signal (Conjugate Matching) ................................ 47
2.6 Experiments ............................................................................................................... 48

## 3 LAB #3 RF LOW PASS FILTER DESIGN

3.1 Objective ..................................................................................................................... 50
3.2 Background of RF Filters .......................................................................................... 50
3.2.1 Filter Basics .................................................................................................................. 50
3.2.2 Low Pass Filter Design ................................................................................................. 54
3.3 Simulation .......................................................................................................................... 64
3.4 Fabrication and Measured Results ..................................................................................... 66
3.5 Experiments ........................................................................................................................ 68

4 LAB #4 MIXER CHARACTERISTICS AND MEASUREMENTS ..................................... 69
4.1 Objective ............................................................................................................................ 69
4.2 Background Information .................................................................................................... 69
4.2.1 Conversion Loss ............................................................................................................. 71
4.2.2 VSWR ............................................................................................................................. 71
4.2.3 Isolation .......................................................................................................................... 71
4.2.4 Noise Figure ................................................................................................................... 72
4.2.5 1-dB Compression Port ................................................................................................ 72
4.2.6 Intercept Point ............................................................................................................... 73
4.2.7 Mixer Spurs .................................................................................................................... 74
4.3 Diode Mixers ....................................................................................................................... 74
4.3.1 Single-Ended Mixer ....................................................................................................... 74
4.3.2 Single-Balanced Diode Mixer ....................................................................................... 76
4.4 Mixer Characteristics Measurements ............................................................................... 77
4.4.1 Conversion Loss and Compression Point Measurements ............................................. 77
4.4.2 Isolation and Return Loss ............................................................................................ 79
4.4.3 IP3 Measurements ......................................................................................................... 81

Reference List .............................................................................................................................. 84
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1-1</td>
<td>A two-port network</td>
<td>1</td>
</tr>
<tr>
<td>Figure 1-2</td>
<td>Microstrip line</td>
<td>3</td>
</tr>
<tr>
<td>Figure 1-3</td>
<td>Transmission line equivalent circuit</td>
<td>4</td>
</tr>
<tr>
<td>Figure 1-4</td>
<td>Transmission line terminated by an arbitrary load</td>
<td>5</td>
</tr>
<tr>
<td>Figure 1-5</td>
<td>Impedance varies along a short circuit transmission line</td>
<td>7</td>
</tr>
<tr>
<td>Figure 1-6</td>
<td>Impedance varies along an open circuit transmission line</td>
<td>8</td>
</tr>
<tr>
<td>Figure 1-7</td>
<td>A two-port network with traveling waves</td>
<td>10</td>
</tr>
<tr>
<td>Figure 1-8</td>
<td>S-parameter notation</td>
<td>11</td>
</tr>
<tr>
<td>Figure 1-9</td>
<td>A 5-dB Pi-type attenuator</td>
<td>12</td>
</tr>
<tr>
<td>Figure 1-10</td>
<td>A directional coupler block diagram</td>
<td>13</td>
</tr>
<tr>
<td>Figure 1-11</td>
<td>A simplified network analyzer block diagram</td>
<td>14</td>
</tr>
<tr>
<td>Figure 2-1</td>
<td>Resistance mapping from Z plane to ( \Gamma ) plane</td>
<td>21</td>
</tr>
<tr>
<td>Figure 2-2</td>
<td>Reactance mapping from Z impedance plane to ( \Gamma ) plane</td>
<td>21</td>
</tr>
<tr>
<td>Figure 2-3</td>
<td>An impedance Smith Chart</td>
<td>22</td>
</tr>
<tr>
<td>Figure 2-4</td>
<td>Adding series inductors and capacitors in the Smith Chart</td>
<td>22</td>
</tr>
<tr>
<td>Figure 2-5</td>
<td>Admittance Smith Chart</td>
<td>23</td>
</tr>
<tr>
<td>Figure 2-6</td>
<td>Admittance manipulations on the admittance Smith Chart</td>
<td>24</td>
</tr>
<tr>
<td>Figure 2-7</td>
<td>Adding lumped components on the Smith Chart</td>
<td>24</td>
</tr>
<tr>
<td>Figure 2-8</td>
<td>An equivalent DC circuits</td>
<td>25</td>
</tr>
<tr>
<td>Figure 2-9</td>
<td>An equivalent circuit in AC</td>
<td>25</td>
</tr>
<tr>
<td>Figure 2-10</td>
<td>The L network matching</td>
<td>26</td>
</tr>
<tr>
<td>Figure 2-11</td>
<td>Equivalent circuits of the L section matching networks: (a) ( R_L ) is greater than ( Z_0 ) (b) ( R_L ) is smaller than ( Z_0 )</td>
<td>26</td>
</tr>
<tr>
<td>Figure 2-12</td>
<td>Example design</td>
<td>27</td>
</tr>
<tr>
<td>Figure 2-13</td>
<td>The Smith Chart used for simple L impedance match network</td>
<td>28</td>
</tr>
<tr>
<td>Figure 2-14</td>
<td>Impedance and ( \Gamma ) transformation on transmission line</td>
<td>29</td>
</tr>
<tr>
<td>Figure 2-15</td>
<td>Transmission line manipulations on the Smith Chart</td>
<td>30</td>
</tr>
<tr>
<td>Figure 2-16</td>
<td>Shunt microstrip line stubs manipulation on the admittance Smith Chart</td>
<td>30</td>
</tr>
<tr>
<td>Figure 2-17</td>
<td>Design example using a cascade microstrip line and a shunt stub</td>
<td>31</td>
</tr>
<tr>
<td>Figure 2-18</td>
<td>Design example using the cascade microstrip line and an open stub</td>
<td>31</td>
</tr>
<tr>
<td>Figure 2-19</td>
<td>A PUFF screen snap shot from a VGA display</td>
<td>33</td>
</tr>
<tr>
<td>Figure 2-20</td>
<td>A screen dump of the design example of part b sweep</td>
<td>41</td>
</tr>
<tr>
<td>Figure 2-21</td>
<td>A screen dump of part c sweep in step 6</td>
<td>42</td>
</tr>
<tr>
<td>Figure 2-22</td>
<td>A screen dump of the final matching circuit analysis</td>
<td>43</td>
</tr>
<tr>
<td>Figure 2-23</td>
<td>Screen dumps of the PUFF analysis on impedance matching example</td>
<td>44</td>
</tr>
<tr>
<td>Figure 2-24</td>
<td>Implementation of the matching design sample</td>
<td>46</td>
</tr>
<tr>
<td>Figure 2-25</td>
<td>Test results of the matching circuit on PCB</td>
<td>47</td>
</tr>
<tr>
<td>Figure 2-26</td>
<td>Block diagram of a single amplifier inserted between the source and load</td>
<td>47</td>
</tr>
<tr>
<td>Figure 2-27</td>
<td>Block diagram for the maximum gain amplifier design</td>
<td>48</td>
</tr>
<tr>
<td>Figure 3-1</td>
<td>Four basic type filters: (a) Low pass filter (b) Higher pass filter (c) Band pass filter (d) Band stop filter</td>
<td>50</td>
</tr>
<tr>
<td>Figure 3-2</td>
<td>Butterworth (Maximum flatness) filter</td>
<td>52</td>
</tr>
<tr>
<td>Figure 3-3</td>
<td>The Chebyshev (equal ripple) filter with 3 dB ripple</td>
<td>53</td>
</tr>
</tbody>
</table>
Figure 3-4  The Chebyshev (equal ripple) filter with 0.5 dB ripple ........................................53
Figure 3-5  Two different low pass filter prototypes: (a) Starting with a shunt component (b) Starting with a series component ..........................................................55
Figure 3-6  Butterworth filters attenuation versus filter orders ............................................56
Figure 3-7  Chebyshev filter with 0.5 dB ripple attenuation versus filter orders .................57
Figure 3-8  3.0 dB in-band ripple Chebyshev filter attenuation versus filter order ............58
Figure 3-9  Two Kuroda identities ...................................................................................60
Figure 3-10  Step 1 of design example ..............................................................................61
Figure 3-11  Step 2 of design example ..............................................................................61
Figure 3-12  Step 3 (a) of the design example ..................................................................61
Figure 3-13  Step 3 (b) of the design example ..................................................................61
Figure 3-14  Step 3 (c) of the design example ..................................................................62
Figure 3-15  Step 4 of the design example ......................................................................63
Figure 3-16  Final layout of the design example ..............................................................63
Figure 3-17  Approximate equivalent circuits for a short length transmission line (a) Short transmission line T-equivalent circuit (b) Large Zo (c) Small Zo .........................63
Figure 3-18  Prototype for stepped-impedance design .....................................................64
Figure 3-19  PUFF simulation for the insertion-loss design example .................................65
Figure 3-20  PUFF simulation results for the stepped-impedance design example ...........65
Figure 3-21  Implementation of insertion-loss filter ..........................................................66
Figure 3-22  Implementation of stepped-impedance filter ...............................................66
Figure 3-23  Low-pass filter measured results from the VNA (a) Insertion-loss method design (b) Stepped-impedance method design ........................................67
Figure 3-24  Comparison of two stepped-impedance filters with different layout ............68
Figure 4-1  A simplified receiver front-end block diagram ..............................................69
Figure 4-2  An ideal mixer .............................................................................................69
Figure 4-3  Frequency spectrum of an ideal mixer .........................................................70
Figure 4-4  Isolations of a mixer ....................................................................................71
Figure 4-5  Compression point of a mixer .......................................................................72
Figure 4-6  Intermodulation distortions spectrum ............................................................73
Figure 4-7  Mixer intercept point ....................................................................................73
Figure 4-8  Mixer IP3 calculation ....................................................................................74
Figure 4-9  A single-ended mixer ..................................................................................75
Figure 4-10  A double-ended mixer ................................................................................76
Figure 4-11  Hybrid coupler ..........................................................................................76
Figure 4-12  Conversion loss measurements setup .........................................................78
Figure 4-13  Isolation measurements setup ....................................................................80
Figure 4-14  IP3 measurement setup ...............................................................................82
Figure 4-15  IMD suppression ......................................................................................83
LIST OF TABLES

Table 1-1  Two power ratios used in VNA for measurements ........................................... 15
Table 1-2  VNA transmission calibration functions ............................................................ 16
Table 2-1  Part types in PUFF ............................................................................................ 37
Table 2-2  Parameters in Board window and their descriptions ........................................ 39
Table 3-1  Coefficients for Butterworth filters .................................................................... 56
Table 3-2  Coefficients for Chebyshev filters with 0.5dB ripple ......................................... 57
Table 3-3  Coefficients for Chebyshev filters with 3 dB ripple ........................................... 57
Table 3-4  Sections list of the stepped-impedance design example ................................... 64
Table 4-1  IP3 Spectrum Analyzer Setup .......................................................................... 82
1 LAB#1 S-PARAMETERS AND VECTOR NETWORK ANALYZER

1.1 Introduction

Lab #1 is designed to enhance the understanding of transmission line theory and the characteristics of traveling waves. In this lab, we will also be reviewing the concept and use of scattering parameters. The vector network analyzer (VNA), one of the most valuable RF measurement systems, will be used to implement the experiments.

1.2 Background

1.2.1 Network Description

In the low frequency regime, a network’s properties can be completely defined and described by using four sets of parameters referred to as the impedance (Z), admittance (Y), hybrid (H), and chain (ABCD) parameters. By properly shortening or opening the circuits, we can measure the voltage and current to determine these parameters.

\[
\begin{align*}
Z_{11} & = V_1 / I_1 + Z_{12} * I_2 \\
Z_{21} & = Z_{21} * I_1 + Z_{22} * I_2
\end{align*}
\]

\[
\begin{bmatrix}
V_1 \\
V_2
\end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\
I_2
\end{bmatrix}
\]

\[
\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}
\]

is called the impedance matrix or Z parameters of a two-port network. Similar equations are used to obtain the other parameters.

For the admittance matrix or Y parameters:

\[
I_1 = Y_{11} * V_1 + Y_{12} * V_2
\]

\[
\begin{bmatrix}
V_1 \\
V_2
\end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{bmatrix} I_1 \\
I_2
\end{bmatrix}
\]

\[
\begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}
\]

is called the admittance matrix or Y parameters of a two-port network.

\[
I_1 = Y_{11} * V_1 + Y_{12} * V_2
\]
For the hybrid matrix or H parameters:

\[
\begin{bmatrix}
I_1 \\
I_2
\end{bmatrix} = \begin{bmatrix}
Y_{11} & Y_{12} \\
Y_{21} & Y_{22}
\end{bmatrix} \times \begin{bmatrix}
V_1 \\
V_2
\end{bmatrix}
\]

For the ABCD parameters:

\[
\begin{bmatrix}
V_1 \\
I_1
\end{bmatrix} = \begin{bmatrix}
A & B \\
C & D
\end{bmatrix} \times \begin{bmatrix}
V_2 \\
-I_2
\end{bmatrix}
\]

In two-port networks, we can obtain the Z impedance parameters readily by performing the following:

\[
Z_{21} = \frac{V_2}{I_1} \bigg|_{I_2=0}, \quad \text{Input impedance, determined by enforcing an open-circuit at port 2}
\]

\[
Z_{21} = \frac{V_2}{I_1} \bigg|_{I_2=0}, \quad \text{Transimpedance, determined by enforcing an open-circuit at port 2}
\]

\[
Z_{12} = \frac{V_1}{I_2} \bigg|_{I_1=0}, \quad \text{Transimpedance, determined by enforcing an open-circuit at port 1}
\]

\[
Z_{22} = \frac{V_2}{I_2} \bigg|_{I_1=0}, \quad \text{Output impedance, determined by enforcing an open-circuit at port 1}
\]

The Y admittance parameters can be derived by properly shortening each of the ports in the same way.

However, when the frequency increases into the microwave range, accurately measuring the voltage and current becomes very difficult because of the requirements of short-circuit or open-circuit test conditions\(^1\). In addition, shortening or opening active circuits may cause unwanted oscillations, which can damage the components of the network permanently. Therefore, a new set of parameters must be used to describe the networks inserted into transmission lines for higher frequencies. In order to introduce these new parameters, we have to know the characteristics of the traveling wave instead of the voltage and current that flow into and out of the network.
1.2.2 Transmission Line Concepts

At low frequencies, the discrete component’s geometric size is much shorter than the length of the transmitted signal wavelength. This characteristic allows us to use Kirchhoff’s laws for voltage and current relationships. However, in microwave systems the length of the component or circuits is comparable to the signal wavelength. Because the propagation time is relatively significant, both the magnitude and the phase of the current and voltage are dependent on the position along propagation path. Kirchhoff’s laws do not directly apply in this situation; therefore, we have to treat the circuit conductors as transmission lines. In general, the transmission line concept is applied for analyses where the conductor’s physical length is equal to or greater than 1/10 of the signal wavelength.

A transmission line consists of at least two conductors to guide the electromagnetic wave in propagating from one point to another point. There are many types of transmission lines, and one of the most wildly used in microwave systems is the microstrip line.

For electronics, an ideal transmission line would be some type of conductor with no resistance or parasitic characteristics connecting two points. However, conductors are never ideal and have a variety of non-ideal characteristics including having small inductive effects, having a small amount of distributed capacitance and resistance. In the case of conductors used in transmission line, there are also leakage or conductance effects due to the non-ideal substrate materials. All these transmission line properties are determined by the conductors and materials used, along with the transmission line’s physical geometry.

A uniform transmission line can be subdivided into smaller sections, which are represented by the equivalent electric circuit in Figure 1-3, consisting four distributed parameters R, L, G and C in per unit length.

We can analyze and calculate the impedance of this equivalent circuit with Kirchhoff’s laws if we assume the length of this piece of short conductor is infinitely small. Under the steady state sinusoidal conditions, we obtain the simplified telegrapher equations:

\[ dV = -(Rdz + jwLdz)I \]
\[ dI = -(Gdz + jwCdz)V \]
Transmission line equivalent circuit

By solving these two equations simultaneously, we can obtain the propagation constant $\gamma$ and the characteristic impedance $Z_0$ of the transmission lines, as well as the waveform solutions:

$$\frac{d^2V}{dz^2} - \gamma^2 V = 0$$
$$\frac{d^2I}{dz^2} - \gamma^2 I = 0$$

where $\gamma = \alpha + \beta = \sqrt{(R + jwL)(G + jwC)}$

$\alpha$ is the attenuation constant.

$\beta$ is the phase constant in radians/unit length and equals to $\frac{2\pi}{\lambda}$.

$$V(z) = V^e^{-\beta z} + V^e^{\beta z}$$
$$I(z) = I^e^{-\beta z} + I^e^{\beta z}$$

$$Z_0 = \frac{V}{I} = \sqrt{\frac{R + jwL}{G + jwC}}$$

If $R \leq jwL$ and $G \leq jwC$, we can ignore the $R$ and $G$, and the transmission lines are ideally lossless, simplifying the equation for $Z_0$:

$$Z_0 = \frac{V}{I} = \sqrt{\frac{jwL}{jwC}} = \sqrt{\frac{L}{C}}$$

From the simplified equation, $Z_0$ is real and independent of frequency; furthermore, the voltage and current along the line are always in phase. In the microwave range, typically $Z_0$ is 50 Ohm, a trade-off between the transmission line's minimum attenuation and power handling capabilities.

When a transmission line is infinitely long, the signal could propagate forward forever along the line. However, in reality transmission lines have only a finite length in which to
transmit their signal to a destination. Nevertheless, if the destination has the same impedance as the characteristic impedance $Z_0$, all the traveling signals will be absorbed by this load without any echo along the transmission line. This scenario is equivalent to an ideal infinitely long transmission line.

When the transmission line has an open-circuit or short-circuit termination, the entire signal will be reflected back to the source because there is no other way for the signal to travel forward. As a result, two traveling waves exit along the same transmission line. One travels forward, and another travels in the opposite direction. The first one is called the incident wave, and the second one is called the reflected wave.

As long as the transmission lines are terminated by an arbitrary load, unequal to the characteristic impedance $Z_0$, part of the incident wave will be transmitted and absorbed by the load. The rest will be reflected back to the source due to the impedance mismatch. The degree of mismatch between the load and the characteristic impedance dictates how much of the signal is reflected.

![Transmission line terminated by an arbitrary load](image)

**Figure 1-4** Transmission line terminated by an arbitrary load

The incident wave and the reflected wave along the transmission lines can be expressed by the following equations respectively:

$$ V^+ = V_0^+ e^{-jeta z} $$

$$ V^- = V_0^- e^{jeta z} $$

Setting the termination port as the reference plane, where $z = 0$, we obtain the incident voltage $V_0^+$ and the reflected voltage $V_0^-$. The vector sum of the voltages at the reference plane is thus given by:

$$ V = V_0^+ + V_0^- $$

and the total current given by:

$$ I = I_0^+ - I_0^- $$

because the currents travel in opposite directions.
Applying Ohm's law and using the previous equations, the impedance at the load is then determined by the following:

\[ Z_L = \frac{V}{I} = \frac{V_0^+ + V_0^-}{I_0^+ - I_0^-} \]

while along the transmission line it is given by:

\[ Z_0 = \frac{V^+}{I^+} = \frac{V^-}{I^-} \]

A new parameter called the reflection coefficient is used to show the ratio of these two voltages at the reference plane \( z = 0 \):

\[ \Gamma_0 = \frac{V_0^-}{V_0^+} \]

\( \Gamma_0 \) is determined by the amount of signal wave that is reflected or the degree of the impedance mismatch at the termination port. Substituting it into the equation for the load impedance, we obtain the following:

\[ Z_L = \frac{1 + \Gamma_0}{1 - \Gamma_0} \cdot Z_0 \]

\[ \Gamma_0 = \frac{Z_L - Z_0}{Z_L + Z_0} \]

If the load is not a purely resistance load, \( \Gamma_0 \) will be a complex number.

Besides the reflection coefficient \( \Gamma_0 \) at \( z = 0 \), we would also like to be able to determine the reflection coefficient \( \Gamma_\ell \) along the transmission line.

If \( z = -\ell \), the voltage for the incident wave is expressed as \( V_0^+ e^{j\beta\ell} \), and the reflected wave has the form of \( V_0^- e^{-j\beta\ell} \). Thus,

\[ \Gamma_\ell = \frac{V_\ell^-}{V_\ell^+} = \frac{V_0^- e^{-j\beta\ell}}{V_0^+ e^{j\beta\ell}} = \Gamma_0 e^{-j2\beta\ell} \]

and the input impedance at \( \ell \) is given by:

\[ Z_\ell = \frac{V_\ell^+}{I_\ell^+} = \frac{V_\ell}{I_\ell} = \frac{V_0^+ e^{j\beta\ell} + V_0^- e^{-j\beta\ell}}{I_0^+ e^{j\beta\ell} - I_0^- e^{-j\beta\ell}} ; \]

\[ = \frac{V_0^+ [e^{j\beta\ell} + \Gamma_0 e^{-j\beta\ell}]}{V_0^+ [e^{j\beta\ell} - \Gamma_0 e^{-j\beta\ell}]} ; \text{ because } \Gamma_0 = \frac{V_0^-}{V_0^+} \]
\[ Z_0 \frac{(Z_L + Z_0)e^{j\beta \ell} + (Z_L - Z_0)e^{-j\beta \ell}}{(Z_L + Z_0)e^{j\beta \ell} - (Z_L - Z_0)e^{-j\beta \ell}}; \text{ because } \Gamma_0 = \frac{Z_L - Z_0}{Z_L + Z_0} \]

\[ = Z_0 \frac{Z_L \cos \beta \ell + jZ_0 \sin \beta \ell}{Z_0 \cos \beta \ell + jZ_L \sin \beta \ell}; \text{ because } e^{j\beta \ell} = \cos \beta \ell + j\sin \beta \ell \]

\[ = Z_0 \frac{Z_L + jZ_0 \tan \beta \ell}{Z_0 + jZ_L \tan \beta \ell}; \tan \beta \ell = \frac{\sin \beta \ell}{\cos \beta \ell} \]

The last equation is one of the most important equations in transmission line theory. It shows how the load impedance is transformed along the line as a function of the length \( \ell \) and the frequency.

When the transmission line is terminated with a short-circuit at the end, the above equation can be simplified to the following:

\[ Z_\ell = jZ_0 \tan(\beta \ell) \]

**Figure 1-5  Impedance varies along a short circuit transmission line**

When the transmission line has an open-circuit termination, the input impedance changes along the line and the equation is simplified to the following:

\[ Z_\ell = -jZ_0 \cot(\beta \ell) \]
The previous two equations and figures demonstrate periodic and purely reactive impedance changes. The transmission line in this case was a microstrip line fabricated on FR4 material with an effective wavelength 0.150 m at 1GHz. For both the open-circuit and short-circuit lines, the purely reactive impedance is transformed from short to open or open to short for every quarter wavelength. When the electrical length is less than $\ell = \pi/2$ from the reference plane, open-circuit transmission lines act like capacitors; while the behavior of short-circuit transmission lines are more like inductors.

From the mismatched impedance at the termination end of a transmission line, we know that there are two different waves traveling along the line in opposite directions simultaneously. Consequently, the vector sum of the incident and reflected waves forms a standing wave on the line. To further illustrate the standing wave, consider a transmission line terminated with a short-circuit at the end, $Z_L = 0$. Under this condition, the equation for the reflection coefficient $\Gamma_0$ simplifies to the following:

$$\Gamma_0 = \frac{Z_L - Z_0}{Z_L + Z_0} = -1$$

The vector sum of the voltages at the termination end is also zero due to the short-circuit termination. Thus, the incident and reflected waves are out of phase. However, when we move to $\frac{\lambda}{4}$ away from the $Z_L$, which equals $\ell = \pi/2$ of the electrical length, the incident wave phase is
earlier than the reference plane and the reflected wave phase is $\frac{\pi}{2}$ later than the reference plane. The difference in phases results in the two waves adding constructive such that:

$$V_\ell = V_\ell^+ + V_\ell^- = 2V_0^+$$

Moving another $\frac{\lambda}{4}$ further, the waves will be out of phase again and voltage sum would be zero. As a result, the sum of the incident and reflected waves along the line produces a new standing wave pattern. The envelope of the new wave pattern is stationary, in other words not moving on the line.

Using the same method, an open-circuit transmission line can be shown to have its own standing wave, but with a peak voltage that’s double the incident voltage at the open plane. The voltage sum becomes zero when we move $\frac{\lambda}{4}$ away from the open point. Compared to the short-circuit line standing wave the open-circuit line standing wave has shifted in phase by $\frac{\pi}{2}$.

When the transmission line is terminated by a load $Z_L$ such that:

$$-1 \leq \Gamma_0 = \frac{Z_L - Z_0}{Z_L + Z_0} \leq 1$$

the peak voltage of the standing wave can be any value between 0 and $2V_0^+$, and the following values are defined:

$$E_{\text{max}} = |V_0^+| + |V_0^-| \text{ when they are in phase}$$
$$E_{\text{min}} = |V_0^+| - |V_0^-| \text{ when they are out phase.}$$

The ratio of $E_{\text{max}}$ to $E_{\text{min}}$ represents another very important relationship called the voltage standing wave ratio, VSWR:

$$\text{VSWR} = \frac{E_{\text{max}}}{E_{\text{min}}} = \frac{|V_0^+| + |V_0^-|}{|V_0^+| - |V_0^-|} = \frac{1 + |\Gamma_0|}{1 - |\Gamma_0|}$$

The primary factor for the VSWR is $|V_0^-|$ or the degree of impedance mismatch since $V_0^+$ is constant along the transmission lines. The standing wave ratio (SWR) can also be expressed as a current ratio.

There is another parameter called return loss (RL) which is based on the reflection coefficient and provides another measure for the impedance mismatch at the termination of a transmission line:

$$\text{RL} = -20\log(|\Gamma|)$$
1.2.3 Scattering Parameters

Having reviewed transmission line theory, we now use the incident and reflected (scattered) waves flowing into and out of a two-port network to introduce the scattering or S-parameters. The results also apply to an n-port network.

First, we define the normalized incident and reflected power waves $a_i$ and $b_i$ at port $i$ ($i = 1, 2$):

$$a_i = \frac{V_i^+}{\sqrt{Z_{0i}}} \text{ and } b_i = \frac{V_i^-}{\sqrt{Z_{0i}}}$$

Where $Z_{0i}$ is the characteristic impedance of the transmission line at port $i$.

The magnitude of $a_i$ is the square root of the incident wave power. Similarly, the magnitude of $b_i$ is the square root of the reflected wave power. The power at port $i$ can then be expressed as $P_i = |a_i|^2 + |b_i|^2 = a \times a^* + b \times b^*$.

![Two-port Network](image)

**Figure 1-7 A two-port network with traveling waves**

If the network is linear, we can relate the reflected waves $b_1$ and $b_2$ to the incident waves $a_1$ and $a_2$ by a matrix:

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \times \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$

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

The matrix $S = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix}$ is called the scattering matrix, and its elements $S_{ij}$ are called the scattering parameters.
Generally $S_{ij}$ is a complex number, and its magnitude is usually expressed in dB.

\[
S_{11} = \frac{b_1}{a_1} \quad \text{Forward reflection coefficient (input match) by terminating port 2;}
\]

\[
S_{12} = \frac{b_1}{a_2} \quad \text{Reverse transmission coefficient (isolation) by terminating port 1;}
\]

\[
S_{21} = \frac{b_2}{a_1} \quad \text{Forward transmission coefficient (gain or loss) by terminating port 2;}
\]

\[
S_{22} = \frac{b_2}{a_2} \quad \text{Reverse reflection coefficient (output match) by terminating port 1;}
\]

Clearly, approximation terminations are required when we measure the S-parameters. For example, the transmission line connected to port 1 should be terminated into its characteristic impedance, not the port-1 impedance, in order to determine $S_{22}$.

For a lossless two-port network, the power flowing into and out of the network would be the same and the scattering matrix $S$ would be unitary:

\[
|S_{11}|^2 + |S_{21}|^2 = 1.0
\]

\[
|S_{12}|^2 + |S_{22}|^2 = 1.0
\]

If the network was reciprocal, $S_{ij}$ would be equal to $S_{ji}$, and the scattering matrix would be symmetric.

Pi or T type attenuators consisting of three resistors are widely used in RF circuit design to either attenuate the power, improve the input or output VSWR, or to mitigate the mismatch between two stages. We use the 5 dB attenuator, a simple two-port network, inserted into a 50 Ω transmission line to illustrate how to computer the S-parameters by their definitions.
To determine $S_{11}$, we have to terminate the second port by a 50 Ohm resistor, $Z_L$.

Because $\Gamma_L = \frac{Z_L - Z_0}{Z_L + Z_0}$, we have $\Gamma_L = 0$ and the following:

$$S_{11} = \left| \frac{b_1}{a_1} \right| = \Gamma_{in} = \frac{Z_{in} - Z_0}{Z_{in} + Z_0}.$$

$$Z_{in} = ((Z_0 || R_2) + R_1) = \left( \frac{180 \times 50}{180 + 50} \right) + 30 \right| \frac{69.13 \times 180}{69.13 + 180} = 49.95$$

$$S_{11} = \Gamma_{in} = \frac{Z_{in} - Z_0}{Z_{in} + Z_0} = \frac{49.95 - 50}{49.95 + 50} = 0.000523$$

$S_{11}$ can also be expressed in dB, $20 \log(\Gamma_{in}) = -65.63$dB

Similarly for determining $S_{21}$:

$$S_{21} = \frac{V_1}{V_2}$$

and

$$V_L = \frac{V_S}{2} \times \frac{(R_2 || R_1)}{(R_2 || R_1) + R_2} = \frac{V_S}{2} \times 0.5660$$

$$S_{21} = \frac{V_1}{V_2} = 0.5660, \text{ or } 20 \log(0.5660) = -4.94$dB

Since the circuit is symmetric, $S_{11}$ is equal to $S_{22}$ and $S_{12} = S_{21}$.

### 1.2.4 Directional Couplers

In the transmission line, we know there are two traveling waves when the line is not terminated by its characteristic impedance. To determine the S-parameters, we would like to separate the reflected wave from the incident wave in order to measure them. A feasible way of measuring the two waves is to use directional couplers.
The directional couple is a passive 4-port network. It divides an input signal between two other three ports. There are several different ways of constructing directional couplers. One of the most common and simple forms is the two parallel microstrip lines architecture which are easy to design and fabricate. One of the lines is used for wave propagation while another one placed close by is used to couple the traveling wave. The coupled amount depends on both the space between the two lines and the length of the coupling lines. Usually odd numbers of quarter-wavelength are used.

![Figure 1-10 A directional coupler block diagram](image)

In Figure 1-10, an incident traveling wave flows in at the input port and passes to port 2 as the output. Port 3 is the coupled port while port 4 is the isolated port and is usually internally terminated. If port 4 is not terminated then the directional coupler is called a bi-directional coupler.

If an incident signal exists at port 1, we can have two output signals at port 2 and the coupling port 3. However, if we apply an incident signal wave on port 2, ideally we would only have the signal at port 1. The coupled signal at port 4 would be absorbed by the termination load.

Four parameters, the coupling coefficient, mainline insertion loss, couple directivity, and coupler isolation are defined based on the various power ratios between each of the ports:

- **Coupling coefficient**
  \[
  \text{Coupling coefficient} = 10 \times \log\left(\frac{P_3}{P_1}\right)
  \]

- **Mainline insertion loss**
  \[
  \text{Mainline insertion loss} = 10 \times \log\left(\frac{P_2}{P_1}\right)
  \]

- **Directivity**
  \[
  \text{Directivity} = 10 \times \log\left(\frac{P_3}{P_2}\right)
  \]

- **Isolation**
  \[
  \text{Isolation} = 10 \times \log\left(\frac{P_3}{P_1}\right) = 10 \times \log\left(\frac{P_2}{P_1}\right) = \text{Coupling coefficient} + \text{Directivity}
  \]
1.2.5 VNA

1.2.5.1 Introduction of VNA

Because S-parameters are significantly useful to describe network characters, network analyzers typically have built-in directional couplers used to measure and determine the S-parameters by properly terminating the network ports.

A Vector Network Analyzer (VNA) is a fully integrated RF test system, which is used to determine the reflection coefficients and transmission coefficients by measuring both the magnitude and phase of the incident and reflected waves. Directional couplers make it possible for the VNA to separately sample the incident and reflected power. Once known, the incident and reflected power allows the voltage, current, impedance and other parameters to be deduced from the measured power and the known system characteristic impedance. Typically, the system characteristic impedance is 50 Ω.

![A simplified network analyzer block diagram](image)

**Figure 1-11 A simplified network analyzer block diagram**

In Figure 1-11, Port 1 is the source of a calibrated incident signal. An amplitude controlled sinusoid signal is applied from port 1 to the device under test (DUT) and the reflected signal wave from the DUT is also measured. At port 1, two directional couplers are used to measure the incident signal and the reflected signal waves. The first directional coupler is employed to couple the known frequency, amplitude and phase incident signal, and it is set as the reference(R). The second one is used to couple and measure the reflected power (A) from the DUT. Base upon the separately measured reflected wave and incident wave, we can then determine the reflection coefficient.
Port 2 on the VNA is used to measure the transmitted signal from the DUT after an incident signal is applied at port 1. With the measured transmitted power (B) and the incident power (R), we are able to determine the transmission coefficient of the DUT. In more advanced VNAs, such as the HP 8710B, port 2 can also be a source for full two-port measurements.

The VNA primarily uses two ratios to implement the testing: the reflected wave/incident wave and the transmitted wave/incident wave:

<table>
<thead>
<tr>
<th>REFLECTED</th>
<th>INCIDENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>A</td>
</tr>
<tr>
<td></td>
<td>R</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TRANSMITTED</th>
<th>INCIDENT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>B</td>
</tr>
<tr>
<td></td>
<td>R</td>
</tr>
</tbody>
</table>

Table 1-1 Two power ratios used in VNA for measurements

<table>
<thead>
<tr>
<th>REFLECTED</th>
<th>INCIDENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>B</td>
</tr>
</tbody>
</table>

- Used for:
  - VSWR
  - $S_{11}/S_{22}$
  - Reflection Coefficient
  - Impedance
  - Return Loss

- Used for:
  - Insertion Loss
  - Gain
  - $S_{21}/S_{12}$
  - Group Delay
  - Transmission

After the VNA simulates an incident signal with sweeping frequency, both the reflected and transmitted signals from the DUT are tested. The internal processor then processes the detected signals and displays the results in various ways, such as relative ratio (dB), absolute power (dBm), as well as phase relationships.

1.2.5.2 Calibration

There are three types of errors during the network analyzer measurements. The first type is random errors because of the noise and the connector repeatability. The second type is drift errors such as frequency and temperature drift during two consecutive calibrations. Both random and drift errors are neither predictable nor repeatable. The third type is systematic errors caused by the mismatch between the port impedance and the system impedance, signal leakage from crosstalk, the use of non-ideal directional couplers in the signal paths, and frequency response errors. However, unlike the other two types of errors, systematic errors are repeatable and predictable.

The VNA performs a calibration to correct for systematic errors. One of the common calibration methods used for the VNA is to measure some precise calibration standards with known magnitude and phase response as a function of frequency. Typically, there are four calibration standards: open, short, load and through.
During a calibration, the VNA measures these standards in a sequence and compares the measured results with the ideal measurements to create error models which are then used to correct the actual measurements on the DUT. Since the standards are used directly, their quality is critical in order for the calibration to be accurate and effective.

More frequently, an external test fixture consisting of two cables, adaptors and connectors is used between the network analyzer and the DUT. As a result, the effects of this external test fixture on the measurements must be considered and accounted for by performing the calibration at the same points that the DUT is placed.

For the transmission measurement, there are several calibration options:

<table>
<thead>
<tr>
<th>Table 1-2</th>
<th>VNA transmission calibration functions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Calibration Options for Transmission</strong></td>
<td><strong>Error Correction</strong></td>
</tr>
<tr>
<td>Default Response</td>
<td>For quick but not accurate test</td>
</tr>
<tr>
<td>Response</td>
<td>Frequency response errors</td>
</tr>
<tr>
<td>Response &amp; isolations</td>
<td>Frequency response and crosstalk errors</td>
</tr>
<tr>
<td>Enhance response</td>
<td>Frequency response and mismatch errors</td>
</tr>
</tbody>
</table>

There are also default one-port and manual one-port calibrations for the reflection measurements.

The user-defined one-port reflection calibration measures three known standards to remove frequency response errors and directional coupler directivity error, as well as source mismatches. The Short-Open-Load-Through (SOLT) full two-port calibration is also commonly used in the VNA calibration.

1.3 Experiments

In this experiment, we are going to setup and calibrate a network analyzer to measure the S-parameters of RF components or devices.

1.3.1 Important Precautions

The network analyzer is deliberated equipment and is very expensive to repair. When used a foot or wrist strap MUST be worn to avoid damaging it by electrostatic discharges. All the calibration kits should be treated very gently and please do not rotate the body of the standard. If any standard is off, it will extremely affect the measurements but you probably never know it. Before connecting any DUT to the VNA, it is important to know whether it is a passive or active RF device and what the maximum applied input and output power will be for DUT. High input power can damage the DUT and if the output power from the DUT is higher than the
maximum power rating of the VNA, the VNA input port will be damaged permanently. If you are not sure on the details regarding the DUT being tested or how to use the equipment after reading the user manual carefully, please ask the lab instructor for help prior to starting any experiments.

1.3.2 Lab Calibrations and Measurements

1.3.2.1 Full Two-port Calibration of HP 8710B

Choose a calibration kit for the type of connectors you are using.

1. Set your frequency range. On the STIMULUS control, press start and type the frequency you want to start. Do the same for the stop frequency.

2. On the MENUS control, press CAL.

3. Delete a calibration set to make room for yours by pressing MORE on the on-screen soft menu, then DELETE CAL SET and choose a number.

4. Select CAL 1 (85052A) because you will be using cables with 3.5 mm connectors attached to the network analyzer.

5. Choose REFLECT'N soft key.

6. You will be prompted to put calibration standards on the end of your cable. They can be found in the wooden calibration kit.

7. Proceed to connect each of the calibration terminations for open, short, and matched load standards for S11 (port 1) and S22 (port 2) as described below. After each termination is placed on the cables, press the respective soft key, remove the termination, and repeat the procedure until all of the calibrations have been made for both ports:

   i. Start with the short standard (it is labeled). Connect the short, press the soft key SHORT and when the 8710B has done its quick measurement it will underline SHORT on-screen. Remove the short and replace the plastic cover.

   ii. Find the open standard. It is the same length as the short, but with a smaller radius and a hole in the end. Press OPEN and then 8048A. Wait until OPEN is underlined and then until DONE OPEN. Remove the open and replace the plastic cover.

   iii. Find the load standard. Connect it, and press LOAD and then press BROADBAND. Wait until OPEN is underlined and then until DONE LOADS. We are assuming our load is accurate over the whole frequency range of calibration and is therefore "broadband".
8. Press REFLECT'N DONE.
9. Choose the TRANSMISSION soft key. Connect cable 1 to cable 2 using the through standard (1/2" long connector with screw threads on both sides). Press each of the four soft keys and wait until each key is underlined. Press TRANS. DONE.
10. Press the ISOLATION soft key. Select the OMIT ISOLATION soft key. Press ISOLATION DONE.
11. The calibration is now complete and must be saved. Select SAVE two-port CAL. Save the calibration to the location (1-8) deleted previously. The exact location used doesn't matter and when performing various tests over different frequency ranges several calibrations can be used at a time. The network analyzer should now say "Correction ON."
12. Test your calibration. Put the load standard on again. On the FORMAT menu, press SMITH CHART. You should see a dot in the center of the smith chart. If this is not "clean", redo the calibration. Factors that can affect your calibration: tightness of cables and connectors specifically between elements and the network analyzer.

1.3.2.2 Impedance Measurements
1. On the FORMAT menu, press Smith Chart.
2. On the MENU menu, press MARKER>.
3. Use the thumb dial to move the marker to the frequency you are interested in (printed on the top left side of the screen). You will also have two numbers followed which are the impedance (real and imaginary) in ohms.

1.3.2.3 Reflection Measurements
1. For determining S11 or S22, on the FORMAT menu, press LOG MAG.
2. Press MENU (on the FORMAT menu), and this will give you many options for what to print. Originally, it is showing LOG MAG and the value is shown in the upper left corner for the marked frequency. Other options for measurements are LIN MAG, SWR, REAL, IMAG, POLAR, etc. For the measurements, you can change the scale of a graph by going to the RESPONSE menu, pressing SCALE, and turning the thumb knob until the graph looks the way you want it. Alternatively, the scale can be entered using the number pad and then pressing x1.

1.3.2.4 Saving Data
On the PC screen, there is already one program running for saving s-parameters:
1. Go to Calibrate in menu and then click two-port s-parameters if you already performed two-port calibration. The program will show the frequencies of your calibration. In order to make sure that the program has frequencies that you calibrated on, click on [Re-read VNA frequencies].

2. Click on [VNA cal is ready]. Go to Menu→Measure, S-parameters, and click on two-port.

3. Click on [Measure]. This step may take several seconds and then the [Enter label for s-parameter data] window pops up.

4. Press [Don't change] and the program will then show you [Save as] to save S-parameters and Smith Chart results. You can save your parameters to the Desktop and email them to yourself.

5. [Esc] to go back to the first screen. Repeat Step 3-5 for other measurements.

Saved files (s2p extension) have all 4 s-parameters for two-port and in magnitude-phase formats, so in order to plot in dB you’ll need to convert the magnitudes.

This program is used only to save s-parameter data. In the case you need to record other data from the analyzer, you’ll need to record the data separately through other means (text file, piece of paper, etc.).
2 LAB #2 IMPEDANCE MATCHING

2.1 Introduction

Lab#2 is intended for a review of the impedance matching in microwave circuits. We will go through the Smith chart first and then examine how to achieve a conjugate match by manipulating both discrete components and microstrip line stubs matching in the Smith Chart. The computer-aided-design tool PUFF will also be introduced to simulate the matching network performance. After the simulation analysis, fabrication of the circuit will also be performed.

2.2 Background

2.2.1 Smith Chart

Before reviewing the impedance matching, we will introduce a very popular and tremendously useful graphical tool in RF circuit design and analysis, the Smith Chart. Named after Phillip Smith, an engineer at Bell Laboratories who conceived and developed the Chart in 1930s, the chart allows RF engineers to avoid using many tedious RF theory equations repetitively.

In Lab#1, we had reviewed the transmission line theory and essential concepts with some basic relationships between the incident and reflection waves along the line. When a transmission line with characteristic impedance $Z_0$ is terminated by an arbitrary load $Z_L$, the reflection coefficient at the termination plane is expressed as,

$$\Gamma_L = \frac{Z_L - Z_0}{Z_L + Z_0} = \frac{Z_L}{Z_0} - \frac{Z_0}{Z_L} = \frac{Z_L - 1}{Z_L + 1}, \quad z_L = r + jx$$

$$\Gamma_L = |\Gamma| \angle \theta = \Gamma_r + \Gamma_j$$

where $z_L$ is the normalized complex load impedance for the transmission line with characteristic impedance $Z_0$. In the reflection coefficient equation, there is a bilinear transformation between $\Gamma$ and it’s corresponding $z_L$. The reflection coefficient $\Gamma$ varies in the range from -1 to 1.

The Smith Chart is a polar chart mapping the impedance to the reflection coefficient plane on a one-to-one basis via bilinear transformations.
In Figure 2-1, constant resistance (normalized on impedance plane) lines are mapped to constant resistance circles in the $\Gamma$ plane. These circles can be expressed by their parametric equation:

$\left( \Gamma_r - \frac{r}{r+1} \right)^2 + \Gamma_j^2 = \left( \frac{1}{r+1} \right)^2$.

The centers of the circles are at $\left( \frac{r}{r+1}, 0 \right)$, whose radius are given by $\frac{1}{r+1}$. The $r$ in these equations is the normalized impedance. Similarly, as show in Figure 2-2, the constant reactance circles also have their parametric equation

$\left( \Gamma_r - 1 \right)^2 + \left( \Gamma_j - \frac{1}{x} \right)^2 = \left( \frac{1}{x} \right)^2$, in which $x$ is the normalized reactance.

Combining Figure 2-1 and Figure 2-2, we obtain the whole impedance Smith Chart.
At this point, we can now manipulate the impedances on the Smith Chart. For example, with an impedance $Z_L=100+j100$, which is normalized by 50 Ohms to $2+j2$, adding series passive components can be performed easily on the impedance or $Z$ Smith Chart.
The reflection coefficients can also be expressed in terms of admittances:

$$\Gamma_L = \frac{Z_L - Z_0}{Z_L + Z_0} = \frac{\frac{1}{Y} - \frac{1}{Y_0}}{\frac{1}{Y} + \frac{1}{Y_0}} = \frac{Y_0 - Y}{Y_0 + Y} = \frac{\frac{1}{Y_0} - \frac{1}{Y_0}}{1 + \frac{1}{Y_0}} = \frac{1 - y}{1 + y}$$

where $Y_0$ is the characteristic admittance, which equals to $\frac{1}{Z_0}$.

Due to the relationship between impedance and admittance, the same mapping as the impedance Smith Chart can be used for the admittance Smith Chart just by rotating the whole chart by 180°.

For an impedance of $Z_L = 75 + j75$ in a 50 Ohm system, in order to manipulate it using the admittance chart, we have to normalize and convert it into an admittance value, resulting in $y_L = 0.4 - 0.4j$. Adding parallel inductors and capacitors on the Y Smith Chart is illustrated in Figure 2-6.

**Figure 2-5  Admittance Smith Chart**

For an impedance of $Z_L = 75 + j75$ in a 50 Ohm system, in order to manipulate it using the admittance chart, we have to normalize and convert it into an admittance value, resulting in $y_L = 0.4 - 0.4j$. Adding parallel inductors and capacitors on the Y Smith Chart is illustrated in Figure 2-6.
Combining Figure 2-4 and Figure 2-6, we obtain Figure 2-7 showing the movement when a component is added to an impedance or admittance on the Z-Y Smith Chart.

**Figure 2-6**  Admittance manipulations on the admittance Smith Chart

**Figure 2-7**  Adding lumped components on the Smith Chart
2.2.2 Impedance Matching

RF circuits are generally used to propagate the signals from one point or one subsystem to another point or subsystem. In order to maximize the power transmission and minimize the reflection in an interested frequency range, we need to ensure the impedances are matched between the source and the destination of the signal for optimal power flow. For example, consider the following DC circuit:

![Figure 2-8 An equivalent DC circuits](image)

In Figure 2-8, we have a source with resistance $R_s$ and voltage $V_s$, and the load with resistance $R_L$.

$$V_L = \frac{V_s \cdot R_L}{R_s + R_L}$$

$$P_L = \frac{V_L^2}{R_L} = \frac{V_s^2 \cdot R_L}{(R_s + R_L)^2}$$

When the source resistance $R_s$ is equal to $R_L$, the load $R_L$ will absorb maximum power, although the efficiency is low because half the power is dissipated by the resistive source\(^9\). However in the AC range, both the load and the source impedances could have a resistive part as well as a reactive part, implying that the impedances are usually complex numbers. Like the DC circuits, we still want to transfer the maximum power to the load by equalizing the source and load impedance. Even after making the real parts equal, the impedance matching will still be more complicated and challenging than in DC due to the reactance. For example, consider the following AC circuit where the source impedance is $Z_s = R_s + X_s$ while the load impedance is $Z_L = R_L + X_L$:

![Figure 2-9 An equivalent circuit in AC](image)

In this case, when $R_s$ and $R_L$ are equal and $X_s$ and $X_L$ have opposite sign, $X_s = -X_L$, the source and load impedances are complex conjugates and the maximum power will be delivered from the source to the load after the reactive parts cancel each other out.
RF circuits may not have the source resistance and load resistance equal, and the source
and load reactive impedances are typically not conjugate matched. To match the impedances of
the source and load in this case, a lossless impedance matching network is inserted between the
source and the load to make the real resistances equal and to let the reactance parts
complementary each other. The conjugate matching results in the reactive parts canceling out,
leaving the purely resistive parts of the source and load equal. Under these conditions, the
maximum power will be transferred from the source to the load.

2.2.2.1 Matching with Lumped Components

The simplest matching network is the L section impedance matching network.

![L network matching](image)

The equivalent circuits for these 8 possible configurations of the L sections can be
expressed as:

![Equivalent circuits of the L section matching networks](image)

**Figure 2-10** The L network matching

For (a) in Figure 2-11, the analytic solutions are given by:

\[
B = \frac{R_L Z_0}{\sqrt{R_L^2 + X_L^2}} \left( \sqrt{R_L^2 + X_L^2} - Z_0 R_L \right) \left( R_L^2 + X_L^2 \right)
\]

\[
X = \frac{1}{B} + \frac{X_L Z_0}{R_L} - \frac{Z_0}{BR_L}
\]
For (b), the analytic solutions are given by:

\[ X = \pm \sqrt{R_L (Z_0 - R_L) - X_L} \]
\[ B = \pm \sqrt{\frac{(Z_0 - R_L)/R_L}{Z_0}} \]

There are two possible solutions for both B and X. The positive solution means the element is an inductor while the negative solution implies a capacitor.

The L sections impedance matching networks are simple to use, but the disadvantage is that the circuit Q (quality factor) is fixed. We have to use more components to achieve a desired flexible Q, such as the Pi or T networks.

Beside the analytical solutions for the L sections impedance matching networks, another useful and easy way to design and analyze microwave circuits is to use the Smith Chart.

For the Smith Chart, let’s look at an example and derive the elements values analytically and graphically:

Design an L network to match a load with impedance \( Z_L = 100 + j100 \) to the system characteristic impedance \( Z_0 = 50 \) Ohms. The network should block DC and work at a center frequency of 0.8 GHz. Determine the L and C values.

Because \( R_L \) is greater than \( Z_0 \) and the network needs to block the DC, the L matching network should look like Figure 2-12.

![Figure 2-12 Example design](image)

The Analytical solutions for Figure 2-12 are then as the following:

\[ Z_L = 100 + j100 \], and \( Z_0 = 50 \)

\[ B = \frac{X_L + \sqrt{\frac{R_L}{Z_0} \sqrt{R_L^2 + X_L^2} - Z_0 R_L}}{R_L^2 + X_L^2} = \frac{100 + \sqrt{\frac{100}{50} \sqrt{100^2 + 100^2} - 50 \times 100}}{100^2 + 100^2} = \frac{1 - \sqrt{3}}{200} \]
Using the Smith Chart, the first step is to normalize the $Z_L$ by the characteristic impedance $Z_0$ of 50 Ohm resulting in $z_L = 2 + j2$. Our target is to match this result with the 50 Ohm characteristic impedance or the center of the Smith Chart.

In the admittance Smith Chart, following the shunt inductor we move anticlockwise from start point A ($y_A = 0.25 - j0.25$) to point B along the $0.25 + jB$ constant conductance circle. Point B ($y_B = 1 - j0.43$) is the intersection point of the constant resistance circle $1 + jx$ and the constant conductance circle $0.25 + jB$. Thus, the amount of added shunt susceptance is equal to $-j0.18$, resulting from $-j0.43$ minus $-j0.25$.

\[
X = \frac{1}{B} + \frac{X_L Z_0}{R_L} \frac{Z_0}{BR_L} = \frac{200}{1 - \sqrt{3}} + \frac{100 \times 50}{100} = \frac{50 \times 200}{(1 - \sqrt{3})100} = -87.1
\]

\[
L_{\text{sh}} = \frac{1}{2\pi f_{\text{GHz}} B} = \frac{0.159}{f_{\text{GHz}} B} = 54.3 \text{nH}
\]

\[
C_{pF} = \frac{1}{2\pi f_{\text{GHz}} X} = \frac{159}{f_{\text{GHz}} X} = 2.28 \text{pF}
\]

Using the Smith Chart, the first step is to normalize the $Z_L$ by the characteristic impedance $Z_0$ of 50 Ohm resulting in $z_L = 2 + j2$. Our target is to match this result with the 50 Ohm characteristic impedance or the center of the Smith Chart.

In the admittance Smith Chart, following the shunt inductor we move anticlockwise from start point A ($y_A = 0.25 - j0.25$) to point B along the $0.25 + jB$ constant conductance circle. Point B ($y_B = 1 - j0.43$) is the intersection point of the constant resistance circle $1 + jx$ and the constant conductance circle $0.25 + jB$. Thus, the amount of added shunt susceptance is equal to $-j0.18$, resulting from $-j0.43$ minus $-j0.25$.

**Figure 2-13** The Smith Chart used for simple L impedance match network

The next step is to move from point B ($z_B = 1 + j1.7$) to the target point C along the constant resistance circle $1 + jx$. The series reactance that needs to be added is $-j1.7$, which comes from $j0$ minus $j1.7$. 

28
For the design frequency of 0.8 GHz, the inductor and capacitor values are given by the following:

\[
\frac{1}{j2\pi fL} = \frac{B}{Z_0} \Rightarrow L = \frac{Z_0}{j2\pi fB} = \frac{50}{j2\pi \times 0.8 \times 10^9 \times j0.18} = 54.4 \text{nH}
\]

\[
\frac{1}{j2\pi fC} = X \times Z_0 \Rightarrow C = \frac{1}{j2\pi fXZ_0} = \frac{1}{j2\pi \times 0.8 \times 10^9 \times -j1.7} = 2.3 \text{pF}
\]

### 2.2.2.2 Matching by Using Microstrip Lines

In Lab#1, we know if a length transmission line is terminated by a load \(Z_L\), the transformed impedance can be expressed as:

\[
Z_{\text{in}} = Z_0 \frac{Z_L + jZ_0 \tan \beta \ell}{Z_0 + jZ_L \tan \beta \ell}
\]

\[
\Gamma_{\text{in}} = \Gamma_{\text{t}} e^{-j\beta \ell} = \Gamma_{\text{t}} e^{j\beta \ell}
\]

\[
\Gamma_{\text{L}} = \frac{Z_L - Z_0}{Z_L + Z_0} = \frac{Z_L/Z_0/Z_0/Z_0/Z_0/Z_L - 1}{Z_L + Z_0/Z_0/Z_0} = \frac{z_L - 1}{z_L + 1}
\]

**Figure 2-14**  Impedance and \(\Gamma\) transformation on transmission line

When we move from \(Z_L\) along the transmission line with characteristic impedance \(Z_0\), the variation of the input impedance \(Z_{\text{in}}\) can be easily illustrated on the Smith Chart since the VSWR keeps constant along the line as in Figure 2-15.

Our analysis and design will focus on microstrip lines, especially the shunt microstrip line stubs, because they are easy to be fabricated on the PCB. For a short-circuit microstrip line, the input impedance is \(Z_t = jZ_0 \tan(\beta \ell)\). The input impedance of an open-circuit microstrip line is \(Z_t = -jZ_0 \cot(\beta \ell)\). The impedances for both of lines are purely periodic and reactive varying between short and open along the transmission line in Figure 2-16.

On the Smith Chart, moving \(\frac{\lambda}{4}\) along a short-circuit transmission line will transform the impedance from short to open-circuit. If we move one more \(\frac{\lambda}{4}\), the impedance will be transformed back from open to short-circuit. The open-circuit transmission lines have the same properties.

We'll now use the same example as previously used in Section 2.2.2.1 to show the convenience of using the Smith Chart in manipulating the transmission lines for impedance matching.
Figure 2-15  Transmission line manipulations on the Smith Chart

Figure 2-16  Shunt microstrip line stubs manipulation on the admittance Smith Chart
Figure 2-17  Design example using a cascade microstrip line and a shunt stub

Figure 2-18  Design example using the cascade microstrip line and an open stub
Using the transmission lines, match a load with impedance \( Z_L = 100 + j100 \) to the system characteristic impedance \( Z_0 \) of 50 Ohms and determine the length of microstrip lines when the design frequency is 0.8 GHz.

The design steps for the example in Figure 2-17 and Figure 2-18 are as the followings:

1. Normalize the source and load impedance by the system characteristic impedance, usually 50Ω. In the above design sample, \( Z_L = 100 + j100 \) is normalized to \( z_L = 2 + j2 \), because the system characteristic impedance is 50 Ohms.

2. Determine if the matching routine is from source to load or the other way. In our example, we start from load to source and then try to match the load impedance to the conjugate of the source. However, in this sample the source has real impedance, thus its complex conjugate is itself.

3. Mark the start point impedance on the normalized z Smith Chart as point A.

4. Draw the constant VSWR circle, which is a concentric circle through \( z_L \) or point A in the Smith Chart.

5. Move along the VSWR circle clockwise to the intersection (point B) of the VSWR circle and the \( 1 + jb \) constant conductance circle on the y Smith Chart. The series microstrip line electrical length is equal to half of the moving angle from point A to point B on the constant VSWR circle.

6. Read out the intersection point B admittance \( 1 + jb \) or \( 1 - jb \) on y Smith Chart.

7. Use the shunt short or open stubs with admittance \( -jb \) or \( jb \) to cancel out the admittances in step 6.

8. Use Figure 2-17 or Figure 2-18 to determine the electrical lengths for the shunt open or short-circuit microstrip line stubs.

9. Check the matching by moving from the source (point C) to the end of the cascade microstrip line. The ending point on the Smith Chart should be the complex conjugate of the load (point A).

### 2.3 Simulation

Computer-aided-design programs play a significant role in modern RF circuit and system design. In this lab, we will introduce an inexpensive and simple program called PUFF to perform the circuit analysis. PUFF is a non-profit educational RF/microwave circuit simulation program, and it was developed by Dr. Scott W. Wedge, Dr. Richard Compton, and Dr. David Rutledge in California Institute of Technology.
2.3.1 Getting Started

2.3.1.1 Installing

1. Create a PUFF folder on your computer, like C:\PUFF.
2. Copy all the files in the original PUFF disk to the PUFF folder you just created.
3. Run PUFF by double clicking the PUFF_xp.bat or PUFF.com in the PUFF folder.
4. A blue display screen will appear on your computer screen with information such as copyright notice, hardware information, etc.
5. Press any key. The PUFF will run and load the default circuit file setup.puf. There will then the main display screen as follows:

![Figure 2-19 A PUFF screen snap shot from a VGA display](image)

6. If there are any display problems, please go to the website
   http://www.its.caltech.edu/~Emmic/PUFFindex/PUFFE/winpub.html for further help.
7. A laser printer with at least 600 dpi resolutions is required to print out the layout, your fabrication artwork.
8. When the PUFF runs, the numeric keypad is disabled. You have to use the number keys at the top of the keyboard.
2.3.1.2 PUFF Display Screen

1. There are four different windows in the PUFF screen: Layout window (F1), Plot window (F2), Parts window (F3), and Board window (F4).
2. By pressing the designated function keys F1 through F4 you can move from one window to another.
3. The window at the top center of the screen is the Layout window, where the circuits are built up, and all the parts are visible. There are four ports around the side of the window.
4. By pressing F2, you can reach the Plot window located on the upper left corner of the screen. The Plot window is used to analyze the circuits displayed in the Layout window. In the Plot window, you can set up the frequency points, as well as the desired scattering parameters you would like to display during the analysis.
5. Below the Plot window, there is a message box which is designed to display the error messages and file name requests.
6. The Part window is just below the message box. All the parts used in the circuit will be edited and listed in this window.
7. The window located in the bottom left corner is the Board window. This window specifies the physical dimensions of the Layout window, normalizing impedance \( z_d \), and the design frequency \( f_d \), as well as the substrate material properties and transmission line types.
8. You can have a small help window by toggling F10 in PUFF.
9. By pressing Esc key, you can exit PUFF.
10. All the circuit information and setups in each window will be stored in a file called saved-file-name.puf after you save your design and analysis in Plot window.

2.3.1.2.1 The parts window

- The Parts window can be reached by typing F3.
- The initial parts list in the Parts Window is loaded from the default setup.puf file. You can also read a circuit file saved earlier with extension name .puf by pressing Ctrl-r in parts window.
- You can edit the part list with the Arrow, Backspace, Enter, Ins and Del keys. By hitting the Tab key, you can double the part numbers which can be used in the circuit.
- When you leave the Parts Window, PUFF will check the parts list and also redraw the circuit in the layout window. If the length of a part is greater than the layout board
dimensions or shorter than the circuit resolution parameter $r$ in the circuit file, there will be an error message.

- There are seven different part types available in PUFF:
  1. **atten** (attenuator) part

    The *atten* is the simplest part type in PUFF. The *atten* part is ideal, which means it is matched to the normalizing impedance $zd$ and independent of frequency. Its unit is in dB.

  2. **xformer** (transformer)

    The *xformer* part represents an ideal transformer which is lossless and frequency independent. The unitless turns ratio is the only numerical parameter needed to define the *xformer* part.

  3. **lumped** part

    The lumped part denotes series and parallel combinations of resistance, capacitance, and/or inductance. In PUFF, there are four acceptable units for impedance and admittance:
    
    - $\Omega$ (Ohms, typed as Alt-o)
    - $S$ (Siemens)
    - $z$ (normalized impedance)
    - $y$ (normalized admittance)

    Capacitance and inductance have the units of Farads (F) and Henries (H) respectively. Reactance is expressed by placing a "j" before or after a real number together with the units $\Omega$ or $z$. For example, $100j\Omega$ means $100\Omega$ reactance at design frequency $fd$. PUFF also interprets the positive and negative reactance as inductive and capacitive reactance respectively. PUFF also scales both types of reactance proportionally and inversely with frequency. For example, at $2fd$, $100j\Omega$ will be $200j\Omega$, and $-100j\Omega$ becomes $-50j\Omega$. For series RLC circuits can be simply expressed as a lumped part in PUFF. For example, a lumped part, $1+j10-j10z$, represents a series resonant circuit at design frequency $fd$ with a quality factor of 10. There is also a special character, $\|$ (press Alt-p), used to express a lumped part as a parallel circuit. For example, a parallel RLC resonant circuit can be easily expressed as $50\Omega || 1nH || 1pF$, when the resonant frequency $fd$ is equal to $5GHz$.

  4. **tline** part

    The tline part in PUFF represents an ideal transmission line with characteristic impedance or admittance and finite physical length. The unit of the line length can
be meters (m with an approximate prefix), h (the substrate material thickness), and ° (degree by pressing Alt –d). For example, tline 50Ω 90° denotes a 50 Ω transmission line with a quarter wavelength at design frequency fd. When you have to compensate for discontinues of the transmission line, you can add an addition or subtraction term to make the lines longer or shorten. For instance, tline 50Ω 90°-0.5h will be treated in the analysis as a quarter wavelength line, but it will be drawn shorter by -0.5h (h is the substrate material thickness) on the screen and in the artwork. You can also check the physical length of a tline part by placing the cursor on the part and hitting the “=” key to display the value in the message box. When the physical length of a tline part is larger than the layout board dimensions, there will be an error message. As a result, you have to adjust the board size or use the Manhattan length in the tline part.

5. The qline part

The qline is similar to the tline part but lossy. There is one more parameter Q, quality factor, in the qline part description. The attenuation at design frequency fd is calculated based on the Q value. There are two models for the attenuation calculation outside the design frequency fd. Qd or Q is for a dielectric loss model. Qc is for a conductor loss model. Artwork correction is not allowed for the qline part.

6. The cline part

The cline part denotes a pair of coupled transmission lines. In the part description, there might be one or two impedances or admittances given. If the impedance is greater then zd, PUFF treats it as even-mode impedance, ze; otherwise, it will be treated as odd-mode impedance zo.

7. The device part

The device part in PUFF is used to load saved files, which contain the S parameters over a frequency range. The device part description has the format of device filename.dev.

- Table 2-1 summarizes the seven PUFF part types.

The fhx04 is a sample device file in the PUFF folder. When PUFF calculates the S-parameters during the circuit analysis, it will interpolate linearly between the points in the device file. Therefore, the analysis frequency range should be within the frequency range of the device file. Otherwise, there will be an error message.

Any circuit files saved earlier can be read in as device files to build a complex network containing many smaller circuits. However, the saved circuit file should contain
all the needed S-parameters for interpolation during the analysis. For example, when you have created a two-port circuit and have calculated all of its S-parameters, use the Ctrl-s in the Plot window to save it as a file. Once saved, you can then read it in as a device file using the device part in the Parts window.

In addition to device files, PUFF can also read in S-parameter data files in the EEsof format. For example, a Touchstone file S1P can be used for a 1 port circuit. A two-port circuit needs a S2P Touchstone file containing the network parameters.

PUFF can only read the S-parameters in the magnitude/angle format when you use device part to load any device file. The prefix for fd and zd used in the board window should coincide with those used in the device files, as well as the frequency prefix in the Plot window.

Table 2-1 Part types in PUFF

<table>
<thead>
<tr>
<th>Part types</th>
<th>Part description and examples</th>
</tr>
</thead>
</table>
| 1. atten   | An ideal attenuator with unit in dB  
                      Part description format: atten 5dB |
| 2. xformer | An Ideal transformer  
                      Part description format: xformer 3:l (its turns ratio) |
| 3. lumped  | Resistance and reactance  
                      lumped 50Ω+j50Ω+j50Ω (a series RLC circuit)  
                      lumped 50Ω 1nH 1pF (a parallel RLC circuit)  
                      lumped 100+j100Ω impedance at fd |
| 4. tline   | Ideal transmission line  
                      tline 50Ω 90° (50 Ω transmission line, quarter wavelength at fd)  
                      tline 50Ω 90°-1.0h (artwork length correction added) |
| 5. qline   | Finite Q tline  
                      qline 100Ω 40° 50Qd (Q=20 due to dielectric losses)  
                      qline 100Ω 40° 50Qc (Q=20 due to conductor losses) |
| 6. cline   | Ideal coupled transmission lines  
                      cline 70Ω 90° (treated as even-mode impedance)  
                      cline 70Ω 30Ω 20° (treated as even and odd-mode impedance) |
| 7. device  | Read file containing S parameters data  
                      device fhx04.dev  
                      device ah118.s2p (load file in the Touchstone format) |

2.3.1.2.2 The layout window

The Layout Window is used in PUFF to draw the parts you created in the Parts window. By typing F1, you can move to the Layout Window.

In order to draw the circuit in the layout window, you have to know how to draw a single part first.

1. Type the part number, like a, to select it.
2. Press an Arrow key to draw the selected part in the direction of the arrow.
3. Use the Arrow keys to move the cursor to the place where you want to draw next part.
4. The circuits can be grounded at any point by pressing the "=" key.
5. You can make a path to a connector by typing the port number. The electric length of this connection path does not affect the analysis.
6. The Shift key can be used to erase and move around the layout. You can hold the Shift key down while pressing an Arrow key to erase a part in the arrow direction.
   The ground can be removed by pressing "=" key while holding Shift. The path to any port can be removed by Shift + port number.
7. The Shift key + Arrow key can move the cursor when there is no part to erase.
8. You can also use the Shift key + port number to move the cursor to that port if this port is not connected to any path. This is useful when you want to start your layout from that port rather than the center of the board.
9. If there is already a part in the direction of the Arrow key, PUFF will move over to its end rather than draw overlap on it.
10. If the ends of two parts are closer than the circuit resolution $r$, PUFF will connect them together.
11. PUFF will stop you from drawing a part off the board, but it will not stop you from crossing over a previous part.
12. You can use Ctrl-e to erase the whole layout.
13. Ctrl-n can be used to move the cursor to the nearest node.

2.3.1.2.3 The plot window

When you have finished your layout, you can move to the Plot window by pressing F2 to analyze your circuit and plot the results in a Smith Chart, as well as a rectangular plot. To start the analysis press Ctrl-p. After the analysis has been completed, the S-parameter values will be displayed in the Plot Window at the design frequency. If you would like to read the values at other frequencies, you can use the PgUp and PgDn to move the marker after moving the cursor on the S-parameter symbol.

PUFF displays $S_{11}$ as part of the default setup, but you are able to select more S-parameters to display. Moving the cursor down to the bottom of the Plot window, a marker with the letter S will appear. You can then type the port numbers for the desired S-parameters. If you leave a line blank, it will be erased when you move the cursor.
The frequency point can be set in the Plot window for cubic spline interpolation between calculated points. The point should be less than 500.

Both the Smith Chart and the rectangular plot are controlled by the Plot window. You are able to change the display parameters in the saved circuit .puf file. The Smith chart can be toggled between an impedance or admittance chart by using the Tab key. You can also plot an impulse response or a step response of the network by pressing “i” or “s” respectively.

In the Plot Window, you can use Ctrl-s to save the circuit file. Later you can use Ctrl-r to read the file into the Parts Window. Alt-g can also perform a screen dump of the whole screen to a PCX file.

The layout can be printed out as your fabrication artwork by using Ctrl-a. PUFF will prompt for titles to be placed on top of the printout artwork; however, the artwork will be magnified by the photographic reduction ratio (p) in the circuit file. If the ratio p is not 1 in the circuit file, you have to use a copy machine to reduce the artwork to 1:1 scale.

2.3.1.2.4 The board window

The Board window is used to setup the important parameters for the Layout board, the properties of the substrate material, the normalizing impedance, the design frequency, as well as the transmission line types.

Table 2-2 Parameters in Board window and their descriptions

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description and setup</th>
</tr>
</thead>
<tbody>
<tr>
<td>zd</td>
<td>Normalizing or characteristic impedance used for the S parameters calculation. Unit Ω with approximate prefix.</td>
</tr>
<tr>
<td>fd</td>
<td>Design frequency used for electrical length calculation. Unit Hz with an approximate prefix which should coincide with the prefix used in the device file and in the Plot window.</td>
</tr>
<tr>
<td>er</td>
<td>Relative dielectric constant of substrate materials.</td>
</tr>
<tr>
<td>h</td>
<td>Thickness of substrate materials.</td>
</tr>
<tr>
<td>s</td>
<td>Board dimensions used to specify the Layout window.</td>
</tr>
<tr>
<td>c</td>
<td>Connector separation between ports 1 and 3, and ports 2 and 4 in the layout.</td>
</tr>
<tr>
<td>Tab</td>
<td>Functional key used to change the transmission line type.</td>
</tr>
</tbody>
</table>

2.3.1.2.5 The component sweep

Component sweep is used to optimize the circuits. A question mark (?) is placed before the part parameter in the part list, such as lump?10nH to sweep an inductor value during the circuit analysis.

When PUFF plots in the Plot window, the frequency will be held constant at the design frequency $f_d$. The values specified in the x-axis of the rectangular plot will be substituted for the
component sweep values, but only single component sweep is allowed by PUFF. For example, a series RLC circuit \(10 + 50j - 50j \Omega\) is not allowed in PUFF component sweep.

### 2.3.2 An Example Using PUFF

This section provides a step-by-step example on how to use PUFF for RF or microwave circuit design and analysis. **Example:** Use transmission lines to match a load with impedance \(Z_L = 100 + j100\) to the system characteristic impedance \(Z_0\) (50 Ohms) and determine the lengths of the microstrip lines when the design frequency is 0.8 GHz, the substrate material is FR4, \(E_r = 4.75\), and \(h = 1.6\)mm.

#### Step 1
1. Press the F4 to reach the Board window.
2. The active Board window is indicated by the flashing cursor for \(zd\) and the highlighted F4 at the top of the Board window.
3. Using the arrow, backspace, Ins and Del keys set up the following: \(zd = 50 \Omega, fd = 0.80\) GHz, \(E_r = 4.75, h = 1.6\)mm, \(s = 200\)mm, and \(c = 60\)mm. Tab toggle to microstrip type.

#### Step 2
1. Use F3 to make the Parts window active, which is indicated by the flashing cursor for part \(a\) and the highlighted F3 at the top of Part window.
2. Edit part \(a\) description as: \(a\) lumped \(100 + 100j \Omega\) (Alt-o for \(\Omega\)).
3. Edit part \(b\) description as: \(b\) tline \(50 \Omega\) \(^\circ\) (Alt-d for \(^\circ\)).
4. Edit part \(c\) description as: \(b\) tline \(50 \Omega\) \(^\circ\).

#### Step 3
1. Move to the Layout window by typing F1.
2. Press “a” to select part \(a\) lumped \(100 + 100j \Omega\) to be drawn in the layout area.
3. Use arrow \(\downarrow\) to draw part \(a\).
4. Use “=” to connect one end of part \(a\) to ground.
5. Use arrow \(\uparrow\) to move the cursor to another end of part \(a\).
6. Press “b” to select part \(b\) and arrow \(\uparrow\) to draw \(b\) tline \(50 \Omega\) \(^\circ\) at the top of part \(a\).
7. Press “1” to connect the port 1 to part \(b\) for analysis.

#### Step 4
1. Press F2 to reach the Plot window.
2. Set the frequency point to 200.
3. Use Ctrl +s to save the circuit file and give the file a name, for example Sample.puf.
4. Type Esc key to exit PUFF.
**Step 5**

1. Use Notepad to open the file Sample.puf.
2. Set up the $f_l$ to 0 and $f_u$ to 200, which will be the sweep range for part b.
3. Exit and save the Sample.puf file.

**Step 6**

1. Run PUFF again.
2. In Part window, type Ctrl-r to read in Sample.puf file.
3. Pressing F2 to make Plot window active.
4. Press Ctrl-p to sweep the part b.

![Figure 2-20 A screen dump of the design example of part b sweep](image.png)

5. Press Tab key to toggle the Smith Chart to admittance circles.
6. Move the cursor on the S11 symbol. If making a path to port 2, you have to change the s-parameter to S22 for display.
7. Use the Page Up or Page Down keys to move the maker to the intersection point of the $g = 1$ constant conductance circle and the input impedance of the circuit on the upper half of the Smith Chart.
8. At $b = 130.2^\circ$, on the upper half of the $g = 1$ normalized conductance circle, an open-circuit stub can be used for matching.
9. Go back to Part Window to edit the part b length to 130.2 degrees.
10. Change the \( c \) parameter in the Board window to 150mm.
11. In the Layout window, use the arrow keys to move the X to the top of part \( b \), then press “c” to highlight part \( c \) and use \( \rightarrow \) to place part \( c \).
12. In Plot window, save (Ctrl-s) and exit (Esc) PUFF and setup the \( fu \) as 75 in the sample.puf file to sweep the part \( c \) value when reloaded in PUFF.

**Step 7**

1. Repeat step 6 to find the length of part \( c \) to finish the impedance matching.
2. At 58.0 degrees, we have the matching circuit at \( fd \).

![Figure 2-21 A screen dump of part c sweep in step 6](image)

**Step 8**

1. In Part window, edit part \( c \). Now the electrical lengths of both part \( b \) and part \( c \) are held constant.
2. In Plot window, save the file and exit PUFF.
3. Edit the \( fu \) in sample.puf to 1.5.
4. Run PUFF.
5. In the Part window, read in the saved sample.puf file.
6. In the Plot window, press “p” to analysis the circuit again. Now PUFF will sweep the frequency from zero to 1.5 GHz.
Step 9

1. Press Alt-g to save a screen dump in your PUFF folder.
2. Add the corrections on the tline parts due to discontinuities of microstrip lines.
3. In the Plot Window, use Ctrl-a to print out the layout as the fabrication artwork after the analysis has been completed.

Having introduced the PUFF, we can use it to analyze the matching network for the design example in the previous section. The results are showed in Figure 2-23.
Figure 2-23  Screen dumps of the PUFF analysis on impedance matching example
2.4 Fabrication

Warning:

*In the following prototyping process, some highly corrosive chemicals will be used. Before you start your experiments, please check the laboratory safety requirements for the storing, using, and disposing of the chemicals used.*

2.4.1 Preparing the Artwork

1. After you finish your layout and the simulation from the previous sections, activate the Plot window and press CTRL + “a” to print out your layout on transparency films from a Laserjet printer, the print quality has to be at least 600 dpi.

2. If the transmission lines are too long, you might need to adjust the photographic reduction ratio “p” in your saved PUFF setup files in order to print out the artwork on one page. For example, if the reduction ratio is 1.25 when you print out your artwork, you have to use the photocopy machine to reproduce the artwork using a magnification of 80% in order to have a 1:1 scale.

3. Due to the discontinuities of microstrip lines, the lines have to be compensated by adjusting the microstrip line lengths in your artwork.

2.4.2 Exposing the Board

1. The exposing process should be implemented in a safelight environment.

2. Setup your 416-x exposure kit from M.G. Chemicals following instructions provided with the kit.

3. Cut the board according to your circuit size with a sharp saw. Leave some spare room on your board because the cutting could damage the photoresist on the board edges.

4. Peel off the white protective coating sheet of only one side of your double-sided copper clad board.

5. Place the presensitized board underneath the fluorescent lamp having the side without protective coating toward the tube. Put your artwork on the top of the board and make sure the printed side faces down. If you need two positive films, align them carefully.

6. Put the Acrylic weight on the top of your artwork. The weight can be replaced with a heavier quartz glass to prevent light leakage under the traces.

7. Turn on the lamp to expose the board for 10 minutes.
2.4.3 Developing the Exposed Board

1. Rubber glove and eye protection are strongly recommended in this process due to corrosive chemicals used.

2. In a plastic tray, dilute the M.G. chemical developer with cold water by 1:12, and stir it carefully while diluting.

3. Put the board in the tray with the exposed side facing up. Use a plastic pair of tweezers to gently shake the board while immersed in the developer. The developer will quickly remove the exposed green photoresist and your circuit patterns will show up clearly. The developing process takes about 30 seconds.

4. Stop the developing process once able to see copper without any green photoresist, except your circuit areas.

5. Rinse the board immediately with cold water.

2.4.4 Etching the Developed Board

1. Ensuring there is enough ventilation, warm up the ferric chloride to 50 °C in a plastic tray.

2. Place the board with the developed side facing up into the tray with enough ferric chloride solution to submerge the board.

3. Shake the tray gently to speed up the etching reaction until all the copper on the developed area is completely removed.

4. Rinse the board with water thoroughly.

Figure 2-24 Implementation of the matching design sample
2.5 Design of Maximum Gain for Small Signal

(Conjugate Matching)

In Lab#1, we reviewed sets of significantly useful parameters in RF circuit design and analysis. We especially looked at S-parameters, which can describe the characters of a network and can be measured easily by the network analyzers.

A typical single stage amplifier with measured S-parameters inserted between the source $Z_s$ and load $Z_L$ has the following block diagram.

\[
S_{11} + \frac{S_{12}S_{21}}{1 - S_{22}} \Gamma_L = \Gamma_{in}
\]
\[
\Gamma_{ox} = S_{22} + \frac{S_{12}S_{21}}{1 - S_{11}} \Gamma_s
\]

Figure 2-26  Block diagram of a single amplifier inserted between the source and load
The transducer power gain $G_T$ is given by:

$$G_T = \frac{P_{\text{LOAD}}}{P_{\text{Source}}} \frac{(1 - |\Gamma_L|^2)(|S_{21}|^2)(1 - |\Gamma_S|^2)}{(1 - S_{11}\Gamma_S)(1 - S_{22}\Gamma_L) - S_{12}S_{21}\Gamma_L\Gamma_S}$$

The $G_T$ is dependant on the source and load terminations. To obtain a maximum gain from the available active device, we should find simultaneous conjugate matches at both sides of the device.

The requirements are as follows:

$$\Gamma_S = \Gamma_m^*, \text{ and } \Gamma_L = \Gamma_{\text{out}}^*, \text{ or}$$

$$\Gamma_S = (S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L})^*, \text{ and } \Gamma_L = (S_{22} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{11}\Gamma_S})^*.$$ 

The two simultaneously matched reflection coefficient solutions are given by:

$$\Gamma_{MS} = \frac{B_1 \pm \sqrt{B_1^2 - 4|C_1|^2}}{2C_1}, \quad \text{and} \quad \Gamma_{ML} = \frac{B_2 \pm \sqrt{B_2^2 - 4|C_2|^2}}{2C_2}$$

$$B_1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta|^2, \quad \text{and} \quad B_2 = 1 + |S_{22}|^2 - |S_{11}|^2 - |\Delta|^2$$

$$C_1 = S_{11} - S_{22}^* \Delta, \quad \text{and} \quad C_2 = S_{22} - S_{11}^* \Delta$$

$$|\Delta| = |S_{11}S_{22} - S_{12}S_{21}|$$

![Block diagram for the maximum gain amplifier design](image)

Figure 2-27  Block diagram for the maximum gain amplifier design

After we have the simultaneous solutions for both sides to obtain the maximum gain, the next step is to use the previous impedance matching approaches to match the two desired impedances with the given source and load impedances for maximum gain.

2.6 Experiments

Using a pre-stabilized amplifier, for which the measured S-parameters are specified for operational conditions at 0.8 GHz:
Design the input and output matching networks by using the microstrip line stubs. Specifically, the shunt open-circuit stubs are recommended due to easy fabrication.

**Guidelines:**

1. A MATLAB routine can be used to find the simultaneously matched reflection coefficients for the design of maximum gain. Impedance matching methods can also be performed by using the microstrip line stubs through the Smith Chart.

2. After you finish your design, use PUFF to simulate your results. Refer to design example 1 for your simulation.

3. If your simulation results agree with your expectations, you can finish the fabrication after you correct the microstrip line discontinuity effects by adjusting the lengths of the shunt arm with \( lh \).

4. Using the Network Analyzer, measure the boards at your design frequency with 150 MHz span.

5. Your report should include the MATLAB results, the Smith Chart design routine, a PUFF simulation screen dump, the completed board, and the board's completed measurement results.

6. If there are any discrepancies between the real test results and your expectations, provide your proposals.

\[
S_{11} = 0.29 \angle -75, \\
S_{12} = 0.072 \angle 58, \\
S_{21} = 7.20 \angle 78, \\
S_{22} = 0.36 \angle -6.0, \\
\]
3 LAB #3 RF LOW PASS FILTER DESIGN

3.1 Objective

The objective of Lab#3 is to review the fundamentals and how to design low pass RF filters using the insertion loss and stepped-impedance methods. PUFF is used to analyze the filter circuit design before the fabrication is implemented.

3.2 Background of RF Filters

3.2.1 Filter Basics

RF filters play a very important role in microwave communications. There is always a need for the ability to select signals within a special desired frequency range and reject or block all other unwanted frequency components. Thus, filters are frequency selective networks with both pass and stop bands.

Based on the frequency responses, there are four types of filters: low pass, high pass, band pass, and band stop filters.

Figure 3-1 Four basic type filters: (a) Low pass filter (b) High pass filter (c) Band pass filter (d) Band stop filter
3.2.1.1 Basic Filter Types

1. Low pass Filter
   A low pass filter passes signals from zero Hz up to a special frequency called the cutoff frequency. As the input frequency increases beyond the cutoff frequency, the filter gradually attenuates more of the signal. A series inductor and a shunt capacitor can form a simple LC low pass filter.

2. High pass Filter
   The name implies that high pass filters attenuates the signal for frequencies below the cutoff frequency and pass signals higher than the cutoff frequency. The high pass and low pass filters have mirror images for frequency responses.

3. Band pass Filter
   Band pass filters only allows signals within a special frequency range called the bandwidth to pass and gradually rejects all other frequencies out the band. A simple band pass filter can be made by cascading a low pass followed by a high pass filters.

4. Band stop Filter
   The frequency responses of band stop filters are similar to band pass filters except that it allows signals outside the band to pass through but blocks signals within the band.

3.2.1.2 Filter Parameters

1. Insertion Loss
   The insertion loss refers to the ratio of the power delivered to the filter load versus the power available from the source when the filter is matched at both ends. A minimum insertion loss in the pass band is desirable.

2. Bandwidth
   The bandwidth usually refers to the frequency range where the attenuation is less than 3 dB, relative to the minimum attenuation. Bandwidth may also be referred to as the 3dB bandwidth where $\text{BW} = f_{\text{up}}^{3\text{dB}} - f_{\text{low}}^{3\text{dB}}$.

3. In-band Ripple or Flatness
   Ripple is a gain fluctuation within the bandwidth, while it’s another measure is called the band’s flatness, the difference of maximum and minimum loss in the band.

4. Shape Factor
   $\text{Shape factor} = \frac{\text{BW}^{60\text{dB}}}{\text{BW}^{30\text{dB}}}$ and indicates how fast the attenuation around the cutoff frequency changes.
5. Terminations and VSWR

Because the filter generally is designed to be terminated by resistive source and load, its overall performance is mainly determined by the quality of its terminations.

6. Group delay

Group delay is the signal propagation time through the filter. Unequal group delay within the filter bandwidth causes distortion in the communication.

3.2.1.3 Typical Filter Frequency Responses

1. Butterworth (Maximum flatness) Filter

This type of filter has a maximum flatness gain response, or minimum ripple, in the pass band. The insertion loss (IL) of the low pass Butterworth filters can be expressed as a binomial function:

\[ IL = -10 \log \left[ 1 + \alpha^2 \left( \frac{f}{f_c} \right)^{2n} \right] = -10 \log \left[ 1 + \alpha^2 \left( \frac{\omega}{\omega_c} \right)^{2n} \right] = -10 \log \left[ 1 + \alpha^2 \left( \frac{\omega}{\omega_c} \right)^{2n} \right] \]

where \( f_c \) is the cutoff frequency, \( n \) is the filter order or the number of the reactive components, and \( \Omega = \frac{\omega}{\omega_c} \) is the normalized frequency.

![Butterworth filter graph](image)

**Figure 3-2** Butterworth (Maximum flatness) filter

At the cutoff frequency, \( f = f_c \) or \( \Omega = 1 \), the IL equals to \( 1 + \alpha^2 \). The insertion loss will be -3 dB, if \( \alpha = 1 \). From the insertion loss formula, it should be apparent that when \( n \) increases, the attenuation increases proportionally.
2. The Chebyshev (equal ripple) Filter

The insertion loss for the low-pass Chebyshev filter is expressed as a Chebyshev polynomial:
\[
\text{IL} = -10 \log \left( 1 + \alpha^2 T_n^2 \left( \frac{f}{f_c} \right) \right) = -10 \log \left( 1 + \alpha^2 T_n^2 (\Omega) \right)
\]

Where \( \Omega = \frac{\omega}{\omega_c} \)

\( \alpha \) is a constant used to control the ripple in the pass band, \( \alpha = \sqrt{10^{10} - 1} \),
\( r \) is ripple in the pass band, expressed in dB,
\( T_n \) is the Chebyshev polynomial dependent on the frequency,
\( f_c \) is the 3dB cutoff frequency.

3. The Elliptic and Bessel filter

Elliptic filters have a faster increase in attenuation than Chebyshev filters beyond the cutoff frequency but have ripples in the stop band. Unlike the Butterworth or Chebyshev filter, whose attenuations increase monolithically after the cutoff frequency, for Elliptic filters there is a minimum attenuation in the stop band.

Bessel filters have the same attenuation responses as the Butterworth filters in their pass band but the out of band attenuations are less than the Butterworth filter. Bessel filters have the advantage of a flatter delay.

The Elliptic low pass filter has the best selectivity due to the fastest or sharpest attenuation after the cutoff frequency but the distortion from group delays is the worst. The Bessel low pass filter exhibits the best flat group delay but has the worst selectivity due to the flat attenuation above the cutoff frequency.

3.2.2 Low Pass Filter Design

An ideal filter would have a minimum insertion loss, a maximum flatness gain response in the pass band, and the sharpest attenuation after the cutoff frequency without any group delay distortion. However, in designing real filters, compromises for the specifications are usually required to meet special application requirements.
3.2.2.1 Insertion Loss Method

3.2.2.1.1 Low pass filter prototypes

The insertion loss (IL) refers to the ratio of power available from the source to the power delivered to the load when the filter source and the load are both matched. The IL ratio equals the reciprocal of the transducer power gain:

$$IL = 10 \log \frac{1}{|S_{21}|^2} = 20 \log \frac{1}{|S_{21}|}$$

It can also be expressed as the ratio of power delivered to the load with a filter inserted to the power delivered to the load without the same filter.

The amount of insertion loss for a particular filter is given by the amount of reflection due to mismatches in the pass band and the amount of power dissipated in the resistive components of the filter. If the filter components are ideal, the insertion loss entirely comes from the reflection. As a matter of fact, the filter rejects the unwanted frequencies by using the out of band mismatch; the unwanted signals are reflected back.

The insertion loss method is a systematic way to achieve a trade off between the desired responses and specifications for the filter verses the filter’s application requirements.

To examine the insertion loss method, let’s review and implement the Butterworth and Chebyshev low pass filter designs by using existing normalized prototypes and then transforming their frequency and impedance to create a real filter. The normalized low-pass filter prototypes have a cutoff frequency $\omega_c = 1$ and unity load. Although we’re working with low pass filters here, the low pass filter prototype can also be converted into a high pass, band pass, or band stop filter through frequency and impedance transformations.

Figure 3-5 Two different low pass filter prototypes: (a) Starting with a shunt component (b) Starting with a series component
In Figure 3-5 (a), $g_0$ is the source resistance, $g_{n+1}$ is the load resistance and $g_1, g_2, \ldots, g_n$ are the inductance for the series inductors or capacitance for shunt capacitors.

In Figure 3-5 (b), $g_0$ is the source conductance, $g_{n+1}$ is the load conductance. $g_1, g_2, \ldots, g_n$ are the inductance for the series inductors or capacitance for shunt capacitors.

For a low pass filter design, we have to determine the value for each of the above components, the filter’s type and order, and its size based on the desired performance. The filter’s size mainly depends on the desired attenuation in the stop band and the insertion loss characteristics in the pass band.

![Figure 3-6 Butterworth filters attenuation versus filter orders](image)

**Table 3-1 Coefficients for Butterworth filters**

<table>
<thead>
<tr>
<th>N</th>
<th>$g_1$</th>
<th>$g_2$</th>
<th>$g_3$</th>
<th>$g_4$</th>
<th>$g_5$</th>
<th>$g_6$</th>
<th>$g_7$</th>
<th>$g_8$</th>
<th>$g_9$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.0000</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.4142</td>
<td>1.4142</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1.0000</td>
<td>2.0000</td>
<td>1.0000</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.7654</td>
<td>1.8478</td>
<td>1.8478</td>
<td>0.7654</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.6180</td>
<td>1.6180</td>
<td>2.0000</td>
<td>1.6180</td>
<td>0.6180</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.5176</td>
<td>1.4142</td>
<td>1.9318</td>
<td>1.9318</td>
<td>1.4142</td>
<td>0.5176</td>
<td>1.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.4450</td>
<td>1.2470</td>
<td>1.8019</td>
<td>2.0000</td>
<td>1.8019</td>
<td>1.2470</td>
<td>0.4450</td>
<td>1.0000</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.3902</td>
<td>1.1111</td>
<td>1.6629</td>
<td>1.9615</td>
<td>1.9615</td>
<td>1.6629</td>
<td>1.1111</td>
<td>0.3902</td>
<td>1.0000</td>
</tr>
</tbody>
</table>
For practical Butterworth filter design, after we determine the filter's order based on the attenuation requirements in the stop band, all the g values can be obtained from Table 3-1. Chebyshev filters refer to Table 3-2 and Table 3-3 with different pass band ripples\textsuperscript{12}.

**Table 3-2**  
Coefficients for Chebyshev filters with 0.5dB ripple  
<table>
<thead>
<tr>
<th>N</th>
<th>g₁</th>
<th>g₂</th>
<th>g₃</th>
<th>g₄</th>
<th>g₅</th>
<th>g₆</th>
<th>g₇</th>
<th>g₈</th>
<th>g₉</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.6986</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.4029</td>
<td>0.7071</td>
<td>1.9841</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1.5963</td>
<td>1.0967</td>
<td>1.5963</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.6703</td>
<td>1.1926</td>
<td>2.3661</td>
<td>0.8419</td>
<td>1.9841</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1.7058</td>
<td>1.2296</td>
<td>2.5408</td>
<td>1.2296</td>
<td>1.7058</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1.7254</td>
<td>1.2479</td>
<td>2.6064</td>
<td>1.3137</td>
<td>2.4758</td>
<td>0.8696</td>
<td>1.9841</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>1.7372</td>
<td>1.2583</td>
<td>2.6381</td>
<td>1.3444</td>
<td>2.6381</td>
<td>1.2583</td>
<td>1.7372</td>
<td>1.0000</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>1.7451</td>
<td>1.2647</td>
<td>2.6564</td>
<td>1.3590</td>
<td>2.6964</td>
<td>1.3389</td>
<td>2.5093</td>
<td>0.8796</td>
<td>1.9841</td>
</tr>
</tbody>
</table>

**Table 3-3**  
Coefficients for Chebyshev filters with 3 dB ripple  
<table>
<thead>
<tr>
<th>N</th>
<th>g₁</th>
<th>g₂</th>
<th>g₃</th>
<th>g₄</th>
<th>g₅</th>
<th>g₆</th>
<th>g₇</th>
<th>g₈</th>
<th>g₉</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.9953</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3.1013</td>
<td>0.5339</td>
<td>5.8095</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3.3487</td>
<td>0.7117</td>
<td>3.3487</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>3.4389</td>
<td>0.7487</td>
<td>4.3471</td>
<td>0.5920</td>
<td>5.8095</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>3.4817</td>
<td>0.7618</td>
<td>4.5381</td>
<td>0.7618</td>
<td>3.4817</td>
<td>1.0000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>3.5045</td>
<td>0.7685</td>
<td>4.6061</td>
<td>0.7929</td>
<td>4.4641</td>
<td>0.6033</td>
<td>5.8095</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>3.5182</td>
<td>0.7723</td>
<td>4.6386</td>
<td>0.8039</td>
<td>4.6386</td>
<td>0.7723</td>
<td>3.5182</td>
<td>1.0000</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>3.5277</td>
<td>0.7745</td>
<td>4.6575</td>
<td>0.8089</td>
<td>4.6990</td>
<td>0.8018</td>
<td>4.4990</td>
<td>0.6073</td>
<td>5.8095</td>
</tr>
</tbody>
</table>

Figure 3-7  
Chebyshev filter with 0.5dB ripple attenuation versus filter orders
3.2.2.1.2 Filter Transformations

The previous section showed how to use the low pass filter prototype responses and how to obtain the components’ values from tables in designing a normalized low pass filter. Next, we illustrate how to transform the normalized values to real values in terms of frequency and impedance.

1. Frequency Transformation

Frequency transformation is used to scale the normalized capacitor and inductor values to real values based on the operational cutoff frequency. In the low pass filter prototypes, the cutoff frequency is normalized to \( 1 = \Omega = \frac{\omega}{\omega_c} \). To obtain a cutoff frequency from 1 to \( \omega_c \), we have to replace \( \Omega \) with \( \Omega \omega_c \), while the inductor and capacitor reactance should remain the same.

\[
j \Omega L = j \frac{\omega}{\omega_c} L = j \omega L' \Rightarrow L' = \frac{L}{\omega_c}
\]

\[
j \Omega C = j \frac{\omega}{\omega_c} C = j \omega C' \Rightarrow C' = \frac{C}{\omega_c}
\]

L and C are the values in the prototypes, and \( L' \) and \( C' \) are the real values after scaling.

2. Impedance Transformation
In the low pass filter prototypes, both the source and the load are equal to 1 (with the exception of the even order Chebyshev filters). If the real source and load impedance is not equal to 1, an impedance transformation is required. For example, if the real source resistance is $R_s$, the required new parameters can be obtained by multiplying the prototype impedance by the given $R_s$:

$$R'_g = 1 \times R_s$$
$$R'_L = R_L \times R_s$$
$$L' = L \times R_s$$
$$C' = \frac{C}{R_s}$$

Applying both the frequency and impedance transformations, we obtain the new filter component values:

$$L' = \frac{R_g \times L}{\omega_c} \quad \text{and} \quad C' = \frac{C}{R_g \times \omega_c}$$

By using appropriate frequency and impedance transformations, low pass filters can be converted into high pass filters and/or band pass filters into band reject filters.

### 3.2.2.1.3 Filter Implementation

When the frequency rises up into the microwave range, the lumped components have difficulties for practical filter realization; consequently, the use of distributed components becomes desired. Microstrip lines are wildly used in PCB to integrate with other surface mount devices because they are easy to fabricate.

From Lab#1, we know that a short-circuit transmission line acts like an inductor and an open-circuit transmission line behaves like a capacitor when their electrical length is less than $\frac{\pi}{2}$. For a short-circuit microstrip line with a physical length of $\ell$, the input impedance is given by:

$$Z_{in} = jZ_0 \tan(\beta \ell)$$

If $\ell = \frac{1}{8} \lambda_c$ at the cutoff frequency then,

$$Z_{in} = jZ_0 \tan(\beta \ell) = jZ_0 \tan\left(\frac{\beta \ell}{\frac{1}{8} \lambda_c}\right) = jZ_0 \tan\left(\frac{\pi f}{4 f_c}\right) = Z_0 S$$
S is called the Richards transformation, which equals to $j\tan\left(\frac{\pi}{4}\Omega\right)$.

An open-circuit microstrip line is similar to the short-circuit microstrip line:

$$Z_{in} = -jZ_0\cot(\beta\ell)$$

$$Y_{in} = \frac{1}{Z_{in}} = jY_0\tan(\beta\ell) = jY_0\tan\left(\frac{\pi}{4}\Omega\right)$$

The result from these formulas is that we can use the short and open-circuit microstrip lines with $1/8$ wavelength at the cutoff frequency to replace the inductor and capacitor respectively in the low pass filter prototypes. The frequency responses of this implementation, however, are band limited due to the periodic reactive properties of the transmission lines. In addition, using series short-circuit microstrip lines as inductors for the prototype filters are not easy to fabricate. To combat this problem, we introduce two of the four Kuroda identities which are used to convert the series short-circuit microstrip line stubs into shunt open-circuit microstrip line stubs for practical implementation.

In the Kuroda identities, each box presents a transmission line with the characteristic impedance indicated in the box and length $\frac{1}{8}\lambda_c$ at cutoff frequency. The inductors and capacitors represent the short and open-circuit transmission line stubs, respectively.

![Figure 3-9 Two Kuroda identities](image)

### 3.2.2.1.4 Design Example

Design a Chebyshev low pass filter from the given specifications: cutoff frequency is 0.8 GHz, in-band ripple is 0.5 dB, attenuation of at least 25 dB at $\Omega = 1.5$ in a 50 Ohm system, and fabricated on FR4 material with the relative dielectric constant of $\varepsilon_r = 4.75$ and a thickness 60 mils.
Step 1
To start, we look up the 0.5dB ripple Chebyshev low pass filter prototype response in Figure 3-7, and determine that a 5th order filter is required. Using the corresponding look-up table, gives the prototype design values:

\[ g_1 = g_5 = 1.7058; \quad g_2 = g_4 = 1.2296; \quad g_3 = 2.540; \quad g_6 = g_7 = 1.0 \]

![Figure 3-10 Step 1 of design example](image1.png)

Step 2
Using the Richards transformation, we replace the inductors and capacitors with microstrip line stubs, \( L = Z_0 \) and \( C = 1/Z_0 \).

![Figure 3-11 Step 2 of design example](image2.png)

Step 3
Using the Kuroda identities, we convert all the short-circuit microstrip lines to open-circuit lines. Starting at both sides, we insert a unit element, which has unity impedance and will not affect the filter performance:

![Figure 3-12 Step 3 (a) of the design example](image3.png)
Next, we apply two more unit elements at both sides again as shown in Figure 3-14.

**Step 4**

In this last step as shown in Figure 3-15, we have to adjust the unity impedances to our system impedance of 50 Ohms by multiplying each microstrip stubs by 50, and making the length of the microstrip stubs $\frac{1}{8}\lambda_{0}$ at the cutoff frequency of 0.8 GHz.
3.2.2.2 Stepped-Impedance Method

For a short length transmission line, if its electrical length is significantly less than $\frac{\lambda}{2}$ and with high characteristic impedance, the transmission line will act like a series inductor. However, its behavior looks like a shunt capacitor if the characteristic impedance is relatively low. As a result, alternating short high impedance ($Z_a$) and low impedance ($Z_b$) transmission lines form series inductors and shunt capacitors respectively, allowing the creation of low pass filters.

From the equivalent circuits in Figure 3-17 and the impedance transformations in the previous insertion loss design section, we have the following equations:

$$\beta = \frac{LR}{Z_0}$$ (Inductor)
\[ \beta_f = \frac{CZ_L}{R} \] (Capacitor)

In which \( R \) is the desired real resistance of the filter, and \( L \) and \( C \) are the component values in the low pass filter prototype.

Using the stepped-impedance method, we now repeat the low pass filter design that we performed in the previous section using the insertion-loss method.

\[ g_1 = g_{20} = 1.705; \ g_{50} = 1.2296; \ g_2 = 2.540; \ g_3 = g_1 = 1.0 \]

Figure 3-18 Prototype for stepped-impedance design

\( Z_h \) and \( Z_l \) are set to be the highest and lowest characteristic impedances respectively for the microstrip line stubs on a given substrate material. The fabrication accuracy dictates the microstrip line's highest impedance that can be used because the line width will be very thin at higher characteristic impedances. The lowest impedance is determined by the practical width of the microstrip line stubs.

<table>
<thead>
<tr>
<th>Table 3-4 Sections list of the stepped-impedance design example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Section</td>
</tr>
<tr>
<td>---------</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>4</td>
</tr>
<tr>
<td>5</td>
</tr>
</tbody>
</table>

In this lab, we’ll use the FR4 substrate with a dielectric constant of \( \varepsilon_r = 4.75 \) and 60 mils thickness. The highest practical impedance that can be used is about 120 \( \Omega \), and the lowest is about 10 \( \Omega \).

3.3 Simulation

Using the PUFF program introduced in Lab #2, we’ll analyze the above low pass filter designs by both insertion loss and stepped impedance methods. The results are illustrated by Figure 3-19 and Figure 3-20 respectively.
Figure 3-19 PUFF simulation for the insertion-loss design example

Figure 3-20 PUFF simulation results for the stepped-impedance design example
3.4 Fabrication and Measured Results

Figure 3-21 Implementation of insertion-loss filter

Figure 3-22 Implementation of stepped-impedance filter
Figure 3-23  Low-pass filter measured results from the VNA (a) Insertion-loss method design (b) Stepped-impedance method design
Comparing the Lab measurements and the PUFF simulation results, we can see that the low pass filter designed by the insertion loss method has a sharper cutoff than the one designed by the stepped impedance method. Furthermore, there is a discrepancy between the simulation from PUFF and the measured results due to the microstrip loss, discontinuities, as well as the imperfect connectors and fabrication tolerances.

3.5 Experiments

1. Using the Insertion Loss method, design a Chebyshev low pass filter with the following specifications: a cutoff frequency of 0.85 GHz, an in-band ripple of 0.5 dB, a characteristic impedance 50 Ohm, and using a 3rd order filter.

2. Repeat the above Chebyshev filter design with the stepped-impedance method. The minimum microstrip line width should be 0.2mm, and maximum width is 25mm using an FR4 substrate material with \( \varepsilon_r = 4.75 \) and 60mils thick.

3. For the lab write-up, it should include the necessary theory analysis, design procedure, PUFF simulation results (screen dump with different scale/Div), fabrication, and test results from the VNA. There should also be a comparison between your test results with the PUFF simulation and the low pass filter prototypes responses. Any discrepancies encountered between the results, you will want to explain in your report.
4 LAB #4 MIXER CHARACTERISTICS AND MEASUREMENTS

4.1 Objective

This lab is designed to review some important characteristics of RF mixers in communication systems. The main focus will be on the operation and use of passive diode mixers.

4.2 Background Information

Mixers are a critical part for any frequency translation function in modern RF systems. For instance, in RF receivers, mixers are used to convert the input RF frequency into an intermediate frequency (IF) for signal decoding. They are also employed in transmitter paths to help transmit the signals efficiently. By converting the frequency, the mixer makes signal processing easier because the overall gain of the receiver or the transmitter becomes distributed in different frequency spectrums. The distributed gain reduces the potential for oscillations; furthermore, the frequency conversion helps to achieve higher selectivity.

![Antenna](Antenna) ![Mixer](Mixer) ![LO](LO) ![LNA](LNA) ![LPF](LPF) ![IF Amplifier](IF Amplifier)

**Figure 4-1** A simplified receiver front-end block diagram

Using down-conversion mixers in the above receiver as an example, we’ll examine how the mixers convert from one frequency to another.

A mixer usually consists of three ports, the RF input port, the local oscillator (LO) input, and the IF output port. The LO is used to drive the mixer for frequency translation.

\[
I_{IF} = V_{IF} \cos(o_{IF} t) \\
I_{LO} = V_{LO} \cos(o_{LO} t) \\
I_{RF} = V_{RF} \cos(o_{RF} t)
\]

**Figure 4-2** An ideal mixer
Mixers actually use the nonlinear or the time varying properties of the electrical components or circuits to produce other frequencies based upon the two input frequencies.

An ideal mixer would multiply the two input frequencies, for example if we had the following:

**RF signal:** \( f_{RF} = V_{RF} \cos(o_{RF}t) \)

**LO:** \( f_{LO} = V_{LO} \cos(o_{LO}t) \)

The output would equal:

\[
\begin{align*}
  f_{out} &= f_{RF} \times f_{LO} = V_{RF} \cos(o_{RF}t) \times V_{LO} \cos(o_{LO}t) \\
  &= \frac{V_{RF}V_{LO}}{2} \left[ \cos(o_{RF} - o_{LO}t) + \cos(o_{RF} + o_{LO}t) \right]
\end{align*}
\]

The mixer output has both the sum and difference frequencies for the input RF and LO components. To obtain the desired down-conversion \( o_{RF} - o_{LO} \), or up-conversion \( o_{RF} + o_{LO} \), a filter is required to select the desired frequency. In addition, the LO can be either lower than the RF (lower side injection) frequency or higher than the RF (high side injection).

![Image of Frequency spectrum of an ideal mixer](image)

**Figure 4-3** Frequency spectrum of an ideal mixer

In an ideal mixer, we would only have the sum (high sideband) and difference (low sideband) of the two input frequencies; however, in practical there are many undesirable higher ordered frequency products resulting from the mixer’s nonlinear performance.

The image frequency (IF), the mirror image of RF signals about the LO frequency, can also be translated to the same IF frequency. As a result, a pre-selective filter before the mixer or an image reject mixer must be used to suppress the mirror image frequency. Otherwise, the strong interfering image signal could block the receiver or worsen the noise figure.
4.2.1 Conversion Loss

Power conversion gain or loss is the ratio of the IF signals power and the input RF signal power. When an input signal is converted to the low sideband and high sideband identically by the mixer, a filter is used to remove the undesired lower or upper sideband output. The filter automatically removes half of the signal power. This signal loss represents the minimum single sideband conversion loss of 3 dB. Moreover, because of the undesirable higher order products, mixer internal resistance and the mismatch at the mixer’s ports, the single sideband conversion gain is usually higher than 3 dB and dependent on the LO’s power level.

When the input RF is small, the mixer conversion loss will be constant and linear. Lastly, for an active mixer, there is the possibility of having positive gain which can boost the strength of certain signals.

4.2.2 VSWR

From Lab#1, we know that VSWR is used to measure the degree of impedance mismatch. If the port impedance of the mixer is not matched to the characteristic impedance of the system, some input signal power will be reflected back. Because the LO power level affects the mixer’s properties significantly, the VSWR at each mixer port is a function of the LO power level, the operational temperature and the mixer internal circuits design.

4.2.3 Isolation

Port to port isolation, the measurement of insertion loss between any two ports of a mixer, is another critical parameter of a mixer. Ideally, we would like to minimize any interaction between the RF, LO, and IF ports of the mixer, especially for the interactions for the LO since it is generally much stronger than the RF and IF signals. The LO presence at the IF port might obscure the IF output and cause desensitization in the subsequent stages. The LO reverse leakage could cause the LO to be fed to the antenna and cause interference to other receivers if there is no any reverse isolation between the RF port and the antenna.
4.2.4 Noise Figure

The definition of the noise figure of a mixer is given by:

\[
NF = 10 \times \log_{10}\left(\frac{\text{SNR}_{\text{INPUT}}}{\text{SNR}_{\text{OUTPUT}}}ight)
\]

where \(NF\) is usually measured in dB and \(\text{SNR}\) is the signal to noise ratio. The overall noise figure of a mixer is the sum of the conversion loss and the noise added to the IF output by the mixer. When only the lower sideband or the upper sideband is needed at the IF output, we then consider the single sideband noise figure.

A mixer’s noise figure is usually higher than an amplifier’s mainly because of the noise folding from other mixer output products. The mixer additive noise is also from the thermal, shot and flicker noise.

The single sideband noise figure equals the double sideband plus 3 dB because the single sideband conversion loss is 3 dB higher than the double sideband.

4.2.5 1-dB Compression Port

When the mixer input signal power is at low level or small-signal range, its conversion loss remains constant. The output power follows the input power in a dB-to-dB linear manner. That means the output power will increase 1 dB if the input signal power increases by 1 dB. However, this relationship only exists if the input power level is less than a certain point. If the input power increases beyond that point, the mixer goes into saturation and the conversion loss increases. The 1-dB compression point is defined in dBm of either the input signal level or output power level where the real output power deviates 1 dB from its ideal linear response.

![Figure 4-5 Compression point of a mixer](image)

The 1-dB compression point indicates the mixer’s maximum output, two-tone performance, and the upper limit of its dynamic range.

72
4.2.6 Intercept Point

The two-tone third order intercept point is another measurement of the linearity of a mixer. Like the 1-dB compression point, the intercept point can be expressed in dBm referenced to a mixer’s input power or output power. When both a desired signal and a potential interferer are present at the input, there will be some two-tone intermodulation (IM) distortion products, \((m_1 f_{RF_1} + m_2 f_{RF_2}) \pm n \cdot LO\), generated by the mixer’s nonlinear properties. Among all these two-tone IM products, the 3\textsuperscript{rd} order product (the sum of coefficient \(m_1\) and \(m_2\) is equal to 3) is more critical because it is closest to the desired IF product in the frequency spectrum. The 3\textsuperscript{rd} order product is not easily removed by filters and also causes trouble as an undesired frequency translation product. For a lower sideband down-conversion, the 3\textsuperscript{rd} order IM products have the expressions:

\[
\begin{align*}
(2m_{RF_1} - m_{RF_2}) - LO \\
(2m_{RF_2} - m_{RF_1}) - LO
\end{align*}
\]

Figure 4-6 Intermodulation distortions spectrum

In Figure 4-7, when the input signal RFI and RF\(_2\) increase by 1 dB, the 3\textsuperscript{rd} order IM products will increase the order amount 3 dB. Thus, there will be an intersect point between the IF product and the 3\textsuperscript{rd} order intermodulation distortion (IMD\(_3\)) products if the mixer does not compress.

Figure 4-7 Mixer intercept point
The 3\textsuperscript{rd} order intercept point is defined as the input power level or output power level where the desirable IF power level is equal to IMD\textsubscript{3} product's power level. We cannot directly measure the 3\textsuperscript{rd} order intercept point since the mixer will reach the compression point before the fundamental IF product power equals the IMD\textsubscript{3} products' power. Thus, the 3\textsuperscript{rd} order intercept point is calculated by extending the ideal linear responses from the intermodulation IM suppression at a low input power level\textsuperscript{14}.

![Figure 4-8 Mixer IP3 calculation](image)

\[ I_{P3} = P_{\text{in}} - \frac{\text{IM suppression (dBc)}}{2} \]

\[ OIP3 = I_{P3} + \text{Conversion gain} \]

4.2.7 Mixer Spurs

In a real mixer, both the input and LO signals have harmonics generated by the mixer's nonlinear performance. The harmonics can be mixed together and result in undesired frequency components; some of them might be close to the IF and be problematic:

\[ f_{\text{spur}} = m f_{\text{RF}} \pm n f_{\text{LO}} \]

Choosing the IF frequency carefully can minimize the unwanted spurious responses at the mixer's output.

4.3 Diode Mixers

4.3.1 Single-Ended Mixer

Single-ended mixers are the simplest and one of the most fundamental diode mixers: only one Schottky barrier diode is used in the design. The RF and LO signals are superimposed.
through the diplexer or coupler making the diode work at two different points on its I-V curve to perform mixer functions. In the schematic, the diode is biased with a DC power supply, implying that two DC block capacitors are used. An RF choke is employed to block the AC from the DC source. The AC output will pass through a low pass filter to deliver the desired IF.

The DC forward-biased single-ended mixer needs less LO power to optimize the conversion efficiency. In some cases, there is no DC forward biasing and the diode is driven to switch between short and open-circuits operation by only high LO power.

In the single-ended mixer, the Schottky diode can be treated as a nonlinear resistor and the mixer approximates the second nonlinearity or square law behavior. The application of the mixer determines the type of Schottky barrier diode that is used in the circuit design.

![Figure 4-9 A single-ended mixer](image)

In the single-ended mixer, based on the diode’s V-I relationship, the small signal approximation of the pumped Schottky diode is simply expressed as:

\[
I = I_{dc} + I_{ac} = I_{dc} + V G_d + \frac{V^2}{2} G_d
\]

where \( I_{dc} \) is the DC biasing current, \( G_d \) is the dynamic conductance of the diode, and \( V \) is the combined AC of RF input signal and LO signals.

If we have \( V = V_R \cos(\omega_R t) + V_{LO} \cos(\omega_{LO} t) \), the diode \( I \) can be rewritten as:

\[
I = I_{dc} + \left( V_R \cos(\omega_R t) + V_{LO} \cos(\omega_{LO} t) \right) G_d + \frac{1}{2} \left( V_R \cos(\omega_R t) + V_{LO} \cos(\omega_{LO} t) \right)^2 G_d
\]

The first term, \( I_{dc} \), will be blocked by the DC block capacitors in the circuit. The second term will also be filtered away by the IF low pass filter. We are interested in particular with the third term which can be rewritten as:

\[
\frac{\left( V_R \cos(\omega_R t) + V_{LO} \cos(\omega_{LO} t) \right)^2}{2} G_d
\]

\[
= \frac{\left( V_R \cos(\omega_R t) \right)^2 + \left( V_{LO} \cos(\omega_{LO} t) \right)^2 + 2V_R V_{LO} \cos(\omega_R t) \times \cos(\omega_{LO} t)}{2} G_d
\]

75
The low pass filter will remove the first two square terms, leaving the cross products with the desirable frequency translation components. By using trigonometric identities, we have:

\[ I_f = \frac{G_d}{2} \left[ V_{RF} V_{LO} \cos(\alpha_{RF} - \alpha_{LO}) + V_{RF} V_{LO} \cos(\alpha_{RF} + \alpha_{LO}) \right] \]

The IF filter is then used to extract the desired products from the mixer's output and provide impedance matching at the IF frequency.

The major drawback of the single-ended diode mixer is the poor isolation between the RF and LO port. The leakage of the LO can travel to the antenna and become interference for other receivers.

### 4.3.2 Single-Balanced Diode Mixer

Balanced diode mixers are used to improve the isolation between any two ports of the mixer, to suppress the nonlinear IMD products, and to broaden the bandwidth.

In a single-balance diode mixer circuit, two Schottky barrier diodes are connected back-to-back, and a 90 degree or 180 degree hybrid coupler is used for the RF and LO input. Using the quadrature (90°) hybrid coupler construction, we'll show how the single-balanced mixer works.

**Figure 4-10** A double-ended mixer

**Figure 4-11** Hybrid coupler

If the hybrid coupler showed in Figure 4-11 is used for a power split, Pin 1 would be the input port; Pin 2 would be the isolation port and can be connected to a resistive terminator, while Pin 3 and Pin 4 are the two output ports. The outputs on Pin 3 and Pin 4 would have the same...
amplitude but with a 90-degree phase shift. If the coupler's two-output ports were mismatched; the terminator at the isolation port would absorb all the reflected power.

Because of the symmetric construct, the input incident signal can be applied to any port of the coupler. The isolation port is on the same side as the input port while the two output ports are on the opposite side. Because of the mutual isolation between the input and isolation ports, the hybrid coupler is suitable for the RF and LO power injection.

In the single-balanced mixer, the LO power should be high enough to switch both $D_1$ and $D_2$ on and off at the same time. Therefore, the LO power will be 3dB higher than the power in the single diode mixer. $L_1$ and $L_2$ are employed to provide the DC path and IF return. The loss pass filter has the same function as in the single diode mixer.

The mathematical analysis of the single-balanced diode is the same as the single-ended diode except that there are two diodes instead of one. The analysis also differs since the IF currents are combined at the parallel junction point where the spurious products with even LO and RF harmonics will be rejected.

If identical diodes are used, the single-balance mixers will have almost the same conversion loss as the single-ended diode mixers, but the IP3 will be 3dB higher than the single-ended type's. The inherent isolation of hybrid couplers provide better isolation between the LO and RF in comparison to the single-ended diode mixers.

Other diode mixers types include double-balanced, double double-balanced mixers and image reject diodes. These other types are good for isolation, IMD product suppression, bandwidth, and image component rejection.

4.4 Mixer Characteristics Measurements

4.4.1 Conversion Loss and Compression Point Measurements

Conversion loss of a given mixer specifies the frequency translation efficiency. The loss is a function of the applied LO power level to the mixer and also varies with the input RF frequencies. In order to measure the conversion loss flatness in the entire operational bandwidth, we can sweep the input RF frequency in the band with a fixed LO frequency to have a variable IF, or we can sweep the LO frequency to track the input frequency for a fixed IF. In this lab, we will measure one RF frequency point to show the process.

4.4.1.1 Calibrations

1. The VNA frequency span should be setup wide enough to cover the whole frequency range you will be testing.
2. The VNA shall be calibrated for transmission measurements using an approved calibration kit.
3. The loss of assembly cable 1 is measured at the input RF frequencies by the VNA, and the amplitude offset of the signal generator 1 is to be setup with your measured loss.
4. The loss of assembly cable 2 is measured at the LO frequency by the VNA, and the amplitude offset of signal generator 2 is to be setup with your measured loss.
5. The total loss of cables 3 & 4 and the low pass filter is measured at the IF frequency by the VNA, and the attenuation level offset of spectrum analyzer is to be set up with your measured loss.
6. The spectrum analyzer should be calibrated by its automatic alignment function.

4.4.1.2 Measurements

Figure 4-12 Conversion loss measurements setup

The setup in Figure 4-12 is used to measure the conversion loss of a mixer.
1. Connect the RF port to signal generator 1.
2. Connect the LO port to signal generator 2.
3. Connect the IF port to one of the low pass filter ports and the other filter port to the spectrum analyzer.
4. Set the frequency for signal generator 1 to 900 MHz at the RF input and its amplitude to -15 dBm with the modulation function off.
5. Set the frequency for signal generator 2 to 750 MHz at the LO input and its amplitude to 16 dBm with the modulation function off.
6. Setup the spectrum analyzer with the following settings to view the mixer frequency responses without the IF filter: center frequency to 1.5 GHz, span to 3 GHz, RBW to auto, Average to 10, and internal attenuation level to Auto.

7. Using the marker peak search function, read the different frequencies and their raw power level.

8. Setup the spectrum analyzer with the following settings to view the mixer frequency responses without the IF filter: center frequency to 150 MHz, span to 200 MHz, RBW to Auto, Average to 10, and the internal attenuation level to Auto.

9. The conversion loss equals the power difference between the IF power reading from the spectrum analyzer and the RF power level for signal generator 1.

10. Put a 20 dB attenuator at the end of cable 4 and measure the total loss for cable 3 & 4, the LPF, and the 20 dB attenuator. Reset the Spectrum Analyzer with the new measured loss.

11. Measure the mixer’s input compression point by increasing the RF input power until the conversion loss increases one 1 dB.

4.4.2 Isolation and Return Loss

Isolation measurements are used to test the leakage or feed through between any two ports of the mixer. We will use the transmission measurement of the VNA to perform the isolation measurement. Due to the high power level of the LO, the isolation between the LO & the RF, the LO & the IF, and the RF & the IF ports are critical and need to be specified. When performing isolation measurements between any two ports, the other unused port should be terminated by its characteristic impedance.

Performing reflection measurements with amplifiers and filters is similar to mixers. Here we will use the 1-port reflection measurement on the VNA for the VSWR measurements. When we perform the reflection measurements, all other two ports should be terminated properly. If we measure the VSWR of the RF input port and IF port, the full power level should be applied to the LO port and the IF & RF ports should be terminated respectively.
4.4.2.1 Calibrations

1. The VNA frequency span should be setup wide enough to cover the entire test frequency range, RF, LO and IF.
2. The VNA MEAS 1 should be calibrated for transmission measurement with the approved calibration kit and the calibration reference plane should be at the end of cable 1 and cable 2.
3. The VNA MEAS 2 should be calibrated for one-port reflection measurement with the approved calibration kit and the calibration reference plane should be at the end of the cable 1 for Open, Shorts and Loads.

4.4.2.2 Isolation Measurements

The setup in Figure 4-13 is used to measure the isolation between two ports of the mixer.

1. Connect the LO port to the signal generator, and set the generator to produce the full LO power level.
2. Connect the RF input port to the VNA output port through cable 1 and connect the IF port to the VNA input port. Set the VNA frequency range to be the same as the RF input frequency range.
3. Varying the VNA power level in the linear range, use the standard transmission method to measure the insertion loss between the RF and IF ports.
4. Terminate the RF input ports and connect the VNA output port to the LO port. Set the power level of the VNA CW signal to the LO full power level and set the VNA frequency to the LO frequency. Connect the IF port to the VNA input port.
5. Using the transmission measurement on the VNA, directly measure the LO-to-IF isolation.
6. Connect the IF port to the LPF and terminate the other port of the LPF filter. Connect the VNA input to the RF input port.
7. Using the transmission measurement on the VNA, directly measure the LO-to-RF isolation.

4.4.2.3 VSWR Measurements
The setup in Figure 4-13 is also used to measure the VSWR of the mixer. Here the one-port reflection method shall be used for the VSWR measurements.

1. Connect the LO port to the signal generator using the full LO power level.
2. Connect cable 1 to the VNA output and the RF input. Connect the IF port to the LPF whose other port is terminated. Set the VNA frequency range to the same range as the mixer’s RF range.
3. Using the VNA, measure the RF input’s VSWR.
4. Terminate the RF input and switch the cable to IF port 1. Change the VNA frequency to the IF frequency and measure the IF port’s VSWR.
5. Terminate the RF input and connect the IF port to the LPF whose other port is terminated. Connect the cable to the LO port. Set the VNA frequency equal to the LO frequency and the CW power level equal to the LO power level.
6. Using the VNA, measure the LO port’s VSWR.

4.4.3 IP3 Measurements
The IP3 setup is used to measure the Input 3rd Order Intercept Power level.
4.4.3.1 Calibrations

1. Make sure the spectrum analyzer has been properly calibrated.
2. Program the spectrum analyzer per Table 4-1

Table 4-1  IP3 Spectrum Analyzer Setup

<table>
<thead>
<tr>
<th>Description</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center Frequency</td>
<td>200 MHz</td>
</tr>
<tr>
<td>Span</td>
<td>10 MHz</td>
</tr>
<tr>
<td>Reference Level</td>
<td>-10 dBm</td>
</tr>
<tr>
<td>Ref Level Offset</td>
<td>0 dBm</td>
</tr>
<tr>
<td>Attenuation</td>
<td>10 dB</td>
</tr>
<tr>
<td>Averaging</td>
<td>10</td>
</tr>
<tr>
<td>Resolution Bandwidth</td>
<td>30 kHz</td>
</tr>
<tr>
<td>Video Bandwidth</td>
<td>300 Hz</td>
</tr>
</tbody>
</table>

4.4.3.2 Measurements

1. Make sure that the setup is connected properly and the equipment is programmed to the correct values.
2. Read the levels of the four tones on the spectrum analyzer. Calculate IIP3 level using $IIP3 \text{ (dBm)} = \frac{1}{3} \text{MIN} (AB, DC) + \text{Pin} (-15 \text{ dBm})$ where AB and DC are the IMD3 suppressions in dBc as showed in Figure 4-15.
3. Change the input frequency and vary the LO frequency. Track the RF frequency to have a fixed IF output and measure the IIP3 at various frequencies.
4. Record the results.

Figure 4-15  IMD suppression
REFERENCE LIST


84