# DESIGN OF WIDEBAND POWER AMPLIFIER FOR GAN HEMT FOR RADIO APPLICATION

LIM WEN JUN

# FACULTY OF ENGINEERING UNIVERSITY OF MALAYA KUALA LUMPUR

2017

# DESIGN OF WIDEBAND POWER AMPLIFIER FOR GAN HEMT FOR RADIO APPLICATION

LIM WEN JUN

# DISSERTATION SUBMITTED IN FULFILMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTER OF ENGINEERING SCIENCE

# FACULTY OF ENGINEERING UNIVERSITY OF MALAYA KUALA LUMPUR

2017

# UNIVERSITY OF MALAYA ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: Lim Wen Jun

Matric No: KGA140060

## Name of Degree: Master in Engineering Science

Title of Project Paper/Research Report/Dissertation/Thesis ("this Work"):

# Design of Wideband Power Amplifier for GaN HEMT for Radio Application

## Field of Study: Electronics

I do solemnly and sincerely declare that:

- (1) I am the sole author/writer of this Work;
- (2) This Work is original;
- (3) Any use of any work in which copyright exists was done by way of fair dealing and for permitted purposes and any excerpt or extract from, or reference to or reproduction of any copyright work has been disclosed expressly and sufficiently and the title of the Work and its authorship have been acknowledged in this Work;
- (4) I do not have any actual knowledge nor do I ought reasonably to know that the making of this work constitutes an infringement of any copyright work;
- (5) I hereby assign all and every rights in the copyright to this Work to the University of Malaya ("UM"), who henceforth shall be owner of the copyright in this Work and that any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of UM having been first had and obtained;
- (6) I am fully aware that if in the course of making this Work I have infringed any copyright whether intentionally or otherwise, I may be subject to legal action or any other action as may be determined by UM.

Candidate's Signature

Date:

Subscribed and solemnly declared before,

Witness's Signature

Date:

Name:

Designation:

#### ABSTRACT

In the field of communication, the vast developing of the communication system in past decades that changes the way of interaction between mankind. In this thesis, the focus has been given to the design of single system where it can operate at wideband frequency in simplest circuit and enhanced performances. In this project, a wideband power amplifier (PA) is designed using real frequency technique (RFT). The motivation of this project is to show that by using the idea of RFT and practical implementation of distributed elements and lumped elements (mixed-lumped elements design), the PA designed is able to achieve satisfying results with wide bandwidth range of 80 MHz to 2200 MHz. The measurement results reported good performance over the bandwidth of the interest (i.e. power of 41 dBm, efficiency about 67% and gain more than 13 dB), and reasonable agreement with the simulated data. The results are significant for single-ended GaN HEMT device for the wideband operation starting from low frequency 80 – 2200 MHz.

### ABSTRAK

Beberapa tahun kebelakangan ini, perkembangan pesat dalam bidang komunikasi telah mengubah cara interaksi di antara manusia. Dalam thesis ini, tumpuan diberikan dalam merekabentuk sistem tunggal yang beroperasi pada frekuensi jalur lebar dengan litar yang paling mudah dan menambahbaikkan prestasi. Dalam kajian ini, sebuah penguat kuasa berjalur lebar di rekabentuk menggunakan '*Real Frequency Technique'* (*RFT*). Motivasi dalam kajian ini adalah untuk membuktikan dengan menggunakan kaedah RFT dan pelaksanaan praktikal unsur diedarkan dan unsur tergumpal (rekabentuk unsur tergumpal campuran). Penguat kuasa yang direka tersebut boleh mencapai keputusan yang memuaskan dengan lebar jalur di antara 80 MHz ke 2200 MHz. Keputusan pengukuran menunjukkan prestasi yang baik mengatasi lebar jalur terpilih (contoh: kuasa 41 dBm, kecekapan 67% dan keuntungan melebihi dari 13 dB), dan menyetujui dengan data simulasi. Keputusan tersebut adalah sangat ketara untuk peranti tunggal-akhir GaN HEMT untuk operasi jalur lebar bermula frekuensi 80 – 2200 MHz.

#### ACKNOWLEDGEMENTS

First of all, I would like to express my gratitude towards my supervisor, Dr. Narendra Kumar for provide the endless support and guidance throughout this project. His kindness and encouragement always inspired me to do my best for this work. I am very grateful to my supervisor for all the kind help and patience advice all the while.

I would also like to acknowledge Motorola Solutions, the main collaborator and supporter of this project in terms of funding. I would like to acknowledge CREST for the help of financial management. I also would like to express my appreciation to the companies and vendors who providing us the components or services for prototype development. I would like to thanks my project partner Ms. Huan Hui Yan who helped me a lot in terms of technical writing, component purchasing and experiment equipment setup so that my project can be carried out smoothly.

Finally, I would like to thank my parents for their unconditional love and support. They supported me spiritually whenever I felt like giving up. Their love and support inspired me to strive for the best.

# TABLE OF CONTENTS

Abstractiii
Abstrakiv
Acknowledgementsv
Table of Contentsvi
List of Figuresix
List of Tablesxi
List of Symbols and Abbreviationsxii
CHAPTER 1: INTRODUCTION1
1.1 Background1
1.2 Problem statement
1.3 Research aim2
1.4 Objectives
1.5 Organization of Dissertation
CHAPTER 2: LITERATURE REVIEW
2.1 Introduction of Power Amplifier
2.1.1 Power Amplifier Basics
2.1.2 Power Amplifier Classification
2.1.2.1 Transconductance PA
2.1.2.2 Switch Mode PA7
2.1.3 Recent works in PA design
2.2 Power Amplifier Figures-of-Merits
2.2.1 Efficiency
2.2.1.1 Drain Efficiency11

		2.2.1.2 Power Added Efficiency (PAE)	11
	2.2.2	Gain	12
	2.2.3	Output power	14
2.3	S-parai	meter	15
2.4	Broadb	and Impedance matching	19
2.5	Coplan	ar Waveguide with Ground (CPWG)	23
	2.5.1	Introduction of CPWG	23
	2.5.2	Advantages of CPWG compared to microstrip	23
2.6	Real F	requency Technique	24
	2.6.1	Background	24
	2.6.2	Theory of Real Frequency Technique	26
2.7	Summa	ary	29
CHA	PTER	3: METHODOLOGY	31
3.1	Design	of matching network	31
	3.1.1	Initial Guess Technique	31
	3.1.2	Initial Guess Simulation using MATLAB	31
3.2	Mixed	-lumped and distributed elements implementation	36
	201	Distributed and Lawrence delements	26

3.2	Mixed-lumped and distributed elements implementation		
	3.2.1	Distributed and Lumped elements	5
	3.2.2	Mixed Lumped and distributed elements matching network	7
	3.2.3	CPWG Implementation	3
	3.2.4	Capacitor implementation	2
3.3	DC Bia	using Circuit44	1
3.4	Advanc	ce Design System (ADS) Simulation47	7
	3.4.1	Simulation of Matching Network with Mixed-Lumped Design47	7
	3.4.2	3.5.2 Simulation of Complete PA Circuit	3
3.5	PCB la	yout Design49	)

	3.5.1	Layout Construction using OrCAD CIS Capture Software	49
	3.5.2	Footprint construction for Each Component	50
	3.5.3	Layout Design	51
3.6	Heat S	ink Design	53
3.7	PCB Fabrication55		
3.8	Measurement Setup		
3.9	Summary		59
			~ 0

CHA	PTER	4: RESULTS AND DISCUSSION	60
4.1	Matchi	ng Network Development	60
4.2	Mixed-	Lumped Element Implementation	63
	4.2.1	CPWG Parameter Selection	64
	4.2.2	ATC Capacitor Implementation	66
4.3	Simula	tion Results	67
	4.3.1	Simulation of Matching Networks with Mixed-Lumped Design	68
	4.3.2	4.3.2 Simulation of GaN HEMT PA	71
4.4	Measu	rement Results	76
4.5	Compa	rison of performances with various works	80
4.6	Summa	iry	81
СНА	PTER	5: CONCLUSION	82
5.1	Conclu	sion	82
5.2	Future	Work	82
Refe	rences		84
List	of Public	cations and Papers Presented	87

# LIST OF FIGURES

Figure 2.1: The generic schematic of a power amplifier
Figure 2.2: The reduced conduction angle current waveform
Figure 2.3: Basic circuit of Class-E power amplifier
Figure 2.4: Class-F power amplifier using a quarter-wave transmission line9
Figure 2.5: General two-port network with characteristic impedance Zo17
Figure 2.6: Source and load impedance circuit
Figure 2.7: Impedance matching equalizer network
Figure 2.8: Simplified diagram of CPWG23
Figure 2.9: Lossless matching network in terms of Darlington's driving point impedance
Figure 3.1: Command window for parameter insertion
Figure 3.2: Example of matching network obtained from execution of RFT algorithm 35
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm 36 Figure 3.4: LineCalc tool interface for the synthesis of <i>l</i> and <i>w</i>
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of <i>l</i> and <i>w</i>
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of <i>l</i> and <i>w</i>
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of l and w  40    Figure 3.5: Schematic of S-parameter conversion of 2-port to 1-port  41    Figure 3.6: Schematic circuit for capacitor simulation  43    Figure 3.7: Simulation result of the capacitors  43    Figure 3.8: Basic DC biasing circuit  44    Figure 3.9: Basic DC biasing circuit  46
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of <i>l</i> and <i>w</i> 40    Figure 3.5: Schematic of S-parameter conversion of 2-port to 1-port  41    Figure 3.6: Schematic circuit for capacitor simulation  43    Figure 3.7: Simulation result of the capacitors  43    Figure 3.8: Basic DC biasing circuit  44    Figure 3.9: Basic DC biasing circuit  46    Figure 3.10: Graph of current against voltage for GaN biasing.  46
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of l and w  40    Figure 3.5: Schematic of S-parameter conversion of 2-port to 1-port  41    Figure 3.6: Schematic circuit for capacitor simulation  43    Figure 3.7: Simulation result of the capacitors  43    Figure 3.8: Basic DC biasing circuit  44    Figure 3.9: Basic DC biasing circuit  46    Figure 3.10: Graph of current against voltage for GaN biasing  46    Figure 3.11: Construction of IMN with mixed-lump design  47
Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm  36    Figure 3.4: LineCalc tool interface for the synthesis of <i>l</i> and <i>w</i> 40    Figure 3.5: Schematic of S-parameter conversion of 2-port to 1-port  41    Figure 3.6: Schematic circuit for capacitor simulation  43    Figure 3.7: Simulation result of the capacitors  43    Figure 3.8: Basic DC biasing circuit  44    Figure 3.9: Basic DC biasing circuit  46    Figure 3.10: Graph of current against voltage for GaN biasing  46    Figure 3.11: Construction of IMN with mixed-lump design  47    Figure 3.12: PA circuit construction in ADS  48

Figure 3.14: PCB layout of the PA design	52
Figure 3.15: Heat Sink Layout Top View	53
Figure 3.16: Heat Sink Layout Side View	53
Figure 3.17: PCB fabrication by Brusia Engineering Sdn Bhd	56
Figure 3.18: Complete PA PCB (top view)	57
Figure 3.19: Complete PA PCB with heat sink (side view)	57
Figure 3.20: Experiment setup for data measurement	58
Figure 4.1: Schematic circuit for IMN	61
Figure 4.2: Graph of TPG against frequency for IMN	61
Figure 4.3: Schematic circuit for OMN	62
Figure 4.4: Graph of TPG against frequency for OMN	63
Figure 4.5: Schematic circuit of IMN with mixed-lumped design	68
Figure 4.6: Graph of TPG against frequency for IMN with mixed-lumped design	69
Figure 4.7: Schematic circuit of OMN with mixed-lumped design	69
Figure 4.8: Graph of TPG against frequency for OMN with mixed-lumped design	70
Figure 4.9: Complete schematic circuit of GaN PA	71
Figure 4.10: Graph of PAE against frequency of GaN PA (Simulation)	73
Figure 4.11: Graph of gain against frequency of GaN PA (Simulation)	74
Figure 4.12: Graph of output power against frequency of GaN PA	74
Figure 4.13: Simulated and Measured result of PAE for GaN PA	77
Figure 4.14: Simulated and Measured result of gain for GaN PA	78
Figure 4.15: Simulated and Measured result of output power for GaN PA	79

# LIST OF TABLES

Table 3.1: Input Description	
Table 3.2: Properties of Rogers's 4350B PCB	material55
Table 4.1: Input Parameters	
Table 4.2: Zo and $\theta$ computation for IMN	
Table 4.3: Zo and $\theta$ computation for OMN	
Table 4.4: Capacitance computation and comp	parison for IMN66
Table 4.5: Capacitance computation and comp	parison for OMN67
Table 4.6: Length and width of CPWG	
Table 4.7: ATC capacitor value	
Table 4.8: Comparison of Performances of Wi	deband Power Amplifier in different works

# LIST OF SYMBOLS AND ABBREVIATIONS

Q	:	Quality factor
$R_S$	:	Source resistance
$R_L$	:	Load resistance
$X_S$	:	Source reactance
$X_L$	:	Load reactance
а	:	Incident wave
b	:	Reflected wave
S	:	Scattering matrix
Zin	:	Input impedance
Zout	:	Output impedance
<i>S</i> <sub>11</sub>	:	Input wave reflection with output terminated
$S_{21}$	:	Forward transmission ration with output terminated
$V_S$	:	Source voltage
P <sub>RFout</sub>	:	Output power of RF
$P_{DC}$	:	Power of DC supply
$\eta_{PAE}$	:	Power added efficiency
Zo	÷	Characteristic impedance
Θ	:	Electrical length
ADS	:	Advance Design System
ATC	:	American Technical Ceramics
CPWG	:	Coplanar waveguide with ground
DC	:	Direct current
FET	:	Field effect transistor
GaN	:	Gallium Nitride

- HBT : Heterojunction bipolar transistor
- HEMT : High-electron-mobility transistor
- IMN : Input matching network
- OMN : Output matching network
- PA : Power amplifier
- PAE : Power added efficiency
- RF : Radio frequency
- RFT : Real frequency technique
- TPG : Transducer power gain
- VSWR : Voltage standing wave ratio

## **CHAPTER 1: INTRODUCTION**

#### 1.1 Background

Wireless radio technologies play an important role in our daily life, which enables us to communicate with other parties no matter how far is the distance. In recent years, radio technology has been undergo vast development in terms of features and physical characteristics. Nowadays, it is better for wireless device to be as small as possible for the ease of mobility. The features of wireless radio devices are very compact to meet the specific requirements of the users with different preference in various fields. With more features in the device, the power consumption will be increased. More components need to be added into the design in order to ensure the efficiency of the device is not compromise, and also the battery life is able to sustain a certain period of time for it to be utilized. However, as the components in the device increases, the size of the device will increase. Therefore, design engineers nowadays are using various methodologies to design the system not only smaller in size, but the performance of the system will also not to be compromised. For radio system design, PA is an essential element in the system which will affect the overall performance. In order to obtain good efficiency, linearity and gain, PA is the stage that needed to be carefully design. One of the key challenges to the development is PA design whereby the requirements such as bandwidth, power level, efficiency, gain, etc are design trade-off.

In this work, a design method of wideband PA employing RFT as a guideline to obtain input matching and output matching network (IMN and OMN), respectively from the actual source and load pull measured results. Note that this work is dedicated to collaboration with Motorola Solutions. The topology interest of this work is required to cover from low frequency such as 80 MHz up to 2200 MHz. The performances of this design is excellent (output power above 41 dBm, gain more than 13 dB and maximum efficiency of 67%). Furthermore, from realization in printed circuit board (PCB) level, where a technique with mixed-lumped elements for smaller form factor. The performance of the prototype showed the approach is acceptable to the radio applications over the entire bandwidth operation.

### **1.2 Problem statement**

In the process of PA design, the performances of the PA is the main concern of the engineers and designers. As the development of the radio communication systems is become vast emerging, it is required by the system to support as high data rate as possible. Therefore, as a key component of the radio communication system, the performance of PA is very vital to ensure the entire communication system to function accordingly. However, due to the complexity of the design process, it is a challenge to design a PA to meet specific requirement. Therefore, researchers are constantly finding new methodology to simplify the design work and produce effective outcome. Also, the future generation of communication systems have more functions integrated in one device. This will result in the complexity of the circuit of the component of the system, particularly for PA design. Therefore, the design of PA should be as simple as possible, which directly will affect the production cost and size, but the performances of the PA design should not compromised the performances.

Hence in this work, it focuses on methodology to design wideband PA in a simplest circuit network, and also with enhanced efficiency and gain performances.

## **1.3** Research aim

The aim of this work is to design a wideband PA using RFT and mixed-lumped distribution elements implementation to obtain a simplest circuit network and achieved enhanced performances. With the proposed technique, the PA is develop with using GaN HEMT technology with efficiency over 40%, output power of 10 W and achieved gain

performance more than 11 dB, in the desired frequency of 80 MHz to 2200 MHz. The measurement data of fabricated prototype will be collected and analyzed.

# 1.4 Objectives

The objectives of this research are as follow:

- 1. Design of wideband PA using mixed-lumped elements implementation with enhanced performances in terms efficiency, gain and output power.
- To investigate and develop wideband PA covering bandwidth of 80 MHz to 2200 MHz using simplest circuit network.
- 3. To develop the prototype board based on industrial requirements with the

## **1.5** Organization of Dissertation

The outline of this dissertation is structured as follows.

**Chapter 2** describes the literature review where the power amplifier is introduced and the classification of power amplifier is presented. A technical view of recent work has been explained.

**Chapter 3** demonstrated the methodology of the GaN PA design. From the development of matching network to the implementation of mixed-lumped elements design, the detail of each procedures are explained. Design of the layout of PCB and heat sink also discussed in this chapter. Experimental setup is presented for measurement data collection.

**Chapter 4** presents the simulation data and measured data of the performances (PAE, gain, and output power) of the designed GaN PA. Both of the data are compared and discussed.

**Chapter 5** presents the conclusion of this work and also discussed about recommendation of future work.

## **CHAPTER 2: LITERATURE REVIEW**

#### 2.1 Introduction of Power Amplifier

#### 2.1.1 **Power Amplifier Basics**

In the past decade, wireless technology emerged as a vital communication tool in our daily life. Wireless power transfer system has been investigated by many scientists and researchers in order to improve the current technology for high efficiency transfer. Power amplifiers are key elements of most microwave and communication systems. Specifications of the power amplifier are vital in designing microwave system. Figure 2.1 shows the generic schematic of a power amplifier.



Figure 2.1: The generic schematic of a power amplifier

# 2.1.2 Power Amplifier Classification

### 2.1.2.1 Transconductance PA

The classification of the transconductance type power amplifiers is based on the conduction angle of the drain current. The conduction angle is defined as the fraction of a period during which the transistor is carrying current. Classes A, B, AB and C are

classified in this category. The reduced conduction angle current waveform is shown in Figure 2.2 and  $\alpha$  is the conduction angle.



# Figure 2.2: The reduced conduction angle current waveform

In terms of design, Class-A power amplifier is the simplest amplifier type. Its quiescent point is biased at halfway between device pinch-off and saturation region. Hence, the transistor operates during the complete cycle and the conduction angle  $\alpha$  is 360<sup>0</sup>. Since Class-A power amplifier operates between cut-off and saturation, it is considered as the most linear power amplifier. However, the disadvantage of this type of power amplifier is low efficiency, with a theoretical maximum of 50%. So Class-A PA is mostly found in those microwave system that require high linearity and can withstand the trade-off of low efficiency.

For Class-B PAs, the quiescent point is at the threshold voltage of the device. Its conduction angle  $\alpha$  is 180<sup>0</sup>, and according to theory, its maximum efficiency is 78.5%. The linearity degrades somehow due to the fact that Class-B power amplifiers only conduct half of the cycle, while in theory also for Class B excellent linearity can still be obtained, this is less trivial than for Class A. Class-B power amplifiers are often

implemented using a push-pull configuration, which uses two transistors in parallel; each amplifying one half of the RF input signal. In push-pull amplifiers, all even harmonics of the load current are eliminated.

The conduction angle  $\alpha$  of Class-AB PAs is between 180<sup>0</sup> and 360<sup>0</sup>. So its quiescent point are flexible which can behave more like a Class-A or Class-B amplifier. This class allows a trade-off between linearity and efficiency. Class-AB power amplifiers can also be implemented using push-pull configuration.

Class-C power amplifiers are biased below the threshold voltage of devices, and hence the conduction angle  $\alpha$  is between 0<sup>0</sup> and 180<sup>0</sup>. Their efficiency can be high but somehow the maximum output power has been sacrificed. Naturally they are quite nonlinear, so they can be only used in applications whose linearity requirement is not too high, or they can be used together with linearization techniques.

### 2.1.2.2 Switch Mode PA

Switch mode PAs can provide larger efficiencies than the transconductance PAs. In switch mode PAs, the transistor is driven by a large input signal (i.e. into compression), such that the transistor operates as a switch rather than as a current source. The overlap of the non-zero drain current and non-zero drain voltage versus time is minimized, as a result higher efficiency can be obtained. In principle, the drain efficiency of switch mode PAs can reach 100% without sacrificing output power. In practice, various loss mechanisms degrade the efficiency of switch mode PAs. In order to improve the linearity of switch mode amplifiers, several advanced amplifier systems have been proposed, like LINC (Linear amplification using Nonlinear Components) and EER (Envelop Elimination and Restoration).

Class E power amplifier is one of the most straightforward switch mode amplifier implementations, which uses transistor operated as an on-to-off switch. The drain current and voltage waveforms do not overlap simultaneously, which minimizes the power loss and maximizes the efficiency. (Andrei Grebennikov, Sokal, & Franco, 2012) A typical Class-E circuit is shown in Figure 2.3. The device output capacitance is included in the shunt capacitor C. A series resonator, a reactive component jX, and a load R<sub>L</sub> are connected in series. The output signal of power amplifier is sinusoidal due to the fact that the series resonator resonates at fundamental frequency. The phase shift between the output voltage in the load and the drain voltage is adjusted by the reactive component jX.



Figure 2.3: Basic circuit of Class-E power amplifier

In Class-F PAs, harmonic traps are used to make the even harmonics and odd harmonics to be short and open respectively. As a result the drain waveform can be shaped, yielding a square wave for the drain voltage and a half-sinusoidal wave for the drain current. High efficiency can be achieved due to the minimized overlapping of the two waves. In practical, harmonic tank resonators are impossible to connect in infinite number. Quarter-wave transmission lines are often used in Class-F design. (El Din, Geck, & Eul, 2009) The quarter-wave transmission line is equivalent to an infinite number of series-resonate circuit. The schematics of Class F using a quarter-wave transmission line and a LC tank are shown in Figure 2.4. Impedance transformation can be provided by quarter-wave transmission line. Class F power amplifier has low peak voltage and current, an advantage compared to Class E amplifier, but it is a complex load network, which is the disadvantage of this type.



Figure 2.4: Class-F power amplifier using a quarterwave transmission line

To summarize, switch-mode PAs have higher efficiencies than transconductance PAs. However, they are inherently more narrowband, because resonators are needed at the output tuned to the fundamental frequency. Class A is suitable for broadband amplifier design. For the efficiency consideration, Class AB bias like conditions can also be used in wideband PA design. Here the self-biasing properties of the device can push it to Class-A operation at higher drive levels. In truly wideband amplifier design (more than octave), the harmonic matching condition (e.g. shorted 2nd harmonics) can no longer be established without additional measures.

#### 2.1.3 Recent works in PA design

In order to develop an efficient PA system, research work has to be done on previous work to investigate identify the weakness and improve the design in our own implementation. Some of the previous works have been carried out to propose the design methodology of PA and enhancement has been done in terms of performances. PA designed in (Chen, Xia, Merrick, & Brazil, 2017) using Bayesian optimization strategy provides results of average efficiency around 45%-53%. The work carried out in (Mughal, Kashif, Cheema, Imran, & Azam, 2015), PA designed using GaN able to achieve maximum efficiency of 67% and output power of 37 dBm in the frequency range of 150-550 MHz. The work presented in (Mohadeskasaei & Zhou, 2016) proposed design method with Butterworth/Chebyshev low pass filter topology to obtain a microstrip-based matching network achieved a satisfactory results of 63.6% and output power of 43 dBm in the desired frequency range of 1.2-1.4 GHz. A PA designed in (Narendra et al., 2012) proposed a new technique by vectorially combined the output current sources when the transistors are loaded by their distributed output networks. The peerformances of this design is excellent (output power more than 43 dBm, gain of 37 dB and maximum efficiency of 57%)

# 2.2 **Power Amplifier Figures-of-Merits**

The performances of the power amplifiers are the main concern of the designers. Several parameters of the power amplifiers provide indications to the designers regarding the performances of the power amplifier. In this section, we are particularly focus on a few parameters that are commonly used for performance indications.

# 2.2.1 Efficiency

Efficiency is the measure on how good of the conversion of one energy to another energy in a device. The energy that does not undergo conversion in this process will be dissipated as heat energy. In designers' point of view, heat energy is the bad by-product of energy conversion. It is represented by the Greek symbol  $\eta$  in all of the mathematical expression. Conversion of DC power to RF power is the primary concern in microwave engineering, which constitute the efficiency of the device. There are many ways to calculate the efficiency of the power amplifier, which will be introduced in this section.

#### 2.2.1.1 Drain Efficiency

Drain efficiency is the efficiency that measured from the drain terminal network of the transistor. Drain efficiency gets its name from Field Effect Transistor (FET) devices, which is the primary terminal that DC power supply will directly supply to. In terms of mathematical expression, drain efficiency can be expressed as follows:

$$\eta_{drain} = \frac{P_{RFout}}{P_{DC}} = \frac{P_{RFout}}{V_{DC} \times I_{DC}}$$
(2.1)

Where P<sub>RFout</sub> =output power of RF

P<sub>DC</sub> =power of DC supply

Drain efficiency is the measure of how much the DC power is converted into RF power in a single device. However, there is a problem when using drain efficiency as a benchmark of the performance. This is due to the fact that RF input power is not considered in the calculation the drain efficiency. RF input power can be significant because the gain for the device is low. Drain efficiency mostly use in FET-based device, such as pHEMT, but in the case of bipolar transistor device such as HBT, this parameter can be referred as "collector efficiency".

#### 2.2.1.2 Power Added Efficiency (PAE)

Power added efficiency (PAE) takes into consideration of input RF power of the device in its calculation, although the computation way is quite similar with drain efficiency. PAE is widely used by designer to benchmark the efficiency of the designed device. The mathematical expression for PAE is:

$$\eta_{power-added} = \frac{P_{RFout} - P_{RFin}}{P_{DC}} = \frac{P_{RFout} - P_{RFin}}{V_{DC} \times I_{DC}}$$
(2.2)

Where P<sub>RFout</sub> =output power of RF

$$P_{RFin}$$
 = input power of RF

In terms of theory, the PAE will be similar to drain efficiency in the case of power amplifier with infinite gain. However in practical case, drain efficiency will be always higher than PAE. When the gain of the power amplifier is low, the input RF power is included in the computation of efficiency. Once the gain of the device achieve 30dB or more, the value of PAE and drain efficiency will become very close to each other due to the fact that the input power will be less than 0.1% of the output power (30dB is 1000 in linear scale).

$$\eta_{power-added} = \eta_{drain} \frac{G-1}{G}$$
(2.3)

Due to the fact that the maximum gain has the tendency to decrease with frequency, the maximum PAE of a device will also decrease with frequency.

#### 2.2.2 Gain

In radio frequency power amplifier, it is important for designer to consider power gain rather than voltage gain. This is because at this frequency range reflections (both current and voltage) become to occur in the transmission lines. So, characterization of the circuit is done by s-parameter and it is based on incoming and outgoing powers. (Zhang et al., 2016) The most commonly used definition of gain is transducer gain which is expressed as:

$$G = \frac{P_{load}}{P_{avail}} \tag{2.4}$$

Where  $P_{load}$  = power deliver to load by amplifier

P<sub>avail</sub>= power available from the source

The power available from the source is actually the same as power delivered to the amplifier input by the source with the condition that the input impedance and source impedance are matched. (In the case of real impedances, Ri=Rs and impedances in complex form, Zi=Zs)

If *Ri=Rs*, the power available (power delivered to matched amplifier input) can be expressed as:

$$P_{avail} = P_{input-matched} = \frac{V_i^2}{R_i}$$
(2.5)

The power delivered to load resistance R<sub>L</sub> by amplifier's output is:

$$P_{load} = \frac{V_L^2}{R_L} \tag{2.6}$$

With the available expression of  $P_{avail}$  and  $P_{load}$ , these two equations are substituted into equation (2.4) and the transducer gain expression is obtained in terms of *V* and *R*.

$$G = \frac{\frac{V_L^2}{R_L}}{\frac{V_i^2}{R_i}} = \frac{\frac{V_L^2}{V_i^2}}{\frac{V_L^2}{R_L}} \times \frac{R_i}{R_L}$$
(2.7)

$$G = A_{\nu \, load}^2 \frac{R_i}{R_L} \tag{2.8}$$

Where  $Av_{load}$  =voltage gain at load.

Please note that power gain in dB is  $10 \log (G)$  where voltage gain in dB is  $20 \log (A_v)$ .

The maximum power will be transfer from amplifier to load if the load is matched to the amplifiers' output impedance ( $Z_o=Z_L$ ), or in the case of resistive load and output impedance ( $R_o=R_L$ ). Therefore transducer gain G is maximum when the output is matched as well as input.

#### 2.2.3 Output power

In the design of RF microwave circuit, output power measurement is one of the crucial factor that we should focus on in order to obtain satisfactory performances of the design. In a system, the signal is passing from one component to the succeeding component. If the output power signal is too high, the performance will be nonlinear and distortion will occur, or even damage the whole circuit. On the other hand, if the output power signal is too low, the signal can be obscured in noise.

At DC and low frequency, current and voltage measurements are simple and straightforward. Power of the system can be easily compute by the following expression:

$$P = IV = \frac{V^2}{R} = I^2 R$$
 (2.9)

The measurement of current and voltage becomes more difficult when the frequency approaches 1 GHz, therefore power measurement is preferable in most application for high frequency. This is because power is maintained constantly throughout the frequency bandwidth along the transmission line but current and voltage may vary

along the transmission line as standing waves produced by the incident and reflected travelling waves. Hence, in RF and microwave frequency, power is more easily to measured and understand, also it is a very useful fundamental quantity compared to voltage and current for performance measurement.

A figure of merit that often employed in the transmission theory is voltage standing wave ratio (VSWR). For maximum power transfer, the load impedance should be equal to the source impedance. However, in almost every practical case the incident signal is reflected back to the source by the load. Therefore, when incident wave and reflected wave are present, standing wave is produced. It is called standing wave due to the fact that the envelope of the wave does not change with time but remain stationary. The ratio refers to the ratio of maximum value and minimum value of envelope, which is a measure of relative amounts of opposite travelling waves.

#### 2.3 S-parameter

In RF circuit design, S-parameter is a very important component to carry out RF circuit design, analysis and measurement. In fact in earlier times, Y-parameter is used to carry out the tasks of the RF circuit design and analysis, but it requires opens or shorts on ports during the measurement, which is very inconvenient for high frequency broadband measurement. (Bowick, Blyler, & Ajluni, 2008) Therefore, the invention of S-parameter has solved the problem of measurement for high-frequency applications. S-parameter are defined and measured with terminated ports in a characteristic reference impedance. S-parameter is well-received by most of the modern network analyzers due to the fact that the networks being analyzed are often employed by insertion of characteristic reference impedance in a transmission medium. In this regard, S-parameter has an advantage which it is related to the commonly specified performance parameters such as return loss and insertion loss.

S-parameters, also known as scattering matrix is a mathematical expression that quantifies propagation of RF energy through multi-port network. It allows us to understand and accurately analyzed the characteristics of the complicated network as simple "black box". During the propagation of RF signal, some incident signals will be reflected back out of incident port, some of it might be entered into the incident ports and scattered to other ports. The remaining incident signal will be disappeared as heat or even electromagnetic radiation.

S-parameters describe the response of signals incident to any or all of the ports in an N-port network. It is represented in a matrix format, with the number of rows and column equal to the number of ports. Normally, the incident port is represented by the second number in the subscript, while the responding port is represented by the first subscript. Therefore the representation of  $S_{12}$  refers to response of port 1 due to the signal at port 2. One-port and two-port network are the most commonly found networks in RF and microwave analysis.

In this section, much attention is given to two-port network. Two-port S-parameters are defined by a set of voltage travelling waves. As mentioned, when a voltage wave from source is incident to a network, some part of voltage wave is transmitted through the network; some part of the voltage wave is reflected back to the source. A diagram of two-port network is shown in Figure 2.5.



Figure 2.5: General two-port network with characteristic impedance Zo

The variables shown in the diagram represent the incident waves and reflected waves at each port. The magnitude of a and b can be thought of as voltage-like variables, normalized using a specified reference impedance. In fact, the square of these magnitudes represent the travelling power waves.

- $|a_1|^2$  = incident power wave at the network input
- $|b_1|^2$  = reflected power wave at the network input
- $|a_2|^2$  = incident power wave at the network output
- $|b_2|^2$  = reflected power wave at the network output

From Figure 2.5, the matrix algebraic expression of 2-port network S-parameters is shown below.

$$\begin{pmatrix} b_1 \\ b_2 \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{pmatrix} \times \begin{pmatrix} a_1 \\ a_2 \end{pmatrix}$$
(2.10)

The equations of the 2-port network can be derived from the above matrix expression.

$$b_1 = a_1 S_{11} + a_2 S_{12} \tag{2.11}$$

$$b_2 = a_1 S_{21} + a_2 S_{22} \tag{2.12}$$

For each S-parameter, its value can be determined by setting either the value of  $a_1$  or  $a_2$  to 0. For example, if the value of incident angel  $a_2$  is set to 0,  $S_{11}$  and  $S_{21}$  can be simply derived from equations (2.11) and (2.12).

$$S_{11} = \frac{b_1}{a_1}$$
(2.13)  
$$S_{21} = \frac{b_2}{a_1}$$
(2.14)

 $S_{11}$  is the network input reflection coefficient whereas  $S_{21}$  is the gain/loss of the network. Similarly, the value of  $S_{12}$  and  $S_{22}$  can be obtained by setting the value of  $a_1$  into 0. The expression for both of these S-parameters are:

$$S_{12} = \frac{b_1}{a_2} \tag{2.15}$$

$$S_{22} = \frac{b_2}{a_2} \tag{2.16}$$

 $S_{22}$  is the network output reflection coefficient whereas  $S_{12}$  is the reverse gain/loss of the network.

There are 2 ways to present S-parameter magnitude, either in liner magnitude or logarithm based decibel (dB). Since S-parameter are complex voltage ratio, therefore they are converted to decibel by multiplying the log of linear ratio by 20.

 $S_{11}$  (dB) = input reflection gain =  $20 \log S_{11}$ 

 $S_{22}$  (dB) = output reflection gain = 20 log $S_{22}$ 

$$S_{21}$$
 (dB) = forward gain = 20 log $S_{21}$ 

$$S_{12}$$
 (dB) = reverse gain =  $20 \log S_{12}$ 

 $S_{11}$  and  $S_{22}$  are less than 1 and they are negative numbers in the case of passive networks with positive resistance.  $S_{21}$  and  $S_{12}$  are forward and reverse gain/loss when network is terminated with reference impedance.

VSWR can be related with S-parameters. For instance, output VSWR and  $S_{22}$  are related by the following equation:

$$VSWR = \frac{1+|S_{22}|}{1-|S_{22}|} \tag{2.17}$$

Similar situation occurs in the case of input VSWR and  $S_{11}$  with the same equation. On the other hand,  $S_{11}$  and  $S_{22}$  also related to input and output impedance respectively. Output impedance and  $S_{22}$  are related with it by following expression:

$$Z_{output} = Z_o \frac{1 + |S_{22}|}{1 - |S_{22}|} \tag{2.18}$$

Input impedance is similarly related to  $S_{11}$ .

#### 2.4 Broadband Impedance matching

Impedance matching is a design technique of matching the input impedance of electrical load or output impedance of signal source in order to maximize the power transfer from source to load, at the same time minimize the signal reflection from load. A diagram of load and source impedance circuit is shown in Figure 2.6.



Figure 2.6: Source and load impedance circuit

For complex load impedance  $Z_L$  and source impedance  $Z_s$ , the maximum power transfer is achieved when the following condition is met:

$$Z_s = Z_L * \tag{2.19}$$

The asterisk indicates the complex conjugate of the variable. On the other hand, the minimum reflection of the transmission line is achieved with the following condition:

$$Z_s = Z_L \tag{2.20}$$

As mentioned before, the whole point of carry out the impedance matching is to ensure the maximum transfer of power. The maximum power transfer theorem states that the impedance of the load should match the impedance of the source in order to transfer maximum power from source to load.

## Broadband matching problem

Designing an impedance matching equalizer network is an important part in most of the electronic devices such as amplifier, transmitter and other RF applications. The diagram of the basic network is shown in Figure 2.7. The aim of the equalizer is to transfer power from load to source by transform complex load impedance  $Z_L=R_L+jX_L$  to match source impedance in either resistive or complex impedance form (Zs=Rs+jXs) over the desired broad frequency band. In most of the case these impedances are measured at finite number of radio frequencies.



**Figure 2.7: Impedance matching equalizer network** 

Sinusoidal voltage source E at any particular frequency is applied to the lossless equalizer input port through Zs. When the input impedance  $Z_1=R_1+jX_1$  is equal to the conjugate of the impedance of the source  $Zs^*=Rs-jXs$ , the maximum available source power *PaS* is able to supply to the load Z<sub>L</sub>. Else, there is some power mismatch. This statement can be represented by the following equations:

$$G_T = \frac{P_L}{PaS} = 1 - MM^2$$
 (2.21)

$$MM = \left| \frac{Z_1 - Z_S^*}{Z_1 + Z_S} \right|$$
(2.22)

Where  $G_T$  = Transducer gain

 $P_L = Load power$ 

PaS= Source power

MM= power mismatch

Power mismatch can be express as return loss as shown in the formula below:

$$RL = -20\log(MM) \tag{2.23}$$

The aim is to determine an equalizer network that minimizes and reduces the mismatch, which results in maximizing the transducer gain  $G_T$  in equation 2.21. Conjugate matching is not physically practical for finite frequency band. (H. Carlin & Civalleri, 1998)

With the reference of Figure 2.7, real power is absorbed by load impedance  $Z_L$  is equal to the power entering a lossless passive network, which is the difference between reflected power and PaS namely  $|a1|^2 - |b1|^2 = |b2|^2 - |a2|^2$ . The power mismatch mentioned also can be represented by both of the equations:

$$MM = \left| \frac{Z_2 - Z_L^*}{Z_2 + Z_L} \right| \tag{2.24}$$

$$MM = \left| \frac{S_1 * - S_2}{1 - S_1 \times S_2} \right|$$
(2.25)

The generalized reflection coefficient in equation (2.22) and (2.24) at any given frequency are also equal to the hyperbolic distance metric in equation (2.25) associated with impedances Z1 and Z2 through reflection coefficients S1 and S2 in the following equation:

$$S_i = \frac{Z_i - 1}{Z_i + 1}, i = 1, 2$$
 (2.26)

Impedances are normally normalized to 1  $\Omega$  and frequency to 1 radian/second (r/s). Due to the fact that the inclusion of Xs or X<sub>L</sub> added to X1 or X2 respectively, reflection coefficients S1 and S2 in equation (2.26) is differ from equation (2.22) and (2.24).
## 2.5 Coplanar Waveguide with Ground (CPWG)

#### 2.5.1 Introduction of CPWG

Coplanar Waveguide (CPW) acts as an alternative to stripline and microstrip, where its structure consists of ground currents and signal currents where both of these are placed on the same layer. CPWG is selected in this project because it provides high frequency response and capable of wider effective bandwidth and impedance range. There are two ground planes on both sides of the CPWG. Two narrow gaps have been found in the device to separate the ground planes from the center strip that make up of the conductors. The attenuation of the line, characteristic impedance and effective dielectric constant are determined by the properties of the CPWG, which are thickness and permittivity of substrate, the gap and also the dimension of the center strip. A simplified diagram of CPWG is shown in Figure 3.4.



Figure 2.8: Simplified diagram of CPWG

## 2.5.2 Advantages of CPWG compared to microstrip

Transmission line is often used by designer in circuit design and also implementation on prototype level in order to achieve optimum performances. Microstrip and CPWG are the most common transmission line used by the designer in most circuitry design. In this section the strength of CPWG is discussed and comparison is made with microstrip. CPWG circuits are able to adapt wider effective bandwidth due to the fact that they have enhanced ground structures. With this property, CPWG has also a wider range of impedances compared to microstrip. However, due to the fact that the straight forward structure of microstrip, it is relatively easier to fabricate microstrip rather than CPWG on PCB. On the other hand, CPWG circuits are able to operate at higher frequencies with lower loses compared to microstrip. For example, with thick copper conductor, CPWG may have a greater impact which will result in increasing EM fields between the ground-signal-ground structures on PCB. Therefore, the effective dielectric constant will be reduce and leading towards the decreased of conductor loss.

## 2.6 Real Frequency Technique

#### 2.6.1 Background

Design engineers play a vital role to design the electrical circuit with best performance possible, regardless of how well the circuit production technology. In communication system design, the most crucial issue is to design a system with error free transmission and reception over desired frequency bandwidth. With proper designed antenna and its matching network, the signal will be transmitted without any hassle, although it may encounter some loss during the transmission. Hence, with a certain understanding of design problems of the matching network construction, we are aim to construct an antenna matching network with high gain and frequency bandwidth as wide as possible.

Therefore, the designers have faced broadband matching problem and this particular problem is the main issue in the existing literature of antenna design at both receiver and transmitter side. Researchers have been trying to find solutions to solve the gainbandwidth limits of the designed system. The analytic theory behind of this design problem is difficult to comprehend and hard to access beyond simple problems.(Fano, 1950; Kuh & Patterson, 1961) In some circumstances, solution may not be obtained by using analytic theory. Even with the existing solution it may result in complex circuit structure and sub-optimal performances. (H. J. Carlin, 1977) Hence, computer-aided adhoc or brute force design techniques are the favorite methods for design engineers. With this method, first a circuit topology is chosen for matching network to be design regardless of what gain bandwidth limits of the device to be matched. Next, the elements value of the circuit is determined to optimize the performance of the designed circuit. However, when deal with complicated circumstances, using of ad-hoc design techniques will compromise the performance of the designed circuit although the use of outstanding manufacturing technology. This means waste of the designer's valuable time and also money.

A new vision to the broadband matching problem has been introduced and this methodology is called "Real Frequency Technique (RFT)". (H. J. Carlin, 1977) With this technique, the analytic theory can be by-passed and provides a reasonable estimation for gain bandwidth limits of the devices to be matched. The lossless matching network constructed with this technique automatically achieved almost optimum gain-bandwidth performance. The initial version of RFT is also called "real frequency-line segment technique" was good enough to solve only single stage matching problems and also design single stage microwave amplifiers. After a period of time, a solution has been developed to handle any kind of matching network and microwave amplifier design problem with scattering approach called "Simplified Real Frequency Technique (SRFT)". (H. J. Carlin & Civalleri, 1985) As a result, SRFT is suitable to design microwave circuit and it is very practical and straight forward to be implemented. (B. Yarman, 1982) By using this particular technique, matching networks have been designed for several microwave amplifiers for satellite transponders and antenna arrays.(Binboga Siddik Yarman & Carlin, 1982)

RFT has been widely used in the field of microwave engineering and the usage of this technique has been extended to construct matching networks and amplifier in two-kinds of elements, which are lumped and distributed elements. (Sertbas & Yarman, 2004) RFT also has been employed to model measured data obtained from active and passive device. (B Siddik Yarman, Aksen, & Kilinç, 2001) Several multiband antenna matching and switching networks for cellular communication systems are designed and completed by employing RFT. (Lindberg et al., 2006)

## 2.6.2 Theory of Real Frequency Technique

In principle, RFT is a lossless equalizer is simply described in terms of its unit normalized scattering parameter. (Aridas, Yarman, & Chacko, 2014; Kılınç, Köprü, Aksen, & Yarman, 2014) Scattering parameter is the method used in computational for the blocks to be matched, results in numerically well behaved gain optimization of the system. Therefore, RFT is a favorite method to design microwave wideband amplifiers due to its simplicity and faster compared to other existing algorithm. (Binboga Siddik Yarman, 2010) Fig.1 shows the lossless matching circuits in terms of Darlington's driving point impedance.



# Figure 2.9: Lossless matching network in terms of Darlington's driving point impedance.

The lossless matching network is described by the Darlington's driving point impedance. It is represented by the following expression:

$$Z_B(\rho) = \frac{N(\rho)}{D(\rho)} \tag{2.27}$$

Where  $\rho$  is a complex variable represented as  $\rho = \sigma + j\omega$ . (Binboga Siddik Yarman, 2010)

In any general case, the positive real impedance is represented as

$$Z_B = Z_M(\rho) + Z_F(\rho) \tag{2.28}$$

In this equation,  $Z_M$  is a minimum reactance function which is free of Right Half Plane (RHP) and  $j\omega$  poles. (Narendra Kumar, Prakash, Grebennikov, & Mediano, 2008)It is represented by the following expression:

$$Z_M(j\omega) = R_M(\omega^2) + jX_M(\omega)$$
(2.29)

 $Z_F$  is the Foster function and it is represented as:

$$Z_M(j\omega) = jX_F \tag{2.30}$$

$$X_F = \frac{1}{C_{n+1} \times \omega} \tag{2.31}$$

In the expression in (2.31),  $C_{n+1}$  is the term that represent the DC blocking capacitor.

The minimum reactance impedance function can be represented in mathematical form, as shown in equation (2.29). The term  $R_M(\omega)$  is a non-negative even function in angular frequency  $\omega$ . When it is terminated in a resistance, it represents the lossless lumped element network. The general form of  $R_M(\omega)$  is shown as follows:

$$R_{\omega^{2}} = \frac{A_{1}\omega^{2} + A_{2}\omega^{2(n-1)} + \dots + A_{n}\omega^{2} + A_{n+1}}{B_{1}\omega^{2} + B_{2}\omega^{2(n-1)} + \dots + B_{n}\omega^{2} + 1} \ge 0; \ \forall \omega$$
(2.32)

The above equation represents a highly complicated circuit topology and the parameter

*Ai;* i=1,2,...(n+1), the numerator of the coefficient is the important factor that influenced of circuit structure. For example, if  $A_0=0$ , at least one transmission zero at DC would be occurred in the circuit. The simpler form of the equation is obtained for producing an "*n*" element *LC* lowpass ladder circuit topology terminated at  $Z_M(0) = R_M(0) = R_L$  as desired. The equation is shown below:

$$R_{\omega^{2}} = \frac{R_{L}}{B_{1}\omega^{2} + B_{2}\omega^{2(n-1)} + \dots + B_{n}\omega^{2} + 1} \ge 0; \ \forall \omega$$
(2.33)

It will eventually come out with the schematic of the matching network and also a graph of transducer power gain (TPG) against the desired bandwidth. The unknown coefficient of the matching network in (2.33) are determined by the optimization of TPG. (Aridas et al., 2014) In terms of theory, the TPG of the circuit can be expressed in mathematical form as shown below:

$$T(\omega) = \frac{4R_M R_L}{(R_M + R_L)^2 + (X_F + X_M + X_L)^2}$$
(2.34)

In an ideal case,  $T(\omega) = 1$  over a specified angular frequency band  $B(\omega) = \omega_2 - \omega_1$ , otherwise it is zero. Over the band of operation  $\omega_1 \le \omega \le \omega_2$ , error function is defined as:

$$\varepsilon(\omega) = (R_M + R_L)^2 + (X_F + X_M + X_L)^2$$
(2.35)

The particular point of optimization is to determine  $R_M(\omega)$  and  $X_F(\omega)$  such that the error function  $\varepsilon(\omega)$  is minimized within the band we are interested in, which is clearly a non-linear optimization problem. The success of the non-linear optimization depends on the degree of non-linearity of error function. If the error function is quadratic in terms of the unknowns, it is possible to achieve global minimum; or else the solution may be jeopardized.

TPG graph provides an indication to the user on how well the performance of the matching network. It is the best that to achieve TPG as low as possible and within the range of 0 and -1 dB. (Binboga Siddik Yarman, 2008). For instance, for GaN HEMT

device (CGH40010F), source and load pull measurement are performed to identify the optimum impedances over the entire bandwidth operation, which are represented by the term  $Z_{in}$  and  $Z_{out}$  in Fig.1. Thus, the initial values of the matching elements presented through this method is optimized where necessary in order to obtain a better performances, refer to Fig. 1 to see the IMN circuit. The series DC capacitor is included to block the DC to disturb RF signal.

RFT is commonly employed by designer to design and construct PA with desired performances. The reason of using RFT to obtain the matching networks are there is no need to determine any form of equivalent circuit or analytical impedance function for the antennas and generator. Also, algebraic transfer function as a measure of performance and the assumption of a topology for the matching networks in advance is avoided. Therefore, the analytic procedure can be bypass to reduce the complexity of the matching network construction. While other method such as Smith chart is available for matching network construction, RFT is still a favorable method compared to Smith chart, particularly for broadband network design due to the fact that Smith chart does not optimized well as compared to RFT for wideband network solution, and the complexity of RFT is lesser compared to Smith chart method in terms of matching network construction.

#### 2.7 Summary

In this chapter, we have done the review of few aspect of the power amplifier. In the first part, basic power amplifier theory has been introduced. Also we have discussed the power amplification classification, which is transconductance power amplifier, which based on the conduction angel of drain current, and also switch mode PA, which the transistor operates as a switch rather than current source. The example of transconductance PA is class A, class B, class AB and class C power amplifier whereas class D, class E, class F are categorized as switch-mode power amplifier. Switch mode

PA has higher efficiency compared to tranconductance PA, however it is not suitable for wideband design due to the need of resonators at the output stage to tune to fundamental frequency.

In the second part of this chapter, power amplifier figure-of-merits are discussed. The parameters discussed in this section can be used as an indicator to observe how well the designed power amplifier perform in prototype level, eventually to the industrial aspect. The parameters studied in this section are power added efficiency (PAE), output power, and gain, which are also the core parameters studied in this work. The theory for S-parameter and broadband impedance matching are discussed in this chapter.

Lastly, an impedance matching technique called "Real Frequency Technique" is discussed. A brief background on the development of Real frequency Technique has been investigated. Invented by Professor Carlin in order to solve broadband matching problems, this technique has improved all this year with the collaboration with Professor Yarman. This technique, at first which can only solve single impedance matching problem, improved towards a stage where it can handle all kinds of problems regarding matching network. The theory of the real frequency technique is discussed.

The literature review in this chapter allow us to analyzed in depth various works and understand different methodology to design wideband PA with desired specification. Several issues arise on the methodology to design wideband PA with simplest network and enhanced performances. By implementing the knowledge from this chapter, the methodology of designing a wideband PA with desired frequency can be proposed in order to develop PA design with enhanced performances and in simplest circuit form factor. This is important from industrial point of view as it will able to develop a new product with reduced cost and compact size, together with more integrated functions.

## **CHAPTER 3: METHODOLOGY**

#### **3.1** Design of matching network

#### 3.1.1 Initial Guess Technique

In this project, RFT is employed in order to obtain the IMN and OMN of the power amplifier. With the aid of MATLAB software, the algorithm of RFT can be executed and the structure of the matching networks with values for each of the elements are presented. Also, a graph of transducer power gain (TPG) is shown together with the schematic structure. TPG is an indicator for the designer to check on the performance validity of the matching network design. Due to the fact that RFT provides an initial idea of the structure of the matching network to designer, this technique is also called initial guess technique, where the structure, elements, and values for each matching network are produced base on the optimization algorithm of the RFT. However, the values that presented by this technique are not finalize. Optimization is necessary on the final level in order to achieve satisfactory performances.

### 3.1.2 Initial Guess Simulation using MATLAB

The execution of RFT algorithm in MATLAB is performed by entering the desired value for each parameter. User will be prompt by the command window for parameter insertion, which will result in the execution and optimization process by the algorithm. Eventually, the structure of the matching network and the graph of TPG will be shown. The command window of the MATLAB program is shown in Figure 3.1 below.

C	ommand Window	O	
	Normalized freq in Mhz 2300	^	
	f0 =		
	2300		
	Enter desired flat gain level IO(db)= 0.8		
	10_db =		
	0. 8000		
	ntr=0 No transformer ntr=1 With transformer 0		
	ntr =		
f	¥	_	
C	ommand Window	0	
	0		
	WNumer of transmission zeros at DC: k=0 Lowpass Ladder Structure 0		
	k -		
	0	- Í	
	<pre>%Initial value for h=[hn, hn-1, hn-2h0] [1 1 1 1 1 1]</pre>		
	h -		
	1 1 1 1 1 1 1		
f	X Enter ws1(Mhz) Lower stopband =500		
C	ommand Window	0	
	Enter ws1(Mhz) Lower stopband =500	í	
	wlows1 =		
	500		
	Enter ws2(MHZ) Upper Stopband=2300		
	whighs2 =		
	2300		
	Enter wc1(Mhz) Lower Passband=500		
	wlow1 =		
ј» С	ommand Window	C	
	500		
	Enter wc2(Mhz) Upper Passband=2300		
	whigh1 =		
	2300		
	# of sample points in the optimization 80		
	N opt =		
	80		
1		- 1	
£	r telanno tanà imandrana in inandrana faranta talang tanà ina dara dia 201 ferrata di		

Figure 3.1: Command window for parameter insertion

In Table 3.1, each of the command prompt for parameter input will be explained briefly.

fo	Normalized frequency in MHz.
$T_o(\mathrm{dB})$	Targeted flat gain level
ntr	Control flag which is set according to
	designer's desire. When ntr=1, ideal
	transformer would be included into the
	design; for <i>ntr</i> =0, designer must work
	with low-pass prototype with no ideal
( )	transformer
k	Order of transmission zeros in DC
$h = [h_1, h_2, h_3, \dots, h_o]$	Input of coefficients for $h$ to determine
	the elements of matching circuits. An ad
5	hoc direct choice for the coefficients (h=
	$\pm 1$ ) would provide satisfactory
	initialization to start RFT algorithm.
	(Binboga Siddik Yarman, 2008)
wlow	Low end of optimization frequency
	(normalized respect to selected
	frequency)
whigh	High end of optimization frequency
	(normalized respect to selected
	frequency)
# N_opt	Sample points of optimization

**Table 3.1: Input Description** 

The algorithm of RFT will be assigned to extract the input and output impedances data from a file. The optimum impedances are obtained from load and source pull measurement respectively. Input and output impedance of the PA depends on the active device that used, in this case, GaN HEMT. In each frequency in the desired bandwidth, the value for the input and output PA impedances are different. The data of impedances in this work are obtained from Motorola Solutions Penang, with the guidance and help of the engineers. The graph of the TPG is the main reference of the performance of the matching network. The TPG of the matching network obtained is preferable to be as flat as possible. Furthermore, it is advisable that the graph obtain should be in the range of 0 to -1. (Aridas et al., 2014). The input coefficient of h is the coefficient that based on the ad hoc choices  $(\pm 1)$ . The number of elements (inductor and capacitor) presented are based on the number of coefficient, e.g. [1 1 1 1 1] represents 4 elements and the last value represents the termination of 50  $\Omega$ . Another indication that the designer should take note is the value of the elements obtained from the optimization algorithm. The value of the elements have a great effect on the performance of the matching network, and also the implementation on the printed circuit board (PCB) level. For example, if the value obtained for the inductors are too big, the performance of the overall design will be affected when both of the matching network are combined together with transistor. Furthermore, implementation on printed board circuit (PCB) level will face difficulties and in terms of cost production. The higher the value of the components, the more expensive the production.

When the program is executed, the user will be prompt by the command window to enter the desired value for each of the parameter. The algorithm of RFT will be executed according to the parameters that inserted in the command. As the program goes on, the PA impedances, either source or load impedances will be extracted and in cooperate with the PA design, and the optimization process will be executed by the algorithm.

34

Eventually, a schematic diagram will be presented with elements and its suggested values, and a TPG graph against frequency will also be presented.

An example of schematic diagram is shown in Figure 3.2. From the diagram, the component of the matching network, which are inductors and capacitors are presented. The value for each inductor and capacitor are shown below the schematic diagram. The value of component are calculated based on the optimization algorithm of RFT.



Figure 3.2: Example of matching network obtained from execution of RFT algorithm

At the same time, a graph of TPG against frequency is shown. An example of TPG graph obtained from the simulation is shown in Figure 3.3. As mentioned before, the graph is preferable to be as flat as possible and the graph should be better in the range of -1 and 0.



Figure 3.3: TPG against frequency for matching network obtained from RFT algorithm

## **3.2** Mixed-lumped and distributed elements implementation

## 3.2.1 Distributed and Lumped elements

The design of integrated circuit, particularly power amplifier design is generally based on distributed elements, lumped elements, or combination of both type elements. Distributed elements are normally represented by any types of transmission line with different characteristic impedance, length and type. Generally, lumped element are represented by capacitor and inductor. Both of these lumped elements are usually small in size when compared to transmission-line wavelength  $\lambda$ , which their linear dimension are normally  $\lambda/6$  or  $\lambda/16$  smaller. (N. Kumar & Grebennikov, 2015) As mentioned before, the size of the lumped elements are very small, therefore this characteristics gives them an advantage towards other components. However, when compared with distributed elements, lumped elements lack of quality factor and power-handling capability.

#### **3.2.2** Mixed Lumped and distributed elements matching network

Mixed lumped and transmission-line elements are generally implemented in matching network in most of the hybrids and monolithic design techniques. It is very convenient when designing push-pull PA with effect grounding, which the capacitor is in shunt configuration in between a transmission lines, in this case, a coplanar waveguide with ground (CPWG). (Andrei Grebennikov, 2005)There are four basic steps in designing a mixed lumped and distributed elements matching networks:

- 1. Choosing appropriate lumped element schematic which achieving nearmaximum gain over the desired bandwidth
- 2. Breakdown of the whole matching network into subsections
- 3. Conversion into transmission line with equivalent value
- Optimization technique to overcome power variation within desired frequency range. (Binboga Siddik Yarman & Aksen, 1992)

Normally under the low frequencies, the lumped elements or mixed lumped matching networks are working well which they achieved the required performances. However, it is extremely difficult for lumped components to be implemented at microwave frequencies where they can be treated as distributed elements. This situation is more evident for inductor, due to the fact it has low quality factor which will result in additional loss. (Andrei Grebennikov, 2005)

In the case of implementation of distributed elements in the matching network, most of the designs are based on the approximation equivalent value of lumped and distributed elements with the reference and application by Richard transformation. (Richards, 1948) This implies that transformation can be carried out for lumped element to distributed elements with equal-length open-circuited or short-circuited transmission lines. The mathematical representation of this transformation is shown below:

$$s = jtan\frac{\pi\omega}{2\omega_o} \tag{3.1}$$

Where  $s=j\omega/\omega_c$  is the conventional normalized complex frequency variable and  $w_o$  is the radian frequency. Therefore, the reactive impedance of lumped inductor  $Z_L$  can be represented by the following expression which is corresponded to the one-port impedance of a short-circuited transmission line.

$$Z_L = sL = j\omega L = jLtan \frac{\pi\omega}{2\omega_0}$$
(3.2)

The equation (3.2) shows that it is possible that an inductor can be represented by a transmission line with characteristic impedance of  $Z_o=L$  and electrical length  $\theta = \pi \omega/2\omega_o$ .

In this work, the IMN and OMN of the power amplifier design is obtained from initial guess technique. The matching network is than realized with mixed-lumped design. In specific, the inductor is converted into distributed elements, which is CPWG as a transmission line, and high-Q discrete capacitor is used to represent capacitor in shunt configuration. In this case, ATC 100A series is selected as model to be implemented on the porotype level.

#### 3.2.3 CPWG Implementation

Basically, a transmission line involves inductance associate with the flow of current in the conductor and capacitance association with the strip separated from the ground by the dielectric substrate. These properties contribute to the "distributed" term of the model of L-C circuit with the inductor and capacitor. A narrow, high-impedance line with impedances less than 90° electrical length will behave like an inductor. (Rhea, 2006) Analytically, the transmission line conversion can be examined from the impedance point of view. A transmission line with characteristic impedance  $Z_o$  and electrical length  $\theta$ , the

impedance  $Z_s$  at the input of the transmission line can be represented by the expression below with load termination  $Z_L$ :

$$Z_s = Z_o \frac{Z_L - jZ_o tan\theta}{Z_L + jZ_L tan\theta}$$
(3.3)

As mentioned before in previous section, it is possible to represent transmission line as inductor in a circuit design. Consider the case which  $Z_L$  is short-circuited, 0 + j0. The expression of (3.3) can be simplified to the following:

$$Z_s = jZ_o \tan\theta \tag{3.4}$$

The input impedance of the short inductor is  $jX_L$ , therefore

$$X_L = Z_o \tan \theta \tag{3.5}$$

From equation (3.5), once again proved that the lumped element can be represented by distributed elements, in specific transmission line with characteristic impedance of  $Z_o$  and electrical length  $\theta$ . With increased frequency, the reactance of the inductor also increased linearly. Equation (3.5) shows the relationship between lumped element (inductor) and distributed element. Generally, it is preferable that the equivalent distributed element has higher impedance and shorter electrical length. (Rhea, 2006)

After the computation of  $Z_o$  and  $\theta$ , the parameter of length (*l*) and width (*w*) are synthesized with the aid of LineCalc tool from ADS. Note that the gap of the CPWG (*g*) is set to 0.9mm with reference of the spec sheet of ATC 100 model capacitor, due to the fact that capacitor is placed in between the gap of the CPWG on the PCB level. The LineCalc tool interface is shown in Figure 3.4.



Figure 3.4: LineCalc tool interface for the synthesis of *l* and *w* 

Next, after obtained the parameter value of l and w, a verification method is carried out in order to show that the CPWG obtained is equivalent to the inductance that obtained from RFT simulation. Therefore, S-parameter conversion of 2-port to 1-port is carried out. The significance of this method is to make good optimization or tuning of the overall matching networks (with mixed lumped elements) from effective individual series inductor that converted.

A circuit of S-parameter conversion of 2-port to 1-port is constructed in ADS and the schematic circuit is shown in Figure 3.5. The inductor and CPWG are connected in parallel configuration with termination of 50  $\Omega$  at both of the end of the circuit.



Figure 3.5: Schematic of S-parameter conversion of 2-port to 1-port

Therefore, the parameter is defined in ADS, for example  $S_{11}=S_{11}(2p)$ ,  $S_{12}=S_{12}(2p)$ and so on where 2p is used to defined the respective terms in 2-port system. The equation of converting 2-port system to single port system is:

$$S_{11}(1p) = S_{11}(2p) - S_{12}(2p) * \frac{S_{21}(2p)}{(1+S_{22}(2p))}$$
(3.6)

Where  $S_{11}(1p) = S_{11}$  in single port device

 $S_{11}(2p) = S_{11}$  in 2-port device

 $S_{12}(2p) = S_{12}$  in 2-port device

 $S_{21}(2p) = S_{21}$  in 2-port device

$$S_{22}(2p) = S_{22}$$
 in 2-port device

As a result, the effective impedance of the element from two-port can be determined from the as shown in following equation:

Impedance 
$$(1p) = \frac{50*(1+S_{11}(1p))}{(1-S_{11}(1p))}$$
 (3.7)

and the impedance which leads towards the equation of inductor for single-port.

$$L(1p) = imag \frac{Impedance(1p)}{2\pi f}$$
(3.8)

#### **3.2.4** Capacitor implementation

In this section, the implementation of high-Q discrete capacitor on simulation level will be discussed. As mentioned before mixed-limped design is used to realize the circuit of the PA designed. Inductor in this work has been represented by distributed element, which is CPWG transmission line; on the other hand lumped element is represented by capacitor, which is available from vendor. The capacitance obtained from RFT algorithm simulation is ideal, which lack of the consideration of loss and parasitic from they are produced with technology under construction.

In this project, ATC 100A series capacitor from ATC Company has been chosen to be implemented on PCB prototype. The reason of choosing this model of capacitor to be used in this project is ATC 100A series capacitor is considered one of the most versatile high-Q, high self-resonant multilayer capacitor.

The value of capacitor needed to be given in order to carry out the simulation of PA, eventually to the porotype production process. The value of the capacitor is obtain through the initial guess technique, however on the simulation level, there are only a certain values of ATC 100A series capacitors are available for simulation on ADS from ATC Company. Therefore, a simple circuit is construct on ADS to compare the ATC capacitor available to the capacitor value simulated from initial guess technique. The significant of this method is to provide a good tuning and optimization for matching network from individual capacitor. The circuit constructed are shown in Figure 3.6. From the schematic, it is observed that two circuits are constructed in order to compare and optimize the simulated capacitor and ATC capacitor. Both of the circuits are terminated

with 50  $\Omega$  termination. For the first circuit, the capacitor value is obtained from the simulation from initial guess technique; on the other hand, the second circuit is connect with a component named "2-port S-parameter", which is used to extract the simulation data of available ATC capacitor,



Figure 3.6: Schematic circuit for capacitor simulation

An example of simulation result of graph of capacitance against frequency is shown in Figure 3.7. The capacitance of ATC capacitor sweep across the desired frequency is in blue graph.



Figure 3.7: Simulation result of the capacitors

## **3.3 DC Biasing Circuit**

In most of the electronic components such as transistors, diode and many others, where the alternating current (AC) is present in the operation of the circuit involved with these devices, biasing is a necessary procedure to determine a fixed DC current and voltage that applied to the terminal of the circuit. In an electronic amplifier, a bias voltage is applied to the transistor in order to allow it to operate at a certain region of transconductance curve.

It is necessary to design a biasing circuit carefully in order to provide optimum operation over the frequency response. A basic structure of dc biasing circuit is shown in Figure 3.8.



Figure 3.8: Basic DC biasing circuit

A dc feeding network at drain region together with  $L_d$  and  $C_d$  with the recommend value of 180nH and 33pF respectively. (Krishnamurthy et al., 2000) The selection of value for these components must fulfil the condition which they must provide high impedance over the desired frequency response, so that the AC would not pass through the DC feeding network to mix with DC signal. In this work a ceramic 4310LC model RF choke from Coilcraft Inc. is chosen for feeding network, together with three ATC 100A series capacitors with the value of 100nF, 33pF and 10 $\mu$ F respectively. The RF choke is selected base on its self-resonant frequency, where it is near to the operating frequency. Inductor tends to become a high impedance element at high frequency. The reason of using multiple capacitors is to cater the broadband frequency range of 80 MHz up to 3000 MHz

In the case of DC biasing network, the selection of  $L_g$  and  $C_g$  must be able to provide stable quiescent point for the consistent operation on the bias region. Therefore, a high-Q chip inductor of 220nH is selected along with three ATC 100A series capacitors with the value of 100nF, 33pF and 10µF respectively. The reason is the same as feeding network, which is to cater the broadband frequency response. It is observed that the capacitors are placed at the input and output side of the RF signal. It is the coupling capacitor which provides blocking purpose to prevent the DC from flowing into the RF path. The value selection of the coupling capacitor must incorporate with the desired frequency bandwidth in order to operate smoothly. In this case, 120pF ATC 100A series is selected as the value of coupling capacitor.

In order to protect the circuit from any possible damage, the dc turn-on sequence is necessary to be complied with. A negative bias voltage is applied to the biasing circuit  $(V_{gate} < 0)$  first before drain voltage  $(V_{drain} < 0)$  and it is vice versa for the case during turning off. Drain voltage should be turned to 0V first followed by gate voltage.

A circuit is constructed in ADS in order to determine the bias point of GaN HEMT for Class AB operation. The circuit constructed is shown in Figure 3.9.



Figure 3.9: Basic DC biasing circuit

A graph of current against voltage of the GaN transistor is obtained and it is shown in Figure 3.10. With regard to class AB biasing, gate voltage Vgs of -2.8 V is found as an appropriate point to bias the transistor to the mentioned class of operation. The drain current corresponds to the selected Vgs is 0.154 A.



Figure 3.10: Graph of current against voltage for GaN biasing

#### **3.4** Advance Design System (ADS) Simulation

After the matching networks are obtained, the design is than transfer into Advance Design System (ADS) software to perform simulation. ADS has been widely used in industry and academia to design and simulate RF electronic circuit due to the fact that it allows designer fully characterize and optimize RF circuit conveniently. In this work, frequency-domain circuit simulation is the feature that mostly used in ADS system.

## 3.4.1 Simulation of Matching Network with Mixed-Lumped Design

After the matching networks are obtain from the initial guess technique, the data obtained is transferred to ADS for simulation procedure. Instead of using inductor in the ADS, CPWG and ATC 100A series capacitor is directly applied in this simulation. An example of construction of the matching network is shown in Figure 3.11.



Figure 3.11: Construction of IMN with mixed-lump design

The component that represent capacitor in this construction is 2-port S-parameter, which can be used to extract the data of ATC 100A series capacitor. The simulation results are observed and discussed in next chapter. The TPG graph obtained is optimized by adjusting the either the value of width (w) or length (l) of the CPWG. Occasionally, the value of ATC capacitor is substitute with suitable value is necessary.

## 3.4.2 3.5.2 Simulation of Complete PA Circuit

After the matching networks are optimized and the TPG graph achieved a satisfactory level, the next procedure is to simulate the complete PA circuit. In detail, the IMN and OMN is connected with GaN transistor, along with DC biasing circuit. The PA circuit is simulated with drain voltage of 28 V and gate voltage of -2.8 V, according to the transcondcutance characteristic curve of GaN that has been investigated in previous section. The schematic diagram of full PA design in ADS is shown in Figure 3.12.



Figure 3.12: PA circuit construction in ADS

By simulating the full PA circuit, the performances of the designed PA can be observed and analysis. PAE, output power and gain are the parameters that we put much attention to in this work.

## 3.5 PCB layout Design

The simulation result is carried out in order to observe the performances of the designed PA. Next, the PCB layout is design for the prototype fabrication. In this section, the method of PCB layout design will be presented in details. The software used in the design of the layout is OrCAD, a PCB design product developed by Cadence. OrCAD software is a powerful software mainly used in electronic design automation. It is been used by most of the engineers to design circuit schematics and also electronic prints for manufacturing PCB. In this work, there are two tools under software that will be used to design the PCB layout, which are CIS Capture tool and Layout tool.

## 3.5.1 Layout Construction using OrCAD CIS Capture Software

First, the circuit of the designed PA is constructed using a software called CIS capture, a tool under OrCAD software. The circuit of the PA design needs to be constructed in this tool first instead of directly design the layout of the PCB is due to the fact that it is much convenient to optimize the circuit in this tool, especial for the routing of one component to another. The schematic constructed using CIS capture is shown in Figure 3.13.



Figure 3.13: Schematic circuit construction on CIS Capture tool

The schematic circuit construct in CIS Capture tool is almost similar with the schematic in the ADS. However, the main purpose of the construction of circuit in this tool is to transfer the design into the Layout tool, with the appropriate connection between components and also routing. The reason of not directly design the layout in the Layout tool is the complexity of routing and connection process between connections in that particular software. It is possible but not advisable. Hence using CIS Capture would greatly reduce the complexity to design the layout. In this tool, the number of pin of the component and pin-to-pin connection of the component are stressed throughout the construction. Note that the component used to represent transistor in the figure. A component with two pins is used to represent GaN transistor instead of component with three pins, this is because the routing of the component will be complex even though the third pin is connected to ground. Therefore in order to simplify the work, the component with two pins is used to represent transistor.

## **3.5.2** Footprint construction for Each Component

After the construction of schematic circuit, the next step is to provide the characteristic of each component. Therefore, footprint need to be inserted into component by using the property command of the system. By insertion of footprint into the component, the parameter of the component such as width, length, gap and others are able to be realized with exact value in the layout. However, the footprints of the components used in this work are not available in the existing library. Hence, these footprints needed to be created manually.

First of all, the parameters of each and every component needed to be verified. In order to obtain these information, spec sheet of each component is used as reference. The height, width, gap between two plates are the common parameters for each component. Layout tool is used to create the footprint of component. The "Padstack" command on the Layout tool is used to characterize the shape, length and width of the footprint. The name of the footprint created is defined, than the shape of the component for this particular footprint is identified. In this work, a two-layer Roger 4350B PCB is used as the prototype for fabrication, so the dimension of the top and bottom layer are the main concern. The layer for Top, SMTop, SPTop and SSTop are referred to top layer; Bottom, SMBot, SPBot and SSBot are refereed to bottom layer. For component that required to present at both top and bottom layer such as via-holes, SMT connectors,  $V_{dc}$  and  $V_{ga}$  connectors, all of the mentioned layer needed to be defined. Else, only top layers needed to be defined. Hence, a footprint is created and can be used to insert into the property of component CIS Capture tool.

## 3.5.3 Layout Design

After schematic construction of PA design in CIS Capture, the schematic design is than transfer into Layout tool. In this part of the work, the components and routing are already completed by generate the file from CIS Capture. Components of the PA needed to be arranged accordingly with the reference of the design completed previously. The length (l) and width (w) of the CPWG transmission line are adjusted according to the calculation value previously. The thickness of the DC routing should be carefully selected so that the appropriate amount of dc is able to flow to the transistor. The layout design is shown is Figure 3.14.



Figure 3.14: PCB layout of the PA design

The area of blue color indicates ground layer whereas the area of red represents empty layer, meaning that neither transmission nor ground present in this area. There are six big circle in the middle of the layout design which represent screw hole for heat sink. The diameter of the outer radius of screw hole is 9.5mm whereas the inner radius is 5mm diameter. The inner radius of the screw hole is represented by brown color, which is the drill hole. Also, the small circle scattered on the PCB layout are via-holes. The outer radius of the via-hole is 5mm whereas the drill hole of the via-hole is 5mm whereas the drill hole of the via-hole is measured 3mm. The reason of including via-holes is to enhance the grounding effect of the system. The layout design of PCB is completed and is ready for fabrication.

## **3.6 Heat Sink Design**

A heat sink is used to attach under the PCB in order to allow maximum heat dissipation. In this work, a standard aluminum heat sink is used. Normally the material of the heat sink is copper, which has better conductivity than aluminum. However, copper is much more dense and also expensive compared to aluminum. Therefore aluminum is an obvious choice for this particular work. The heat sink in this work has a dimension of 108.5mm x 74mm with the thickness of 25mm. The layout design of the heat sink top view and side view is shown in Figure 3.15and Figure 3.16 respectively.



Figure 3.16: Heat Sink Layout Side View

The alignment of screw hole for heat sink and also GaN transistor should be accurate so that the component can be screwed on the PCB. For example, the distance between screw holes for transistor is measured 9.45mm according to the measurement provided by spec sheet. The thickness of the fin on the heat sink is measure 2mm and the distance between each fin is measured 4mm. A small cavity measured with depth of 1.7mm is designed in the area where GaN transistor is placed. This is due to a thick gold layer of the GaN transistor which is thicker than the PCB. Without the cavity, the contact of the surface of PCB and the heat sink will be greatly reduced.

## **3.7 PCB Fabrication**

After completing the layout design, the Gerber file of the design is generated and sent to manufacturer for fabrication. The company appointed for this task is Brusia Engineering Sdn. Bhd. The materials for the PCB production is Roger 4350B, and the properties of this material are presented in Table 3.2.

Value
3.66
0.508 mm
4.1 x 10 <sup>7</sup> S/m
1
0.0034

 Table 3.2: Properties of Rogers's 4350B PCB material

Note that the information provided in Table 3.2 is also crucial for LineCalc tool for generation of w and l of CPWG transmission line, as shown in Figure 3.5. The PCB fabricated is shown in Figure 3.17.



Figure 3.17: PCB fabrication by Brusia Engineering Sdn Bhd

Components such as capacitor, RF choke, and chip inductor are ready to be soldered onto the PCB. GaN transistor should be handle with care by not touching it with bare hands, as the electrostatic discharge (ESD) will degrade the quality performance of the transistor. Heat sink is mounted under the PCB in order to provide maximum heat dissipation. A cavity is available at the center of the heat sink for the placement of GaN transistor. To ensure maximum contact between transistor and heat sink, a few pieces of indium foil are placed between the active device and heat sink to enhance the contact, and also to provide better heat dissipation. The complete PCB with complete components with mounted heat sink is shown in Figure 3.18 (top view) and Figure 3.19 (side view).



Figure 3.18: Complete PA PCB (top view)



Figure 3.19: Complete PA PCB with heat sink (side view)

The PCB fabrication is competed with soldering of all components on the board and also heat sink mounting under the board. The complete GaN PA board is ready for testing.

## 3.8 Measurement Setup

The equipment is setup in order to obtain measurement results. The experiment equipment used are Rohde and Schwarz SMA 100A signal generator, Rohde and Schwarz FSG spectrum analyzer, GW Instek GPS-4303 power supply, attenuator, and cables.

The cable used in the experiment setup is N-type cable. Attenuator is used in this setup as a protective measure to avoid high RF signal entering to vector network analyzer that would eventually damage the equipment. Signal generator provides the input signal power to the device under test, which is the PA. Power supply is used to provide DC to the device. Precaution should be noted when operating the power supply to the device. Gate voltage should be turn on first before drain voltage, and vice versa when the device is turn-off to avoid current overflow to the circuit. The diagram of experiment setup is shown in Figure 3.20.



## Figure 3.20: Experiment setup for data measurement

Measurement data is obtained and analysis of the performances has been made for the designed PA. The measurement data is also compared with the simulation result.
#### 3.9 Summary

In this chapter, the methodology of wideband PA design is discussed. The matching network of the PA is obtained through real frequency technique. This technique provides us the initial structure of the matching networks, together with the suggested value for each elements by executing the optimization algorithm. Therefore it is also called initial guess technique.

The matching networks are realized with mixed-lumped designed, in other word the inductor is represented by series distributed element which is transmission line CPWG and high-Q capacitor is selected for the realization of circuits on PCB level. The reason of this method is used because it is more efficient in terms of prototype fabrication and also the performance of the designed device can be enhanced. The definition of lumped element and distributed element are discussed in this section. Method of conversion of inductor to CPWG transmission line is also presented in this section, along with the method to verify the equivalent of CPWG to the inductor. The approach of selection of capacitor value by using ADS is discussed in this section.

By using the design software OrCAD, the layout of PCB is able to be constructed. The method of layout design is discussed in this section, from circuit construction in CIS Capture to ensure the routing of each component to footprint development for each component. Also, PCB is fabricated and the complete PCB with components soldered on it with mounted heat sink is presented. Heat sink is designed to ensure maximum heat dissipation of the device. The measurement setup is presented to obtain experiment data of PAE, gain and output power.

#### **CHAPTER 4: RESULTS AND DISCUSSION**

#### 4.1 Matching Network Development

In this section, the IMN and OMN are developed using the RFT algorithm or initial guess technique. This algorithm enables designer to obtain the initial structure of the matching networks after a series of optimization provided by the algorithm. The elements of the matching networks also provided with the suggested values.

In previous chapter, the algorithm of this technique has been discussed. In order for the algorithm to be executed, designer has to insert parameters through the command prompt. The parameters required by the algorithm have been discussed in section 3.1.2. In Table 4.1, the values of parameters are presented. These values have been chosen in this work based on series of experiments, hence a satisfying performance of the matching networks are obtained with the reference of TPG graph.

	fo	2300MHz
	$T_{o}(dB)$	0.8
	ntr	0
	k	0
	$h=[h_1, h_2, h_3, \dots, h_o]$	[1 1 1 1 1 1]
	wlow	500MHz
	whigh	2300MHz
	# N_opt	80

**Table 4.1: Input Parameters** 

The simulation results are shown in Figure 4.1 and Figure 4.2 respectively. Figure 4.1 presents the schematic IMN obtained from the RFT algorithm whereas Figure 4.2 shows the graph of TPG against frequency for the input matching network obtained.



Figure 4.1: Schematic circuit for IMN



Figure 4.2: Graph of TPG against frequency for IMN

Refer to Figure 4.1, the IMN obtained from the simulation consists of seven elements: three inductors, three capacitors and a 50  $\Omega$  termination. The values of the passive components are in acceptable range. If the value of the component is too large, it will behave as open or short circuit and definitely will impact the overall performance of the PA, especially when obtaining measurement data. Refer to Figure 4.2, the graph of TPG against frequency for IMN is shown. The TPG for this IMN achieved the conditions for an acceptable matching network, which are the graph is in the range between -1 to 0, and the graph is as flat as possible. Therefore, the IMN obtained is valid in terms of theory and is ready to be further tested in terms of simulation and experimental.

As for OMN, the same procedures are applied, with the same values of input parameters for command prompt. However in this case, the impedance data extracted by the algorithm have to modify to load impedances instead of source impedances for the case of IMN. The simulation results are shown in Figure 4.3 and Figure 4.4 respectively. Figure 4.3 presents the schematic OMN obtained from the RFT algorithm whereas Figure 4.4 shows the graph of TPG against frequency for the input matching network obtained.



Figure 4.3: Schematic circuit for OMN



Figure 4.4: Graph of TPG against frequency for OMN

Same as IMN, the OMN obtained from the RFT algorithm consists of seven elements: three inductors, three capacitors and a 50  $\Omega$  termination. The value of passive components are also acceptable. In Figure 4.4, the graph of TPG against frequency for OMN is shown. The TPG graph is in the range of -1 to 0 and demonstrated the trend of flat respond throughout the frequency range. Therefore, the design of OMN obtained from RFT algorithm is acceptable and ready to be tested in simulation and experimental level.

Both of the matching networks are developed using RFT algorithm. The simulation results obtained will be implemented in ADS software for further optimization before realization on prototype level.

# 4.2 Mixed-Lumped Element Implementation

The matching networks are realized with mixed-lumped designed, in specific the inductor will be represented by distributed element which is CPWG transmission line and high-Q discrete capacitor from vendor is chosen as capacitor. In previous section, the method of implementation of CPWG as well as capacitor is discussed. Therefore, in this section, the discussion are further extended to the results of the selection of parameter of CPWG as well as capacitor value.

#### 4.2.1 CPWG Parameter Selection

In order to realize inductor on PCB board, it is represented by CPWG transmission line for a better performance in the wideband RF application. Using LineCalc tool in ADS the value of width (*w*) and length (*l*) of CPWG can be synthesized given the value of impedance  $Z_o$  and electrical length in degree,  $\theta$  with the value of gap thickness (*g*) remain constant.

Computation of  $Z_o$  and  $\theta$  involved two equations. The first equation using the mathematical relationship between reactance and inductance, which is shown below:

$$X = 2\pi f l \tag{4.1}$$

Also, another equation is used which involves the relationship between reactance,  $Z_o$  and  $\theta$ . The equation is shown below:

$$X = Z_o \sin\theta, \theta < 45^o \tag{4.2}$$

By using these two equations, the value of  $Z_o$  and  $\theta$  can be found and the value of w and l can be synthesized through the LineCalc tools. The computation of Zo and  $\theta$  for IMN and OMN are shown in Table 4.2 and Table 4.3 respectively.

l (nH)	1.59	3.03	3.97
f(Mhz)	2300	2300	2300
$X\left(\Omega ight)$	22.9	43.8	57.3
heta	10	15	20
$\sin(\theta)$	0.173648178	0.258819	0.34202
$Zo(\Omega)$	132	169	168

Table 4.3: Zo and  $\theta$  computation for OMN

l (nH)	1.87	3.61	4.57			
f(Mhz)	2300	2300	2300			
$X(\Omega)$	27.1	52.2	66.0			
θ	12	18	24			
$\sin(\theta)$	$\sin(\theta) = 0.207911691$		0.406737			
Ζο (Ω) 130		169	162			

The value of  $\theta$  should be less than 45° and it is determined through a series of experiment with different values. With the value of *Zo* and  $\theta$ , the value of *w* and length *l* can be synthesized with LineCalc tool. Than the values obtained are applied in the circuit of S-parameter conversion 2-port to 1-port shown in Figure 3.5 to determine the equivalent of the transmission line to the inductance. The simulation results with less deviation with the actual inductor value will be chosen as the *w* and *l* of the CPW

# 4.2.2 ATC Capacitor Implementation

For capacitor, the capacitance obtained from RFT algorithm is substituted with the ATC 100A series capacitor for realization on PCB level. Before that, simulation is performed in order to determine the suitable value of ATC capacitor for each matching network. The simulation data for ATC capacitor is obtained from ATC Company, which contain several values of capacitor, but some values are not available to match with the capacitance required. Therefore, a simple simulation is carried out to determine the nearest value of the ATC capacitor to the capacitor value obtained from the algorithm. The construction of circuit for simulation is shown in Figure 3.6. The results of the comparison of ATC capacitor and capacitance from algorithm are shown in Table 4.4 and Table 4.5 respectively.

Capacitance	ATC	Highest Lowest		Average
(pF)	Capacitor	Capacitance	Capacitance Capacitance	
	available (pF)	(pF)	(pF)	
6.345	4.7	6.613	6.613 4.824	
	5.6	8.114	5.658	6.886
	6.8	10.98	6.942	8.961
4.479	3.3	4.027	3.4	3.7135
	3.9	5.088	3.903	4.4955
	4.7	6.613	4.824	5.7185
2.07	1.8	2.035	1.824	1.92595
	2.2	2.249	2.163	2.206
	2.7	3.345	2.828	3.0865

Table 4.4: Capacitance computation and comparison for IMN

Capacitance	ATC	Highest	Lowest	Average
(pF)	Capacitor	Capacitance (pF)	Capacitance (pF)	Capacitance
	available (pF)			
				(pF)
5.168	3.9	5.088	3.903	4.4955
	4.7	6.613	4.824	5.7185
	5.6	8.114	5.658	6.886
3.908	2.7	3.345	2.828	3.0865
	3.3	4.027	3.4	3.7135
	3.9	5.088	3.903	4.4955
1.889	1.5	1.63	1.52	1.575
	1.8	2.035	1.824	1.9295
	2.2	2.249	2.163	2.206

Table 4.5: Capacitance computation and comparison for OMN

For each capacitance, a few sets of ATC capacitor nearest to the value required is chosen to compare with the capacitance. The simulation result shows the capacitance for ATC capacitor in each frequency in the desired range. Therefore, the average capacitance is calculated for the simulated data and the average capacitance nearest to the required capacitance is chosen as the ATC capacitor used in the simulation. The ATC capacitor chosen is highlighted for each set of capacitor.

# 4.3 Simulation Results

In this section, simulation results of the PA design are discussed. First, the matching networks with mixed-lumped element design are simulated to observe the performance with the reference of TPG graph. Next, the matching networks with mixed-lumped implementation are connected with GaN transistor and DC biasing circuit to form a complete PA circuit. Simulation is performed to observe the performance of the PA. Attention is given to the efficiency, gain and output of the designed PA in this work.

#### 4.3.1 Simulation of Matching Networks with Mixed-Lumped Design

For each matching network, mixed-lumped design is implemented. Inductors are represented by CPWG transmission line and ATC capacitor is selected to represent capacitor on PCB level. The value of *w* and *l* computed in previous section are applied to CPWG which replaced the inductors in the network. ATC capacitor is selected based on series of computation and comparison with the required capacitance, which is used in this simulation. The performance of the matching network with mixed-lumped design is observed with the reference of the TPG graph. The schematic of IMN with mixed-lumped design is shown in Figure 4.5



Figure 4.5: Schematic circuit of IMN with mixed-lumped design

The graph of TPG against frequency of the IMN is shown in Figure 4.6. According the thoery, the performance of this IMN design is acceptable within the desired range, which can be applied to PA construction and subject to optimization where necessary.



Figure 4.6: Graph of TPG against frequency for IMN with mixed-lumped design

For OMN, CPWG transmission line and ATC capacitor are also applied with the parameters obtained from previous calculation. The schematic of OMN with mixed-lumped design is shown in Figure 4.7.



Figure 4.7: Schematic circuit of OMN with mixed-lumped design

The graph of TPG against frequency of OMN is shown in Figure 4.8. The performance of OMN with mixed-lumped design is acceptable according to the theory. The graph lies between -1 to 0 and the graph of TPG obtained is as flat as possible within the desired frequency.



# Figure 4.8: Graph of TPG against frequency for OMN with mixed-lumped design

The matching networks with mixed-lumped design are simulated and the performance are observed. From the results obtained, both of the IMN and OMN with mixed-lumped elements implementation are able to achieve a satisfactory performance. Both of the matching networks with this implementation meet the conditions for acceptable matching networks. Hence, IMN and OMN are ready to combine with GaN transistor to form a complete PA circuit for simulation.

70

#### 4.3.2 Simulation of GaN HEMT PA

The matching networks obtained and optimized with mixed-lumped designed are used to construct the PA circuit, together with GaN HEMT. The model of GaN used in this work is CGH40010F GaN from Cree Company. DC biasing circuit is also involve in the circuit construction of PA. Note that in simulation, both of the biasing circuit are in ideal condition. The complete circuit of GaN HEMT PA is shown in Figure 4.9.



Figure 4.9: Complete schematic circuit of GaN PA

In this complete form of PA circuit simulation, the length of CPWG for IMN and OMN are standardize to a certain value to avoid discontinuity of the CPWG which will degrade the performances of PA design. The value of *l* and *w* of the CPWG are shown in Table 4.6. The value chosen for ATC capacitor based on computation and comparison is shown in Table 4.7. Note that the value of *l*, *w* and the value of ATC capacitor listed are selected after series of optimization.

CPWG	Width (w) (mm)	Length ( <i>l</i> ) (mm)
L <sub>1</sub>	1.301	13.3168
$L_2$	1.301	13.4037
L <sub>3</sub>	1.301	7.1124
L <sub>4</sub>	1.301	9.3381
L <sub>5</sub>	1.301	7.9622
L <sub>6</sub>	1.301	12.0596

Table 4.6: Length and width of CPWG

Table 4.7: ATC capacitor value

Capacitor	ATC Capacitor Value (pF)
C1	1.2
$C_2$	3.3
$C_3$	2.7
$C_4$	1.5
$C_5$	1.0
$C_6$	1.0

Simulation is performed in order to obtain and observed the performances of the design PA. In this work, much attention is given to the efficiency, gain and output power of the designed PA. The power added efficiency of the PA is shown in Figure 4.10.



Figure 4.10: Graph of PAE against frequency of GaN PA (Simulation)

From the simulation result, the efficiency of the designed PA achieved a satisfactory performance. In the desired frequency range from 80 Mhz to 2200 MHz, the PAE of the designed PA achieved around of 55% to 69%.

The simulation result of gain of the PA is shown in Figure 4.11. The gain obtained from simulation shows a stable performance across the desired bandwidth, which is around 14 to 15 dB in the frequency range of 80 MHz to 2200 MHz



Figure 4.11: Graph of gain against frequency of GaN PA (Simulation)

For output power, the overall output power achieved by the designed PA is considered remarkable. The output power achieved by PA is in the range of 39 dBm to 42 dBm. The simulation result is shown in Figure 4.12.



Figure 4.12: Graph of output power against frequency of GaN PA

In this section, the simulation results of the GaN PA are observed and discussed. In general, the performances of the designed PA based on simulation are considered remarkable. The PA is able to achieved maximum PAE of 69%. The output power of the designed PA is around 41 dBm and the device shows a stable gain of 14 to 15 dB across the desired frequency range of 80 MHz to 2200 MHz.

University

#### 4.4 Measurement Results

In this section, the measurement results of the designed PA are presented. The measurement results are analyzed and compered with the simulation results obtain. Experimental results usually would be deviated with simulation results, as the simulation results are obtain in the ideal situation. For example in this work, DC biasing circuit in simulation is in the ideal condition, therefore the results obtain are often better than expected. Before discussion of measurement results, fabrication of PCB is briefly presented.

In this section, experiment is carried out in order to obtain measurement results. The measurement setup is discussed in section 3.8. Measurement results obtained are compared with simulation results. The spectrum analyzer is set to the measured frequency with zero span and the marker is placed at the peak of the signal to obtain accurate displayed output power. The measured efficiency, output power and gain of the designed PA are presented and compared with simulation results.

The measured PAE of the GaN wideband PA is shown in Figure 4.13. The graph of measured PAE and simulated PAE are shown in the same graph for comparison purpose.



Figure 4.13: Simulated and Measured result of PAE for GaN PA

From the graph, it is observed that the measured PAE of the designed PA is in agreement with the simulated data in the range of 80 MHz to 1800 MHz, which the PAE achieved around 54% to 67%. However, the performance of the efficiency degraded after the frequency of 1800 MHz, which fall towards 40% efficiency. The outcome is still acceptable for wide bandwidth application.

The measured gain of the designed PA is shown in Figure 4.14. Simulated data and measured data are presented in a same graph.



Figure 4.14: Simulated and Measured result of gain for GaN PA

The gain of the GaN PA demonstrated a stable performance across the bandwidth, and also less deviated from the simulated results. Measured gain achieved 13dB to 18dB in the range of 80 MHz to 2200 MHz, which is considered as a remarkable achievement. The performance of the PA declined after 2000 MHz to around 11dB but the overall gain is acceptable for wideband PA.

The measured output power is shown in Figure 4.15. Both of simulated and measured data are shown in the graph for comparison purpose.



Figure 4.15: Simulated and Measured result of output power for GaN PA

The measured output power of the GaN PA showed a satisfactory achievement. Output power measured 40 dBm to 42 dBm in the frequency range of 80 MHz to 2200 Mhz, although at high frequency (2000 Mhz and above) the output power demonstrated a fall towards 35 dBm. In general the measured data did not show much deviation from simulated result.

Overall, the PAE, gain and output power of the GaN PA achieved satisfactory performances with reasonable agreement with simulated data.

# 4.5 Comparison of performances with various works

In this section, the performances of recent work in the context of GaN wideband PA are summarized in Table 4.8. The significant of this work compared to other work is this work gives good performance from 80 MHz to 2200 MHz for high operating gain.

Work	Device	Bandwidth (GHz)	Efficiency (%)	Output Power (dBm)	Gain (dB)
(Smith & Cripps, 2016)	GaN HEMT	0.5-1.5	44-75	45	10
(A. Grebennikov, 2016)	GaN HEMT	1.4-2.7	68	41	9
(Bae & Kim, 2016)	GaN HEMT	2-6	18-22	41-43	22
(Ma, Zhou, & Yu, 2015)	GaN HEMT	2.1-2.6	70.7	40.5	12.5
(Aridas et al., 2014)	LDMOS	0.27-0.56	65	38.5	10
(Wu, Mkadem, & Boumaiza, 2010)	GaN HEMT	1.9-2.9	63	45.8	10.8
This work	GaN HEMT	0.08-2.2	45-67	34-41	13-18

 Table 4.8: Comparison of Performances of Wideband Power Amplifier in different works

#### 4.6 Summary

In this chapter, the results obtained from experiment are presented and discussed. First, IMN and OMN are developed from the RFT algorithm and the schematic of each matching network is presented. The performance of the matching network obtained is based on the TPG graph.

Next, mixed-lumped design is implemented in both of the matching networks. Inductors are replaces with CPWG transmission line which contributes to the distributed elements and high-Q ATC capacitor is selected to represent capacitor on PCB level. The method of choosing the parameter of CPWG transmission line and method of selection of ATC capacitor are discussed.

Simulation of matching networks with mixed-lumped design is carried out in order to validate the designed matching networks with the reference of TPG graph. Then, the matching networks with mixed-lumped designed connected with GaN transistor and biasing circuit are simulated to observe the PAE, gain and output power of the PA.

The completed PA PCB is used to carry out experiment in order to obtain measurement data. The measurement data of PAE, gain and output power are compared with simulation data. Comparison of performances of wideband PA in various work has been presented.

# **CHAPTER 5: CONCLUSION**

#### 5.1 Conclusion

In this work, a wideband PA for GaN HEMT is designed for tow-way radio application. The methodology of the PA designed is discussed from the development of matching networks towards the fabrication of prototype for measurement purpose. A technique called RFT is employed in the process of matching networks development. Mixed-lumped elements design is implemented in the matching networks, in specific the inductor in represented by CPWG transmission line and high-Q ATC 100A series capacitor is selected to represent lumped element. For measurement data, the proposed PA design with GaN HEMT presents a satisfactory achievement in terms of PAE, output power and gain in the frequency range of 80 MHz to 2200 MHz, and have reasonable agreement with simulated data. The maximum PAE of the PA achieved 69% (range of 39% to 69%) and the gain is more than 13 dB which considered good gain achievement. The output power of the PA achieved 40 dBm to 43 dBm in the frequency range. The PA designed for wideband operation across low frequency (80 MHz) to higher frequency (2200 MHz) achieved state-of-the-art performances. Hence, the proposed methodology in this paper is practical with simplest circuit configuration and enhanced performances, which fulfills the objectives of this research work.

#### 5.2 Future Work

This research work has develop a wideband PA employing RFT and mixed lumpeddistributed elements implementation. The key point of this designed is to employ the methodology mentioned to obtained wideband PA with enhanced performances and in simplest circuit form. The measurement results of the designed PA are presented in Chapter 4. As we can observed the although the measurement performances of designed PA is in good agreement with the simulation data, the results at the higher frequency end (after 2 GHz) showed degradation due to loss, limitation of equipment and errors on the circuit construction. This design can be further improved by adding figure-of-merits such as 3db back-off region, linearity, 1db compression point in order investigate in depth of the PA designed. Also, this designed can be extended to high frequency application (up to 20 Ghz), such as millimeter wave application which is an important application in future 5G communications.

university

#### REFERENCES

- Aridas, N. K., Yarman, B. S., & Chacko, P. (2014). Wideband power amplifier for twoway radio applications via real-frequency technique. *Electronics letters*, 50(23), 1762-1764. doi:10.1049/el.2014.2972
- Bae, K. T., & Kim, D. W. (2016). 2–6 GHz GaN distributed power amplifier MMIC with tapered gate - series/drain - shunt capacitors. *International Journal of RF and Microwave Computer - Aided Engineering*, 26(5), 456-465.
- Bowick, C., Blyler, J., & Ajluni, C. (2008). Chapter 5 The Transistor at Radio Frequencies *RF Circuit Design (Second Edition)* (pp. 103-123). Burlington: Newnes.
- Carlin, H., & Civalleri, P. (1998). Electronic engineering systems series. Wideband Circuit Design.
- Carlin, H. J. (1977). A new approach to gain-bandwidth problems. *Circuits and Systems, IEEE Transactions on*, 24(4), 170-175.
- Carlin, H. J., & Civalleri, P. P. (1985). On flat gain with frequency-dependent terminations. *Circuits and Systems, IEEE Transactions on, 32*(8), 827-839.
- Chen, P., Xia, J., Merrick, B. M., & Brazil, T. J. (2017). Multiobjective Bayesian Optimization for Active Load Modulation in a Broadband 20-W GaN Doherty Power Amplifier Design. *IEEE Transactions on Microwave Theory and Techniques*, 65(3), 860-871. doi:10.1109/TMTT.2016.2636146
- El Din, M. G., Geck, B., & Eul, H. (2009). Adaptive matching for efficiency enhancement of GAN class-F power amplifiers. Paper presented at the Wireless Sensing, Local Positioning, and RFID, 2009. IMWS 2009. IEEE MTT-S International Microwave Workshop on.
- Fano, R. M. (1950). Theoretical limitations on the broadband matching of arbitrary impedances. *Journal of the Franklin Institute*, 249(1), 57-83.
- Grebennikov, A. (2005). *RF and microwave power amplifier design*: McGraw-Hill New-York.
- Grebennikov, A. (2016). High-Efficiency Class-E Power Amplifier With Shunt Capacitance and Shunt Filter. *IEEE Transactions on Circuits and Systems I: Regular Papers*, 63(1), 12-22. doi:10.1109/TCSI.2015.2512698
- Grebennikov, A., Sokal, N. O., & Franco, M. J. (2012). *Switchmode RF and microwave power amplifiers*: Academic Press.
- Kılınç, S., Köprü, R., Aksen, A., & Yarman, B. S. (2014, 12-14 Dec. 2014). Mixed element wideband microwave amplifier design via simplified real frequency technique. Paper presented at the Proceedings of 2014 Mediterranean Microwave Symposium (MMS2014).

- Krishnamurthy, K., Vetury, R., Keller, S., Mishra, U., Rodwell, M. J., & Long, S. I. (2000). Broadband GaAs MESFET and GaN HEMT resistive feedback power amplifiers. *IEEE Journal of Solid-State Circuits*, 35(9), 1285-1292.
- Kuh, E., & Patterson, J. (1961). Design theory of optimum negative-resistance amplifiers. *Proceedings of the IRE, 49*(6), 1043-1050.
- Kumar, N., & Grebennikov, A. (2015). *Distributed Power Amplifiers for RF and Microwave Communications*: Artech House Publishers.
- Kumar, N., Prakash, C., Grebennikov, A., & Mediano, A. (2008). High-efficiency broadband parallel-circuit class E RF power amplifier with reactancecompensation technique. *IEEE Transactions on Microwave Theory and Techniques*, 56(3), 604-612.
- Lindberg, P., Sengul, M., Cimen, E., Yarman, B., Rydberg, A., & Aksen, A. (2006). *A* single matching network design for a dual band PIFA antenna via simplified real frequency technique. Paper presented at the Proc. 1st European Conference on Antennas and Propagation.
- Ma, L., Zhou, J., & Yu, Z. (2015, 6-9 Dec. 2015). *Design of a high-efficiency dual-band harmonic-tuned power amplifier via simplified real frequency technique*. Paper presented at the 2015 Asia-Pacific Microwave Conference (APMC).
- Mohadeskasaei, S. A., & Zhou, X. (2016, 27-29 July 2016). A 30 watt high efficient high power RF pulse power amplifier. Paper presented at the 2016 IEEE MTT-S International Conference on Numerical Electromagnetic and Multiphysics Modeling and Optimization (NEMO).
- Mughal, F. A., Kashif, A., Cheema, N. B., Imran, M., & Azam, S. (2015, 13-17 Jan. 2015). Design and fabrication of a GaN HEMT based amplifier for wideband applications. Paper presented at the 2015 12th International Bhurban Conference on Applied Sciences and Technology (IBCAST).
- Narendra, K., Limiti, E., Paoloni, C., Collantes, J.-M., Jansen, R., & Yarman, S. (2012). Vectorially combined distributed power amplifiers for software-defined radio applications. *Microwave Theory and Techniques, IEEE Transactions on, 60*(10), 3189-3200.
- Rhea, R. (2006). The Yin-Yang of Matching: Part 1—Basic Matching Concepts. *High Frequency Electronics*, 5(3), 16-25.
- Richards, P. I. (1948). Resistor-Transmission-Line Circuits. *Proceedings of the IRE*, 36(2), 217-220. doi:10.1109/JRPROC.1948.233274
- Sertbas, A., & Yarman, B. S. (2004). A computer-aided design technique for lossless matching networks with mixed, lumped and distributed elements. *AEU-International Journal of Electronics and Communications*, 58(6), 424-428.
- Smith, R. M. H., & Cripps, S. C. (2016, 3-4 Oct. 2016). Broadband push-pull power amplifier design at microwave frequencies. Paper presented at the 2016 11th European Microwave Integrated Circuits Conference (EuMIC).

- Wu, D. Y. T., Mkadem, F., & Boumaiza, S. (2010, 23-28 May 2010). Design of a broadband and highly efficient 45W GaN power amplifier via simplified real frequency technique. Paper presented at the 2010 IEEE MTT-S International Microwave Symposium.
- Yarman, B. (1982). A SIMPLIFIED REAL FREQUENCY TECHNIQUE FOR BROAD-BAND MATCHING A COMPLEX GENERATOR TO A COMPLEX LOAD. *RCA review*, 43(3), 529-541.
- Yarman, B. S. (2008). Design of ultra wideband antenna matching networks: via simplified real frequency technique: Springer Science & Business Media.
- Yarman, B. S. (2010). *Design of ultra wideband power transfer networks*: John Wiley & Sons.
- Yarman, B. S., & Aksen, A. (1992). An integrated design tool to construct lossless matching networks with mixed lumped and distributed elements. *Circuits and Systems I: Fundamental Theory and Applications, IEEE Transactions on, 39*(9), 713-723.
- Yarman, B. S., Aksen, A., & Kilinç, A. (2001). An immitance based tool for modelling passive one-port devices by means of darlington equivalents. *AEU-International Journal of Electronics and Communications*, 55(6), 443-451.
- Yarman, B. S., & Carlin, H. J. (1982). A simplified" real frequency" technique applied to broad-band multistage microwave amplifiers. *Microwave Theory and Techniques*, *IEEE Transactions on*, 30(12), 2216-2222.
- Zhang, W., Yamamoto, Y., Oehme, M., Matthies, K., Raju, A. I., Srinivasan, V. S., ... Funk, H. (2016). S-parameter characterization and lumped-element modelling of millimeter-wave single-drift impact-ionization avalanche transit-time diode. *Japanese journal of applied physics*, 55(4S), 04EF03.

# LIST OF PUBLICATIONS AND PAPERS PRESENTED

**ISI-Cited** Journal papers:

- Narendra Kumar, Wen Jun Lim, B. S. Yarman, Prakash Chacko, "A Practical Design Methodology of Wideband UHF RF Power Amplifiers for Two-Way Radio Applications" in IEEE Transaction on Microwave Theory and Techniques. (Under Revision with Major Correction)
- Wen Jun Lim, Narendra Kumar, B. S. Yarman, Prakash Chacko, "Ultrawideband GaN HEMT Power Amplifier with Practical Mixed Lumped Approach Employing Real-Frequency Technique" in IEICE Electronic Express. (Published on 10/7/2017)

Conference:

 Wen Jun Lim, Narendra Kumar, "Design of Wideband Power Amplifier for GaN HEMT for Two-Way Radio Application." 2016 IEEE Industrial Electronics and Applications Conference, Kota Kinabalu, Sabah, Malaysia, 20-22 November 2016.