

This is to certify that the
thesis entitled

The Rapid Design of RF Amplifiers

presented by

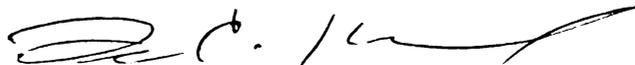
Alexander N. Stewart

has been accepted towards fulfillment
of the requirements for the

Master of
Science

degree in

Electrical and Computer
Engineering



Major Professor's Signature

14 Dec 2004

Date

PLACE IN RETURN BOX to remove this checkout from your record.
TO AVOID FINES return on or before date due.
MAY BE RECALLED with earlier due date if requested.

<u>DATE DUE</u>	<u>DATE DUE</u>	<u>DATE DUE</u>

The Rapid Design of RF Amplifiers

By

Alexander N. Stewart

A THESIS

Submitted to
Michigan State University
in partial fulfillment of the requirements
for the degree of

MASTER OF SCIENCE

Department of Electrical and Computer Engineering

2004

ABSTRACT

The Rapid Design of RF Amplifiers

By

Alexander N. Stewart

Today's electronics market has seen an increase for high frequency, wireless technology. Associated with this trend is the need for suitable RF amplifiers to handle the reception and transmission of these signals. Unfortunately, the cost of these amplifiers from traditional vendors is quite high. It is therefore advantageous to investigate methods to reduce the acquisition cost, such as a custom design.

The design of RF amplifiers is a time consuming and repetitive process if the calculations were to be done by hand. There are commercially available packages which can aid in the design process; however, this is an expensive alternative for creating these types of amplifiers.

The purpose of this thesis will be to develop such a design methodology and tool. First, all of the necessary background in electromagnetic theory is needed to develop the models used in RF amplifier design. Then there is a discussion on how to design these types of amplifiers while pointing out the shortcomings of doing these designs by hand. Next, a Matlab based GUI is introduced which will reduce the time required in the design process. Other software packages are also discussed used in the design process. Finally, results are shown to verify the design process. It is the intent of this work to provide an educational tool for undergraduate students.

Copyright © by
Alexander N. Stewart
2004

To my parents, Robert and Jane

ACKNOWLEDGMENTS

First, I would like to thank my family for their support and giving me a hand when I needed it. Without it, college would have been near impossible.

I would also like to thank my graduate advisor, Dr. Leo Kempel, for taking me on as his student and allowing me the opportunity to attend graduate school at Michigan State University.

Next, I would like to thank Catherine Beauduy and her family for their support. They have been there to offer their advice and support and have been like a second family to me.

Finally, I would like to thank the National Science Foundation (NSF) for their support by funding my way through graduate school with grants ECS-0134236 and DUE-0231312.

TABLE OF CONTENTS

LIST OF TABLES	viii
LIST OF FIGURES	ix
1 Introduction	1
2 Theory	3
2.1 Transmission Lines	3
2.2 Microstrip	9
2.3 Two Port Networks	12
2.3.1 S-Parameters	12
2.3.2 Power Waves	15
2.4 Matching Networks	21
2.5 Signal Flow Graphs	27
2.6 Noise	36
2.6.1 Noise Sources	36
2.6.2 Noise in Systems	37
3 Amplifier Design	40
3.1 Stability	40
3.2 Standing Wave Ratio	43
3.3 Bilateral Conjugate Match	46
3.3.1 Design Procedure	48
3.4 Operating Power Gain	48
3.4.1 Design Procedure	50
3.5 Available Power Gain	51
3.5.1 Design Procedure	52
3.6 SWR Circles	53
3.6.1 Design Procedure	55
3.7 Noise Figure Circles	57

3.7.1	Design Procedure	58
4	Software Tools with Updated Design Procedures	60
4.1	Using the Design Program	60
4.2	Design using the GUI	64
4.3	Circuit Design	67
4.3.1	Microstrip Implementation	67
4.3.2	Lumped Element Design	71
5	Designs and Analysis	73
5.1	S-Parameter Measurement	73
5.2	Amplifier Designs	81
5.2.1	AT-41486 Design	81
5.2.2	MPSH10 Design	87
5.2.3	MPSH17 Design	94
5.2.4	Summary	96
6	Conclusion	98
	BIBLIOGRAPHY	101

LIST OF TABLES

5.1	Transistor S Parameters used in Designs	81
5.2	Design Values	81
5.3	Measured Values Summary	97

LIST OF FIGURES

2.1	Transmission Line Model	3
2.2	Finite Transmission Line	7
2.3	Microstrip Geometry	10
2.4	Two Port Network	13
2.5	BJT Two Port Model	15
2.6	Two Port Model with Load	16
2.7	Reactance of an Open Transmission Line	22
2.8	Reactance of a Shorted Transmission Line	23
2.9	Smith Chart	25
2.10	Two port signal flow graph model	27
2.11	Port 1 equivalent circuit and signal flow graph	29
2.12	Port 2 equivalent circuit and signal flow graph	30
2.13	Two port signal flow graph	31
2.14	Model for Γ_{IN} computation	32
2.15	Model for P_{AVS} computation	33
2.16	System noise figure model	37
3.1	Reflection coefficient diagram of a two port model.	40
3.2	Transistor amplifier with input and output matching networks.	41
4.1	Screenshot of the Microwave Amplifier Design Program	61
5.1	DC Bias Circuit Used in S-Parameter Measurement for the MPSH10	74
5.2	DC Bias Circuit Used in S-Parameter Measurement for the MPSH17	74
5.3	S_{11} of MPSH10	76
5.4	S_{12} of MPSH10	77
5.5	S_{21} of MPSH10	77
5.6	S_{22} of MPSH10	78
5.7	S_{11} of MPSH17	78
5.8	S_{12} of MPSH17	79
5.9	S_{21} of MPSH17	79

5.10	S_{22} of MPSH17	80
5.11	Schematic of AT-41486 Amplifier Bias Circuit	82
5.12	Layout of AT-41486 Amplifier on Microstrip	83
5.13	S_{11} and SWR_{in} of Compensated AT-41486 Amplifier	84
5.14	S_{21} of Compensated AT-41486 Amplifier	85
5.15	S_{22} and SWR_{out} of Compensated AT-41486 Amplifier	87
5.16	Biconjugate Match Network for MPSH10 at 100MHz	88
5.17	S_{11} of Biconjugate Matched MPSH10 Amplifier	89
5.18	S_{21} of Biconjugate Matched MPSH10 Amplifier	90
5.19	S_{22} of Biconjugate Matched MPSH10 Amplifier	90
5.20	Mismatched Network for MPSH10 at 100MHz	91
5.21	S_{11} of Mismatched MPSH10 Amplifier	92
5.22	S_{21} of Mismatched MPSH10 Amplifier	92
5.23	S_{22} of Mismatched MPSH10 Amplifier	93
5.24	Biconjugate Matched Network for MPSH17 at 100MHz	94
5.25	S_{11} of Biconjugate Matched MPSH17 Amplifier	95
5.26	S_{21} of Biconjugate Matched MPSH17 Amplifier	95
5.27	S_{22} of Biconjugate Matched MPSH17 Amplifier	96

CHAPTER 1

Introduction

Amplifiers are common parts of any electronic circuit. In wireless systems, they are typically the first element after the receive antenna. When used in this sort of application, it is important that the input impedance of the amplifier be very close to the impedance of the antenna and add little noise to the system. This increases the dynamic range of the receiver, thereby improving its performance [6]. Finding a commercial amplifier which meets these criteria may be difficult, leaving the designers to develop their own amplifier.

The theory of designing RF transistor amplifiers requires a broad range of topics to be discussed. Chapter two is dedicated to giving the reader the basic knowledge of these topics including transmission line theory, two port networks, and signal flow graphs, to name a few.

Chapter three pools all of this basic knowledge together and creates the framework on how to design an RF amplifier without the aid of any design tools. This leads to the discussion of how difficult the design process can be without the aid of a design tool.

Chapter four presents the reader with the design tool developed by the author of this thesis and explains how to use this program. This design tool computes design parameters necessary to develop the amplifier, and proceeds to explain how to

implement this design. Also, other software packages are discussed which aid in the design process.

Chapter five presents a few designs that were made by the author and discusses the results of these designs as compared to the design goals.

CHAPTER 2

Theory

2.1 Transmission Lines

For a better understanding of the means in which microstrip geometries will be used, it is necessary to include a brief explanation about transmission lines. The model that will be used to develop the theory is shown in Figure 2.1 where the wave is TEM with propagation in the \hat{z} direction [2].

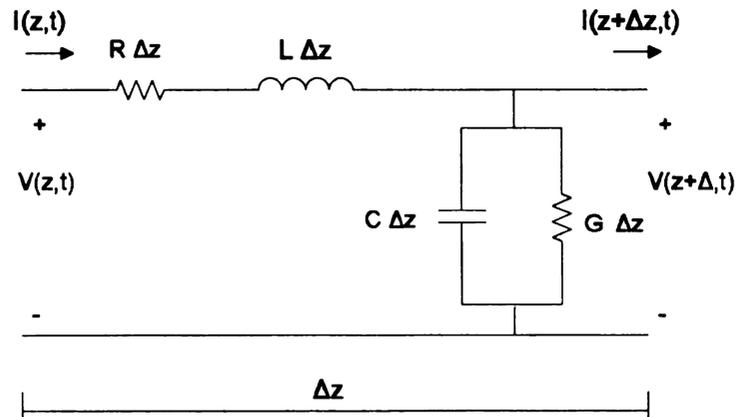


Figure 2.1. Transmission Line Model

It will be assumed that there will be a series resistance R per unit length as well as an inductance per unit length L . Between the two conductors there will be some capacitance per unit length C as well as an conductance per unit length G . Applying Kirchhoff's voltage law around the loop yields

$$\frac{v(z + \Delta z, t) - v(z, t)}{\Delta z} = -Ri(z, t) - L\frac{\partial i(z, t)}{\partial t} \quad (2.1)$$

Taking the limit of (2.1) and allowing $\Delta z \rightarrow 0$ shows

$$\frac{\partial v(z, t)}{\partial z} = -Ri(z, t) - L\frac{\partial i(z, t)}{\partial t} \quad (2.2)$$

Performing the Kirchhoff current law at the $v(z + \Delta z, t)$ node gives

$$\frac{i(z + \Delta z, t) - i(z, t)}{\Delta z} = -C\frac{\partial v(z + \Delta z, t)}{\partial t} - Gv(z + \Delta z, t) \quad (2.3)$$

Taking the limit of (2.3) as $\Delta z \rightarrow 0$ leads to

$$\frac{\partial i(z, t)}{\partial z} = -C\frac{\partial v(z, t)}{\partial t} - Gv(z, t) \quad (2.4)$$

It is convenient to assume that the system involved will be time-harmonic, therefore phasor notion can be used to simplify the equations by defining two new terms:

$$v(z, t) = \text{Re}[V(z)e^{j\omega t}]$$

$$i(z, t) = \text{Re}[I(z)e^{j\omega t}]$$

and so the time convention $e^{j\omega t}$ will be assumed and suppressed. The motivation for this is to turn a partial differential equation into an ordinary differential equation. The simultaneous solving of equations (2.2) and (2.4) give the following two second

order ordinary differential equations

$$\frac{d^2V(z)}{dz^2} = \gamma^2V(z) \quad (2.5)$$

$$\frac{d^2I(z)}{dz^2} = \gamma^2I(z) \quad (2.6)$$

where

$$\gamma = \alpha + j\beta = \sqrt{(R + j\omega L)(G + j\omega C)} \quad (2.7)$$

Equation (2.7) has the terms α and β which represent the attenuation constant and the phase constant respectively.

The solutions to (2.5) and (2.6) are

$$V(z) = V^+e^{-\gamma z} + V^-e^{\gamma z} \quad (2.8)$$

$$I(z) = I^+e^{-\gamma z} + I^-e^{\gamma z} \quad (2.9)$$

where the superscripts represent the direction of travel for each wave. The positive superscript is for wave travel in the $+\hat{z}$ direction and negative superscript is for wave travel in the $-\hat{z}$ direction when referring to Figure 2.1.

Rewriting equation (2.2) into the time-harmonic form yields

$$\frac{dV(z)}{dz} = (-R - j\omega L)I(z) \quad (2.10)$$

Taking equations (2.8) and (2.9) then substituting into (2.10) gives

$$\begin{aligned} \gamma(V^+e^{-\gamma z} - V^-e^{\gamma z}) &= (R + j\omega L)(I^+e^{-\gamma z} + I^-e^{\gamma z}) \\ \frac{\gamma(V^+e^{-\gamma z} - V^-e^{\gamma z})}{(R + j\omega L)} &= I^+e^{-\gamma z} + I^-e^{\gamma z} \\ \sqrt{\frac{(G + j\omega C)}{(R + j\omega L)}}(V^+e^{-\gamma z} - V^-e^{\gamma z}) &= I^+e^{-\gamma z} + I^-e^{\gamma z} \end{aligned} \quad (2.11)$$

Equation (2.11) is similar to Ohm's Law if we define

$$\sqrt{\frac{(R + j\omega L)}{(G + j\omega C)}} = Z_o$$

where Z_o is the characteristic impedance of the transmission line. This will allow the writing of equation (2.11) in the form

$$\frac{(V^+e^{-\gamma z} - V^-e^{\gamma z})}{Z_o} = (I^+e^{-\gamma z} + I^-e^{\gamma z}) \quad (2.12)$$

which now resembles Ohm's Law as desired. Going one step further and grouping the forward traveling wave quantities and reverse traveling wave quantities together yields

$$\frac{V^+(z)}{Z_o} = I^+(z) \quad (2.13)$$

$$-\frac{V^-(z)}{Z_o} = I^-(z) \quad (2.14)$$

Equations (2.8) and (2.9) show that there are four unknowns and only two equations, but if the relationships of (2.13) and (2.14) are used, this reduces the number of unknowns down to two with the resulting equations

$$V(z) = V^+e^{-\gamma z} + V^-e^{\gamma z} \quad (2.15)$$

$$I(z) = \frac{V^+}{Z_o}e^{-\gamma z} - \frac{V^-}{Z_o}e^{\gamma z} \quad (2.16)$$

These equations then allow a solution to be found for any arbitrary position along the transmission line assuming that at least two boundary conditions are provided. Using Figure 2.2, by letting $z = l$ and then solving for V^+ and V^- gives

$$V^+ = \frac{1}{2}(V_L + I_L Z_o)e^{\gamma l} \quad (2.17)$$

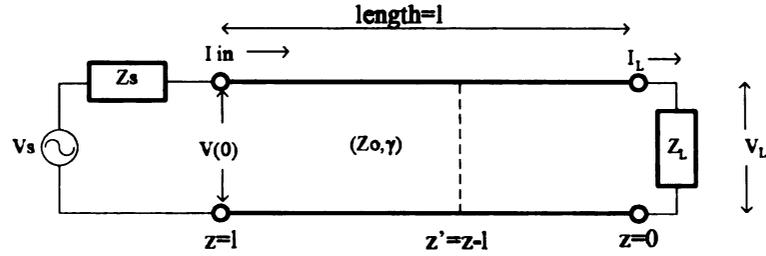


Figure 2.2. Finite Transmission Line

$$V^- = \frac{1}{2}(V_L - I_L Z_o)e^{-\gamma l} \quad (2.18)$$

Knowing that $I_L Z_L = V_L$ and substituting into (2.17) and (2.18) gives the following

$$V^+ = \frac{1}{2}I_L(Z_L + Z_o)e^{\gamma l} \quad (2.19)$$

$$V^- = \frac{1}{2}I_L(Z_L - Z_o)e^{-\gamma l} \quad (2.20)$$

Since $z = l$ is at the same location as $z' = 0$ along the transmission line, they have the same electrical properties. Taking the ratio of equations (2.20) and (2.19) gives a term known as the reflection coefficient and is shown mathematically as

$$\Gamma = \frac{V^-}{V^+} = \frac{Z_L - Z_o}{Z_L + Z_o} \quad (2.21)$$

This quantity represents the electric field intensity that is reflected due to an impedance mismatch at $z = l$. Note that if $Z_o = Z_L$, there is no reflection; a condition which is referred to as a perfect match. This quantity will also be used to represent current reflection as well.

Now to find the voltage or current at any arbitrary point ($z' = z - l$) on the

transmission line and using (2.8) and (2.9) with (2.19) and (2.20) gives

$$V(z') = \frac{I_L}{2} [(Z_L + Z_o)e^{\gamma z'} + (Z_L - Z_o)e^{-\gamma z'}] \quad (2.22)$$

$$I(z') = \frac{I_L}{2Z_o} [(Z_L + Z_o)e^{\gamma z'} + (Z_L - Z_o)e^{-\gamma z'}] \quad (2.23)$$

To find the impedance of a wave that is terminated by some Z_L translated by a distance z' , divide (2.22) by (2.23). Using the identities $e^{\gamma z'} + e^{-\gamma z'} = 2 \cosh(\gamma z')$ and $e^{\gamma z'} - e^{-\gamma z'} = 2 \sinh(\gamma z')$ gives

$$Z(z') = Z_o \frac{Z_L \cosh(\gamma z') + Z_o \sinh(\gamma z')}{Z_o \cosh(\gamma z') + Z_L \sinh(\gamma z')}$$

which can further simplify down to

$$Z(z') = Z_o \frac{Z_L + Z_o \tanh(\gamma z')}{Z_o + Z_L \tanh(\gamma z')} \quad (2.24)$$

Since the transmission line is of length l , a better definition of (2.24) is

$$Z_i = Z_o \frac{Z_L + Z_o \tanh(\gamma l)}{Z_o + Z_L \tanh(\gamma l)} \quad (2.25)$$

where Z_i is the input impedance looking into the transmission line terminated by some arbitrary impedance, Z_L . Equation (2.25) is known as the impedance transformation.

Before moving on to the discussion of microstrip transmission lines, it will be assumed that the transmission lines are lossless. When referring to Figure 2.1, it is possible to assume that the attenuation constant $\alpha = 0$ in the special case that $G = R = 0$. This is not an impossible assumption since the lengths of the conductors on microstrip are relatively short causing $R \approx 0$, and that the conductivity of the dielectric is quite low making $G \approx 0$. Using this assumption allows γ to be rewritten as

$$\gamma = j\beta = \sqrt{(j\omega L)(j\omega C)} = j\omega\sqrt{LC}$$

This allows us to define $\beta = \omega\sqrt{LC}$. Also note that equation (2.25) is now changed. This makes $\tanh(\beta l) = j \tan(\beta l)$ giving the final working equation of (2.25)

$$Z_i = Z_o \frac{Z_L + jZ_o \tan(\beta l)}{Z_o + jZ_L \tan(\beta l)} \quad (2.26)$$

Equation (2.26) is the basis by which most of the networks for the transistor amplifiers will be designed.

2.2 Microstrip

Some of the transistor amplifiers for this thesis were built with microstrip transmission lines. The main reason for this is that the lumped element concept of discrete components is no longer valid since the phase change through the element is not negligible at higher frequencies. These discrete components also have an associated quality factor (Q) to them where they may no longer act optimally. For example, a surface mount resistor will have a capacitance associated with it due the the metallic pad on each end of the resistor. These disadvantages of discrete components can be overcome since equation (2.26) allows one to design a component which has the desired impedance at a certain point with the use of transmission lines.

To use the equation (2.26), the first assumption made was that the mode of propagation was TEM. This however is not valid in the strictest sense for the microstrip. The reason this is the case is because microstrip has half of the conductor exposed to freespace and the other side is in contact with the dielectric. Figure (2.3) shows the geometry where W is the width of the transmission line, h is the distance between

the ground plane and the transmission line, and ϵ_r is the relative permittivity of the dielectric. Since there is a dielectric with a permittivity ϵ_r between the two layers

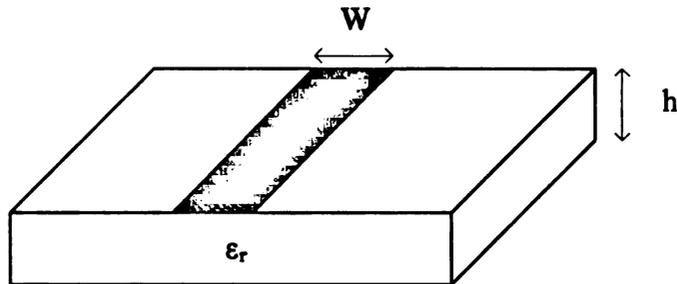


Figure 2.3. Microstrip Geometry

of copper, then the majority of the fields emitted from the top conductor will tend to concentrate inside the material. However, Pozar [6] states that the exact field distribution is a hybrid TE-TM wave, but since the dielectric layer is electrically thin ($h \ll \lambda$), then the field distribution inside the dielectric is almost identical to the fields outside creating a quasi-TEM mode.

The propagation velocity in free-space is defined as

$$v_p = \frac{1}{\sqrt{\mu_0 \epsilon_0}} = c = \frac{\omega}{\beta}$$

where c is the speed of light, μ_0 is the permeability of free-space, ϵ_0 is the permittivity of free-space. If the propagation is in a material other than free-space with some (ϵ_r, μ_r) , then the velocity is scaled by the following relationship

$$v_p = \frac{c}{\sqrt{\mu_r \epsilon_r}} \quad (2.27)$$

Since the microstrip being used is non-magnetic, then $\mu_r = 1$. This changes equation

(2.27) to

$$v_p = \frac{c}{\sqrt{\epsilon_r}} \quad (2.28)$$

In freespace $c = \lambda f$, where f is the frequency of the wave, but as was shown in equation (2.28) the velocity of the wave is scaled by the dielectric constant for that material, and thus the wavelength λ is dependant on the material properties since the frequency does not change and is shown by

$$v_p = \frac{c}{\sqrt{\epsilon_r}} = \lambda f \quad (2.29)$$

To use equation (2.26) for microstrip, it is desirable to find an approximate value for the effective dielectric constant which exists for the microstrip. An approximation for the effective dielectric constant is given [6] below

$$\epsilon_e = \frac{\epsilon_r + 1}{2} + \frac{\epsilon_r - 1}{2} \frac{1}{\sqrt{1 + 12h/W}} \quad (2.30)$$

where $1 < \epsilon_e < \epsilon_r$ if $\epsilon_r > 1$ assuming that the microstrip structure is immersed in free-space. Equation (2.30) allows the computation of the wavelength of the wave that travels on the microstrip allowing us to use (2.26) for designing microstrip lines with the desired impedance. The only thing missing that is needed for full implementation of (2.26) is the characteristic impedance of the transmission line.

Approximate closed form equations for the characteristic impedance are given by

$$Z_o = \frac{60}{\sqrt{\epsilon_e}} \ln \left(\frac{8h}{W} + \frac{W}{4h} \right) \quad \text{for } \frac{W}{h} \leq 1$$

$$Z_o = \frac{120\pi}{\sqrt{\epsilon_e} [W/h + 1.393 + 0.667 \ln(W/h + 1.444)]} \quad \text{for } \frac{W}{h} \geq 1$$
(2.31)

where W is the width of the trace and h is the height of the substrate. There will be times where it will be necessary to design a transmission line for a desired impedance. If that is the case then use the following set of equations

$$\frac{W}{h} = \begin{cases} \frac{8e^A}{e^{2A} - 1} & \text{for } \frac{W}{h} \leq 2 \\ \frac{2}{\pi} \left[B - 1 - \ln(2B - 1) + \frac{\epsilon_r - 1}{2\epsilon_r} \left\{ \ln(B - 1) + .39 - \frac{0.61}{\epsilon_r} \right\} \right] & \text{for } \frac{W}{h} \geq 2 \end{cases} \quad (2.32)$$

where

$$A = \frac{Z_o}{60} \sqrt{\frac{\epsilon_r + 1}{2}} + \frac{\epsilon_r - 1}{\epsilon_r + 1} \left(0.23 + \frac{0.11}{\epsilon_r} \right)$$

$$B = \frac{377\pi}{2Z_o\sqrt{\epsilon_r}}$$

In general, the larger the dielectric constant means that the traces can be thinner versus boards with smaller dielectric values. The transmission line thickness becomes important when deciding which transistor to use since the transistor pin spacing may be narrower than the transmission line width being used and this makes mounting the device quite difficult.

2.3 Two Port Networks

2.3.1 S-Parameters

The transistors that are used for the amplifiers being designed will be treated as two port networks. This has the advantage of allowing the designer to not worry too much about the actual physical phenomena such as junction capacitances and lead inductance that occurs inside the transistor. Instead, all of the physical phenomena are accounted for by the measurement of its two port parameters. The two port pa-

Parameters that are of primary concern are the scattering parameters, or S-parameters, which allows one to measure the wave-like behavior of the device.

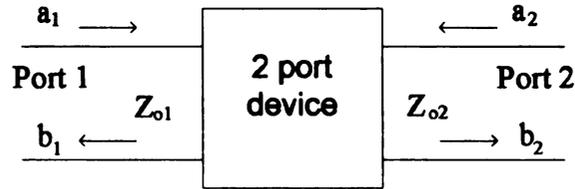


Figure 2.4. Two Port Network

Figure (2.4) is a block diagram of a two port device. On port 1, a_1 represents the incoming wave to the device and b_1 represents the wave leaving the device. Similarly on port 2, a_2 represents the incident wave into the device and b_2 represents the wave leaving the device. In general, there will be some transmission through the network so both b_1 and b_2 will have some of its energy come from a_2 and a_1 , respectively. There will also be some reflection at either port 1 and port 2, which also comprise a portion of b_1 and b_2 , due to the waves impinging on these respective ports.

To express the above mathematically

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} \quad (2.33)$$

where

$$S_{11} = \left. \frac{b_1}{a_1} \right|_{a_2=0} = \begin{matrix} \text{Reflection coefficient at port 1 with} \\ \text{a matched load on port 2} \end{matrix} \quad (2.34)$$

$$S_{12} = \left. \frac{b_1}{a_2} \right|_{a_1=0} = \begin{array}{l} \text{Transmission coefficient output to} \\ \text{port 1 due to an input on port 2} \\ \text{with a matched load at port 1} \end{array} \quad (2.35)$$

$$S_{21} = \left. \frac{b_2}{a_1} \right|_{a_2=0} = \begin{array}{l} \text{Transmission coefficient output to} \\ \text{port 2 due to an input on port 1} \\ \text{with a matched load at port 2} \end{array} \quad (2.36)$$

$$S_{22} = \left. \frac{b_2}{a_2} \right|_{a_1=0} = \begin{array}{l} \text{Reflection coefficient at port 2 with} \\ \text{a matched load on port 1} \end{array} \quad (2.37)$$

The entries of the scattering matrix are typically complex numbers and are frequency dependant. This means that a phase plane must be established for a reference point at each port. If the phase plane is elsewhere along the transmission line, then the given S-parameter must be multiplied by $e^{j\beta l}$, where l is the distance moved from the given reference plane.

Using S-parameters to model the transistors as wave-like devices first requires one to choose the configuration for which the transistor will be operated. The most fundamental configurations for a BJT amplifier are the common-emitter, common-collector, and common-base. Manufacturer data sheets typically give the values of the S-parameters when oriented as a common-emitter amplifier. Other properties which are given are the collector-emitter voltage and collector current. These are the bias conditions and they have a direct effect on the S-parameters as well as the noise figure of the transistor.

Below is the model that will be used for measuring the transistor's 2-port properties configured as a common-emitter. Notice the two port model does not have the external bias circuitry included. This does not invalidate the model since by using either an RF choke or other means, it is possible to make externally connected circuitry

appear as if it were disconnected at higher frequencies.

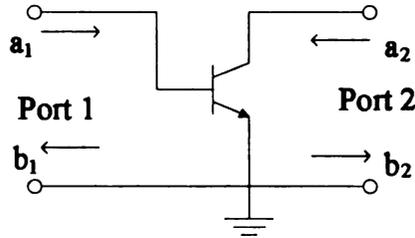


Figure 2.5. BJT Two Port Model

2.3.2 Power Waves

Since we are interested in the wave-like nature of the transistor, it is also reasonable to discuss the amount of power that is either being transmitted or reflected by the device. Ideally, the characterization will be related to the already known S-parameters of the device. Some notation must be defined first before proceeding. It should be noted that the following derivations come from [3]. Referring back to equations (2.8) and (2.9), $V(z)$ and $I(z)$ will be normalized by $\sqrt{Z_o}$ and the assumption of a lossless transmission line (i.e. $\gamma = j\beta$) will be used creates the following definitions

$$\begin{aligned}
 v(z) &= \frac{V(z)}{\sqrt{Z_o}} \\
 i(z) &= I(z)\sqrt{Z_o} \\
 a(z) &= \frac{V^+(z)}{\sqrt{Z_o}} e^{-j\beta z} \\
 b(z) &= \frac{V^-(z)}{\sqrt{Z_o}} e^{j\beta z}
 \end{aligned}$$

The terms $v(z)$ and $i(z)$ are the voltage and current respectively, whereas $a(z)$ and $b(z)$ are the forward and reverse traveling waves as functions of position. These definitions are consistent with equations (2.13) and (2.14). Using these equations we can rewrite (2.8) and (2.9) as the following

$$v(z) = a(z) + b(z) \quad (2.38)$$

$$i(z) = a(z) - b(z) \quad (2.39)$$

which again will allow equations (2.17) and (2.18) to be rewritten as

$$a(z) = \frac{1}{2} [v(z) + i(z)] = \frac{1}{2\sqrt{Z_o}} [V(z) + Z_o I(z)] \quad (2.40)$$

$$b(z) = \frac{1}{2} [v(z) - i(z)] = \frac{1}{2\sqrt{Z_o}} [V(z) - Z_o I(z)] \quad (2.41)$$

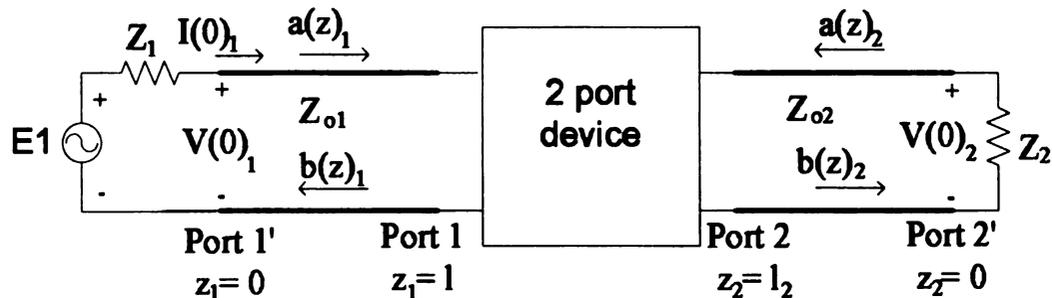


Figure 2.6. Two Port Model with Load

Figure 2.6 shows a source connected to a two port network and a load. The source provides the network with power. It is desirable to find out how much power is being delivered into the system as well as how much power is received at whatever load is

attached to the network, which in this case is Z_2 . The average power of an incident wave is found in the following way

$$\begin{aligned}
 P(z) &= \text{Re}[V^+(z)(I^+(z))^*] \\
 &= \frac{1}{2} \text{Re} \left[V^+(z) \frac{(V^+(z))^*}{Z_o} \right] \\
 &= \frac{1}{2} \frac{|V^+(z)|^2}{Z_o} \\
 &= \frac{1}{2} |a(z)|^2
 \end{aligned} \tag{2.42}$$

The reflected wave's power can be derived in a similar manner as, the incident wave's power; specifically

$$P(z) = \frac{1}{2} |b(z)|^2 \tag{2.43}$$

From Figure 2.6, a Kirchhoff Voltage Law expression at $z_1 = 0$ gives

$$V(0)_1 = E_1 - Z_{o1} I(0)_1 \tag{2.44}$$

if $Z_{o1} = Z_1$. Substituting this expression into (2.40) gives

$$a_1(0) = \frac{E_1}{2\sqrt{Z_{o1}}} \tag{2.45}$$

or

$$|a_1(0)|^2 = \frac{|E_1|^2}{4Z_{o1}} \tag{2.46}$$

Substituting (2.46) into (2.42) yields

$$P_1(0) = \frac{1}{2} |a_1(0)|^2 = \frac{|E_1|^2}{8Z_{o1}} = P_{AVS} \tag{2.47}$$

The result of (2.47) represents the power available from the source (P_{AVS}) when the

source impedance equals the transmission line impedance (i.e. $Z_{o1} = Z_1$, or a matched source). The term P_{AVS} is important and will appear later when discussing network power gains in Section (2.5). It is also worth noting that the power input from the source reaches $z_1 = l_1$ without any loss since there is no power loss in the transmission line.

Suppose that the impedance is not matched at $z_1 = 0$ (i.e. $Z_{o1} \neq Z_1$). In this case there is a reflection at $z_1 = 0$ as given by equation (2.21). This results in $P_1(0)$ not equaling P_{AVS} . It is thus desirable to find what fraction of power does reach $z_1 = l_1$. Start again with finding the input power to the system using equations (2.40) and (2.44) this power is given by

$$\frac{1}{2}|a_1(0)|^2 = \frac{1}{8Z_{o1}}(|V_1(0)|^2 + Z_{o1}I_1(0)V_1^*(0) + Z_{o1}V_1(0)I_1^*(0) + Z_{o1}^2|I_1(0)|^2) \quad (2.48)$$

Similarly using equations (2.41) and (2.44)

$$\frac{1}{2}|b_1(0)|^2 = \frac{1}{8Z_{o1}}(|V_1(0)|^2 - Z_{o1}I_1(0)V_1^*(0) - Z_{o1}V_1(0)I_1^*(0) + Z_{o1}^2|I_1(0)|^2) \quad (2.49)$$

Taking the difference between the above equations gives

$$\begin{aligned} \frac{1}{2}|a_1(0)|^2 - \frac{1}{2}|b_1(0)|^2 &= \frac{1}{4}(I_1(0)V_1^*(0) + I_1^*(0)V_1(0)) \\ &= \frac{1}{2}\text{Re}[I_1(0)V_1^*(0)] \end{aligned} \quad (2.50)$$

which represents the power delivered to the two port network at $z_1 = 0$ which is the same as the power delivered to $z_1 = l_1$. The left had side of (2.50) will be denoted by $P_1(0) = P_1(l_1) = P_{IN}$. The term P_{IN} is another term that will appear often later in the work, and is defined as the power input to the network. Therefore (2.50) can be

rewritten with a substitution from (2.47) as

$$\frac{1}{2}|b_1(0)|^2 = P_{AVS} - P_1(0) \quad (2.51)$$

which is equivalent to

$$\frac{1}{2}|b_1(l_1)|^2 = P_{AVS} - P_1(l_1) \quad (2.52)$$

Using the definition of the S-parameters at $z_1 = l_1$ shows

$$|S_{11}|^2 = \frac{|b_1(l_1)|^2}{|a_1(l_1)|^2} = \frac{P_{AVS} - P_1(l_1)}{P_{AVS}}$$

which can be rearranged to

$$P_1(l_1) = P_{AVS}(1 - |S_{11}|^2)$$

The above equation relates S_{11} in power terms to show how much power is reflected from the port due to an input at port one.

Now a derivation for how much power is transmitted through the two port network will be found by relating it to S_{21} . First using the definition of S_{21} , it is found that

$$\begin{aligned} S_{21} &= \left. \frac{b_2(l_2)}{a_1(l_1)} \right|_{a_2(l_2)=0} = \left. \frac{\sqrt{Z_{o2}}I_2^-(l_2)}{\sqrt{Z_{o1}}I_1^+(l_1)} \right|_{I_2^+(l_2)=0} \\ &= - \left. \frac{\sqrt{Z_{o2}}I_2(l_2)}{\sqrt{Z_{o1}}I_1^+(l_1)} \right|_{I_2^+(l_2)=0} \end{aligned} \quad (2.53)$$

For the rest of the derivation, assume that there is no transmission line between the the voltage source and the 2 port network of Figure 2.6 and that the location they meet is at $z_1 = l_1$ as well as letting $Z_2 = Z_{o2}$ at $z_2 = l_2$. This allows the rewriting of (2.40) as

$$I_1^+(l_1) = \frac{a_1(l_1)}{\sqrt{Z_{o1}}} = \frac{1}{2Z_{o1}}[V_1(l_1) + Z_{o1}I_1(l_1)] \quad (2.54)$$

Another KVL is written at $z_1 = l_1$ as

$$V_1(l_1) = E_1 - Z_{o1}I_1(l_1) \quad (2.55)$$

Substituting (2.55) into (2.54) gives

$$I_1^+(l_1) = \frac{E_1}{2Z_{o1}} \quad (2.56)$$

At port 2 of Figure 2.6, the current is found to be

$$-I_2(l_2) = \frac{V_2(l_2)}{Z_{o2}} \quad (2.57)$$

Substituting (2.56) and (2.57) into (2.53) yields

$$S_{21} = \frac{2\sqrt{Z_{o1}}}{\sqrt{Z_{o2}}} \frac{V_2(l_2)}{E_1} \quad (2.58)$$

Finding the power equivalent term of (2.58) gives

$$|S_{21}|^2 = \frac{\frac{1}{2}|V_2(l_2)|^2/Z_{o2}}{|E_1|^2/8Z_{o1}} \quad (2.59)$$

Recognizing that the denominator is what was defined earlier as P_{AVS} and that the numerator represents the power that is delivered to the load, a new term is defined called the transducer power gain and is denoted as G_T .

If $Z_1 = Z_2 = Z_{o1} = Z_{o2}$, then it follows from (2.58) that

$$S_{21} = \frac{V_2}{\left(\frac{E_1}{2}\right)}$$

which leads to

$$G_T = |S_{21}|^2 = \left| \frac{V_2}{E_1/2} \right|^2$$

Similar derivations of equations (2.59) and (2.47) starting at port 2 can be used to find equations for S_{12} and S_{22} .

2.4 Matching Networks

In most cases, it is desirable to deliver power to a load in the most efficient manner. The concepts presented thus far have shown that power can be reflected at the interface between a transmission line and source impedance, or at the interface between the transmission line and the two port network. It is possible to match the transmission line to either the source impedance or the two port network, but typically the transmission line will be matched to the source impedance since that is usually a system wide impedance. This means that there will be a mismatch between the transmission line and the 2 port network interface. The mismatch can be overcome by the design of matching networks which will transform the impedance of the 2 port network to appear as a matched load which maximizes power transfer. This is accomplished by creating transmission line elements which act as circuit elements by means of equation (2.26) when assuming a lossless transmission line. The primary method that equation (2.26) will be implemented is with microstrip matching networks.

For microstrip geometries, it is practical to use shunt elements which are either open or shorted to the ground plane. This then defines what the load impedance is in (2.26). Exploring the open case ($Z_L = \infty$) shows

$$\begin{aligned}
 Z_{in} &= \lim_{Z_L \rightarrow \infty} Z_o \frac{1 + j(Z_o/Z_L) \tan(\beta l)}{(Z_o/Z_L) + j \tan(\beta l)} \\
 &= -\frac{jZ_o}{\tan(\beta l)} \\
 &= -jZ_o \cot(\beta l)
 \end{aligned} \tag{2.60}$$

Figure 2.7 shows a plot of the above equation. Note that that values repeat over

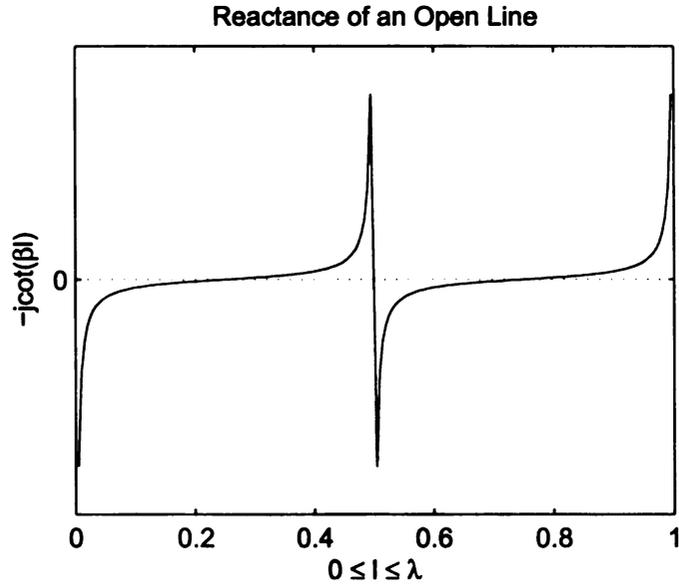


Figure 2.7. Reactance of an Open Transmission Line

a period of $n\lambda/2$ where n is any integer and that the values are both positive and negative over the interval.

Examining the case of the short circuit ($Z_L = 0$) shows

$$\begin{aligned} Z_{in} &= \lim_{Z_L \rightarrow 0} Z_o \frac{Z_L + jZ_o \tan(\beta l)}{Z_o + jZ_L \tan(\beta l)} \\ &= jZ_o \tan(\beta l) \end{aligned} \quad (2.61)$$

with a graph of the resulting equation in Figure 2.8. Just as in the case of the open line, the reactance is periodic every $n\lambda/2$ but is shifted to the left by $\lambda/4$. This implies that when designing a matching network that either the shorted line or open line can give the same desired reactance but one tradeoff then is the transmission line may be longer. The other major tradeoff lies in the fact that it is easier to realize a short as compared to a true open.

Another transmission line that is of note is of length $\lambda/4$. This changes the

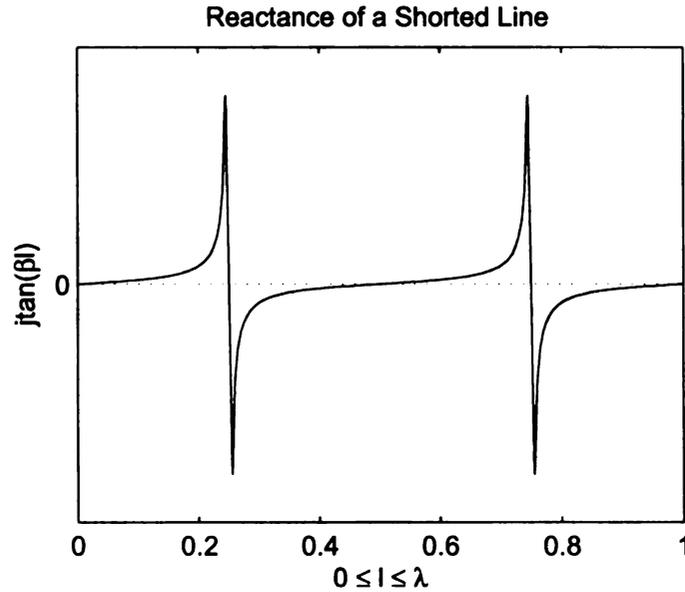


Figure 2.8. Reactance of a Shorted Transmission Line

argument of tangent to the following

$$\begin{aligned}\beta l &= \frac{2\pi}{\lambda} \frac{\lambda}{4} \\ &= \frac{\pi}{2}\end{aligned}$$

where $\tan(\pi/2)$ is undefined. Through some manipulation, equation (2.26) reduces down to

$$\begin{aligned}Z_{in} &= Z_o \frac{jZ_o}{jZ_L} \\ &= \frac{Z_o^2}{Z_L}\end{aligned}\tag{2.62}$$

Equation (2.62) has a special name called a quarter-wave transformer, and is usually used only if Z_o and Z_L are real valued. If Z_L and Z_o are real valued, then Z_{in} will also be real valued, thereby transforming from one resistance to another. All the designer

must do is find the appropriate Z_o to transform between the two desired impedances. Since microstrip is being used, then the designer uses equations (2.32) to find the appropriate width to height ratio.

The last case of note is a transmission line of length $\lambda/8$. The value of $\tan(\pi/4)$ or $\cot(\pi/4)$ simply equal 1, which reduce equations (2.61) to $Z_{in} = jZ_o$ and equation (2.60) to $Z_{in} = -jZ_o$. Again the designer just needs to design for the desired impedance of the transmission line to create the purely reactive element. The length does not need to be restricted to $l = \lambda/8$. Values of $l = (2n - 1)\lambda/8$ where n is any integer are also acceptable. The only thing that will change will be the sign of Z_{in} depending on which quadrant βl ends up in.

Matching networks will be designed using various forms of the above equations as either series elements or parallel elements. For series elements, it makes sense to use the given impedance equations since impedances add in series. In practice, there will typically be some element that will be in parallel since these element themselves are purely reactive when used as opens or shorts on microstrip, thereby changing the reactive component of of a desired impedance. The problem is that the total impedance of elements in parallel have the form

$$Z_{total} = \frac{Z_1 Z_2}{Z_1 + Z_2} \quad (2.63)$$

which when dealing with complex numbers is quite cumbersome mathematically. To make the computation of elements in parallel easier, first change the impedance of the element to an admittance which is done by the relation

$$Y = \frac{1}{Z}$$

Using the relationship of impedance to admittance and equation (2.63), it can be

shown that admittances add when in parallel.

$$\begin{aligned}\frac{1}{Z_{total}} &= \frac{Z_1 + Z_2}{Z_1 Z_2} \\ \frac{1}{Z_{total}} &= \frac{1}{Z_1} + \frac{1}{Z_2} \\ Y_{total} &= Y_1 + Y_2\end{aligned}$$

Finally, most all of the matching network design will be done on a Smith chart. A Smith chart shown in Figure 2.9 is a tool which allows for graphical solutions of

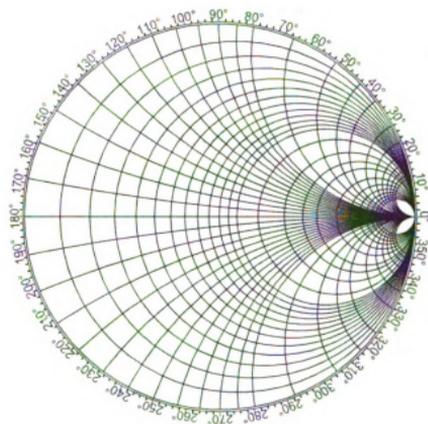


Figure 2.9. Smith Chart

equation (2.26). The circles which are completely contained in the chart represent

the real part of the impedance, whereas the circles which are partially contained are the reactances with those circles in the upper half of the plane being positive valued (e.g. inductance) and those in the lower half being negative (e.g. capacitance). The outer circle represents a reflection coefficient $\Gamma = 1$ while $\Gamma = 0$ when at the origin. Hence, a short would be located on the centerline at the extreme right while an open lies on the centerline of the extreme left. Note that from equation (2.21), Γ can be complex valued and therefore also has an angle associated with it when in polar form. The numbers that are drawn on the outside of the circle represent these angles.

To use the Smith chart, the impedance of a load or source must be normalized by the characteristic impedance of the transmission line. The impedance is then found on the Smith chart and graphed. A line is drawn from the center of the circle through the impedance. The linear distance from the center of the circle and the angle that the ray passes through represent the reflection coefficient of that load when normalized to the given impedance. If it is desired to see what the input impedance would be if a wave traveled of length l , then another circle is drawn centered about the origin in a clockwise manner until the length l has been traveled. The length l is normalized by the actual wavelength in the media of propagation. The value where the circle stops being drawn is the normalized input impedance as seen by the source. Multiplying this number by the characteristic impedance gives the actual resistance value. Also, the reflection coefficient magnitude did not change; however, the polar angle did. This method may sound complicated at first, but when compared to the complex math which is necessary for equation (2.26), matching network design is usually quicker and easier when using a Smith Chart.

2.5 Signal Flow Graphs

The S-parameters described in section 2.3.1 can be implemented by means of a signal flow chart. By using this method, it is possible to define additional terms which are associated with RF transistor amplifiers. This is by no means the only way to derive the following information (see [3] for two derivation methods), but it allows for insight without being overly concerned with the details.

The signal flow model for a two port device is shown in Figure 2.10 The nodes

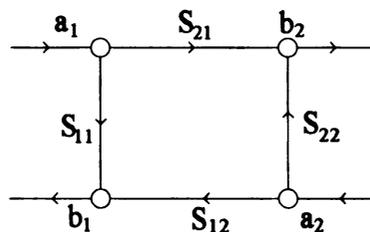


Figure 2.10. Two port signal flow graph model

represent signals and the branches represent systems [5]. Some general rules that will apply to signal flow graphs are listed below [3]

1. Each variable is designated as a node.
2. The S-parameters and reflection coefficients are represented by branches.
3. A branch enters dependent variable nodes and emanates from independent variable nodes. The independent variable nodes are the incident waves and the reflected waves are dependent variable nodes.
4. A node is equal to the sum of the branches entering it.

5. Travel from node to node is only permitted in the direction as indicated by the arrow, and when that does occur, the signal is multiplied by the value adjacent to that arrow.

For example,

$$b_2 = a_1 S_{21} + a_2 S_{22}$$

which is identified as part of the definition of S parameters.

Moving between nodes in Figure 2.10 is quite simple. There will be other systems which are more complicated and require more complex methods for analysis. One way to do this is through Mason's Rule [5] which provides a formulaic way of analyzing these systems. Mason's Rule gives the transfer function of a system when starting from an independent node and traveling to another node and is shown as

$$G = \frac{\text{Ending Node}}{\text{Starting Node}} = \frac{\sum_k T_k \Delta_k}{\Delta} \quad (2.64)$$

where

- k = the number of forward paths
- T_k = the k th forward-path gain
- Δ = $1 - \Sigma$ loop gains + Σ non-touching-loop gains taken two at a time
 $- \Sigma$ non-touching-loop gains taken two at a time + ...
- Δ_k = $\Delta - \Sigma$ loop gain terms in Δ that touch the k th forward path

This equation can become quite complex for large systems, but since there are relatively few forward paths in the two port model being used, the equation (2.64) does not become unmanageable.

A representation of Figure 2.6 as a signal flow graphs must be made to allow for

use of Mason's rule. To modify port one, an equivalent circuit must first be developed which is shown in Figure 2.11. Performing a KVL on the circuit shows

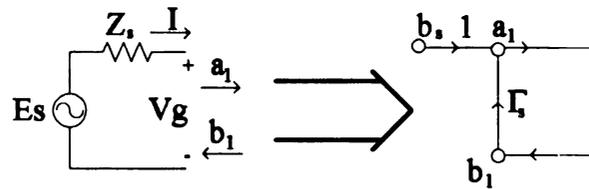


Figure 2.11. Port 1 equivalent circuit and signal flow graph

$$E_s = IZ_s + V_g$$

Recalling that a $V = V^+ + V^-$ and $I = I^+ - I^-$ for waves gives

$$E_s = \frac{Z_s}{Z_o}(V_g^+ - V_g^-) + (V_g^+ + V_g^-)$$

Solving for V_g^+ yields

$$\frac{V_g^+}{\sqrt{Z_o}} = \frac{V_g^-}{\sqrt{Z_o}} \frac{Z_s - Z_o}{Z_s + Z_o} + \frac{E_s \sqrt{Z_o}}{Z_s + Z_o} \quad (2.65)$$

which can be rewritten as

$$a_1 = b_1 \Gamma_s + b_s \quad (2.66)$$

where

$$a_1 = \frac{V_g^+}{\sqrt{Z_o}}$$

$$b_1 = \frac{V_g^-}{\sqrt{Z_o}}$$

$$b_s = \frac{E_s \sqrt{Z_o}}{Z_s + Z_o}$$

$$\Gamma_s = \frac{Z_s - Z_o}{Z_s + Z_o}$$

Equation (2.66) can be represented in signal flow graph terms and is shown on the right side of Figure 2.11.

The output signal flow graph must also be derived. The model for the output is shown in Figure 2.12. Similar to the derivation for the input signal flow graph, a

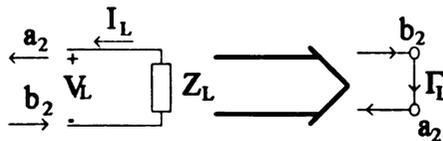


Figure 2.12. Port 2 equivalent circuit and signal flow graph

KVL is performed around the circuit giving the following equation

$$V_L = -I_L Z_L$$

when written in terms of traveling waves gives

$$V_L^+ + V_L^- = -\frac{Z_L}{Z_o}(V_L^+ - V_L^-)$$

Solving for V_L^+ and normalizing by $\sqrt{Z_o}$

$$\frac{V_L^+}{\sqrt{Z_o}} = \frac{V_L^-}{\sqrt{Z_o}} \frac{Z_L - Z_o}{Z_L + Z_o} \quad (2.67)$$

This can be rewritten in terms of wave quantities as

$$a_2 = b_2 \Gamma_L \quad (2.68)$$

where

$$a_2 = \frac{V_L^+}{\sqrt{Z_o}}$$

$$b_2 = \frac{V_L^-}{\sqrt{Z_o}}$$

$$\Gamma_L = \frac{Z_L - Z_o}{Z_L + Z_o}$$

A composite model can now be defined from the two port network in Figure 2.10 and the networks developed in Figures 2.11 and 2.12 and is shown in Figure 2.13.

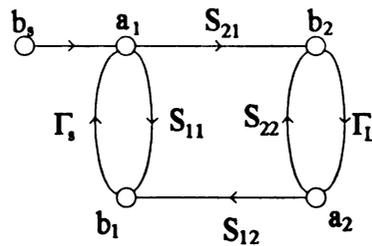


Figure 2.13. Two port signal flow graph

An important quantity for amplifiers is the input reflection coefficient, Γ_{IN} , due to an arbitrary termination at the output as shown in Figure 2.14. The mathematical definition of the input reflection coefficient is $\Gamma_{IN} = b_1/a_1$. This is in the form that allows for use of Mason's rule since a_1 is now an independent variable. The first step for Mason's rule is to find all of the forward paths from a_1 to b_1 . There are two, one

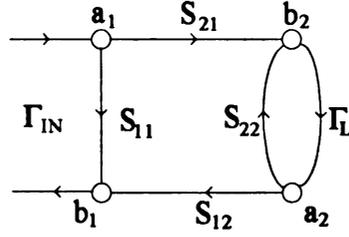


Figure 2.14. Model for Γ_{IN} computation

of which is $T_1 = S_{21}\Gamma_L S_{12}$ and the other is $T_2 = S_{11}$. Loop gains must also be found. There is only one loop gain, which is $S_{22}\Gamma_L$. Next, Δ must be found which is simply $1 - S_{22}\Gamma_L$. Finally, Δ_k must be found for each T_k , so $\Delta_1 = 1$ and $\Delta_2 = 1 - S_{22}\Gamma_L$. Substituting into equation (2.64) gives

$$\begin{aligned}\Gamma_{\text{IN}} &= \frac{S_{11}(1 - S_{22}\Gamma_L)}{1 - S_{22}\Gamma_L} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L} \\ \Gamma_{\text{IN}} &= S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L}\end{aligned}\quad (2.69)$$

A similar derivation can be obtained at the output to find the output reflection coefficient which is defined as $\Gamma_{\text{OUT}} = b_2/a_2$ where the input has an arbitrary source impedance, $b_s = 0$, and a_2 is an independent node. By applying Mason's rule again, the following equation results

$$\Gamma_{\text{OUT}} = S_{22} + \frac{S_{12}S_{21}\Gamma_S}{1 - S_{11}\Gamma_S}\quad (2.70)$$

The power delivered to the load is just the power of the incident wave minus the power reflected from the load, or

$$P_L = \frac{1}{2}|b_2|^2 - \frac{1}{2}|a_2|^2 = \frac{1}{2}|b_2|^2(1 - |\Gamma_L|^2)\quad (2.71)$$

If the circuits from Figures 2.11 and 2.12 were not connected to a two port network, then the signal flow graph in Figure 2.15 would be created.

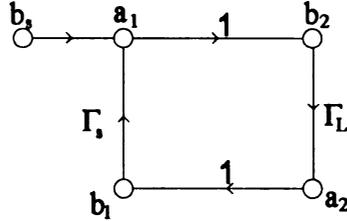


Figure 2.15. Model for P_{AVS} computation

Recall that the definition for P_{AVS} requires that a matched load be attached to the network. The P_{AVS} is equal to the power leaving the source minus the power that is returned to the source giving,

$$P_{AVS} = \frac{1}{2}|a_1|^2 - \frac{1}{2}|b_1|^2 \quad (2.72)$$

If the load attached is a matched load (i.e. $\Gamma_L = \Gamma_S^*$), then $a_1 = b_s + \Gamma_S \Gamma_S^* a_1$ and $b_1 = \Gamma_S^* a_1$, which gives

$$a_1 = \frac{b_s}{1 - |\Gamma_S|^2} \quad (2.73)$$

$$b_1 = \frac{b_s \Gamma_S^*}{1 - |\Gamma_S|^2} \quad (2.74)$$

Substituting equations (2.73) and (2.74) into (2.72) yields

$$P_{AVS} = \frac{\frac{1}{2}|b_s|^2}{1 - |\Gamma_S|^2} \quad (2.75)$$

Recalling that the transducer power gain (G_T) is the ratio of the power delivered to the load (P_L) to the power available from the source (P_{AVS}), the following equation is now created

$$G_T = \frac{P_L}{P_{AVS}} = \frac{|b_2|^2}{|b_s|^2} (1 - |\Gamma_L|^2)(1 - |\Gamma_S|^2) \quad (2.76)$$

The ratio of b_2/b_s can be solved with Mason's rule and after some simplification is given by

$$\frac{b_2}{b_s} = \frac{S_{21}}{(1 - S_{11}\Gamma_S)(1 - S_{22}\Gamma_L) - S_{12}S_{21}\Gamma_L\Gamma_S} \quad (2.77)$$

Substituting equation (2.77) into equation (2.76) gives

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

which simplifies to

$$G_T = \frac{1 - |\Gamma_S|^2}{|1 - \Gamma_{IN}\Gamma_S|} |S_{21}|^2 \frac{1 - |\Gamma_L|^2}{|1 - S_{22}\Gamma_L|^2} \quad (2.78)$$

The operating power gain (G_P) is defined as the ratio of the power delivered to the load, P_L , to the power input into the network, P_{IN} . Describing P_{IN} in terms of signal flow graphs is shown here.

$$P_{IN} = \frac{1}{2}|a_1|^2 - \frac{1}{2}|b_1|^2$$

P_{IN} has a similar definition as P_{AVS} , but the distinction lies in the fact there is no restriction placed on the load impedance, therefore the definition is modified to

$$P_{IN} = \frac{1}{2}|a_1|^2 - \frac{1}{2}|b_1|^2 = \frac{1}{2}(1 - |\Gamma_{IN}|^2)$$

where Γ_{IN} is a function of the S parameters as well as the load impedance.

Now G_P can be written as

$$G_P = \frac{P_L}{P_{IN}} = \frac{|b_2|^2(1 - |\Gamma_L|^2)}{|a_1|^2(1 - |\Gamma_{IN}|^2)}$$

Dividing the numerator and denominator by b_s gives

$$G_P = \frac{P_L}{P_{IN}} = \frac{\left| \frac{b_2}{b_s} \right|^2 (1 - |\Gamma_L|^2)}{\left| \frac{a_1}{b_s} \right|^2 (1 - |\Gamma_{IN}|^2)} \quad (2.79)$$

then finding the ratios of b_2/b_s and a_1/b_s by Mason's rule gives the final form of G_P which is

$$G_P = \frac{1}{1 - |\Gamma_{IN}|^2} |S_{21}|^2 \frac{1 - |\Gamma_{IN}|^2}{|1 - S_{22}\Gamma_L|^2} \quad (2.80)$$

One last power term which needs to be defined is the power available from the network which is the power delivered to a conjugately matched load and is written as

$$\begin{aligned} P_{AVN} &= P_L|_{\Gamma_L = \Gamma_{OUT}^*} = \left[\frac{1}{2}|b_2|^2 - \frac{1}{2}|a_2|^2 \right]_{\Gamma_L = \Gamma_{OUT}^*} = \\ & \left[\frac{1}{2}|b_2|^2(1 - |\Gamma_L|^2) \right]_{\Gamma_L = \Gamma_{OUT}^*} = \frac{1}{2}|b_2|^2(1 - |\Gamma_{OUT}|^2) \end{aligned} \quad (2.81)$$

This allows one more gain term to be defined, which is called the available gain and is shown as

$$G_A = \frac{P_{AVN}}{P_{AVS}} = \frac{|b_2|^2}{|b_s|^2} (1 - |\Gamma_{OUT}|^2)(1 - |\Gamma_S|^2) \quad (2.82)$$

Using Mason's rule again on the ratio b_2/b_s , substituting into equation (2.82) gives

$$G_A = \frac{1 - |\Gamma_S|^2}{|1 - S_{22}\Gamma_S|^2} |S_{21}|^2 \frac{1}{1 - |\Gamma_{OUT}|^2} \quad (2.83)$$

2.6 Noise

Noise is a phenomenon which is both naturally generated as well as man made. In either case it is undesirable when developing systems which require a high sensitivity to low powered signals. Noise is typically classified into two broad categories [11]. One category refers to the types of noise generated by devices inside of a receiver such as resistors, electron tubes, and semiconductors. The other category refers to those noise sources which are external to the receiver and examples of these would be lightning discharge and cosmic radiation. Obviously the designer has no control of the latter of these types of noises, but can control those of the former.

2.6.1 Noise Sources

One source for noise is thermal noise which is defined as the noise generated by the random motion of electrons in a conductive medium. Nyquist was one of the first people to have studied thermal noise. His theorem on noise states that the root-mean-square (RMS) noise voltage appearing across a resistor of value R ohms at temperature T in kelvin across a bandwidth B in hertz is given as

$$v_{n,RMS}^2 = 4kTRB \quad (2.84)$$

where k is Boltzmann's constant.

Another type of noise is called shot noise. Shot noise is caused by the quantized and random nature of current flow [4]. When a DC current is flowing in a device, there is some average current flowing in it, I_o , but does not indicate any variation or frequency of this variation that is indicated from the average current. Schottky's theorem relates this average current to the noise generated as shown below

$$i_{n,RMS}^2 = 2eI_oB \quad (2.85)$$

where e is the charge of an electron in coulombs and B is the bandwidth in hertz of the circuit.

Other types of sources of noise are generation-recombination, temperature fluctuation, and flicker noise. Generation-recombination is caused by electron hole pairs being recombined in a semiconductor and is typically classified as a shot noise. Temperature fluctuation noise is caused by the temperature interaction of the device with its environment. Flicker noise is a type of noise that is characterized by its spectral density that increases with decreasing frequency [11].

2.6.2 Noise in Systems

Figure 2.16 shows two amplifiers which are cascaded and have available gains of G_{A1} and G_{A2} . The resistor is acting as a noise source and has a signal input power P_N ,

$$P_{Ni} = \frac{v_{n,rms}^2}{4R} = kT_0B$$

where T_0 is a temperature of 290°K (for room temperature). Each amplifier has its own contribution to the noise of the system and is denoted as P_{n1} and P_{n2} for amplifiers one and two respectively. The noise output from the system is simply labeled as P_{No} .

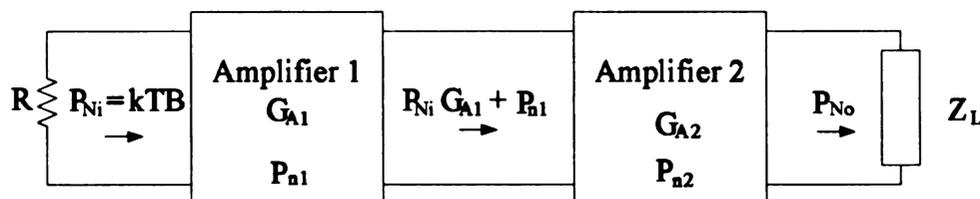


Figure 2.16. System noise figure model

One characteristic of an amplifier is its noise figure which defined as the ratio of the signal-to-noise ratio (SNR) power at the output of the amplifier to the signal-to-noise ratio power at the input of the amplifier [10], or expressed mathematically as

$$F = \frac{P_{Si}/P_{Ni}}{P_{So}/P_{No}} \quad (2.86)$$

where P_{Si} is the input signal power and P_{So} is the output signal power. Equation (2.86) can be rewritten by using the fact that $G_A = P_{So}/P_{Si}$ as

$$F = \frac{P_{No}}{P_{Ni}G_A} \quad (2.87)$$

Referring to Figure 2.16, it is easily seen that the output noise power is

$$P_{No} = G_{A2}(G_{A1}P_{Ni} + P_{n1}) + P_{n2} \quad (2.88)$$

Dividing both sides by $G_{A2}G_{A1}P_{Ni}$ gives

$$F = \frac{P_{No}}{G_{A2}G_{A1}P_{Ni}} = 1 + \frac{P_{n1}}{G_{A1}P_{Ni}} + \frac{P_{n2}}{G_{A2}G_{A1}P_{Ni}} \quad (2.89)$$

or alternatively

$$F = F_1 + \frac{F_2 - 1}{G_{A1}} \quad (2.90)$$

where

$$F_1 = 1 + \frac{P_{n1}}{P_{Ni}G_{A1}}$$

$$F_2 = 1 + \frac{P_{n2}}{P_{Ni}G_{A2}}$$

F_1 and F_2 are called the individual noise figures of the first and second amplifiers respectively.

Closer inspection of equation (2.90) shows that the overall system noise figure is reduced by the gain of the first stage. The problem is that the higher the gain, the higher the noise figure for that stage typically. Thus, there is a tradeoff in the first stage amplifier; does one increase the gain of the first stage to reduce the noise contribution from the second stage or does one design for reducing the noise added by the first stage amplifier by lowering the gain at the cost of having a greater contribution from the second stage? The answer to this lies in a term called the noise measure (M) and is found to be

$$M = \frac{F - 1}{1 - \frac{1}{G_A}} \quad (2.91)$$

If equation (2.91) is computed for each stage, the amplifier with the lower M should be the first stage amplifier. This will ensure that the system noise figure be at its lowest.

CHAPTER 3

Amplifier Design

3.1 Stability

A stable amplifier by definition is one which does not oscillate. A two port network is unconditionally stable if the reflection coefficients at both ports are less than one. When referring to Figure 3.1, $|\Gamma_{IN}|$ and $|\Gamma_{OUT}|$ are these port reflection coefficients. A value of greater than one represents that the port being measured has a negative

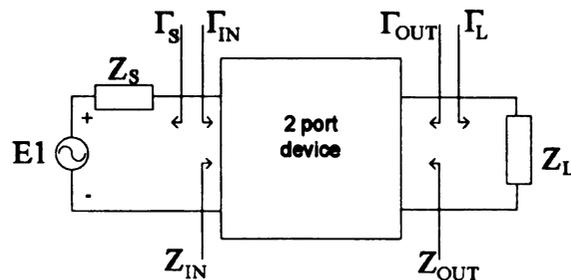


Figure 3.1. Reflection coefficient diagram of a two port model.

resistance, which is shown by solving equation (2.21) for Z_L .

$$Z_L = Z_o \frac{1 + \Gamma}{1 - \Gamma}$$

If Z_L is real valued and $\Gamma > 1$, then the denominator is negative which in turn implies $\text{Re}[Z_L]$ is negative as well.

Figure 3.2 shows the transistor amplifier with both input and output matching networks attached. Recall that Γ_{IN} (equation 2.69) is a function of the S-parameters

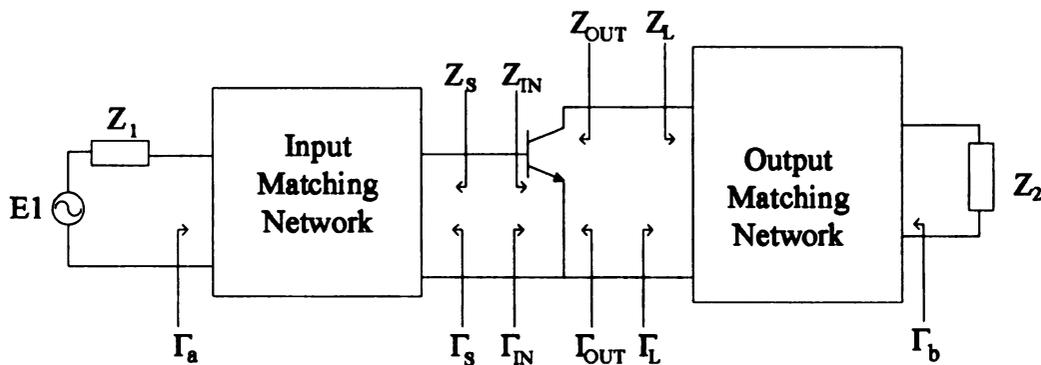


Figure 3.2. Transistor amplifier with input and output matching networks.

of the transistor as well as Γ_L and that Γ_{OUT} (equation 2.70) is a function of the S-parameters of the transistor as well as Γ_s . The designer will have some nominal frequency when designing the amplifiers, so the S-parameters for the transistor will not vary at that frequency. The only values that can change for a given frequency are Γ_s and Γ_L . The arbitrary selection of these could cause an instability, thus it is desirable to know which values of Γ_s and Γ_L will prevent this from happening. Since most of the design process will be done on the Smith chart, it is desirable to be able to show what regions where these passive terminations could cause an instability.

The derivation of the stability circles is given in [3], but the general procedure is to let $|\Gamma_{\text{IN}}| = |\Gamma_{\text{OUT}}| = 1$ and find for which values those correspond to for Γ_L and Γ_S in equations (2.69) and (2.70) respectively. The Γ_L stability circle is given by

$$r_L = \left| \frac{S_{12}S_{21}}{|S_{22}|^2 - |\Delta_s|^2} \right| \quad (3.1)$$

$$C_L = \frac{(S_{22} - \Delta_s S_{11}^*)^*}{|S_{22}|^2 - |\Delta_s|^2} \quad (3.2)$$

where r_L is the radius of the circle, C_L is the location of the center of the circle is on the Smith chart, and

$$\Delta_s = S_{11}S_{22} - S_{12}S_{21}$$

Similarly for Γ_S

$$r_S = \left| \frac{S_{12}S_{21}}{|S_{11}|^2 - |\Delta_s|^2} \right| \quad (3.3)$$

$$C_S = \frac{(S_{11} - \Delta_s S_{22}^*)^*}{|S_{11}|^2 - |\Delta_s|^2} \quad (3.4)$$

where r_S is the radius of the circle, C_S is the location of the center of the circle, and Δ_s has the same definition as above.

The drawing of these circles on the Smith chart will likely create an overlap with the Smith chart. The region of overlap is the area in which the amplifier is unstable as long as $S_{nn} < 1$, where S_{nn} is either S_{11} or S_{22} . The region of stability is anywhere else on the Smith chart. If the stability circles do not overlap on any region of the standard Smith chart, then the transistor is then classified unconditionally stable at that frequency since no selection of a passive Γ_L or Γ_S network will cause an instability.

The necessary and sufficient means of showing unconditional stability is given in

Appendix B of [3] but here is summarized as

$$K > 1$$

and

$$|\Delta_s| < 1$$

where

$$K = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |\Delta_s|^2}{2|S_{12}S_{21}|^2} \quad (3.5)$$

If neither of these two conditions are met, then the transistor is potentially unstable and the stability circles must be plotted to find which regions are considered stable.

3.2 Standing Wave Ratio

One important aspect of any amplifier design is the standing wave ratio (SWR) on that transmission line. The definition of SWR is the ratio of the maximum voltage (or current) to the minimum voltage (or current) on a transmission line. For VSWR, this is shown as

$$\text{SWR} = \frac{|V_{max}|}{|V_{min}|} = \frac{1 + |\Gamma|}{1 - |\Gamma|} \quad (3.6)$$

Recall that Γ in equation (2.21) is a function of the characteristic impedance as well as the load impedance. If $\Gamma = 0$ then it is easily seen that equation (3.6) is equal to one. If $Z_L \neq Z_o$ then the reflection coefficient is non-zero. Any non-zero value for Γ causes (3.6) to become a value greater than one. From this, it is intuitive that an SWR value near one is desirable as it implies that the impedances are matched.

In transistor amplifiers, a SWR can not only be due to an impedance mismatch, but is also due to the actual matching networks which are connected to the input and outputs of the transistor, as will be shown below.

Before explaining how the input and output networks affect SWR, a small discussion must be made about gain and power relationships. This will lead to the derivation of how SWR is affected by these networks. A summary of the gain terms of importance defined in section 2.5 is given by

$$\begin{aligned}
 G_T &= \frac{P_L}{P_{AVS}} = \frac{\text{power delivered to the load}}{\text{power available from the source}} \\
 G_P &= \frac{P_L}{P_{IN}} = \frac{\text{power delivered to the load}}{\text{power input to the network}} \\
 G_A &= \frac{P_{AVN}}{P_{AVS}} = \frac{\text{power available from the network}}{\text{power available from the source}}
 \end{aligned}$$

If the ratio of G_T and G_P is taken, the P_L terms will cancel and the following expression is found

$$\frac{G_T}{G_P} = \frac{P_{IN}}{P_{AVS}}$$

If this ratio is equated to some constant called the mismatch factor, then the following equations are defined

$$P_{IN} = P_{AVS}M_s \quad (3.7)$$

$$G_T = G_P M_s \quad (3.8)$$

where M_s is called the source mismatch factor and can be written as

$$M_s = \frac{(1 - |\Gamma_S|^2)(1 - |\Gamma_{IN}|^2)}{|1 - \Gamma_S \Gamma_{IN}|^2} \quad (3.9)$$

Similarly, an output mismatch term can be found by taking the ratio of G_T and G_A with the final resulting equations

$$P_L = P_{AVN}M_L \quad (3.10)$$

$$G_T = G_A M_L \quad (3.11)$$

where M_L is called the load mismatch factor and can be written as

$$M_L = \frac{(1 - |\Gamma_L|^2)(1 - |\Gamma_{OUT}|^2)}{|1 - \Gamma_{OUT}\Gamma_L|^2} \quad (3.12)$$

Referring to Figure 3.2, the term Γ_a is a reflection coefficient as seen by the source for the entire transistor network. It follows from here that the input SWR will be defined as

$$SWR_{in} = \frac{1 + |\Gamma_a|}{1 - |\Gamma_a|} \quad (3.13)$$

The input power into the system from the source is $P_{IN} = P_{AVS}(1 - |\Gamma_a|^2)$. Using the above equation and equation (3.7) gives the following result

$$M_s = 1 - |\Gamma_a|^2$$

or

$$|\Gamma_a| = \sqrt{1 - M_s} \quad (3.14)$$

Substituting the definition of M_s into (3.14) gives

$$|\Gamma_a| = \sqrt{1 - \frac{(1 - |\Gamma_S|^2)(1 - |\Gamma_{IN}|^2)}{|1 - \Gamma_S\Gamma_{IN}|^2}} = \left| \frac{\Gamma_{IN} - \Gamma_S^*}{1 - \Gamma_{IN}\Gamma_S} \right| \quad (3.15)$$

Examination of equation (3.15) shows that SWR_{in} is a function of Γ_{IN} and Γ_S , where Γ_{IN} itself is a function of the S-parameters of the transistor as well as the output matching network, Γ_L . Equation (3.15) also shows that if $\Gamma_{IN} = \Gamma_S^*$, then $|\Gamma_a| = 0$ which will cause SWR_{in} to equal one.

The output SWR is defined as

$$SWR_{out} = \frac{1 + |\Gamma_b|}{1 - |\Gamma_b|} \quad (3.16)$$

where Γ_b is the reflection coefficient at the output of the transistor network as seen by the load. Using the concept of the load mismatch factor then $|\Gamma_b|$ can be written as

$$\Gamma_b = \sqrt{1 - M_L} \quad (3.17)$$

Substituting (3.12) into (3.17) gives

$$|\Gamma_b| = \sqrt{1 - \frac{(1 - |\Gamma_L|^2)(1 - |\Gamma_{OUT}|^2)}{|1 - \Gamma_{OUT}\Gamma_L|^2}} = \left| \frac{\Gamma_{OUT} - \Gamma_L^*}{1 - \Gamma_{OUT}\Gamma_L} \right| \quad (3.18)$$

This shows that $|\Gamma_b|$ is a function of Γ_{OUT} and Γ_L where Γ_{OUT} is a function of the transistor S-parameters as well as the input matching network Γ_S . If $\Gamma_L = \Gamma_{OUT}^*$, then $|\Gamma_b| = 0$ thereby causing SWR_{out} to equal one.

3.3 Bilateral Conjugate Match

By definition, a bilateral conjugate match occurs if

$$\Gamma_S = \Gamma_{IN}^*$$

and

$$\Gamma_L = \Gamma_{OUT}^*$$

Recalling equations (2.69) and (2.70) and using the conditions for a bilateral match give

$$\Gamma_S^* = S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L} \quad (3.19)$$

$$\Gamma_L^* = S_{22} + \frac{S_{12}S_{21}\Gamma_S}{1 - S_{11}\Gamma_S} \quad (3.20)$$

The simultaneous solutions of (3.19) and (3.20) give the following results

$$\Gamma_{Ms} = \frac{B_1 \pm \sqrt{B_1^2 - 4|C_1|^2}}{2C_1} \quad (3.21)$$

$$\Gamma_{ML} = \frac{B_2 \pm \sqrt{B_2^2 - 4|C_2|^2}}{2C_2} \quad (3.22)$$

$$B_1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta_s|^2 \quad (3.23)$$

$$B_2 = 1 + |S_{22}|^2 - |S_{11}|^2 - |\Delta_s|^2 \quad (3.24)$$

$$C_1 = S_{11} - \Delta_s S_{22}^* \quad (3.25)$$

$$C_2 = S_{22} - \Delta_s S_{11}^* \quad (3.26)$$

where Γ_{Ms} and Γ_{ML} represent the bilateral conjugate matched values. The next question to ask is what sign is used in equations (3.21) and (3.22). The proofs for which sign to use are in Appendix E of [3]. The results say that if $|B_1/2C_1| > 1$ and $B_1 > 0$ for equation (3.21), then the solution with the minus sign is appropriate to produce $|\Gamma_{Ms}| < 1$. If $|B_1/2C_1| > 1$ and $B_1 < 0$, then the plus sign is used to produce $|\Gamma_{Ms}| < 1$. Similar requirements are used for equation (3.22) on which sign to choose.

A bilateral conjugate match can occur only if the the transistor is unconditionally stable for that given frequency. This is the case since only in an unconditionally stable transistor will all of the Smith chart be available for the selection of Γ_{Ms} and Γ_{ML} . Under a bilateral conjugate match condition, then the transducer power gain, equation (2.78), then can be simplified to

$$G_{Tmax} = \frac{1}{1 - |\Gamma_{Ms}|^2} |S_{21}|^2 \frac{1 - |\Gamma_{ML}|^2}{|1 - S_{22}\Gamma_{ML}|^2} \quad (3.27)$$

which after substituting equations (3.21) and (3.22) reduces to

$$G_{Tmax} = \frac{|S_{21}|}{|S_{12}|} (K - \sqrt{K^2 - 1}) \quad (3.28)$$

As a side note, under bilateral conjugate match conditions $G_T = G_P = G_A$, which also implies $G_{Tmax} = G_{Pmax} = G_{Amax}$.

There will always be cases where the transistor is not unconditionally stable, but it is desirable to know what the maximum gain could be and still be stable. Under this scenario then a different type of measure is used to determine what the maximum stable gain is. The maximum gain is simply found by

$$G_{MSG} = \frac{|S_{21}|}{|S_{12}|} \quad (3.29)$$

3.3.1 Design Procedure

The design procedure for unconditionally stable transistor is quite straight forward.

1. Compute Γ_{Ms} and Γ_{ML} from equations (3.21) and (3.22) respectively.
2. Design an appropriate matching network to the values found for Γ_{Ms} and Γ_{ML} .

The properties associated with a bilateral conjugate matched amplifier are $SWR_{in} = 1$, $SWR_{out} = 1$, and that the maximum gain possible is being used. If these are the design criteria then this is the ideal, but it will be shown later that an amplifier in this configuration will not always have the best noise performance. Also, a bilateral conjugate match will not always be possible, so other design methods must be developed to achieve the desired operating performance.

3.4 Operating Power Gain

When the bilateral conjugate match is not necessary or if the transistor is potentially unstable for the desired operating frequency, then a design utilizing operating power offers a viable alternative. Gonzalez says that since the operating power gain is inde-

pendent of the source impedance, then this is the recommended method for practical amplifier design [3].

Starting with equation (2.80) and substituting equation (2.69) for Γ_{IN} gives

$$G_P = \frac{|S_{21}|^2(1 - |\Gamma_L|^2)}{\left(1 - \left|\frac{S_{11} - \Delta_s \Gamma_L}{1 - S_{22} \Gamma_L}\right|^2\right) |1 - S_{22} \Gamma_L|^2}$$

which is rewritten in compact form

$$G_P = |S_{21}|^2 g_p$$

where

$$\begin{aligned} g_p &= \frac{1 - |\Gamma_L|^2}{|1 - S_{22} \Gamma_L|^2 - |S_{11} - \Delta_s \Gamma_L|^2} \\ &= \frac{1 - |\Gamma_L|^2}{1 - |S_{11}|^2 + |\Gamma_L|^2(|S_{22}|^2 - |\Delta_s|^2) - 2\text{Re}(\Gamma_L C_2)} \end{aligned} \quad (3.30)$$

Notice that g_p is a function of Γ_L and the transistor S-parameters. Since the S-parameters are fixed for a given frequency, the only true variable is Γ_L ; thus, it necessary to find which values of Γ_L will give the desired value for g_p . A mapping of these values of Γ_L will be graphed on a Smith chart to aid in the design. A derivation of these equations is in Appendix G of [3] and are merely presented here. The radius of the circle drawn will be denoted as r_p and the location of the circle center is given as C_p . Mathematically

$$|\Gamma_L - C_p| = r_p \quad (3.31)$$

where

$$C_p = \frac{g_p C_2^*}{1 + g_p(|S_{22}|^2 - |\Delta_s|^2)} \quad (3.32)$$

and

$$r_p = \frac{\sqrt{1 - 2K|S_{21}S_{12}|g_p + |S_{21}S_{12}|^2g_p^2}}{1 + g_p(|S_{22}|^2 - |\Delta_s|^2)} \quad (3.33)$$

3.4.1 Design Procedure

There are two possible cases that must be considered when designing by this method and those are if the transistor is unconditionally stable or not.

Unconditionally Stable

1. Choose a $G_P < G_{Pmax}$ and find g_p .
2. Find the center and the radius for the given operating power gain by equations (3.32) and (3.33) respectively and plot the circle onto the Γ_L Smith chart.
3. Select the desired Γ_L somewhere along the plotted circle.
4. Compute Γ_{IN} from equation (2.69).
5. Maximum power transfer will occur if $\Gamma_{IN}^* = \Gamma_S$ so this is the value that should be chosen for Γ_S .
6. Since both Γ_S and Γ_L have been found, design the matching network to these values.

Conditionally Stable

1. Choose a $G_P < G_{MSG}$ and find g_p .
2. Find the center and the radius for the given operating power gain by equations (3.32) and (3.33) respectively and plot the circle onto the Γ_L Smith chart.
3. Select the desired Γ_L somewhere along the plotted circle that is inside the Smith chart and away from the instability circle.

4. Compute Γ_{IN} from equation (2.69).
5. Since the transistor is only conditionally stable, the choice of $\Gamma_{IN}^* = \Gamma_S$ may not be available due to the fact that the region of instability may be covering that value of Γ_S . Instead choose a value of Γ_S far enough away from the stability circle but still somewhat near Γ_{IN}^* . This will help keep SWR_{in} as small as possible.
6. Once a value for Γ_S has been chosen, then design the input and output matching networks for the values found for Γ_S and Γ_L .

Suppose that Γ_L is chosen such that $\Gamma_L \neq \Gamma_{ML}$ for the design specification of some $G_P < G_{Pmax}$. This will create an $SWR_{out} > 1$. This shows that there is a trade-off in one parameter for another, where in this case the gain is desired to be reduced but has the overall effect of changing SWR_{out} . These decisions will be the theme throughout any transistor amplifier design.

Also, the choice of an operating power gain does not imply that is the total gain of the amplifier. Instead it will be the maximum possible gain the amplifier could have. This is also true for the available power gain scenario to be discussed in the next section. The true gain of the amplifier is the transducer power gain, G_T , and corresponds to the measured S_{21} on a network analyzer.

3.5 Available Power Gain

Available power gain design procedures are similar to that of the operating power gain. The difference lies in the fact that available power gain can be found to be independent of the load impedance attached.

Starting with equation (2.83) and substituting equation (2.70) for Γ_{OUT} gives

$$G_A = \frac{|S_{21}|^2(1 - |\Gamma_S|^2)}{\left(1 - \left|\frac{S_{22} - \Delta_s \Gamma_S}{1 - S_{11} \Gamma_S}\right|^2\right) |1 - S_{11} \Gamma_S|^2} = |S_{21}|^2 g_a \quad (3.34)$$

where

$$g_a = \frac{G_A}{|S_{21}|^2} = \frac{1 - |\Gamma_S|^2}{1 - |S_{22}|^2 + |\Gamma_S|^2(|S_{11}|^2 - |\Delta_s|^2) - 2\text{Re}(\Gamma_S C_1)}$$

The term g_a is a function of only Γ_S and the transistor S-parameters. Similar to the operating power design, the only true variable for the available power gain is Γ_S since the S-parameters are fixed for a given frequency. Circles will be drawn onto a Smith chart to find all values of Γ_S which will solve for the desired g_a . The radius of the circle and center of the circle will be defined as r_a and C_a respectively and are found to be

$$r_a = \frac{\sqrt{1 - 2K|S_{21}S_{12}|g_a + |S_{21}S_{12}|^2g_a^2}}{|1 + g_a(|S_{11}|^2 - |\Delta_s|^2)|} \quad (3.35)$$

$$C_a = \frac{g_a C_1^*}{|1 + g_a(|S_{11}|^2 - |\Delta_s|^2)|} \quad (3.36)$$

3.5.1 Design Procedure

There are two possible cases again that one must consider and those are if the transistor is unconditionally stable or not. Both procedures are similar, however the choices for Γ_S and Γ_L are limited in the conditionally stable case.

Unconditionally Stable

1. Choose a desired $G_A < G_{Amax}$ then find g_a .
2. Compute r_a and C_a from equations (3.35) and (3.36) respectively and plot the circle onto the Γ_S Smith chart.

3. Select a value for Γ_S along the constant gain circle.
4. Compute Γ_{OUT} from equation (2.70).
5. Choose $\Gamma_{OUT}^* = \Gamma_L$ since this maximizes the power transfer as well as makes $SWR_{out} = 1$.
6. Design the matching network from values found for Γ_L and Γ_S .

Conditionally Stable

1. Choose a desired $G_A < G_{MSG}$ then find g_a .
2. Compute r_a and C_a from equations (3.35) and (3.36) respectively and plot the circle onto the Γ_S Smith chart.
3. Select a value for Γ_S along the constant gain circle which is in the stable region and is not too close to the unstable region circle.
4. Compute Γ_{OUT} from equation (2.70).
5. Choose a Γ_L in the stable region near Γ_{OUT}^* but not too close to the stability circle. This will help to minimize SWR_{out} .
6. Design the matching network from the values found for Γ_L and Γ_S .

3.6 SWR Circles

As was seen in the potentially unstable cases for the designs of both the operating and available power gains, a conjugate match could not be obtained in most cases. Instead the designer had to choose between amplifier gain versus SWR. Suppose that there was a specification for a fixed SWR at either the input or output of the amplifier, then it would be nice to know for which values of Γ_L or Γ_S would give that SWR to meet the

design requirements. Since all designs are done on the Smith chart, it is desired to find an equation which will draw constant SWR circles onto their respective planes; SWR_{in} circles will be plotted on Γ_S and SWR_{out} circles will be plotted on Γ_L .

The equations used to draw the constant SWR_{in} circles on the Γ_S plane are

$$C_{V_i} = \frac{\Gamma_{IN}^*(1 - |\Gamma_a|^2)}{1 - |\Gamma_a\Gamma_{IN}|^2} \quad (3.37)$$

$$r_{V_i} = \frac{|\Gamma_a|(1 - |\Gamma_{IN}|^2)}{1 - |\Gamma_a\Gamma_{IN}|^2} \quad (3.38)$$

where C_{V_i} is the circle center location, r_{V_i} is the radius of the circle, and

$$|\Gamma_a| = \frac{SWR_{in} - 1}{SWR_{in} + 1} \quad (3.39)$$

The circles that are drawn represent the all of the values for a chosen Γ_S which will give a constant SWR_{in} at the desired level.

Similarly for the output, the equations are

$$C_{V_o} = \frac{\Gamma_{OUT}^*(1 - |\Gamma_b|^2)}{1 - |\Gamma_b\Gamma_{OUT}|^2} \quad (3.40)$$

$$r_{V_o} = \frac{|\Gamma_b|(1 - |\Gamma_{OUT}|^2)}{1 - |\Gamma_b\Gamma_{OUT}|^2} \quad (3.41)$$

where C_{V_o} is the location of the constant SWR_{out} circle center, r_{V_o} is the radius of this circle, and

$$|\Gamma_b| = \frac{SWR_{out} - 1}{SWR_{out} + 1} \quad (3.42)$$

The circles that are drawn represent those values for a chosen Γ_L which will give a constant SWR_{out} at a desired value.

3.6.1 Design Procedure

Designing for a desired SWR value can be quite complicated. There many different values that one can chose for either Γ_S or Γ_L , all which will have an effect throughout the design. The procedure is more or less a guideline since there is no set way to do this.

Suppose that SWR_{in} is the restricted parameter in the design and SWR_{out} is less important. The general procedure then would be as follows:

1. Choose a value for G_P such that $G_P < G_{Pmax}$ if the transistor is unconditionally stable or $G_P < G_{MSG}$ if the transistor is conditionally stable.
2. Find the center and the radius for the given available power gain by equations (3.32) and (3.33).
3. Choose a value for Γ_L along the G_P circle that is in the stable region.
4. Compute Γ_{IN} from equation (2.69).
5. The ideal choice for Γ_S is the case where $\Gamma_S = \Gamma_{IN}^*$. That choice may not be available since the transistor may be conditionally stable. If SWR_{in} has a design limit, use that value in equation (3.39) to solve for $|\Gamma_a|$
6. Use equations (3.37) and (3.38) to compute the constant SWR_{in} circle. Choose a value for Γ_S that is in the stable region not too close the stability circle.
7. Compute Γ_{OUT} from equation (2.70) using the value selected for Γ_S .
8. Compute SWR_{out} by first finding $|\Gamma_b|$ from equation (3.18) then use equation (3.16).

As can be seen, the procedure is quite involved. Even worse, initial choice for Γ_L and the final answer of Γ_{OUT} may cause an unacceptable result for the output

SWR. In this case then one would have to start almost completely over to find values for Γ_L and Γ_{OUT} which would provide an acceptable result for SWR_{out} . However, there will still be a limit to how well SWR_{out} can be. One way to achieve a better SWR_{out} is to loosen the requirements on the input SWR. This will only work so well, but it illustrates again this concept of give and take on operating parameters for the amplifier.

A similar procedure is developed in the case where the output SWR needs to be a certain value.

1. Choose a value for G_A such that $G_A < G_{Amax}$ if the transistor is unconditionally stable or $G_A < G_{MSG}$ if the transistor is conditionally stable.
2. Find the center and the radius for the given available power gain by equations (3.36) and (3.35).
3. Choose a value for Γ_S along the G_A circle.
4. Compute Γ_{OUT} from equation (2.70).
5. The ideal choice for Γ_L is the case where $\Gamma_L = \Gamma_{OUT}^*$. That choice may not be available since the transistor may be conditionally stable. If SWR_{out} has a design limit, use that value in equation (3.42) to solve for $|\Gamma_b|$
6. Use equations (3.40) and (3.41) to compute the constant SWR_{out} circle. Choose a value for Γ_L that is in the stable region not too close the stability circle.
7. Compute Γ_{IN} from equation (2.69) using the value selected for Γ_L .
8. Compute SWR_{in} by first finding $|\Gamma_a|$ from equation (3.15) then use equation (3.16).

3.7 Noise Figure Circles

One final design consideration given for transistor amplifiers is the noise figure. In some applications, such as telecommunications, it is desirable to have a low noise receiver so that a minimal amount of noise is added to the signal for processing of the information. In this regard, a designer may deem the noise performance more important than those discussed above which again adds another layer of complexity to the design of the amplifier.

The noise figure of a two port amplifier is found to be

$$F = F_{\min} + \frac{r_n}{g_s} |y_s - y_{\text{opt}}| \quad (3.43)$$

The derivation of this equation is found in Appendix L of [3]. The goal is to manipulate equation (3.43) in a way such that it can be plotted onto the Smith chart such that a graphical approach can be maintained in the design. The first step is to express y_s and y_{opt} as reflection coefficients defined as

$$\begin{aligned} y_s &= \frac{1 - \Gamma_S}{1 + \Gamma_S} \\ y_{\text{opt}} &= \frac{1 - \Gamma_{\text{opt}}}{1 + \Gamma_{\text{opt}}} \end{aligned}$$

Substituting these equations into (3.43) gives

$$F = F_{\min} + \frac{4r_n |\Gamma_S - \Gamma_{\text{opt}}|^2}{(1 - |\Gamma_S|^2) |1 + \Gamma_{\text{opt}}|^2} \quad (3.44)$$

The terms F_{\min} , r_n , and Γ_{opt} are all provided on the data sheets for a given RF transistor. Note that sometimes F_{\min} is called Nf_0 by some manufacturers. Allowing

$F = F_i$, where F_i is the desired noise figure and using

$$N_i = \frac{F_i - F_{\min}}{4r_n} |1 + \Gamma_{\text{opt}}|^2 \quad (3.45)$$

in equation (3.44) gives

$$\frac{|\Gamma_S - \Gamma_{\text{opt}}|^2}{1 - |\Gamma_S|^2} = N_i \quad (3.46)$$

where N_i is referred to as the noise figure parameter. Equation (3.46) can be manipulated to

$$\left| \Gamma_S - \frac{\Gamma_{\text{opt}}}{1 + N_i} \right|^2 = \frac{N_i^2 + N_i(1 - |\Gamma_{\text{opt}}|^2)}{(1 + N_i)^2} \quad (3.47)$$

which is easily identified as a circle centered at

$$C_{F_i} = \frac{\Gamma_{\text{opt}}}{1 + N_i} \quad (3.48)$$

with radius

$$r_{F_i} = \frac{1}{1 + N_i} \sqrt{N_i^2 + N_i(1 - |\Gamma_{\text{opt}}|^2)} \quad (3.49)$$

Also from the left hand side of equation (3.47), the noise figure circles are plotted on the Γ_S plane. To use equations (3.48) and (3.49), equation (3.45) is first used to find a value for N_i . Equation (3.45) requires the desired design noise figure F_i must be chosen such that it is greater than F_{\min} .

3.7.1 Design Procedure

If noise is the most important aspect of a design and must be at a minimum, then the choice for Γ_S is quite easy. It is given on the data sheet. The procedure for available gain circles is then followed. The draw back to this choice for Γ_S is that the input SWR may be beyond the desired specification. If the restrictions on the noise figure are relaxed, then one can tune the other performance characteristics.

There is no true procedure that can be broken down into discrete steps for this design. Instead, adapting the idea that the noise figure circles are more of a guide will be beneficial. The only part that can be done procedurally is as follows

1. Choose a value for a target noise figure (F_i).
2. Compute N_i from equation (3.45).
3. Compute the circle center and radius from equations (3.48) and (3.49) respectively.
4. Plot the circle on the Γ_S plane.

The next steps (such as gain and SWR) are up to the designer as to which aspects are most important. Use any of the above procedures as a guide on what to do next. The only addendum to the above procedure is that the choices for Γ_S should be near the desired noise figure circle if all other choices for Γ_S are otherwise equally as good.

CHAPTER 4

Software Tools with Updated Design Procedures

Last chapter dealt with design procedures which were to be done by hand on Smith charts. As was touched on before, this technique can be, and is, quite tedious. Even after going through one of the design procedures mentioned, the results may be unsatisfactory for the design requirements which leads to going through the procedure again however many times until an acceptable design is found. This sort of repetitive nature of the above design process lends itself perfectly to the use of a computer application to aid in the design.

4.1 Using the Design Program

Matlab was chosen for the computing environment due to its presence on university campuses. In addition to having the main program files, there are some thirty other files necessary for various function calls and scripts. Ensure that the folder location for all of the files has been added to the path, or change the active directory to this folder location. Once the path has been set, type “AmpDesigner.m” in the command window. The GUI then opens as well as two other figures which have more detailed

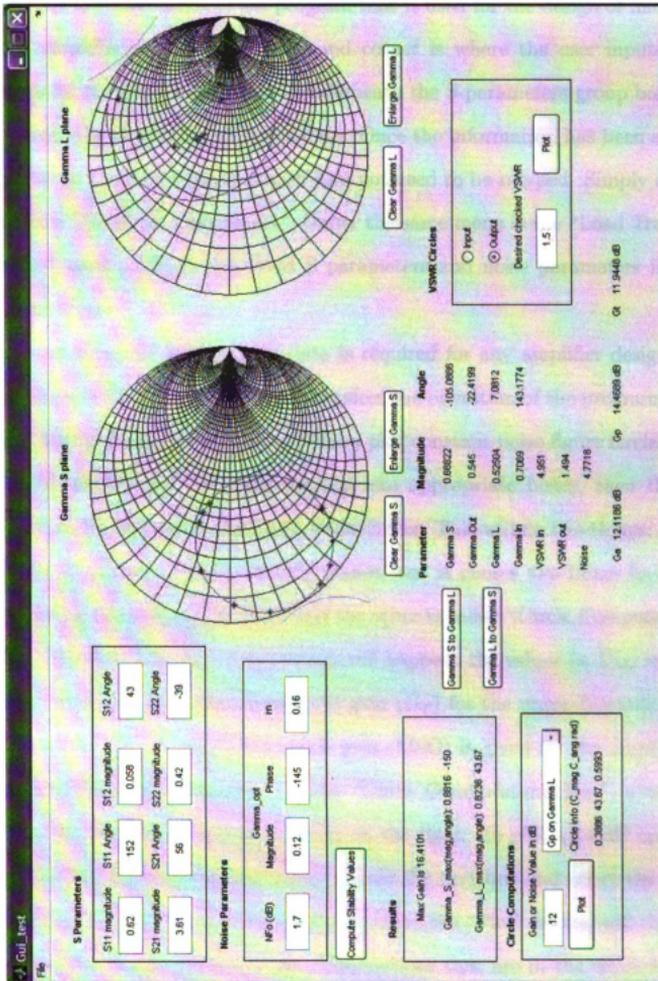


Figure 4.1. Screenshot of the Microwave Amplifier Design Program

Smith charts for Γ_S and Γ_L .

Figure 4.1 is a screenshot of the program that is used for the design of microwave transistor amplifiers. The upper left hand corner is where the user inputs the S parameters for the desired frequency. Underneath the S-parameters group box is the place to enter the the noise parameter data. Once the information has been entered, it is possible to save the data so that it does not need to be retyped. Simply click on “File → Save Transistor Parameters.” Under the same menu is the “Load Transistor Parameters” used to fill in the saved S parameters and noise parameters into the appropriate boxes.

As a minimum, the S parameter data is required for any amplifier design. The noise parameters are optional and will not affect the operation of the program if they are blank; the user will simply not be able to plot constant noise figure circles.

Once the information has been entered into appropriate boxes, then the user should click on the “Compute Stability Values” box. This will do two things; it plots the stability circles onto the Γ_S and Γ_L planes, and it causes two boxes to appear. One of these boxes is called “Results” and the other is called “Circle Computations”. Inside the “Results” box, one of two things will happen: the values for Γ_{Ms} and Γ_{ML} will appear with the associated transducer gain (G_T) for the unconditionally stable transistor, otherwise the maximum stable gain (MSG) is given for the amplifier for the conditionally stable transistor. In the “Circle Computations box”, a text box appears on the left with a pull-down menu on the right. To plot a 14 dB operating gain circle on the Γ_L plane, simply enter 14 into the text box and select the proper option on the pull down menu to the right. Pressing the “Plot” button will then plot the circle on the on appropriate plane. Two options that are in the window which are not discussed are the “Ga on Gamma L” and “Gp on Gamma S” options. As the name implies, these are for plotting the gain circles on the plane other than what they are normally plotted on. Consult [3] on designs using these circles.

There is a color code to the circles that are plotted on Γ_S and Γ_L . Maroon circles are plotted for G_A on Γ_S and G_P on Γ_L . Green circles are plotted for G_A on Γ_L and G_P on Γ_S . Yellow circles represent constant SWR circles. Finally, noise circles are graphed in red.

Underneath the Γ_S plane, there are two buttons: “Gamma S to Gamma L” and “Gamma L to Gamma S”. These are used to activate the continual updating feature of the cursor’s location on either the Γ_S or Γ_L plane. Use of this feature will be discussed more in detail later.

At the bottom right is where the user will input a desired input or output SWR value and selects which type to plot via the radio buttons. It is important to note, that a gain is required for the plotting of these functions. The way that this feature operates is that it reads the value from the gain box at the bottom left of the GUI inside the “Circle Computations” box prior to computing the SWR circle. When these constant SWR circles are plotted, the user will see ten red stars appear on both Γ_S and Γ_L . These points represent the ideal solutions for a given constant SWR. For example, if a constant SWR circle is plotted on Γ_L for a specific G_P , then on the Γ_L plane, these red stars appear at various points along the constant gain circle. The ten points on along the constant gain circle have a one-to-one correspondence to those values that appear on the Γ_S plane and represent the minimal values for SWR_{in} for a constant SWR at the output.

There are two buttons under each Smith chart. The “Clear Gamma X” button is for clearing that Smith chart if it becomes too cluttered in both the figure and main GUI. The button called “Enlarge Gamma X” is there in case the figure that represents that figure is closed, then this button provides a means of recreating that figure in the background.

4.2 Design using the GUI

As was discussed in Chapter 3, there are two ways to design an amplifier: start with either G_A or with G_P and then incorporate the SWR limitation if needed. Below is a procedure starting with a desired G_P .

1. Enter the values for the transistor S parameters in the appropriate box. If available, enter the noise figure data as well. It is recommended that these values be saved.
2. Press the “Compute Stability Values”.
3. If the transistor is unconditionally stable and that is what is desired, then simply read the values for Γ_{Ms} and Γ_{ML} ; your design is done. If a gain other than the MAG is desired or for noise considerations, then continue on with the procedure.
4. Choose a value for G_P in dB less than the gain shown in the “Results” box and enter it into the “Circle Computations” text box.
5. On the pull-down menu select “Gp on Gamma L” and click on “Plot”. This graphs the constant operating power gain circle onto the Γ_L plane. Repeat this step as many times as necessary for different gain values.
 - (a) If noise figure data has also been entered, then enter noise figure values greater than those that have been entered into the box labeled “Nfo (dB)”.
6. Since the procedure being considered is using G_P as a starting point and is plotted on Γ_L , then click on the button labeled “Gamma L to Gamma S”.
7. The user should notice that the value for “Gamma L” is now changing in the middle of the GUI with the location of the cursor. Choose some point along

the desired constant G_P circle by right clicking on the Γ_L Smith chart. Right clicking on Γ_L will fix the value for Γ_L , and now shifts control to the Γ_S plane. Also, the user should see that the values for Γ_S , Γ_{OUT} , Γ_{IN} , SWR_{in} , SWR_{out} , and noise are now all changing as the cursor changes its location.

8. To plot a constant SWR circle, ensure that value for the gain circle that was clicked on in Γ_L is in the gain value box of the lower left hand corner. Ensure the radio button for “Input” is selected in the “VSWR Circles” box and select a value. This step can be repeated any number of times by selecting a different Γ_L . Simply right click on a new value for Γ_L and click on “Plot” again.
9. Choose a value for Γ_S based on the circles that have been plotted by pressing the space bar. If noise is a consideration, choose the value which has a red star near the desired constant noise figure circle. Pressing the space bar will fix all of the values that are updated on the GUI as well as tell the designer what the values are for G_T , G_A , and G_P .
10. The two Smith charts in the background have been updating all actions that have been occurring on the main GUI. At this point, the designer may wish to print off these Smith charts to complete the design of the amplifier.

That is all for the procedure, but there are few things on which to comment. First, notice that the value for G_P is the same as was used in the design. This would simply be just a self check for the user to ensure that points chosen during the design procedure are indeed the value which was desired.

Also note that if a point for Γ_L is chosen on one of the red stars that appear during the plotting of a constant SWR circle, the new constant SWR_{in} circle touches only one of these points. If the user were to move to this choice for Γ_L to an adjacent red star in the clockwise direction and plotted a new constant SWR_{in} circle, then the new

SWR_{in} circle would also be adjacent to the previous choice in a clockwise direction as well.

Finally, when constant SWR circles are graphed and have centers that are in the origin, then that represents a match at either the input or the output (i.e. $\Gamma_S = \Gamma_{IN}^*$ or $\Gamma_{OUT} = \Gamma_L^*$). Specifically this can be shown to be true if the following equation constant input SWR equation is examined

$$\left| \Gamma_S - \frac{\Gamma_{IN}^*(1 - |\Gamma_a|^2)}{1 - |\Gamma_a \Gamma_{IN}|^2} \right| = \frac{|\Gamma_a(1 - |\Gamma_{IN}|^2)|}{1 - |\Gamma_a \Gamma_{IN}|^2} \quad (4.1)$$

where Γ_a is defined as it is in equation 3.15 and is rewritten here for convenience.

$$\Gamma_a = \left| \frac{\Gamma_{IN} - \Gamma_S^*}{1 - \Gamma_{IN} \Gamma_S} \right| \quad (4.2)$$

Since a calculation for constant SWR_{in} circles is being considered, a value for Γ_L has likely been chosen and Γ_{IN} has been calculated and is fixed. When the user is moving the cursor, the variable that is changing in equation 4.1 is Γ_S . As can be seen in equation 4.2, $\Gamma_a = 0$ if $\Gamma_S^* = \Gamma_{IN}$. When $\Gamma_a = 0$, the term on the right hand side of equation, or the radius, 4.1 goes to zero. Also, the second term on the left hand side of equation 4.1 equals Γ_{IN}^* since $\Gamma_a = 0$ and the denominator equals one. A similar proof can be given for the constant output SWR circle equation.

The design procedure for constant G_A circles is essentially the same as above. The only difference would be to click on the “Gamma S to Gamma L” button. The advantage of starting with available gain circles is that it allows the designer to choose Γ_S immediately, thereby setting the noise figure at the desired value.

4.3 Circuit Design

Once the designer has the two design values for Γ_S and Γ_L , then it is up to the designer as how to implement the design; will the design be done on microstrip or will the design be done using lumped elements. Each design technique will be discussed here.

4.3.1 Microstrip Implementation

If the designer is implementing a microstrip based design, then the Smith charts which have been plotting in the background can be used. These Smith charts have degree measurements around the perimeter as well as a linear scale at the bottom to accommodate the use of a compass.

When using the Smith chart to match to either Γ_S or Γ_L , take note on the definition of these terms (Figure 3.2); they are the reflection coefficients looking into the matching network and terminate at the system characteristic impedance. This means that the first rotation will be in the “Wavelengths towards load” direction from this impedance/admittance. Depending on what type of network being used, the goal is to move all of the components added toward the center of the Smith chart.

The values necessary for the matching network should be extracted such as the lengths in fraction of wavelengths as well as what type of transmission lines used. Next, find the effective wavelength of the for the design frequency on the board to be used. This allows the designer to turn the normalized lengths into actual lengths. If using impedances other than those at the system impedance, such as for quarter wave transformers, use equation 2.32 to calculate the necessary transmission line width.

Ansoft Designer

This next step is optional, but is recommended as a sort of verification of the design. For this thesis, a program called Ansoft Designer has been used for this step.

When the program opens, by default a new project opens. Click on the icon which looks like a transistor at the top. This opens a dialog box which asks what type of microstrip is being used. If the type is not there that is being used, then choose any type that is listed. It can be overridden later.

A blank schematic sheet now appears as well as more icons at the top of the page. The next thing to do is add a two port network to the schematic. Click on 'Draw→N-Port...'. A dialog box appears. On the first tabbed sheet, there are three radio buttons under 'Data Source'. Select 'Enter data in spreadsheet'. Select the tab labeled 'Network Data'. Select the proper units for the frequency. On the first row, first column of the spreadsheet enter the frequency that the operation will be performed. Tab to the next square and enter S_{11} in that block formatted as the magnitude-space-angle. Do this for the other S parameters noting that S_{12} is in the first row, last column and S_{21} is in the second row, middle column. Select the tab 'Noise Data' if there is noise data to enter. Press 'OK' and place the network on the schematic.

On the left hand side of the program, there are a set of three tabs labeled, project, components, and search. Click on 'Components'. A list of different types of elements then become available including microstrip elements and lumped elements. Select the folder labeled microstrip. There are three options here that will be of use to the designer: shorted stubs, open stubs, and transmission lines. Add whatever elements are necessary to the schematic. Select the type out of these three menus that are denoted as near ground and electrical length.

Double click on any one of the transmission lines just added. This will bring up another dialog box. Click on the box on the line labeled 'SUB'. This is where the microstrip specific information can be added. This will pop up another dialog box. Click on 'New'. Enter all of the pertinent information for the board being used. Click 'Ok' twice. Finally enter the information necessary for all of the microstrip lines such

as electrical length and width.

Next, RF sources need to be added. Select 'Draw→Interface Port' and add two of these. Double click on each of these ports and select the radio button which turns the port into a microwave port.

After connecting all of the components, it is time to simulate the circuit. Under the menu 'Circuit', select 'Add Analysis Setup...'. Click on 'Next'. Click on the 'Add' button. Select the radio button labeled 'Single Value' and enter the frequency at which to simulate. Press the button labeled 'Add>>'. Click 'Ok' then 'Finish'. To simulate the circuit, go to 'Circuit→Analyze'. A box at the lower middle will tell if the analysis has been successful.

To view a report, go to 'Circuit→Create Report...' and simply add what variables are of concern. There are options such as G_T , S_{21} , and SWR. Simply add what is desired to look at. If these values are close to what is expected, then the design is as it should be and is ready for the next step.

Sonnet

To model the matching networks on microstrip, the program Sonnet was used to tune the network to the desired impedance values. This was done by drawing the geometry to the approximate values given by using the Smith charts. When using Sonnet, it is important make sure that the proper dielectric is being used and that the cell sizes are reasonable for the given frequency. Also, make sure that there is sufficient space all around the board to minimize interactions with the edges as well as for leaving room for changing the transmission line lengths as will be discussed later.

Place two ports onto where the transmission line will be attached and where the transistor will be attached. Double click on the port that will be attached to the transistor. Make the port an autoground since the port is fed from the ground plane to the trace. This allows the user to modify the port impedance. To figure out

the values to be entered for the impedance, simply enter $\text{Re}[\Gamma_S]$ times the characteristic impedance in the 'Resistance' block and enter $-\text{Im}[\Gamma_S]$ times the characteristic impedance into the reactance block. The simulations should show how well the initial design compares to the actual layout.

Sonnet also comes with an optimization feature. To use this feature, there must be elements on the design called 'parameters' for Sonnet to vary. To add a parameter, go to 'Tools→Add Parameter'. From there the user has the choice of either anchored or symmetric. The anchored option sets one point which will not move from that point on the board and uses this as a reference when changing the other dimensions. The symmetric case changes the lengths equally in opposite directions about it's midpoint.

Once the parameter components have been placed, it is required to setup the program to optimize. Go to 'Tools→Setup...'. Under the box labeled "Analysis Control", select the option called Optimization. This will cause two other boxes underneath to appear. The upper box is called 'Parameters'. This is where the user will set the constraints on the parameters that have been added to the drawing. Select the 'Edit' feature. Another dialog box appears and allows the user to select which length will be varied as well as what values to limit the possible lengths. Once the constraints have been entered, click on 'OK'. The second box which appears is the place where the user enters the design goals. Next, the user should click on 'Add...'. Here the user can enter a frequency range over which to optimize the design as well as what the design goal should be. It is suggested that only one frequency be entered and that S_{11} be set to less than -30dB.

If a designed stub happens to be a shorted stub, then it is possible to create a short by adding a via to the board in Sonnet. When using the optimization routine, ensure that all corners of the via have been selected while adding a parameter. This will move the via and maintain it's position at the edge of the stub.

Once networks have been developed, then merge all of the components onto one

board to be made. There are practical aspects to be considered before completing the design. First, the vias that were used to simulate shorts for the shorted stubs need to be removed. If a DC bias is applied to the shorted stub, then there will be a short circuit to ground. The way to get around this problem is to make the shorted stubs open but then attach a bypass capacitor from the stub edge to a grounded pad near the edge of the stub. This will preserve the DC bias, but will act as a short at the higher operating frequency of the amplifier.

4.3.2 Lumped Element Design

Microstrip designs start being practical when the design frequency is above 1GHz, because the free-space wavelength is about 30cm. This is still large, but the wavelength can be significantly reduced by choosing microstrip board with higher dielectric values, but still may be large. Thus, for frequencies less than about 1GHz, it makes sense to use lumped elements.

After obtaining the design values for Γ_S and Γ_L , then it is better, if not a necessity, to use an admittance-impedance Smith chart (or sometimes called ZY Smith chart.) A ZY Smith chart has a normal impedance Smith chart on as well as a mirror image superimposed on top of the impedance Smith chart.

To explain the reason for using this type of Smith chart, it is better to examine what can be done on the standard Smith chart. The circles which have origins along the central axis represent constant $\text{Re}[Z]$. To add a series reactive element then it is only required that the designer move along the perimeter on one of these circles. To add a shunt reactive element on the standard Smith chart is not as easy. There are no circles on which to traverse. By adding the admittance equivalent of the Smith chart, then the designs have been reduced to traveling along circles for shunt elements.

The key to the design with lumped element is to start at the design value for Γ_S or Γ_L and move towards the center to the Smith chart. Clockwise motion along a

constant $\text{Re}[Z]$ circle represents adding a series inductor, whereas counterclockwise rotation represents adding a series capacitor. Likewise on a constant $\text{Re}[Y]$ circle, clockwise rotation represents adding a shunt capacitor and counterclockwise rotation represents adding a shunt inductor. If starting at the wrong point, then the wrong element will be inserted.

Once the values for the inductors and capacitors are found, then it is possible to test this design in Ansoft as well. Instead of using microstrip elements there is a folder labeled “lumped” which then allows the user to add these types of elements.

CHAPTER 5

Designs and Analysis

A design tool or procedure is only as good as the the finished product that is produced as well as its ease of use. To verify the design tool works, a few designs were made with different transistors and different parameters.

The foundation for any of the designs that are to be built in this manner are the S parameters. If these values are incorrect, then the rest of the design will be wrong; thus, it is important to have accurate S parameter measurement. The S parameters for the Agilent transistor AT-41486 were extracted from the datasheet [9]. This might seem contrary as to the previous statement about accurate S parameter measurement, but also notice that only one design was made using this information. On the other hand, the S-parameters for the MPSH10 and MPSH17 transistors were actually measured for the designs made, and as will be seen, made the measured values very close to the design goals.

5.1 S-Parameter Measurement

Accurate measurement of the transistor S-parameters will improve the accuracy of the design of the amplifier. Figures 5.1 and 5.2 are the bias circuits used in measuring these transistor's S-parameters. This circuit is designed to give a 5V collector-emitter

voltage with a collector current of 10mA. The ports labeled “NA” on either end of the circuit are where the network analyzer was attached. Capacitor C1 is used as a bypass capacitor to effectively short the emitter to ground, and capacitors C2 and C3 are used as DC blocks. Both circuits had identical current gains of approximately 100, so the DC bias circuits end up being the same.

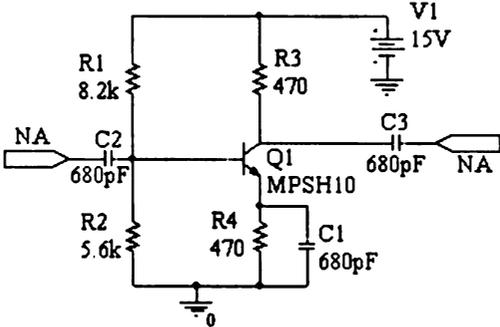


Figure 5.1. DC Bias Circuit Used in S-Parameter Measurement for the MPSH10

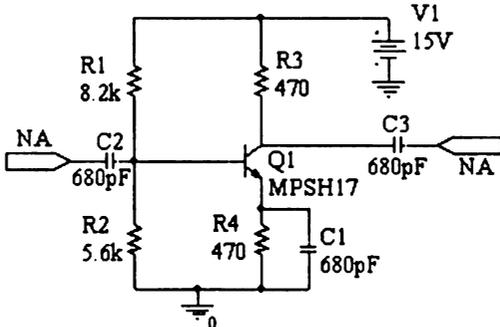


Figure 5.2. DC Bias Circuit Used in S-Parameter Measurement for the MPSH17

Neither of these designs have RF chokes to isolate the transistor from the bias circuitry. This means that the bias circuitry was included in the measurement of the networks S-parameters. This approach seemed to work well for when the matching networks were placed on the network analyzer side of the blocking capacitors and treating the entire amplifier with bias circuitry as a two port network. If the desire is to find the actual transistor S parameters, then it would be necessary to include the RF chokes.

The circuit was built by using the “dead bug” or ground plane technique as discussed in [7]. This technique eliminates most stray capacitance which may appear if the circuit were to be built in a prototyping board or perforated board. A short coaxial cable was then cut and stripped down to expose the center conductor such that a connection could be made to the circuit. The exposed length of the center conductor was minimized so that the cable could maintain its characteristic impedance. The length of the transmission line was short to ensure that the attenuation in the transmission line was negligible.

To ensure that a proper phase measurement was obtained, the center conductor of the coaxial cable was shorted to the outer braided shielding while connected to the network analyzer. Since the shorted transmission line is known to have a reflection of $1\angle 180^\circ$, the user can then change the electrical delay of the network analyzer to this value at the desired frequency. When the cable is open circuited, it should have a reflection coefficient near $1\angle 0^\circ$ and should have angle measures near these points. It is important that both cables be near identical. This means that the lengths of the cable and the amount that has been stripped should be about the same. This should yield similar values for the electrical delay of each cable. If the shorted and opened transmission lines have electrical delay values, then the cables must be adjusted until they agree. Once a value for electrical delay has been found for S_{11} and S_{22} , then ensure that the delays for S_{21} and S_{12} are adjusted to these same values. Figures 5.3

through 5.6 are the measured S-parameters of the circuit shown in 5.1, and Figures 5.7 through 5.10 are the MPSH17 S-parameters measured using the circuit in Figure 5.2.

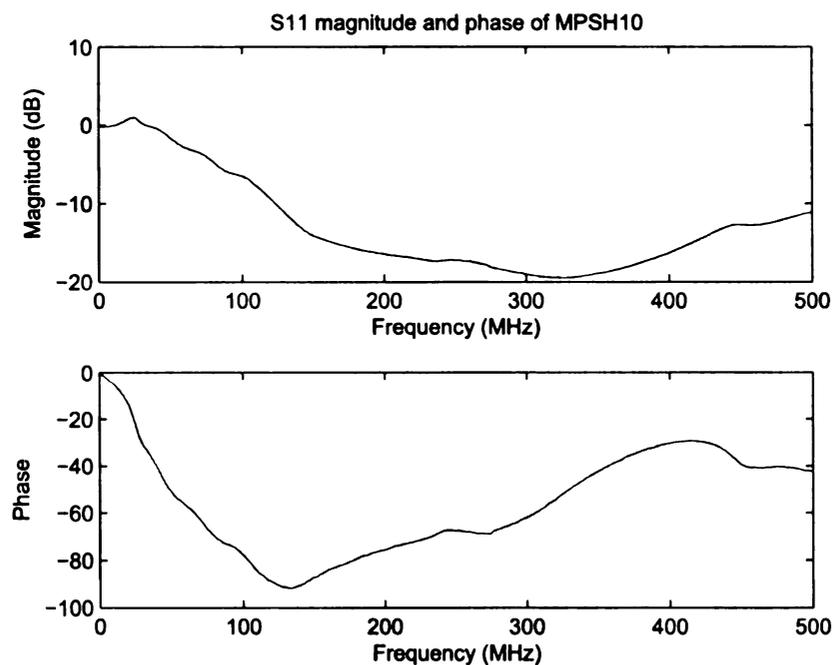


Figure 5.3. S_{11} of MPSH10

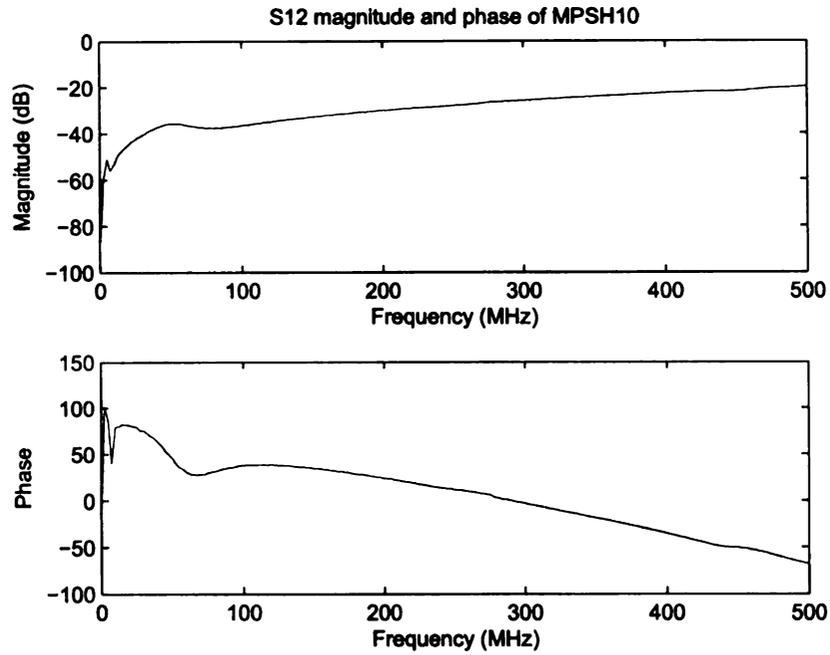


Figure 5.4. S_{12} of MPSH10

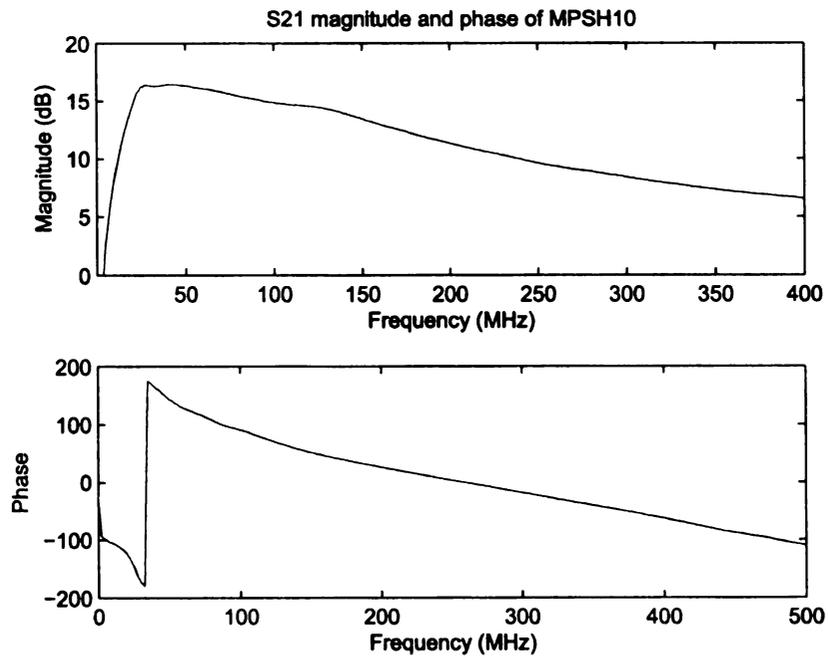


Figure 5.5. S_{21} of MPSH10

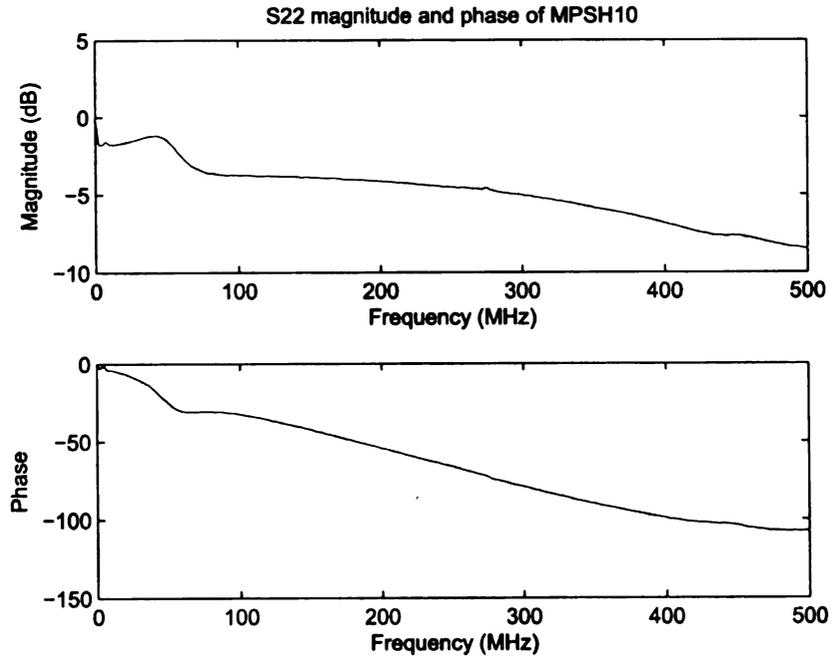


Figure 5.6. S_{22} of MPSH10

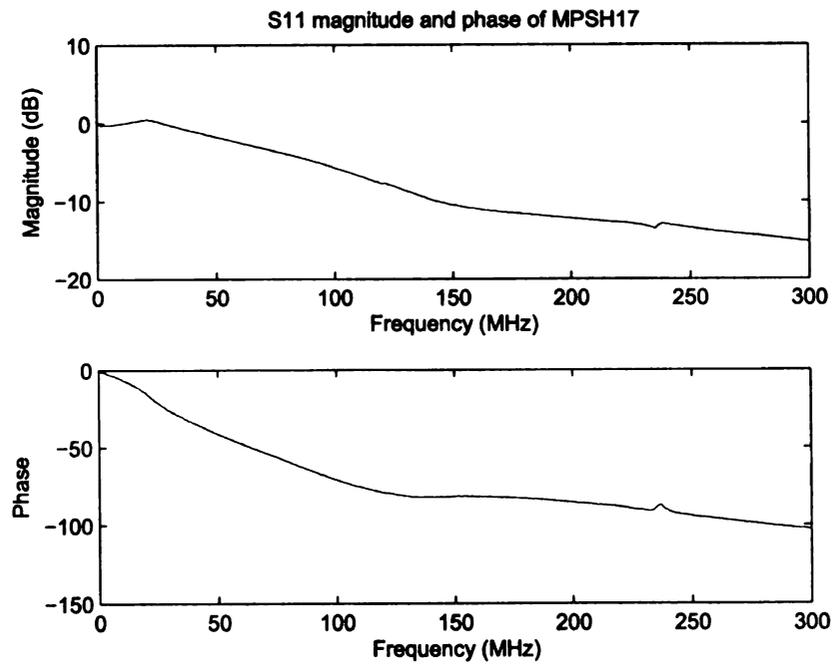


Figure 5.7. S_{11} of MPSH17

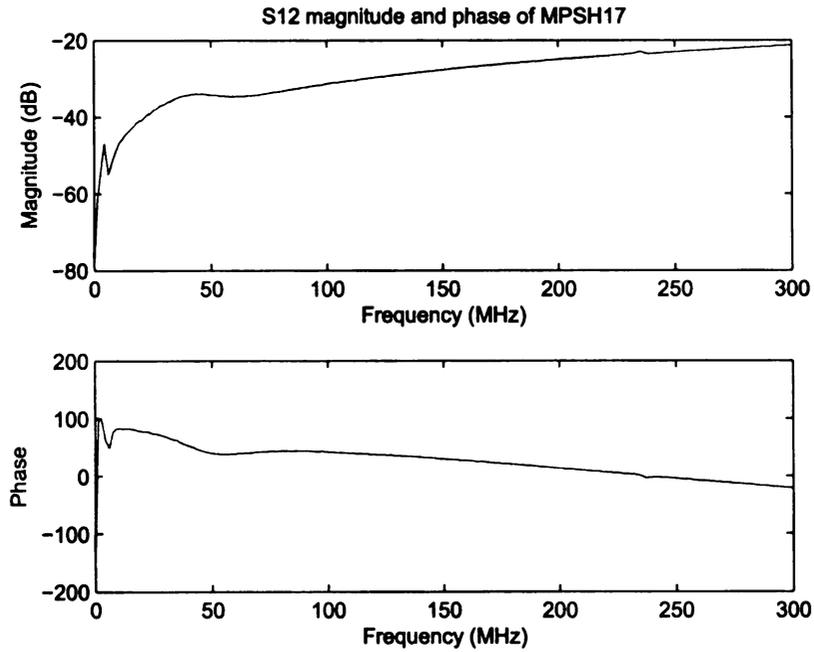


Figure 5.8. S_{12} of MPSH17

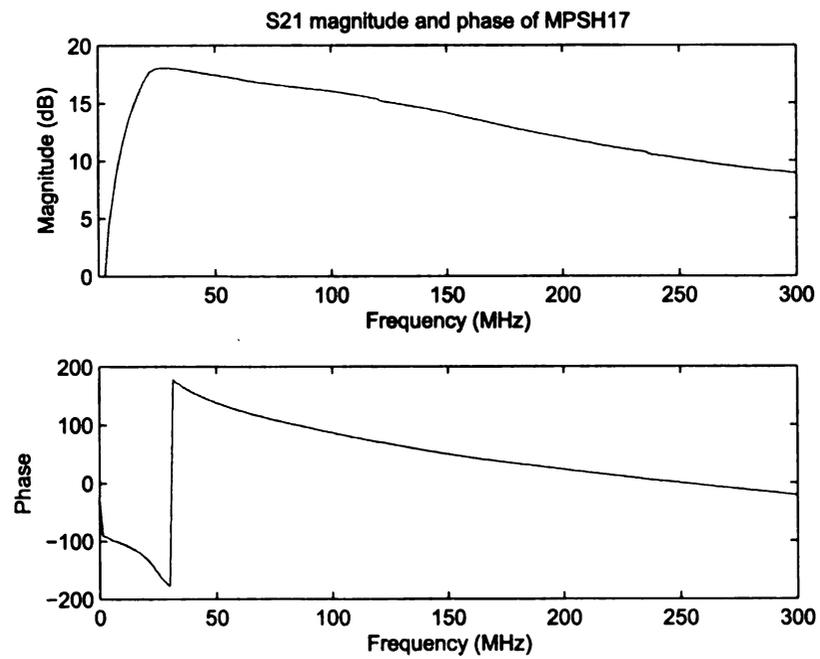


Figure 5.9. S_{21} of MPSH17

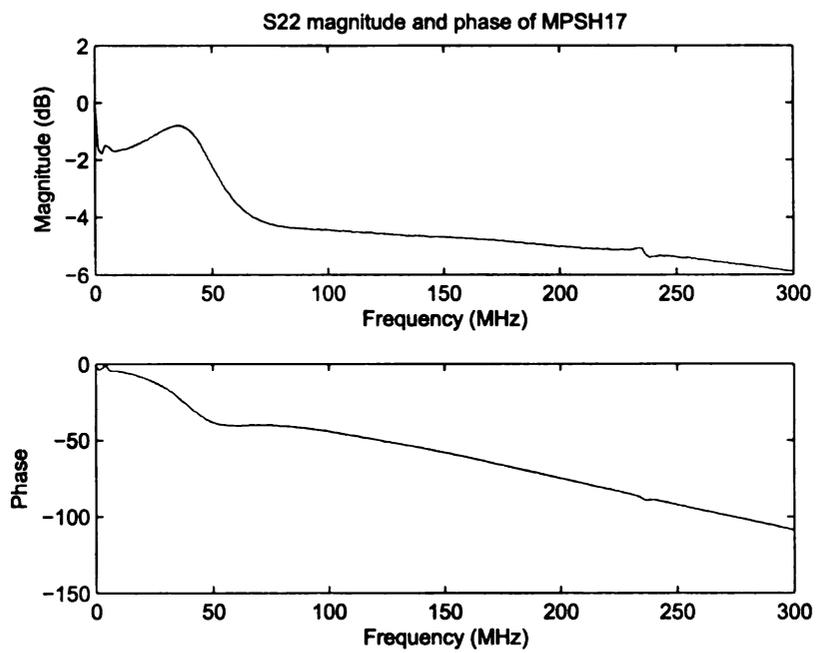


Figure 5.10. S_{22} of MPSH17

5.2 Amplifier Designs

Table 5.1 has extracted S-parameter values for the MPSH10 and MPSH17 transistors, whereas the values for the AT-41486 were from the datasheet [9]. Table 5.2 represents the target design values given the transistor S parameters and the values for Γ_S and Γ_L . The “AmpDesigner” program was used to compute all of the design values.

Transistor	AT-41486 (2GHz)	MPSH10 (100MHz)	MPSH17 (100MHz)
S_{11}	0.62 \angle 152	0.473 \angle -77.6	0.515 \angle -71.0
S_{12}	0.058 \angle 43	0.015 \angle 38.0	0.027 \angle 42.1
S_{21}	3.61 \angle 56	5.55 \angle 91.23	6.335 \angle 86.26
S_{22}	0.42 \angle -39	0.650 \angle -32.42	0.600 \angle -44.2

Table 5.1. Transistor S Parameters used in Designs

Transistor	AT-41486	MPSH10	MPSH10	MPSH17
Center Frequency	2GHz	100MHz	100MHz	100MHz
G_T (dB)	14.43	18.2	14.8	19.2
Γ_S	0.470 \angle -148.2	0.444 \angle 88.3	0.490 \angle 55.6	0.520 \angle -71.0
Γ_L	0.557 \angle 41.1	0.638 \angle 36.4	0.510 \angle -37.5	0.600 \angle 55.6
SWR_{in}	3.07	1	1.5	1
SWR_{out}	1.59	1	5.59	1

Table 5.2. Design Values

5.2.1 AT-41486 Design

Figure 5.11 shows the DC bias circuitry with the RF amplifier attached. This is known as an active biasing circuit since there is another transistor used in the bias circuit. The data sheet states that the bias conditions on the AT-41486 for the S

parameters listed in Table 5.1 required that the collector current be 10mA and a collector-emitter voltage of 8V. This circuit will allow the user to adjust both of these values by adjusting the two potentiometers in the circuit.

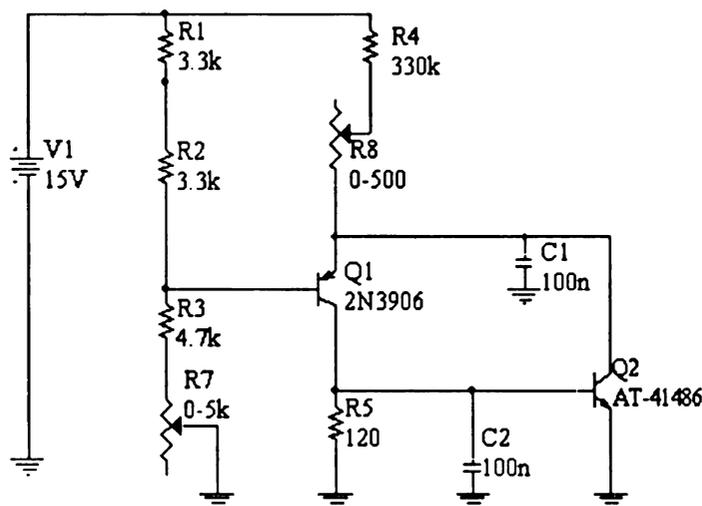


Figure 5.11. Schematic of AT-41486 Amplifier Bias Circuit

This design was done at 2GHz so a microstrip implementation was utilized for the amplifier. Figure 5.12 shows the microstrip layout. The microstrip board is Taconic TLC-32 with a thickness 20 mils. On the left edge is the input to the amplifier. The gap that is seen immediately to the right is for a DC blocking chip capacitor. The length of microstrip before the open stub is simply a 50Ω transmission line. In the center of the board is a gap for the transistor to be placed. The two pieces of microstrip above and below this gap are pads for the transistor to be soldered to and are grounded. On the far right edge is where the output port was attached with a gap for a DC blocking capacitor to be placed again. The stub on the right was to be shorted for this design. To implement a short at the higher frequencies but maintain

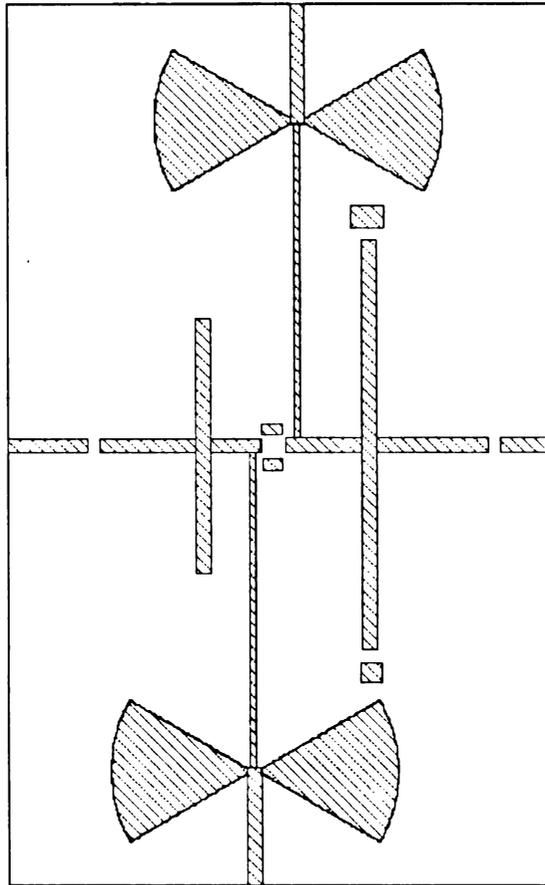


Figure 5.12. Layout of AT-41486 Amplifier on Microstrip

the DC bias level, two grounded microstrip pads were placed on the ends of the stubs where capacitors could be placed to act as a short. Finally, the devices which look like a bow tie are actually broadbanded stubs [1, 8]. These are used in conjunction with high impedance quarter-wave transformers. These elements together create a microstrip version of an inductor, which at high frequencies will block RF energy from getting onto DC supply lines.

The results for the amplifier of Figure 5.12 are shown in Figures 5.13 through 5.15. When comparing the results of these figures to the target values shown in Table 5.2, it is easily seen that the values are off by a substantial margin. This is likely due

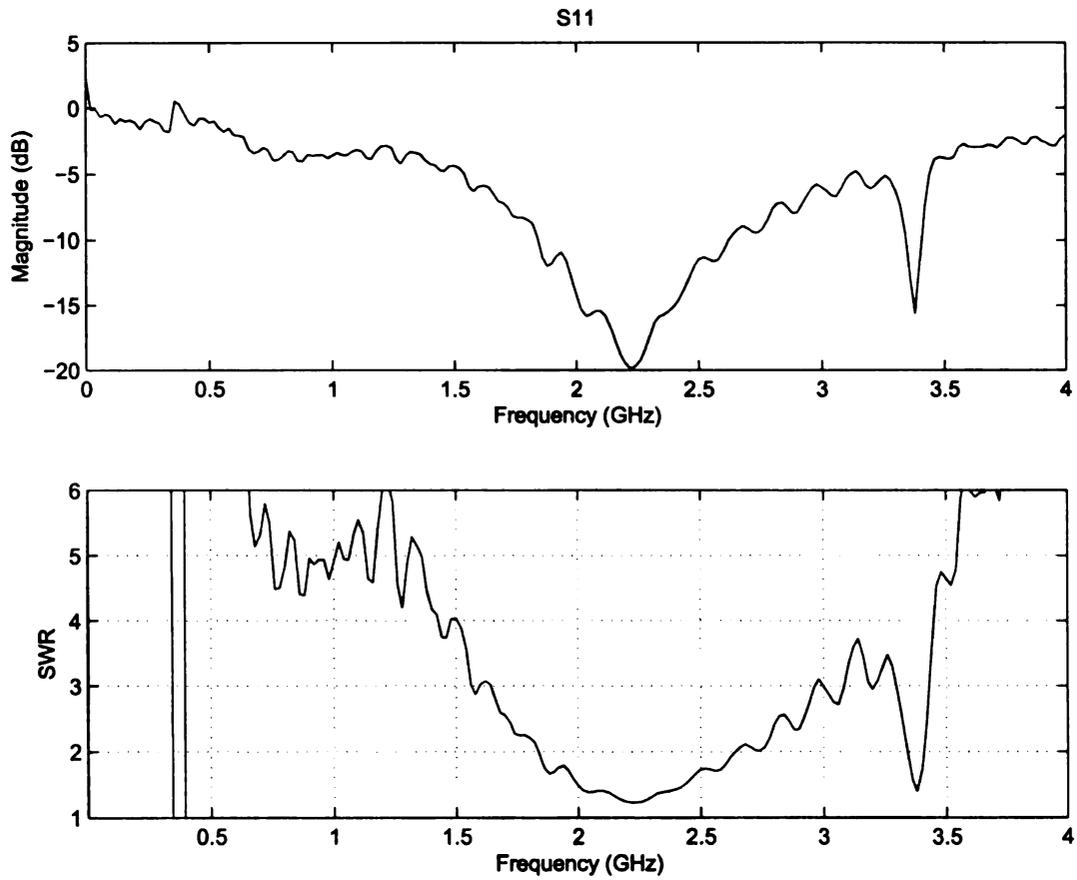


Figure 5.13. S_{11} and SWR_{in} of Compensated AT-41486 Amplifier

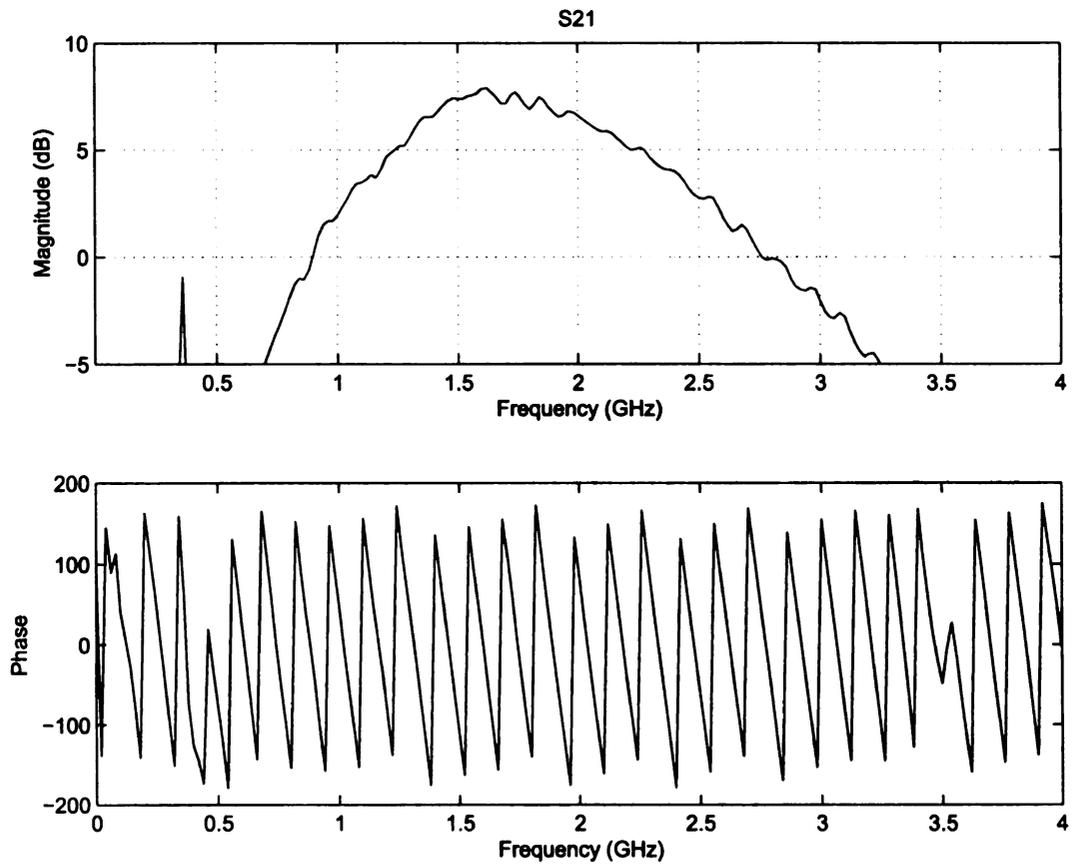


Figure 5.14. S_{21} of Compensated AT-41486 Amplifier

to the S-parameters used for the design being incorrect.

Under a closer examination, this model of transistor themselves seemed to have issues that would make them unreliable to achieve a reproducible design. First, the datasheet for this transistor had a range for the current gain, β or h_{fe} , which varied between 30 and 270 with the average value being about 150. This broad range of possible current gains would definitely have an affect on the S_{21} of the transistor making the S_{21} labeled on the datasheet as an approximation at best. An attempt was made to measure the β of each transistor by connecting them to a curve tracer. Doing this led to an another interesting finding. In the ideal case, the curve tracer will give a plot of collector current vs. the collector-emitter voltage for fixed values for the base current. When connecting the transistor to the curve tracer, the distinct lines formed for different base currents were not always present. In a few cases, these lines began to oscillate as the collector-emitter voltage increased. Other transistors had these constant base current line converge to a point that almost resembled a diode curve.

The amplifier that is shown in Figure 5.12 had a transistor which did not exhibit any of these anomalies that were discussed above, but the design is still way off. The best possible explanation for this is that the S-parameters were not actually measured for that transistor. Since β had such a large variation between each transistor, there is no conceivable way that those S parameters were correct. Even if the S-parameters were correct, the transistor data sheet did not explain where the phase plane for the measurement of the S-parameters was located. At lower frequencies, one would expect that the actual location of the phase plane would be less of a factor in the design, but at higher frequencies, even 1mm could be a substantial enough change in phase that the matching network could be off margin.

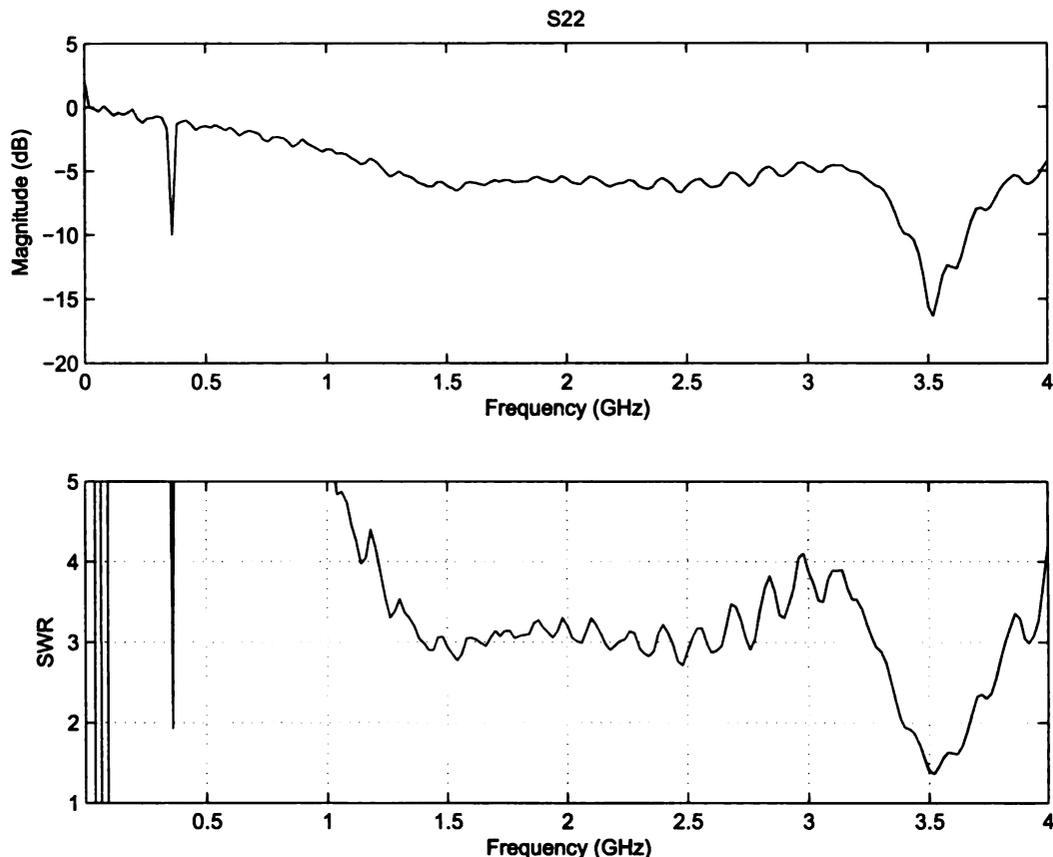


Figure 5.15. S_{22} and SWR_{out} of Compensated AT-41486 Amplifier

5.2.2 MPSH10 Design

The S-parameters for this amplifier were measured as described above. The design for both the MPSH10 and MPSH17 were chosen to be at 100MHz so that the measured results would in no way be erroneous due to parasitics of the components used in the design. Also, these designs required the use of hand-wound air core inductors. The equation used in the fabrication of these inductors is found in [7] and is written here for convenience

$$n = \frac{\sqrt{L(18d + 40l)}}{d} \quad (5.1)$$

where

n = Number of turns

L = Inductance in microhenrys

d = Coil diameter in inches

l = Coil length in inches

with the assumption that the length of the coil is much greater than the diameter. The coils that were used in these matching networks were wound to have 0.125" diameter and were about 0.25" in length.

Biconjugate Match

Figure 5.16 shows the matching network that was used for a biconjugate match to the circuit shown in figure 5.1. The capacitor values shown in Figure 5.16 were those actually used in the measurements and are slightly different than those that were computed to be necessary for the matching network. The difference between these capacitor values was at most 1pF for this design.

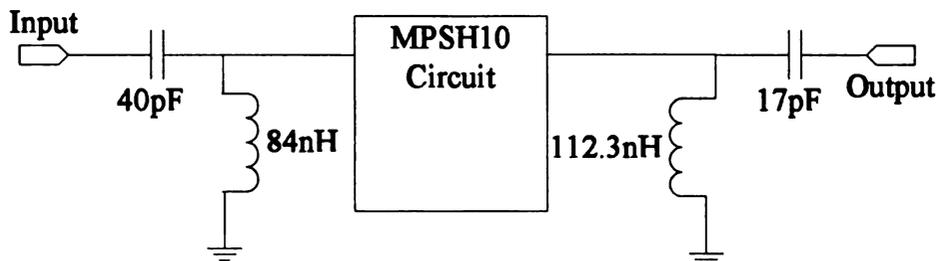


Figure 5.16. Biconjugate Match Network for MPSH10 at 100MHz

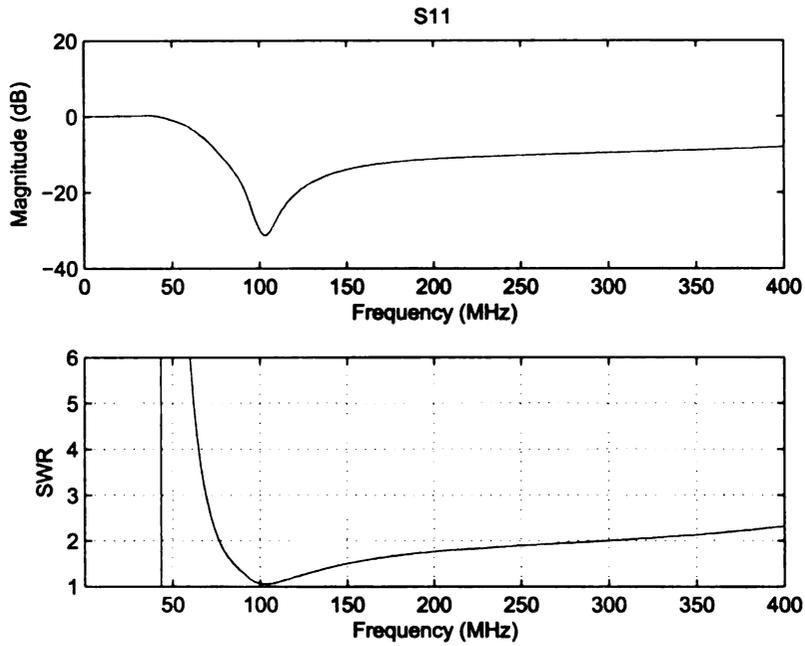


Figure 5.17. S_{11} of Biconjugate Matched MPSH10 Amplifier

The results for this amplifier are shown in Figures 5.17 through 5.19. Comparing these results to Table 5.2 shows that the predicted does well versus the measured. The actual measured gain is 16.85 dB compared to a predicted value of 18.2 dB. Also, SWR_{in} is negligibly close to one, but SWR_{out} measured is at 1.325 which most likely accounts for the 1.15 dB difference in the gain.

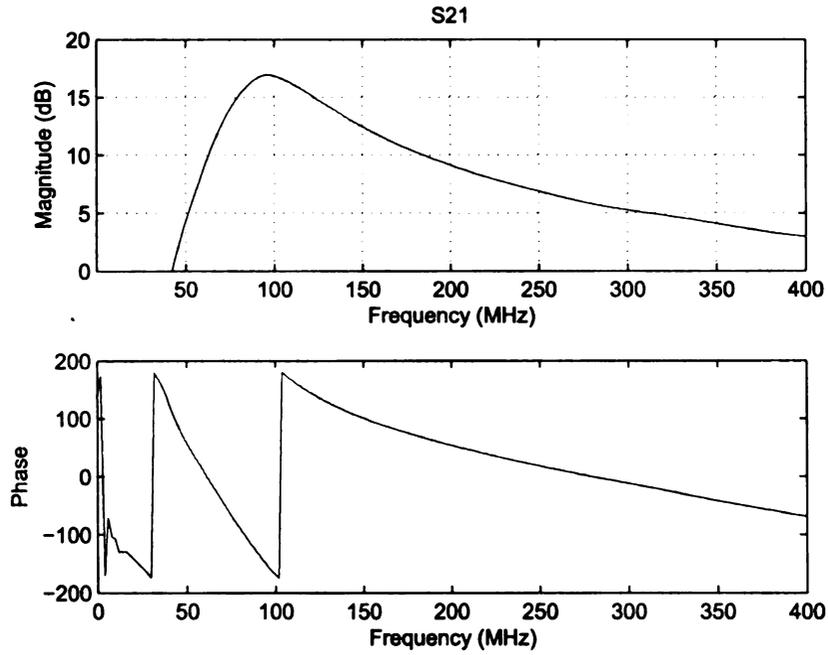


Figure 5.18. S_{21} of Biconjugate Matched MPSH10 Amplifier

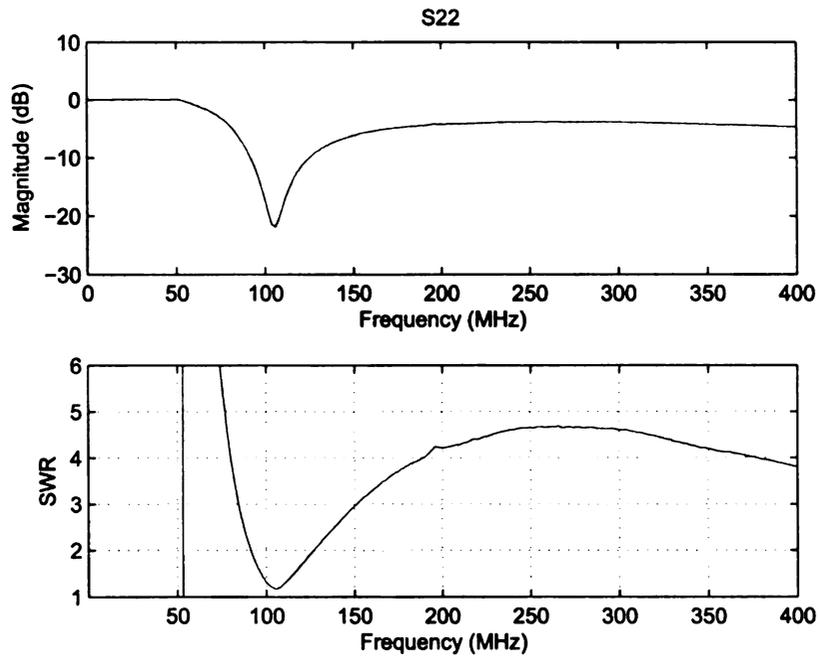


Figure 5.19. S_{22} of Biconjugate Matched MPSH10 Amplifier

Mismatched Design

Figure 5.20 is the matching network that has been attached to the circuit in Figure 5.1. This design has been made to have parameters other than a biconjugate match to show that the design parameters can be achieved. As was the case for the biconjugate match, the capacitor values for the design were rounded to the closest standard value.

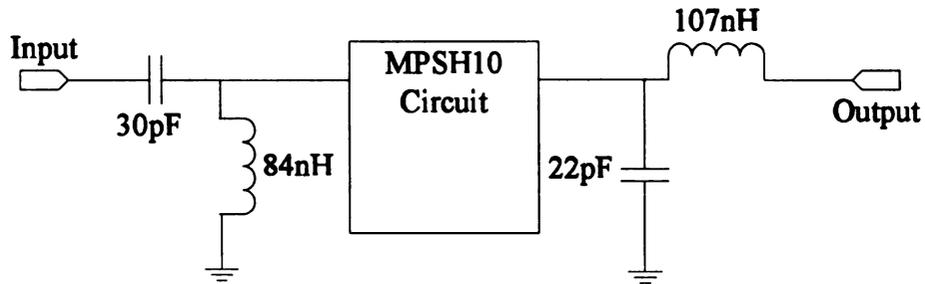


Figure 5.20. Mismatched Network for MPSH10 at 100MHz

The results for the other MPSH10 transistor are shown in Figures 5.21 through 5.23. In comparison to the values predicted in Table 5.2, the measured data is quite close. The target gain for the amplifier was 14.8 dB compared to a measured of 13.4 dB. Also, SWR_{in} had a target value of 1.5 compared to a measured value of 1.241. Finally, SWR_{out} was designed to be 5.59 and the measured value is 5.4.

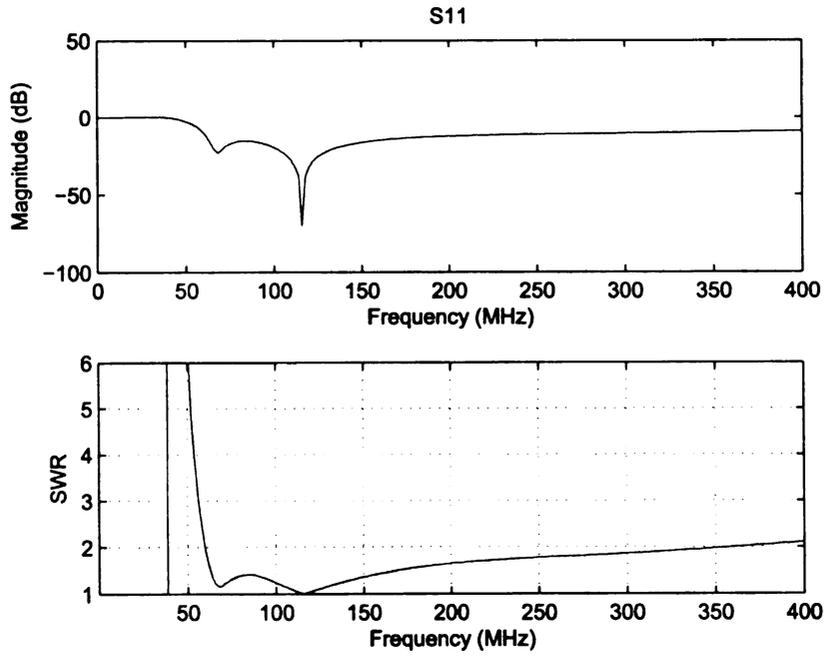


Figure 5.21. S_{11} of Mismatched MPSH10 Amplifier

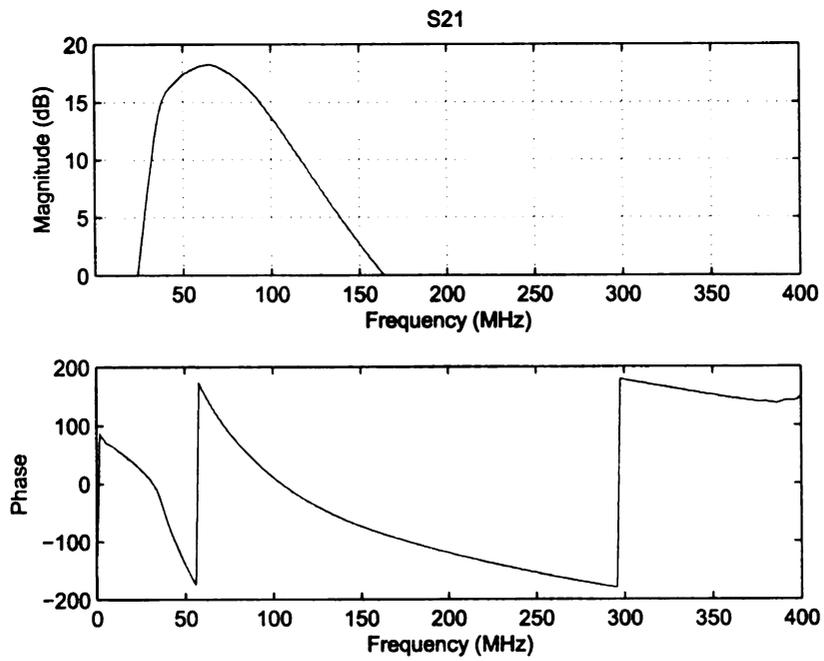


Figure 5.22. S_{21} of Mismatched MPSH10 Amplifier

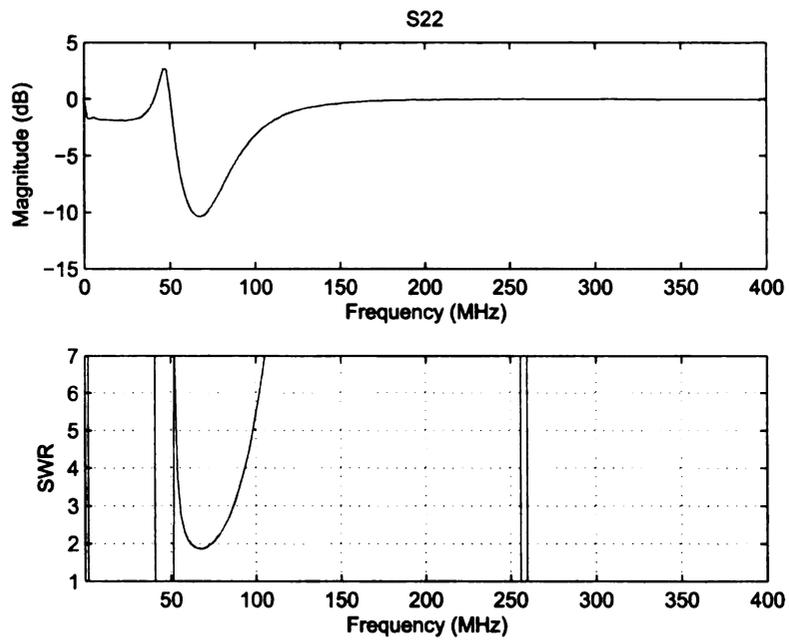


Figure 5.23. S_{22} of Mismatched MPSH10 Amplifier

5.2.3 MPSH17 Design

Figure 5.24 is the matching network used for the circuit shown in Figure 5.2. As was the case in all the other designs, the capacitor values were rounded to the nearest standard values but the difference in the capacitor values was less than one.

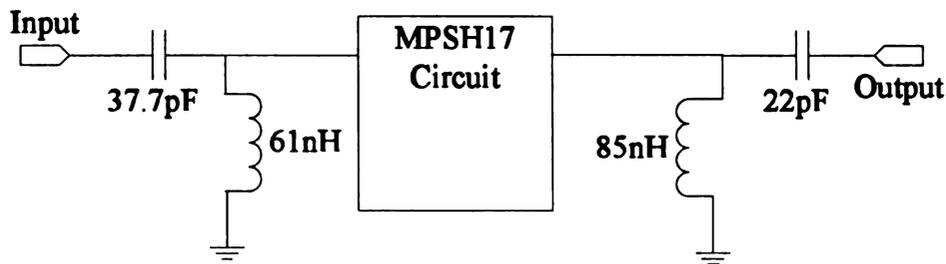


Figure 5.24. Biconjugate Matched Network for MPSH17 at 100MHz

Figures 5.25 through 5.27 show the measured results of this amplifier. As can be seen, the reflection coefficient minimums (S_{11} and S_{22}) do not occur at 100MHz as desired. Attempts to tune the circuit by adjusting the spacing in the inductors' windings were ineffective. The capacitor values were very close to standard values so this was likely not the source of the error. Even though these minimums do not occur at 100MHz, the values measured at 100MHz are still acceptable. The measured gain was at 18.6 dB when compared to 19.2 dB for the predicted. The SWR_{in} was measured to be 1.46 whereas SWR_{out} was measured to be 1.325.

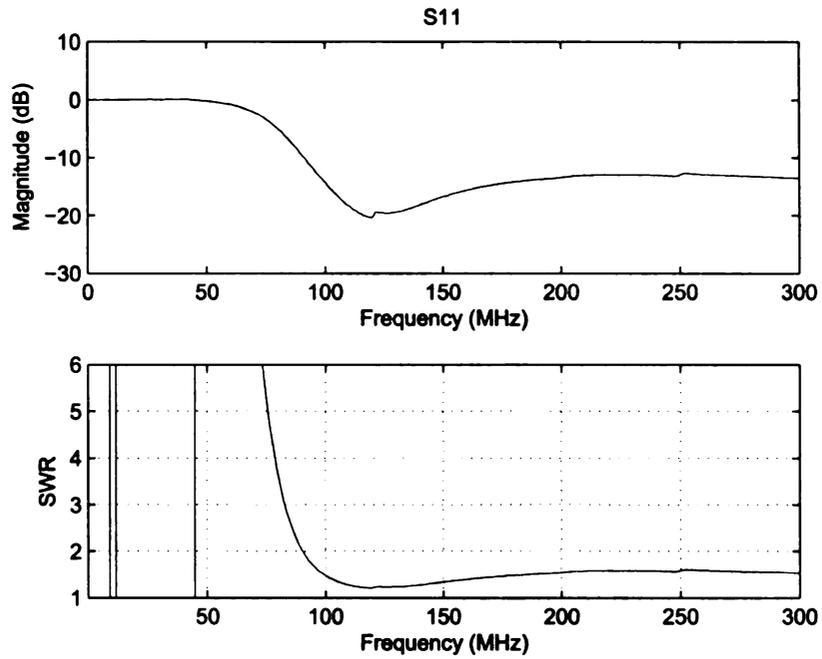


Figure 5.25. S_{11} of Biconjugate Matched MPSH17 Amplifier

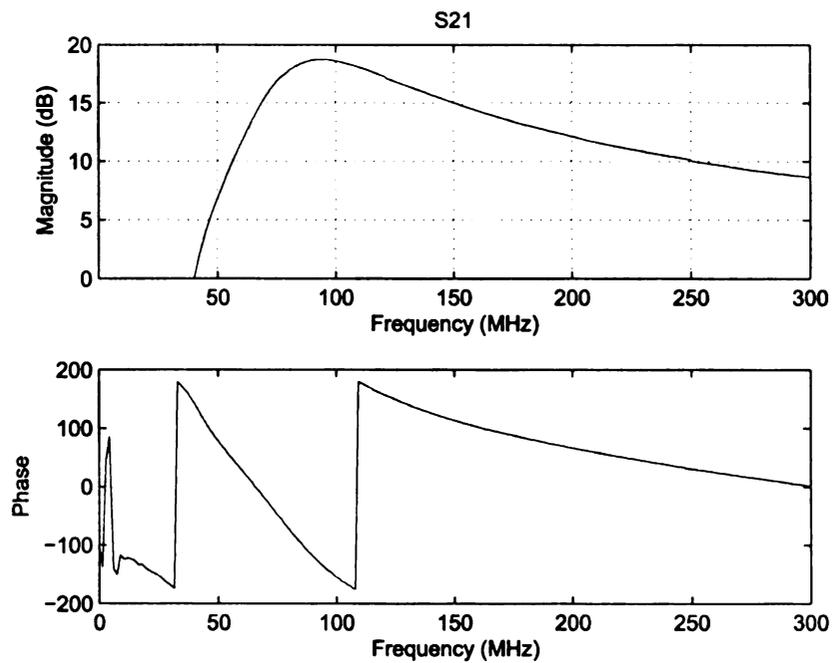


Figure 5.26. S_{21} of Biconjugate Matched MPSH17 Amplifier

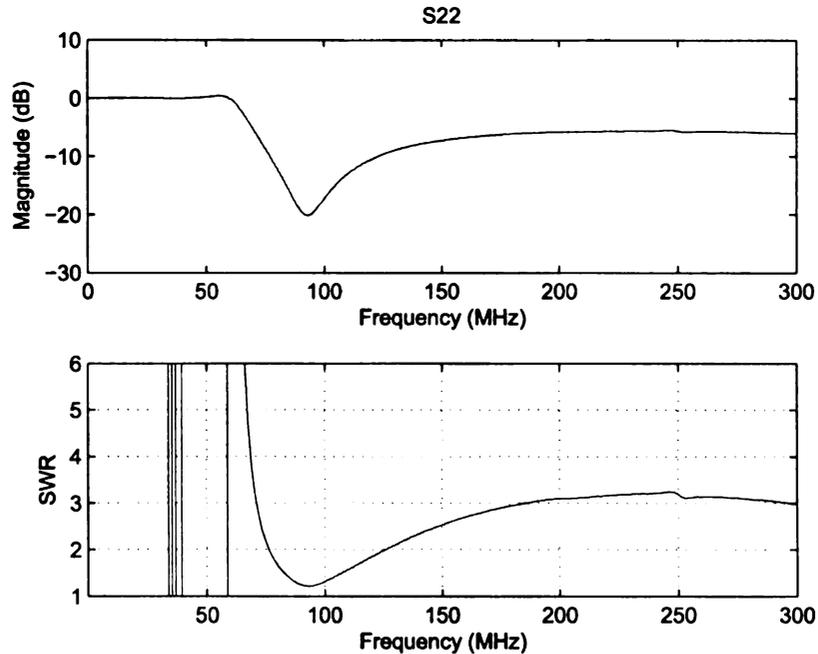


Figure 5.27. S_{22} of Biconjugate Matched MPSH17 Amplifier

5.2.4 Summary

Table 5.3 shows a summary of the values attained from the above measurements. for the most part, the values agree well with the predicted values found in Table 5.2. For all of the amplifiers, other than the AT41486, placing adjustable capacitors will allow the ability to better tune the circuit to the desired values. Surprisingly, the inductors made from equation 5.1 were quite accurate and could be slightly adjusted by changing the spacing between the coils for tuning purposes.

Noting that the amplifier based on the AT-41486 had measured data that was far away from any of the predicted data shows the importance of accurate S-parameter measurements. All of the other amplifiers had their own S-parameter measurement

Transistor	AT-41486 Mismatch	MPSH10 Biconjugate	MPSH10 Mismatch	MPSH17 Biconjugate
Center Frequency	2GHz	100MHz	100MHz	100MHz
G_T (dB)	6.76	16.85	13.4	18.6
SWR_{in}	1.49	1.07	1.24	1.46
SWR_{out}	3.21	1.325	5.4	1.325

Table 5.3. Measured Values Summary

data used in the designing process.

CHAPTER 6

Conclusion

As can be seen from the amplifiers in Chapter five, the design tool, “AmpDesigner” will give accurate results for those amplifiers which have accurate S-parameter measurements. This program aids in the design process by giving the user all of the pertinent amplifier information that may be of concern in the design process. Once the design values for Γ_S and Γ_L have been obtained from the program, then the user just needs to concentrate on the physical layout of the amplifier.

However, this program does have some limitations. It will not give the frequency dependant characteristics, such as bandwidth, of the amplifier with the matching networks connected. To implement this type of analysis, a few things would be needed. First, the entire spectrum of S-parameters for the transistor without matching networks would be required. This is likely not a problem if the S-parameters are measured by the designer, but if the data is extracted from a data sheet, then this may be a problem since the values listed tend to be sparse. Second, the specific network that is connected needs to have its frequency dependant characteristics modeled. This is so the values for Γ_S and Γ_L can be modeled over the frequency range of interest. Once this is accomplished, then the amplifier can be characterized over frequency and showing its true predicted performance.

Despite these drawbacks, “AmpDesigner” will give the user the ability to reliably

achieve the desired performance at the frequency of interest.

BIBLIOGRAPHY

BIBLIOGRAPHY

- [1] H.A. Atwater. The design of the radial stub: A useful microstrip circuit element. *Microwave Journal*, November 1985.
- [2] David K. Cheng. *Field and Wave Electromagnetics*. Addison-Wesley, second edition, 1989.
- [3] Guillermo Gonzalez. *Microwave Transistor Amplifiers*. Prentice Hall, second edition, 1997.
- [4] J.P. Humbert. Microwave measurements. In Bradford L. Smith and Michel-Henri Carpentier, editors, *The Microwave Engineering Handbook*, volume 2. Van Nostrand Reinhold, 1993.
- [5] Norman S. Nise. *Control Systems Engineering*. John Wiley & Sons, Inc., third edition, 2000.
- [6] David Pozar. *Microwave Engineering*. John Wiley & Sons, Inc., third edition, 2005.
- [7] Dana G. Reed, editor. *ARRL Handbook*. ARRL-The national association for amateur radio, eighty-first edition, 2004.
- [8] B.A. Syrett. A broad-band element for microstrip bias or tuning circuits. *IEEE Transactions on Microwave Theory and Techniques*, MTT-28(8):925–927, August 1980.
- [9] Agilent Technologies. Up to 6 ghz low-noise silicon bipolar transistor, at-41486. Technical report, Aglient, 1999.
- [10] Agilent Technologies. Fundamentals of rf and microwave noise figure measurements, application note 57-1. Technical report, Aglient, March 2004.
- [11] Rodger Ziemer and William Tranter. *Principles of Communications*. John Wiley & Sons, Inc., fifth edition, 2002.

MICHIGAN STATE UNIVERSITY LIBRARIES



3 1293 02504 0191