AN INTRODUCTION TO
THE DESIGN AND TESTING OF MICROWAVE AMPLIFIERS

LABORATORY MANUAL
for the
EE 723 Adjunct Microwave Laboratory

Patrick Roblin
Professor
The Ohio State University
2005 (13th Edition)
Contents

Acknowledgements 5

1 INTRODUCTION 6
1.1 Laboratory Overview ........................................ 6
1.2 The Laboratory Facilities ..................................... 6
1.3 Laboratory Policies .......................................... 7
1.4 Laboratory Schedule and Report ............................ 8

2 LABORATORY 1 with HP8573C:
Microwave Measurement of a Microwave Amplifier 9
2.1 Preparation for Lab 1 ........................................... 9
2.2 PART I: Frequency and Gain Compression Measurement .......... 9
2.3 PART II: Measurement of the Noise Figure ...................... 13

3 LABORATORY 1 with Agilent E5071B:
Microwave Measurement of a Microwave Amplifier 17
3.1 Preparation for Lab 1 ........................................... 17
3.2 PART I: Frequency and Gain Compression Measurement .......... 17
3.3 PART II: Measurement of the Noise Figure ...................... 21

4 LABORATORY 2:
Nonlinear Response of Amplifiers under Multitone Excitations 25
4.1 Introduction and System Diagram ............................ 25
4.2 EXPERIMENT I ................................................. 27
4.2.1 Laboratory Goal ........................................... 27
4.2.2 Output Data To Be Recorded ............................. 27
4.2.3 Measurement Procedure .................................. 28
4.2.4 Data Analysis ............................................. 32
4.3 EXPERIMENT II ............................................... 33
4.3.1 Laboratory Goals ........................................... 33
4.3.2 Output Data To Be Recorded ................................. 33
4.3.3 Measurement Procedure ...................................... 33
4.3.4 Data Analysis ................................................. 35
4.4 EXPERIMENT III .................................................. 35
4.4.1 Laboratory Goals ............................................... 35
4.4.2 Output Data To Be Recorded ................................. 35
4.4.3 Measurement Procedure ...................................... 36
4.4.4 Data Analysis ................................................. 38
4.5 Appendix .......................................................... 38

5 LABORATORY 3:
Design of a Microwave Amplifier ............................... 39
5.1 Design .......................................................... 39
  5.1.1 Important Design Constrains: ............................ 39
  5.1.2 Design Procedure: .......................................... 39
  5.1.3 Summary of the Design rules associated with the Fabrication .... 42
5.2 Fabrication Procedure .......................................... 44
5.3 Measurement of Your Circuit. .................................. 45
5.4 Write Up ....................................................... 46
  5.4.1 The Laboratory Report ................................. 46
  5.4.2 Poster Page .............................................. 46

6 APPENDICES ....................................................... 48
6.1 How to Use ADS ................................................. 49
6.2 How to Use the Layout Tool ................................... 50
6.3 How to Use LineCalc .......................................... 52
6.4 Tips for Soldering .............................................. 53
6.5 Caring About Connectors ...................................... 55
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.6</td>
<td>AT42085 S-Parameters and Noise Parameters</td>
<td>56</td>
</tr>
<tr>
<td>6.7</td>
<td>Computer Data Acquisition for 2-Port S-parameters</td>
<td>57</td>
</tr>
<tr>
<td>6.8</td>
<td>Loading S-parameter data in ADS and MATLAB</td>
<td>58</td>
</tr>
<tr>
<td>6.9</td>
<td>Active Biasing Circuit</td>
<td>61</td>
</tr>
<tr>
<td>6.10</td>
<td>Laboratory Report Forms</td>
<td>63</td>
</tr>
</tbody>
</table>
Acknowledgements

This laboratory would not have been possible without the help and support of many people whose contributions I would like to acknowledge.

Most of all I am very grateful to the Hewlett Packard/Agilent Technologies Company for donating our first Network Analyzer (now retired), the Spectrum Analyzer HP8592A and Noise Figure meter HP8970B. Particular I would like to thank our famous OSU alumini Dr. John Moll of HP Laboratories as well as Howard Boyd, Tom Nebel, and Jeff Skokal of the local HP sale office for their active support.

The Network Analyzer HP8753C currently used in the lab was acquired thanks to a National Science Foundation education equipment grant. The most recent Network Analyzer Agilent E5071B was acquired thanks to a grant of the Ohio Board of Regents. Finally we are greatful to Motorola for donating to our lab in 2003 a set of unused pieces of equipment which we used for developing Lab #2 on nonlinear RF.

Several parts and supplies have been generously donated by the microwave industry. I would like to thank the Rogers Corporation for the donation of microstrip circuit boards; and Agilent/Avantek for the donation of microwave transistors.

The first demonstration microwave amplifier was developed for this laboratory by Truong X. Nguyen during the course of independent study projects.

I am most grateful to my graduate students Sung Choon Kang, Chih Ju Hung, Siraj Akhtar, Xian Cui for their dedicated contribution to the development of this laboratory since 1992. Specifically I would like to acknowledge the help of Xian Cui with the development of the new Lab #2 experiments.

Finally I would like to thank Professors. H.C. Ko, B. Mayan, D. Hodge, Y. Zheng, and F. Ozguner for the full support provided to this project as chair of the EE department. Prof. Klein on-going support is also acknowledged.

Patrick Roblin
January 2005
1. INTRODUCTION

1.1 Laboratory Overview

The EE 723 adjunct microwave laboratory is designed to run in parallel with EE 723. An introduction to microwave amplifier and oscillator design is given in the EE 723 lecture. In support to the theory, the simulation and optimization of the microwave amplifiers studied are performed using an advanced microwave circuit simulator (ADS) in our ER4 computer facilities. To complete the learning cycle the EE 723 adjunct microwave laboratory provides the EE 723 students with the opportunity to experimentally verify the device principles and design theory of the microwave devices and circuits studied. The two main objectives of the laboratory are therefore:

1. To introduce the EE 723 student to the techniques and principles of microwave measurement with a modern Network Analyzer as well as an introduction to non-linear RF measurement.

2. To involve the EE 723 student in the design of an amplifier or an oscillator. The project includes first principle design, CAD simulation and optimization, CAD layout, fabrication of a prototype, and testing.

Note that the students who are interested in pursuing a design project of their own will have the opportunity to do so under a follow-up independent study. Successful design projects will be used as demo in the microwave laboratory.

1.2 The Laboratory Facilities

The measurement and fabrication facilities of the Education Microwave Laboratory are located in CL 305.

The measurement stations include of a HP 8753C and an Agilent E5071B Network Analyzer, the HP 8970B Noise Figure meter, the HP 6237B power supplies, the HP54503A Digitizing oscilloscope, the RD Intermodulation distortion simulator and the Anritsu MS610B Spectrum Analyzer.

A bench rack is available for storing your circuits under development. Please use the white stickers available in CL 305 to write your name(s) on your circuits.
The circuits designed by the students are fabricated in CL305 using computer aided machining by the EE 723 teaching assistant. CL305 features a workbench, a tool box, various drills, two different soldering irons, and a parts cabinet.

1.3 Laboratory Policies

For a proper and efficient sharing of the laboratory equipment a set of rules is necessary. The latter make up the Laboratory policy. A sample of the Laboratory Policy is given below. You are asked to read it and obide by its rules to participate in this laboratory.

EE 723 Laboratory Policy

1. Schedule your measurement and fabrication time (CL 305) in advance on the sign up sheet on the door of CL 305. You can reserve up to 2 hours/per week. Open time is however available on a first come, first served basis.

2. Read your laboratory manual and make a plan of work before coming to the lab.

3. You can obtain the laboratory key against your I.D card at the reception desk of the EE department office. The receptionist should check that you are on the EE 723 class roster; remind him to do so!

4. Make sure that the laboratory door is closed when you leave the room even for a short period.

5. Care for the measurement cables and connectors (refer to Appendix 6.5 of your laboratory manual) and the equipment in general. The measurements will be made using SMA test cables. Do not disconnect the SMA test cables from the Network Analyzer. Dirty cables lead to bad data.

6. Read soldering, drilling and cutting tips in the Appendices D, E and F of your laboratory manual to ensure a proper use of the tools and to assure your protection. If you are uncomfortable with a particular laboratory procedure request help from the instructor.

7. Report damage or malfunction immediately (you will not be penalized) to the instructor or teaching assistant.
1.4 Laboratory Schedule and Report

A laboratory report is due for each laboratory and should be turned in by the due date specified in your EE723 syllabus. In addition, a poster page summarizing your design should be included with the last laboratory report. A few guidelines are given below:

- use for first page of your lab report the laboratory report sheet given in Appendix 6.10,
- include a table of contents describing the material appended,
- write a caption on each figure and plot,
- discuss any problems encountered.

Only one report per team should be turned in. The reports do not need to be long but must be well organized. The first two laboratory reports do not need to be typed. However, to help with clarity please write the laboratory 3 (design project) report and poster page using a word processor. Include ADS simulation plots in your report. An electronic version of your report/poster and presentation will be archived on the web so that it can be used as a reference by future EE723 students and the best posters will displayed in the display case in the Dreese-Caldwell bridge.
2. LABORATORY 1 with HP8573C: Microwave Measurement of a Microwave Amplifier

Laboratory Goal: This laboratory is divided into two parts. The purpose of Part I of this laboratory is to get familiarized with the Network Analyzer and to perform measurements on a microwave amplifier. In Part II of this laboratory, the measurement of the Noise Figure of the microwave amplifier is performed using a Noise Figure meter. Approximate Duration: 4-5 hours.

2.1 Preparation for Lab 1

1. Preliminary Reading:
   Read in the User’s Guide manual of the HP 8753B Network Analyzer (available on-line in the course webpage) the following chapters:
   - Chapter 1: Operating the HP 8753B
   - Chapter 3: Scattering Parameter Measurements with the HP 8753B
   - Chapter 4: Time Domain Analysis

   Do not forget to bring your Network Analyzer User’s guide with you to the lab for quick reference.

2. Laboratory Overview:
   Read the remaining of the instructions for Lab 1 before coming to your scheduled laboratory time. Determine the goals to be attained.

3. Connector and Cable Care:
   The measurement of microwave circuit is a science. In this laboratory and the subsequent ones you will physically connecting many microwave devices to the test cables of the network analyzer. The quality of your measurement and the lifetime of the devices and cables will dependent critically on this simple process. Please read Appendix 6.5 which gives some key advices about it. It is indeed important to learn how to do it right and develop habits which you can keep during your professional life.

2.2 PART I: Frequency and Gain Compression Measurement

Measurement Procedure:
1. **Clean the SMA connectors of the cables, and standards:**
   The relatively inexpensive SMA connectors are not made to be connected more than a few times. You might notice gold particles on the white dielectric of the cables, and standards which will affect the quality of your calibration and measurement. The standards which are sketched below are available in a small wooden box. Clean the cables and standards (50 Ω and short) using a cotton swab, alcohol and compressed air. See connector care in Appendix 6.5.

2. **Power Up:**
   Turn the Network Analyzer on if it is off or press PRESET if it is already on.

3. **Perform a 2-Port Calibration of the Network Analyzer**
   You might find it convenient to select the SMITH CHART format (p. 10) for all the scattering parameters before calibrating. Simply press MEAS to select a S-parameters and then press FORMAT and select SMITH CHART.

   **Do NOT HOLD the cable when you calibrate as this introduces calibration instabilities.** Just let the coaxial cable lying at rest on the table before you select SHORT, OPEN, 50 Ω LOAD and the four THRU in the calibration procedure.

   To start the calibration press CAL and select the 2-port Calibration (p. 13). Use the calibration short, open, 50 Ω load in the SMA calibration kit (available in the small wooden box) for the reflection calibration of ports 1 and 2. A sketch of the standards is given below. Use the thru for the THRU calibration. For the isolation calibration select Omit Isolation.

   Press DONE when done. Save your calibration in a register (p. 31).

   Verify the calibration by testing the 50Ω load, the open and short and thru connections. Note that this calibration is only valid for the particular test cables you are using. **Please do not remove the test coaxial cables!** You are now ready to perform a two-port measurement.

   ![Diagram](image)

   Figure 2.1: Calibration Standards

Before you continue a few comments on calibration are in order.
The transmission calibration required the use of a thru standard. A thru is supposed to have no length. We have a mating problem since our SMA connectors are both male can be connected together. Since an error in the thru calibration is not too critical at 3 GHz you used the short female to female barrel available. A more rigorous approach at high-frequency requires the use of two equal length barrels. The female to female barrel is connected between port 1 and port 2 during the thru calibration. The female to male which provides an equal electric length is connected say to port 1 for the S11 calibration and for the subsequent S-parameter measurements. [Note: Once you have become familiar with the calibration procedure you can avoid the calibration step by recalling the 2-Port calibration data from a previous calibration which have been stored on disk for your convenience (check with the instructor to verify if this is available).]

To retrieve calibration data use the following input procedure:

Press [LOCAL], select [SYSTEM CONTROLLER].
Press [RECALL], select [LOAD FROM DISK], [LOAD TWOFULL].
Press [LOCAL], select [TALKER/LISNTER].

The last step is intended to free the HPIB (IEEE488) network so that another piece of instrument (the other Network Analyzer or the Noise Figure meter) can also become temporarily a system controller to plot or access the disk drive. Once these calibration data have been loaded the network analyzer should be calibrated. You can verify it with the open short, thru and 50Ω to make sure. Note: The calibration data are only valid for the pair of coaxial cables used. The network analyzer will be improperly calibrated if someone has changed the coaxial cables. Do not disconnected the coaxial cables from the network analyzer itself!

4. S-parameters of a microwave amplifier:

We shall now measure the S parameters of a low noise narrow-band microwave amplifier.

- Clean the connector of the amplifier (see Appendix 6.5 on connector care).
- Connect the low noise amplifier model 13LNA (aluminium body with 2 SMA connectors, blue label) between port 1 (input) and 2 (output).
- Turn on the HP6237B power supply. The operating voltage should already be set to 12 V on the lower scale of the power supply meter (Do not increase it as this would damage the amplifier). Note: The red banana cable should be connected to +18 V and the black cable to COM. The meter switch should indicate +18 V.
• Connect the power supply to the amplifier. Respect the polarity! Red is positive. Black is negative. To prevent power gain compression of the front end 13LNA amplifier we need to reduce the power used by the Network Analyzer for the S-parameter measurements to -10 dBm:

\[
\text{[MENU]} \\
\text{[POWER] [-10] [×1]}
\]

Make a hardcopy output of \( S_{21} \) on the plotter (Follow the instruction given in the User’s guide manual p. 17). Find the frequency \( f_0 \) at which \( S_{21} \) is maximum using the marker MKR. Find the 3 dB bandwidth \( B \) of the amplifier. Record \( |S_{21}| \) and \( |S_{12}| \) in dB at this frequency and measure \( |S_{11}| \) and \( |S_{12}| \) using VSWR units. [Note: The network analyser should only become the system controller for the time necessary to plot (Press SYSTEM and select CONTROLLER). Once you are done free the HPIB (IEEE488) network (Press SYSTEM and select TALKER/LISTEN) to give access to the plotter and disk drive to other users.]

5. 1 dB Power Compression:

We shall now measure the 1 dB Power compression point at the frequency \( f_0 \) of the amplifier under test.

The network analyzer HP 8753B offers the capability (not always associated with a network analyzer) to sweep the RF input power. An accurate measurement of the output power of an amplifier requires a special calibration procedure to account for the loss through the test set between the amplifier output and the receiver input. For the sake of simplicity we shall bypass here this special calibration. Our measurement should give us the amplifier’s response to a power ramp. Record \( |S_{21}| \) in dB of the amplifier for -10 dBm input power \( P_{in} \). [Note: \( P_{in} \) in dBm is defined as \( 10 \log (P_{in} / 1 \text{ mW}) \). From this response we shall determine the input power at which the gain \( |S_{21}| \) is compressed by 1 dB.

Measurement procedure:

• Select a power sweep at the frequency \( f_0 \) (e.g. 2 GHz)

\[
\text{[MENU]} \\
\text{[SWEEP TYPE MENU]} \\
\text{[POWER SWEEP]} \\
\text{[RETURN]} \\
\text{[CW FREQ] [2] [G/n]}
\]

• Set the stimulus parameters. Power levels must be set so that the amplifier is forced into power gain compression. The range of the HP 8753’s source is from
-10 to +10 dBm.

[MEAS]
[S21[B/R]]
[START] [-10] [×1]
[STOP] [10] [×1]

- Use the marker [MKR] to find the input power for which a 1 dB drop in the amplifier’s gain occurs relative to the small signal gain.

6. Fill the Lab Report #1 given in Appendix 6.10. Append the plots which documents steps 4, and 5.

2.3 PART II: Measurement of the Noise Figure

Low noise RF and microwave amplifiers are used at the front end of microwave communication systems to amplify the weak signals received by the antenna or dish. The signal received from a satellite are typically in the order of -120 dBm (10^{-15} Watt) which corresponds approximately to a voltage of 0.2 μV across a resistor of 50 Ω.

The Noise Figure F is defined (see Gonzalez p. 296) as the ratio of the output noise power \( P_{No} \) of the amplifier divided by the amplifier gain \( G_A \) and the input noise power \( P_{ni} \).

\[
F = \frac{P_{No}}{P_{ni} G_A}
\]

The noise figure F of an amplifier is larger than 1 because it is a measure of the noise added by the amplifier to the output noise power. If the amplifier did not add noise we would have \( F = 1 \) or 0 dB. The bandwidth \( B \) of the amplifier and its noise figure F will permit us to calculate the minimum detectable signal input power \( P_{i,mds} \) (see Gonzalez p. 354).

The measurement of the noise figure is quite simple with the HP8970B meter.

1. **Power up:**
   Turn the HP8970B Noise Figure meter by pressing the LINE switch ON. If the noise figure meter is already on press RESET. The Noise Figure Meter performs a quick internal check and frequency calibration.

2. **Set frequency parameters:**
   The HP8970B measures the noise figure versus frequency from 10 to 1800 MHz. The default START, STOP and START frequencies are 10, 1600 and 20 MHz respectively. To extend it to 1800 MHz simply press STOP FREQ 1800 ENTER. **Note:** The noise
figure of the low noise amplifier 13LNA at its center frequency $f_0$ cannot be measured as it is above 1800 MHz.

3. **Calibration:**
   Connect (using a thru) the Noise Figure Meter as shown below
   ![Diagram of Noise Figure Meter Calibration](image)

   Press CALIBRATE twice. The HP8970B measures its own noise figure. The noise figure meter should make two frequency sweeps. If an error occurs you might have to increase the number of measurements averaged by pressing INCREASE in the SMOOTHING block and try again. If after several trials an error is still occurring you might have to limit the measurement to 1600 MHz.

4. **Measurement of the Noise Figure of the Amplifier:**
   Once the noise figure meter has been calibrated it is possible to measure the corrected noise figure and gain of the device under test (DUT). [Note: This noise figure is corrected because the meter removes its own noise figure from the measurement results.]
   - Connect the amplifier between the noise source output and the meter input as shown below:
   - To measure the noise figure and gain press the key NOISE FIGURE AND GAIN. The INSERTION GAIN display shows the device gain and the NOISE FIGURE display shows the noise figure of the DUT at room temperature at the displayed frequency.
   - Connect the HP6237 power supply to the amplifier. Respect the polarity! See earlier instructions regarding the use of the power supply. Notice the reduction of the noise figure once the amplifier is powered.
   - Measure the noise figure and gain at 1800 MHz by entering the frequency.
Figure 2.3: Noise Figure Measurement

- Measure the noise figure versus frequency by pressing the SINGLE key to obtain a single sweep.

- Plot the data measured. In order to plot, the noise figure meter must be the controller of the HPIB (IEEE488) network.

Verify that nobody is plotting or using the disk and inform the other users that the noise figure meter will temporarily become the controller of the HPIB network.

Turn the plotter on and introduce a sheet of paper.

Press 48.0 SPECIAL function for the noise figure meter to become the controller.

Press 47.0 SPECIAL FUNCTION to verify that the plotter is on the HPIB network.

Press 25.0 SPECIAL FUNCTION to start the plot.

Once the plot is done press 48.1 SPECIAL FUNCTION to free the HPIB network from the control of the noise figure meter.

5. Complete the Laboratory Report #1 with the noise figure data. Append you noise figure plot. Calculate the power of the minimum detectable input signal $P_{i,mds}$ of this amplifier at $f_0$ (you will need to extrapolate $F$ at $f_0$).
Laboratory Goal: This laboratory is divided into two parts. The purpose of Part I of this laboratory is to get familiarized with the Network Analyzer and to perform measurements on a microwave amplifier. In Part II of this laboratory, the measurement of the Noise Figure of the microwave amplifier is performed using a Noise Figure meter. Approximate Duration: 4-5 hours.

3.1 Preparation for Lab 1

1. Preliminary Reading:
   Read in the User’s Guide manual of the E5071B Network Analyzer (available on-line in the course webpage) the following chapters:
   - Chapter 4: Full 2-port Calibration (p. 100)
   - Chapter 6: Data Analysis
   - Chapter 9: Data Output
   - Chapter 14: Measurement Examples

   A PC and an HP workstation are available in the microwave lab for a quick reference to this manual.

2. Laboratory Overview:
   Read the remaining of the instructions for Lab 1 before coming to your scheduled laboratory time. Determine the goals to be attained.

3. Connector and Cable Care:
   The measurement of microwave circuit is a science. In this laboratory and the subsequent ones you will physically connecting many microwave devices to the test cables of the network analyzer. The quality of your measurement and the lifetime of the devices and cables will depend critically on this simple process. Please read Appendix 6.5 which gives some key advices about it. It is indeed important to learn how to do it right and develop habits which you can keep during your professional life.

3.2 PART I: Frequency and Gain Compression Measurement

Measurement Procedure:
1. **Clean the SMA connectors of the cables, and standards:**
   The relatively inexpensive SMA connectors are not made to be connected more than a few times. You might notice *gold particles* on the white dielectric of the cables, and standards *which will affect the quality of your calibration and measurement*. The standards which are sketched below are available in a small wooden box. Clean the cables and standards (50 Ω and short) using a cotton Q-tips, alcohol and compressed air. See connector care in Appendix 6.5.

2. **Power Up:**
   Turn the Network Analyzer on if it is off or press [PRESSET] if it is already on.

3. **Perform a 2-Port Calibration of the Network Analyzer**
   You might find it convenient to select the Z-SMITH CHART format for all the scattering parameters before calibrating. For example for $S_{21}$:
   
   press [Meas]
   click “S21”
   press [Format]
   click “Smith”
   click ”$R+jX$”

   **Do NOT HOLD** the cable when you calibrate as this introduces calibration instabilities. Just let the coaxial cable lying at rest on the table before you select SHORT, OPEN, 50 Ω LOAD and the four THRU in the calibration procedure.

   To start the calibration press [CAL] and select the 2-port Calibration (p. 101). Use the calibration short, open, 50 Ω load in the SMA calibration kit (available in the small wooden box) for the reflection calibration of ports 1 and 2. A sketch of the standards is given below. Use the thru for the THRU calibration. Just skip the isolation calibration step ”Isolation (Optional)”.

   Click ”Done” when done.

   Save your calibration in a register (p. 244):
   
   press [Save/Recall]
   click ”Save Type”
   click ”State & Cal”
   click ”Save State”
   click ”State 01” (or any other)
Verify the calibration by testing the 50Ω load, the open and short and thru connections. Note that this calibration is only valid for the particular test cables you are using. Please do not remove the test coaxial cables!. You are now ready to perform a two-port measurement.

![Calibration Standards](image)

Figure 3.1: Calibration Standards

Before you continue a few comments on calibration are in order.

The transmission calibration required the use of a thru standard. A thru is supposed to have no length. We have a mating problem since our SMA connectors are both male can be connected together. Since an error in the thru calibration is not too critical at 3 GHz you used the short female to female barrel available. A more rigorous approach at high-frequency requires the use of two equal length barrels. The female to female barrel is connected between port 1 and port 2 during the thru calibration. The female to male which provides an equal electric length is connected say to port 1 for the S11 calibration and for the subsequent S-parameter measurements.

4. S-parameters of a microwave amplifier:
We shall now measure the S parameters of a low noise narrow-band microwave amplifier.

- Clean the connector of the amplifier (see Appendix 6.5 on connector care).
- Connect the low noise amplifier model 13LNA (aluminium body with 2 SMA connectors, blue label) between port 1 (input) and 2 (output).
- Turn on the HP6237B power supply. The operating voltage should already be set to 12 V on the lower scale of the power supply meter (Do not increase it as this would damage the amplifier). Note: The red banana cable should be connected to +18 V and the black cable to COM. The meter switch should indicate +18 V.
- Connect the power supply to the amplifier. Respect the polarity! Red is positive. Black is negative. To prevent power gain compression of the front end 13LNA amplifier we need to reduce the power used by the Network Analyzer for the S-parameter measurements to -10 dBm:
press [Sweep Setup]
click "Power"
enter [-10] [×1]
Make a hardcopy output of $S_{21}$ on the printer:
press [System]
click "Print"

Find the frequency $f_0$ at which $S_{21}$ is maximum using the marker MKR. Find the $3$ dB bandwidth $B$ of the amplifier. Record $|S_{21}|$ and $|S_{12}|$ in dB at this frequency and measure $|S_{11}|$ and $|S_{12}|$ using VSWR units.

5. 1 dB Power Compression:
We shall now measure the 1 dB Power compression point at the frequency $f_0$ of the amplifier under test.

The network analyzer E5071B offers the capability (not always associated with a network analyzer) to sweep the RF input power. An accurate measurement of the output power of an amplifier requires a special calibration procedure to account for the loss through the test set between the amplifier output and the receiver input. For the sake of simplicity we shall bypass here this special calibration. Our measurement should give us the amplifier’s response to a power ramp. Record $|S_{21}|$ in dB of the amplifier for -10 dBm input power $P_{in}$. [Note: $P_{in}$ in dBm is defined as $10 \log (P_{in}/1 \text{ mW})$.] From this response we shall determine the input power at which the gain $|S_{21}|$ is compressed by 1 dB.

Measurement procedure:

- Select a power sweep at the frequency $f_0$ (e.g. 2 GHz)
  press [Sweep Setup]
click "Sweep Type"
click "Power Sweep"
click "Power"
click "CW Freq"
enter [G/Hz]

- Set the stimulus parameters. Power levels must be set so that the amplifier is forced into power gain compression. The range of the E5071B’s source is from -10 to +10 dBm.
  press [Meas]
click "S21"
press [Format]
click "Log Mag"

[Start][×10][×1]
[Stop][×10][×1]

- Use the marker [MKR] to find the input power for which a 1 dB drop in the amplifier’s gain occurs relative to the small signal gain.

6. Fill the Lab Report # 1 given in Appendix 6.10. Append the plots which documents steps 4, and 5.

3.3 PART II: Measurement of the Noise Figure

Low noise RF and microwave amplifiers are used at the front end of microwave communication systems to amplify the weak signals received by the antenna or dish. The signal received from a satellite are typically in the order of -120 dBm (10^-15 Watt) which corresponds approximately to a voltage of 0.2 \( \mu \)V across a resistor of 50 \( \Omega \).

The Noise Figure \( F \) is defined (see Gonzalez p. 296) as the ratio of the output noise power \( P_{No} \) of the amplifier divided by the amplifier gain \( G_A \) and the input noise power \( P_{ni} \).

\[
F = \frac{P_{No}}{P_{ni} G_A}
\]

The noise figure \( F \) of an amplifier is larger than 1 because it is a measure of the noise added by the amplifier to the output noise power. If the amplifier did not add noise we would have \( F = 1 \) or 0 dB. The bandwidth \( B \) of the amplifier and its noise figure \( F \) will permit us to calculate the minimum detectable signal input power \( P_{i,mds} \) (see Gonzalez p. 354).

The measurement of the noise figure is quite simple with the HP8970B meter.

1. **Power up:**  
   Turn the HP8970B Noise Figure meter by pressing the LINE switch ON. If the noise figure meter is already on press RESET. The noise Figure Meter performs a quick internal check and frequency calibration.

2. **Set frequency parameters:**  
The HP8970B measures the noise figure versus frequency from 10 to 1800 MHz. The default START, STOP and START frequencies are 10, 1600 and 20 MHz respectively. To extend it to 1800 MHz simply press STOP FREQ 1800 ENTER. [Note: The noise figure of the low noise amplifier 13LNA at its center frequency \( f_0 \) cannot be measured as it is above 1800 MHz.]
3. **Calibration:**

   Connect (using a thru) the Noise Figure Meter as shown below

   ![Diagram](image1.png)

   **Figure 3.2: Noise Figure Meter Calibration**

   Press CALIBRATE twice. The HP8970B measures its own noise figure. The noise figure meter should make two frequency sweeps. If an error occurs you might have to increase the number of measurements averaged by pressing INCREASE in the SMOOTHING block and try again. If after several trials an error is still occurring you might have to limit the measurement to 1600 MHz.

4. **Measurement of the Noise Figure of the Amplifier:**

   Once the noise figure meter has been calibrated it is possible to measure the corrected noise figure and gain of the device under test (DUT). [Note: This noise figure is corrected because the meter removes its own noise figure from the measurement results.]

   - Connect the amplifier between the noise source output and the meter input as shown below:

   ![Diagram](image2.png)

   **Figure 3.3: Noise Figure Measurement**
• To measure the noise figure and gain press the key NOISE FIGURE AND GAIN. The INSERTION GAIN display shows the device gain and the NOISFE FIGURE display shows the noise figure of the DUT at room temperature at the displayed frequency.

• Connect the HP6237 power supply to the amplifier. Respect the polarity! See earlier instructions regarding the use of the power supply. Notice the reduction of the noise figure once the amplifier is powered.

• Measure the noise figure and gain at 1800 MHz by entering the frequency.

• Measure the noise figure versus frequency by pressing the [SINGLE] key to obtain a single sweep.

• Plot the data measured.
  
  – In order to plot, the noise figure meter must be the controller of the HPIB (IEEE488) network. Verify that nobody is plotting with the HP8378C and inform the other users that the noise figure meter will temporarily become the controller of the HPIB network.
  
  – Press 48.0 SPECIAL function for the noise figure meter to become the controller.
  
  – Press 47.0 SPECIAL FUNCTION to verify that the plotter is on the HPIB network.
  
  – Turn the plotter on and introduce a sheet of paper.
  
  – Press 25.0 SPECIAL FUNCTION to start the plot.
  
  – Once the plot is done press 48.1 SPECIAL FUNCTION to free the HPIB network from the control of the noise figure meter.

5. Complete the Laboratory Report #1 with the noise figure data. Append you noise figure plot. Calculate the power of the minimum detectable input signal $P_{i,mds}$ of this amplifier at $f_0$ (you will need to extrapolate $F$ at $f_0$).
4. LABORATORY 2:
Nonlinear Response of Amplifiers under Multitone Excitations

4.1 Introduction and System Diagram

Non-linear effects in RF power amplifiers is a topic of increasing importance in modern communication systems. The nonlinearity of an amplifier is indeed responsible for the introduction of spurious signals in the RF frequency band of operation and in adjacent bands. In this laboratory you will conduct 3 different experiments to measure non-linear effects arising under multitone excitations approximating modulated signals used in communication systems. Experiments I, II and III, all make use of the system shown in Fig. 4.1. The system should have been preinstalled by the TA and no additional connection will be required. It is however necessary to power up all the instruments. Please study this circuit diagram and identify the various components before doing this lab. The RF signal is generated by the RDL IMD Simulator which is a multitone generator combining the output of up to 7 (6 in our experiments) independent phase-locked RF sources (the 8th channel has been damaged). The RF signal after a 20 dB attenuation is sent to an HP 8447B amplifier. In its linear regime this amplifier can provide up to 22 dB gain in the 0.4 to 1.3 GHz frequency range. 17 dB gain was measured in our experiment below, certainly due in part to gain saturation as the PA maximum output is stated to be 3 dBm. At the output a 20 dB direction coupler is used to inspect the RF signal using a spectrum analyzer. The RF signal is then sent to a 17 level (needs 17 dBm at the LO input) mixer for down conversion. The conversion loss from RF to IF is around 6 dB. The resulting IF signal is then sent to a digital scope for inspecting the RF envelope.

To acquaint yourself with the typical power levels at the different nodes in the system diagram of Figure 4.1, please refer to Table 4.1 which gives the data measured for an input power of -31 dBm on the RF amplifier.

The power level on each node were measured with the MS601B spectrum analyzer with the IMD Simulators SYS ATTEN (internal system attenuator inside RDL IMD Simulator) set to 15 dB, FREQUENCY set to 869 MHz and ATTEN set to 0 dB for each channel. Corresponding power level values shown on MS610B are listed below.

The 20 dB fixed attenuator creates 20 dB of attenuation from node A to B. The Amplifier amplifies the input signal (node B) and outputs a stronger signal (node C). The directional Coupler let a very small fraction of its input power (node C) couple to node D while the remainder of the signal power is available at node . So the power level at node C and E should be essentially equal. Node D is connected to the input of the Spectrum Analyzer. So
Figure 4.1: EE723 Lab2 System Diagram

<table>
<thead>
<tr>
<th>Node</th>
<th>Measured Power Level (dBm)</th>
<th>Actual Power Level (dBm) with MS610B</th>
<th>After 9dB correction</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>-12</td>
<td></td>
<td>-3</td>
</tr>
<tr>
<td>B</td>
<td>-32</td>
<td></td>
<td>-23</td>
</tr>
<tr>
<td>C</td>
<td>-14</td>
<td></td>
<td>-5</td>
</tr>
<tr>
<td>D</td>
<td>-36</td>
<td></td>
<td>-27</td>
</tr>
<tr>
<td>E</td>
<td>-14</td>
<td></td>
<td>-5</td>
</tr>
<tr>
<td>F</td>
<td>8</td>
<td></td>
<td>17</td>
</tr>
</tbody>
</table>

The power level on node C is approximately 22 dB LARGER than the power level measured by the Spectrum Analyzer at node D. During your data analysis, DO REMEMBER these power level relationships:

\[ P_C = P_E \quad \text{and} \quad P_C_{\text{dBm}} = P_D_{\text{dBm}} + 22 \text{ dB} \]

Serving as a Local Oscillator for the Mixer, the signal on node F which is taken from the rear panel originates from channel #7 of the RDL IMD Simulator. After the down-conversion performed by the Mixer, the resulted IF signal is sent to the input channel #1 of the digitized Oscilloscope.
To simplify the lab procedure we are not calibrating the Spectrum Analyzer MS610B. Normally the GAIN ADJ is used to adjust the reading using a reference signal. Instead in this experiment we will keep the GAIN AGJ in its lowest setting (turned all the way in the reverse clockwise direction). Under such an uncalibrated but sufficiently reproducible condition the displayed power level value is approximately 9 dB lower than the actual value:

\[
Actual\ Power\ Level_{\text{dBm}} = MARKER\ LEVEL_{\text{dBm}} + 9\ dB
\]

DO REMEMBER this fact for your final data analysis.

4.2 EXPERIMENT I

4.2.1 Laboratory Goal

In this first experiment you will learn to use the various pieces of equipment:

* The multisine generator (IMD Simulator)
* The spectrum analyzer (MS610B)
* The digital scope (HP 54503A Oscilloscope)

and develop a method for estimating the peak RF power which you will calibrate for a constant envelope single-tone signal by comparing your time domain and frequency domain measurements.

4.2.2 Output Data To Be Recorded

* power shown on the spectrum analyzer
* max signal level (Vp-p/2) on the digital scope for \( I (\phi_7=0^\circ) \)
* max signal level (Vp-p/2) on the digital scope for \( Q (\phi_7=90^\circ) \)
* These data are recorded for 3 different power levels obtained by setting the IMD Simulators SYS ATTEN. to 5, 10 and 15 dB respectively.
4.2.3 Measurement Procedure

(0) Power Up All Equipment

Power up the following equipment (if not already on):

- ANRITSU SPECTRUM ANALYZER MS610B
- HP 54503A DIGITIZING OSCILLOSCOPE*
- RDL,Inc INTERMODULATION DISTORTION SIMULATOR

*The oscilloscope is powered by using the switch on the right leg of the bench table

(1) Configure Intermodulation Distortion Simulator (IMD Simulator)

The IMD Simulator uses 3 primary menus to access all functions. Each of these menus may be accessed using the right arrow key →. The Data Entry keys include digits from 0 to 9, the 4 arrow keys (←,→,↑,↓) and the ENT key.

In Experiment I only channel #1 and #7 are activated. Channel #7 will be the LO signal in the whole lab. Channel #1 here serves as the RF signal in this first experiment. This one-tone RF signal is amplified by the HP8447B Amplifier and then the amplified signal goes to Mixer ZLW-11H. For Channel #7 which goes to the LO terminal of the Mixer ZLW-11H, we will use 2 different phases for generating the I and Q signals.

Channel #1: $f_1=869$ MHz, $\phi_1=0$ degree
Channel #7: $f_7=868.5$ MHz, $\phi_7=0$ and 90 degree

(1a) Activate Channel #1

- Press right arrow → to switch to the CHANNEL CONFIGURATION menu.
- Press down arrow key ↓ until you have an asterisk "*" next to CHAN #1 in the channel display list.
- Press digit "1" to activate CHAN #1 using TOGGLE ON.
- Press digit "2" to select CHANGE FREQUENCY.
• Press digit ”8”, ”6”, ”9” and then ENT to set 869 MHz for CHAN #1.

• Press digit ”3” to select CHANGE ATTENUATION.

• Press digit ”0” and then ENT to set 0 dB attenuation for CHAN #1.

• Press right arrow → to switch to the PHASE CONTROL menu.

• Press digit ”5” on keypad to select MANUAL PHASE control. Press digit ”0” and then ENT to set 0 degree for the phase of CHAN #1.

(1b) Activate Channel #7 (Set phase to 0 degree)

Procedures are similar to those for activating channel #1.

• Press right arrow → to switch to the CHANNEL CONFIGURATION menu.

• Press down arrow key ↓ until you have asterisk ”*” labeled on CHAN #7 in the channel display list.

• Press digit ”1” to activate CHAN #7 using TOGGLE ON.

• Press digit ”2” to select CHANGE FREQUENCY.

• Press digit ”8”, ”6”, ”8”, ”5” and then ENT to set 868.5 MHz for CHAN #7.

• Press digit ”3” to select CHANGE ATTENUATION.

• Press digit ”0” and then ENT to set 0 dB attenuation for CHAN #7.

• Press right arrow → to switch to the PHASE CONTROL menu.

• Press digit ”5” on keypad to select MANUAL PHASE control. Press digit ”0” and then ENT to set 0 degree for the phase of CHAN #7.

(1c) Set System Attenuation

• Press right arrow → to switch to the SYSTEM CONTROL menu.

• Press digit ”2” to select OUTPUT ATTEN.

• Press digit ”5” and then ENT to set 5 dB system attenuation.
(1d) Check IMD Simulator Status

The four parameters displayed on screen should be the same as shown below:

IEEE:OFF SAMPLE:INTERNAL SYS ATTEN:= 05DB PHASE:MANUAL

Only CHAN #1 and #7 should be ON now, and other channels should be OFF. Follow the same procedure as for setting the channel ON to toggle unwanted channels to OFF if they are ON.

(2) Power up HP8447B Amplifier

(3) Locate Spectrum on Spectrum Analyzer MS610B

- Check equipment status.
  
  TRIG: FREE RUN   SCALE: 10dB/div   VIDEO FILTER: OFF

- Press CENTER FREQ, set to 869 MHz.

- Press FREQ SPAN, set to 20 MHz.

- Press REF LEVEL, set to 20 dBm.

- Press INPUT ATTEN, set to 30 dB.

- Press RBW, set to 9 KHz.

- Press Sweep Time, set to 0.1 S.

- Use the knob on the front panel to move the marker in the spectrum, the value shown in MARKER LEVEL represents the power level (dBm) at your marker position (the vertical center line of the spectrum display screen)

  The Spectrum Analyzer MS610B is not calibrated and the power level value displayed is approximately 9 dB lower than the actual value. So do use the relation:

  \[
  \text{Actual Power Level}_{\text{dBm}} = \text{MARKER LEVEL}_{\text{dBm}} + 9 \text{ dB}
  \]

- Record power on the spectrum analyzer; Record I data on the scope respectively (see Section 4.2.2 for details).
(4) Measure Amplitude of IF Signal on HP 54503A Oscilloscope

- Press CHAN on keypad of HP 54503A Oscilloscope. By using soft keys (one column to the right of display screen), select Channel #1 “on”. Set 100mV/div using the knob or by inputing digit numbers. Set ”Offset”=0 V. Set both ”dc” and ”BW lim” for 50 Ω (not 1MΩ).

- Press TRIG, use soft keys to select ”auto” and ”edge” mode.

- Press TIMEBASE, set ”1us/div” by knob or inputing digit numbers.

- Press DISPLAY, use soft key to set ”normal” mode; Set ”Connect dots” as ”ON”.

- Press DEFINE MEAS, use soft key to set ”statistics” being ”off”.

- Press the only blue colored button on keypad. Press FREQ and then digit ”1” (i.e. channel 1) to obtain the frequency of the signal displayed on the scope.

- Press the blue colored button again. Press V P-P and then digit ”1” (i.e. channel 1) to obtain the signal voltage peak-to-peak value (Vp-p) of the signal displayed on the scope.

- Record Data

(5) Reconfigure CHAN #7 (Set phase to 90 degree)

- Use the same procedure as above to activate CHAN #7 but set now its phase to 90 degree.

- Record Q data.

(6) Acquire Data for Another Two Cases

- Set IMD Simulator’s SYS. ATTEN. to 10 dB, then follow the same measurement procedure.

- Set IMD Simulator’s SYS. ATTEN. to 15 dB, then follow the same measurement procedure.
4.2.4 Data Analysis

A modulated RF signal can be written:

\[ x_{RF}(t) = A(t) \cos[\omega t + \phi(t)] = I(t) \cos(\omega t) - Q(t) \sin(\omega t) \]

with

\[ I(t) = A(t) \cos \phi(t) \]
\[ Q(t) = A(t) \sin \phi(t) \]

The RF signal envelope amplitude \( Env(t) \) is recovered from the measured data using:

\[ Env(t) = |A(t)| = \sqrt{I(t)^2 + Q(t)^2} \]

In this first experiment the envelope is constant in time. Evaluate the mixer attenuation coefficient \( L_c = \alpha^2 \) from the following relations:

\[ V_{rf} = \alpha \times Env \]

\[ P_{rf,avg} = P_{rf,peak} = \frac{V_{rf}^2}{2Z_o} = \alpha^2 \frac{Env^2}{2Z_o} \quad \text{with} \quad Z_o = 50 \, \Omega \]

\[ \alpha^2 = \frac{2Z_o P_{rf,avg}}{Env^2} \]

\[ \alpha (\text{dB}) = 20 \log(\alpha) = 10 \log(\alpha^2) = P_{rf,avg}|_{\text{dBm}} - \frac{Env^2}{2Z_o}|_{\text{dBm}} \]

\( \alpha \) (dB) should be around 6 dB. It results that we have:

\[ P_{rf,peak}|_{\text{dBm}} = \alpha (\text{dB}) + \frac{Env^2}{2Z_o}|_{\text{dBm}} \]

Verify that this relation works for the different IMD Simulator’s SYS ATTEN. levels of 5, 10 and 15 dB by tabulating and comparing the \( \alpha \) (dB) obtained for these different power values.

The envelope \( Env \) will be used to measure the peak RF power \( P_{rf,peak} \) and calculate the Peak to Average Ratio (PAR) in Experiment III.

\[ PAR(\text{dB}) = 10 \log \left( \frac{P_{rf,peak}}{P_{rf,avg}} \right) = P_{rf,peak}|_{\text{dBm}} - P_{rf,avg}|_{\text{dBm}} \]
4.3 EXPERIMENT II

4.3.1 Laboratory Goals

In this experiment you will

* observe the spectral regrowth generated by the amplifier for a multitone RF signal using the spectrum analyzer (the RF envelope displayed on the digital scope will not used here).

* measure the variation of the spectral regrowth generated by the amplifier with the input power level.

4.3.2 Output Data To Be Recorded

* Sketch the power spectrum for 3 different power levels set by the IMD Simulator’s SYS ATTEN (5, 10, 15 dB, respectively).

* Record the signal power in the central band and in the lower and upper side bands.

4.3.3 Measurement Procedure

Do NOT change the circuit connections. Many of the measurement operations will be similar to those in Experiment I.

(1) Configure Intermodulation Distortion Simulator (IMD Simulator)

In Experiment II channels #1 to #7 are activated. Channel #7 still serves as LO signal; Channel #1 to #6 are used to generate a 6-tone RF signal.

In Experiment II, Channels #1 to #6 have frequency spacing of 500 KHz, and all 7 channels have 0 phase configured.

(1a) Activate Channel #1

Using the same procedures as in Experiment I, activate Channel #1, and set the frequency, channel attenuation and phase for channel #1 to 869.0 MHz, 0 dB and 0 degree respectively.
(1b) Activate Channel #7 (Set 0 degree phase)

Using the same procedure as above, activate Channel #7, and set the frequency, channel attenuation and phase for channel #7 to 868.5 MHz, 0 dB and 0 degree respectively.

(1c) Activate Channel #2 to Channel #6 (Channel # 8 set to OFF)

Using the same procedure as above, activate the remaining channel and set the frequency, channel attenuation and phase for channel #2 to #6 (frequency stepped by 0.5 MHz). When done the final configuration obtained should be as follow:

<table>
<thead>
<tr>
<th>CHAN</th>
<th>FREQUENCY</th>
<th>ATTEN</th>
<th>PHASE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ON</td>
<td>869.00 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>2 ON</td>
<td>869.50 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>3 ON</td>
<td>870.00 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>4 ON</td>
<td>870.50 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>5 ON</td>
<td>871.00 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>6 ON</td>
<td>871.50 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>7 ON</td>
<td>868.50 MHZ</td>
<td>00.0 DB</td>
<td>000</td>
</tr>
<tr>
<td>8 OFF</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(2) Power up HP8447B Amplifier

(3) Locate the Spectrum on Spectrum Analyzer MS610B

Starting from the same configuration as in Experiment I (keep SYS. ATTEN=5 dB), only change the following two settings on the spectrum analyzer MS610B:

- Press REF LEVEL, set to 0 dBm.
- Press INPUT ATTEN, set to 10 dB
- Record the power level of the 6 in-band (IB) tones and the 6 adjacent lower side-band (LSB) tones and the 6 adjacent upper side-band (USB) tones (for a total of 18 tones).
Remember that the Spectrum Analyzer MS610B is not calibrated, so the power level value shown is approximately 9 dB lower than the actual value and we have:

\[
\text{Actual Power Level}_{\text{dBm}} = \text{MARKER LEVEL}_{\text{dBm}} + 9 \text{ dB}
\]

(4) **Measure Data for Another Two Cases**

- Set IMD Simulator’s SYS. ATTN. to 10 dB, and follow the same procedure as above to record data (18 tones) shown on spectrum analyzer.

- Set IMD Simulator’s SYS. ATTN. to 15 dB, and follow the same procedure as above to record data (18 tones) shown on spectrum analyzer.

4.3.4 **Data Analysis**

* Tabulate the variation of the ACPR with the output power \( P_{rf,\text{avg}} \). Note that you have to convert back to mW before adding the power levels in the various bands.

The ACPR is defined here as:

\[
ACPR = \frac{P_{IB}(6\text{tones})}{P_{LSB}(6\text{tones}) + P_{USB}(6\text{tones})}
\]

4.4 **EXPERIMENT III**

4.4.1 **Laboratory Goals**

In this experiment you will:

* record the variation of the spectral regrowth with the PAR

* correlate ACPR and PAR under a fixed power level for three different phase conditions

4.4.2 **Output Data To Be Recorded**

For each of the 3 phase conditions to be considered

* Sketch the spectrum displayed on the MS610B.

* Record the signal power in the in-band and in the lower and upper side bands shown on MS610B (18 tones total).
* Measure $I$ and $Q$ on the oscilloscope and calculate the envelope $Env$

### 4.4.3 Measurement Procedure

Do NOT change the circuit connections. Again the measurements are similar to those in Experiment II. The measurement include 3 parts:

* The random phase condition
* The minimum ACPR phase condition
* The maximum ACPR phase condition

#### (1) Measurement for The Random Phase Condition

Starting from same configuration as in Experiment II (with SYS. ATTEN=15 dB), execute the following steps:

- On spectrum analyzer MS610B press REF LEVEL, set to -10 dBm; press INPUT ATTEN, set to 0 dB
- On the IMD Simulator press right arrow key → to switch to the SYSTEM CONTROL menu, press OUTPUT ATTEN and input 15 dB system attenuation.
- On the IMD Simulator press right arrow key → to switch to the PHASE CONTROL menu, press RANDOMIZE PHASE
- Record the 18 tone data on the spectrum analyzer.
- On the IMD Simulator press down arrow key ↓ till the asterisk "*" is located next to CHAN #7 in the channel display list. Press digit "5" for MANUAL PHASE input. Press digit 0 and then ENT to set 0 degree phase for CHAN #7.
- Record the $I$ data on the scope (see Experiment I).
- Change phase of CHAN #7 to 90 degree manually, record the $Q$ data.

#### (2) Measurement for The Minimum ACPR Phase Condition

Procedures are similar to measurement for Random Phase Condition. Starting again from the same configuration as in Experiment II (with SYS. ATTEN=15 dB), execute the following steps:
• On spectrum analyzer MS610B press REF LEVEL, set -10 dBm; press INPUT ATTEN, set 0 dB

• On IMD Simulator press right arrow key → to switch to the SYSTEM CONTROL menu, press OUTPUT ATTEN and input 15 dB system attenuation.

• On IMD Simulator press right arrow key → to switch to the PHASE CONTROL menu, press PHASE PEAKING. Wait for about 20 seconds.

• Once the IMD Simulator is done searching for the PHASE PEAKING condition record the 18 tone data on the spectrum analyzer.

• On IMD Simulator press down arrow key ↓ till the asterisk "*" is located next to CHAN #7 in the channel display list. Press digit "5" for MANUAL PHASE input. Press digit 0 and then ENT to set 0 degree phase for CHAN #7.

• Record the I data on the scope (see Experiment I).

• Change phase of CHAN #7 to 90 degree manually, record the Q data.

(3) Measurement for The Maximum ACPR Phase Condition

Procedures are similar to measurement for Random Phase Condition. Starting from same configuration as in Experiment II (but keep SYS. ATTEN=15 dB), execute the following steps:

• On spectrum analyzer MS610B press REF LEVEL, set -10 dBm; press INPUT ATTEN, set 0 dB.

• On IMD Simulator press right arrow key → to switch to the SYSTEM CONTROL menu, press OUTPUT ATTEN and input 15 dB system attenuation.

• On the IMD Simulator press right arrow key → to switch to the PHASE CONTROL menu, press PHASE NULLING. Wait for about 20 seconds.

• Once the IMD Simulator is done searching for the PHASE NULLING condition record the 18 tone data on the spectrum analyzer.

• On IMD Simulator press down arrow key to get asterisk "*" labeled on CHAN #7 in the channel display list. Press digit "5" for MANUAL PHASE input. Press digit 0 and then ENT to set 0 degree phase for CHAN #7.
- Record the I data on the scope (see Experiment I).
- Change phase of CHAN #7 to 90 degree manually, record the Q data.

### 4.4.4 Data Analysis

* Estimate the total power in the main band (6 tones), 1st lower sideband (6 tones) and 1st upper sideband (6 tones)
* Calculate the ACPR
* Calculate the PAR: \[ PAR(dB) = P_{rf,peak} |dBm - P_{rf,avg}|dBm \]
* Tabulate the variation of the ACPR versus the PAR for the 3 phase conditions and explain the result

### 4.5 Appendix

**Instrument List:**

- ANRITSU SPECTRUM ANALYZER MS610B
- HP 54503A DIGITIZING OSCILLOSCOPE
- RDL,Inc INTERMODULATION DISTORTION SIMULATOR

**Device List:**

- Mini-Circuits ZLW-11H (Mixer)
- HP 8447B AMPLIFIER
- HP 778D DUAL DIRECTIONAL COUPLER
5. LABORATORY 3:
Design of a Microwave Amplifier

Laboratory Goal: The goal of this laboratory is to design, fabricate and test a microwave amplifier. Three possible designs are proposed:

- Narrow band/maximum gain (2 GHz center frequency)
- High Q Narrow band/maximum gain (2 GHz center frequency)
- Narrow band/low noise (1.5 GHz center frequency)

Approximate Duration: 6 hours

5.1 Design

5.1.1 Important Design Constrains:

1. The amplifier must be stable for 50 Ω source and load impedances at all frequencies.

2. The amplifier will be designed using the NPN transistor AT42085 biased with 10 mA collector current and 8 V collector emitter potential. Typical values for the scattering and noise parameters of the AT42085 transistor at this bias point are given in Appendix 6.6. It is expected that the scattering parameters measured in Lab #2 should be more accurate than those of Appendix 6.6 for your amplifier design.

3. The input and output matching network will be designed for a 50 Ω source and load impedance. The matching circuit will be realized using the technique of your choice. Note however that short circuit stubs and lumped elements cannot be used.

4. The microstrip will be constructed on a Duroid substrate. Get the spec of this substrate from your instructor. The pattern will be fabricated by the Quick Circuit machine.

5. The biasing of the bipolar transistor will be realized through the built-in bias Tees of port 1 and port 2 of the Network Analyzer (see Appendix 6.9) using the NPN bias network provided in the laboratory. This active biasing circuit is analyzed in Appendix 6.9.

5.1.2 Design Procedure:

The design procedure of the amplifier is based on the following steps:
1. See the instructor to obtain the values you should use for the dielectric thickness, dielectric constant $\varepsilon_r$ and tangent loss and copper thickness of the Duroid microstrip substrate you will use for your amplifier. For copper we have $\text{RHO} = .84$. Using lineCalc calculate the width and length of a 50 $\Omega$ line of electrical length of 360 degree (length equal to one wavelength).

2. To start your design you need realistic scattering parameters for your transistor which accounts for the way the transistor is connected on the microstrip board. You will use the transistor S-parameter file you acquired in Laboratory 2. **However the via holes used to connect the emitter of the transistor to the ground introduces a parasitic inductance.** It results that the scattering parameters of the transistor plus connecting bed seen by the circuit departs from those you measured for the transistor. This is **very important** as this affects the amplifier design, stability and noise performance. **Via Hole elements must therefore be introduced between the transistor emitters and the ground to account for this parasitic inductance in your simulation.** Also the difference between the width of the transistor pads and the width of your transmission lines must be bridged by a tapered line. TO FACILITATE your design a transistor footprint subcircuit accounting for these parasitics is provided in Appendix 6.2.

3. Using ADS calculate the new scattering parameters for the transistor plus its connecting bed. Make sure that the **correct microstrip substrate** has been used. Make sure that the tapered line connects to the correct width of the line connected to it (for example your design might use a line of 50 $\Omega$). Make sure to use your own S-parameters measured in Laboratory 2. See Appendix 6.7 to learn how to save the new transistor S-parameter data in a file and Appendix 6.8 to verify the format used by ADS.

4. Do a first cut design (see Homework #2) using the Smith Chart and the new scattering parameters resulting from this circuit.

5. Implement your first cut design in ADS using transmission lines. **Do not optimize yet this ideal circuit as you still need to design a realistic microstrip layout.** Compare your first cut design with your Smith Chart design to verify that the first cut design is correct. This first cut design provides you with a starting point for the design of the amplifier with microstrip lines.

6. Using Linecalc (see Appendix 6.3) calculate the width and length of the microstrip line required for your design (use MLIN in linecalc). Actually or a 50 $\Omega$ line you can just scale the wavelength obtained in the first step for $E=360$ by the appropriate electrical length $E$ you need to implement.
7. Implement your amplifier design in ADS using microstrip lines. You can compare the microstrip and transmission line implementation but do not optimize yet as this is not the final circuit yet. Indeed the final circuit you are going to optimize should realistically represent the circuit you are going to fabricate. Introduce MCRoss, MTEE, MSTEP, MOPEN and VIA2 to account for all the discontinuities present in your layout (see Appendix 6.2). Use the transistor TEST BED: trans_sim.dsn introduced in Appendix 6.2 to simulate or optimize your design. Update FILE=at42085N in trans_sim.dsn with the S-parameters measured for your own transistor. Once your circuit has been optimized replace the TEST BED file tran_sim.dsn by trans_lay.dsn to be able to generate the correct transistor layout. Then you can generate the layout of your complete circuit using the synchronize command describe in Appendix 6.2 to verify that the layout is acceptable (not too long) and that all the discontinuities are properly modeled in your circuit. ALSO your layout should not violate the design rules given in the next section. You can use MBEND to reduce the length when appropriate. If you are adding biasing lines (see low noise amplifier design comment below) includes them in your design to verify if they impact the RF performance and make sure they are properly implemented in your layout!

8. The footprint for the input and output SMA connectors will be added to your layout by the TA (unless otherwise instructed). JUST make sure to add a 50 Ω ununcombered microstrip line of a quarter of an inch (1/4 in) at both the input and output of the amplifier to permit the installation of the connectors without interfering with your circuits. The connector footprint should not affect your amplifier design apart from reducing its gain and increase its noise figure due to the SMA connector to microstrip launcher loss.

9. Once a realistic circuit modeling of the final amplifier circuit to be layout has been implemented then it is worthwhile to optimize your design. Plot $|S_{21}|$ (dB) and $|S_{11}|$ and $|S_{22}|$ versus frequency.

10. Investigate the stability of the amplifier from .1 to 6 GHz. For this purpose generate the stability circles of the amplifier using ADS. This permits to verify whether or not the final amplifier is stable for the 50 Ω load used at the input and output. Unconditional stability is usually too much of a constrain to target. It is generally preferable to avoid using resistive loading since it reduces the gain unless if you are purposely designing a broad band amplifier. If resistive loading is to be used note that only series loading can be used since a shunt loading requires a capacitor to prevent shorting the bias (see the instructor if you absolutely need a shunt loading). A resistor in series with a high impedance (very narrow) microstrip line (inductance) and DC
block capacitor will realize a resistive loading with would stabilize the transistor at low frequency but will not decrease its gain at high-frequency.

11. If you are building a low noise amplifier you need to have it fit inside a box to shield it from the environment noise and approach the simulated noise figure. In such a case the entire circuit should fit inside a specific rectangle size with an appropriate safety margin, say of 1/4 of a inch, from the box’s edge. The dimension of the box available for your design is shown in Figure 5.1.

Note that the box will also impact the synthesis of the microstrip line width and you need to set the correct value for HC, W1 and W2 for the distance of the microstrip line to the cover and wall of the box (see Appendix 6.3). Also to measure its noise characteristic, an external biasing TEE will need to be implemented on the microstrip board to bias the transistor (usually the network analyzer is used to perform this function). The best approach is to include the bias TEE as part of the amplifier layout. You can discuss these various options with the instructor.

5.1.3 Summary of the Design rules associated with the Fabrication

Your circuit will be fabricated using the Quick Circuit machine. Your design should therefore account for the physical constrains introduced by the layout and the fabrication.

Read Appendix 6.2 for more information on the layout procedure. In particular the layout tool introduces the following constrains

- Via holes (connection to the ground plane through the microstrip board) should be modeled using the VIA2 model.
- All the line discontinuities should be modeled using elements such as MSTEP, MCROSS, MTEE, MBEND, MCLIN, MGAP and so on.
- Use an MGAP statement for the foot print of chip resistors and chip capacitors.

A few design rules should be respected to permit the succesful fabrication of your design:

- The minimum line width is 10 mils
- The minimum separation between two lines (line gap) is 11 mils.
- Connectors (SMA launchers) should not be closer than 2 cm for easing their connection to the SMA cables.
1: The microstrip lines should not extend beyond the design limits.
2: The center pin of the connectors can be placed anywhere inside the design limits including the edge.

Figure 5.1: Box Dimension
• A 50 Ω line of 1/4 of an inch should be reserved to attach the SMA Connector (SMA launcher).

• MINIMIZE the area of your circuit. The Quick Circuit fabrication time and therefore tool-wear is proportional to the amount of copper to remove! Please minimize the foot-print of your circuit. For the low-noise amplifier the box constrains the size of the layout.

Please set up an appointment with the TA before the Layout Approval by TA date specified in your syllabus, to have your layout checked and approved for fabrication by the TA.

Expect up to a week turn around for fabrication.

The circuits are all fabricated together in a single batch. Your design should therefore be delivered to the Teaching Assistant by the Final Layout Delivery date specified in your syllabus.

5.2 Fabrication Procedure

The fabrication of your circuit with Quick Circuit involves multiple steps which are performed by the Teaching Assistant. First your layout is finalized with the addition of the connector footprints and the transistor footprint determined and the circuit bound is determined. The file is then converted in Gerber format. The fabrication process is manually designed from your layout using a tool call Isolator. Finally the fabrication process is executed.

Your circuit built with the Quick-Circuit machine should already include all the transmission lines and holes required and no additional drilling or cutting should be required. Therefore the remaining fabrication steps are relatively few.

List of parts:

- Microwave transistor
- Microstrip board
- 2 SMA connectors
- PNP biasing network (provided)
- bias TEE are required for oscillators and LNA

Fabrication Procedure:

1. Solder the transistor using the soldering iron (see Appendix 6.4). Use a heat sink to
avoid damaging the transistor. The transistor leads need to be flush with the microstrip board.

2. Fix the connectors on the circuit board. One of the four legs must be removed if this has not been done. The connector NEEDs to be flush with the ground plane. Solder the connectors using the soldering gun (see Appendix 6.4). Soldering to legs on the ground plane should be sufficient.

5.3 Measurement of Your Circuit.

Measurement procedure:

1. Make a 2 port calibration with the Network Analyzer or recall one saved on the HP disk drive for the same test cables.

2. Amplifier Biasing:

   • Connect Port 1 to the input of the amplifier (base of the NPN). Connect Port 2 to the output of the amplifier (collector of the NPN). The bias network should have the BNC connector labeled "base" connected to port 1 and the BNC connector labeled "collector" connected to port 2.

   • Turn on the HP 6237 power supply. The operating voltage should already be set to 10.1 V on the lower scale of the power supply meter (Do not increase it as this could damage the biasing network). [Note: The red banana cable should be connected to +18 V and the black cable to COM. The meter switch should indicate +18 V.]

   • Connect the NPN biasing network to the HP 6237A power supply. Respect the polarity (red with red, black with black)! The read part of the banana plug should be connected to +18 V and the black part of the banana plug to COM.

3. Measure and plot $|S_{21}|$ in dB versus frequency. At which frequency $f_0$ is the maximum gain? What is the 3dB bandwidth B? Measure and plot $|S_{11}|$ in dB.

4. Measure and plot $|S_{11}|$ and $|S_{22}|$ in VSWR units. Check how well the input and output networks are matched at the operating frequency.

5. Trim the open stub to improve the matching if necessary. You could also increase the stubs’ length by soldering a piece of copper tape.
6. If you experience difficulty (apparent noise) with the measurement of the scattering parameters with the Network Analyzer the amplifier might be unstable. You can verify it with the spectrum analyzer. The amplifier can be stabilized with a feedback resistance between collector and emitter or a shunt resistance (in series with a shunt capacitor) between base and emitter or collector and emitter.

7. Measure the input power for which the power gain drops of 1 dB.

8. For low noise amplifiers: Measure the noise figure at the design frequency.

5.4 Write Up

5.4.1 The Laboratory Report

Your report should include:

0. The complete laboratory 3 report sheet available in Appendix 6.10
1. Smith Chart Design
2. ADS circuit file and plot of S11, S22 and S21 (dB).
3. The drawing of the amplifier layout including all dimensions.
4. Plots of measured |S11|, |S22|, |S12|, and |S21| (dB or VSWR) versus frequencies.
5. Compare the measured and simulated data.
6. Discuss the design and fabrication problems encountered.

5.4.2 Poster Page

A poster page summarizing your design and experimental results should be included with the report. This poster page should include:

- Project Title
- Your names
- EE723, Fall Quarter 1995
• A project abstract describing the circuit targeted, the design approach and the performance obtained.

• A figure or two with short captions comparing the measured and simulated (ADS) circuit performance.

• A figure showing the layout generated by ADS.

A prototype Poster page in latex will be made available in the directory:
~roblin/latex/.
6. APPENDICES
6.1 How to Use ADS

The most current information for using ADS is available at:

http://eewww.eng.ohio-state.edu/ads

Usually to start ADS the following commands are used:

```
source /opt/local/ads/Startup.EE
ads
```

**TUTORIAL:**

If you are new to ADS you should go through the OSU tutorial:

http://eewww.eng.ohio-state.edu/ads/tutorial2.pdf

Check also our local ADS webpage: http://eewww.eng.ohio-state.edu/ads for access to the online ADS manual and other various tutorials found on the web.

**MOVING OR SHARING DESIGNS:**

Note that if you are moving your design files to another directory (or want to share them with someone else) you can copy both the .dsn and .ael files of the networks directory of your project directory. The .atf files are generated automatically.

**IMPORTANT NOTE:**

It is important not to exist ADS in an unconventional way. When you do ADS keeps running in the background even if you log off. We have only ten user licenses and we quickly run out of them if such processes are running in the background. The consequence is that other people (including yourself) stop being able to use ADS. **Please exit ADS using the exit command** in the Project menu.

If ever ADS dies or is exited unconventionally you need to kill all the unix process which are still running. For this purpose type:

```
ps
```

or type:

```
ps -ef | grep yourusername
```

Then kill the job number involving your name and hpeesof. For example by typing **ps -ef | grep roblin** the user roblin obtained among other thing:

```
roblin 14776 14775 0 Jan 5 pts/0 0:00 hpeesofviewer
```

To kill that job simply type:

```
kil -9 14776
```
6.2 How to Use the Layout Tool

ADS allows for the automatic generation of the layout from the circuit design.

To generate the layout you must have both the schematic and layout windows open. In the schematic window select the synchronize menu and the first item on this menu. ADS will highlight in red every circuit for which it has a corresponding layout or foot print. It does not have a foot print for capacitors. ADS starts by Port 1 and asks you where you want to locate it. Position (0,0) is just fine. The layout will stop whenever, starting from port 1, ADS runs into a device for which it does not have a layout. If you run again synchronize it will then ask for a new position (x,y) where to initiate the continuation of the layout. The best is to instead introduce elements which specify a foot print. For example for a capacitor you would use a chip capacitors. Simply place in parallel with the ideal capacitor a MGAP (microstrip gap) element and specify the width of the gap (width of the capacitor).

Note that the synchronize tool requires that you make use of the discontinuities MSTEP, MCROSS, MGAP, VIA2 and so on to establish a physical network. This is also required to obtained a realistic simulation of your circuits as these discontinuities introduces parasitics which impact the microwave performance of your circuit.

As you go through several layout trials you will want to RESET the layout window. For this purpose go to the file menu and select: clear layout and then synchronize again to obtain a new layout. Be careful if you synchronize in the layout window your schematic will be updated! Save your schematic ahead of time to avoid an unwanted modification of your schematic.

Sometimes the layout is not what you want. For example one can experience two lines intersecting. In the layout window you can then click on all the items you want to move together and then move them and rotate them where you want. All the editing commands apply to these items.

The layout is really occuring on several layers. The metal (cond) layer is the most important one for our process. However you need also to define the bound (size) of the circuit. In the Layout window go to the Draw menu and select the Select layer menu time. Select (you might have to scroll down the menu) the bound layer (not bond!). Then again select the Draw menu and select rectangle. Place your rectangle with the mouse. The Quick Circuit machine will cut the contour of your circuit using the bound you specified. If your circuit is to be placed in the box available refer to the section below discussing

Holes are also considered as a layer. There are automatically introduced by the via holes: VIA2 (see below).
Ask your instructor for more information on the substrate you will be using. For the final Quick Circuit design we usually use a thin substrate of 30 or 45 mils to obtain smaller microstrip linewidth. Changing the substrate MSUB affects the width of all the lines. Such a change can be easily implemented in your schematic if you define a variable say W1 for microstrip lines of same width. Then you can select the value type of the width to be a variable (instead of a parameter) and select this variable to be W1 in the value option menu.

**Via Holes:**

Via holes are used to establish a connection to ground. They also affect the simulation since a via hole behaves as an inductor. You need therefore to include them both for the simulation and the layout. Use the model VIA2 available in the microstrip library. W is the width of the square pad used on the top of the microstrip. The hole should be smaller than the width of the pad.

So in summary to perform a layout you need to update your schematic with MCROSS, MSTEP, MGAP, VIA2, bias line (for active circuits) and so on to obtain a realistic design. When you are done with your layout schedule an appointment with your instructor to discuss your layout/schematics and simulation before the scheduled fabrication (see EE723 syllabus).
6.3 How to Use LineCalc

How to Start and Use LineCalc:

1. You can start LineCalc from the ADS Circuit window from the TOOLS menu.

2. We will most likely use LineCalc to synthesize a microstrip line. Click on Select... and scroll down the menu to select MLIN (and not MCLIN) and click on OK.

3. Edit the various substrate parameters using the Modify Substrate key (you may need to scroll the menu or make the window bigger): $E_r$ is the effective relative dielectric constant, $Mu_r$ is the permittivity (1.0), $H$ is the thickness of the substrate, $Hu$ is the position of the cover (keep it large if there is none), $T$ the thickness of the copper line $Cond$ is the copper line conductivity: $4.878 \times 10^7 \text{m}^{-1}$, $TanD$ is the loss tangent, $Rough$ is the ideal surface roughness ($Rough=0$ is a very good approximation).

4. Edit the component parameters. Select the frequency $Freq$ targeted. $Wall1$ and $Wall2$ are the distance of the microstrip line from the side metallic walls. Set $Wall1 = 0$ and $Wall2 = 0$ to make them infinite (no walls) except maybe if you are making a low noise amplifier going into a box.

5. Set the characteristic impedance $ZO$ and effective electrical length $E_{Eff}$ you wish to obtain.

6. Click the up arrow to calculate the width $W$ and length $L$ of the microstrip line. Also calculated are $K_{Eff}$ the effective relative dielectric constant, the line attenuation $A_{DB}$ and the skin depth.

7. For more information use the on-line Help command.
6.4 Tips for Soldering

**Tips for soldering the transistor:**

1. Moisten the sponge of the soldering stand.
2. Turn on the soldering iron.
3. Set the temperature around 700°F.
4. The right light is blinking when the tip is hot.
5. Before soldering clean the soldering tip with the moisten sponge.
6. Fast soldering will prevent overheating of both the circuit and elements.
7. Avoid breathing the fumes.
8. Do not use too much solder.
9. When you solder the chip capacitor or resistor, one student can hold the element with tweasers and the other solder it.
10. When you solder the center conductor of the connector, you should solder both sides of the conductor.
11. Be careful when soldering the transistor on the copper tape. To prevent damaging your transistor use a heat sink such as a metal clip or pliers.

**Tips for soldering the connectors**

To solder the SMA connectors to the microstrip board use the soldering gun. **BE CAREFUL. You must plug it and press on the trigger to active it for the time you are soldering ONLY.**

*The SAFETY PRECAUTIONS are:*

1. Keep your soldering gun well away from all flammable material.
2. To avoid burns, always assume that the tip is hot.
3. Be sure the hot metal tip does not come in contact with the electrical power cord.
4. Before making any adjustment-removing or replacing a tip etc- make sure the gun is unplugged and cool.
5. Release the trigger whenever the tip is not in contact with work. NEVER EVER tape back the trigger.

6. Do not hold work in your hand if you can possibly avoid it. Use a vise, clamp or pliers.

7. Do not dip the tool into any liquid.

8. Many materials give off unpleasant fumes when heated-so always work in a well ventilated room.

9. Clean the tip by wiping it, when hot across a damp sponge or cloth- placed on a non-flammable surface, NOT held in the hand.

10. **AFTER USE, DISCONNECT** the soldering gun. allow the tip to cool completely, and store the tool in a safe place (out of reach of children).

11. Safety goggles are recommended to prevent hot materials from entering the eyes.

Do not use much solder to hold the connectors. We intend to reuse these connectors.
6.5 Caring About Connectors

The relatively inexpensive SMA connectors we use are not made to be connected more than a few times in their lifetime. You might notice gold particles on the white dielectric of the cables and standards which will affect the quality of your calibration and measurement. In such a case clean the cables and standards using a cotton swab and alcohol. Here are some general care and maintenance rules.

1. Do

   a) Keep connectors clean
   b) Extend sleeve or connector nut when you store it
   c) Place plastic end-caps after you use it
   d) Inspect all connectors carefully before every connection
   e) Look for metal particles, scratches, dents when you inspect it
   f) Align connectors carefully when you connect them
   g) Make preliminary connection lightly
   h) Turn connector nut only to tighten
   i) Use a torque wrench for final connection. Use the 5 lb-in torque wrench to connect a male SMA to a female SMA or a female precision 3.5 mm. Use the 8 lb-in torque wrench to connect a male precision 3.5 mm to female SMA connectors.

2. Do not

   a) Touch mating plate surfaces
   b) Set connectors contact-end down
   c) Use a damaged connector
   d) Apply bending force to connector
   e) Over tighten preliminary connection
   f) Twist or screw in connectors
   g) Tighten past “break” point of torque wrench
6.6 AT42085 S-Parameters and Noise Parameters

! File AT42085

! Vce=8V Ic=10mA

! S parameters, Common Emitter

| 0.1 | .72 | 50 | 26.52 | 152 | .014 | 73 | .90 | -16 |
| 0.5 | .66 | -139 | 11.23 | 103 | .035 | 36 | .53 | -32 |
| 1.0 | .65 | -168 | 5.95 | 84 | .037 | 39 | .45 | -33 |
| 1.5 | .65 | 175 | 4.06 | 71 | .045 | 46 | .43 | -36 |
| 2.0 | .65 | 163 | 3.06 | 60 | .054 | 51 | .42 | -41 |
| 2.5 | .66 | 157 | 2.51 | 55 | .063 | 60 | .42 | -42 |
| 3.0 | .68 | 149 | 2.07 | 46 | .072 | 65 | .41 | -48 |
| 3.5 | .68 | 141 | 1.79 | 38 | .085 | 64 | .43 | -55 |
| 4.0 | .69 | 133 | 1.57 | 29 | .104 | 64 | .45 | -61 |
| 4.5 | .69 | 125 | 1.41 | 21 | .119 | 63 | .46 | -66 |
| 5.0 | .69 | 114 | 1.28 | 12 | .139 | 58 | .47 | -71 |
| 5.5 | .71 | 103 | 1.17 | 3 | .161 | 55 | .44 | -76 |
| 6.0 | .75 | 91 | 1.07 | -6 | .177 | 49 | .40 | -85 |

! Noise Parameters, Common Emitter

| !FREQ | Fopt | !GAMMA | OPT | !RN/Zo
| !GHZ | dB | MAG | ANG | - |
| 0.1 | 1.1 | 0.05 | 16 | 0.13 |
| 0.5 | 1.2 | 0.06 | 77 | 0.13 |
| 1.0 | 1.3 | 0.10 | 131 | 0.12 |
| 2.0 | 2.0 | 0.24 | -179 | 0.11 |
| 4.0 | 3.5 | 0.46 | -128 | 0.25 |
6.7 Computer Data Acquisition for 2-Port S-parameters

NOTE: A new improved data acquisition tool is now also available for the HP8753C.
For the new Agilent E5071B the data can be saved in a floppy
6.8 Loading S-parameter data in ADS and MATLAB

In laboratory 3 you need to compare the S parameters measured for the device you designed and fabricated with those you obtained with your simulation. This appendix describes several way to do it:

Plotting ADS Data in MATLAB (Recommended Approach)

It is usually more convenient to bring the ADS simulation data in MATLAB for plotting them. This gives you more control on the plot. and allow you to to bring both ADS simulated data and measured data in MATLAB for comparison.

For this purpose you need to save your simulation data in a file while being in ADS. To do so refer to the instruction available at:
verb+ http://eewww.eng.ohio-state.edu/ads/ADSsaving.html+

Assume you saved your data in the file myfile.text You can now load your data in MATLAB using:

```
>> load myfile.text
```

Usually the frequencies are stored in

```
>> myfile(:,1)
```

The amplitude of S11 is stored in

```
>> myfile(:,2)
```

To plot simply type:

```
>> plot (myfile(:,1),myfile(:,2))
```

To save into an encapsulated postscript file type:

```
>> print -deps2 myfile.eps
```

Some of you have several plots they would like to combine together in a single plot and postscript file. This is easily done with MATLAB. Four small plots (2x2) would be generated using:
```matlab
>>subplot(2,2,1), plot (myfile(:,1),myfile(:,2))
>>subplot(2,2,2), plot (myfile(:,1),myfile(:,4))
>>subplot(2,2,3), plot (myfile(:,1),myfile(:,6))
>>subplot(2,2,4), plot (myfile(:,1),myfile(:,8))
```

Type `help plot` or `help subplot` to check how to set your titles and axes.

An alternate approach it to plot experimental data in ADS

Move to the data directory of your project directory To plot experimental data in ADS
(for example ADS/ee723_prj)

```bash
cd ADS/ee723_prj/data
```

You are now going to create a file using an editor (for example vi or emacs) to store your S parameter data. Call it for example mydata.s2p: The file should look like that

```plaintext
! Comments line: My data file ...
# GHz  S   MA R 50.0
! SCATTERING PARAMETERS :
 2 0.95 -26 3.57 157 .04 76 .66 -14
 3 0.93 -40 3.53 147 .05 69 .65 -20
 4 0.89 -52 3.23 136 .06 62 .63 -26
```

Note that in the file above:

- Comment lines start with !

- The # line defines the units. In the example shown the Frequency is entered using GHz. S indicates that S parameters are used. MA indicates that they are entered using the amplitude (mag[Sij]) and the angle (ang[Sij]). R 50.0 means that 50 ohms is the reference characteristic impedance.

- The data are introduced in the following order:


Once you have created your file and located it in the data directory you can load it in ADS as data item.
Note that you can save any files in ADS (circuit, layout) into a postscript file by clicking on file and selecting print/plot setup. Select Graphics, File and Postscript Gray Scale and then OK. Now when you select plot in the file menu you will be prompted for the name of the file you want to save it to: e.g., mylayout.ps
6.9 Active Biasing Circuit

(from Ralph S. Carson, High Frequency Amplifiers, 1982)

An active biasing circuit is shown in Fig. 5.16a. Here, pnp transistor $Q_1$, assumed to operate in its active region, helps stabilize the operating point of transistor $Q_2$, hence the name active biasing. The basic operation of the circuit is as follows. If $I_{C2}$ increases, more voltage of the polarity indicated in Fig. 5.16 appears across $R_3$, and this decreases the forward bias for the emitter-base circuit of $Q_1$, so $I_{E1}$ decreases. Since $I_{C1} = I_{B2} = \alpha_1 I_{E1}$, a decrease in $I_{E1}$ causes $I_{B2}$ to decrease. But a decrease in $I_{B2}$ leads to a decrease in $I_{C2}$, so this opposes the increase in $I_{C2}$ assumed originally. Therefore bias stabilization is achieved.

The analysis of the active biasing circuit cannot follow the $T$-equivalent method because of the active device $Q_1$. Instead, straightforward circuit analysis can be carried out after substituting an appropriate circuit model for each transistor, as shown in Fig. 5.16b. Neglecting $I_{co}$, for Case I, we find, from Fig. 5.16b

$$I_{C2} = \beta_2 I_{B2} = \beta_2 \alpha_1 I_{E1}$$

Since $\alpha_1 \approx 1$, this becomes

$$I_{C2} \approx \beta_2 I_{E1} \quad (1)$$

Also,

$$I_{R_3} = I_{E1} + I_{C2} \approx I_{E1} + \beta_2 I_{E1} = I_{E1}(1 + \beta_2)$$

Using KVL around the $V_{CC} - R_1 - R_2$ loop and substituting $(1 - \alpha) = 1/(1 + \beta)$, we obtain

$$I_{R_1}(R_1 + R_2) + I_{E1} \left( \frac{1}{1 + \beta_1} \right) R_1 = V_{CC} \quad (2)$$

and, using KVL around the $R_2 - R_3 - V_{BE1}$ loop yields

$$I_{R_2}(R_2) - I_{E1}(1 + \beta_2)R_3 = V_{BE1} \quad (3)$$

Solve 2 and 3 for $I_{E1}$ and substitute that result into 1 to obtain, for negligible $I_{co}$,

$$I_{C2} \approx (1 + \beta_1) \frac{\beta_2 \left[ V_{CC} \left( \frac{R_3}{R_1 + R_2} \right) - V_{BE1} \right]}{R_4 + (1 + \beta_1)(1 + \beta_2)R_3} \quad (4)$$

where $R_4 = R_1 R_2/(R_1 + R_2)$. If $(1 + \beta_1)(1 + \beta_2)R_3 \gg R_4$, $I_{C2}$ is independent of $\beta_1$ and varies as $\beta_1/(1 + \beta_2)$, and bias stabilization is obtained. It is seen that the collector current
$I_{C2}$ does not depend on $V_{BE2}$ because the current source $\alpha_1 I_{E1}$, in the simplified transistor model is independent of $V_{BE2}$.

Therefore for large $\beta_1$ and $\beta_2$ we have $(1 + \beta_1) \times (1 + \beta_2) \gg R_4/R_3$ and the collector current of the transistor $Q_2$ (microwave transistor) is

$$I_{CC} = \frac{V_{CC} \frac{R_2}{R_1 + R_2}}{R_3} - V_{BE1}$$

Figure 6.1: (a) Overall active biasing circuit and (b) active bias analysis
6.10 Laboratory Report Forms
Step 4) of Part I:
Center frequency: \( f_0 = \)

Bandwidth: \( B = \)

\[ S_{21}(f_0) \text{ dB} = |S_{21}(f_0)|_{dB} = \]

(Include \( S_{21} \) plot in appendix)
\[ |S_{12}(f_0)|_{dB} = \]

\[ |S_{11}(f_0)| \) (SWR) = \]

\[ |S_{22}(f_0)| \) (SWR) = \]

Step 5 of Part I:
Include the swept power gain compression plot in appendix.
\[ |S_{21}(P_{in} = -10 \text{dBm})| \) (in dB) =

\[ P_{in}(-1 \text{dB drop of } |S_{21}|) = \]

Step 6 of Part II:
Include the Noise Figure versus frequency plot in appendix. Noise Figure at 1.8 GHz (with Power on):
\[ F \) (1.8 GHz) = \]

Minimum detectable input signal (See Gonzalez, p. 354):
\[ P_{in,mds} \text{dBm} = \]

Discussion/Comments:
Name: ____________________________
Date: ____________________________

EE 723 Adjunct Laboratory — LABORATORY #2 Report

Experiment #1: Table of $\alpha$ versus $P_{rf,avg}$

Experiment #2: Table of $ACPR$ versus $P_{rf,avg}$

Experiment #3: Table of $ACPR$ versus $PAR$
Name: ____________________________
Date: ____________________________

EE 723 Adjunct Laboratory — PROPOSAL for LABORATORY #3

Project Title:

Design Goals:

Design Procedure:

List of parts:
1 AT42085 transistor
Duroid circuit board
2 SMA connectors
Amplifier characteristics:

center frequency: $f_0 =$

3dB bandwidth: $B =$

$|S_{21}(f_0)|_{dB} =$

(Include S21 plot in appendix)

$|S_{12}(f_0)|_{dB} =$

$|S_{11}(f_0)| (SWR) =$

$|S_{22}(f_0)| (SWR) =$

Optional: Include the swept power gain compression plot in appendix.

$|S_{21}(P_{in} = -10dBm)|$ (in dB) =

$P_{in}$ ( - 1 dB drop of $|S_{21}|$ ) =

Noise Figure at 1.5 GHz (with Power on):

Minimum detectable input signal (See Gonzalez, p. 176):

$P_{i,mds} =$