AN INTRODUCTION TO
THE DESIGN AND TESTING OF MICROWAVE CIRCUITS

LABORATORY MANUAL
for the
EE 710 Adjunct Microwave Laboratory

Patrick Roblin
Associate Professor

The Ohio State University
2001 (14th Edition)
## Contents

### Acknowledgements

#### 1 INTRODUCTION

1.1 Laboratory Overview .................................................. 1
1.2 Laboratory Facilities .................................................. 1
1.3 Laboratory Policies ................................................... 2
1.4 Laboratory Schedule and Report ....................................... 2

#### 2 LABORATORY 1:

2.1 Preparation for Lab 1 .................................................... 4
2.2 Measurement Procedure ............................................... 4
2.3 Write Up ............................................................... 7

#### 3 LABORATORY 2:

3.1 Preparation for Lab 2 .................................................... 9
3.2 Measurement Procedure ............................................... 9

#### 4 LABORATORY 3:

4.1 Selection of the Laboratory Project ................................... 14
4.2 Simulation and Preparation of the Layout ............................ 15
4.3 Fabrication Procedure ................................................. 17
4.4 Measurement Procedure ............................................... 17
4.5 Write Up ............................................................... 17

#### 5 APPENDICES

5.1 How to Use ADS .......................................................... 20
5.2 How to Use the Layout Tool ........................................... 22
5.3 How to Use LineCalc .................................................... 24
5.4 Tips for Soldering ....................................................... 25
5.5 Caring For Connectors .................................................. 27
5.6 Directional Coupler ...................................................... 28
5.7 Loading S-parameter data in ADS and MATLAB ...................... 30
5.8 Least Square and Plotting with MATLAB ............................ 33
5.9 Making a Poster with Latex ............................................ 34
5.10 Laboratory Report Sheets .............................................. 36
Acknowledgements

The development of the ee710 adjunct laboratory would not have been possible without the help and support of many people whose contributions we would like to acknowledge.
The Network Analyzer HP8753C was acquired thanks to a National Science Foundation educational equipment grant.
Several parts and supplies have been generously donated by the microwave industry. We would like to thank the Rogers Corporation for the continuous support of this laboratory with the donation of microstrip circuit boards and Western Microwave for the donation of a circulator.
We are grateful to Prof. Walter D. Burnside for providing us with demonstration microwave devices.
Demonstration microwave circuits were developed for this laboratory by Patrick Kaiser, Lin Chia-Liang, Truong X. Nguyen, and Ben Johnson during the course of independent study projects.
I am most grateful to my graduate students Sung Choon Kang, Chih Ju Hung, Vakur Erturk and Siraj Akhtar for their dedicated contribution to the development of this laboratory.
I am grateful to Prof. Johnson for recently teaching that course and providing corrections for this manual and offering suggestions for further lab developments.
Finally I would like to thank Professors H.C. Ko, B. Mayan, D. Hodge and Y. Zheng, for the support provided to this laboratory as chairman of the EE department.
1. INTRODUCTION

1.1 Laboratory Overview

The EE 710 adjunct microwave laboratory is designed to run in parallel with EE 710. An introduction to microwave circuit design is given in the EE 710 lecture. In support to the design theory, the simulation and optimization of the microwave circuits studied are performed using an advanced microwave circuit simulator (ADS) installed in the ER4 computer system. To complete the learning cycle the EE 710 adjunct microwave laboratory provides EE 710 students with the opportunity to experimentally verify the device principles and design theory of the microwave devices and circuits studied. The three main objectives of the laboratory are therefore:

1. To introduce the EE 710 student to the techniques and principles of microwave measurement with a modern Network Analyzer.

2. To provide the EE 710 student with the opportunity to test some of the microwave devices discussed in EE 710.

3. To involve the EE 710 student in a design project of the student choice. The project include first principle design, CAD simulation and optimization, CAD layout, fabrication and testing of a prototype.

Note that 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 (see Section 5). Successful design projects will be used as demos in the microwave laboratory.

1.2 Laboratory Facilities

The laboratory facilities located in CL305 provide three functions:
- The measurement stations
- The Quick Circuit fabrication
- The soldering stations

The measurement station used in EE710 consists of one HP 8753C Network Analyzer and the Colorpro HP Plotter.

The design bench includes a workbench, a tool box, a drill, a soldering iron, and a parts cabinet. 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 using computer aided machining by the EE 710 teaching assistant in CL 305.

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 710 Laboratory Policy**

1. Schedule your lab measurement in (CL 305) time in advance on the sign up sheet. You can reserve up to 2 hours/ per week. More time is however allowed 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 710 class roster; remind him/her 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 5.5 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.

6. Read soldering tips in Appendix 5.4 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 EE710 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 5.10,
• 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 on a word processor (latex is strongly recommended). Note that you can store your simulation plots in a postscript file and include them in your report without cutting and pasting. Your report will be archived so that it can be used as a reference by future EE710 students and your poster will displayed in the display case in the Dreese-Caldwell bridge.
2. LABORATORY 1:

Microwave Measurement with the Network Analyzer

Laboratory Goal: The purpose of this first laboratory is to get familiarized with the Network Analyzer and the measurement of one-port devices. In particular the student will perform both frequency domain measurement and time domain analysis to extract the effective lengths of a microstrip open stub and SMA launcher and evaluate the dielectric constant of the microstrip substrate.

Approximate Duration: 2-3 hours

2.1 Preparation for Lab 1

1. Preliminary Reading:
   Read in the User’s guide of the HP 8753C Network Analyzer the following chapters:
   - Chapter 1: Operating the HP 8753C
   - Chapter 3: Reflection Measurements with the HP 8753C
   - Chapter 4: Time Domain Analysis

   A Network Analyzer User (NAU) guide is available in the CL305 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 5.5 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 Measurement Procedure

1. Clean the SMA connectors of the cables, and standards:
   The relatively inexpensive SMA connectors are not really 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 5.5.

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

3. **Perform a 1-Port Calibration of the Network Analyzer**:
   Press MEAS and select S11 (p. 9 of user’s guide).
   You might find it convenient to select the SMITH CHART format (NAU guide p.10) before calibrating. Simply press FORMAT and select SMITH CHART.
   To start the calibration press CAL and select the 1-port Calibration (NAU guide p.11). Use the calibration short, open and 50 Ω load in the SMA calibration kit (available in the small wooden box) for the reflection calibration of port 1. A sketch of the standards is given below.
   **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 and 50 Ω LOAD in the calibration procedure.
   Press DONE when done. Save your calibration in a register (NAU guide p.31).
   Verify the calibration by testing the 50 Ω load, the open and short connections. Note that this calibration is only valid for the particular test cable you are using at port 1 and would no longer be valid if the cable was removed (*Please do not remove/disc connect the test coaxial cables!*). You are now ready to perform a one-port measurement.

   ![Figure 2.1: Calibration Standards](image)

4. **Evaluation of the effective length of an open stub at \(\lambda/4\)**:
   Identify the test board consisting of three 50 Ω microstrip open stubs (3, 6 and 9 cm) available on the rack. Connect the shortest open stub (3 cm) to port 1. Use the Smith Chart format to display S11. Using the marker (press MKR) (NAU guide p.17)
determine at which frequency the open termination of the stub is transformed into a short at the level of port 1. At this frequency the stub is a quarter wave transformer. Plot this Smith Chart (NAU guide p.17) using the plotter. You need will this information in your write up to estimate the length of the stub + connector system. You will assume that the VELOCITY FACTOR \( \left( \frac{1}{\sqrt{c_f(\text{eff})}} \right) \) is 0.7196 for both the connector and the microstrip board used.

**Note:** To make a hardcopy output on the plotter follow the instruction given in the User’s guide manual p. 17. The network analyzer should only become the system controller for the time necessary to plot (Press LOCAL 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. **Evaluation of the effective length of an open stub with a reference plane shift:**

The reference plane for S11 was defined during calibration by the short at the end of the test cable on the dielectric plane. It is possible to electronically move it (NAU guide p.24). Press SCALE REF and select ELECTRICAL DELAY. Vary the knob to move the reference plane up to the level of the open and measure the electrical length. Reset the ELECTRICAL DELAY to zero before continuing. You will use this datum in your report to estimate the effective length of the stub + connector system. Note that you must account for a factor 2 associated with the round trip of the wave. You will assume that the VELOCITY FACTOR \( \left( \frac{1}{\sqrt{c_f(\text{eff})}} \right) \) is approximately 0.7196 for both the connector and the microstrip stub considered.

6. **Evaluation of the effective length of an open stub with time domain:**

Following the procedure described on page 37 of the Network Analyser User guide, generate the impulse response of the microstrip open stub circuit. With TRANSFORM OFF verify that the START and STOP frequencies are 300 kHz and 3 GHz respectively. With TRANSFORM ON select the LOW PASS IMPULSE and a START and STOP time of -10 and 10 ns (or smaller) respectively. Use LOG MAG format.

- Locate the position of the top of the impulse for an open circuit (stub disconnected).
- Measure the time delays associated with all the 3 open stubs (3, 6 and 9 cm). The delay is measured at the motion of the top of impulse relative to the impulse obtained for the open (stub disconnected). Note that you must account for a factor 2 associated with the round trip of the wave. You will use these data in your report to evaluate the effective length of the shortest stub (about 3 cm), the
length of the connector and extract the dielectric constant of the microstrip board. You will assume in your calculation that the VELOCITY FACTOR \(1/\sqrt{\varepsilon_r^{\text{eff}}}(\text{eff})\) is approximately 0.7196 for the microstrip stub considered.

- Record the 3 dB width of the pulse to estimate the smallest distance between two successive reflections which can be resolved.
- Identity the presence of a secondary small peak before the principal impulse peak for the 9 cm stub. Explain its origin.

7. Radiating open stub
A stub of length \(\ell\) terminated by an open at both end is a resonator whose first resonant frequency occurs at half a wavelength \(\ell = \lambda/2\). In wide open stubs the radiation loss becomes important and these stubs can be used as an effective radiating element called a patch antenna. Connect the 2.xx GHz patch antenna. Turn TRANSFORM off and set CAL correction ON (the calibration correction is somehow turned off by TRANSFORM). Display \(S_{11}\) on the Smith chart. Notice that at its resonant frequency of 2.xx GHz the patch antenna is matched. Add an electrical delay of about 105 ps to compensate for the connector delay by moving the reference plane next to the antenna. Plot on the plotter the resulting \(S_{11}\) parameter on the Smith Chart. Determine whether the resonance is that of parallel or series RLC resonant circuit.

2.3 Write Up

- Complete the lab report sheet given in Appendix 5.10.
- Document steps 4, 5, and 7. Include the plots generated.
- Compare the effective length calculated for the short stub (3 cm) using each method. List the source of errors in each measurement. Estimate the different physical contributions to this length. The different physical contributions are

  - the mechanical length of the connector starting from the coaxial dielectric plane to the microstrip board.
  - the mechanical length of the microstrip stub.
  - the length contributed by the fringing field (see Pozar p. 254).

**Note:** The added electrical length \(\Delta \ell\) beyond the mechanical length of an open microstrip line is given by (see Davis p. 128)

\[
\frac{\Delta \ell}{h} = 0.412 \left[ \frac{\varepsilon_{\text{eff}} + 0.3}{\varepsilon_{\text{eff}} - 0.258} \right] \left[ \frac{w}{h} + 0.262 \right] \left[ \frac{w}{h} + 0.813 \right]
\]
where $\epsilon_{\text{eff}} = 1.931$ is the effective relative dielectric constant, $w = 75.577$ mil is the microstrip line width, and $h = 30$ mil is the thickness of the dielectric substrate.

- Plot (using MATLAB, see Appendix 5.8) the time delays measured of the 3 stubs versus the physical length $\ell$ of the stubs (3, 6 and 9 cm). Generate a least square fit (see Appendix 5.8) of the three data point using a straight line $t = a\ell + b$ and extrapolate the delay $b$ contributed by the SMA launcher for a stub of length $\ell = 0$ cm.

- Calculate the length of the SMA launcher using a velocity factor of 0.73.

- Calculate from the slope of your least square fit the estimated effective dielectric for the microstrip lines.

- Identify the presence of a secondary small peak before the principal impulse peak for the 9 cm stub. Explain its origin.

- Discuss any difficulties you experienced during the lab.
3. LABORATORY 2:

Microwave Measurement of 2, 3 and 4 Port Devices

Laboratory Goal: The purpose of this laboratory is to get familiarized with the Network Analyzer and the measurement of 2, 3 and 4 port devices. The student will test various devices including a filter, a circulator, a directional coupler and a dielectric resonator filter.

Approximate Duration: 2-3 hours

3.1 Preparation for Lab 2

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

2. Additional Reading Material:
   This laboratory exposes you to a number of microwave devices before they have been covered in the lecture. You will therefore benefit from reading more about them in our textbook (Pozar, Microwave Engineering) before measuring them. Circulators are covered in Section 9.6. Couplers are covered in Section 7.1 and in Appendix 5.6 of this manual. Dielectric resonators are covered in Section 6.5. Reading the introduction (first two pages) of these sections will suffice for our present purpose.

3.2 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. Clean the cables and standards (50 Ω and short available in the small wooden box and sketched below) using a cotton swab, alcohol and compressed air. See connector care in Appendix 5.5.

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 (NAU guide p.10) for all the scattering parameters before calibrating.
Press CAL and select the 2-port Calibration (NAU guide p.13).

Use the calibration short, open and 50 Ω load in the SMA calibration kit (available in the small wooden box) for the reflection calibration of ports 1 and 2. Use the thru for the four transmission calibrations. A sketch of the standards is given below.

**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, THRU and so on in the calibration procedure.**

```
50 Ω    OPEN

SHORT   THRU
```

![Figure 3.1: Calibration Standards](image)

For the isolation calibration select Omit Isolation.

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

Verify the calibration by testing the 50 Ω load, the open and short and thru connections. Note that this calibration is only valid for the particular test cables you are using. You are now ready to perform a two-port measurement.

Before continuing, a few comments 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 critical at 3 GHz* you can use 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 $S_{11}$ calibration and for the subsequent S-parameter measurements.

4. **Testing of Low Pass Filter:**
   Connect one of the Filters available (black cylindrical body with 2 SMA connectors). Measure $|S_{21}|$ in dB. Find the 3 dB corner frequency using the marker. Make a hardcopy output on the plotter.

5. **Testing of a Circulator:**
   Connect the Western Microwave Circulator (blue body device with 3 SMA female
connectors). Since this is a three-port device you need to use a matched load at the port which is not connected to the Network Analyzer. This circulator is intended to operate around 2 GHz. Measure the amplitude and phase of the circulator scattering parameters at this frequency. Compare the 3 by 3 scattering parameter matrix obtained with that of the ideal circulator.

6. Testing of a Directional Coupler:
Connect the Directional Coupler (grey body with 4 SMA female connectors). Since this is a four-port device you need to use two matched loads at the ports which are not connected to the network analyzer. Use the two 50 Ω loads which look similar to the calibration 50 Ω load but have a silver body. The center frequency of operation is 1.2 GHz. Measure the fractional bandwidth \( \frac{f_{3db}}{f_{center}} \). Show the directional coupler symbol with the proper labeling of the port. Is it an Hybrid coupler? What is the insertion loss, coupled power, directivity and isolation of the Directional coupler at the center frequency (see Appendix 5.6 for their definition).

7. Dielectric Resonator Filter Experiment:
A dielectric resonator is realized by a piece of low loss material with a high dielectric constant which ensures that the EM fields are mostly confined in the dielectric material. In our experiment a cylindrical dielectric resonator made in ceramic with a resonant frequency around 2.4 GHz is used. It is glued to a piece of duroid to slide it on the microstrip board. Please do not touch the ceramic with your fingers as any deposits on the resonator will increase its loss. The frequency of the resonator can be tuned mechanically by moving a piece of metal (we use a microstrip board) on top of it which also shield it from unwanted coupling. In our experiment the dielectric resonator is coupled via its electric field to a microstrip line connecting port 1 to port 2 (2-port thru). The resulting circuit realizes a high-Q band-stop filter at the resonant frequency. Indeed at resonance the dielectric resonator behaves like a nearly ideal shunt LC resonator placed in series between the microstrip line and its ground plane. Therefore at resonance the dielectric resonator introduces an open circuit located at the position of the dielectric resonator.

- Decrease the frequency ranged used for the frequency scanning by selecting MENU.
Press START to select a start frequency of 2.3 GHz. Press STOP to select a stop frequency of 2.6 GHz. Then RECALIBRATE the network analyzer using a 2-port calibration!
• Connect the dielectric resonator circuit between port 1 and port 2 and locate the dielectric resonator near the microstrip line. Then display on the screen of the Network Analyzer both $S_{11}$ using the Smith Chart and $|S_{21}|$ in dB using a log scale. To see both channel press DISPLAY and select DUAL CHANNEL ON. Two displays appear on the CRT. You can switch from one channel to other using the ACTIVE CHANNEL CH1 and CH2 keys.

• Locate the resonance frequency $f_r$ at which $S_{11}$ is tangent to the unit circle on the Smith Chart ($|S_{11}(f_r)| = 1$ ideally, 0.8 is really observed) and port 1 and 2 are disconnected ($S_{21} = 0$ ideally). Move the resonator laterally and evaluate at which approximate distance from port 1 (in terms of wavelength) $S_{11}$ is a short $S_{11}(f_r) = -1$.

• The bandwidth $\Delta f$ can be adjusted by varying the distance between the dielectric resonator and the microstrip line. Observe the variation of the resonant frequency as the top conductive plane (microstrip board) is slightly lowered by gently pressing on it. Give a simple justification using a parallel plate cavity model.

• Determine the resonant frequency $f_r$, the insertion loss at resonance ($-|S_{21}(f_r)|$dB), the +3dB bandwidth $\Delta f$ (measured from the magnitude of $S_{21}$ at the resonant frequency: $|S_{21}(f_r)|$) and the loaded Q ($Q_L$) of the resonator defined as the inverse of the fractional bandwidth:

$$Q_L = \frac{f_r}{\Delta f}$$

These measurements can be realized using the network analyzer. First display both the magnitude of $S_{21}$ and $S_{11}$ using a log scale. Supercpose both of these graphs on a same graticule by pressing the DISPLAY menu key and selecting MORE and SPLIT DISP OFF. Switch to the channel (CH1 or CH2) corresponding to $S_{21}$. To obtain $f_r$, $\Delta f$ and $Q_L$ follow the procedure:

- Clear all previous markers: MKR, all off
- Find the resonant frequency: MKR FCTN, MKR SEARCH, MIN
- Define the center frequency for the 3dB bandwidth: MKR, MKR ZERO
- Enter the search menu: MKR FCTN, MKR SEARCH
- Change the default WIDTH VALUE from -3 dB (low pass/ band pass) to 3 dB (band-stop and high-pass): WIDTH VALUE, $3 \times 1$
- Calculate the bandwidth: WIDTH ON

The $Q$ of the resonator increases as the dielectric resonator is moved away from the microstrip line. Tune the distance between the dielectric resonator and the
microstrip line and repeat the bandwidth measurement steps till you achieve a
loaded $Q$ between 3000 and 6000. Once you achieve it plot $|S_{21}|$ and $S_{11}$. This
provides you with a record $f_r$, $\Delta f$ and $Q_L$.

- Around its resonant frequency a high-$Q$ dielectric resonator can be approximated
  by an RLC shunt resonator and therefore characterised by three quantities: the
  resonant frequency, the load $Q_L$ and the loaded $Q_U$. Measure $S_{21}$ in dB and
calculate the unloaded $Q$ ($Q_U$) of the resonator using the formula connecting it
to the insertion loss at resonance:

$$L_I(\omega_r) = 20 \log \left( \frac{1}{|S_{21}|} \right) = 20 \log \frac{Q_U}{Q_L}$$

3.3 Write Up Write a short report which documents steps 4, 5, 6, and 7. Include
all your plots and calculations. Complete the laboratory report sheet (see Appendix
5.10).
4. LABORATORY 3:

Design of a Distributed Microwave Circuit

Laboratory Goal: The goal of this laboratory is to design, fabricate and test a distributed microwave circuit.

Approximate Duration: 6 hours

4.1 Selection of the Laboratory Project.

A list of possible projects is given below:

- Broadband transformer: (Check with TA on the availability of chip resistors
  Butterworth (Rizzi p.146), (Pozar p.278); Tchebyscheff (Rizzi p.150), (Pozar p.282);
  Tapered (Rizzi p.152), (Pozar p.288)

- Rat race coupler (Rizzi p.363), (Pozar p.401)

- Branch line coupler (Rizzi p.378), (Pozar p.379)

- Power divider (Rizzi p.367 and Wilkinson paper (see instructor)), (Pozar p.363)

- Coplanar microstrip coupler (Rizzi p.382), (Pozar p.383)

- Narrowband bandstop filter (Rizzi p.454 and p.460)

- Narrowband bandpass filter (Rizzi p.462), (Pozar p.490)

- Broadband bandstop filter (Rizzi p.487), (Pozar p.466)

A proposal (see Appendix 5.10) describing the proposed project should be returned by the due date set by your instructor. This proposal should describe the design goals, outline the design procedure, list the parts required and list the measurements to be performed. See the next page for a prototype laboratory proposal.
SAMPLE
PROPOSAL FOR LABORATORY 3

Project Title: Design of a Broadband Transformer
Design Goals:

1. Design of a two-section Chebyshev transformer to match a 100Ω chip resistance to a 50Ω transmission line with a maximum VSWR of 1.2 at 2 GHz.

2. The microstrip design will be optimized with ADS.

3. The microstrip layout will be generated with the layout tool.

4. The substrate used is Duroid.

Design Procedure:

1. The design will start from the first-cut design of this transformer done for transmission lines in homework problem #2.

2. Linecalc (see Appendix 5.3) will be used to calculate the dimensions of the microstrip lines for each section of the transformer. The ADS circuit file of the transformer will be converted from transmission line to microstrips. MSTEP and MOPEN models will be used to account for the discontinuities. The RES element footprint will be realized with the MGAP element.

3. The layout will be generated with ADS.

Measurement Results and Analysis:

$S_{11}$ versus frequency will be measured and compared with the simulated results.

List of parts:

- Size of Duroid circuit board: 2 × 4 inches
- # of SMA launchers: 1
- 1 Chip resistor (100 ohm)

4.2 Simulation and Preparation of the Layout

Your circuit will be fabricated using the Quick Circuit machine. Your design should therefore account for the physical constraints introduced by the layout and the fabrication. Read Appendix 5.2 for more information on the layout procedure. In particular the layout tool introduces the following constraints
• 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 footprint of chip resistors and chip capacitors.

**Design rules**
A few design rules should be respected to permit the successful 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.

• The microstrip lines should be slightly tapered near a SMA connector to avoid shorting of the center conductor by the ground of the SMA coaxial connector.

• MINIMIZE the area (footprint) used by your circuit. The fabrication time and therefore the tool-wear in the Quick Circuit machine is proportional to the amount of copper removed! Also the Duroid substrate used is expensive. Note however that the edge of your circuit need to kept $3 \times H$ away from the edge of the microstrip board.

• Add a ground plane around your circuit. This will improve the grounding of your circuit as well as speed up the fabrication time and minimize the wear of the Quick Circuit machine. Note that you need to allow for at least $3^H$ spacing ($H$ is the board thickness) between your circuit and the ground plane except obviously when connecting to ground sections. A few via holes might be desirable too. See detailed instruction at:

  http://www.eng.ohio-state.edu/~roblin/images/lab/adshelp.html

**Connector less design**
We encourage the fabrication of connector-less circuits for 2-port design projects. These circuits are tested using a special testbed which provides the 2 SMA connectors. This saves about $10 per circuit. The 2-port circuit (bounding box) must be 3494 mils long. The circuit width is of your choice but we recommend keeping it to 2 to 3 inch wide (2000-3000 mils). Note that any grounding pins used should be connected such that the ground plane remains as flat as possible in the testbed. A tapered microstrip line is recommended next to the SMA.
connectors. Cleaning the testbed is recommended before testing to insure a good grounding. Contact the TA to inspect the testbed ahead of time. 

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.

4.3 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 circuit bound determined. The file is converted to Gerber format. Then 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.

1. Fix the connectors on the circuit board.
2. Solder (see Appendix 5.4) all connection points as needed.

4.4 Measurement Procedure

1. Calibrate the Network Analyzer or retrieve the calibration from the HP disk drive.
2. Measure and plot the amplitude of the reflection coefficients $|S_{ij}|$.
3. Record the data manually for 20 frequency points or so.

4.5 Write Up

The final report should include the laboratory proposal in the first page. The design and fabrication steps should be documented so that another student could reproduce your work. The measured and simulated circuit performance should be compared (plotted together on a same graph) and an analysis of the results obtained should be included in the report.
A poster page summarizing your design and experimental results should be included with the report. This poster page should include:

- Project Title
- Your names
- EE710, 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/. See Appendix 5.9 for instructions for making posters with latex.
5. INDEPENDENT STUDIES

Students interested in continuing the laboratory can do so under the form of an independent study. The following format is proposed. The student should first discuss a possible project with the instructor. The student should then write a one or two page independent study proposal which will include a discussion of the goal pursued, a plan of work, an evaluation of the cost of the parts required if any. The EE department can cover the design cost if the latter is reasonable. The prototype circuit developed will be left to the EE department. The project should not last more than a quarter. Successful projects will be included in the Open Laboratory section of this Laboratory Manual for demonstration purposes. Proper credits to their designer will be given in the Laboratory Manual.

A list of possible projects looking for an independent study is given below:

1-Port: Various RLC lump circuits
   Microwave Rectifier Circuit
   2 GHz Bipolar Transistor Oscillator

2-Port: Pin Diode Switch Circuit
   Microstrip Filters and Transformers
   Microwave Transistor Test Set and Deembedding with Touchstone
   Lange Coupler Fabrication
   S-Parameter Capture from IBM-PC using HPIB network.

3-Port: Microstrip Wilkinson Power Divider
   Microstrip Circulator

4-Port: Microstrip Coupler using Coupled Line
   Microstrip Branch Line Coupler
   Microstrip Rat Race Coupler
5. APPENDICES

5.1 How to Use ADS

??
To start ADS use the following commands:

source /apps/hpeesof/SETUP
hpads

Alternately to avoid sourcing SETUP you could enter in your .cshrc file:

setenv HPEESOF_DIR /apps/hpeesof
setenv LM_LICENSE_FILE $HPEESOF_DIR/licenses/license.dat
set path = (. $HPEESOF_DIR/bin $path)

TUTORIAL:
If you are new to ADS you should go through the tutorial. For this purpose click on Help in the Advanced Design System (Main) window. Select Topics & Index. A new window labeled manuals.fm with hypertext mark-up giving you access to the full ADS documentation appears. Select (click on) Getting Started with Circuit Simulation to open the on-line tutorial. You will need to go through Program Basics (Chapter 2) and Introduction to RF Board Simulation (Chapter 5). Additional information is also available in User’s guide and Data Display of the original help window manuals.fm.

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 bf /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 Jan 5 pts/0 hpeesofviewer /apps/hpeesof/bin/hpeesofviewer

To kill that job simply type:
    kill -9 14776
5.2 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 footprint. It does not have a footprint 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 footprint. 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 occurring 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. Holes are also considered as a layer. There are automatically introduced by the via holes: VIA2 (see below).

Ask you instructor for more information on the substrate you will be using. The substrate used for your class will specified in the class webpage. 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 $W_1$ 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 $W_1$ 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).
5.3 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, \( H_u \) 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: \( (4.878 \times 10^7 \omega^{-1}m^{-1}) \), \( TanD \) 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.
5.4 Tips for Soldering

Tips for soldering:

1. Moisten the sponge of the soldering stand.
2. Turn on the soldering iron.
3. Set the temperature around 700oF.
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.

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 activate it for the time you are soldering ONLY.
The SAFETY PRECAUTIONS are:

1. Keep you 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.
5.5 Caring For Connectors

This Appendix gives some important hints on the proper way to connect devices and maintain them. 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 (and not the device or cable)
   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) Overtighten preliminary connection
   f) Twist or screw device or cables in connectors
   g) Tighten past ”break” point of torque wrench
5.6 Directional Coupler

An ideal directional coupler is a loss-less reciprocal \((S_{ij} = S_{ji})\) four-port device (see Figure 5.1) which is matched at all ports \(S_{ii} = 0\), couples the incident power on port 1 to port 3 and 4 while providing isolation between port 1 and 2 \((S_{12} = 0)\) and between port 3 and 4 \((S_{34} = 0)\).

![Diagram of a directional coupler](image)

Figure 5.1: Directional coupler symbol.

An ideal coupler has a S-parameter matrix of the following form:

\[
[S] = \begin{bmatrix}
0 & 0 & S_{13} & S_{14} \\
0 & 0 & S_{23} & S_{24} \\
S_{13} & S_{23} & 0 & 0 \\
S_{14} & S_{24} & 0 & 0 \\
\end{bmatrix}
\]

where the S-parameters verify the following properties:

\[
|S_{13}|^2 + |S_{14}|^2 = 1 \\
|S_{13}| = |S_{24}| \\
|S_{14}| = |S_{21}|
\]

(1)

The coupling coefficient is defined in terms of the incident wave \(a_1\) at port 1 and reflected wave \(b_4\) at port 4 as:

\[
C = -10 \log \left( \frac{|b_4|^2}{|a_1|^2} \right) = -10 \log \left( |S_{41}|^2 \right)
\]

The isolation coefficient is defined as

\[
I = -10 \log \left( \frac{|b_2|^2}{|a_1|^2} \right) = -10 \log \left( |S_{21}|^2 \right)
\]
The directivity coefficient is defined as

$$ D = -10 \log \frac{b_2}{b_3} = -10 \log \left( \frac{|S_{21}|^2}{|S_{31}|^2} \right) $$

Note that the directivity cannot be measured directly but it is expressed in terms of the coupling and isolation coefficient by the relation

$$ I = C + D. $$

For low coupling the directivity is an important measure of the coupler’s quality as we must have $I \ll C$. For example a coupler with a coupling of $C = 20$ dB, and a directivity of $D = 20$ dB would be an excellent coupler as this would provide an isolation of $I = 40$ dB.
5.7 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. Here is how to do it:

Move to the data directory of your project directory (for example hpeesof/ee723.prj)

```
cd
   cd hpeesof/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 by doing the following:

While in the schematics window, click on the library menu and select the menu item: Linear Data File Elements. In the right side of the dialogue box select then the item: S2P(2-Port S-parameter File)

In the new dialogue box which then appears, click on Value Options. The file mydata you have created before and which is located in the data directory should appear. Select it and click on OK. Then click on Apply and the file name is loaded. Click on OK to introduce
this new device on your schematic. Then as usual you need to add two ports and a ground terminal and save your schematic under a name before plotting the S-parameters in a test window.

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

Plotting ADS Data in MATLAB (Recommended Approach)
It might be more convenient to do the reverse: that is bring the ADS simulation data in MATLAB. This gives you much more control. Bringing simulated data in ADS for comparison with measured data can give you more control on the plot.

For this purpose you need to save your simulation data in a file while being in ADS. To do so go in your test window in ADS. Click on Library, scroll down the menu and then click on Output Files. Then select SNP (S-parameter File). In the value field enter the name of the file for example myfile. ADS will automatically add an extension such as .s2p for a 2 port device (myfile.s2p), .s4p for a 4 port device (myfile.s4p).

Locate the OFILE item in your test window and simulate. It would be convenient to select a reasonable number of frequency points such as 20 points. The file will be created in the data subdirectory of your project directory.

Before loading this file on MATLAB you need to clean it. Use an editor and remove the text headers:

! Communications Design Suite 5.0 306 Aug 26 1994 (c) 1993 Hewlett-Packard ! Mon Nov 27 11:34:56 1995 lange_tb\Lange

# MHz S MA R 50.0000

! SCATTERING PARAMETERS :

Note that the header specifies the data type: S stands for scattering parameter, MA indicates that the scattering parameters are represented by their magnitude and angle. 50 gives the characteristic impedance.

For each frequency the data is presented in the following format for a 4 port:

freq  magS11  angS11  magS12  angS12  magS13  angS13  
              magS21  angS21  magS22  angS22  magS23  angS23
              magS31  angS31  magS32  angS32  magS33  angS33
              magS41  angS41  magS42  angS42  magS43  angS43

For each frequency (freq) consolidate with your editor the three lines into a single one
To perform these tasks automatically you can use the script file `s#p2text`:

```
s#p2text <myfile.s4p >myfile.text
```

The files `myfile.s4p` and `myfile.text` and `s#p2text` in the ER4 directory `/tmp_mnt/user2/faculty/roblin/latex` for you to inspect them.

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

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

Now 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 in a postscript file. This is easily done with MATLAB. Four small plots (2x2) would be generated using:

```
>> 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.
5.8 Least Square and Plotting with MATLAB

A least square solution of an overdetermined system \( t = a\ell + b \) can easily be computed using MATLAB. Consider the system

\[
T = MV
\]

where the matrices \( T \), \( M \) and \( L \) are defined by

\[
T = \begin{bmatrix}
t_1 \\
t_2 \\
t_3
\end{bmatrix}, \quad M = \begin{bmatrix}
\ell_1 & 1 \\
\ell_2 & 1 \\
\ell_3 & 1
\end{bmatrix}, \quad V = \begin{bmatrix}
a \\
b
\end{bmatrix}
\]

It results that

\[
V = M^{-1}T
\]

The least square inverse of the non-square matrix \( M \) can be calculated using MATLAB. MATLAB is available on the ER4 network. To invoke it simply type: `matlab` Once in MATLAB you can create the matrix \( M \) as follow

\[
M = \begin{bmatrix}
3 & 1 & 6 & 1 \\
1 & 6 & 1 & 9 \\
1 & 9 & 1
\end{bmatrix}
\]

and the vector \( T \) (for example)

\[
T = \begin{bmatrix}
102 \\
202 \\
302
\end{bmatrix}
\]

The vector \( V \) is then simply given by

\[
M \backslash T
\]

To plot a vector \( y \) versus \( x \) simply type:

`plot(x,y)`

Use the command

`help plot`

to find more about plotting (check also `hold`, `axis`, `xlabel`, `ylabel`).
5.9 Making a Poster with Latex

The latex files for making the poster are in the directory:

`~roblin/latex`

The example files are:

- `poster.tex`
- `setfigfont.tex`
- `langelayout90.ps`
- `langecircuit90.ps`
- `plot4fig.eps`

Note that the first postscript file (here `langelayout90.ps`) would be the layout of your circuit. The second would be (here `langecircuit90.ps`) your ADS circuit. Finally the third file (here `plot4fig.eps`) would be a comparison between measured and simulated results. Refer to Appendix 5.7 of your Lab manual to find out how to make such an `xxx.eps` file using MATLAB.

To compile your latex file type:

`latex poster.tex`

A device independent file `poster.dvi` is created. Then to generate a postscript version use:

`dvips poster.dvi -o file.ps`

To preview it type:

`ghostview file.ps &`

To print it type:

`lp file.ps`

Modify the title, caption and insert your own figure for both of the `xxx.ps` or `xxx.eps` files. If your caption is big you might have to reduce the size of the figures currently (1.7 by 1.7). ADS generates postscript files instead of encapsulated postscript. To use the postscript files you must add a line like:

`%%BoundingBox: 37 70 558 757`
Note that 'showpage' at the end of the file should also normally be removed from the ps file. The postscript figures generated by ADS for the layout and circuit will appear rotated. To correct for this problem you need to edit the layout postscript file and locate the statements:

```
90 rotate
10 mm -210 mm translate
```

and replace them by:

```
%90 rotate
%10 mm -210 mm translate
-10 mm 20 mm translate
```

This will remove the 90 rotation and translate the figure.

In your poster.tex file you will have to experiment with the translation parameters and the graphics size: \texttt{\textbackslash epsfxsize 3.5in \textbackslash epsfysize 3.5in}. The horizontal spacing between graphs: \texttt{\textbackslash hspace{-1 in}} the vertical spacing: \texttt{\textbackslash vspace{-1 in}} can both be negative. Finally when you print the linewidth might be too small. You can edit the postscript file and change the linewidth For example replace

```
0 mm setlinewidth
```

by

```
2 mm setlinewidth
```
5.10 Laboratory Report Sheets
Step 4): Quarter-Wave Transformer

\[ f_{\lambda/4} \text{ (GHz)} = \]
Estimated effective length of the 3 cm stub (cm) =

Step 5): Reference Plane Shift

Delay measured for the 3 cm stub (ps) =
Estimated effective length of the 3 cm stub (cm) =

Step 6): Time Domain

Impulse response delay measured for 3 cm stub (ps) =
Estimated effective length of the 3 cm stub (cm) =
Resolution (3dB width) of the impulse response (ps) =
Impulse response delay measured for 6 cm stub (ps) =
Impulse response delay measured for 9 cm stub (ps) =
Estimated delay of the connector (ps) =
Estimated effective relative dielectric constant =
Estimated length of the connector (cm) =
Effective length added by the fringe capacitance (cm) =

Step 7): Patch Antenna

Type or resonance: series or parallel RLC?
Step 4): Low-Pass Filter

\[ f_{3dB} = \]

Step 5): Circulator

Scattering matrix [ \( S \) ] =

Comments:

Step 6): Directional Coupler

\[ \Delta f_{3dB}/f_{center} = \]

Show the directional coupler symbol with the proper labeling of the port:

Is it an hybrid coupler?

Insertion Loss =

Coupled Power =

Directivity =

Isolation =

Step 7): Dielectric Resonator

\[ f_r = \quad \Delta f = \quad L_I = \]

\[ Q_L = f_r/\Delta f = \quad Q_U = \]

Shunt or Series resonator?
EE 723 Adjunct Laboratory — PROPOSAL for LABORATORY #3

Project Title:

Design Goals:

Design Procedure:

List of parts:
Size of Duroid circuit board:
# of SMA connectors