Lab 3: Couplers and Power Dividers

By: Cynthia Furse, Ray Woodward
ECE 6130 Wireless LAN Lab
Utah State University
USA

Objective:

- Design, optimize, build, and test a 3 dB power divider

Equipment:

- Agilent 8510C Microwave Vector Network Analyzer
  (Replacement model: Agilent E8362B PNA Network Analyzer)
- TRL Calibration Kit

Procedure:

1. Review the text or your portfolio and find the desired parameters for a 3-dB coupler using microstrip line. The coupler should divide the power evenly (3 dB) between the two output branches. It will operate at a frequency of 2.5 GHz. Each student in the class should do his or her own design for this lab and test it in Libra. Your group will then decide which design to mill and test.

2. Use ports, lines, tees and bends as desired in Libra to implement your design.

   A. Design your 3dB power splitter as described in the text. Use MLIN and MTEE to produce it in the schematic.

   B. Design quarter-wave matching sections for the output ports:

   Use Linecalc to find the lengths and width for the quarter-wave transformers of the output sections (which are no longer 50 ohms lines). Also make sure that the widths of the tee correspond to the widths of the line. These widths can be verified using the Design Synchronization and looking at the layout to see if the widths look correct.

   C. Add lengths of line to the input and output ports to allow accurate calibration:

   There are some important specifications that you will need to conform to so that you will be able to get accurate results when testing the circuit. Since the TRL calibration of the network analyzer is calibrated to 1.9304 cm (760 mils= 1/2 of the length of the THRU), you will want to append at least that length of line (with Zo = 50 ohms at 2.5 GHz) to the input and the outputs of the circuit. The width necessary for the 50 ohm impedance can be found by using Linecalc.

   D. Adjust the distance between the output ports so that it can be held in the circuit board holder:

   When using this circuit with the test holder, the output lines need to have 2.54 cm (1000 mil) centers. Make sure that your circuit conforms to this separation so that you can match one output port while finding the S-parameters for the other port using the network analyzer.

   E. The total length of the circuit should not exceed 6.096 cm (2400 mils). This is the maximum amount the test holder can be opened while still being able to place a circuit inside of it.

   F. Save the circuit and test it on the test bench in Libra. Note: this is a multi-port network and has nine S-parameters describing its operation. Decide which parameters are important for your design and prepare to plot them from 2-3 GHz using the values for i

and j that you have chosen. This is done by choosing Select Palette, Network Parameters, and Sij. Set the values of i and j as necessary.

G. Analyze the circuit and plot it. To see all of the parameters, select Edit, Measurements, and then cut and paste as desired to plot the parameters. It may be best to view one parameter at a time to get a good view of the response. Another way to get a good idea of the actual value of the response at a particular frequency is by using View, Cursor and then moving the cross-hairs to the proper frequency to find its value.

H. Optimize the quarter wave transformers:

Use the tune button next to the plot button to adjust the lengths or widths of the quarter-wave transformer sections. After clicking on the tune button, click on the component that you would like to change and then adjust parameters. After tuning the first time, each modification results in a new plot with a different color. As a result, you can check to see if a better response is attained.

I. When you are satisfied with the response of the circuit, double check to see that the parameters in parts A-D above are met and Synchronize the schematic.

J. Follow the steps for exporting the file to the HPGL format as described in the handout for exporting files in Libra

(or see http://www.engineering.usu.edu/ece/faculty/furse/TUTORIAL/libra/circuitcam.html)

K. Save your data from libra on a disk so that you can compare it with the network analyzer plots: On the graph window, click Graph - New - Table, and the data will appear on a table. Use Edit - Measurements (Add) to put all of the data you want in the table. Cut and paste it to a text file for future use. An easy way to do this is to put the cursor before the first frequency. Hold down CTRL - SHFT - END (there is an END key), and this will highlight the whole document. Open a text editor, and press the middle mouse button to paste. Place a variable name at the top with a square brace (a = [ ) and place a closing square brace at the end of the document. Save the file as file_name.m. When you are ready to use it in Matlab, get in the directory where you saved the file and type: >> file_name. In this case the data will all be stored in the variable "a." You can then use the variable a to make plots or calculations as desired.

L. Your group will turn in the disk with the file to be milled along with the sketch and dimensions of the circuit so that it can be milled. Make sure that the disk and sketch are well labeled. Your group will test the circuit on the Network Analyzer after Spring Break.

Measure your circuit:

1. Attach the circuit board holder to the network analyzer and perform a TRL calibration.

2. Attach your circuit to the holder.

3. View the different parameters on the Network analyzer. Make sure that you match the output port(s) that you are not measuring.

4. Plot your network analyzer results against your libra results on the same graph. See the tutorials on the web for network analyzer data plotting.

5. Also measure the 20 dB coupler provided.

6. Draw conclusions: How well matched are all of your ports? Was the power divided as expected? What losses were observed? (Where did this power go?) Was the power divided as expected for the 20 dB coupler?

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家…

课程网址： http://www.edatop.com/peixun/ads/13.html

HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html