UNIVERSITY OF UTAH ELECTRICAL AND COMPUTER ENGINEERING

ECE 5320

LABORATORY #1 USING AGILENT ADS AND ENA - E5071C NETWORK ANALYZER SUPPLEMENTARY HANDOUT

1. Calibration procedure for ENA E5071C NA

Besides all the safety cautions while using NA, we need to calibrate the NA by a 'Cal Kit' every time before testing to ensure an accurate measurement. The 'Cal Kit' consists two parts: hardware and software, we have developed our own Cal Kit for the lab. Ask your TA for two standard testing boards: a 'THRU' microstrip line and a 'LINE' microstrip line (the longer one is the LINE line). Turn on the NA, wait for warming up. Calibration the NA following the below procedure:

- a. GROUND YOURSELF FIRST BEFORE YOU MANIPULATE ANYTHING ON NA, BE GENTLE TO ALL THE EQUIPMENT AND APDAPTORS.
- b. On the *Stimulus* menu:
 - i. Press *Start* key to set the start frequency at 1 GHz.
 - ii. Press *Stop* key to set the stop frequency at 8 GHz
 - iii. Press Sweep Setup and set up the number of measurement points to 201.
- c. On the *Response* menu:
 - i. Press the *Cal* button, set *Correction* on and off to see the difference in measured values.
 - ii. For further measurement after calibration, always make sure that *Correction* is on. Turn the *Correction* on.
 - iii. Make sure that the *Cal Kit* indicates TRL not 850xx or User. If incorrectly set select the Cal kit option to TRL
 - iv. Select Calibrate, and select 2-Port TRL Cal.
- d. In next window it shows the menu *Thru/Line*, *Reflect* and *Thru/Match*.
 - i. Put the *Thru* board into the adaptor, connect it well, and then select *Thru/Line* followed by 1-2 *Thru/Line*. you will see the NA sweeps the frequency range. The 'THRU' function on the screen will be underlined. Select *Return*.
 - ii. Take off the board, **leave the adaptor open and spread apart**, select *Reflect*, followed by *Port1 Reflect* and *Port2 Reflect* and then select Return.
 - iii. Put the *Line* board into the adaptor, select *Thru/Match* followed by *1-2 Thru/Match*, *1-2 Fwd* (S21) and *1-2 Rvs* (S12). You will again see the NA sweeps and underlines as each option completes. Select *Return*.
 - iv. Finally on the 2-Port TRL Cal menu press *Return* and the NA will calculate the calibration setup for your connections. Now you are ready to do the measurement.
- 2. Read data from Agilent ENA E5071C NA to Agilent ADS (Will also be discuss in ADS tutorial)
 - In ADS schematic window, choose 'File/Instrument Server' from 'Windows' in main menu. The instrument window will open. Click 'Read' and select 'Network Analyzer'. Select 'Database Name', enter any name for your reference, and then click

'Read'.

- In ADS schematic window, select 'new display window' from 'Windows', select your reference file name from 'datasets and equations' box.
- Generate the plot which compares simulated and measured data in Data Display Window (on the same plot for a good comparison). Identify and label every curve. (Although the graph shown on computer is in color, we only have a black and white printer.)

3. Lab report requirement

The requirement is as the same as written in Lab handout 'Write-Up'. Every report will count for 20 points. You will need to specify the lab objective (3 points), procedure for simulation, testing, and circuit analysis (7 points), comparison between the simulated and measured data, and explanation of results (7 points), and make conclusion for your work (3 points). You are greatly encouraged to read through the lab handout and get familiar with the lab design before you come to the lab. Usually one lab will last two weeks and sometime overlap with the next lab, for example, we will design the circuit in the first week and get it fabricated, and then test the circuit in the following week before we start the new lab. The lab reports are due one week after testing the circuit, and will be returned one week after the due date.

Agilent Advanced Design Systems (ADS) Tutorial for Lab 1

Agilent ADS is a powerful, convenient and user-friendly design tool for microwave circuits and components. It has been widely used in teaching in universities as well as in product research and development in industry. Generally speaking, it has the functions such as schematic generation, microwave circuit simulation, layout generation (output to CAD compatible drawings), and full-wave simulation (using MoM).

As ECE 5320 students you will use ADS (2012) for laboratory circuit design and layout and homework simulations. You are also encouraged to practice with ADS as much as possible to have some good experience on microwave circuit design and simulation tools. Below is an introductory tutorial for ADS for Lab 1

ADS software manages its design and data files in so-called 'Workspaces'. It is a file folder which includes subfolders for the library of your design and simulation files.

1. Start to design your microwave circuit using ADS by launching the application. Click on *Create a new workspace*.



Give your workspace and name and choose a location. Click finish.

Workspace Nam Choose a nam	e and location for the new workspace.	
Workspace name:	MyWorkspace_wrk	_
Create in:	C: \Users \Student	Browse
The new workspac C: \Users\S	e is: tudent\MyWorkspace_wrk	
	ent workspace settings:	
 Library Na 	e Name: C:\Users\Student\MyWorkspace_wrk me: C:\Users\Student\MyWorkspace_wrk\MyLibrary_lib ibraries: ADS Analog/RF, ADS DSP	
	ate a new workspace with these settings.	

2. Create a Schematic Window in a Cell.

File View Option	ons Tools W	indow Designk	(its Des	ignGuide Help	
	r View Librar	View			
C:\Users\	Student\MyWorl	kspace_wrk	Library:	Schematic	
			Cell:		Browse Cells.
		7	View:	schematic	Edit View Name
				s able the Schematic W atic Design Template	
			<non< td=""><td>e></td><td></td></non<>	e>	

Open the *Window* menu and select *New Schematic Window*. Designs are grouped into cells. Each cell can have a Schematics, Layout and Symbol that are group together. Give the cell a name. Click OK to continue.

3. Configure the simulation setup

By default the circuit wizard runs to help with getting the schematic started.

e cell_1 [MyLibrary_libxcell_1:schematic) (Schematic):1 File Edit. Select: View Intert Options Tools Layout Simil	ell_1 [MyLibrary_lib:cell_1	ischematic] (Schematic):1 Insert Options Tools Layout Sin	
🗈 Schematic Wizard:1	Schematic Wizard:1		
Stort Application Crcuit Smulation Setup Finish Finish Help Cancel Do not show this dalog again Help Cancel Next >	Application Circuit Simulation Setup	ct an application type Linear Circuit 2-Port 3-Port 4-Port N-Port Active Device Characterization BIT FET Amolifier 4 Stack Next>	
	Lunged Bernert Bandpass		Spedfy the smulation template for the 2-Port Linear Frequency Sweep Logarithmic Frequency Sweep
	Sample designs will be copied to current workspace Use existing design MyLibrary_librarie_librari		Description Sparameters versus linear frequency sver * - S-Parameter simulation controller - Display template - Two port terminations
Help	Cancel < Back Next >	Help	Cancel < Back Finish

For this tutorial we will setup a simulation of an open circuited stub line.

- Select the simulation radio button and click *Next>*.
- Select a 2-port circuit from the *Linear Circuit* application type. Click Next>
- Select I will design my own circuit. Click Next>
- Select Linear Frequency Sweep for the simulation template. Click Finish

A schematic window with all of the require part to perform simulation of a two port circuit are automatically provided for you.

s cell_1 (MyLibrary_librcell_1schematic) * (Schematic):1	
<u>File Edit Select View Insert Options Jools Layout Simulate Window DynamicLink DesignGuide Help</u>	
🗋 🗋 🔒 🔖 🖃 🕪 🗶 🍠 연 🖄 🌵 🕄 🧟 🧶 🧶 🖉 📥 🏦 😤 🛣 🛝 🦲	
Lumped-Components 🔹 🔹 🖓 🕂 🖓 🕂 🕲 🦊 🛣 👧	
Palette 5	
arre arre a construction and a construction of a	
	a a a a a a a a
G C Model Giptemp1.	
Confeed Dock	Quad_dB_Smith"
Start=1.0 GHz SHART MURRO E Stop=10.0 GHz	
Step=100 MHz	
R R	
ste ste Term	
Term 1 Term 2 Num=1 Num=2	
Z=50 Ohm Z=50 Ohm	
→ H move holds	
1000 PLOG	
Select: Enter the starting point 0 items ads_device:drawing 4,750, -0.250	in

4. Add and configure the required blocks for Microstrip circuit simulation

To simulate a microstrip circuit the MSUB block most be added to the schematic to configure the microstrip circuit parameters. For this simulation we will assume that the material we are using will be Rogers Corp RO4350 it has the following properties:

- H = 30 mils (thickness of dielectric)
- Er = 3.48 (dielectric constant)
- T = 1.3 mils (copper thickness)
- TanD = 0.0035 (loss tangent)

Change the parts palette to TLines-Microstrip and add an MSUB part by clicking the part and placing it on the schematic.

C_stub [test_lib:OC_stub	:schematic] * (Schematic):5				
Elle Edit Belest Liew	Insert Options Joels Layout	Simulate Window	EynamicLint Design	iulide Help	
	- H H X 7 C 2	2] 💠 💽 🤣 e	🥺 🥪 🔶 🤰	1 2 41	* 🛛 🗮
TLines-Microstrip	→ads_tlines:MSUB → ○		NAME 👷 🥮	🜵 🛕 🚾 😥	
Palette		· · · · · · · · · · · ·		 	· · · · · · · · ·
	S-Parameter Simul	ation 👩	S-PARAMETERS	DisplayTemplate	· · · · · · ·
Maclin Maclin3 =	Linear Frequency Sweep	S.P.	=1.0 GHz	disptemp1. "S_Params_Qua	d_dB_Smith"
MBstub Mcfil		Stop	=10.0 GHz =100 MHz	Sub	· · · · · · · ·
	· · · · · · · · · · · · · · · · · · ·				· · · · · · ·
Morroso Mourve	Terro Terro	Term Term2	Mu	nd=1.0E+50	
	Num=1 +	Num=2 Z=50 Ohm	T="	=3.9e+034 mil I.3 mil ID=0.0035 ugh=0 mil	· · · · · · ·
			Bbs		
MIGAP3 MIGAP4 - 4		0 items	ads device:drawing 6.00	0, 2.875 0.625,	3.750 in
selects circle the starting point		U ILEITIS	aus_uevice.urawing 0.00	0,2.075 0.025,	57.50 III

Now update the MSUB component with the correct parameters. You can either double click on the part or select the part & click right mouse button and choose *Component -> Edit component parameters*.

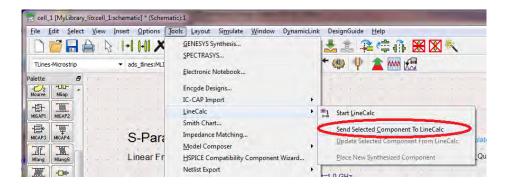
ads_times.MSUB	Parameter Entry Mode Standard
Instance Name (name[<start:stop>])</start:stop>	150-090
elect Parameter	H
H=30 mil	- In [7
Er:: 3.48	
Mur=1	Equation Editor
Cond=1.0E+50	Tune/Opt/Stat/DOE Setup
Hu=3.9e+034 mil	(
T=1.3 mil	
TanD=0.0035	
Rough=0 mil	
Cond1="cond:drawing"	
Cond2="cond2:drawing" *	
4 N F	Display parameter on schematic
Add Cuit Paster	Component Options
: Substrate Brickness	

5. Add an Microstip line and compute the dimensions

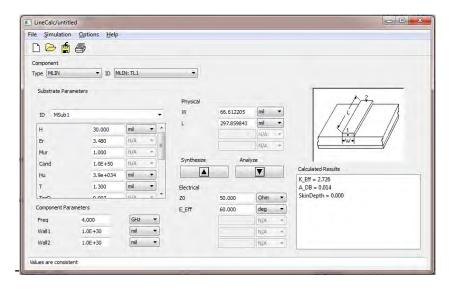
Now it is time to add an MLIN component and compute the microstrip parameters to create a 50 ohm microstrip line. ADS provides an application to compute the required parmeters for the Line.

- Add an MLIN part from the palette. Place it between the *Term* blocks on the schematic
- OC_stub [test_lib:OC_stub:schematic] * (Schematic):5 File Edit Select View Insert Options Tools Layout Simulate Window DynamicLink DesignGuide Help H H X 7 C 2 🕂 C 2 4 🔍 1. 2 1 XX TLines-Microstrip ads_tlines:MLTN 5 alette MSUBST MSUB S-Parameter Simulation DisplayTemplate S-PARAMETERS Maclin3 disptemp1 Maclin Linear Frequency Sweep S_Params_Quad_dB_Smith MSABND MSOBND Start=1.0 GHz Stop=10.0 GHz Step=100 MHz MSub MBstub Mcfil **ISUE** MSubt H=30 mil Er=3.48 Mur=1 ۲ Mcroso Mourve Cond=1.0E+50 Hu=3.9e+034 mil Subst="MSub1" Num=2 W=25.0 mil T=1.3 mil Z=50 Ohm Z=50 Ohm TanD=0.0035 -00-L=100.0 mil Rough=0 mil MGap ·日 MIGAP1 Doeeks= THE MIGAP2 一中 Ш MICAP3 MICAP4 Select: Enter the starting point MLIN TL1 ads_device:drawing 4.625, 3.375 4.000, 3.500 in
- Select the place by click one time on it

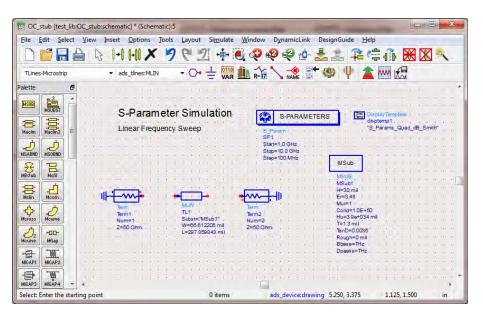
• Open the *Tools* menu and select *LineCalc -> Send Selected Component to LineCalc.* This will open a window to compute the line parameters using the previously define MSUB parameters. You can specify the desire characteristic impedence (Z0) and the desire length of the line in the Electrical area of the window



• Set the Z0 = 50 ignore the E_Eff and click Synthesize. You should find that the interface updates both the W and L of the Physical parameters. The W parameter is the width of a 50 ohm microstrip line on this specific material. The length (L) was calculated using the Freq values of the Component parameters Area and the desired E_Eff. E_Eff is the electrical length in degrees.



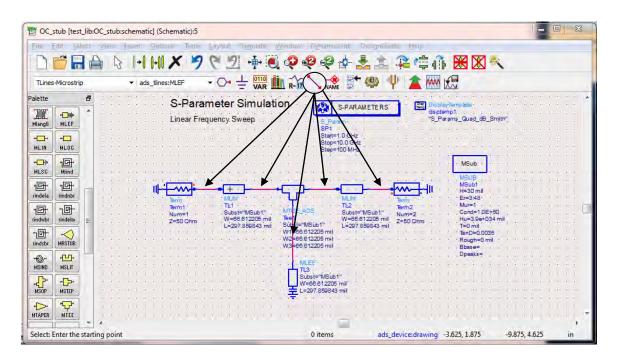
- Lets specify the electrical length to be 60 Degrees at 4 Ghz by changing Freq = 4.0 Ghz and $E_Eff = 60$ degrees and then clicking *Synthesize*.
- Now we want to send the W and L parameters back to the MLIN we created so on the schematic window select the Tools menu and select LineCalc -> Update Selected Component From LineCalc
- The parameters of the MLIN should now match the LineCalc settings.



- Close the LineCalc Window and don't save the configuration.
- 6. Junctions, Open Circuit Parts and Parameters.

The junction parts such as 'MTEE', 'MSTEP', and 'MCROSS' are often used in circuits to model the junction effects. The widths of the various ports should be as the same as the widths of the parts it connects with. There is a small slash mark on one port of the junction part, starting from this port, the port sequence is clockwise counted. There are two types of open circuit parts: MLOC and MLEF. MLOC is an ideal model that the component models the exact length you defined. MLEF will also model the effect effective capacitance which is more realistic.

- Make a copy of the MLIN and place it on the schematic.
- Add an MTEE and set the W1, W2 and W3 values to the width of a 50 ohm line
- Add an MLEF component to the schematic press CTRL + R to rotate the part to a vertical orientation. Set the width and length parameters to match the MLIN parts.
- Connect the parts as shown using the *Wire* button. Click each terminal to connect and hit *Esc* once to end the wire to start a new one.

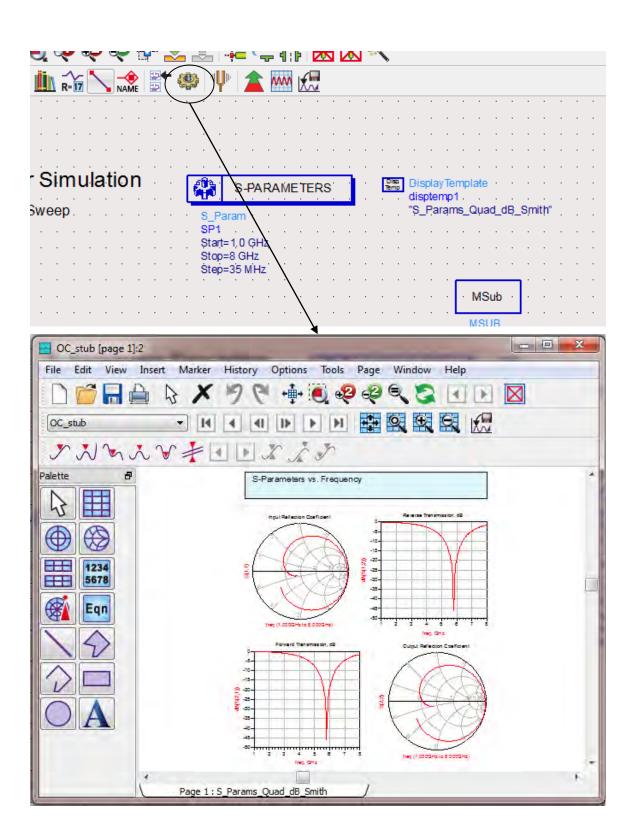


7. S-Parameter Simulation.

We are going to run the S-parameter simulations on ADS in ECE 5320. Set up Sparameter frequency sweeping and terminations (work as both terminations and sources), and grounds. ADS will simulate the S-parameter variations. Double clicking the 'S-Parameters' component we can edit the S-parameter setting in the component box. We usually set the start frequency as 1 GHz, and stop frequency as 8 GHz (which is our calibration range), choose the 'Num. of pts.' as 201 or larger for a smooth curve. (When we do the circuit optimization, the larger number of points means longer convergence time, you may also want to try a smaller number if it takes too long to optimize).

Frequency	Par	rameters	s Noise	e Ou	utput 4
	equency Sweep Ty		Linear	•	
	Start tart	/Stop (Span Hz •	5
	top	8.0		Hz	
	tep-size	35		Hz •	-
	um. of pts				
Ē	Use swe	ep plan		+]

Now that you have setup the sweep, we are ready to simulate. The Nice thing about using the simulation wizard is that it added a component called the Display Template that pre-configures at set of plots showing the S parameters is Smith chart and dB plots depending on the whether the parameter is a reflection or transmission parameters. Click Simulate to see the Display window.



8. Read Data from Network Analyzer.

ADS provides a method to capture data from the NA connected to the computer The application you use depends on the NA that you are using if your at a station next to an 8720 analyzer you will need to find the tutorial to capture for that device on the microwave lab website. *These instructions are for the E5071C*

🔄 [test_prj]	untitled1 (Sch	nematic):1															-	E
File Edit Sele	ct View Inser	rt Options	To	ols Layout Simulate Window Dynamic	Link	E	esigr	nGuid	e I	Help							_	
. 🗅 🕞 😭		0+0 0+00	1	GENESYS Synthesis SPECTRASYS	-2	2	¢				1	6	8	ĵ¢.			3	0
E Lumped-Compon	ents	<]		Electronic Notebook		() NAME	4	0 3	Q	1	~							
Palette	8			Custom Library	•	*	•	• •										
			1	Digital Filter	Ŧ		÷	* 0										
B B Model				Encode Designs				• •										
				IC-CAP Import	1.1		*	• •										
L L Model				LineCalc +														
				Smith Chart														
$\rightarrow \vdash \vdash$				Impedance Matching														
C C_Model		115		Model Composer +	1													
				HSPICE Compatibility Component Wizard		÷.												
DGFeed DGBlck				Netlist Export														
- A				Spice Model Generator														
SHORT MUTIND				User-Compiled Model														
5 - GTP - GTP				Check Representation														
PLC PRC	* • • •			Hierarchy														
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~			6	Info														
PRL PRLC	* * * *		6	Identify														
				Identity				• •			• •							
SLG SRG				Component Palette Configuration	1													
				Hot Key/Toolbar Configuration	1													
SRL SRLC			In	Data File Tool	12													
Select: Enter the s	- I -		6.02		5, 0	0.62	5						in	4	RF	SimS	chei	m
			Í.	Connection Manager Client	É		-			-	-							
			1	Export ADS Ptolemy Design														
				Instrument Server														
				Import System Model														

I. From the schematic window, Click on the *Tools* menu and select *Connection Manager Client* 

The Connection manager Client is used to capture data from the devices attached to the lab computer via a LAN connection and stores it as part of your project data.

<b>B</b> A1	lvanced D	esign Sy	stem 2007	(Main)			
FILE	🖬 [ test.	pri] m	ntitled) (Sci	hematic),1			
	(File )	Agilent	Connection	Manager Clie	ent		
File	📃 File	e Server	Instruments	Measurements	Window	Help	1
	0			Behavioral	Modeling	) I	
	Palette			S-Paramete Spectrum Voltage Wa	1.00	•	8753/8722/8720/8719 Network Analyzer Family 8712/8714 Network Analyzer Family ENA B and PNA Network Analyzer Families
	R					1	8510 Network Analyzer Family
	_2mn_,						
	→ c						
	DCFeed						
C:\use	m						

II. To capture data find the model on the device attached to the computer. For the microwave labs the device is the ENA B (E5071C). Select *Measurements* -> *S*-*Parameters* and the Device.

E Agilent Connection Mana	ger Client	
File Server Instruments	Measurements, Help	
	E Set Server	
	Server Host Configuration Enter the server IP address or DNS name	
	nawave0x.ece.utah.edu	
	Server Port Configuration  Connect to server on default port  Connect to server on non-default port	
	Server port # 4790	
	Do not prompt me again OK Cancel Help	

You will be prompted to Set Server. Click the radio box *Connect to a server on a remote Windows PC*. Set the DNS Name to *nawave0x.ece.utah.edu* where *x* is the number posted on the device. Click OK.

III. The device window for S-parameter Measurements will launch.

Server Instruments Measurements Help	
S-Parameter Measurement - ENA-B Network Analyzer Far	mi 😐 🔍 S
Server nawave01.ece.utah.edu	
Select Instrument	
GPIB1::17::INSTR  Refresh Override inst	rument model check
Select Measurement	
♥ S11 ♥ S12 ■ S13 ■ S14	
▼ 521 ▼ 522 ■ 523 ■ 524 Select All	
S31 S32 S33 S34	
S41 S42 S43 S44	
Export Data	
Export to Text File     File Type     CitiFile	*
File name	Browse
Save Dataset	
Dataset Name Browse	Measure
Block Name CM	Help

IV. Click *Browse*...and give a name to the data set that is alpha-numeric.

Organize 🔻 New folder		<b>ا</b>	0
Name ^	Date modified Type		Size
OC_stub.ds	9/14/2012 10:47 AM DS File		
File name: OC_stub_meas	- 10)		-
Save as type: ADS data sets *.ds			

V. Click *Measure* to perform the data capture. You will notice the displays on the device change as measurement are taken and a window will pop-up to indicate a successful measurement. The captured data should now be available for display in ADS.