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.

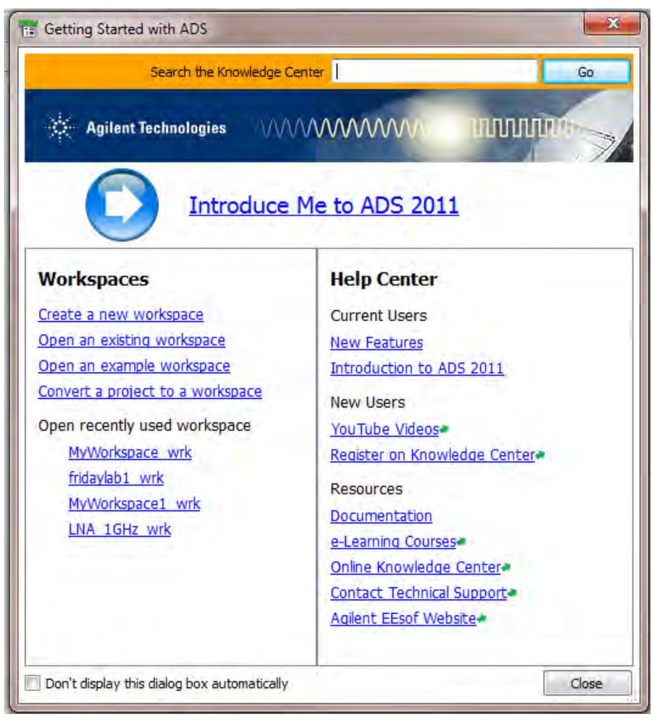
Agilent Advanced Design Systems (ADS) Tutorial for Lab 1

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).

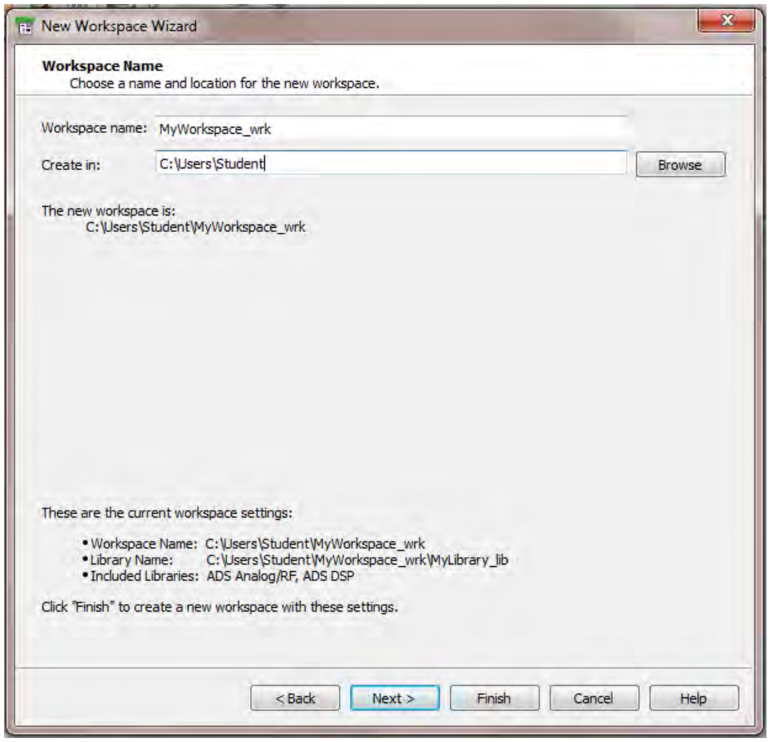
As ECE 5320 students you will use ADS (2012) for laboratory circuit design and layout and homework simulations. You are also encouraged to practice with ADS as much as possible to have some good experience on microwave circuit design and simulation tools. Below is an introductory tutorial for ADS for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

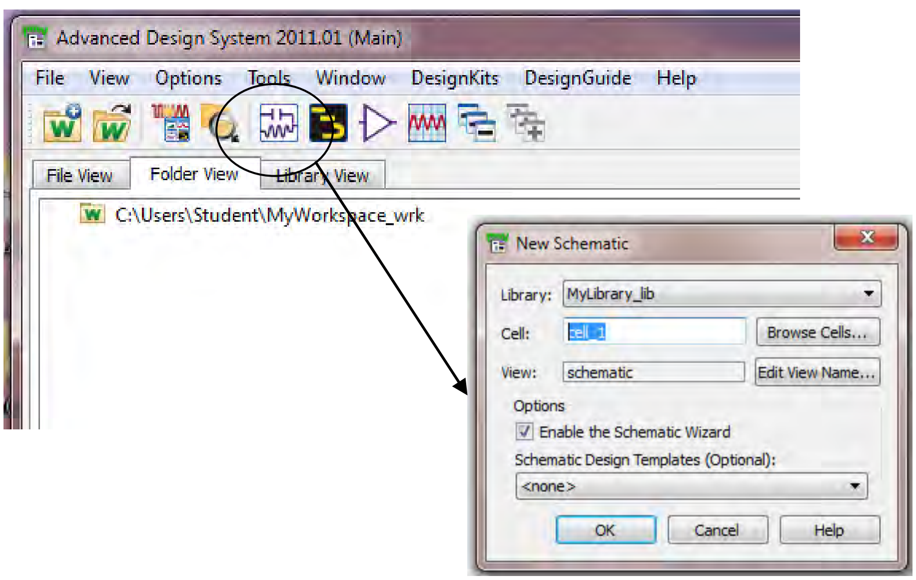
1. Start to design your microwave circuit using ADS by launching the application. Click on *Create a new workspace.*



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

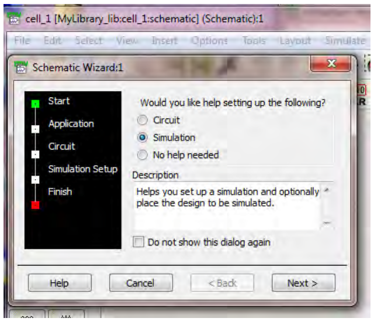


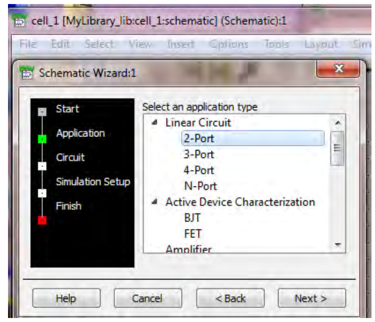
2. Create a Schematic Window in a Cell.

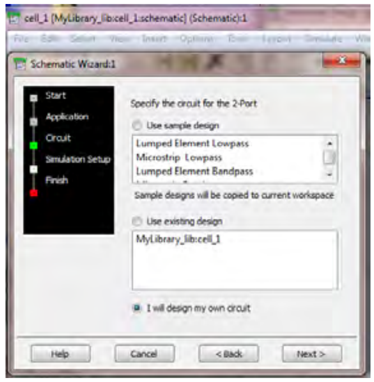


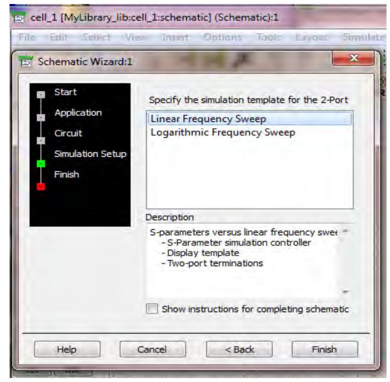
Open the *Window* menu and select *New Schematic Window*. Designs are grouped into cells. Each cell can have a Schematics, Layout and Symbol that are group together. Give the cell a name. Click OK to continue.

3. Configure the simulation setup
By default the circuit wizard runs to help with getting the schematic started.





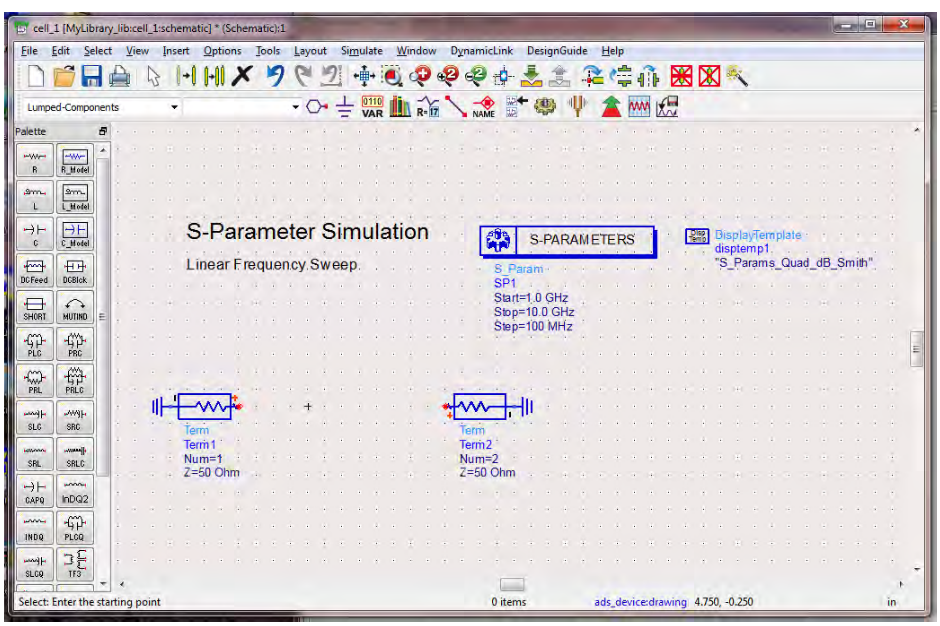




For this tutorial we will setup a simulation of an open circuited stub line.

- 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.

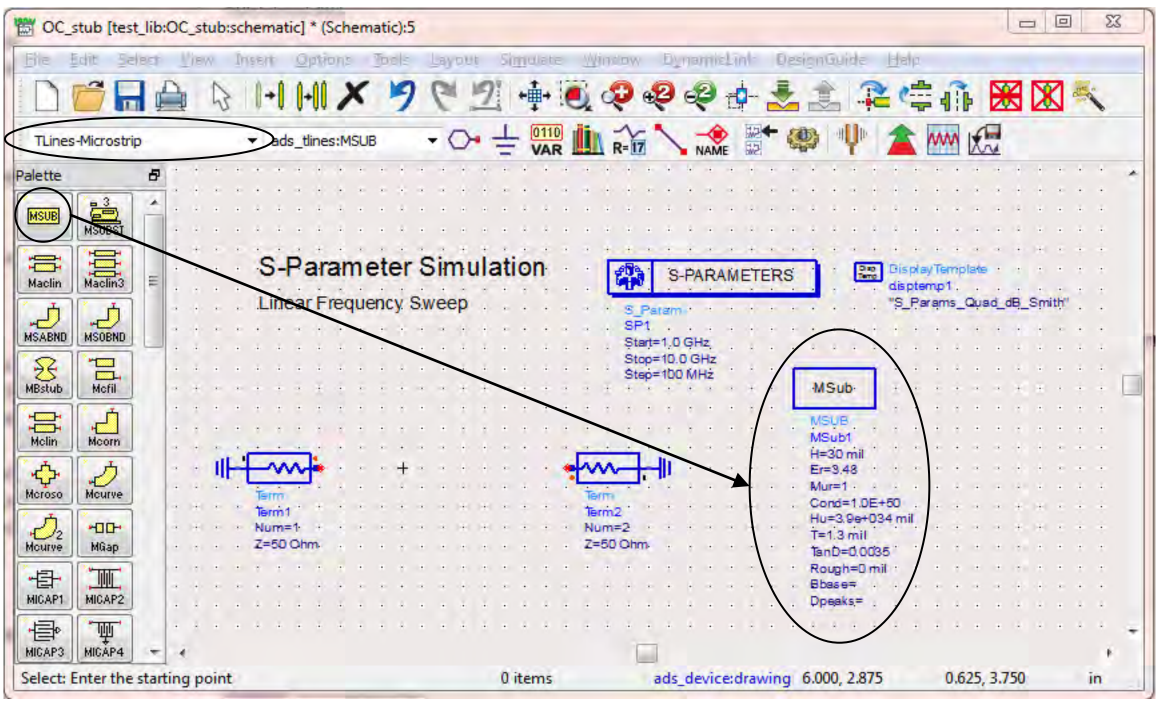


4. Add and configure the required blocks for Microstrip circuit simulation

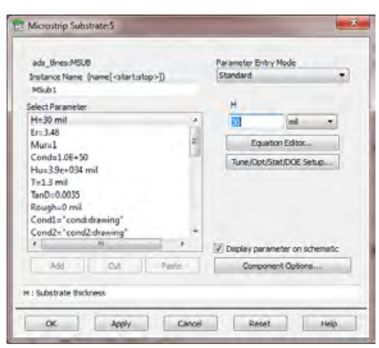
   To simulate a microstrip circuit the MSUB block most be added to the schematic to configure the microstrip circuit parameters. For this simulation we will assume that the material we are using will be Rogers Corp RO4350 it has the following properties:

   - 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.



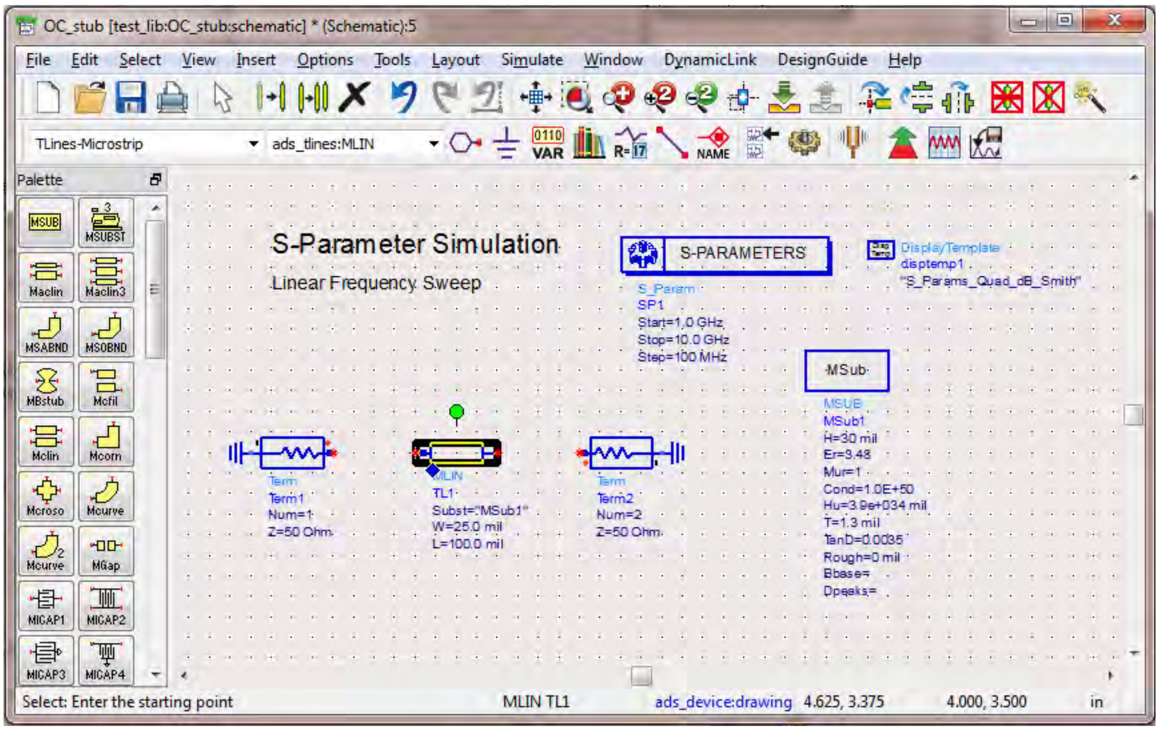
Now update the MSUB component with the correct parameters. You can either double click on the part or select the part & click right mouse button and choose *Component -> Edit component parameters*.



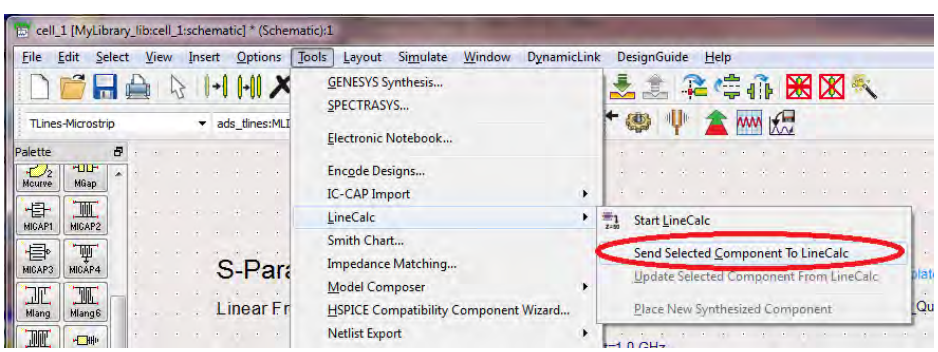
5. Add an Microstip line and compute the dimensions

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.

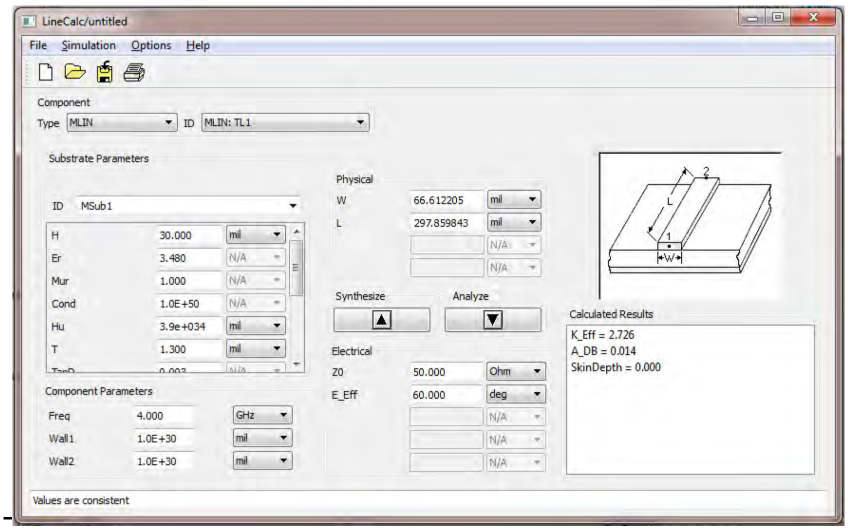
- Add an MLIN part from the palette. Place it between the *Term* blocks on the schematic
- Select the place by click one time on it



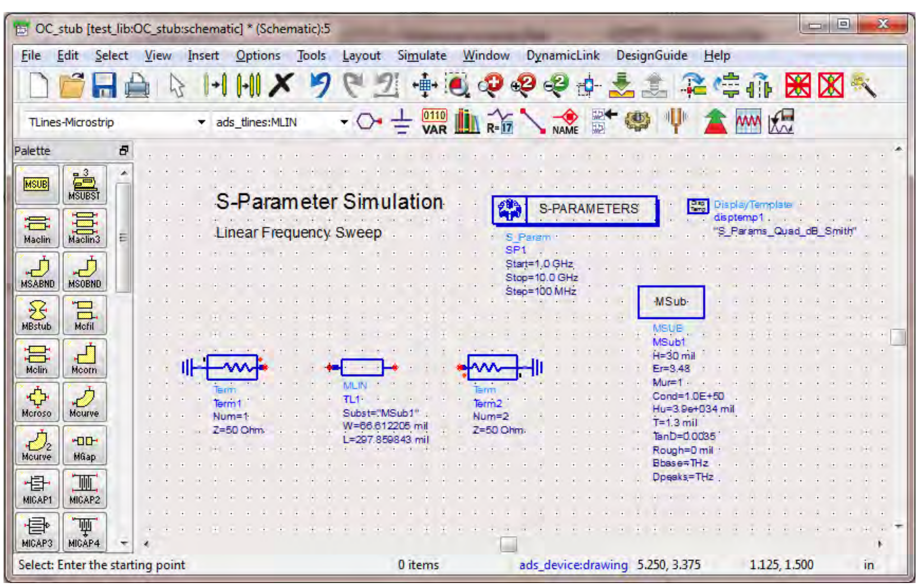
- Open the *Tools* menu and select *LineCalc -> Send Selected Component to LineCalc*. This will open a window to compute the line parameters using the previously define MSUB parameters. You can specify the desire characteristic impedence (Z0) and the desire length of the line in the Electrical area of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

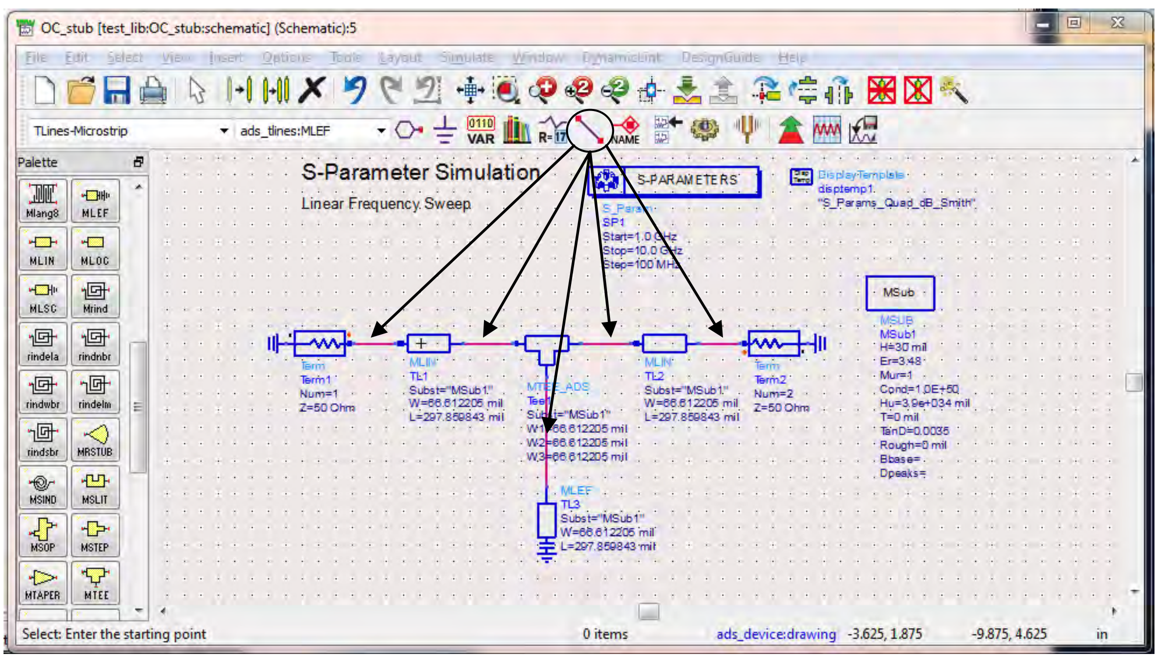


- Close the LineCalc Window and don't save the configuration.

6. Junctions, Open Circuit Parts and Parameters.

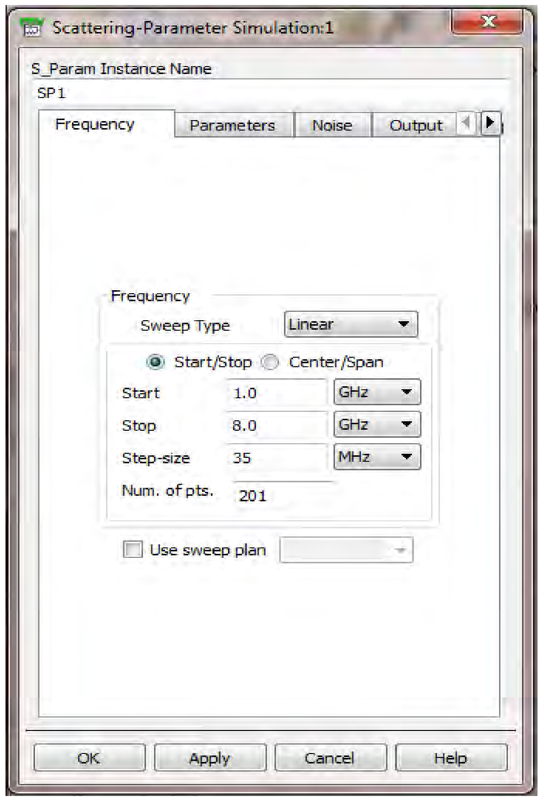
   The junction parts such as 'MTEE', 'MSTEP', and 'MCROSS' are often used in circuits to model the junction effects. The widths of the various ports should be as the same as the widths of the parts it connects with. There is a small slash mark on one port of the junction part, starting from this port, 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.

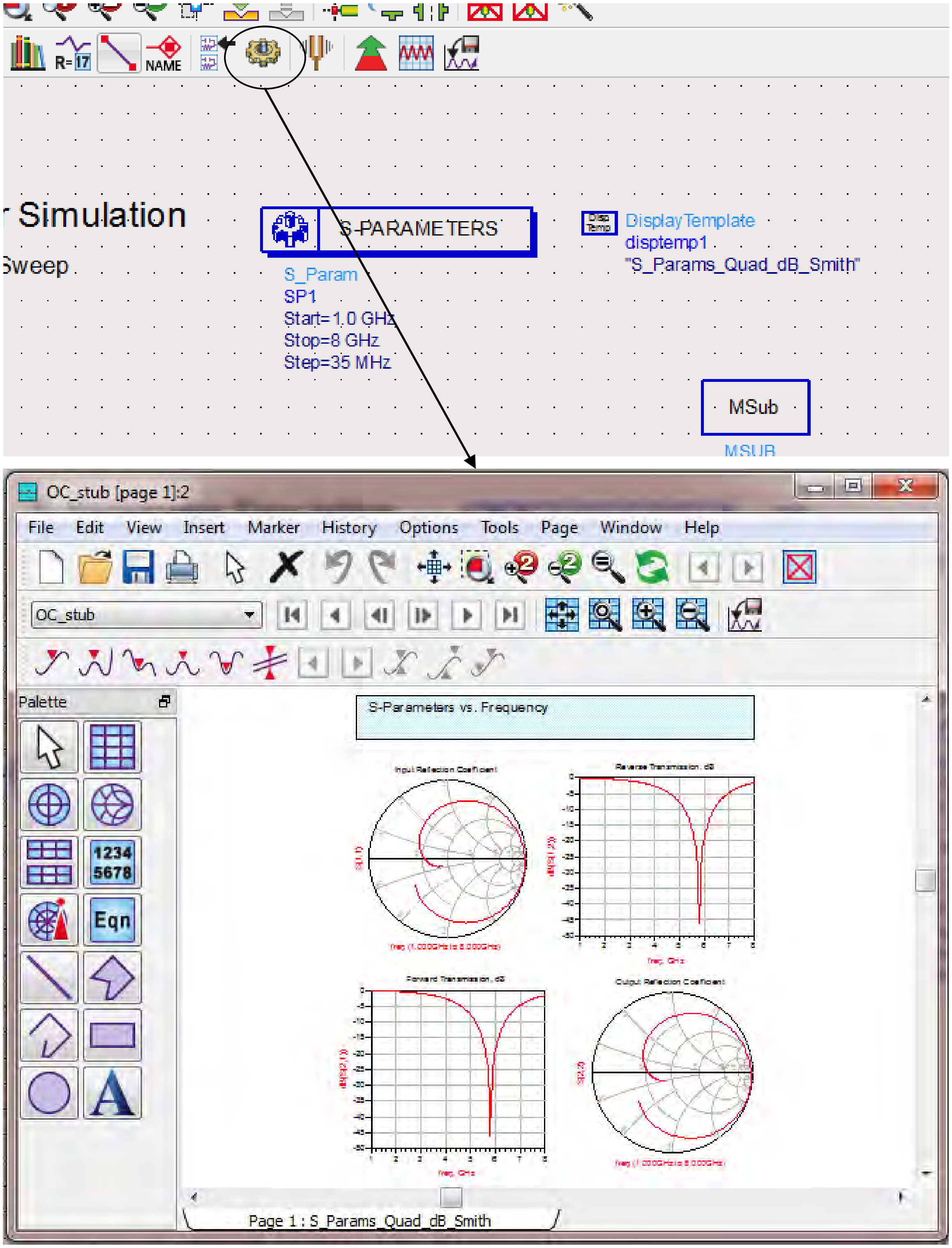


7. S-Parameter Simulation.

   We are going to run the S-parameter simulations on ADS in ECE 5320. Set up S-parameter 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).

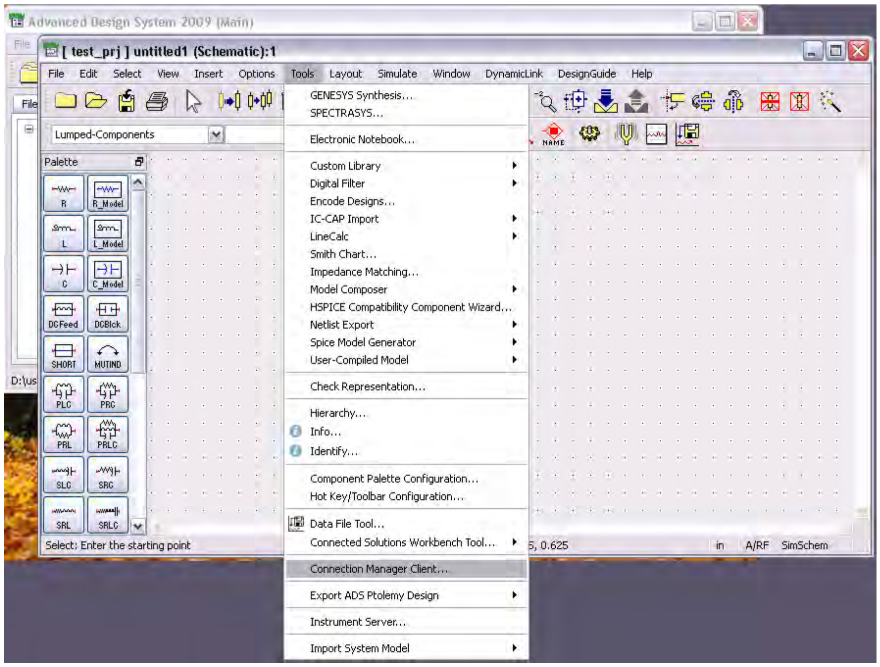


Now that you have setup the sweep, we are ready to simulate. The Nice thing about using the simulation wizard is that it added a component called the Display Template that pre-configures at set of plots showing the S parameters is Smith chart and dB plots depending on the whether the parameter is a reflection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


Simulation
Sweep
S-PARAMETERS
S_Param
SP1
Start=1.0 GHz
Stop=8 GHz
Step=35 MHz
DisplayTemplate
disptemp1
"S_Params_Quad_dB_Smith"
MSub
MSUB
OC_stub [page 1]:2
File Edit View Insert Marker History Options Tools Page Window Help
OC_stub
Palette
S-Parameters vs. Frequency
Page 1 : S_Params_Quad_dB_Smith

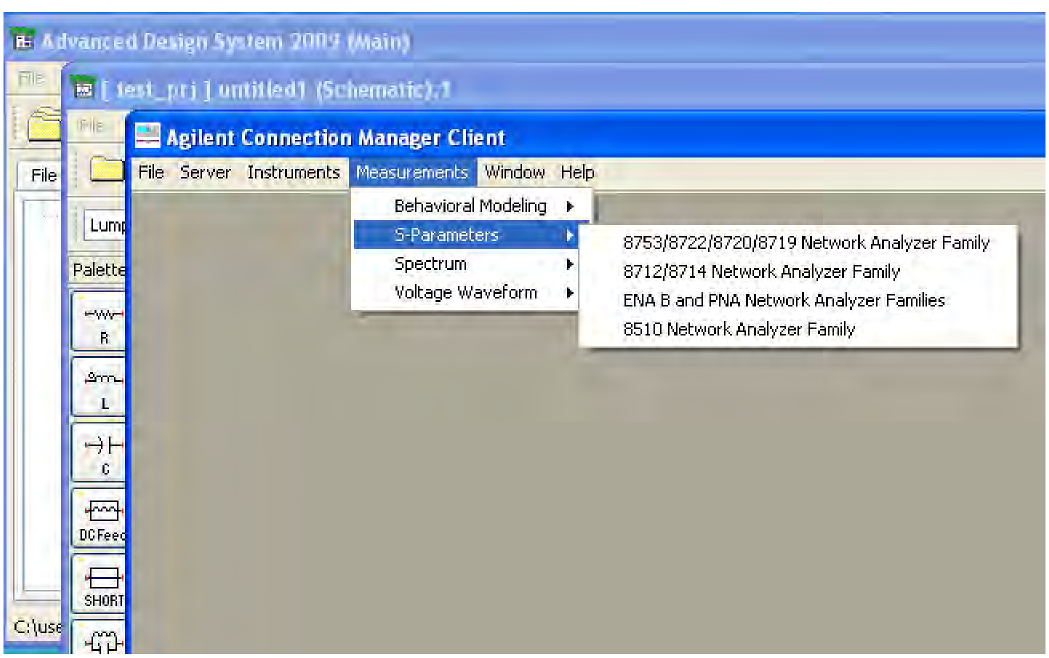
8. Read Data from Network Analyzer.

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***

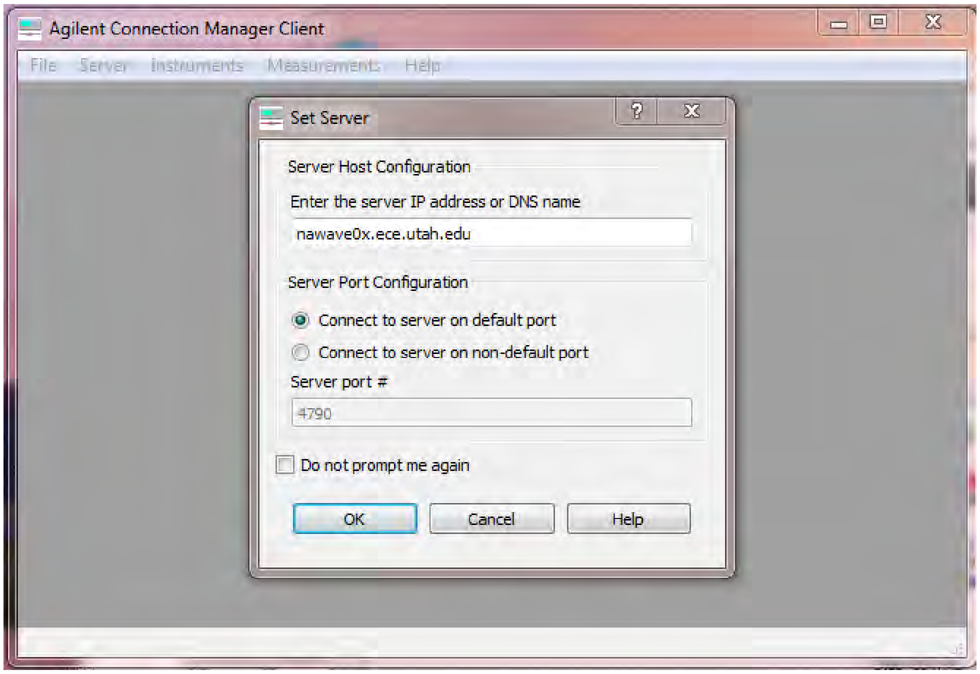


I. From the schematic window, Click on the *Tools* menu and select *Connection Manager Client*

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.

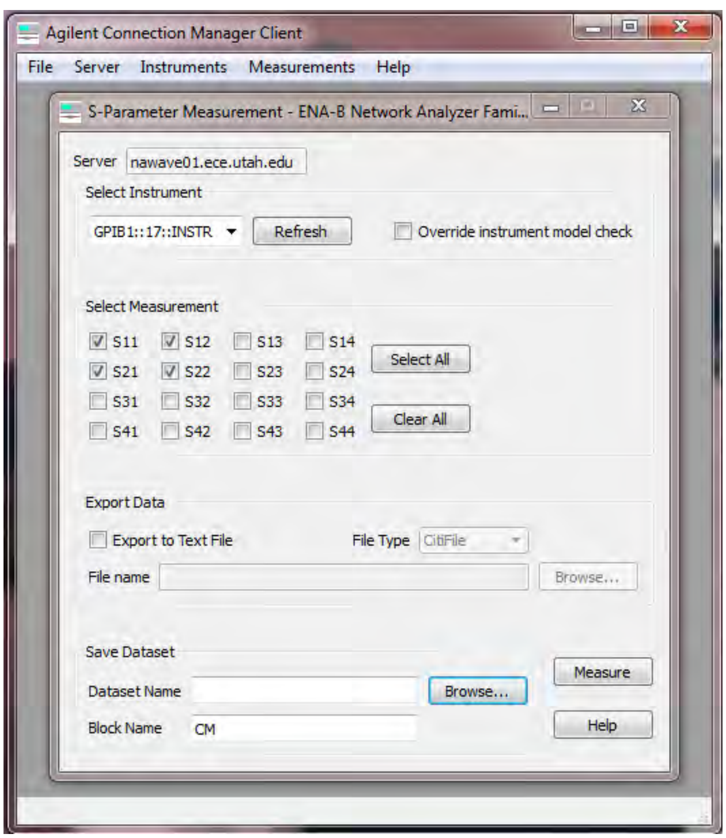


II. To capture data find the model on the device attached to the computer. For the microwave labs the device is the ENA B (E5071C). Select *Measurements -> S-Parameters* and the Device.

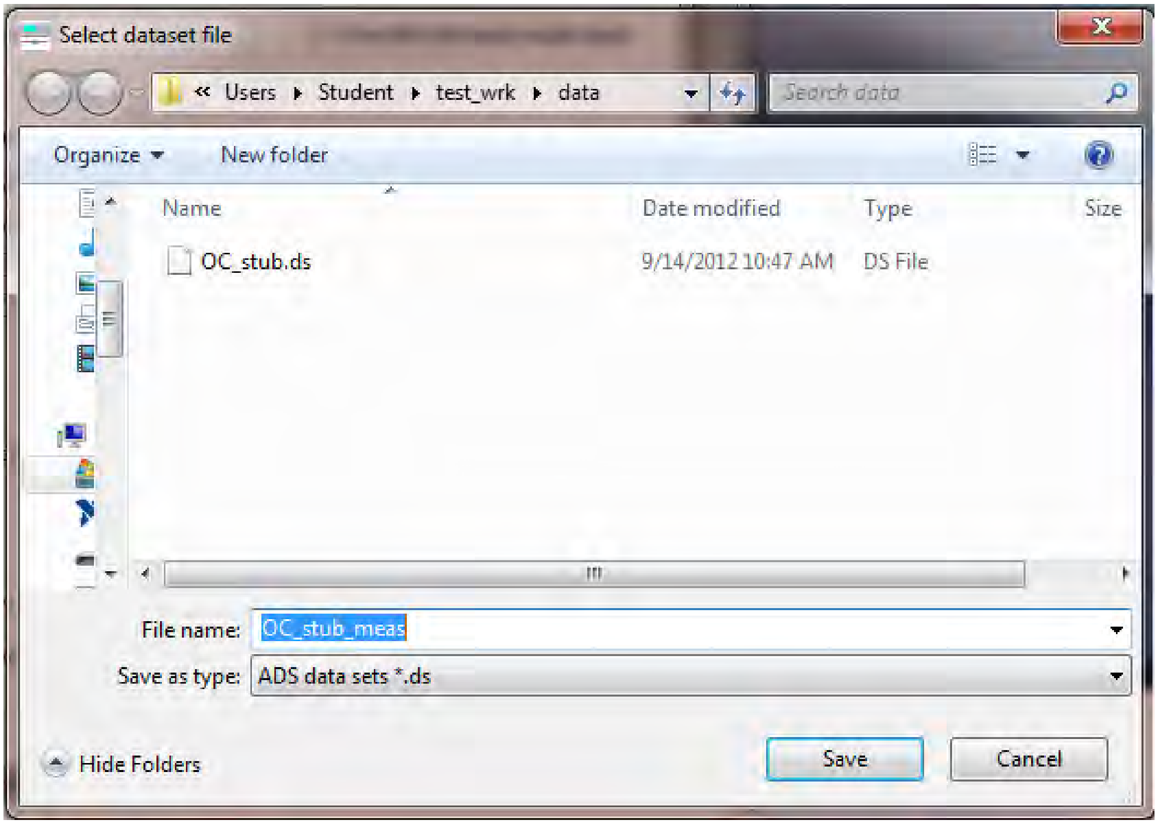


You will be prompted to Set Server. Click the radio box *Connect to a server on a remote Windows PC.* Set the DNS Name to *nawave0x.ece.utah.edu* where *x* is the number posted on the device. Click OK.

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



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



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.