

Wireless Circuits and Systems Laboratory

Procedure #2 Simulation of Multiple DUTs Using ADS

This procedure relies on information contained in WAMI Lab Procedure 1. Contained in this procedure:

- 1) The difference between a “Port” and a “Termination” in ADS
- 2) How to reference a sub-circuit from a higher-level schematic
- 3) A method for simulating two different circuits simultaneously in the same schematic window

In the following exercise, we will run through a procedure for simulating two different circuits simultaneously from one schematic window. This procedure can be useful when we want to compare the frequency response of two or more circuits, when we want to compare a simulated frequency response to measurement data, and for other processes like circuit optimization.

In order to access two or more circuits from one schematic, an understanding of the difference between a “termination” and a “port” is first needed; this topic is addressed in Section 1. In Section 2, the process of referencing a sub-circuit from a higher level schematic is then described. Finally, in Section 3, an example of simulating two sub-circuits from a higher-level schematic is given. (Section 3 makes specific references to circuit schematics associated with Laboratory Experiment #6 – Impedance Matching.)

1. “Ports” versus “Terminations” in ADS

In Procedure #1, a description is given for simulating the S-parameters of a lumped element low-pass filter. The filter schematic was comprised of inductors and capacitors, ground elements, and “Terminations.” The (default) impedance for the terminations was 50 Ohms (see Figure 1). It is ALWAYS necessary to add terminations to a DUT (device under test) in ADS when you are simulating S-parameters, because the Term element indicates the reference impedance at each port - it is a characteristic of S-parameters that they depend on the reference impedance at the ports.

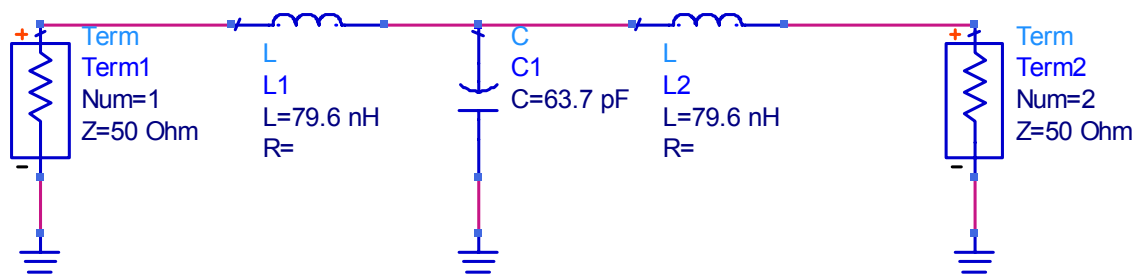


Figure 1. Lowpass filter circuit with “terminations”. This schematic can be used to perform S-parameter simulations.

Another “element” that can be added to a terminal of a circuit (e.g., consider just the inductors and capacitor of the low-pass filter) is a “Port” (see Figure 2). If you only have “Ports” at the terminals of a circuit you will not be able to perform an S-parameter simulation. The Port(s) only indicate the points at which energy (RF and/or DC) can enter or leave the circuit. In the schematic window, you can find a button with the port icon located three to the left of the Library (books) icon.

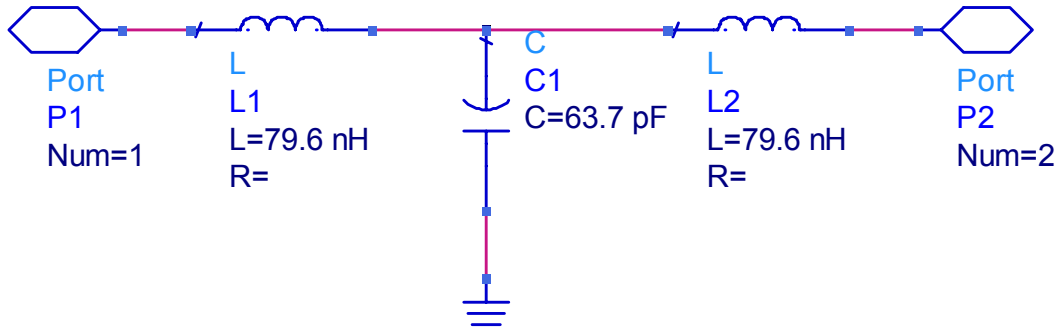


Figure 2. Lowpass filter circuit with "ports". This schematic cannot be used to perform S-parameter simulations, but it can be referenced from a higher-level schematic.

In this procedure, the use of a Port will be demonstrated in the creation of a “higher-level schematic” that references a “sub-circuit.” In other words, we will create the lowpass filter circuit and add ports. Then, we will create a second schematic that only references the lowpass filter circuit – the lowpass filter circuit will show up as a “black box” in the higher-level schematic.

2. How to Reference a Sub-Circuit from a Higher-Level Schematic

When you click on the Library (books) icon in the ADS schematic window, a pop-up window appears with a list of libraries. If you go into the All -> Sub-networks directory (click on the small box to the left of each label, to change the character inside the box from a “+” to a “-”) you will see a list of project directories. Finally, if you click on the name of a project directory that you have created (you may only have one at this point) you will see a list of “Components” that shows up on the right-hand side of the pop-up window. Each of the “components” is actually a circuit schematic that you have already created; any of these can now be referenced from a different “higher-level” schematic. In order to place a copy of the “component” into your higher-level schematic, click on the component name and then click in the schematic window.

For the sake of argument, assume that the schematic shown in Figure 2 (above) was given the name `lpf_1_port` (to distinguish it from the version that uses terminations, as shown in Figure 1). Now assume we create an entirely new schematic, which we will refer to as the “higher-level” schematic. We can reference the `lpf_1_port` circuit following the procedure described in the previous paragraph, and place a “black box” representation of `lpf_1_port` in the higher-level schematic. At this point we can also add “terminations” to the “black box” and perform S-parameter simulations from the higher-level schematic if desired (see Figure 3). (After finding `lpf_1_port` listed in the project directory in the Library, and then selecting it and clicking into the schematic window, the box in the middle of Figure 3 will appear.)

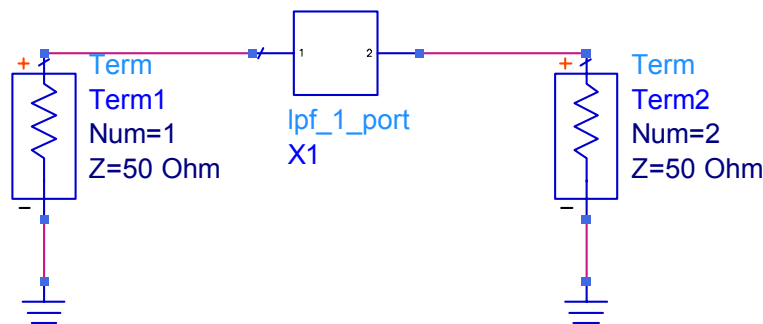


Figure 3. Higher-level schematic that references the lowpass filter circuit shown in Figure 2 (the name of the schematic in Figure 2 is assumed to be `lpf_1_port`). The lowpass filter is represented by the “black box” in the middle of the figure.

3. Simulating Two Different Circuits Simultaneously in the Same Schematic Window

In the following steps we will:

- Create two separate, one-port circuit schematics that use a “port” versus a “termination”
- Create a “higher-level” schematic that references the two one-port circuits, and add a “termination” to each referenced circuit
- Simulate the S-parameters for both circuits simultaneously and view the results in the graphics window

NOTE: The one-port circuit schematics that are created in this procedure are NOT representative of those that are used in **Laboratory #6 – Impedance Matching**. In this procedure, ideal transmission lines are used. In **Laboratory #6** you are required to create schematics using microstrip transmission line sections. The process of creating microstrip-based schematics is outlined in Procedure #6. Aside from the composition of the actual matching circuits, all other steps are identical to those that are required in the impedance matching laboratory.

- Following the steps described in Procedure #1, open the schematic window and create a new schematic called **matcht_fake**. As developed in the following steps, this will become a quarter-wave impedance matching network that uses ideal transmission lines to match a 100 Ohm load to 50 Ohms.
- Click on the Component Palette List pull-down menu and select the Lumped-Components list. Click on the resistor (“R”) button on the upper left-hand side of the window, place a resistor in the schematic, and change its value to 100 Ohms. (You have to double-click on the resistor after placing it in the schematic window.)
- Click on the “ground” button (located two to the left of the Library button) and place a ground to the right of the resistor.
- Click on the Component Palette List pull-down menu and select the TLines-Ideal list. Click on the TLIN button on the upper left-hand side of the window and place an ideal transmission line section in the schematic. Change the characteristic impedance (Z) to 70.7 Ohms, the electrical length (E) to 90 degrees, and the reference frequency (F) to 2 GHz.
- Click on the Port button (3 to the left of the Library button) and place a port on the left-hand side of the circuit. Now use the Wire tool to connect the elements. Your schematic should resemble Figure 4. **SAVE YOUR DESIGN.**

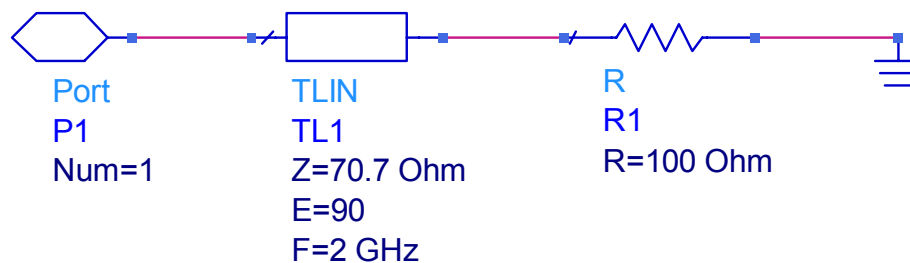


Figure 4. Illustration of the matcht_fake schematic described in the text.

- Now save matcht_fake as matchtb_fake (use the File->Save As command to create a new schematic). Change the load resistor value to 200 Ohms and the characteristic impedance of the transmission line to 100 Ohms.
- Now that we have created our two individual circuits, we will create the “higher-level” schematic and reference these two from it. First, use the File->New command and create a schematic called matcht_lab.

8. Click on the Library (books) icon and open the Sub-networks directory. Now double click on your project directory name.
9. At this point, you should see the matchkt_fake and matchtb_fake schematics in the list on the right-hand side of the window. If these schematic names do not appear, you may have to close the Sub-networks directory and re-open it, in order to refresh the display.
10. Click on the matchkt_fake schematic name, minimize the Component Schematic/Library pop-up window (so you can see your schematic window) and place one copy of the “black box” that references matchkt_fake into your schematic. Repeat the process for matchtb_fake.
11. Click on the Component Palette List pull-down menu and select the Simulation-S_Param list. Place a termination near each “black box”, ground each termination, and then connect the ground/termination/black box using the wire tool.
12. Finally, click on the button on the upper left-hand side of the window and place a S_Param:S-parameter Simulation element in the schematic window. Set the start frequency to 30 KHz, the stop frequency to 3 GHz, and the step size to 0.01 GHz. Your schematic should now resemble Figure 5.

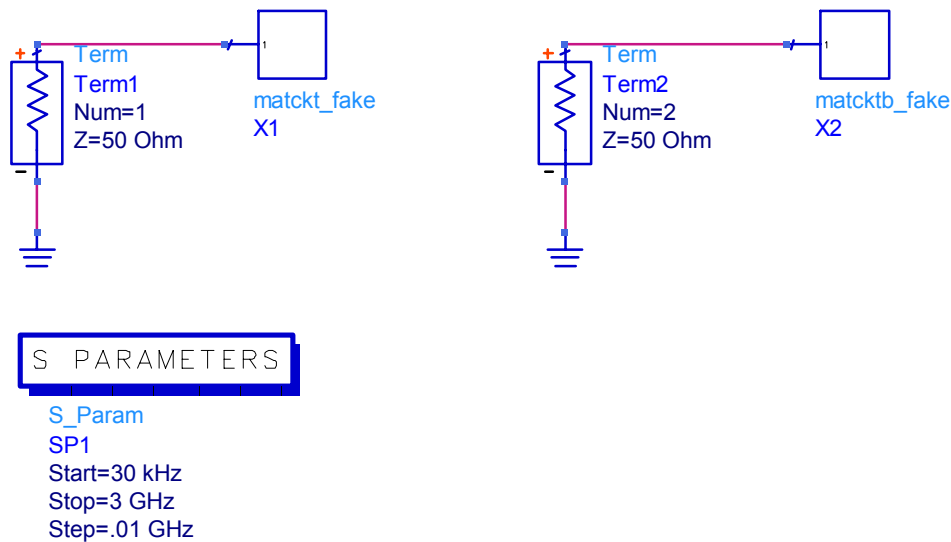


Figure 5. Illustration of the "higher-level" schematic that references two individual circuits as black-boxes.

13. Before proceeding with the simulation, it is important to point out something relating to the numbering of the port terminations. Notice in Figure 5 that Term1 is connected to matchkt_fake and Term2 is connected to matchtb_fake. Since ADS sees TWO terminations in the schematic it will automatically assume that this is a TWO-PORT circuit, while in reality we have two, 1-PORT circuits that are not physically connected to each other. Because ADS assumes this is a 2-port circuit, it will calculate four S-parameters (S_{11} , S_{21} , S_{12} , S_{22}). Clearly, S_{21} and S_{12} have no meaning here and we will thus disregard them. S_{11} refers to the input reflection of matchkt_fake and S_{22} refers to the input reflection of matchtb_fake. (Clearly this procedure becomes increasingly complicated as more black-box circuits are added to a schematic, and when the number of terminations on any given black box increases, e.g., if each black box was a 2-port instead of a 1-port network.)
14. Click on the Simulate button (five to the right of the Library button).
15. Open the graphics display window. Choose a rectangular plot and add S_{11} and S_{22} to the plot (each in dB). Now double click on the plot, click on the Plot Options tab, unset the Auto Scale option and set the minimum value to -50 . Your graph should now resemble Figure 6.

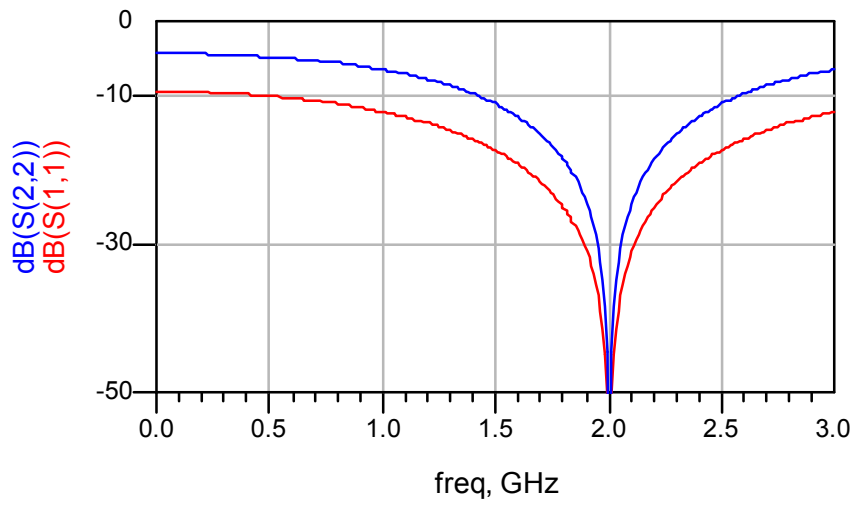


Figure 6. Plots of input reflection coefficient for matcht_fake (S11, lower/red curve) and matchtb_fake (S22, upper/blue curve).