

## *RF/Microwave Circuits I Laboratory #2*

### Circuit Tuning and EM Simulation with *Momentum*

#### Overview

This laboratory continues the ‘basic ADS training’ approach from Laboratory #1. The two main objectives for this assignment are:

- 1) Gain experience using the *tuning* feature in ADS to manually optimize a circuit, and
- 2) Gain experience using the numerical electromagnetic (EM) simulator *Momentum* that is part of the ADS software suite.

As you complete this assignment you will be developing a design for a component that can be used as an open-circuited “harmonic” tuning stub. The harmonic tuning concept will be discussed in more detail later in the semester. For now, it is important to understand that **the stub should provide a high reflection to signals near 5 GHz.**

In the first part of the laboratory you will set up a schematic for the tuning stub using the circuit schematic window in ADS. The design consists of microstrip input and output lines, a tee-junction and a shunt stub in the center. For this design we will use a microstrip ‘radial’ stub, rather than a simple straight stub. The reason for this is that a radial stub can provide more bandwidth than the straight design. After the schematic is setup, you will then use the ‘tune’ feature in ADS to manually adjust the stub length so that it provides the desired response (high  $S_{11}$  near 5 GHz).

In the second part of the laboratory you will use the *Momentum* EM (electromagnetic) simulator to analyze the stub, and compare the resulting data to that from the circuit simulator model. Rather than manually drawing the microstrip layout, you will use the ‘schematic capture’ option to automatically generate the physical layout from the circuit schematic description.

As with Laboratory #1, you will be referring to several of the ADS tutorial procedures that are available from the class web page. The assignment is organized as follows:

- ◆ Follow *Procedure #11 Circuit Tuning using ADS* to create the circuit schematic and tune the stub geometry for optimal response
- ◆ Follow *Procedure #12 Schematic Capture in ADS* to automatically generate a CAD layout from the circuit schematic description
- ◆ Follow *Procedure #8 Using the Momentum EM Simulator in ADS* to perform a numerical electromagnetic analysis of the layout
- ◆ Follow *Procedure #3 Creating a Circuit Element using Measured Data in ADS* to generate a schematic that references the *Momentum* output data, and subsequently compare the circuit simulator and *Momentum* results

**NOTES:**

1. The ‘procedures’ you will use were originally created for ADS Version 1.3. and some updates have been made to retain compatibility with later releases of ADS. You will be using a version > 1.3 in the laboratory assignments this semester. I expect that any discrepancies that result from the version differences will be minimal and easy to work through; if you find that not to be the case, please contact me to resolve any issues that arise.
2. **The ADS procedures are intentionally generic in that the substrate definitions, frequency ranges and even circuit configurations may not exactly match those of this laboratory assignment. Make sure that you use the parameters that are specified in this document.**

**Laboratory Assignment**

- 1) Follow the steps in Procedure #11 to create the circuit schematic and tune the length of the stub. As mentioned, the goal is to find the stub length that provides the maximum reflection (highest  $S_{11}$ ) at 5 GHz.
  - a) Your “design frequency” is 5 GHz whereas the frequency in Procedure #11 is 4 GHz.
  - b) The substrate parameters that you need to use are the following:  $H=0.125$  mm,  $\epsilon_r=9.6$ ,  $\mu_r=1$ ,  $\text{Cond}=1.0E+50$ ,  $H_u=1.0E+33$  mm,  $T=0.005$  mm,  $\text{TanD}=0.0009$ ,  $\text{Rough}=0$  mm. Note that these are also the parameter values you will use in the electromagnetic simulation part of this exercise.
  - c) The width of a  $50\Omega$  line on the substrate you are using should be determined using the Linecalc feature in ADS (refer to Procedure #5 for directions on using Linecalc) – do not use the default value stated in Procedure #11 (3.058 mm).
  - d) The length of a quarter-wavelength line on the substrate you are can also be determined using Linecalc – do not use width stated in Procedure #11 (10 mm).
  - e) Perform the circuit simulations from 0.5 GHz to 8 GHz in steps of 0.1 GHz, instead of using the frequency range given in Procedure #11.
    - i) You may get warnings from the simulator indicating that certain parameters in your design are “out of range”
  - f) When you have finished the steps in Procedure #11, **generate a hardcopy of the circuit schematic (with the final stub length clearly indicated) and the graph showing the magnitude (in dB) and phase of the S-parameters ( $S_{11}$  and  $S_{21}$ )**. Put the magnitude results into one graph and the phase in another. NOTE: To make parameters available for tuning in ADS 2004, you can click on the parameters in the schematic after pressing the “Tune” button. After tuning is complete, click on the “Update Schematic” button in the tuning pop-up window to transfer the new parameter values to the schematic.

- 2) Follow the steps in Procedure #12 to create a layout of the circuit using schematic capture.
  - a) The “optimum stub length” described in Procedure #12 is 6.92 mm, whereas the stub length for your design will be that determined during the steps given above.
  - b) NOTE: When using the auto-generation of the layout from the schematic, after clicking on Layout->Generate/Update Layout...
    - i) Click on Preferences
    - ii) Make sure the Entry Layer box is set to “v,s cond”
  - c) **Generate a hardcopy of the layout when you are finished with Procedure #12.**
- 3) Follow the steps in Procedure #8 to perform a *Momentum* simulation of the layout.
  - a) While you should read the first few pages of Procedure #8, you will actually begin the procedure around the bottom of page 4 (“Add Ports” in Section II). The steps preceding this assume that you are manually creating the layout, and thus don’t apply in this case. You will follow Procedure #8 up to the middle of page 8 (at which point the topic changes to the use of ‘air bridges’ which does not apply in this case).
  - b) Note that the example circuit described in Procedure #8 is different from the one you with which you are working. Also, **you should use the same substrate parameters and frequency range that were used in the circuit schematic.** (For example, you should ‘solve the substrate’ from 0.5 to 8 GHz and set the Mesh Frequency to 8 GHz.)
  - c) When you create/modify the substrate definition there will be a few differences with respect to Procedure #8.
    - i) Because you created your layout using schematic capture, and the schematic contained a microstrip substrate definition, you **may** find that the proper substrate parameters have automatically been transferred to *Momentum* (i.e., a 0.125 mm-thick substrate with a dielectric constant of 9.6 and a loss tangent of 0.0009). If not, you can enter them manually as described in Procedure #8.
    - ii) Similarly, you **may** find that the metal traces (which are automatically generated on the ‘cond’ layer in the layout) have been automatically ‘mapped’ to the proper layer in the substrate definition (between the top of the substrate and free space).
    - iii) **You can ignore the steps relating to the ‘via layer’ that are included in Procedure #8.**
  - d) When the *Momentum* simulation is finished a data display window should open automatically. **Generate a hardcopy of the display window.** (You should also save this display to a file for future reference.)
- 4) Follow the steps in Procedure #3 to create a schematic that references the data generated during the *Momentum* simulation.

- a) Whereas Procedure #3 relates to referencing ‘measured’ data, here you are referencing ‘simulated data’. The basic idea is simply that you are obtaining the information from a file.
  - b) Whereas Procedure #3 refers to the ‘Touchstone’ file type, you should specify the ‘Dataset’ file type. These files have a .ds extension, and are the type automatically generated/stored following the *Momentum* simulation. (Touchstone file types have an .snp extension, where “n” refers to the number of ports.)
  - c) The schematic that you create should resemble the one shown in Figure 1. With regard to example file names used here,
    - i) The circuit schematic that was used to generate the layout (through schematic capture) was called schem\_capture\_radial\_1 (this corresponds to whatever filename you gave the schematic that you will have worked with during Procedures #11 and #12)
    - ii) The layout file and all generated data files **are associated with the name of the original circuit schematic**, and therefore have the same name. As a result, the interpolated dataset created by *Momentum* is called schem\_capture\_radial\_1\_a.ds (recall that the “\_a” indicates data from the adaptive frequency sweep and the “ds” extension indicates dataset).
    - iii) I gave the schematic that I created specifically to reference the *Momentum* dataset a slightly different name, schem\_capture\_radial\_1\_MOM (this is the name of the schematic shown in Figure 1). Notice that the S2P element in the figure references the dataset schem\_capture\_radial\_1\_a.ds
    - iv) The detailed parameters for the S2P element are shown in Figure 2. Note that the Type parameter is set to Dataset.
  - d) When you have created the schematic shown in Figure 1, generate a graph that contains the S-parameters ( $S_{11}$  and  $S_{21}$ , in dB format, from 0.5 GHz to 8 GHz in steps of 0.1 GHz) calculated by the circuit and *Momentum* simulations.
    - i) Analyze the schematic that references the *Momentum* results.
    - ii) Now open (or re-open) the original circuit schematic (from Procedure #11) and analyze this schematic.
    - iii) Both results should now be stored into memory. If you open a graph window, you should be able to select either ‘dataset’ from the pull-down menu, and plot the S-parameters from each dataset on the same graph.
    - iv) **Generate a hardcopy of the comparison graph.**
- 5) Your report should include the following:
- a) Printout of the microstrip circuit schematic and the layout
  - b) Graph of the circuit simulation results
  - c) Graph of the *Momentum* simulation results

- d) Graph comparing the circuit and *Momentum* simulation results.
- e) Answer the following questions:
- i) What is the percent discrepancy in the frequency at which the minimum S21 occurs between the circuit simulation and the Momentum simulation (using the circuit simulation result as the reference)?
  - ii) In general terms, how would you modify the layout/geometry so that the Momentum simulations showed the S21 minimum occurring at 5 GHz?
  - iii) For the circuit simulation results, what is the percent bandwidth over which S21 is less than or equal to  $-20$  dB?

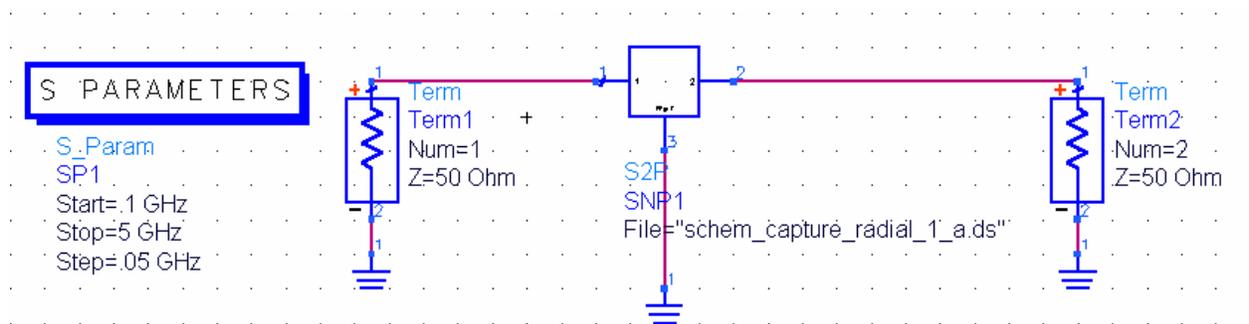


Figure 1. Schematic created to reference the dataset generated by Momentum. This schematic was called *schem\_capture\_radial\_1\_MOM*. (Note: for this laboratory you should set the S-parameter simulation from 0.5 to 8 GHz in steps of 0.1 GHz.)

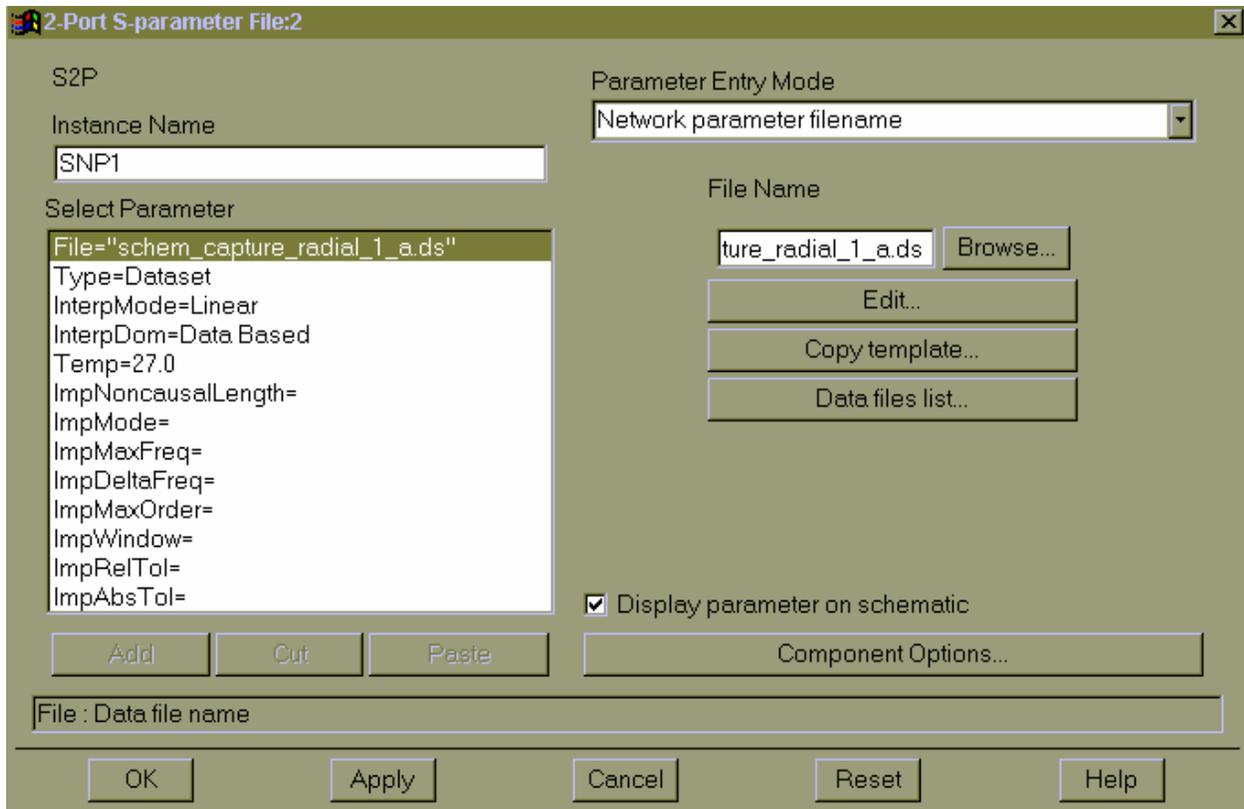


Figure 2. Detailed parameters for the S2P element shown in the previous figure. Notice the 'Type' is set to Dataset, and the filename has the .ds extension.