RF/Microwave Circuits I Laboratory #1

Equivalent Circuit Modeling (and Optimization)

Overview

There are two objectives for this software laboratory assignment:

1) Gain experience developing equivalent circuit models that accurately represent known (in this case, measured) performance of a microwave component, and

2) Learn how to use the *Advanced Design System* (ADS) CAD/CAE software from Agilent Technologies.

The first objective is important because equivalent circuit modeling is an important tool for RF/microwave designers, and in particular because you will need similar models to design your lumped element IF filter later in the semester.

The importance of the second objective is that you will be using ADS in each of the subsequent software laboratories. However, 'learning how to use ADS' is a lengthy process, even to the extent that you will gain experience this semester. For this reason, the steps you are asked to take in this laboratory are outlined in great detail and there is only a minimal amount of work to do on your own. Essentially you should follow step-by-step through a series of procedures that result in the generation of an equivalent circuit model. The only additional step is to repeat the process for a second component; if you have followed the initial steps correctly this should require a small amount of effort.

The organization of this software laboratory deserves some explanation. The ADS tutorial procedures that you will reference have been created for the Wireless Circuits and Systems Laboratory, and are intended to cover quite specific aspects of ADS. (The tutorials are available on the course website.) In order to get up the learning curve to the point where you can generate an equivalent circuit model (using circuit optimization) it is necessary to first understand some of these 'specific aspects of ADS'. Rather than combine ALL of the necessary material into one large document, you will instead have to refer to several documents during the course of this laboratory. However, the majority of the equivalent circuit modeling/optimization topics are discussed in only one procedure (#10).

You will need to refer to the following ADS procedures:

♦ *Procedure #1 ADS Basics*

♦ *Procedure #2 Simulation of Multiple DUTs Using ADS*

♦ *Procedure #3 Creating a Circuit Element Using Measured Data in ADS*

♦ *Procedure #6 Circuit Design Using Microstrip Lines in ADS*

♦ *Procedure #10 Circuit Optimization using ADS*

Comment: 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 the teaching assistant or course instructor to resolve any issues that arise.

Second comment: You may find it possible to obtain copies of the ADS projects and/or schematics from students who have taken the course in previous years. While using these files may save you some time now, you will likely lose time in the later laboratory assignments if you do not have a good grasp on the basics of creating these schematic setups.

Important: The procedures you will use to complete the assignment are generic, in the sense that certain printed circuit board substrate and component parameters may not apply to this specific laboratory. For this lab you are working with a 4.3 pF capacitor – any references, filenames, etc. to different values should be modified or interpreted accordingly. Similarly, for the inductor modeling, you will be working with a 16.0 nH inductor. Finally, please be sure to use the following:

Substrate Parameters:

H = 60 mils, relative dielectric constant = 3.7, loss tangent = 0.003, conductor thickness = 1.7 mils.

Microstrip Taper Dimensions:

Capacitor: L = 72 mils, W1 = 138 mils, W2 = 28 mils

Inductor: L = 72 mils, W1=138 mils, W2 = 27 mils

Laboratory Assignment

1) If you have not already done so, download the zip file from the course webpage that contains the ADS procedures. The filename is ADS-pdf-xxxxxx.zip (where xxxxxx represents the latest revision date). The file is found under the course documents section.

2) From the same web page, download the zip file entitled LC Data-xxxxxx.zip. From this zip files, extract the data files entitled CAP_035_R60_4p3.s2p and IND_051_R60_16p0.s2p. (That's "4 point 3" picoFarads and "16 point 0" nanoHenries).

3) **If you have never used ADS, or have forgotten some of the basics**, complete the steps in ADS Procedure 1 (p01-xxxxxx) that will provide an introduction to the basics of ADS. In completing this procedure, you will create a project directory (that you can use for the remainder of the course) and build and simulate a simple lumped element schematic. It is recommended that you save the project directory to a hard drive or a thumb drive. (Do not try to use a floppy disk!)

4) **If you have never used ADS, or have forgotten some of the basics,** complete the steps in ADS Procedure 2 (p02-xxxxxx) that will explain one approach to simulating more than one circuit/device (device under test, or DUT) simultaneously.

5) At this point, you should have created a project directory. (If you did not complete ADS Procedures 1 and 2 above, create a project directory now.) Copy (or move) the two data files that you extracted from the zip files into the "data" subdirectory of your project. You must have an unzipped version of the files in the directory, and the filename extensions must be .s2p.

6) With these preliminary steps completed, you are ready to begin the main procedure on equivalent circuit modeling. This process is discussed in detail in ADS Procedure 10 (p10-xxxxxx) which is entitled *Circuit Optimization using ADS*.

   a) The first step of Procedure 10 instructs you to create a circuit schematic that references an .s2p data file. **This process is described in detail in ADS Procedure 3** (p03-xxxxxx). You can follow the steps in Procedure 3 almost exactly, but use the schematic file names discussed in Procedure 10. Note that the schematic you create must have Ports and not Terminations. **For this schematic you should reference the** CAP_035_R60_4p3.s2p **data file.**

   b) The second step of Procedure 10 instructs you to create the equivalent circuit model. In order to do this you will follow essentially the same steps as those in Procedure 1. The only difference is that you need microstrip elements in addition to lumped elements. **The use of microstrip elements is described in the first pages of Procedure 6** (p06-xxxxxx). Note that the schematic you create must have Ports and not Terminations.

   c) Follow the remaining steps in Procedure 10 to create an equivalent circuit model that matches the measured S-parameter data for the capacitor. Run the simulations from 0.05 – 6 GHz.

7) Include the following in your laboratory report:

   a) Printout of your equivalent circuit schematic. Make sure that the optimized parameter (variable) values are clearly legible.

   b) Printout of a graph comparing the measured S-parameters to those of the equivalent circuit model (should resemble Figure 7 in Procedure 10). Unless you annotate the graph electronically somehow, you will most likely have to label the curves in pen to indicate which curve corresponds to each parameter (since the colored curves are not very useful in black and white printouts).

8) Finally, repeat steps 6 and 7 to develop an equivalent circuit model for an **inductor** that will match the S-parameter data in the IND_051_R60_16p0.s2p. The topology for the inductor equivalent circuit that you should use is shown in Figure 1. Notice that the topology (and variable names and corresponding elements) are the same as with the chip capacitor model, except that the series capacitor C1 has been removed. Run the simulations from 0.05 – 6 GHz.
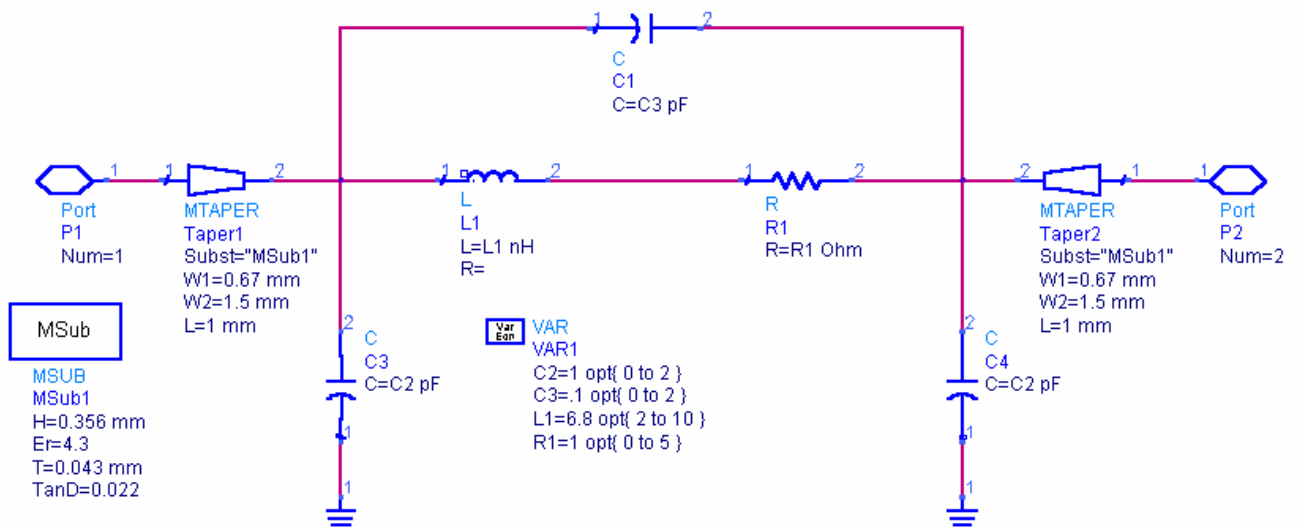
Figure 1. Typical schematic representing the equivalent circuit ("model circuit") of a chip inductor. Note that the substrate parameters, taper dimensions, and parameter values may be different for your laboratory experiment.