Appendix B

MP 0 - Agilent ADS Tutorial: Simulate an RLC Network

B.1 Introduction

This tutorial is designed to familiarize you with Agilent's Advanced Design System (ADS). It is arranged and categorized so that it may be used easily as reference material. Some parts of this reference were taken directly from the on-line User's Guide.

The following conventions are used in this tutorial. Specific buttons in the window environment will be **bold-faced**. Anytime that you are asked to type something, the content will be *italicized*.

B.2 Working with ADS

B.2.1 The Design Environment

The design environment is made up of the **Main** window, the **Schematic** window, the **Data Display** window, and the **Message/Status** window.

From the **Main** window you can:

- Create and manage projects and designs
- Set program preferences
- Change tool-bar configuration and keyboard shortcuts
- Change the type of components loaded on start-up
- Launch the text editor
- Open a data display window

From the **Schematic** window you can:

- Create and edit schematics
- Create variables and equations
- Simulate and optimize
- Open a data display window
- Generate a layout from the schematic

B.2.2 Starting ADS

- If you have not already done so, login into one of the PCs in the lab.
- Create a directory for this lab (e.g. *ECE453\Lab1*)
- Double-click the **Advanced Design System** icon or select it from the start menu. Close the wizard (grey window with buttons) to conform to instructions below). The main ADS window remains.
- Choose **File** > **New Project**, and the **New Project** dialog box appears. In the **New Project** dialog box, navigate to the directory you just created in your account on the network drive *w:* using the **Browse** button.
- type *\rlc_model_prj* in the **Name** field after the path and press **OK**. The **Schematic** window will appear at this point.
- In the **Schematic** window, choose **File** > **New**. In the dialog box, type *rlc_model_sch* as the name for the new design and then click **OK**.
- Save your new design.

B.2.3 Creating a Schematic

Using the **Component Palette** on the left of the **Schematic** window is the easiest way to create a schematic.

- Placing Components
  - Locate and click to select the desired component from the component palette.
  - Move the pointer into the drawing area and click to place component in the desired location. Note the cross point is the location of Pin 1 of the chosen component (Pin 1 is marked with a \ - slash).
  - Pick and drag components with a mouse. Rotate using right mouse button or 'Ctrl-R'.
  - The unconnected pins of a placed component can be identified by the red, diamond-shaped outline around the pins. Once connected, the outline disappears.
- Editing Component Values
  - Once a component is placed in the drawing area, double-click on it to display the dialog box for editing.

  - In the editing dialog box, select the parameter you want to change, then change all fields and the parameter value as needed, and click **OK**.

- Connecting Components
  - click the **Insert Wire** button.
  - Position the pointer on the pin at one end to be connected and click.
  - Position the pointer on the other end and click. A wire (a purple line) should show between the specified points.

- Moving Components
  - Position the pointer over the component you want to move.
  - Press the left mouse button, drag the component to the new location, and release.

B.3 Designing the RLC Circuit Using ADS

So far we have learned the basics of using ADS. This section contains the step-by-step approach for simulating the impedance of an RLC circuit.

B.3.1 Creating a Circuit to be Simulated

- In the **Schematic** window, choose **File** > **Open**. Highlight *rlc_model_sch*, then click **OK**
- Change the palette group to **Sources-Freq Domain**. Click **V_1Tone**, and then place the voltage source on the schematic. Double-click on the source to bring up the dialog box. Select parameter **Freq=1**, and change the unit of frequency to **None**. click **OK** to close the dialog box.
- Change the palette group from **Sources-Freq Domain** to **Simulation-AC**. Click on **AC**. Place this simulation component on the schematic. Double-click on this component to bring up the editing dialog box. Select the **Frequency** tab, change the options to:

  Sweep Type = Linear

  Start = 1 MHz

  Stop = 100 MHz

  Num. of pts. = 100

  click **OK** to close the dialog box.
- Change the palette group back to **Lumped-Components**, and finish wiring the circuit as shown in Figure B.1 shows an example circuit.
- Name the node on the top of the AC source *SRC1* as *Vin*. To do so, click on **Name Node** (second row, sixth button from right), type *Vin* in the dialog box, and then click the wire connecting *SRC1* and *R1*. Click **Cancel** when finished.

B.3.2 Simulating the Circuit

Choose **Simulate** > **Simulate** to start simulation.

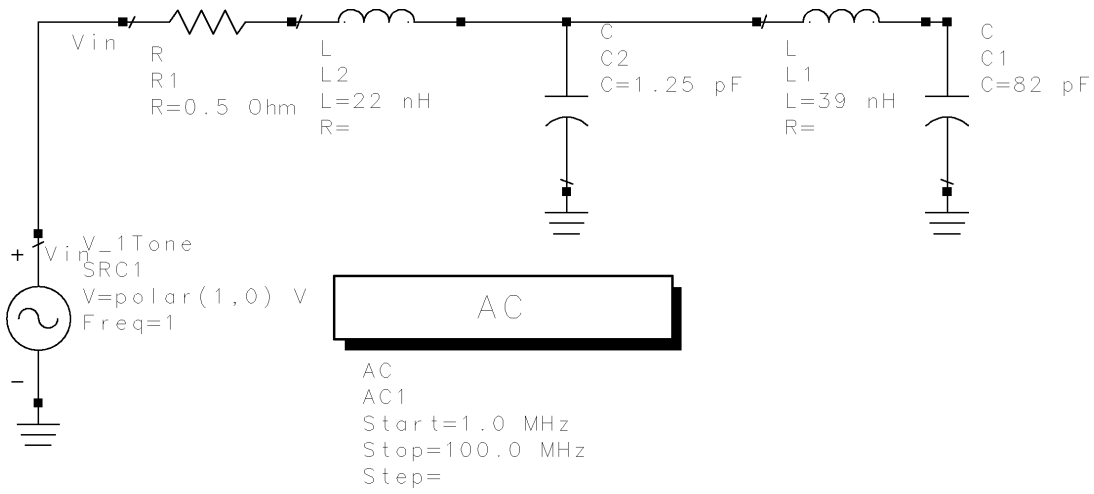

Figure B.1: A series LC circuit driven by a model for the Vector Impedance Meter probe consisting of an AC source and a network that models the series inductance/resistance and shunt capacitance of the probe.

B.3.3 Data Display

The data display window will come up after the simulation has completed.

- click **Eqn** in the menu on the left, and place it in the drawing area. Type *current=*, then highlight **SRC1.i**. click **Insert** and then **OK**.
- Repeat the last step to place the following set of equations in the drawing area:
  $current = rlc\_model..SRC1.i$
  $voltage = rlc\_model..Vin$
  $ReZ = real(voltage/(-current))$
  $ImZ = imag(voltage/(-current))$
  $MagZ = mag(voltage/(-current))$
  $PhsZ = phase(voltage/(-current))$
  Note: a quicker way is just to define complex Z, because ADS will give multiple display options when plotting complex data:
  $Z = rlc\_model..Vin/rlc\_model..SRC1.i$
- You can also add a reflection coefficient equation to plot on the Smith Chart:
  $Z = (voltage/(-current))$
  $gamma = (Z - 50)/(Z + 50)$
- click **Stacked Rectangular Plots** in the menu on the left and place it in the drawing area. In the dialog box, change **Datasets and Equations** to **Equations**.
  Note: in your lab you will need to display multiple traces on the same axis. In such a case its better to use a regular plot rather than stacked plots.

- Highlight *ReZ* and *ImZ* then click **ADD**. click **OK** to close the dialog box.
- Repeat the previous step to obtain plots for *MagZ* and *PhsZ*. Figure B.2 shows the results for the series LC circuit.
- Choose **Marker** > **New** and then click on one of the traces. Click on the marker and drag it to the desired position to display values.
- Choose **File** > **Save As**, and type *rlc_model_dis* to save the display.

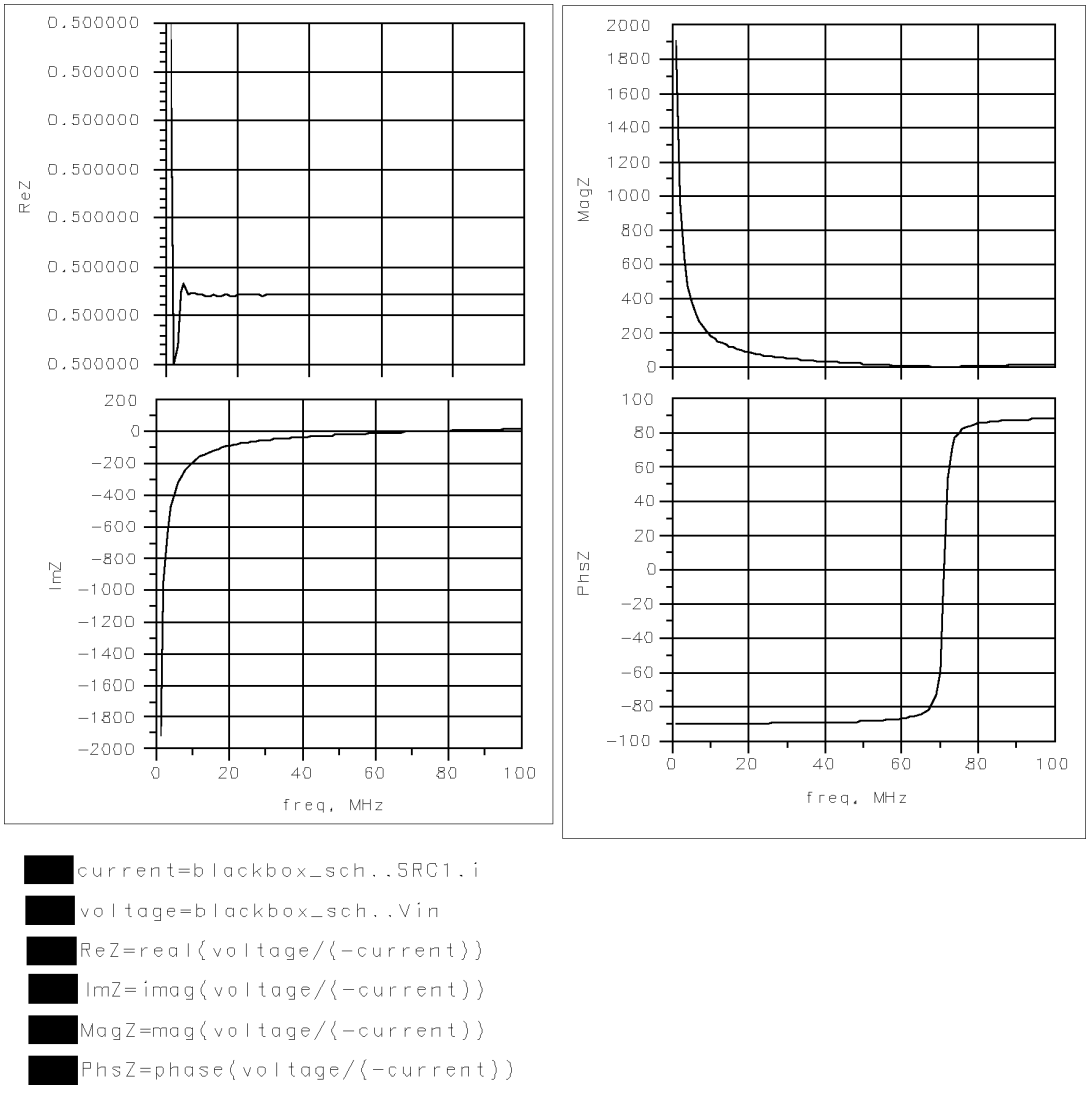

Figure B.2: Display page showing the equations that define the impedance as measured by the VIM. Also shown are plots of the real/imaginary and magnitude/phase of the measured impedance.

B.3.4 Plotting the Results

- In the **Data Display Window**, choose **File** > **Print**.
- You may wish to click **Properties** and change the **Orientation** to **Landscape**.
- click **OK** with the left mouse button.