ADS Fundamentals – 2009

LAB 1: Circuit Simulation Fundamentals

Overview - This lab covers user interface basics, ADS files, schematic capture, simulation, and data display. In addition, tuning and ADS example files are also covered.

OBJECTIVES

- Create a new project and schematic designs
- Setup and perform S-parameter simulation
- Display simulation data and save files
- Tune circuit parameters during simulation
- Use the Examples files and node names
- Perform a Harmonic Balance simulation
- Write an equation in the data display

Agilent Technologies

Table of Contents

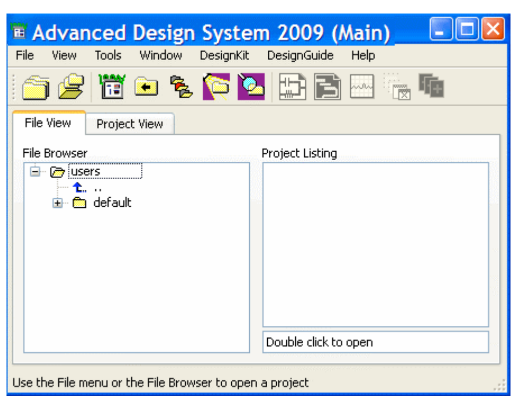
PROCEDURE

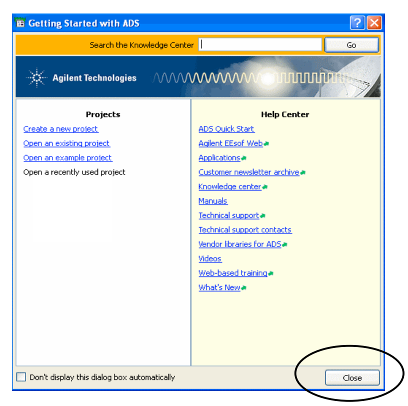
1. Start ADS on the computer.

a. For PCs: Click the shortcut icon for ADS if it appears on your screen, or use the Start > Programs command to find and start Advanced Designs System as shown here. For UNIX: type the script/ command at the terminal prompt (for example: hpads).

b. When the Main Window appears, you should also see the Getting Started dialog. If it appears, close it – you will learn how to do all of the things it asks and much more…if it does not appear, it has already been turned off. Also, do not be concerned about the File View – it will depend upon which directory start-up directory ADS is using.

NOTE: your File View may be different, depending upon where ADS is started…



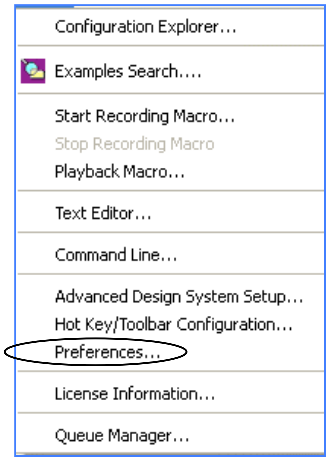


2. **Examine the Main Window preferences.**

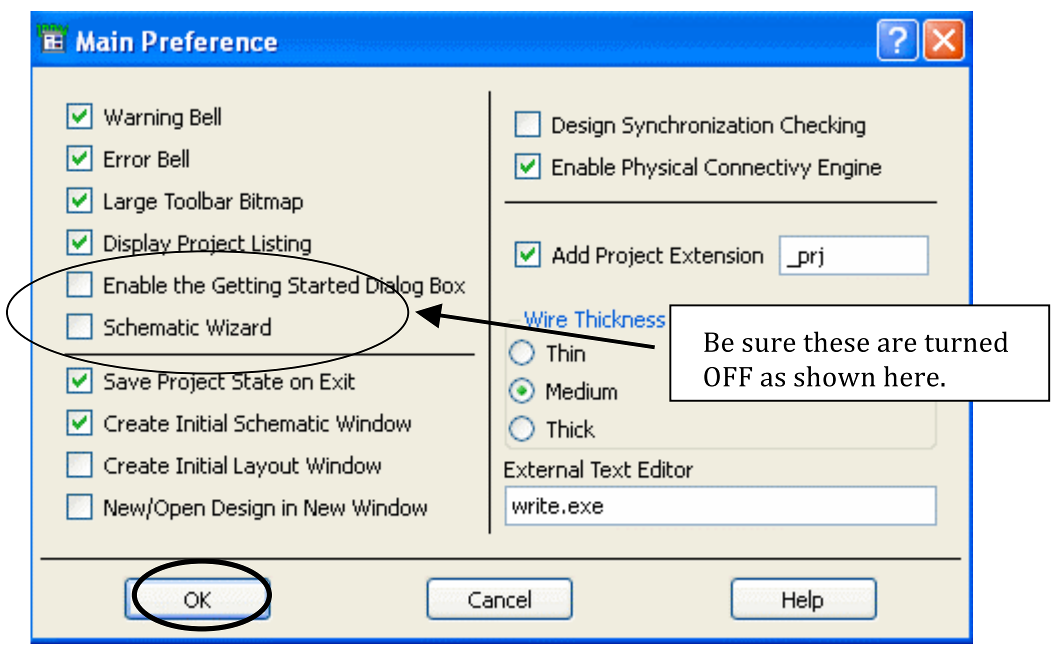
   a. In the Main Window, click **Tools > Preferences**.

   b. When the Main Preference dialog appears, be sure that there are **no checked boxes** for the display greeting or the schematic wizard. You will not need them in this course.

   c. Be sure the Large Toolbar Bitmap is checked (turned on) and leave the other settings as shown in their default condition.



Most ADS windows have a preference dialog so that you can set or customize the window as desired. You can try all of these after this course.

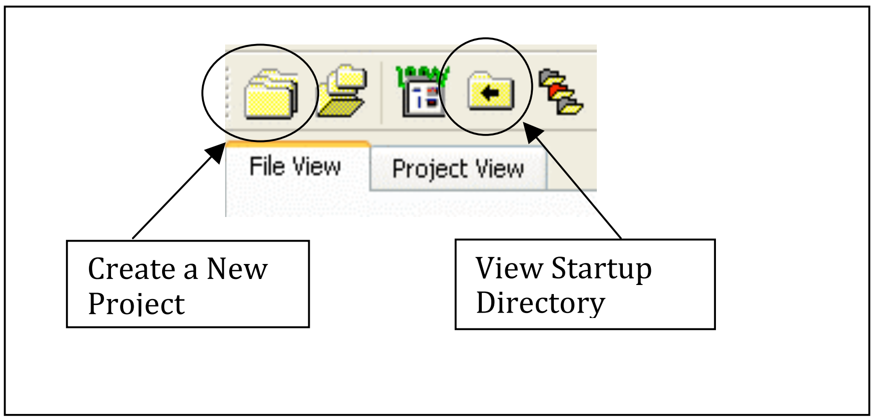


   d. Click **OK** to dismiss the dialog.

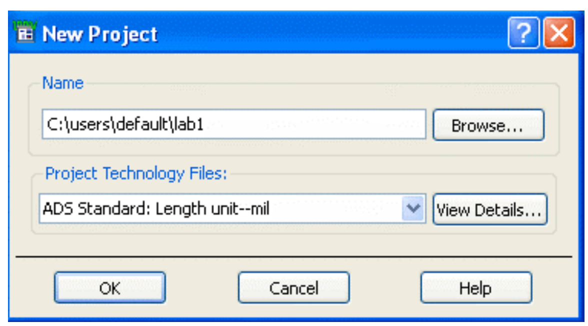
3. Create a new Project.

For this step you will use the icons on the Main window. Using icons to execute commands usually requires fewer mouse clicks. You can identify what an icon does by placing the cursor on the icon to display a text box: this is called *balloon help*.

a. In the Main window, select the File View tab and then click the icon: **View Startup Directory.** This will show that you are in the default starting directory for ADS projects.



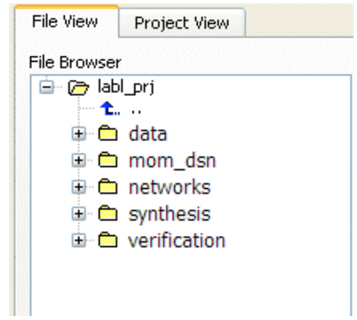
b. Click: **File > New Project or use the the icon**. When the dialog box appears, appears, you will see the default working working directory. On your own computer it will be C:\users\default but C:\users\default but in an ADS classroom it may be different (C:\users\ads) so check with the instructor. Insert the cursor at the end of end of the line and type the name: **lab1**. **lab1**.



Note on Project Technology Files - The ADS Standard Length unit is used in layout. If you had a Design Kit (foundry kit or PDK) installed, you could select it here from the drop-down list (arrow button). For this lab use the default value (mil) as shown.

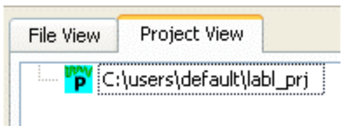
c. Click **OK** and the project is created and a schematic window opens, ready for you to create a design.

4. Examine the files in your new project directory.



a. Look at the Main window File View tab. It should now show now show all the files that are automatically built in the the **lab1** project directory. Notice that the sub-directories directories (data, networks, etc.) were also created automatically, ready for you to create designs, simulate and simulate and plot data.

b. Look in the Project View tab and notice that the project is project is empty at this time – no schematics or data

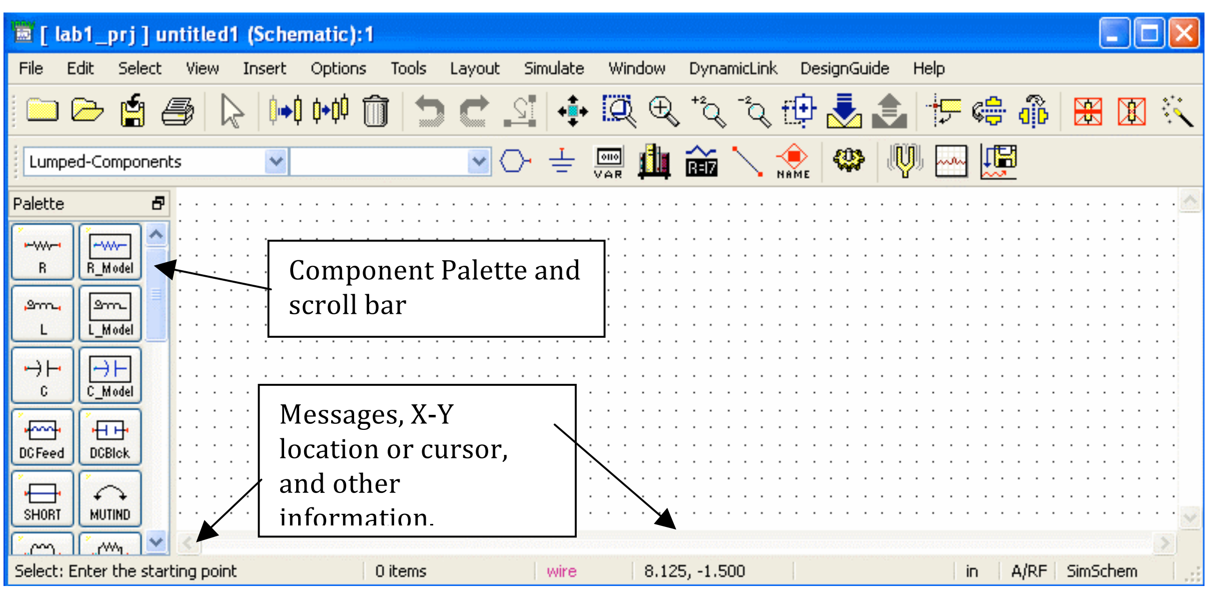


displays yet.

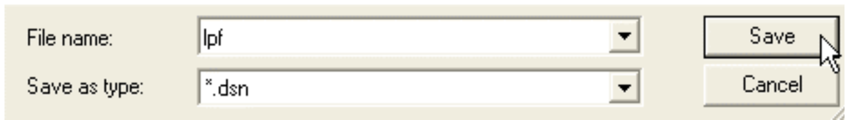
c. Also, notice that the schematic, layout, and data display icons are now activated (not gray). This means you can now open those windows which you will do in the next steps.

5. Create a low-pass filter design.

a. In the Main window, click the New Schematic Window icon (shown here). This is the same as selecting the menu command: **Window > New Schematic Window**. Immediately, the Schematic window will appear. If your preferences are set to create an initial schematic, you will have two schematics now opened – close either one of them.

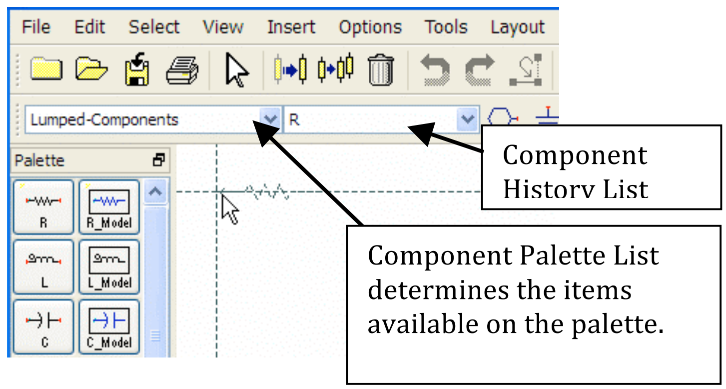


b. **Save the schematic.** Notice the top window border of the schematic shows the schematic name as *untitled*. Click the icon (shown here) and the *Save Design As* dialog will appear. Type in the name **lpf** and click **Save**. This will save it in the *networks* directory of *lab1* project.



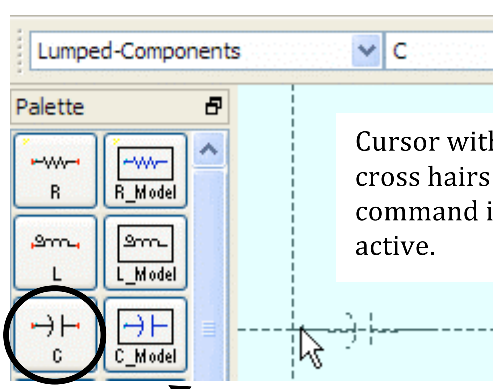
NOTE on saving designs - After naming the schematic, the *Save* icon will not bring up this same dialog box. Instead, it will save the named design. To save the design with a different name, use the command: *File > Save Design As*.

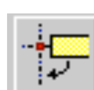
c. **Examine the schematic window commands and icons. and icons.** Click the small small arrow on the Component Component Palette list to see to see the palette choices. Also, choices. Also, move the Scroll Scroll Bar down and up to see to see how it works.



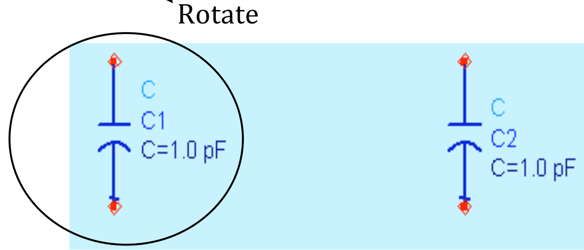
d. In the **Lumped Components** palette, select (click) the **capacitor C** shown here (not the C model). Then click the **Rotate By Increment** icon as needed for the correct orientation and then click to insert the capacitor as shown on the schematic. Next, insert another capacitor.

NOTE on schematics: you can change the color of the schematic background, grid dots, and more using Options > Preferences. This will be covered later in the course.



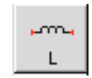


Rotate

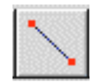


NOTE: some boxed icons (R, L and C) are models – not components.

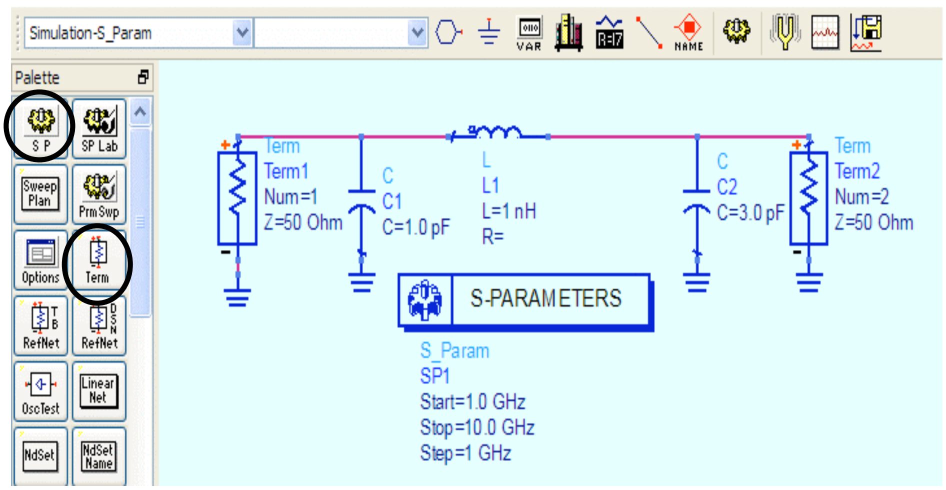
e. Continue creating the low-pass filter as shown by inserting the **inductor** and **grounds** (icons are shown here). Then **wire** the components together. This will give you practice with schematic capture. You can try using the copy, move and other icons or commands.



f. After the filter is built, edit the value of **C2** to be **3** pico-farads. To do this, double click the capacitor symbol or select the capacitor and use the icon (R=17 shown here). When the dialog box appears – change the value: **C=3.0 pF**, click **Apply** and **OK**.

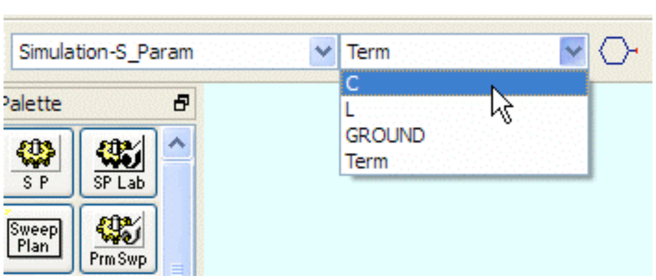


g. Next, select the **Simulation- S_Param** palette and insert the **S-parameter** simulation controller (gear icon). Use the ESC key to end the command.



h. Insert the port terminations: **Term** Num= 1 and **Term** num=2.

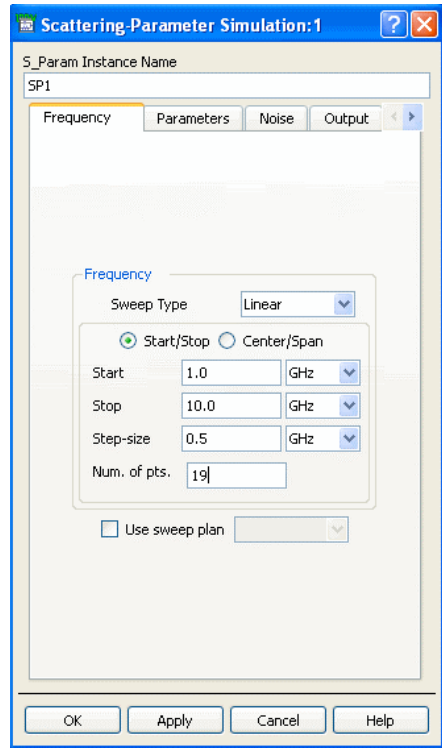
i. **Use Component History**: After the circuit is built, delete capacitor C2 and then reinsert it by typing or selecting (history) the capital letter **C** in the Component History field and press Enter. Next, edit the value directly on the schematic by highlighting the value and typing over it with the value (3.0 pF). Verify that it has changed by looking at the value in the edit dialog box.



COMPONENT HISTORY: You can insert components if they have been inserted previously, or by typing in the component name (C, L,

6. Setup the S-Parameter Simulation.

a. To setup the simulation, double click on the S-
S-parameter simulation controller on the schematic.
schematic. When the dialog box appears, change the
change the **Step-size to 0.5 GHz** and click **Apply**.
Apply. Notice how it updates the value on the
the screen. The *OK* button does the same thing as
thing as Apply and also dismisses the dialog box –
box – **do not** click *OK* yet.



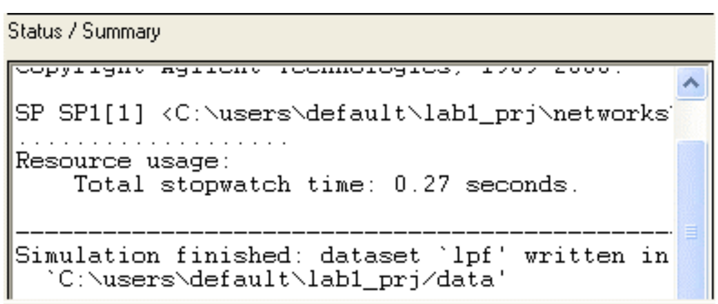
b. Click the **Display** tab and you will see that the Start,
the Start, Stop and Step values have been checked
checked (by default) to be displayed on the
schematic. Later in this course, you will use the
the display tab to check other parameters you want
you want displayed on the schematic.

c. Click the **OK** button to dismiss the dialog box. You
box. You are now ready to simulate.

7. Launch the simulation and display the data.

a. At the top of the schematic window, click the **Simulate** icon gear (shown here) to start the simulation process.

b. Next, look for the **Status window** to appear and you should see messages similar to the ones here, describing the results of the simulation, the writing of the dataset file, and the creation of a display window. If not, ask the instructor for help.

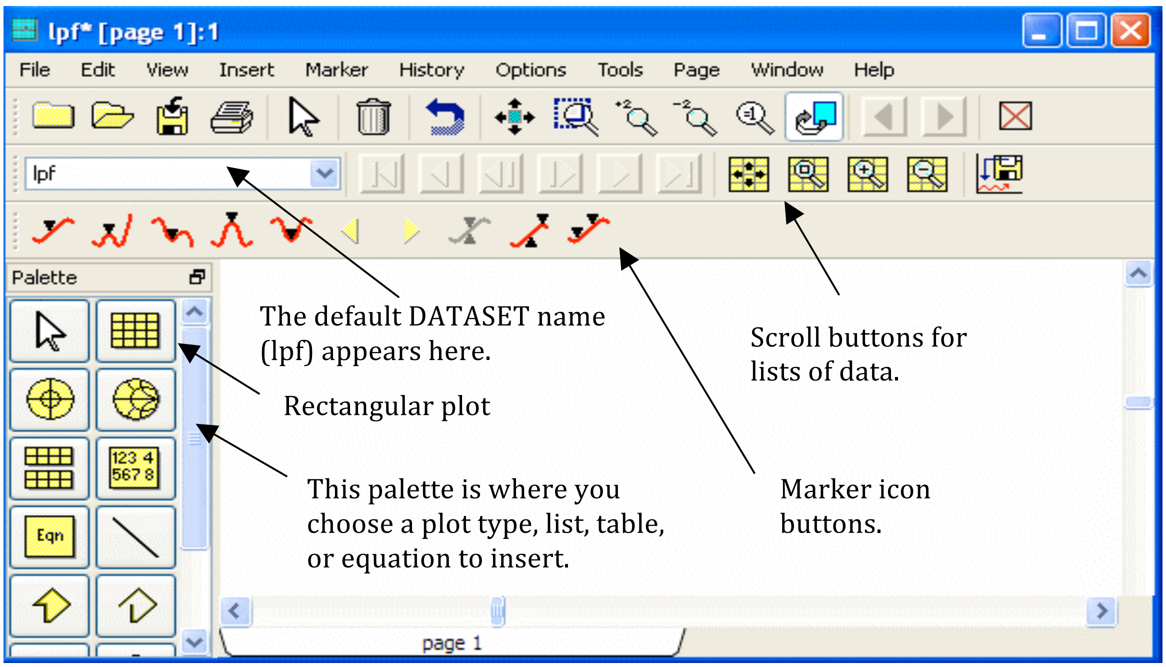


NOTE: If you scroll up, you will see more simulation information.

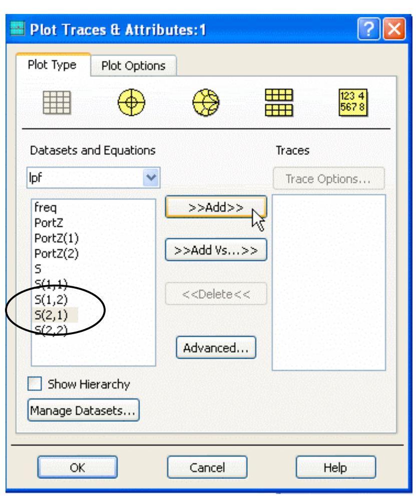
If no simulation errors occurred, close the Status window. You can always recall the status window using the schematic window command: **Window > Simulation Status** (try it).

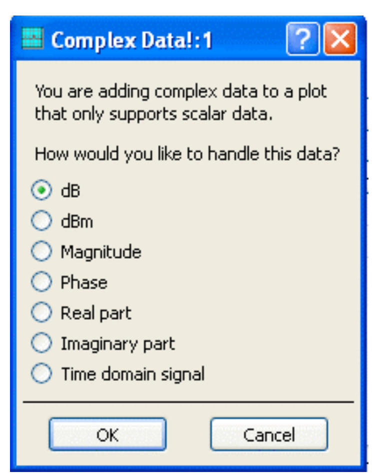
c. The Data Display window will appear with the name **lpf** in the top left corner – this is the same name as your schematic. Also, you are looking at page 1 which is blank at this time. Examine the picture below - the next steps will show how to display the simulation data.

NOTE: An asterisk next to a window name (lpf*) means it is not saved.



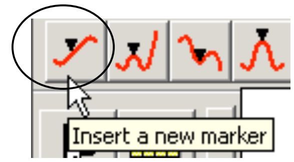
d. To create the plot, click on the **Rectangular Plot** icon and move the cursor (outlined box) into the window and click. When the next dialog box appears, select the **S(2,1)** data and click the **Add** button. Then select **dB** as the format for the data. Click **OK** in both boxes.

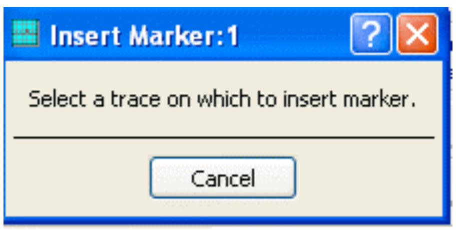


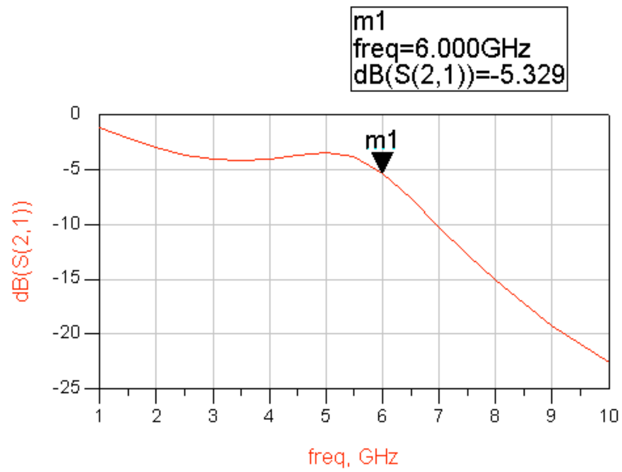


The plot should show a reasonable low pass filter response. Also, if you have a mouse wheel – try using it to zoom in and out.

e. Put a marker on the trace: Click new marker icon on icon on the toolbar (shown here). You will be prompted to select a trace to insert the marker. Next, Next, try using the other marker icons. You can also also move it using the cursor or the keyboard arrow arrow keys. Try deleting the marker or putting another marker on the



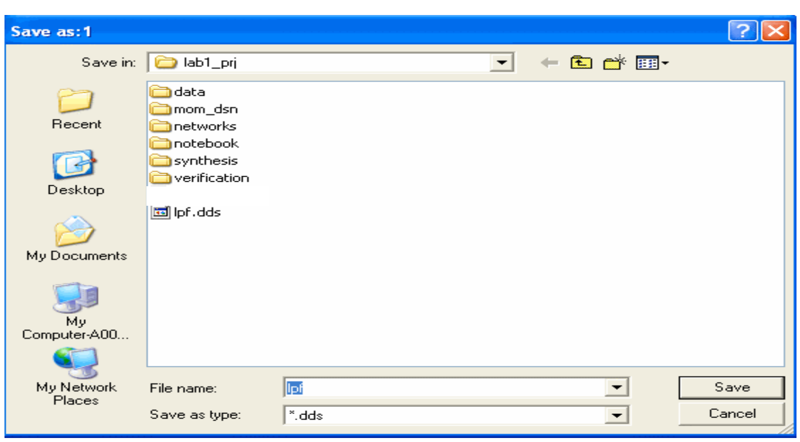




trace.

8. Save the Data Display Window.

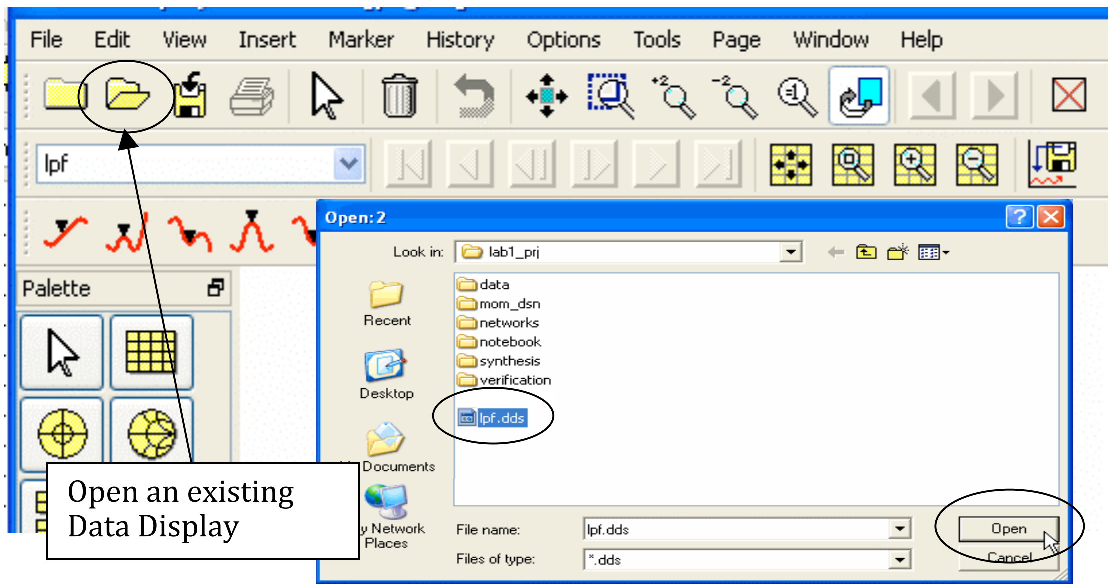
a. Save this data display window: Use the **File > Save As** command and use the next dialog box to save it with the default name **lpf**. This means that it will be saved as a **.dds** file (data display server) in the project directory and it will have access to all data (.ds files or datasets) in the *data* directory. This step shows you that data display windows are saved in the project directory and not in the data directory. Only data (datasets) are stored in the data directory.



b. Close the data display window using: **File > Close**.

c. Re-open the saved data display by clicking the **Data Display icon** (shown here) from the Schematic or Main window. After the window opens, click the Open icon folder (shown here). Select **lpf.dds** in the dialog and click **Open**. It will reappear with your S21 plot. Also, notice that the default dataset name (lpf) remains from your previous simulation. KEEP THIS WINDOW OPEN for the next steps.

Data Display icon:



9. Tune the filter circuit.

This step introduces the ADS tuning feature that allows you to tune parameter values of components and see the simulation results in the data display. In this step, you first select the components and then select the tuning feature. If you select the tuning feature first, you must select the component parameters and not the components.

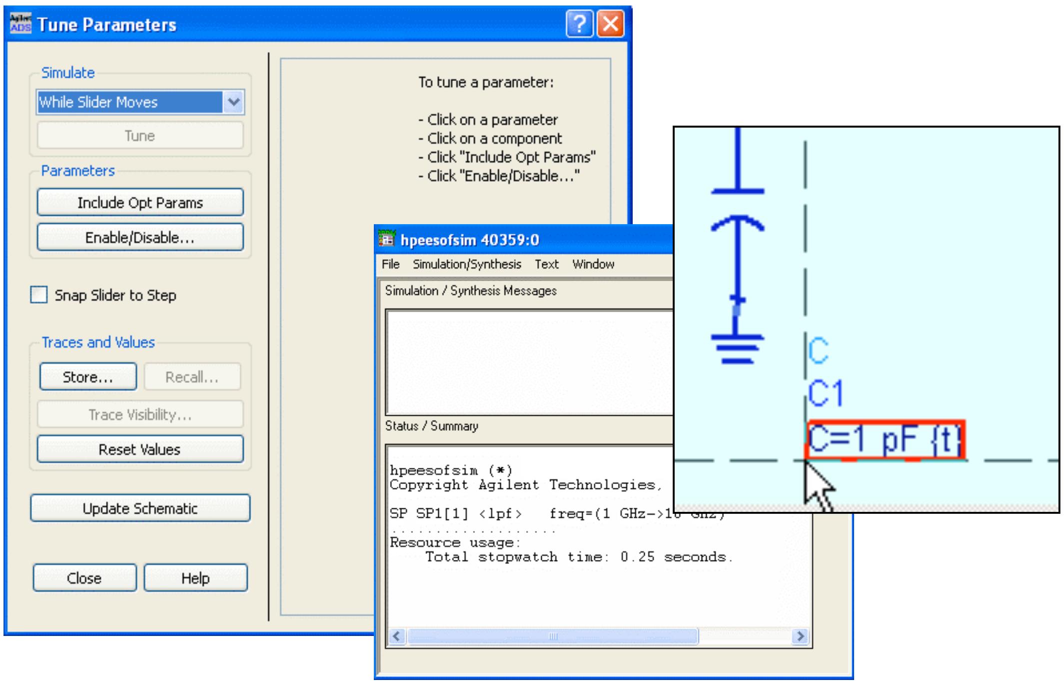
a. Position the <u>Data Display</u> and the **Schematic** windows so you can see them both on the screen. If necessary, re-size the windows and use **View All**.

View All

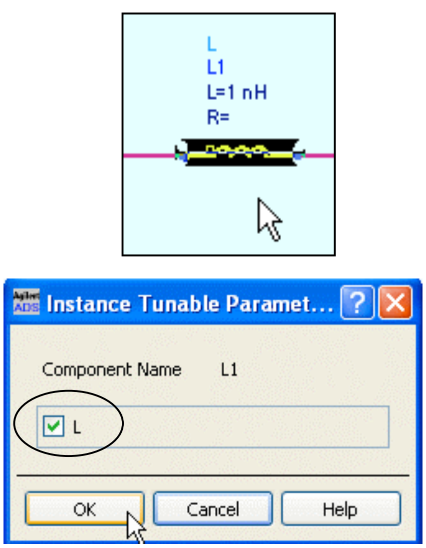
b. Now, start the tuner by clicking the command: **Simulate > Tuning** or click the Tune Parameters icon (shown here).

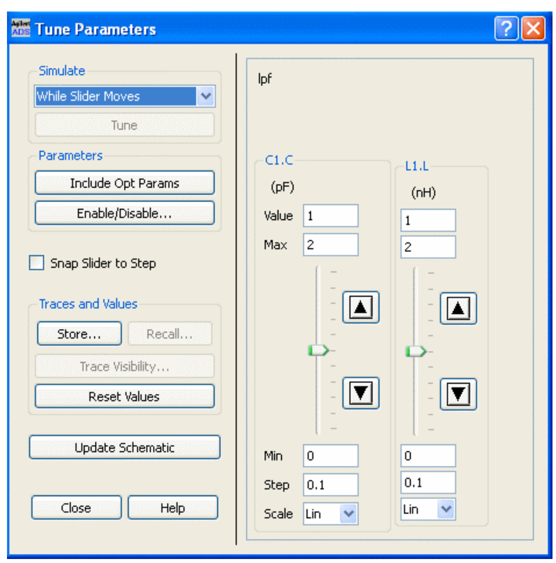
Tune Parameters

c. Immediately, the status (simulation) window will appear along with the Tune Control dialog box (shown here). Go ahead and click on the **C** parameter for the C1 capacitor as shown here. When you do, the tunable parameter will appear in the Tune controller and the parameter will appear with {t} to show that it is enabled for tuning. Another way to select the parameter is next.

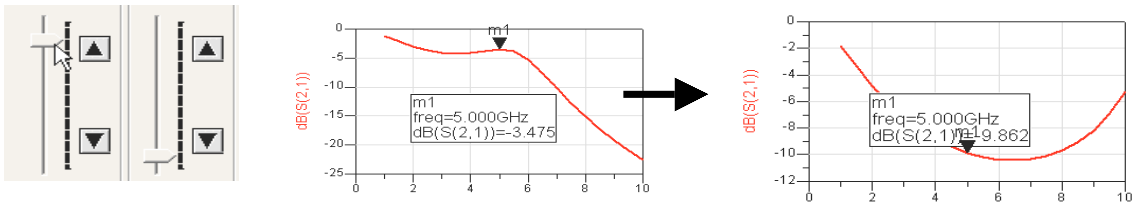


d. Click on the inductor symbol and when the small dialog appears, click on the L and OK. This will add the inductor in the tune controller.

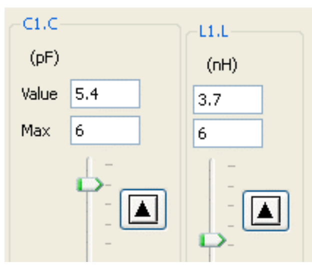


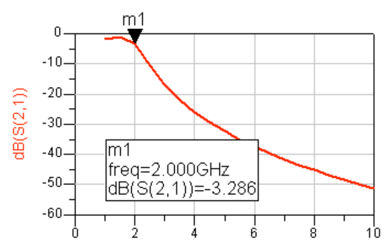


e. Arrange you Data Display window with the S-21 plot so that you can see it along with the Tuner. Place a marker on the 5 GHz data point as shown here. Then try moving the tuner sliders and see how the trace is automatically updated as if you were tuning with an instrument. You can also try typing in the value or change the way the slider is used.

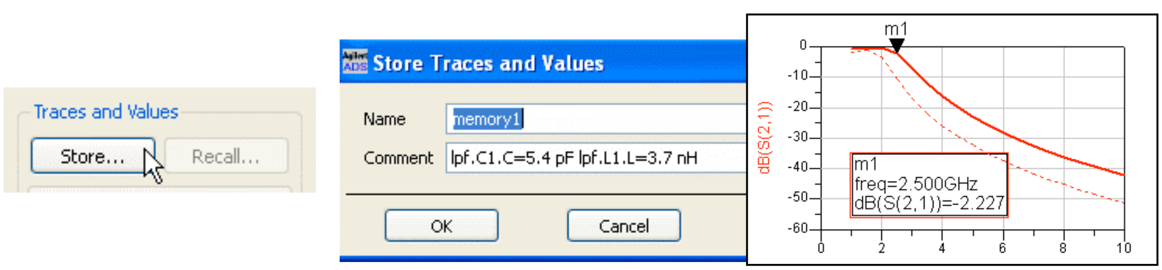


f. Increase the **Max** values to **6** for both parameters and continue tuning to get a typical low-pass filter response: about -3 dB near 2 GHz, as shown here - move the marker to 2 GHz to see this.

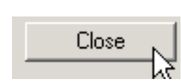


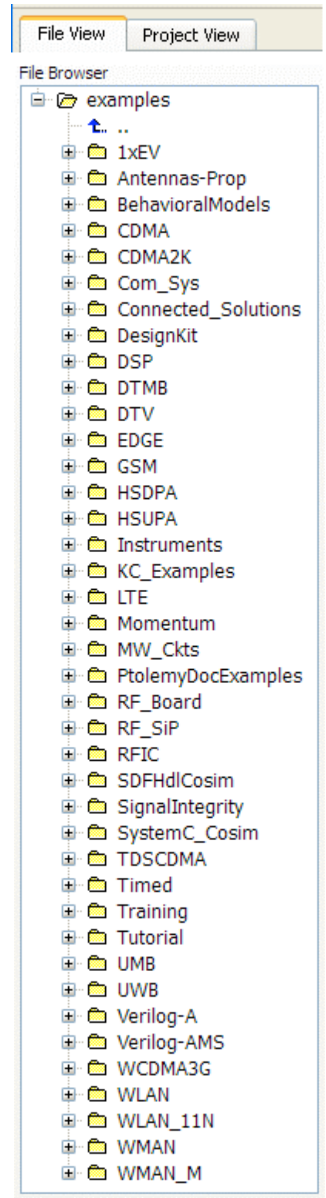


g. Store the trace by clicking on the **Store** button and **OK** as shown here. Then move the tuning sliders again - the stored trace remains (dashed line) while the new trace responds to the tuning. Traces can be stored / recalled and made visible as needed. Go ahead and try these and others to better understand how the tuner works.



h. Use the **Close** button on the tuner to close it. Then **Save** the data display and the schematic using the **Save** icon (in both windows) shown here. Finally, close all the windows, except the ADS Main window. Next, you will use Harmonic Balance with an ADS example.





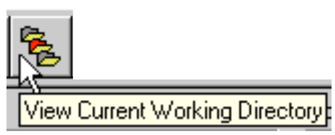
IMPORTANT NOTE: Using ADS Examples

All of the examples shipped with ADS can be examined in the

the Examples directory. However, to use them for your design design work, you must copy the files into another directory. In In general, the examples directory is read–only and the files must must remain unchanged. The following steps will give you some some experience leveraging examples for your own use.

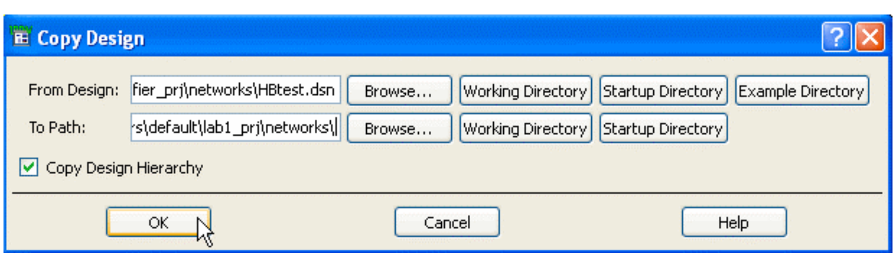
10. Copy an RFIC Harmonic Balance example.

a. Go to the ADS Main window, File View tab, and click on the **View Example Directory** icon to see the list of example topics but do not go any further, simply look at the choices. Afterward, click on the View Current Working Directory icon to see that you are still in the lab1 project.



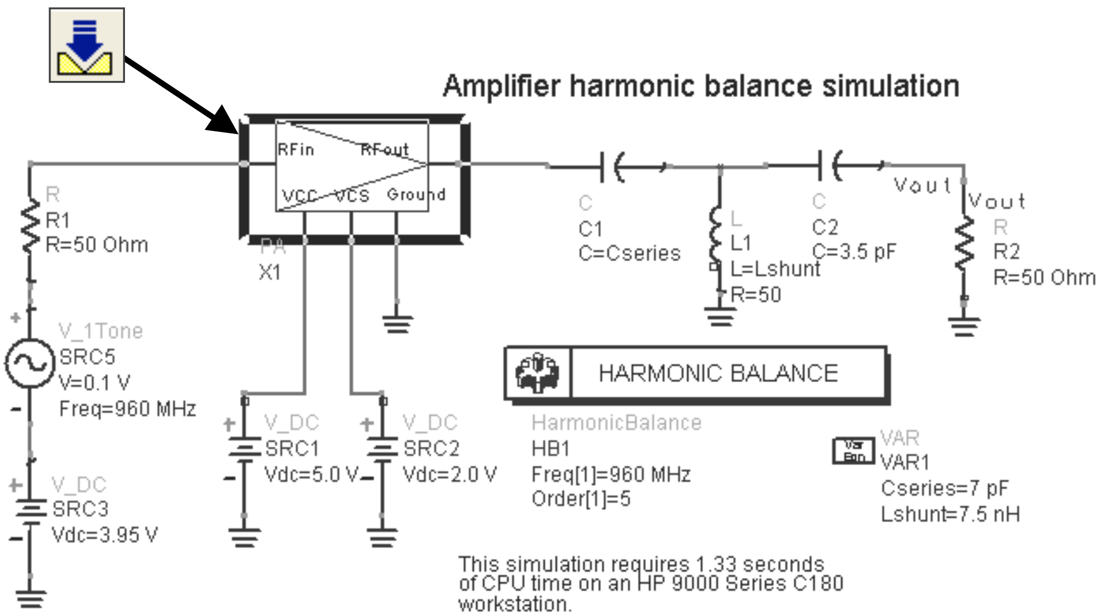
b. You are going to copy a schematic design from one of the example directories into the lab1 project (networks). In the ADS Main window, click **File > Copy Design** and the Copy Design dialog will appear.

c. Select: **From Design**: This is where you get the example design. When the dialog appears, select **Example Directory** and **Browse**. Then use the dialog boxes, double clicking on **RFIC > amplifier_prj > networks > HBtest.dsn** and Open.

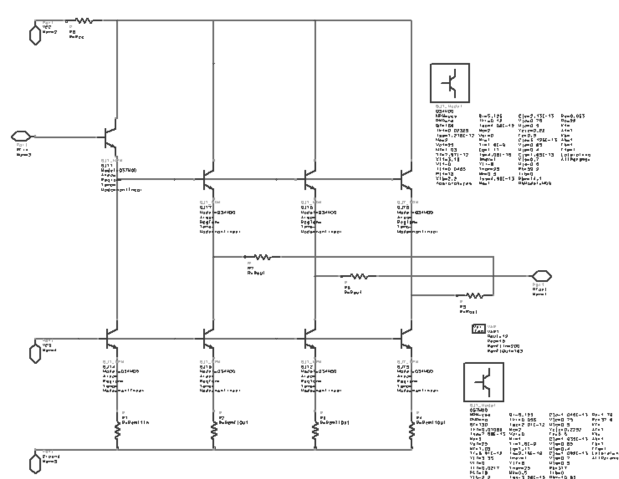


d. Specify the **To Path**: Select **Working Directory**, which should be the **networks** directory of the lab_1 project (shown here). Also check the box: **Copy Design Hierarchy**. Click **OK** and a copy of the HBtest and its hierarchy (sub-circuits) will be copied into your lab1_project.

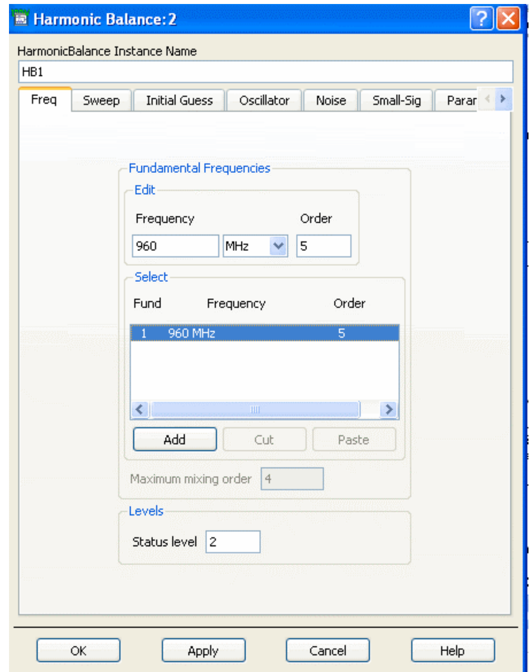
e. After the copy is complete, open a **schematic window** and use the icon or File >Open Design to open the **HBtest.dsn**. As shown here, this is the top-level hierarchy of the HBtest.dsn. This is where the Harmonic Balance simulation is set up. To see the amplifier sub-circuit click on the amplifier symbol and then click the icon: **Push into Hierarchy**.

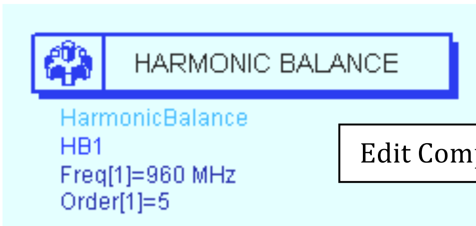


f. Notice that the sub-circuit has several biased devices with the model description (model card) shown. Click the **Pop Out of Hierarchy** to go back to the top level where the simulation is set up.



g. After you return to the upper level, examine the **Harmonic Balance controller** by double clicking on it or by selecting it and clicking the edit icon (shown here).





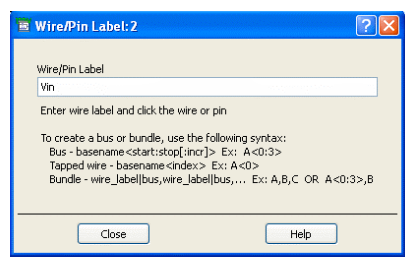
Edit Component

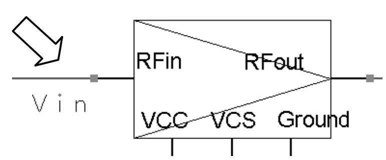
h. The Harmonic Balance controller has many tabs for setting up simulation parameters. The purpose of this step is to get you acquainted with the simulation controller and not to use all the settings. Look through the tabs, do not make any changes, and **Cancel** when you are done.

The simulator is set to calculate 960 MHz with 5 harmonics (order). Notice that the V_1Tone source (schematic) frequency is also 960 MHz.

11. Add a wire label (node name) and simulate.

a. Click on the **Name** icon (shown here). When the dialog appears, type in the name **Vin** and click on the wire or node at the input to the amplifier. Click Close when finished. The schematic now has a Vin and a Vout wire label.



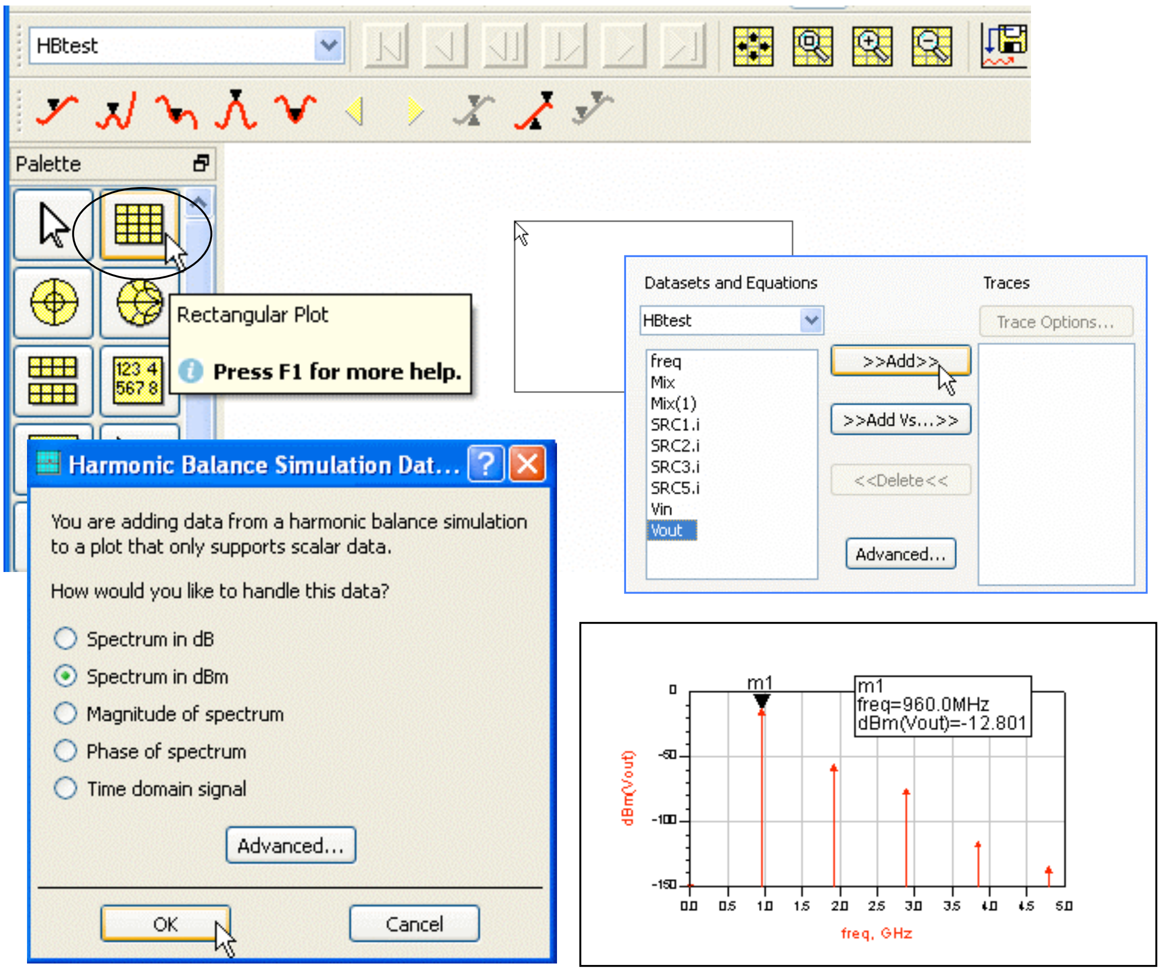


b. Click the **Simulate** button. When the simulation finishes, the node voltages at Vin and Vout will be available in the Data Display.

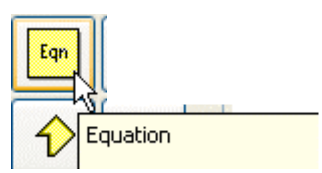
12. Plot the spectrum of Vout in dBm.

a. When the data display window opens, select the Rectangular Plot and insert it. Immediately, another dialog will appear where you select **Vout** and click **Add**. Next, the dialog will ask you for the format: **Spectrum in dBm**. Click **OK** and **OK** again, and the plot will appear.

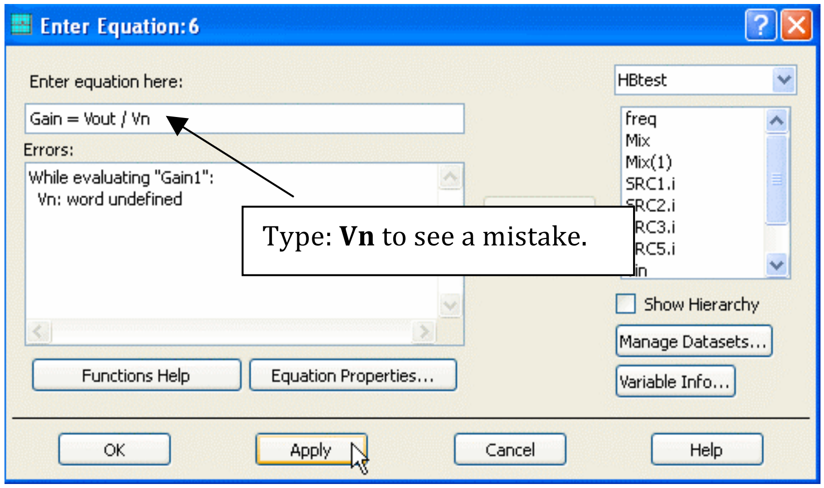
b. Put a **marker** on the first tone to verify that it is 960 MHz.



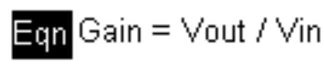
c. Insert an equation by selecting the Eqn icon and inserting it on the data display. Immediately, another dialog will appear.



d. Write the equation in the field with a mistake in spelling the node name. For example, write a voltage gain equation as: **Gain = Vout/Vn**. Click Apply and you will see how the error is recognized.

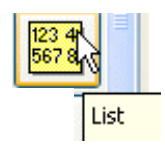


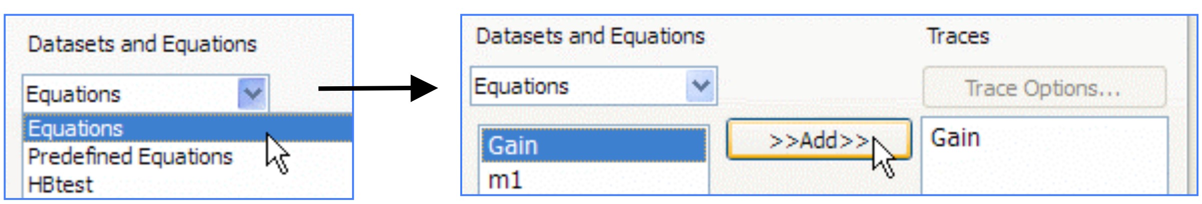
e. Correct the spelling to Vin, click **Apply** again and **OK**. The correct equation will appear.



NOTE on data display equations: If the equation is red in color, then it is invalid.

f. To list the equation value of Gain, insert a list by selecting the **List** the **List** icon. When the dialog appears, click the arrow and scroll scroll down to the Equations list. When it appears select the Gain the Gain equation and Add it and click OK.



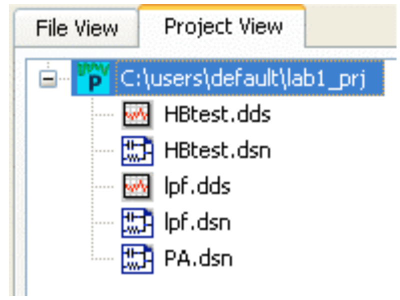


You should see a list of Gain: the complex voltage for each frequency calculated by your equation and the Harmonic Balance data. Actually, this circuit really has more current gain than voltage gain.

freq	Gain
0.0000 Hz	0.000 / 0.000
960.0MHz	0.693 / 60.028
1.920GHz	2.489 / 21.067
2.880GHz	5.303 / 5.198
3.840GHz	1.461 / -159.389
4.800GHz	3.615 / -71.992

g. Close all the windows – **do not save** the files in files in this lab. You will not need this lab or any or any of its files to continue.

13. Examine the Main Window again



a. Notice that the Project View tab in the main main window now shows your saved designs and designs and data displays.

b. Any design or data display can be opened by by double clicking on any of them – try it and and then close them using the Main Window Window command: File > Close All.

<u>END OF LAB EXERCISE</u> – if you have time, try the Extra Exercises.

EXTRA EXERCISES: do these only if you have additional time. Otherwise, you can do them for practice after completing the course.

1. Try searching for an example of **acpr** using the example search icon.

2. Go back to the HBtest simulation and increase the Order in the Harmonic Balance simulator to 7. Simulate again and note what happens. Also, change the value of Freq to 940 and note what happens – you can learn about simulation errors from this type of exercise.

3. Write another equation for the HBtest results:

   Gain_fund = Vout[1] / Vin[1]

   Determine what the bracketed value of [1] is doing in the equation.

4. Go back to the LPF design and open the data display page. Try writing an equation for the phase of S21 and then plot it:

   Eqn lpf_phase=phase(S(2,1))

5. Examine any other examples that are interesting to you.