

This chapter covers the user interface basics for file handling, schematic capture, simulation, and data display. In addition, tuning and the use of ADS example files is also covered.

Lab 1: Basics of using ADS

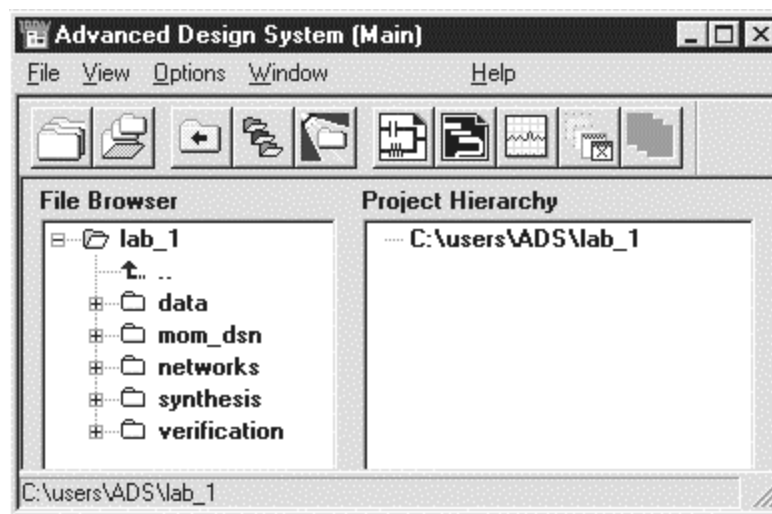
OBJECTIVES

- Examine the Main window commands and icons
- Create a new project and schematic design
- Setup and perform an S-parameter simulation
- Display the simulation data on a plot and save files
- Tune the circuit to refine the response
- Look through the Examples and do a Harmonic Balance simulation

PROCEDURE

1. Start the system (instructor will give you instructions)

- a. Typically, on a PC, you will use standard method for starting a program or on UNIX, you would type: hpads.



Main window (PC version)

NOTE on Interface Differences between UNIX and PC:: The user interface for the PC and UNIX are the same. The only difference is the appearance and some minor features: For example, UNIX has tear-off menus; the PC version has a Toolbar that can be detached from the window. Otherwise, all the functions and commands are the same for both platforms.

2. Examine the Main Window

- a. Click the **File** command.

These commands are for controlling and handling *projects* (directories) and *designs* (files) which are schematics and layouts.

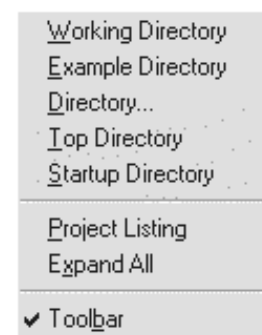
Click on any command with ellipses (...) and examine the dialog. Then click the Cancel button as necessary to return.

This step is only to show you the menus. Later on, you will be using these commands which are superior to using UNIX commands or PC file managers for ADS.



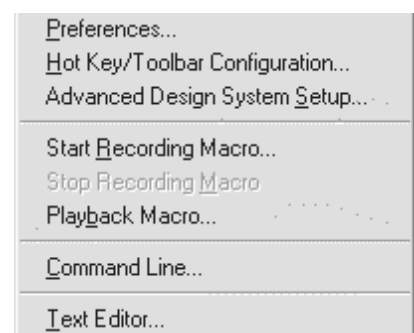
- b. Click the **View** command.

These commands are specifically used for changing and viewing directories. Click on any of the commands to see how they work.

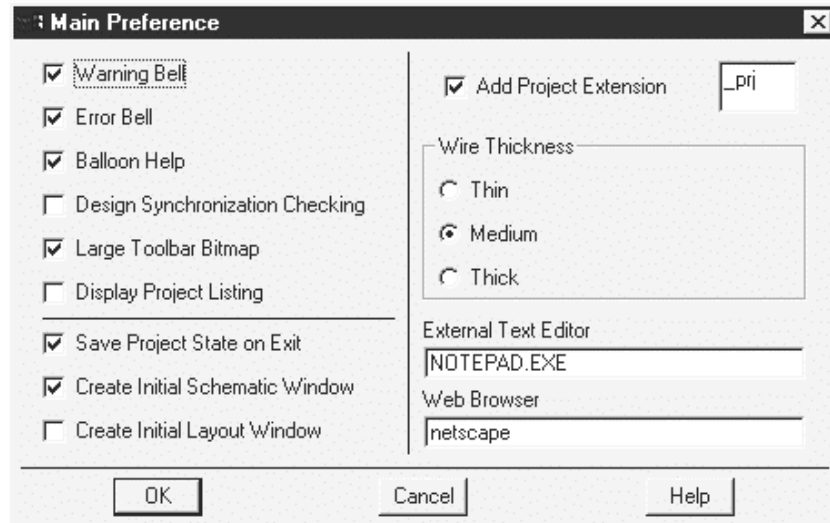


- c. Click the **Options** command.

These are used to setup global elements for the user interface and for macro recordings. For now, click **Preferences...** and a dialog box will appear (shown here).

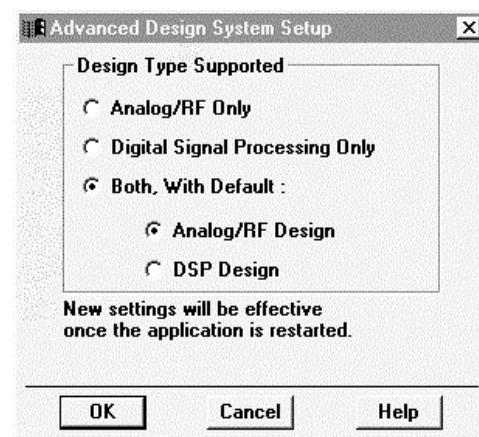


- d. In the Preference dialog, be sure the **Add Project Extension** box is checked. This means that all projects (directories) you create in ADS will automatically be appended with the extension **_prj** so that you will recognize them. For this lab, no other preferences should be set. Click **OK** when finished.



- e. Click **Options > Advanced Design System Setup...**

When you first install ADS or when your ADS system is updated, you will also see this dialog box. It is used to define which type of schematic elements and library elements are the default, depending upon the licenses you have. For this lab, be sure the settings look like the picture here – if they do, select **Cancel**. If not, check with the instructor.



- f. In the Main window, click the **Window** command.

At this time, most of these commands will not be available (inactive) because you have not yet created a project and no other windows are opened. After you create a project, these commands will be available.

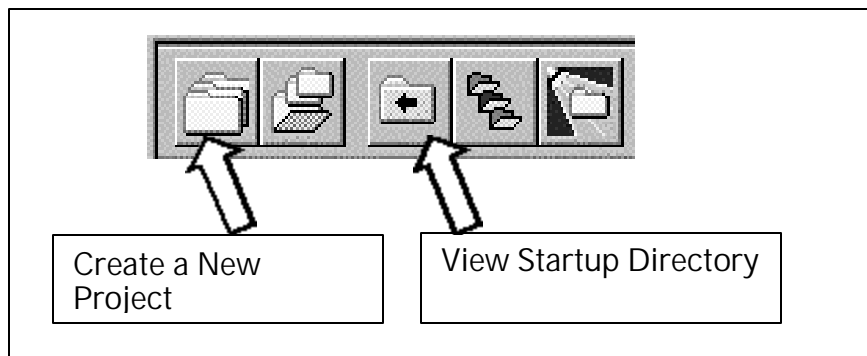
- g. The final Main window command is the Help command. Click **Help > Topics and Index** and a new window will open. ADS has Help topics and on-line manuals. Spend a few moments looking through the topics and then click **Close All**.



3. Create a new Project

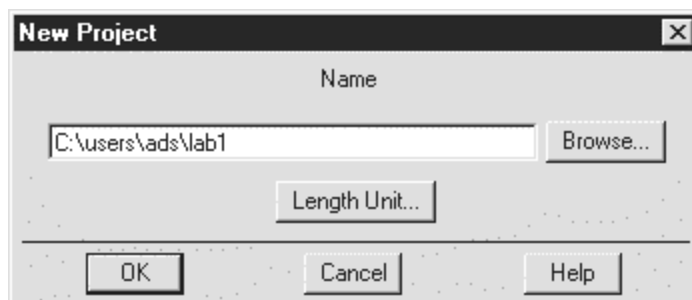
For this step you will use the icons on the Main window. Typically, clicking the correct icon means you have one less mouse click to execute than using the menu commands. In addition, you can identify what an icon does by placing the cursor on the icon. This is called *balloon help* and is one of the preferences you can turn off or on.

- a. Try moving the mouse cursor slowly into the bottom of each icon on the Main window. You will see the balloon help and learn the icon names.
- b. In the Main window, click the icon: **View Startup Directory**. This will put you in the starting directory for ADS.



- c. Click: **File > New Project**.
- d. When the dialog box appears, give the project a name by typing: **lab1**.

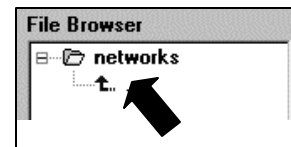
Notice that the length unit is a setting for items such as microstrip lines and also used for layout. For this lab use the default value (mil). Notice that the Browse button allows you to create project directories anywhere you like. Click **OK** to continue.



4. Examine the project File Browser and Project Hierarchy

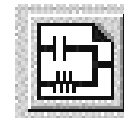
The Main window **File Browser** area should now show that you are in the **lab1** project directory. Notice that the sub-directories (data, networks, etc.) were created automatically. Also, the schematic icon is now activated (no longer gray).

- a. In the main window, double click on the **networks** directory. The file browser now shows you are in that directory which is empty (no schematics exist).

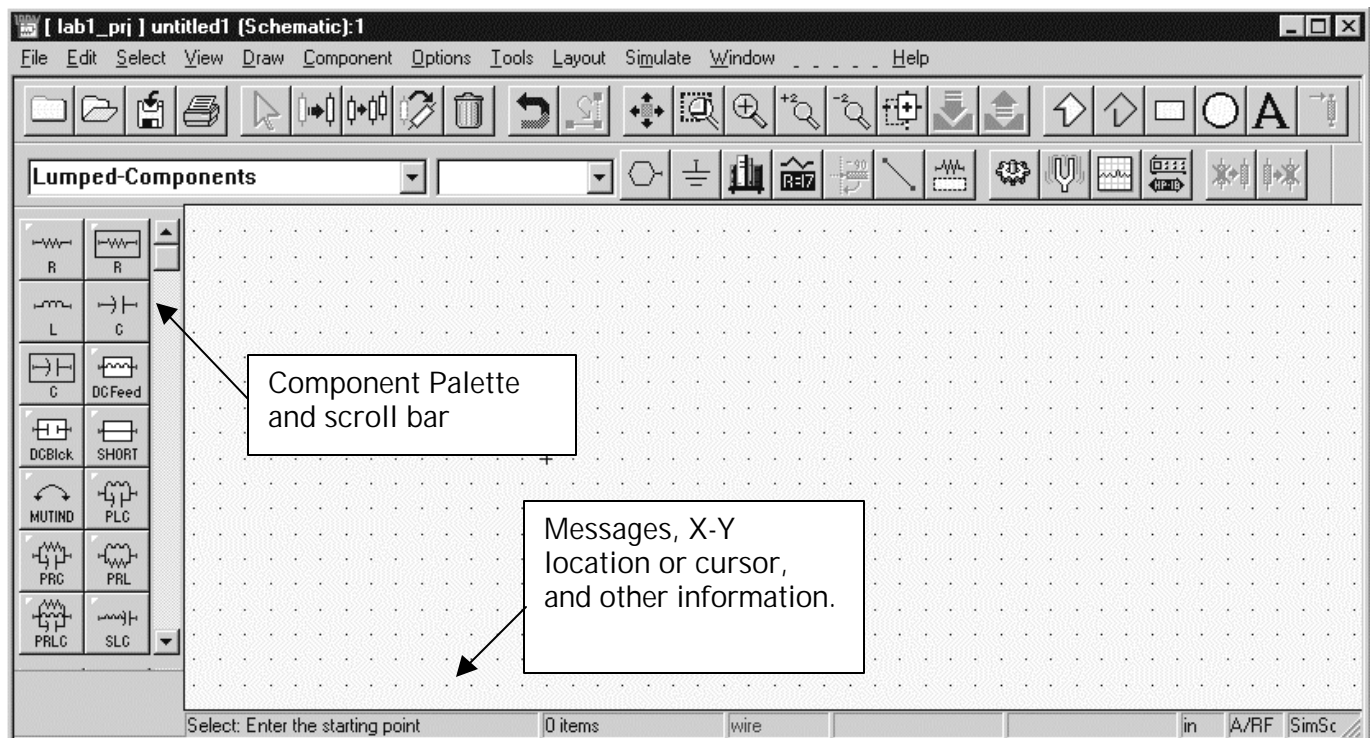


- b. To return, double click on the two dots (..) next to the arrow and you will go up one directory.

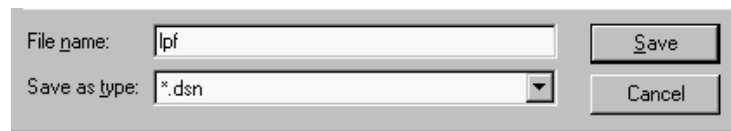
5. Create a Schematic low-pass filter design



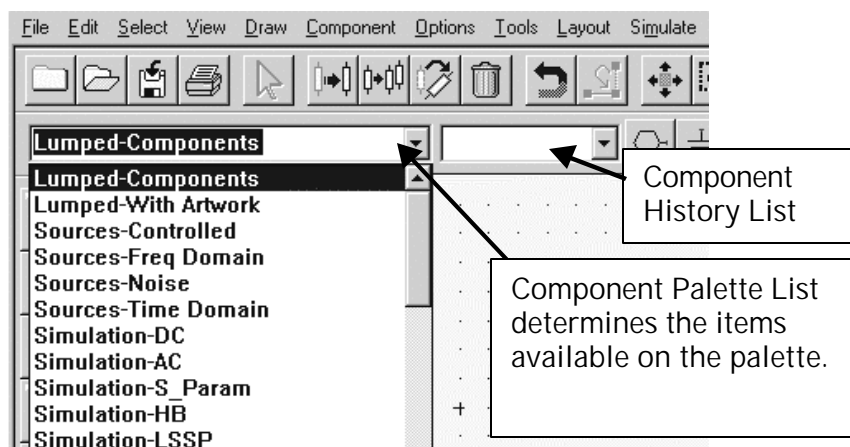
- a. In the Main window, click the **New Schematic Window icon**. This is the same as selecting the menu command: **Window > New Schematic Window**. Immediately, the Schematic window will appear. If your preferences are set to create an initial schematic, you will have two schematics now opened – close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



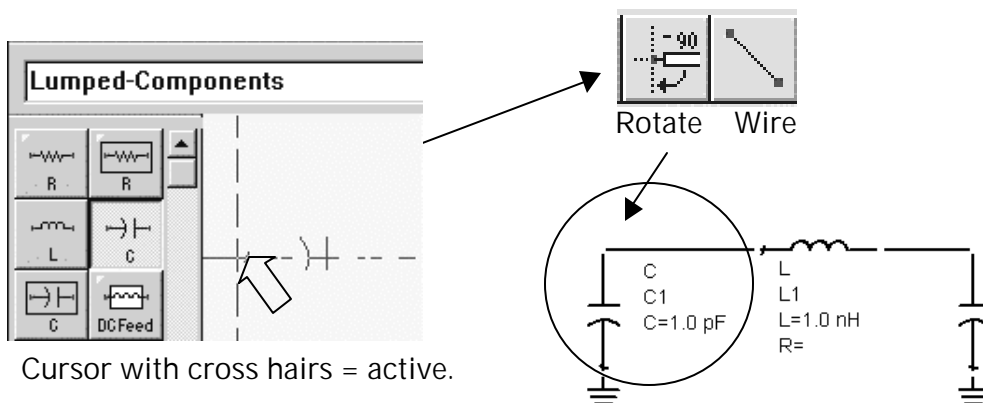
- b. **Save the schematic design:** notice the top line (window border) shows the project name (lab1_prj) and the name of the schematic (untitled) with an incremental number (1, 2, 3, ...) of the schematic window you have open. To name the schematic, click **File > Save As** and type in a name such as **lpf** or **low_pass** and click Save. This will save it in the networks directory of the lab1 project.



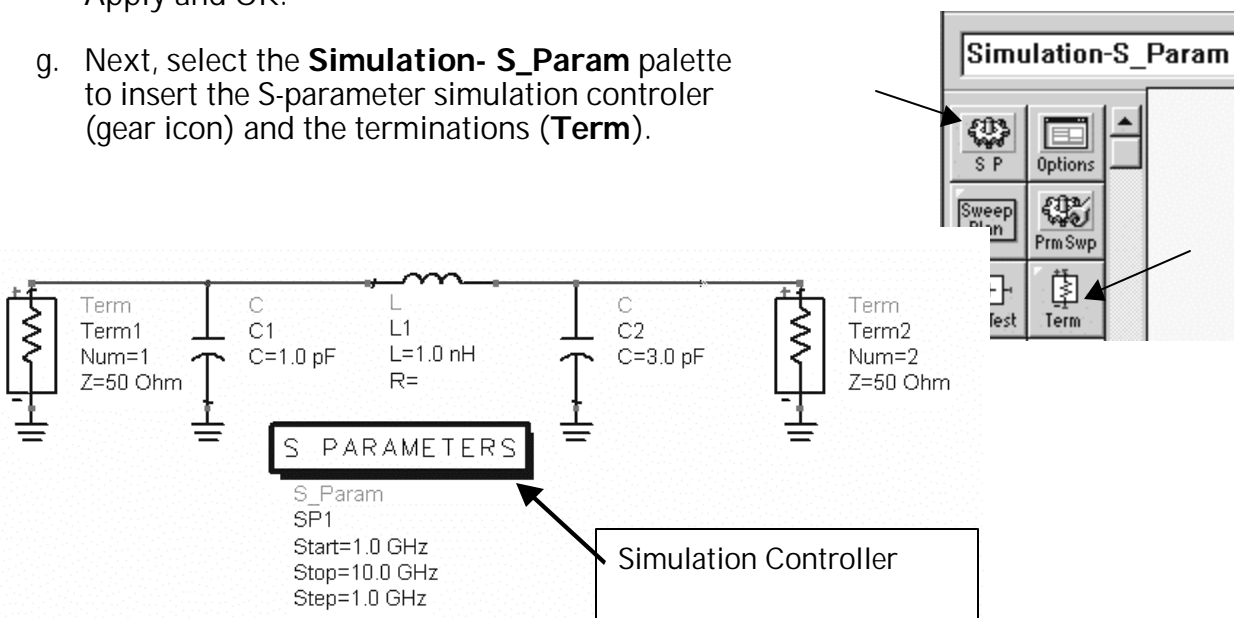
- c. Examine the commands and icons. Click on the small arrow on the Component Palette list to see the palette choices. Also, move the Scroll Bar down and up to see how it works.



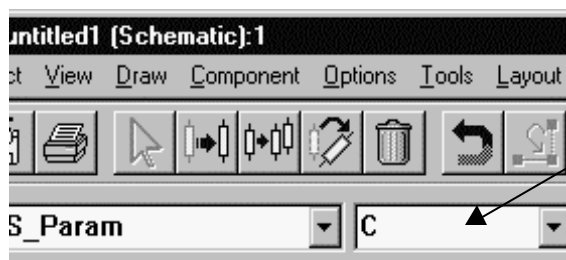
- d. In the **Lumped Components** palette, click on the **capacitor "C"** and click the **rotate** icon as needed to get the correct orientation. Then click to insert the capacitor as shown on the schematic. Next, insert another capacitor.



- e. Continue creating the low-pass filter as shown by inserting the **inductor**. Then insert **grounds** and **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 circuit is built, edit the value of **C2 = 3 pico-farads**. To do this, click on the component and then click the icon: Edit Component Parameters (same as double clicking the capacitor symbol). When the dialog box appears – change the value to 3.0, click Apply and OK.
- g. Next, select the **Simulation- S_Param** palette to insert the S-parameter simulation controller (gear icon) and the terminations (**Term**).



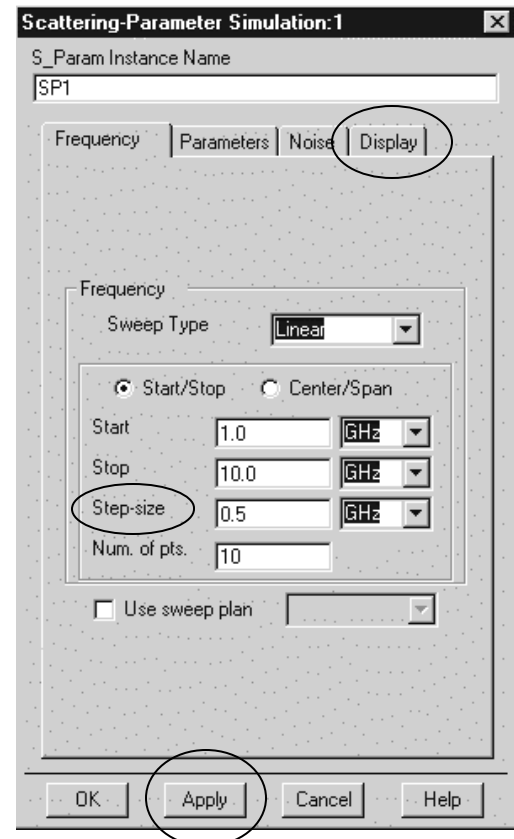
- h. **Using Component History:** After the circuit is built, try deleting a capacitor and then reinserting it by typing in the capital letter **C** in the Component History window and press Enter. Next, edit the value directly on the schematic by highlighting the value and typing over with the new value (3.0). Verify that it has changed by looking at the value in the edit dialog box.



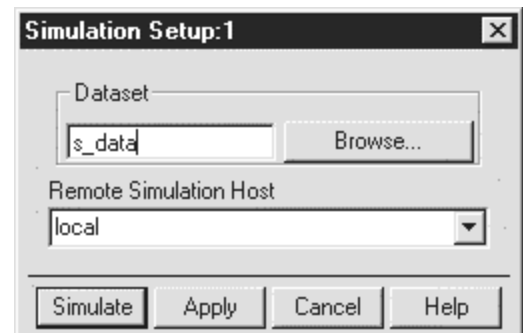
NOTE: You can insert components by typing in the component label (C, L, R etc) instead of using the palette.

6. Setup and Run the Simulation

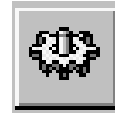
- To setup the simulation, double click on the S-parameter simulation controller on the schematic. When the dialog box appears, change the step size **to 0.5 GHz** and click **Apply**. Notice how it updates the value on the screen as it reads the entries. The **OK** button does the same thing and also dismisses the dialog box.
- Click the **Display** tab and you will see that the Start, Stop and Step values have been checked to be displayed on the simulation controller. You will use the display tab for setting other controllers to display the desired settings during this class.
- Click the **OK** button to dismiss the dialog box.
- Setup the Simulation dataset. The default dataset name is the same as the schematic (*lpf*). But you can give the dataset (a file) a name. To do this, click **Simulate > Simulation Setup**. When the dialog box appears, type in the name like **s_data**. Then click **Apply**. In general, the default dataset name is the same name as the schematic design but you can control it using this method.



DEFINITION of a DATASET: A dataset is a file that may contain matrices, results calculated from equations, node voltages, etc. It has the extension **.ds** which means dataset (results of a simulation). It is important to remember that all datasets are only written into the project **data** directory but the data display windows (.dds files - which means data display server) are not in the data directory, they are under the project directory.



- e. Click the **Simulate** icon (gear) to start the simulation process. This is the same as clicking Simulate in the setup dialog. When you simulate, the resulting data is always written into the current dataset you have setup.



- i. Next, look for the **Status window** to appear and you should see a message similar to the one here, describing the results of the simulation. SP1 refers to the s-parameter simulation controller and its settings. If no errors occurred, the message tells you the simulation is finished and that the dataset has been written into the data directory in the project you are in (here it is lab1_prj).

```
SP SP1[1] <(GEMX netlist)>   freq=(1 GHz->10 GHz)
.....

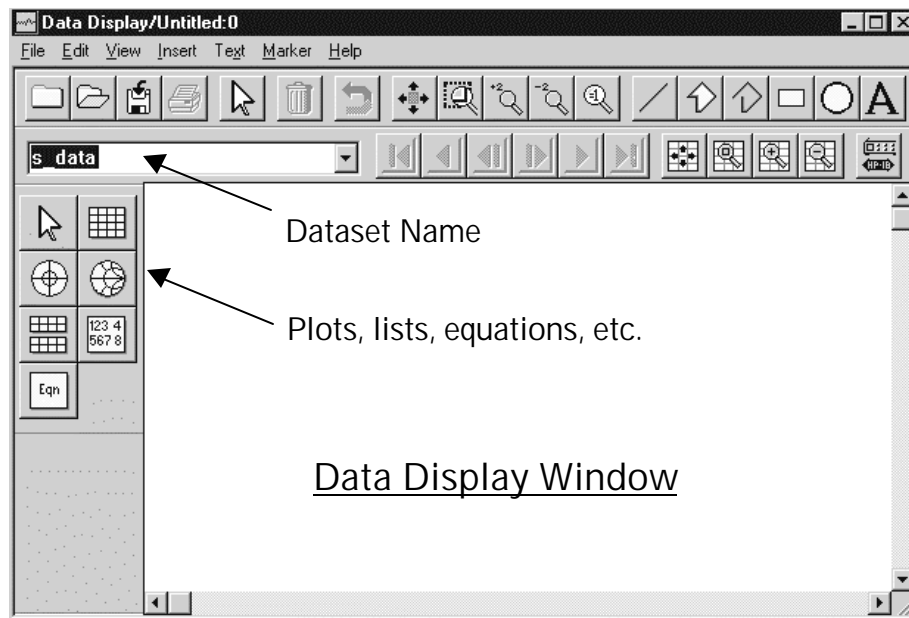
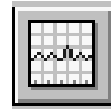
-----
Simulation finished: dataset 's_data' written in 'C:\users\DEFAULT\lab1_prj\data'.
-----

Resource usage:
  Total CPU time: 0.56 seconds.
  Simulation stopwatch time: 1.30 seconds.
  Total stopwatch time: 4.39 seconds.
```

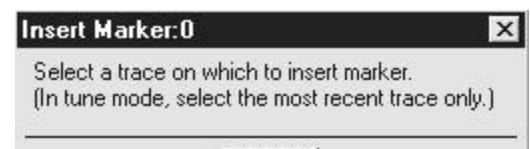
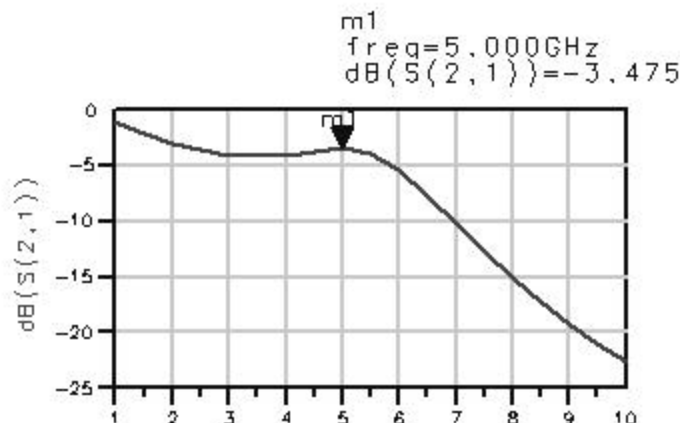
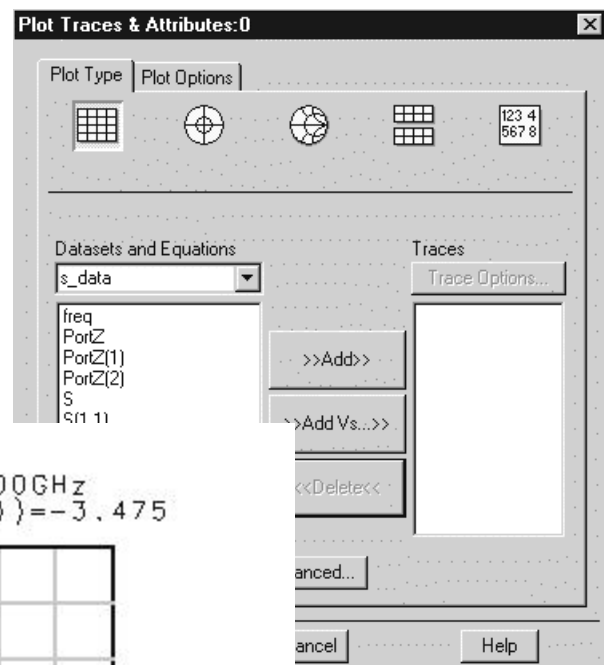
- j. Close the Status window after you see the message. You can always get the status window back using the command: **Window > Restore Status**. The simulation status information can be restored using the Window command in the status window and then selecting the simulation from the list.

7. Display the simulation results (Data Display window)

- Open a data display window from either the Main window or the schematic window by clicking the **Data Display** icon.
- When the Data Display window opens, the name of the dataset will appear in the list.



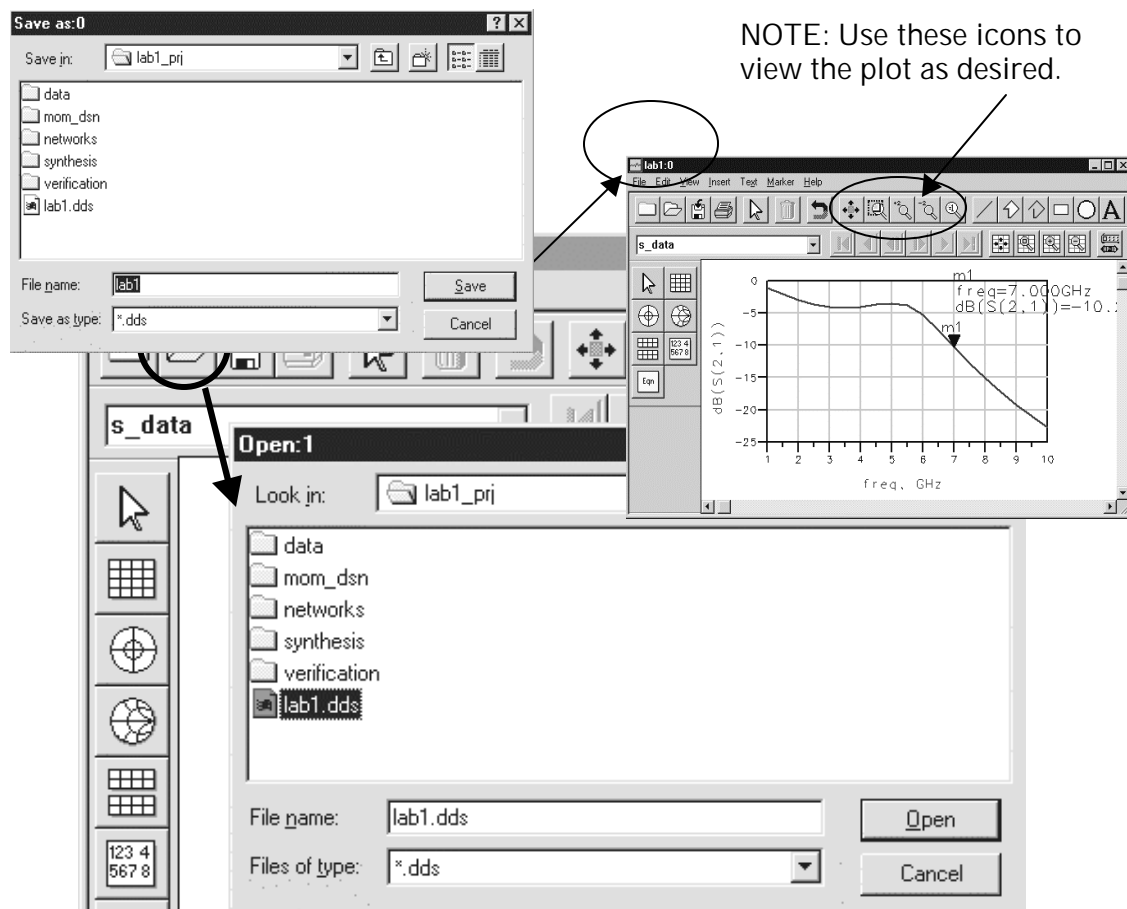
- To create the plot, click on the **Rectangular Plot** icon and move the cursor (with ghost plot) into the window and click. When the next dialog box appears, select the **S21** data and click the **Add** button.
- The next dialog will prompt you to specify the type of data to display. Select **dB**.



Marker > New. Select the trace and click to insert the marker. Move the marker using the cursor or the keyboard arrow keys. Also, move the marker text by selecting it and positioning it as desired. Try deleting the marker or putting another marker on the trace.

8. Save the Data Display and Schematic

- In the Data Display window, notice that it is labeled *Untitled*. To save this data display window with a name, click **File > Save As** and type in the name: **lab1** and click **Save**. This means that it will be saved as a **.dds** (data display server) file in the lab1 project directory and it will have access to all data (.ds files or datasets) in the *data* directory.

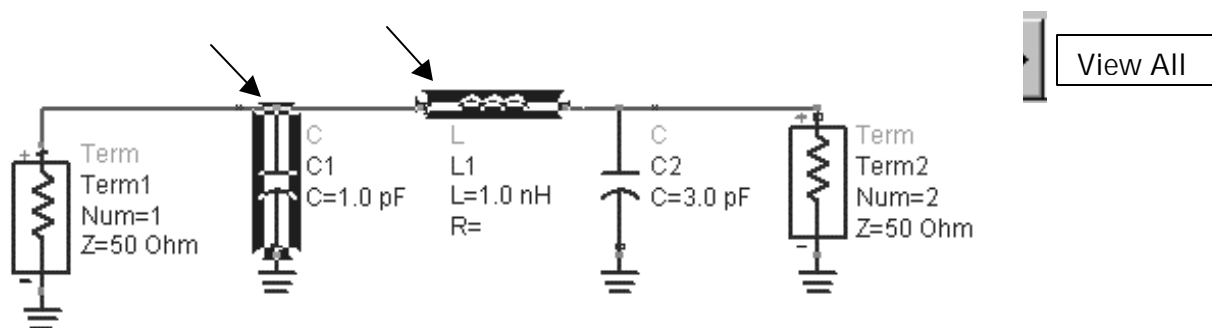


- Close the data display and reopen it: After saving lab1.dds, close it by clicking the **X** in the window corner. Then reopen it as follows: click the **data display icon** to open the window. Click the **File > Open** icon and select **lab1.dds** in the dialog and click **Open** and it will reappear with your S21 plot. Also, notice that the default dataset is **s_data** from your previous simulation.

9. Tune the filter circuit

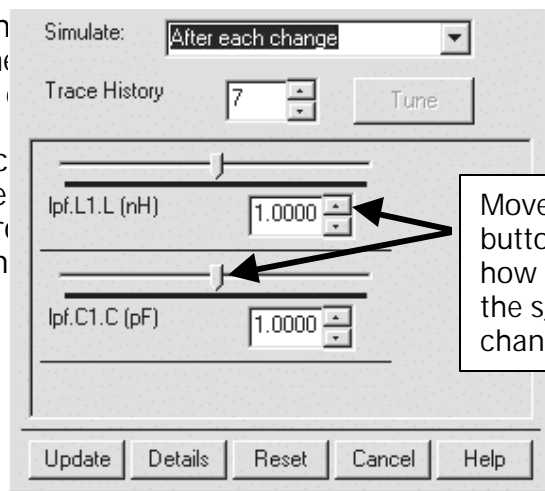
This step introduces the ADS tuning feature that allows you to alter the parameter value(s) of components and see the simulation results. In this step, you first select the components and then select the tuning feature. If you select the tuning feature first, you must select the component parameters and not the components.

- First, in the filter schematic, select (click) the capacitor and inductor to be tuned as shown. Hold down the **SHIFT** or Ctrl key to select multiple components: C1 and L1.



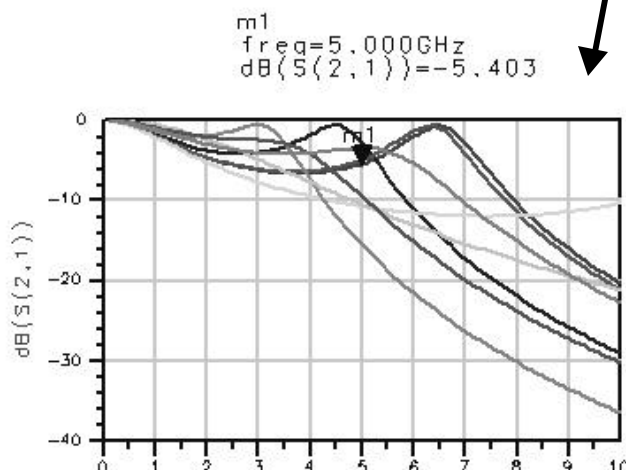
- Position the cursor over the **Tune** button in the simulation control panel. You can see them both on the schematic and in the simulation control panel. Use the **View All** button to view all components.

- Click the **Tune** button. The tuning dialog box appears. The tuning icon.



Move the slider or click on the buttons to tune values. Notice how the new traces appear on the s_data plot after each change.

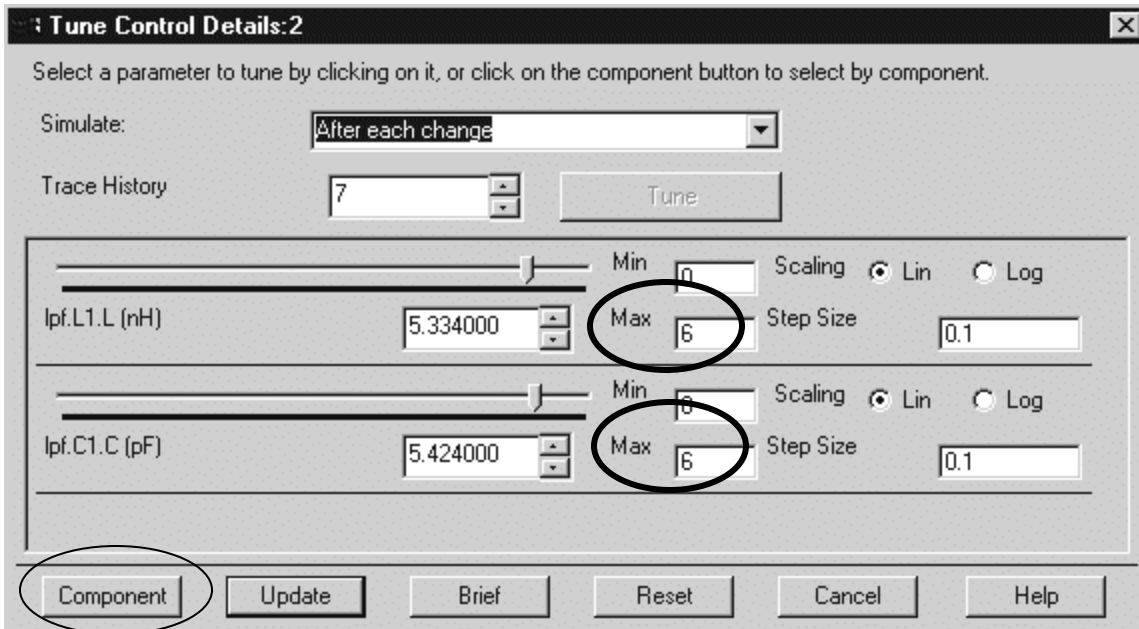
Tuning



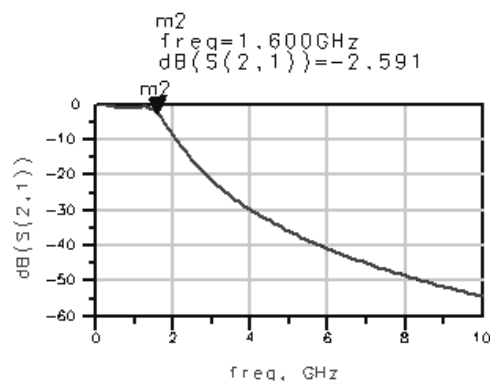
Each tuning creates another trace. The marker moves to the most recent simulated (tuned) trace, which is red. This trace is the s_data dataset which is changed each time you tune (simulate).

- d. Change the tuning range: In the Tune Control dialog, click the **Details** button and watch the dialog change from brief to the detailed. Type in a larger range such as 6 and then tune the filter again. You should be able to see a greater response.

Details



- e. Continue tuning and when you are satisfied with the results, click the **Update** button to have the C and L values updated on the schematic. If you click the **Component** button you will notice that it allows you to add other parameters to the tuning. The Brief button returns to the smaller (brief) Tune Control dialog. When you are satisfied with the tuned response, simply click the **Cancel** button and the plot will contain the final tuned trace such as the one here.



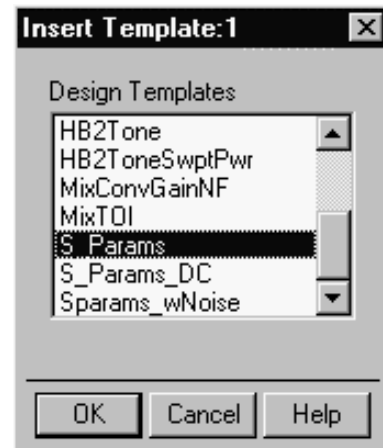
- f. Save the data display and the schematic.



10. Using Templates

Templates make it easy to include the required simulation controllers, ports, and other items used in the simulation. In addition, you can create your own templates or customize the existing ones.

- a. From the Main window, click the **Schematic icon** and another schematic window will open.
- b. In the new schematic, click: **File > Insert Template** and insert the **S_Params** template.



- c. Modify the template in some way - for example, change the simulation controller values and then click: **File > Save As Template**. When the next dialog appears, type in a name for the template: **my_template**.
- d. **Open a new schematic window (from the Main window)** and insert your template in the same way (File > Insert Template). Now you know how to create your own template.
- e. Click: **File > Close Design** (do not save the schematic) and close the window.

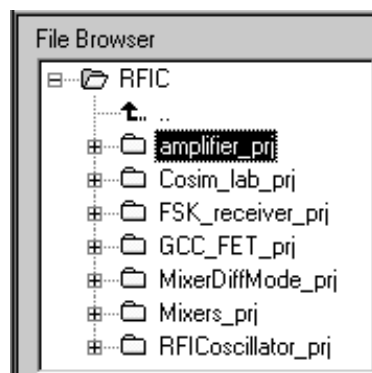
At this point you have stepped through the basics of using Advanced Design System. The following steps will show you the basics of using the Examples directory.

About the Examples Directory

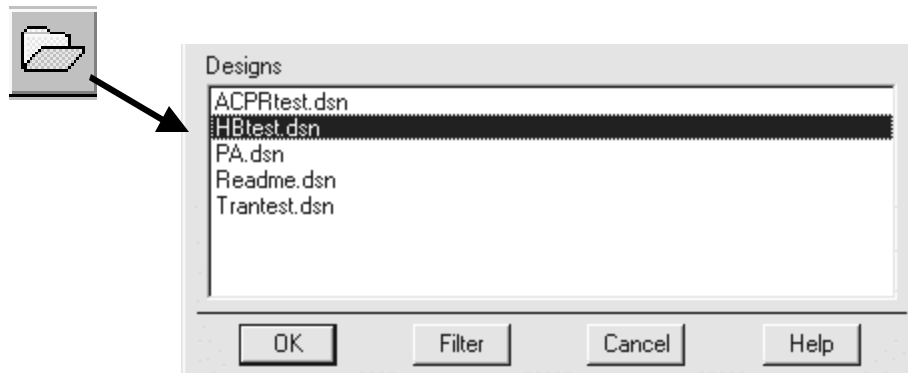
All of the examples can be examined, including the results of the simulations. However, because the example files should remain unchanged, copy them into your own directory to simulate or modify them.

11. Open the Example Directory: RFIC, amplifier_prj, HBtest.dsn

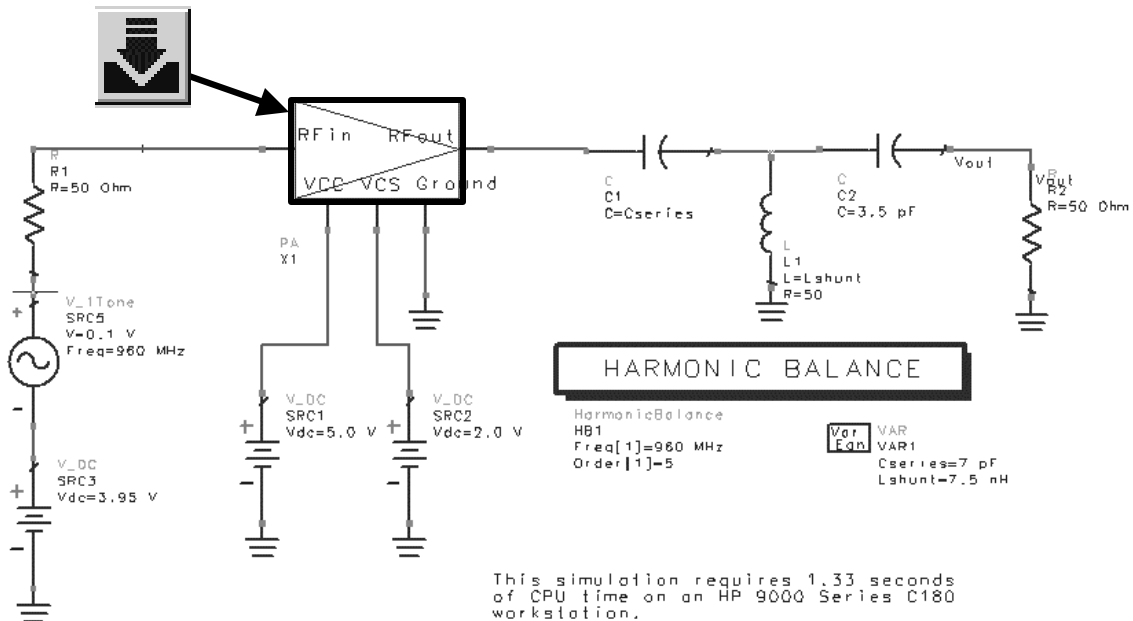
- a. In the Main window, click on the **View Examples Directory** icon. You will be prompted to confirm you are changing directories. Afterward, select the **RFIC** directory and open the **amplifier_prj** directory.



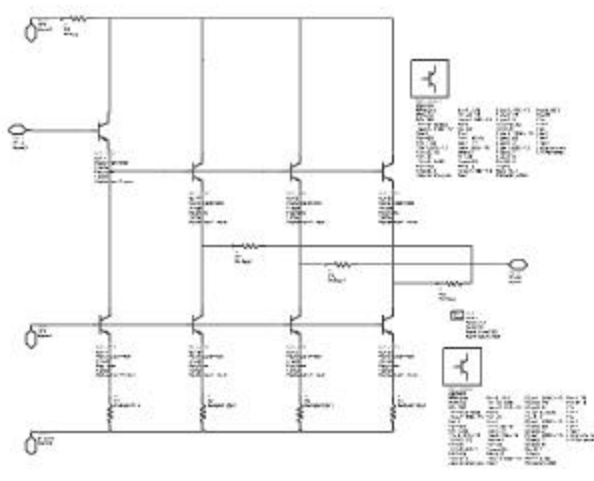
- b. Immediately, two schematics windows will open: Readme and a schematic design (ACPRtest). This is how all example files open with some documentation and a particular example.
- c. In the schematic, click the **File > Open** icon and you will see other RFIC amplifier designs. Now, open the **HBtest.dsn**.



- d. This is the top-level hierarchy of the HBtest.dsn. This is where the simulation is setup and controlled. To see the amplifier the sub-circuit click on the symbol (shown here) and then click the icon: **Push into Hierarchy**.



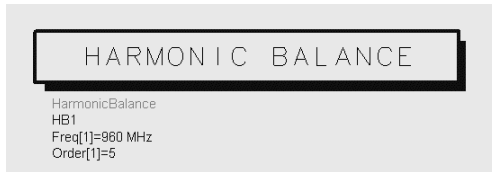
- e. You can go back to the upper level by clicking the reverse arrow



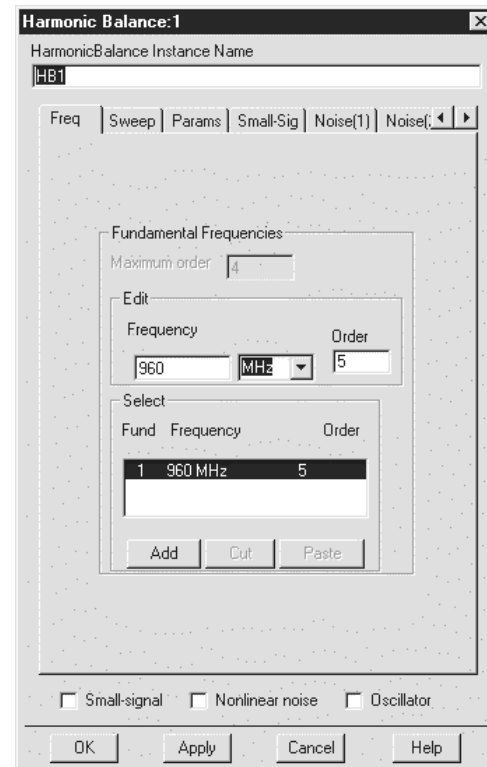
Click here to return to upper level design.

Lower level schematic shows transistor level circuit with model assignments and port connectors.

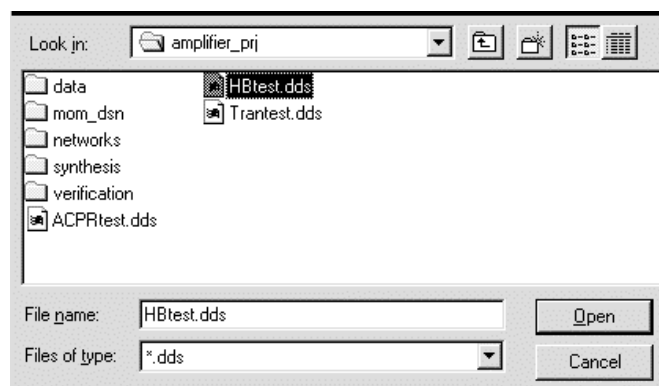
- f. After you return to the upper level, examine the **Harmonic Balance controller** by double clicking on it or by selecting it and clicking the edit icon.



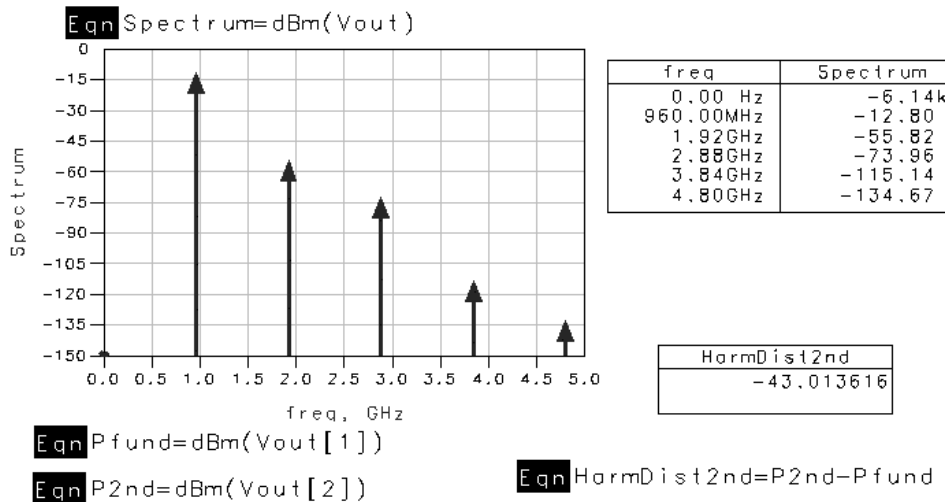
As you can see, the Harmonic Balance controller has many tabs for setting up the type of simulation that you want. The purpose of this step is only to get you acquainted with the simulation controller. Look through the tabs and **Cancel** when you are done.



- g. To see the data from this simulation, click on the icon: **New Data Display**. When the window opens, click on **File > Open**. Then select the **HBtest** data display and click **Open**. This is how you can open your own saved data display files and access the data in the datasets.



The data display window will appear. Examine the data and notice that "Vout" in the equation is a *named node* point on the schematic. When finished, close the display and schematic window. Later on, you will be setting up these same simulations.



12. Delete the lab1 project directory

Because you are in another directory (Examples directory), you can use the Main window command **File > Delete Project** to delete the lab1 project. You must be in a different project to delete another project.

This lab exercise ends here.

EXTRA EXERCISE: If you finish the lab early, spend more time examining the Examples directory designs or try using the Layout window. Or, go back and try building a better low pass filter.