

LAB #01: Introduction to ADS, Tools & Commands

Objective:

To familiarize the students with modeling tools and commands in ADS.

Pre Lab:

What is ADS?

ADS stands for Advanced Design System (ADS). ADS is a simulator like spice, cadence. But it focuses on the RF and microwave design, so most of its devices on the library are microwave devices. Advanced Design System uses projects to organize and store the data generated when you create, simulate, and analyze designs to accomplish your design goals. A project includes circuit, layout, simulation, analysis, and output information on the designs that you create, along with any links you add to other designs and projects.

ADS USES:

Here are some of ADS uses:

- DC analysis:** is used for determining the bias point of the circuit.
- Transient analysis:** runs the time domain analysis on the circuits and considers the nonlinearity of the elements.
- AC analysis:** runs the small signal analysis and use the linear model of elements on their bias point. So the nonlinear elements like transistor are replaced by a linear model (small signal circuit) which includes resistors, capacitors, inductors and voltage & current sources.
- S_parameters analysis:** calculates the Scattering parameters of the components, and shows the variation of the S_parameters over different frequencies. It is also used for calculating noise figure and group delay.

Starting ADS:

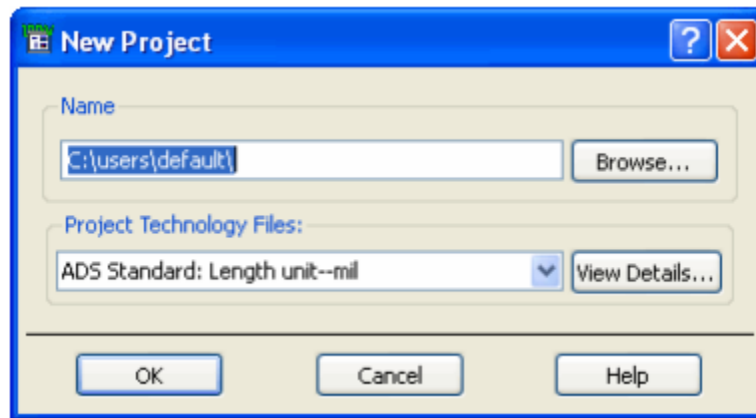
All design work must be done in a Workspace directory. Working in Workspace directories enables you to organize related files within a predetermined file structure. This predetermined file structure consists of a set of sub-directories. Some of those sub-directories are :

- nameofyourworkspace_lib contains schematic and layout information, as well as information needed for simulating
- data is the default directory location for input and output data files used or generated by the simulator

Creating a project in ADS:

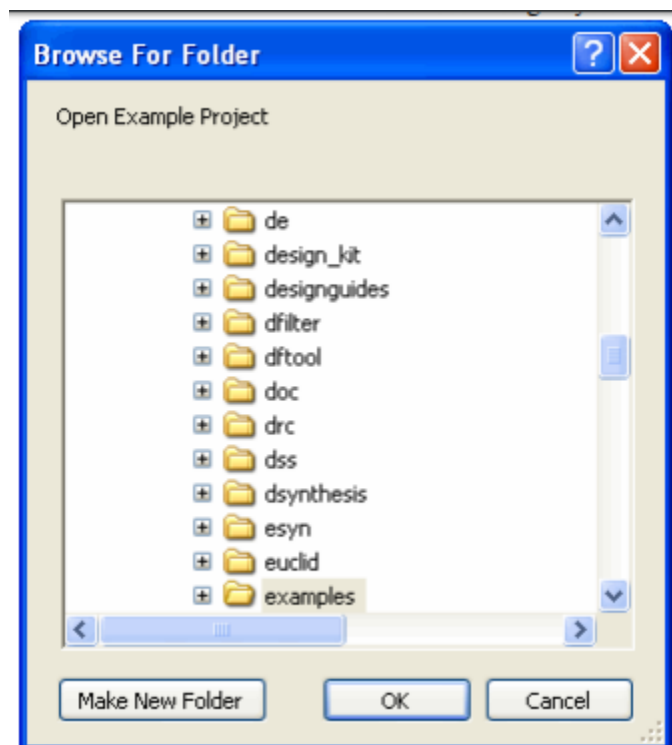
Use the Main window to create a project that you can then use to organize your designs. A project includes circuit, layout, simulation, analysis, and output information on the designs that

you create, along with any links you add to other designs and projects. To create a project... 1. **Choose File > New Project** 2. **Enter Project Name & Location**



Using examples project in ADS:

Advanced Design System offers an extensive set of example projects that demonstrate designing for various technologies. You can view example projects, as well as copy and modify them to create new projects. To view an example project choose **File > Example Project** and use the **Open Example Project dialog box** to select the directory and the project.



To copy an example project choose **File > Copy Project** , click the **Example Directory button**, and use the **dialog box** to locate and **copy the project**.

Using a project in ADS:

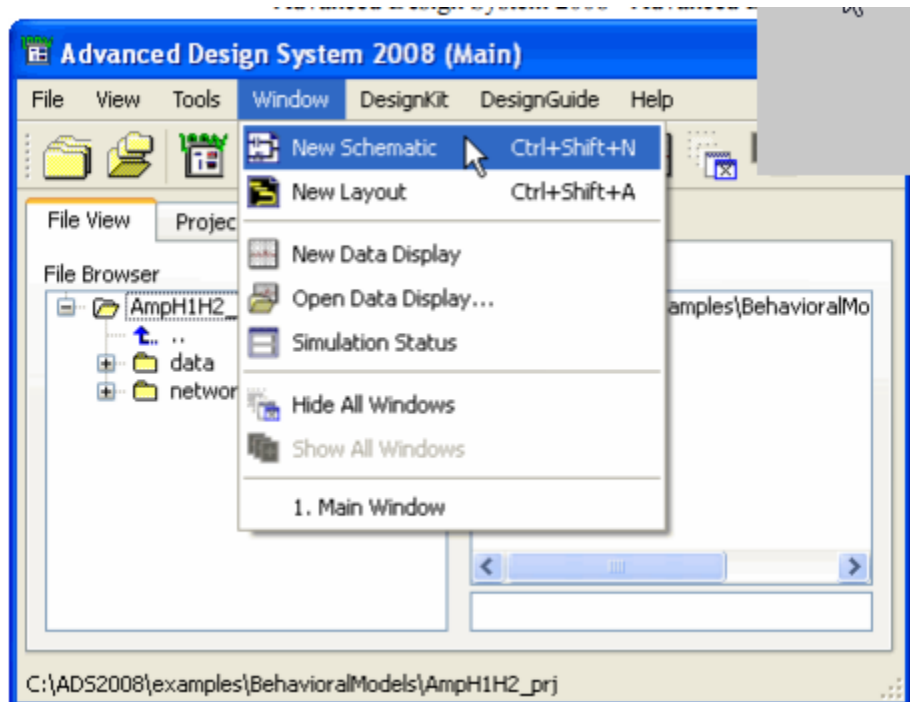
Advanced Design System uses designs to store the schematic and layout information you generate to accomplish your design goals. A design can consist of a single schematic or layout, or it can be made up of a number of schematics and layouts embedded as subnetworks within a single design. All designs in a project can be displayed and opened directly from the Main window or from within a Design window.

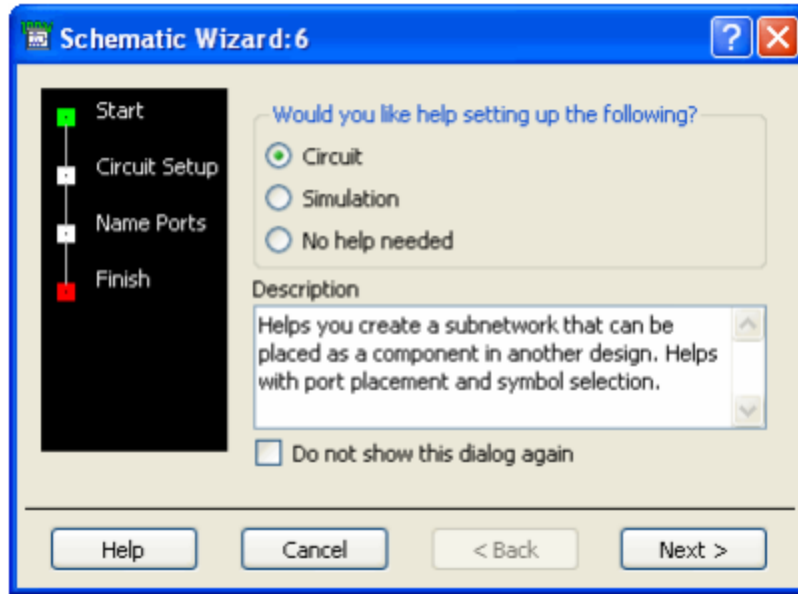
In a Design window you can...

- Create and modify circuits and layouts
- Add variables and equations
- Place and configure components, shapes, and simulation controllers
- Specify layer and display preferences
- Include annotations using text and illustrations
- Generate layouts from schematics (and schematics from layouts)
-

Creating a design project in ADS:

Choose **Window > New Schematic** in the Main window and use the **Schematic Wizard** to set up a design.



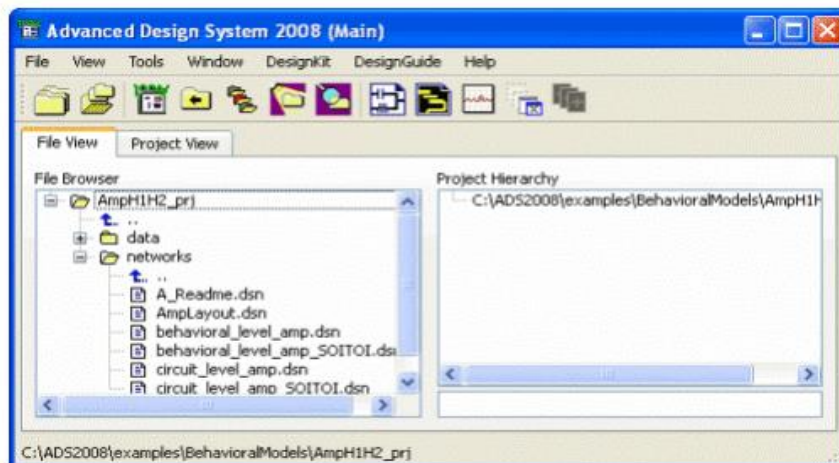


- Choose **Insert > Template** in the Schematic window and select a template for the new file. When you use a template, most of the initial setup and configuration for the schematic, the simulation, and the data analysis is done for you automatically.
- Choose **File > New Design in the Schematic (Layout)** window and use the New Design dialog box to name the file you are creating. You can also open the Schematic Wizard or choose a Schematic Design Template from this dialog box.

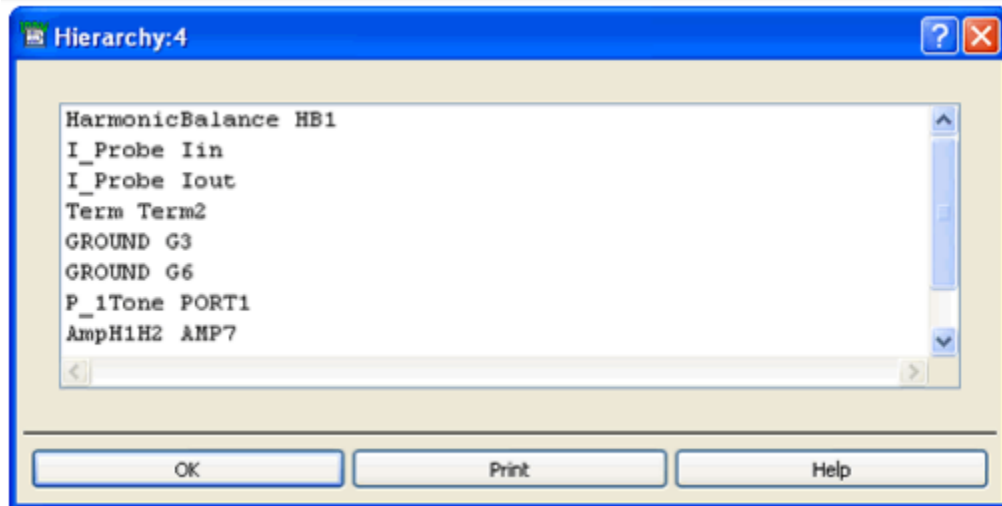
Listing a design project in ADS:

Even after you close all Schematic and Layout windows, designs that you opened remain in memory until you explicitly clear them or exit the program. To list designs, use one of two ways:

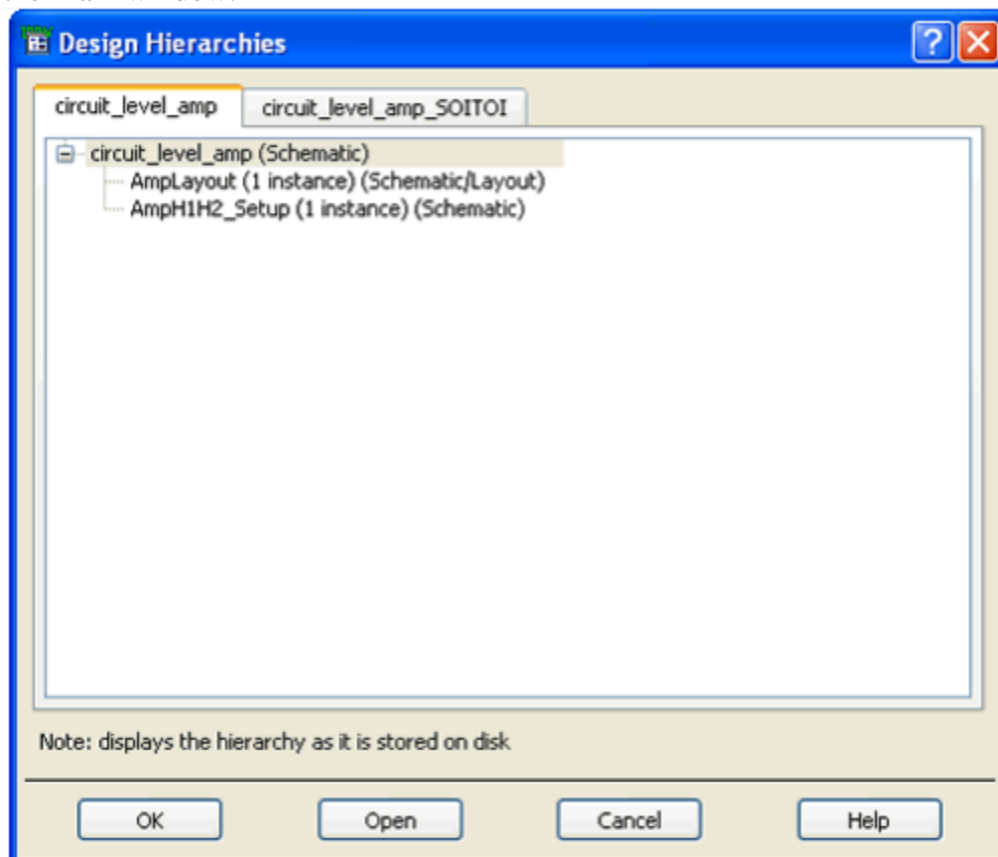
- In the Schematic (Layout) window, choose **Window > Designs Open**.
- In the Main window, double-click the **Networks directory** to display all designs, and then double-click a design to list its schematic, layout, and hierarchical information.



- To view the component hierarchy within a design, choose **Tools > Hierarchy** from the schematic window.

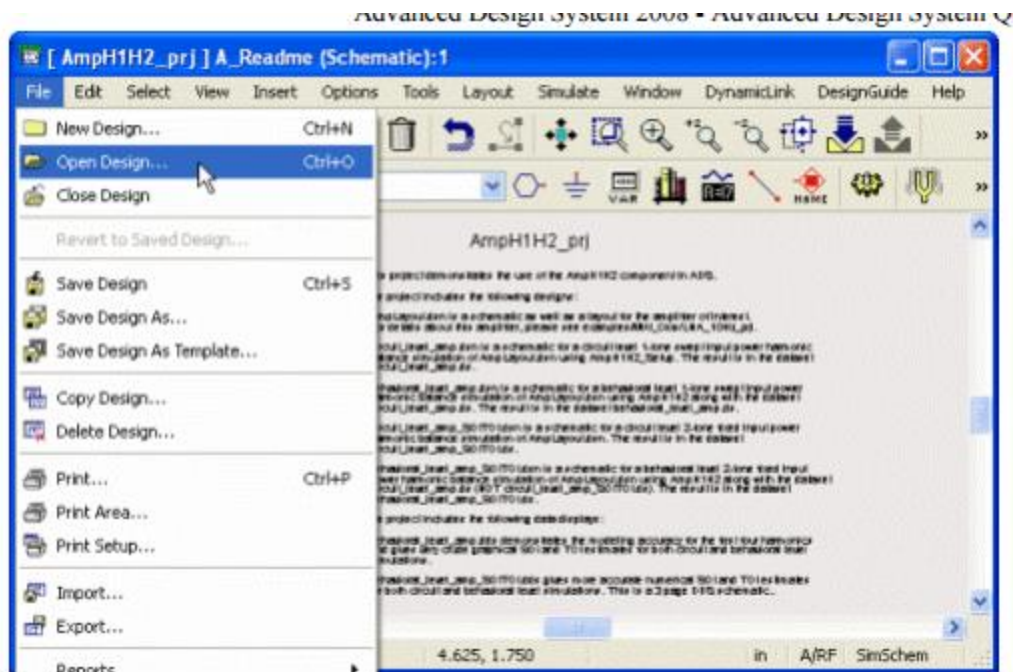


- To view the design hierarchies within a project, choose **View > Design Hierarchies** from the Main window.

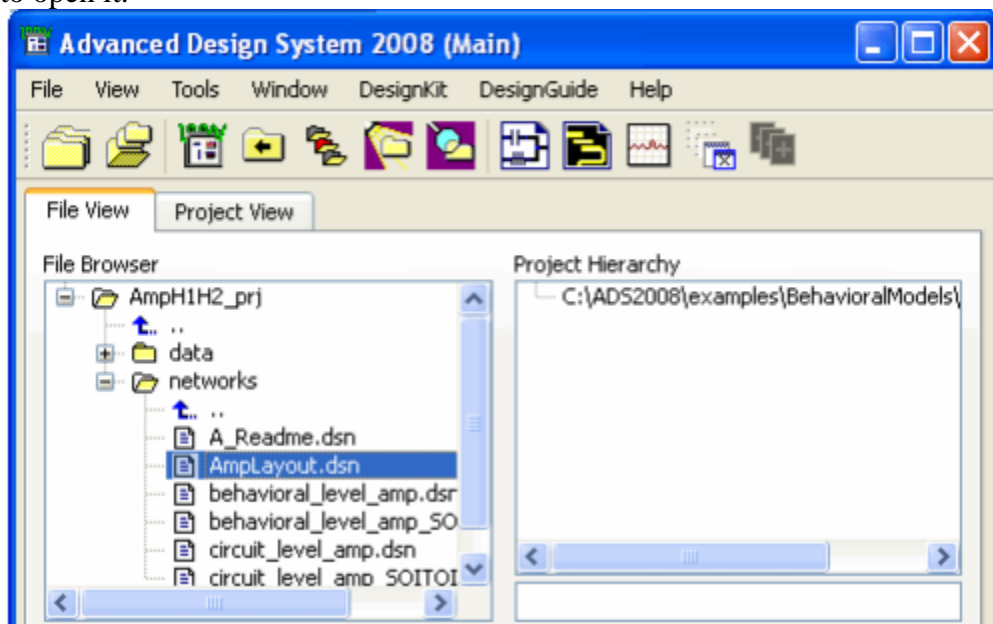


Opening a project in ADS:

You can use either the **Main window** or the Schematic (Layout) window to open a design (layout). Choose **File > Open Design** in the Schematic (Layout) window and use the dialog box to locate and open the design.



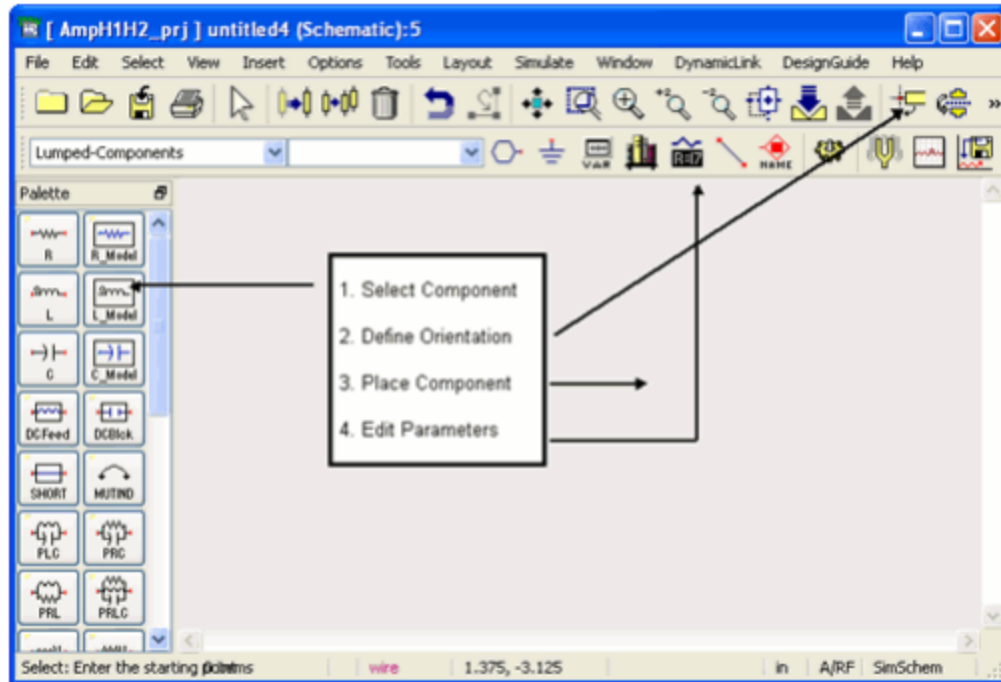
- Use the File Browser pane of the Main window to locate the design (layout) and double-click to open it.



Adding components :

You can place, connect, and configure the following items in the drawing area of your design window to create your design: Components Data items Measurement sources Simulation controllers You can also add entire circuits as subnetworks to create hierarchical designs. Keep in mind that when you begin a design using a template, most of the simulation and analysis setup and configuration is done for you automatically.

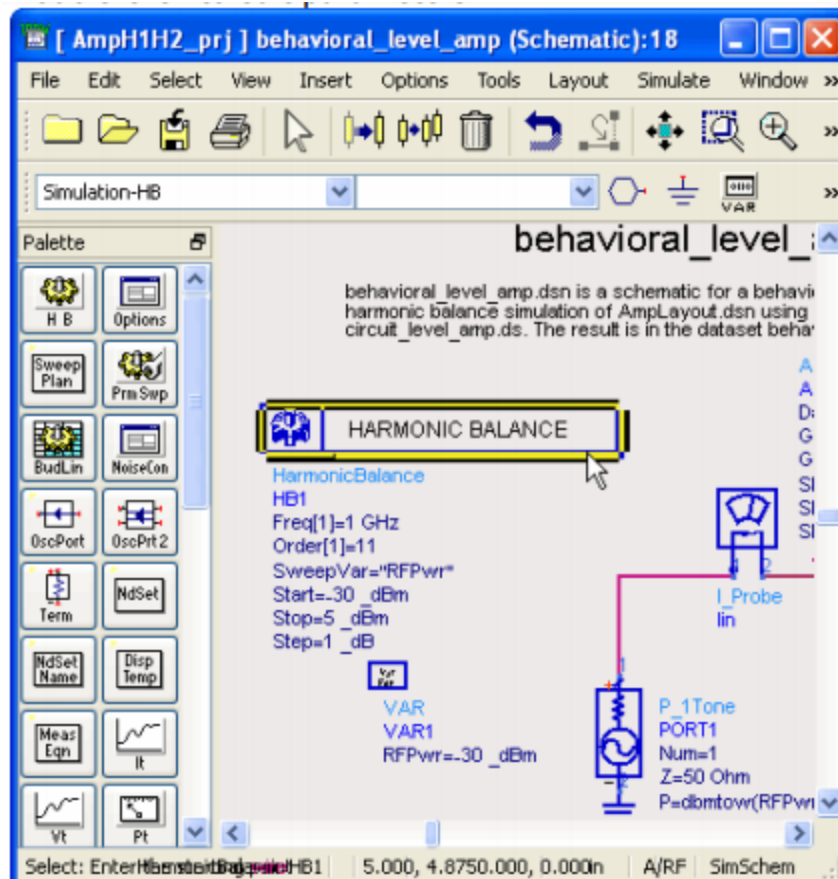
To add a component follow...

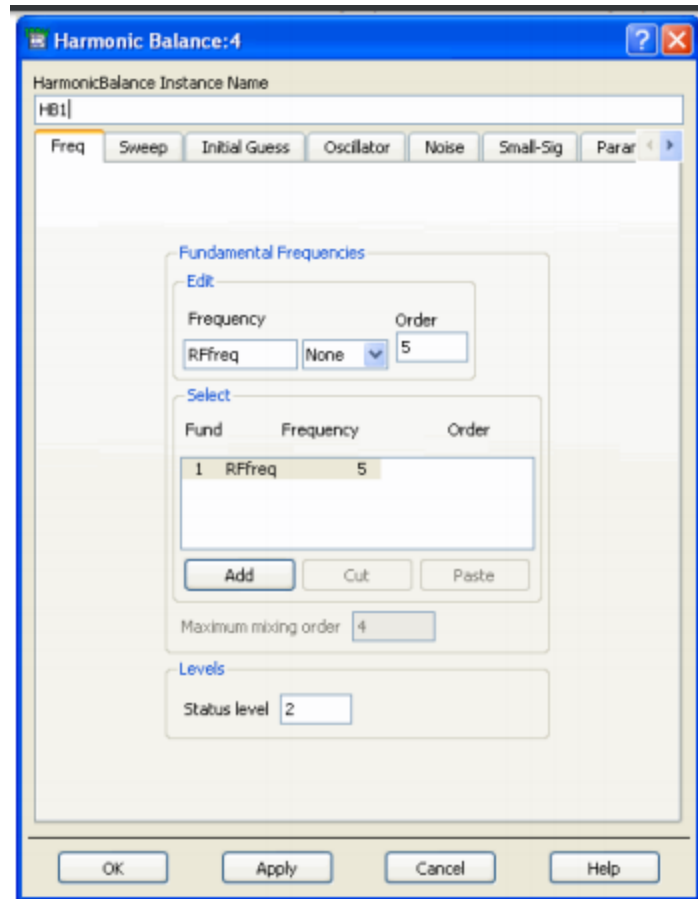


Simulating Design :

Advanced Design System provides controllers that you can add and configure to simulate, optimize, and test your designs. A DSP design simulation requires a Data Flow Controller while an Analog/RF design simulation requires one or more of various controllers. You can either add and configure the appropriate controllers or you can insert a template (choose **Insert** > **Template** from a Schematic window) that contains the appropriate controllers. To simulate a design...

1. Click and place controller
2. Double-click to edit parameters

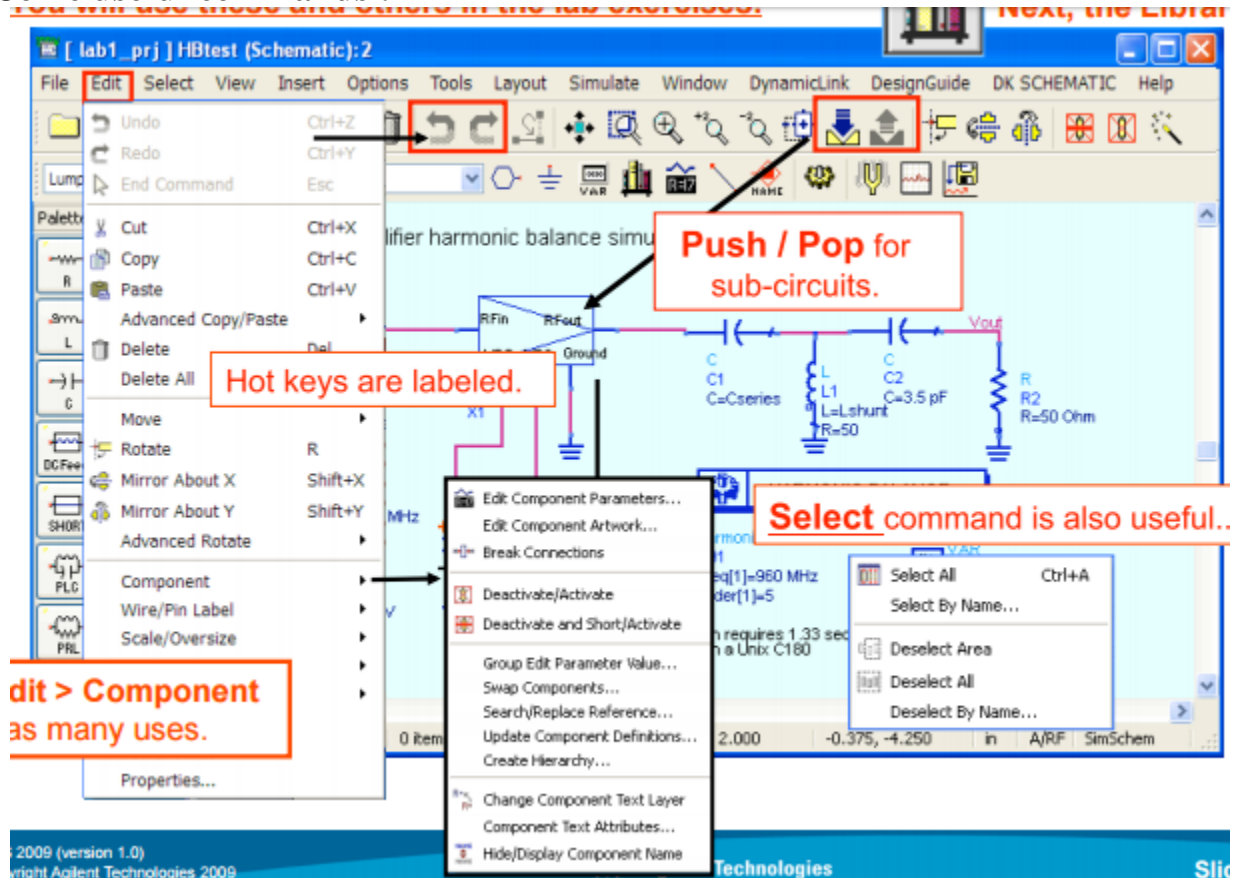




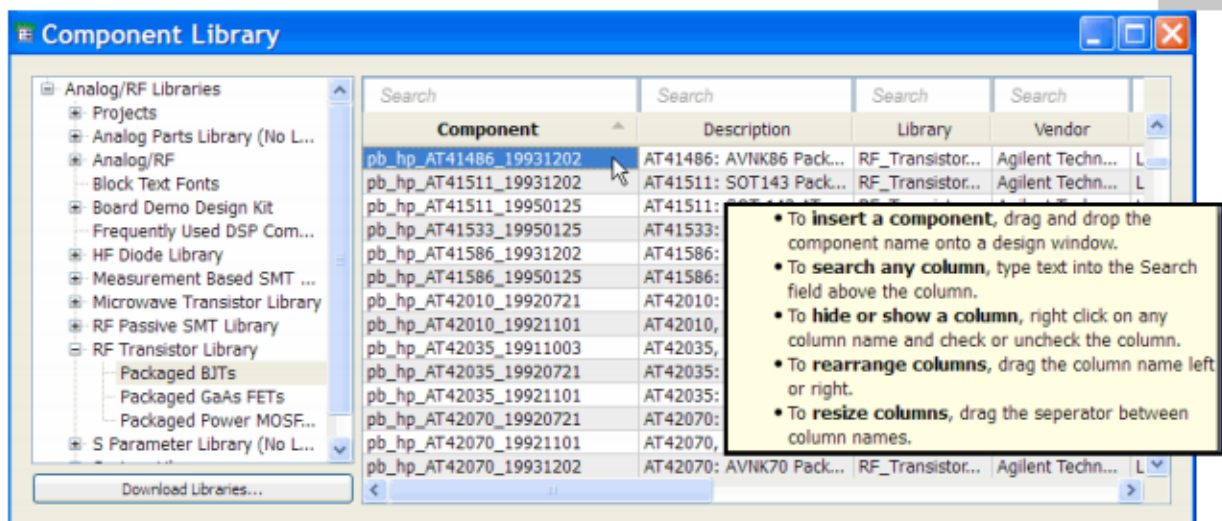
Click to simulate:



Some useful commands :



Library vendor parts + all your circuits:



Example:

Follow the given instructions and make simulation:

Select the palette:

*NOTE asterisk means schematic is not yet saved.

Click the gear and insert the controller.

Other Palette items: Sweep Plan, Options, Meas Eqn. Some items (measurements and components) are simulation specific.

Next, editing the controller! →

Now edit the simulation parameters

S-PARAMETERS

S_Param
SP1
SweepVar="freq"
Start=100 MHz
Stop=2 GHz
Step=100 MHz
FreqConversion=no
CalcNoise=yes

By default, freq is the swept variable here!

Display tab lists all the settings for on-screen display.

Output tab allows you to select what goes to the dataset.

You will learn to use these in the lab exercises!

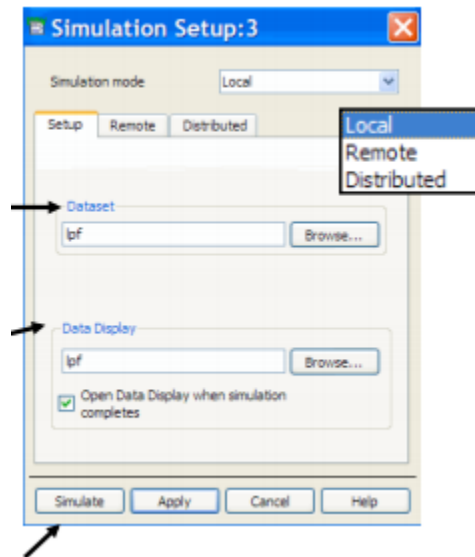
Almost ready to simulate... →

Now run the simulation as given below

- Click: **Simulate > Simulation Setup:**

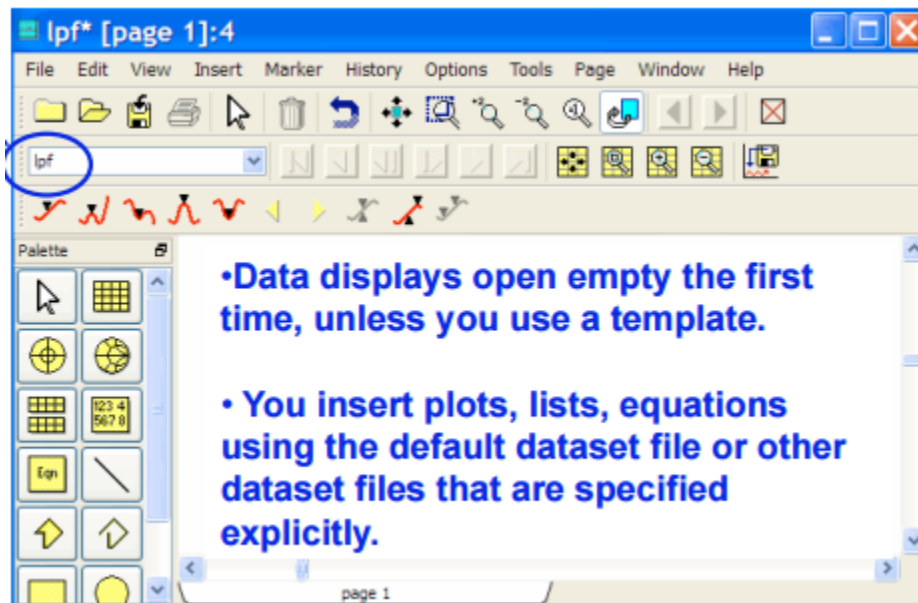
- Before you simulate:
- You can name the dataset file.
- If not, default dataset = schematic name.

Dataset files (.ds) are in the DATA directory. Data Display windows (.dds) are in the PROJECT directory.



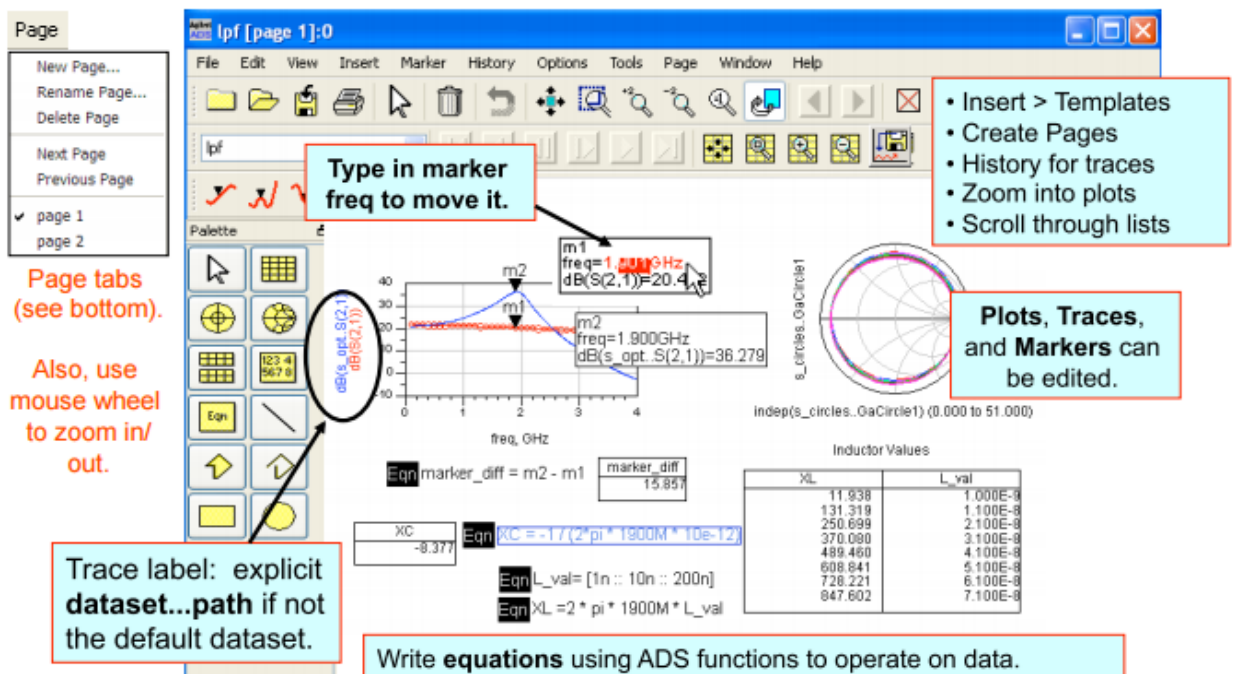
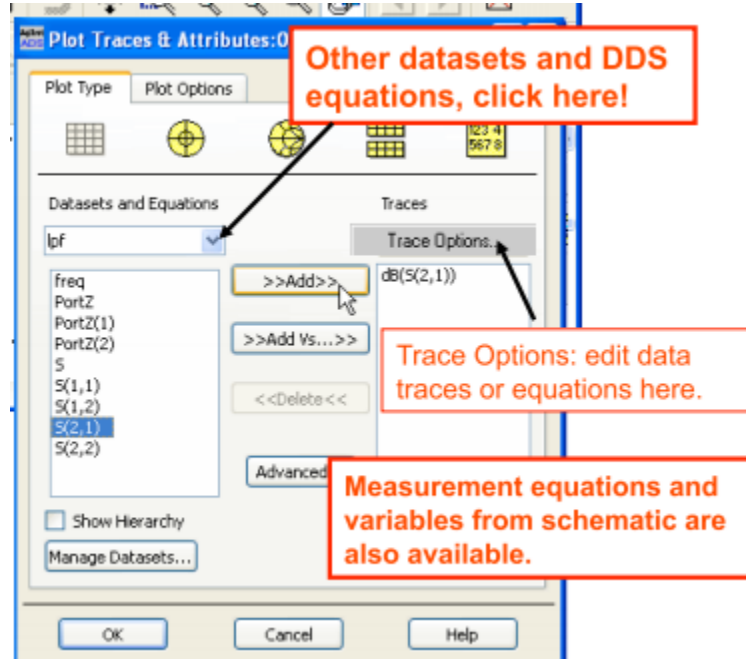
DDS data display window Example:

- Data displays open empty the first time, unless you use a template.
- You insert plots, lists, equations using the default dataset file or other dataset files that are specified explicitly



List or plot simulate data:

- Insert the plot, list or equation.
- Select the data or equations.
- Options edit data or plot.
- Save/name the DDS window



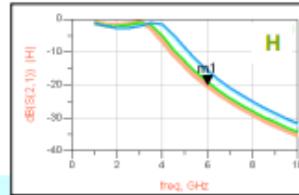
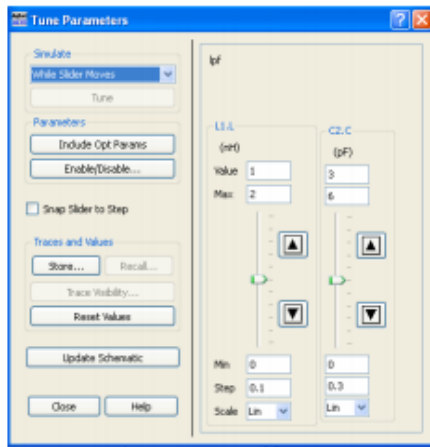
TUNING PARAMETERS:

Simulate> Tunning

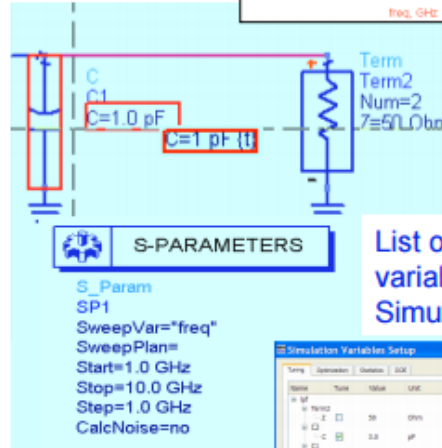


Simulate > Tuning...

Select parameters, tune, and watch the results!



H: means the trace history is turned on.



List of available simulation variables: Simulate > Simulation Variables Setup:

Name	Tune	Value	Unit	Format	Min	Max	Step	Type
R1	<input checked="" type="checkbox"/>	50	Ohms	Decimal				
C1	<input checked="" type="checkbox"/>	1.0	pF	Decimal	0.0	1.0	0.1	pF
C2	<input checked="" type="checkbox"/>	1.0	pF	Decimal	0.0	1.0	0.1	pF

Lab Task:

Design and simulate RC circuit with more than 5 combinations and adjust cutoff frequency at 2GHz.