Advanced Design System

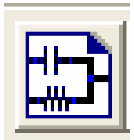
Lecture：劉益良、張訓豪

Date：2005/08/18

Course Topics

- 1:Circuit Simulation Fundamentals
- 2:DC Simulation and Circuit Modeling
- 3:AC Simulation and Tuning
- 4:S-Parameter Simulation and Optimization

Here is ADS Simplified: 3 steps



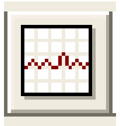
STEP 1: design capture

Insert circuit & system components and set up the simulation.

STEP 2: Simulation

Netlist is automatically sent to the simulator.

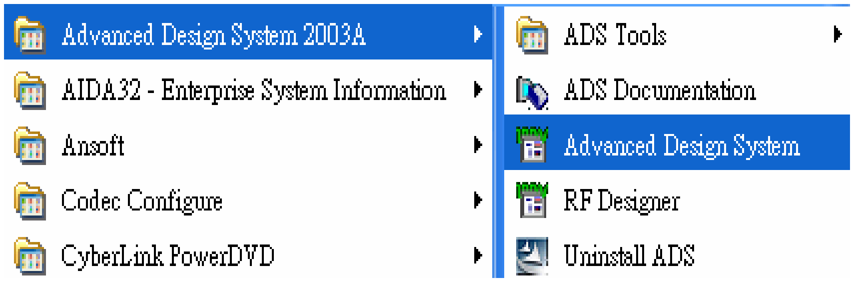
Simulation results(data) are written to a dataset.

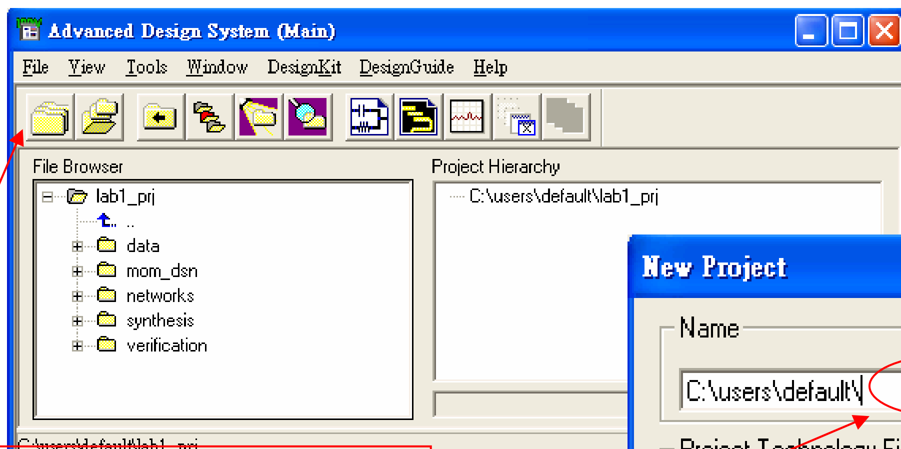


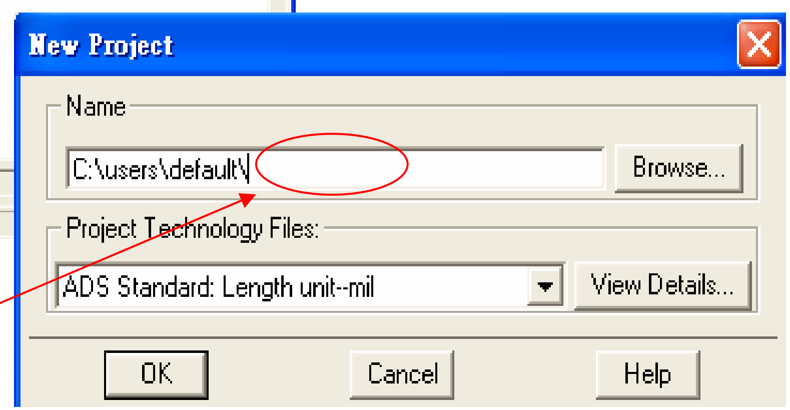
STEP 3: display the results

Plot or list data & write equations.

Starting ADS and creating a project

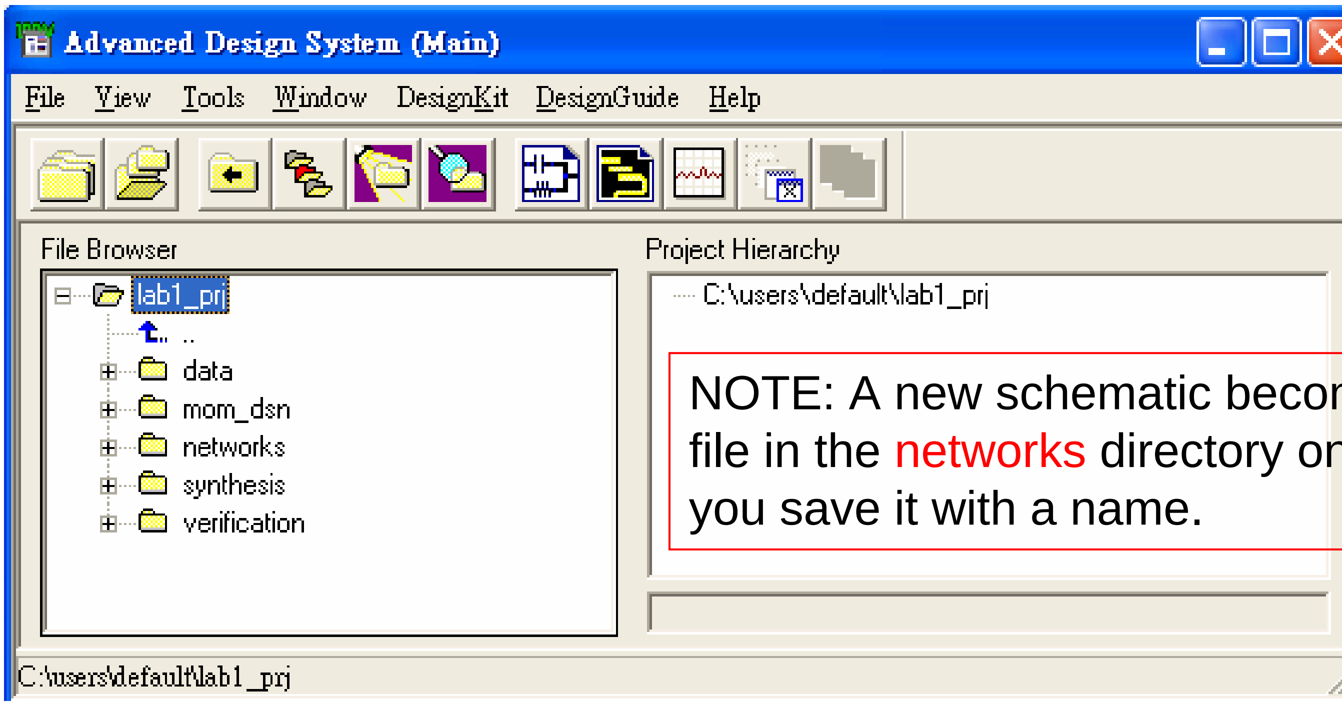






To create a new project: click the folder icon or File > New Project and name it.

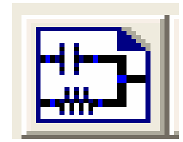
Project directories are created and a blank schematic window opens!



NOTE: A new schematic becomes a .dsn file in the networks directory only after you save it with a name.

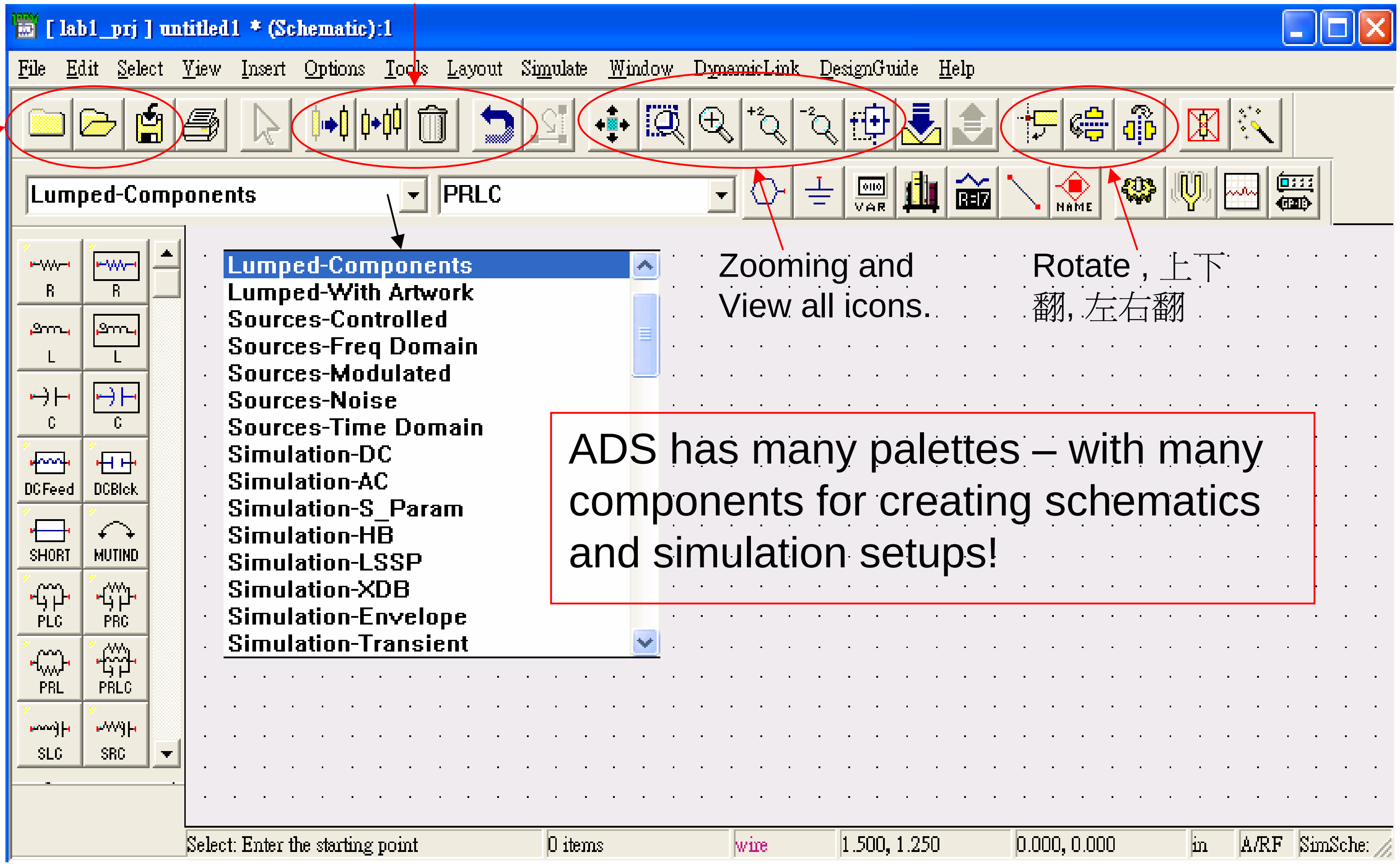
ADS automatically creates these directories for every project. But they are empty until you create the schematics, simulate to produce data, and display the results.

Schematic window



Move, copy, delete, undo

Use icons to create, open, and save designs.

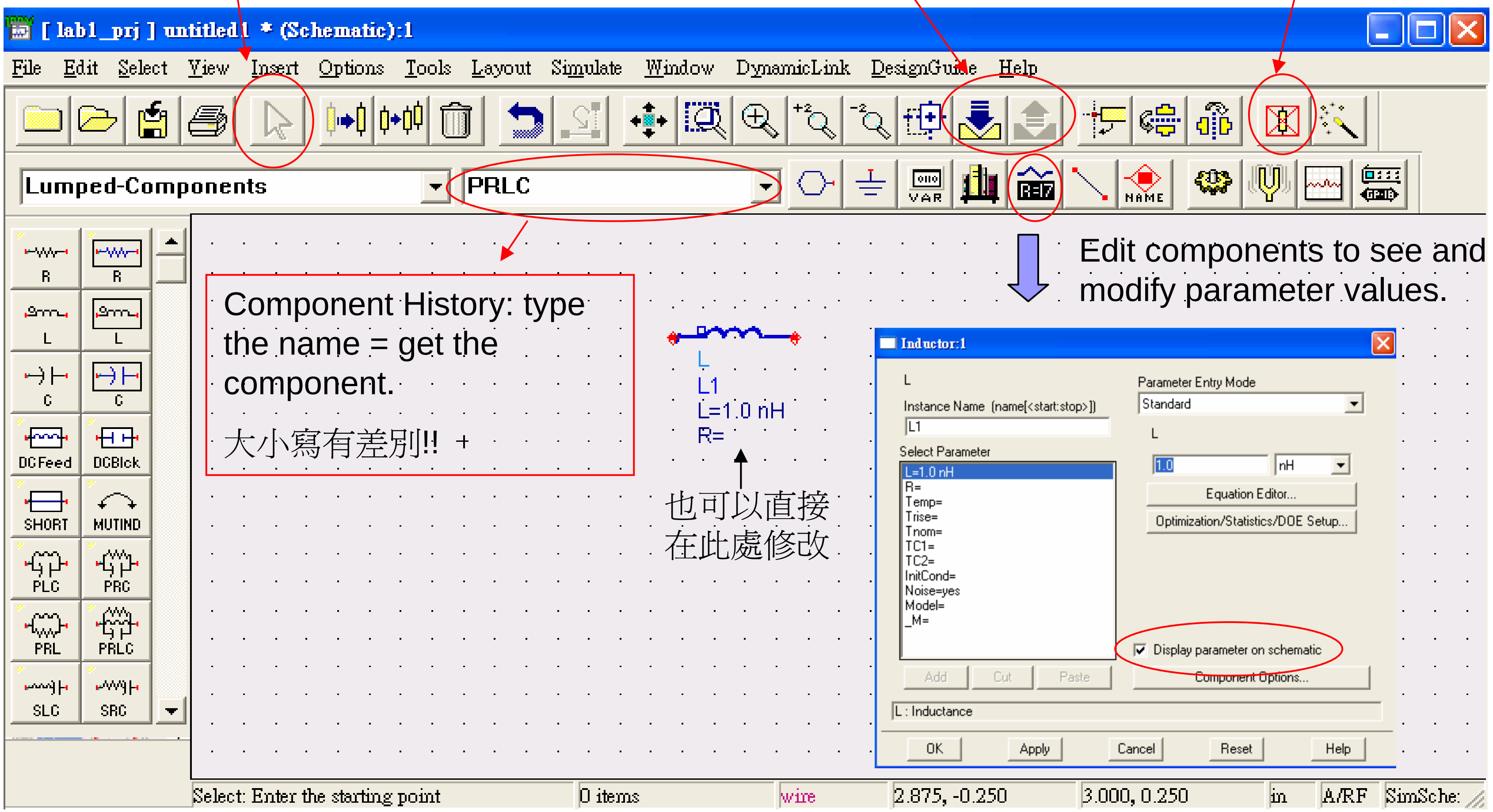


Inserting and editing components

End command or use ESC

Push / Pop for sub-circuits.

Activate / Deactivate



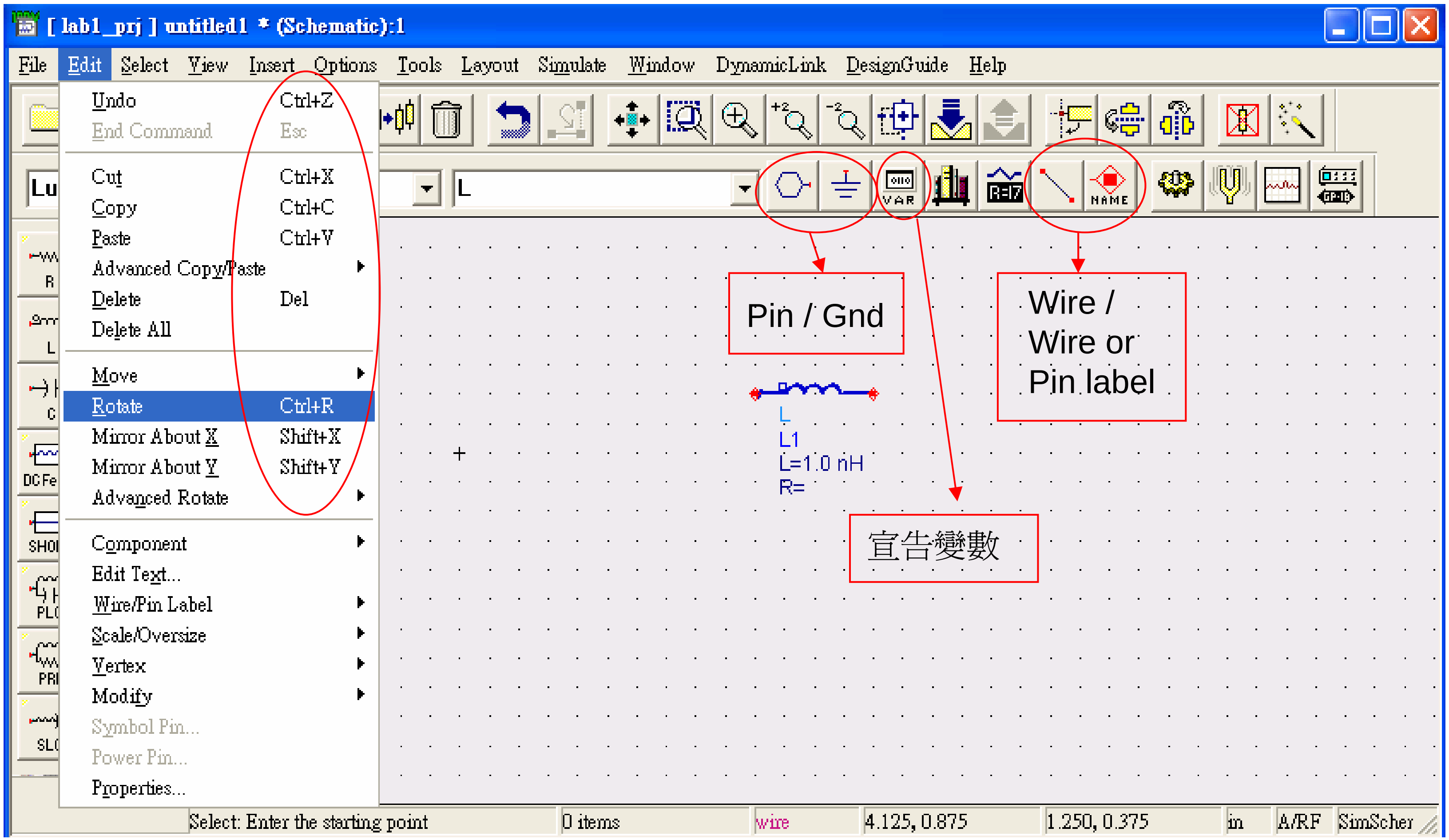
Component History: type the name = get the component.

大小寫有差別!!

Edit components to see and modify parameter values.

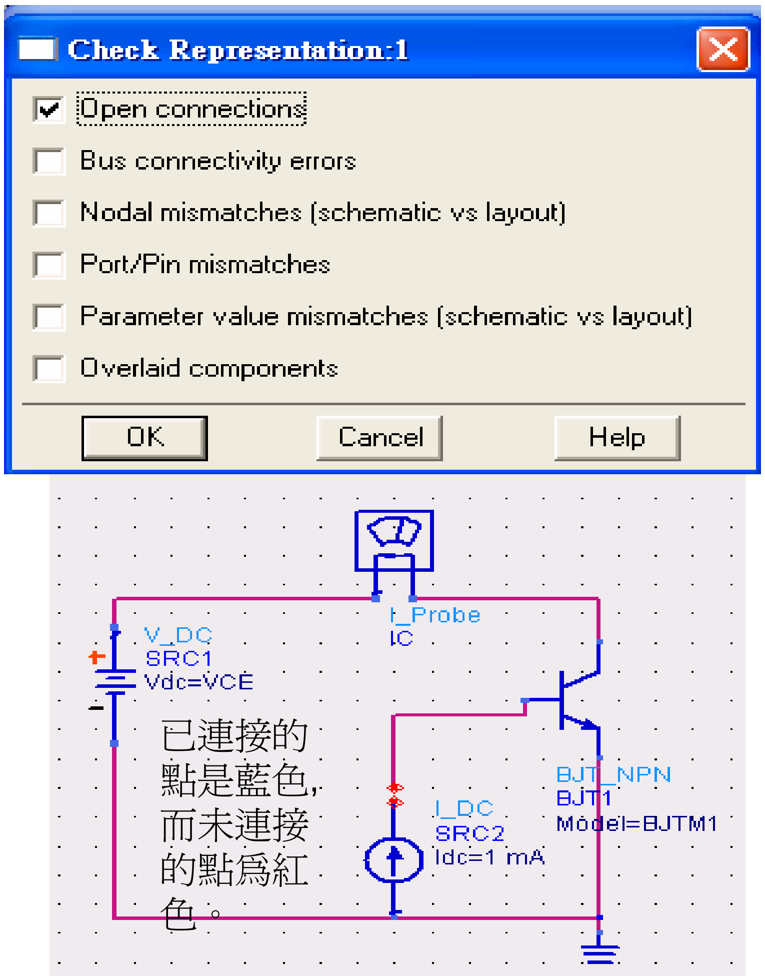
也可以直接在此處修改

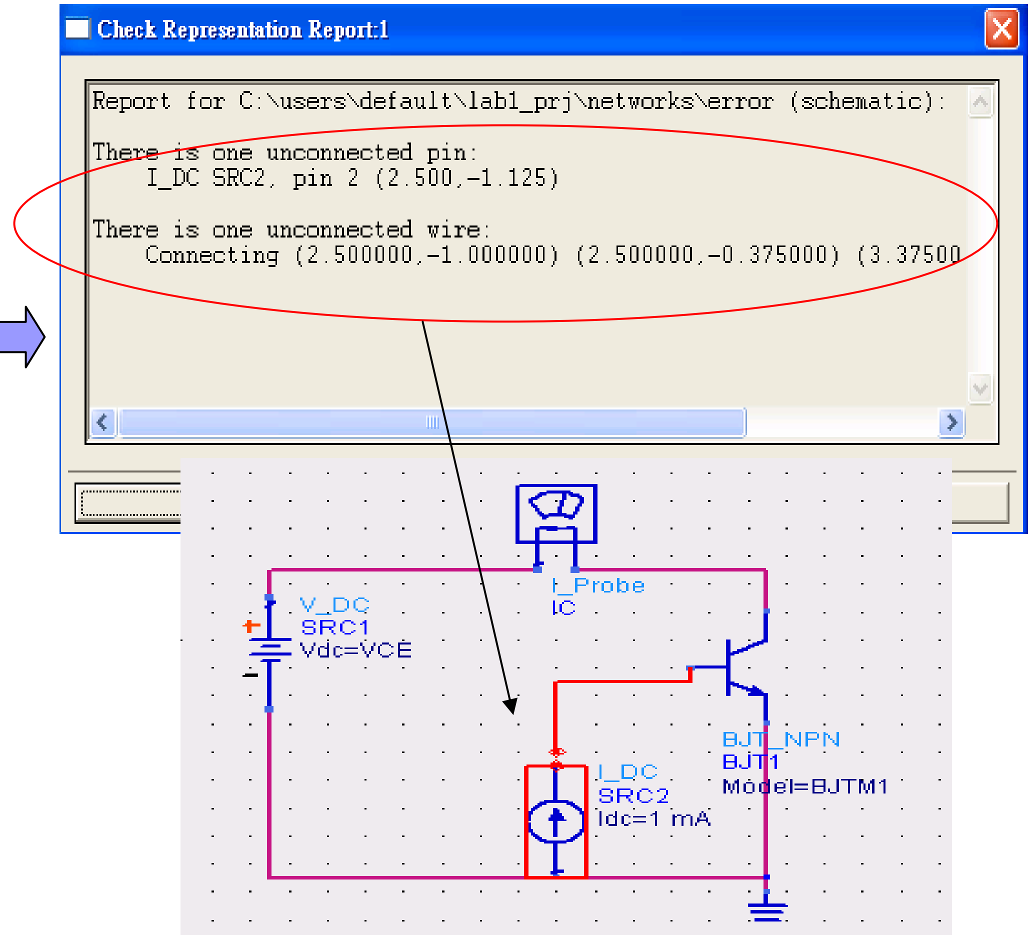
Hot keys are labeled



Check your schematic for errors

Tools > check representation

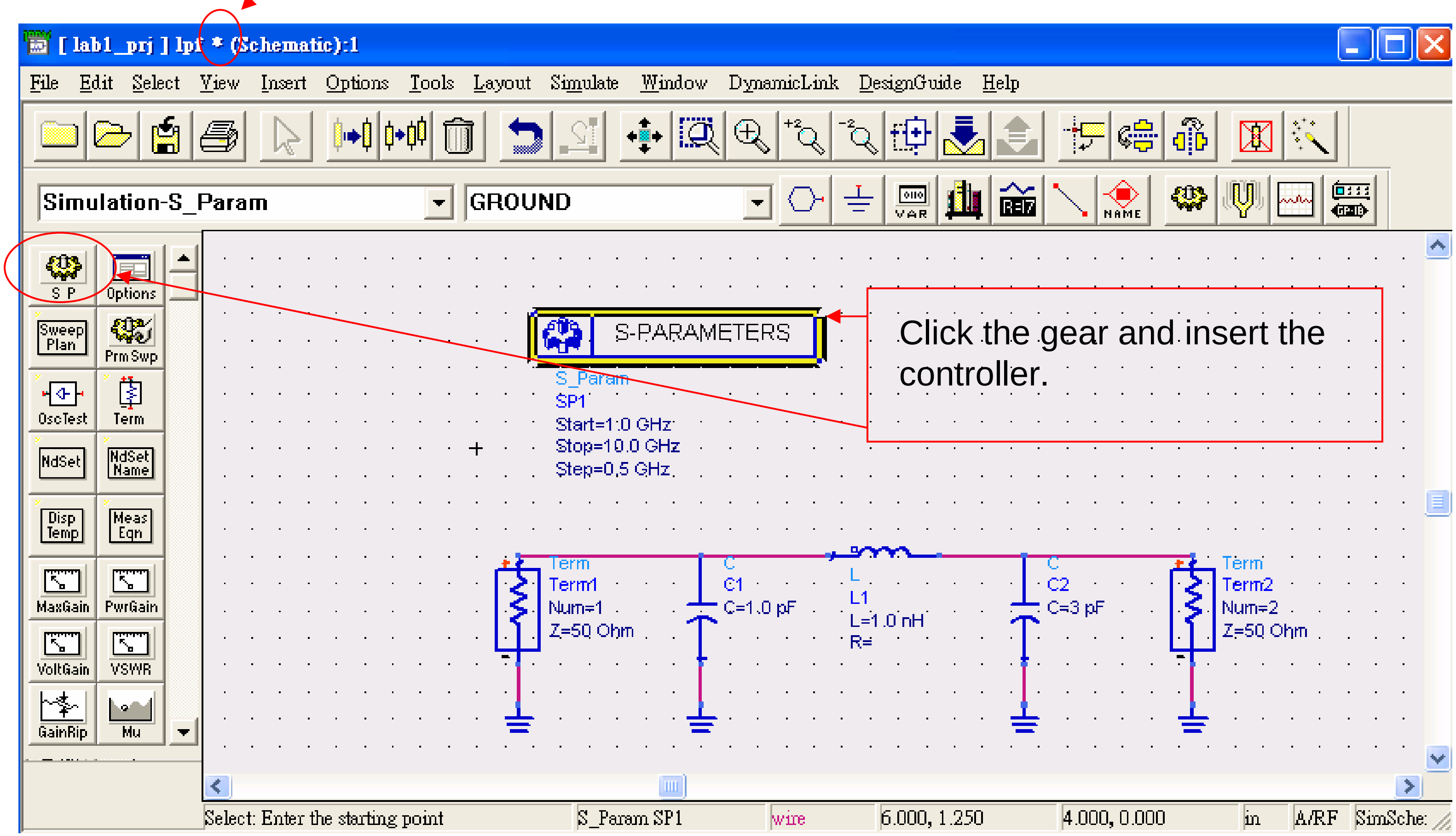




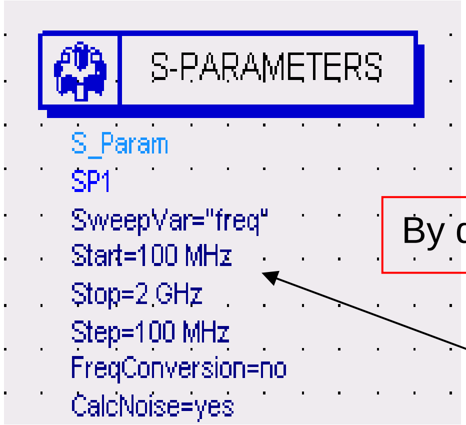
View > clear Highlighting

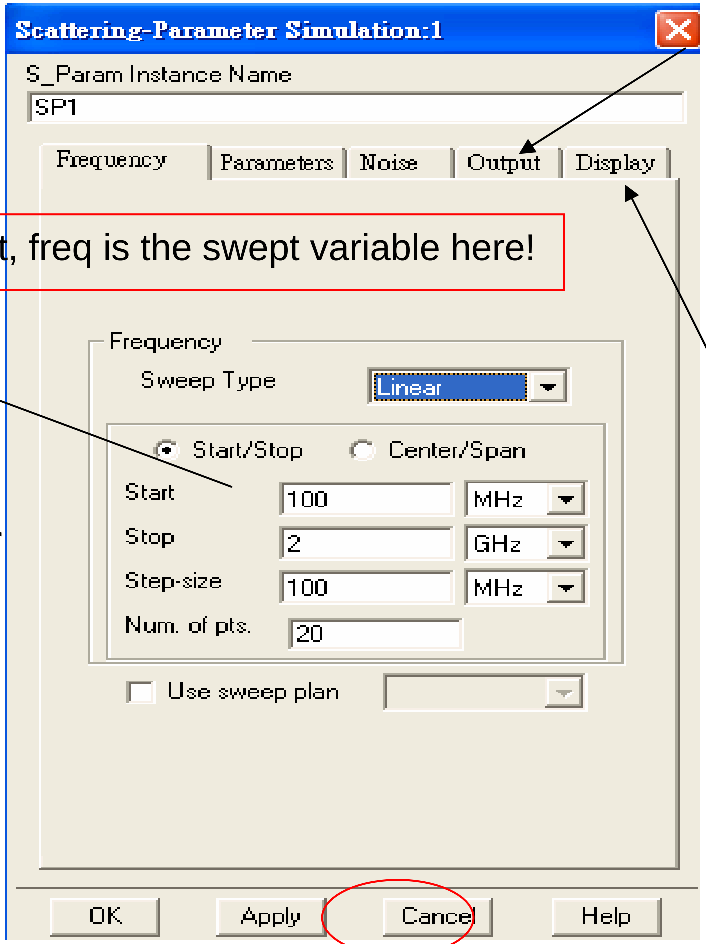
First step: insert a Simulation Controller

*NOTE asterisk means schematic is not yet saved.



Next, edit the Simulation parameters.





By default, freq is the swept variable here!

Output tab allows you to select what goes to the dataset

Display tab lists all the settings for on screen display.

Edit on-screen if the parameter is displayed or use the dialog box.

Running the simulation

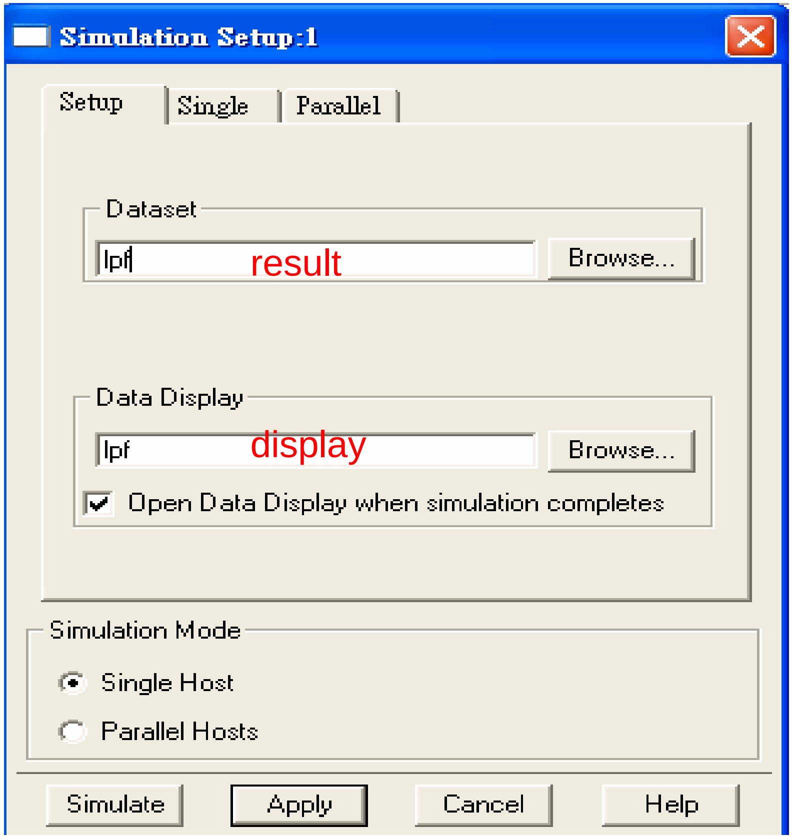
Click: Simulation > Simulation setup:

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.

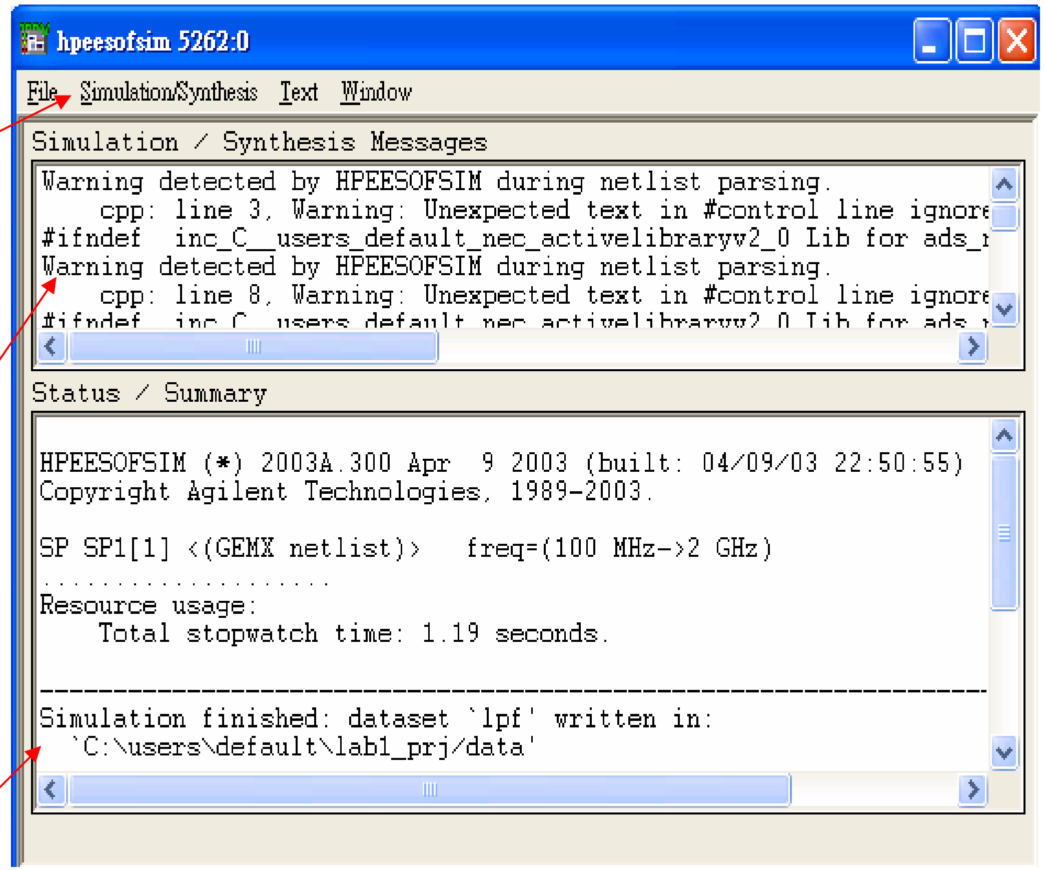


To simulate: use F7 key, click Simulate, or click the gear icon on the schematic.

Simulation information: Status Window

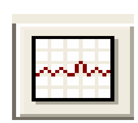
One way to stop a simulation, click:

Simulation/Synthesis > Stop Simulation



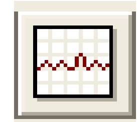
If there is a warning or error, it will appear here.

A successful simulation results in a dataset:



When finished, the Data Display opens…

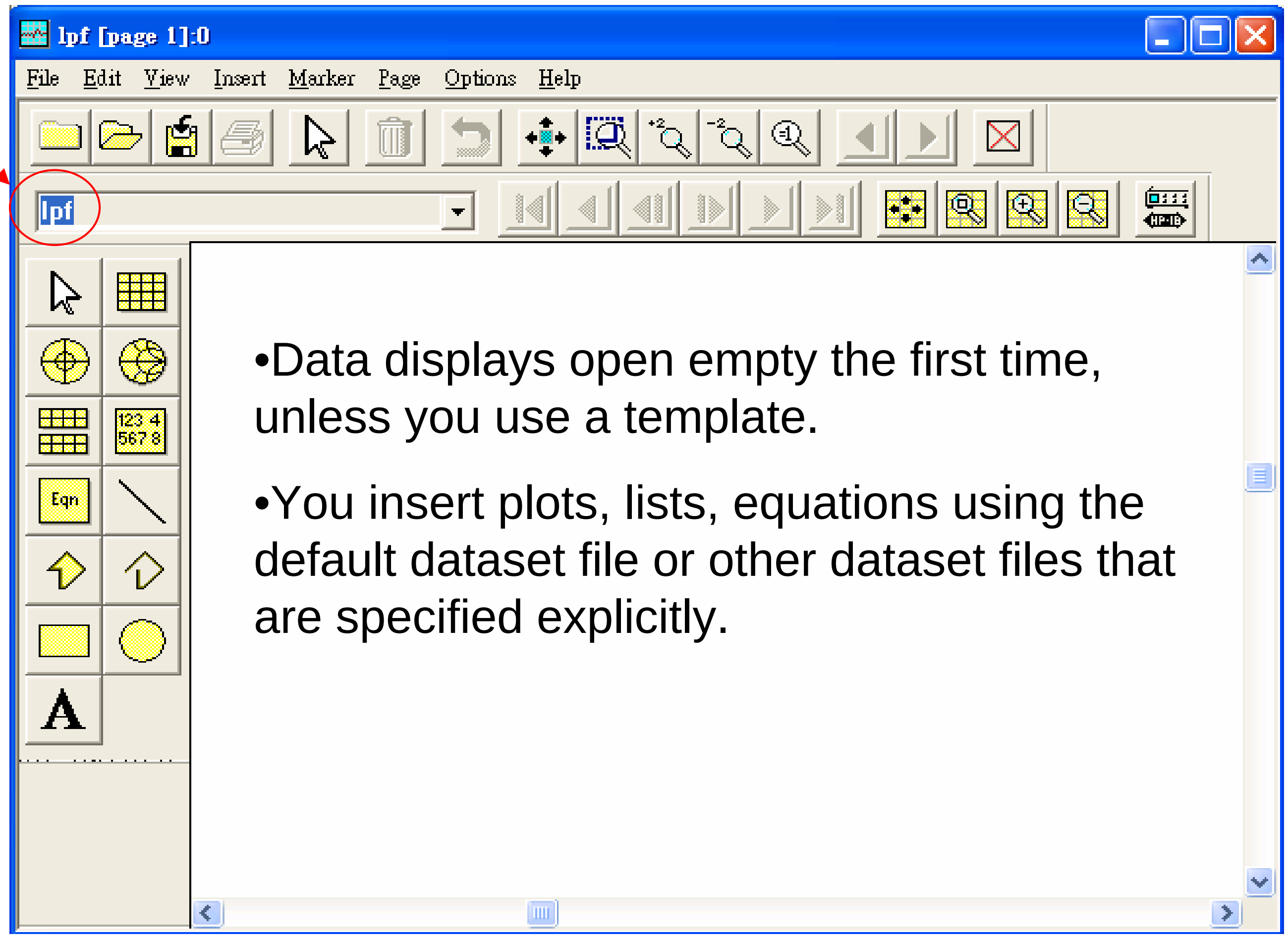
Data Display window



If automatically opens or you can open this window from any Schematic or the ADS Main window:

Default dataset

First, select a plot, list or Eqn for the data….



•Data displays open empty the first time, unless you use a template.

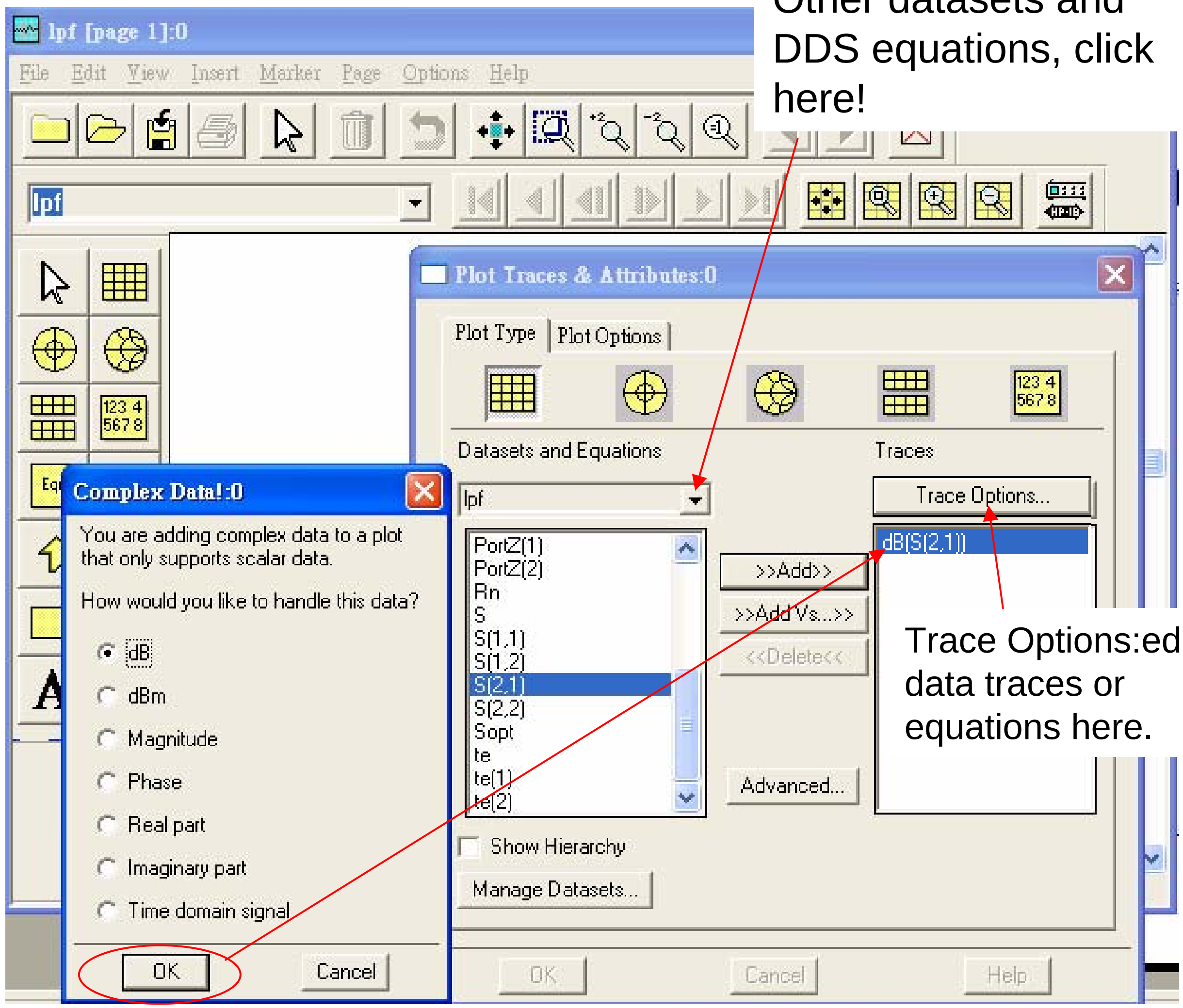
•You insert plots, lists, equations using the default dataset file or other dataset files that are specified explicitly.

List or plot the simulation data...

STEPS:

1. Insert the plot, list, or equation.
2. Select the data or equations.
3. Options – edit data or plot.
4. Save / name the DDS window.

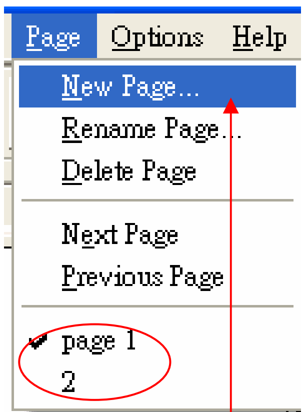
Measurement equations and variables from schematic are also available.



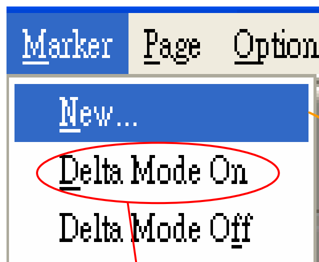
Other datasets and DDS equations, click here!

Trace Options:edit data traces or equations here.

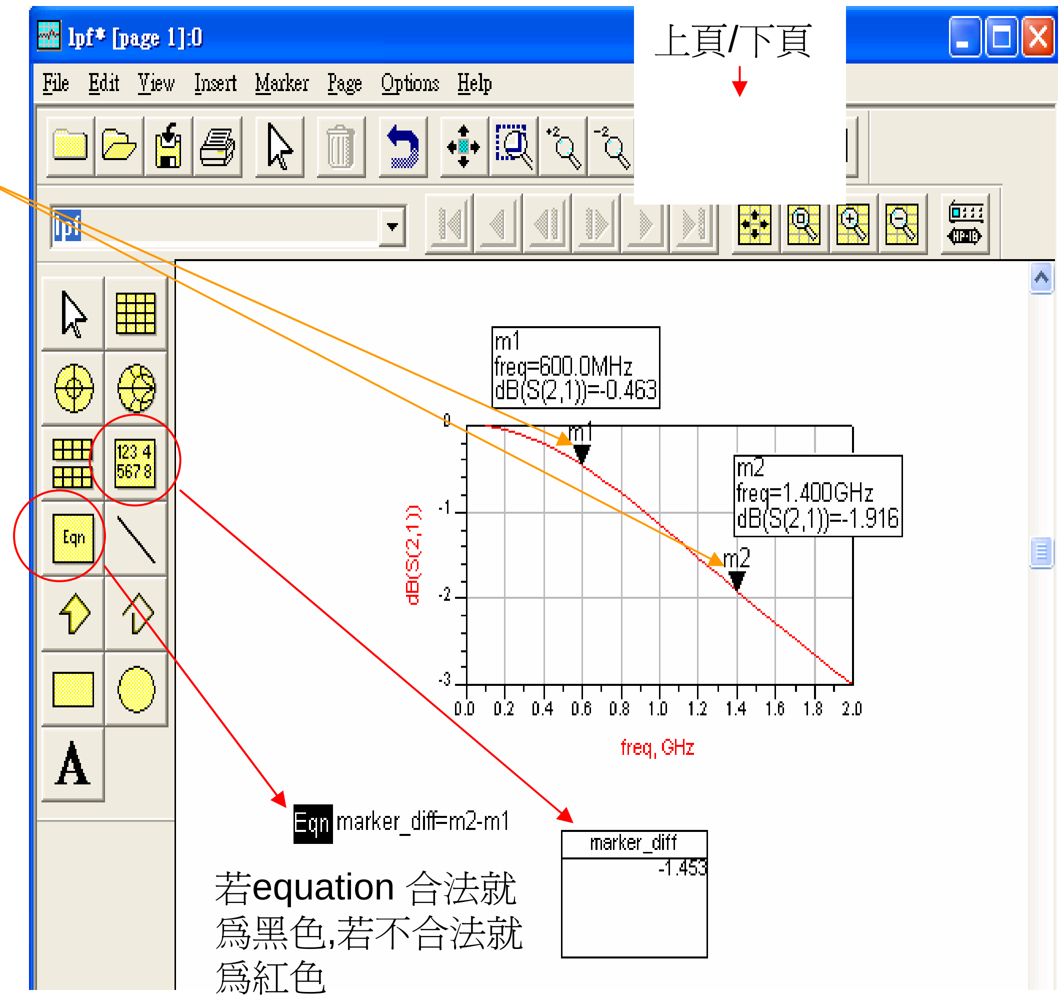
You control all your simualtion data



資料多時可分頁

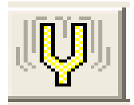


可將兩marker的資料相減



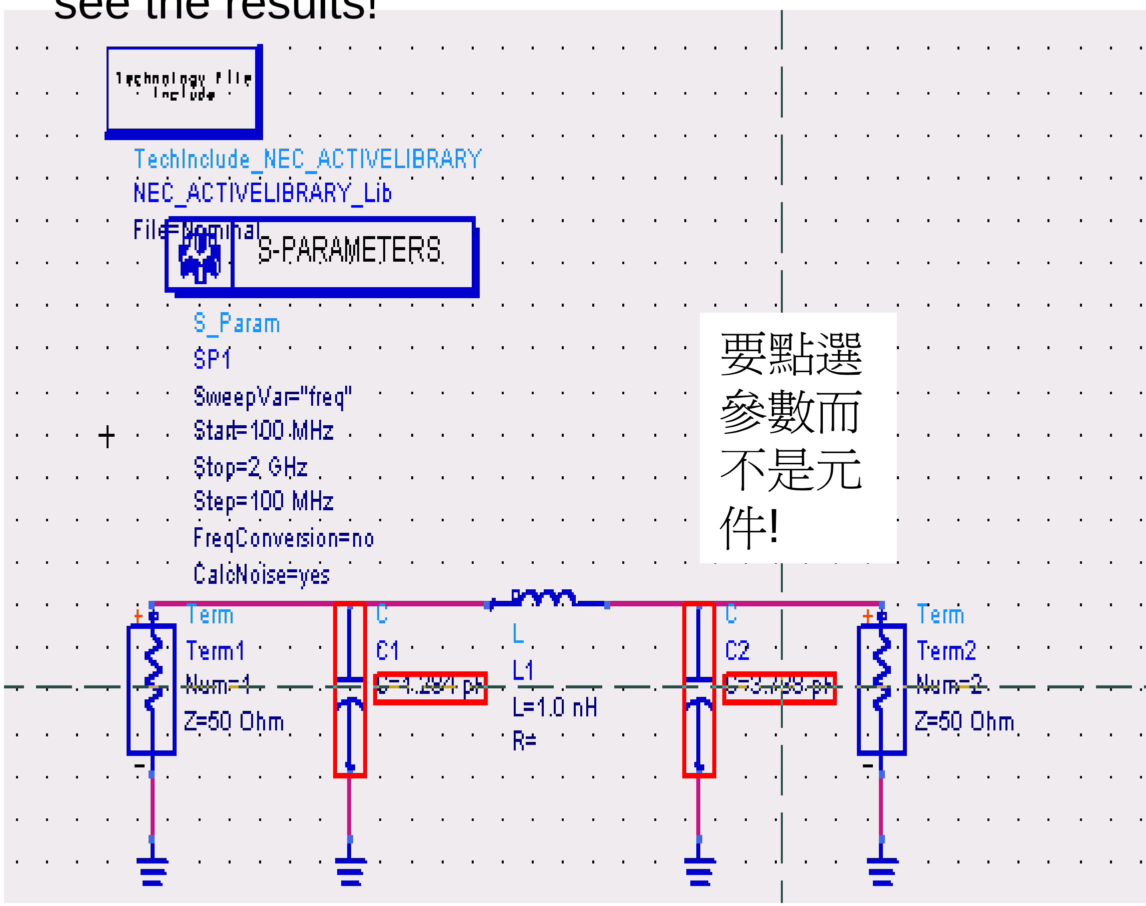
- Insert > Templates
- Create Pages
- Zoom into plots
- Scroll through lists

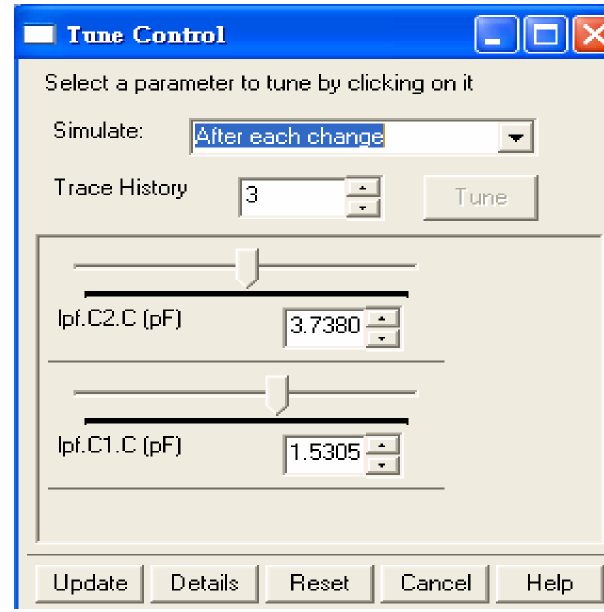
Tuning Parameters: tune mode = simulation

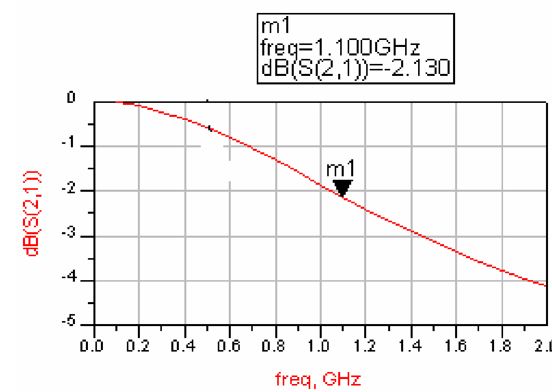


Simulate > Tuning…

Tuning allows you to tweak parameter values and see the results!

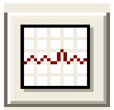




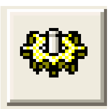


Review: contents of an ADS Project Directory

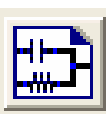
Project



.dds files (data display server): Data display windows you create to display simulation data. You cannot see these in the Main window. (在project下)



data directory contains .ds files (datasets). This is the simualtion data.

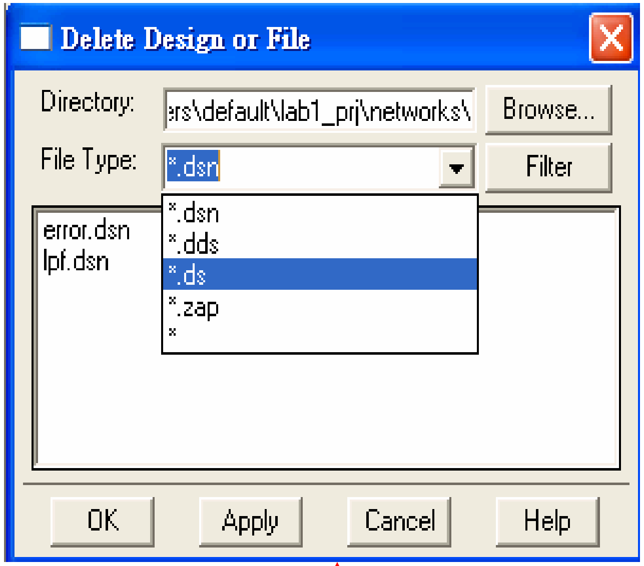


Networks directory contains .dsn files (designs). These are layouts and schematics with simulation setups.

Preference files & ADS netlist.log

- mom_dsn (Momentum)
- Substrates (Momentum)
- Synthesis (used for E-Syn & DSP
- Verification (used for DRC)
- notebook

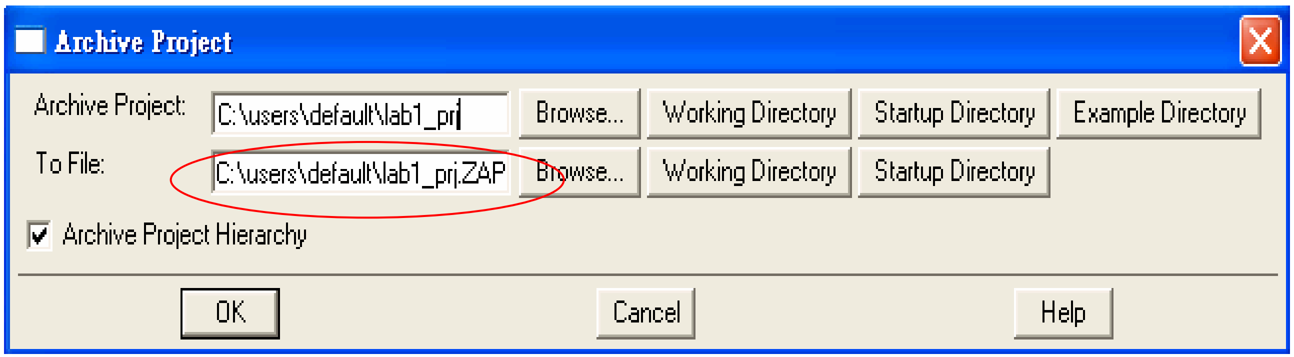
These miscellaneous directories are not required for most circuit simulations.

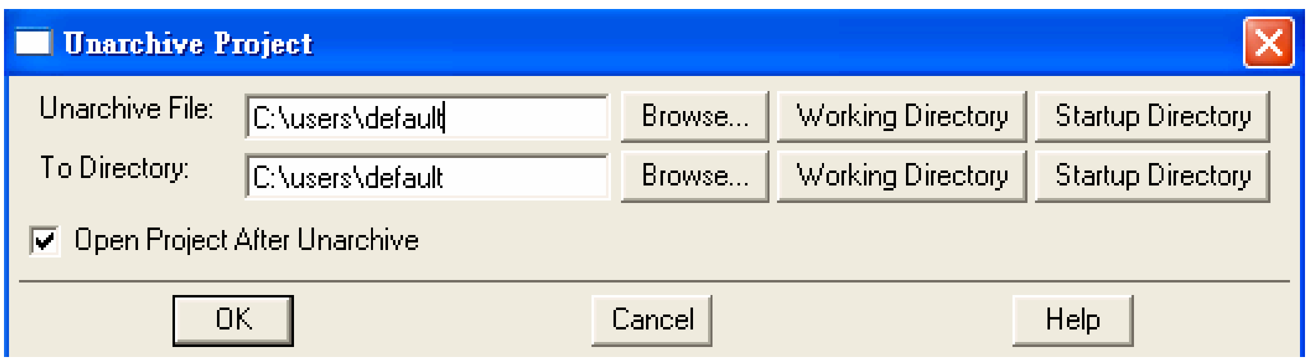


To delete ADS files, use the Main Window command:
File > Delete Design

ZAP your projects for e-mail or disk

From the Main window, click: File > Archive or Unarchive





NOTE: Archive files become .ZAP files.

They can include all networks, data, and display files (entire project).

Course Topics

- 1:Circuit Simulation Fundamentals
- 2:DC Simulation and Circuit Modeling
- 3:AC Simulation and Tuning
- 4:S-Parameter Simulation and Optimization

DC Simulation

- You get steady-state DC voltages and currents according to Ohm's Law: V=IR

•Capacitors = treated as ideal open circuits

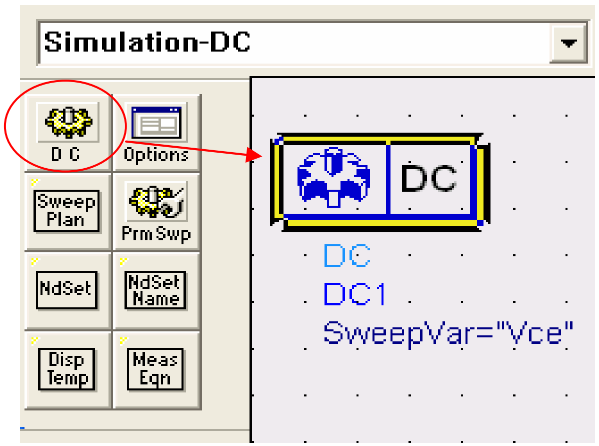
•Inductors = treated as ideal short circuits

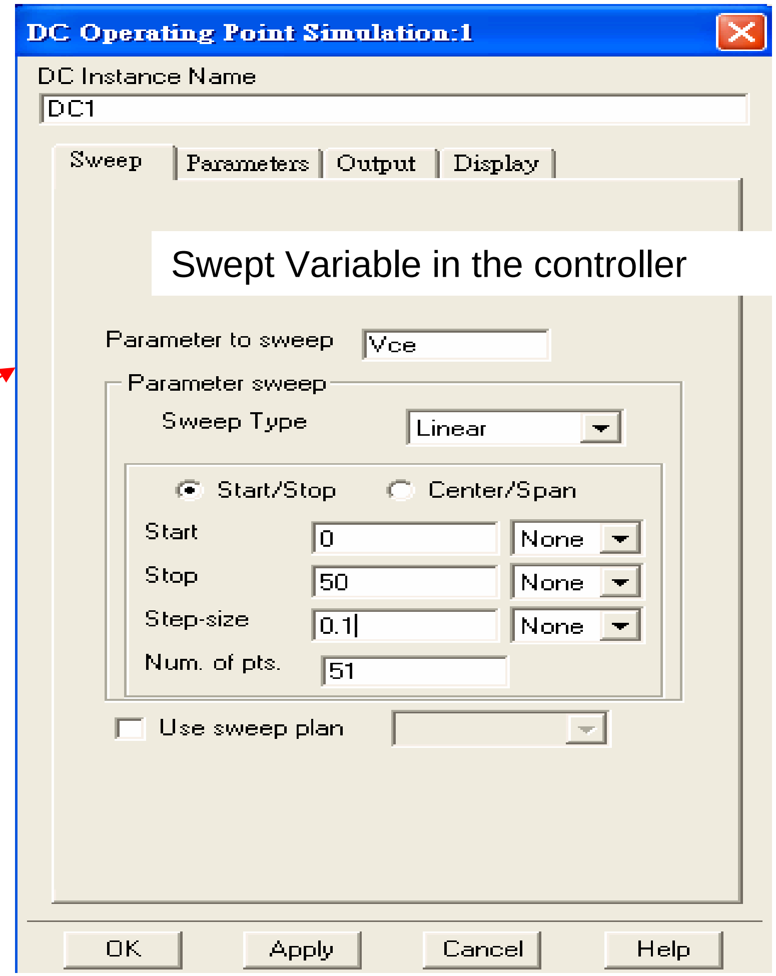
•Topology check: dc path to ground (if not ➔ error message)

•Kirchoff's Law satisfied: sum of node current = 0

DC simulation controller

Palette and editor (dialog box)

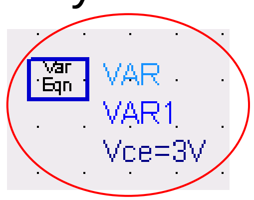




Swept Variable in the controller

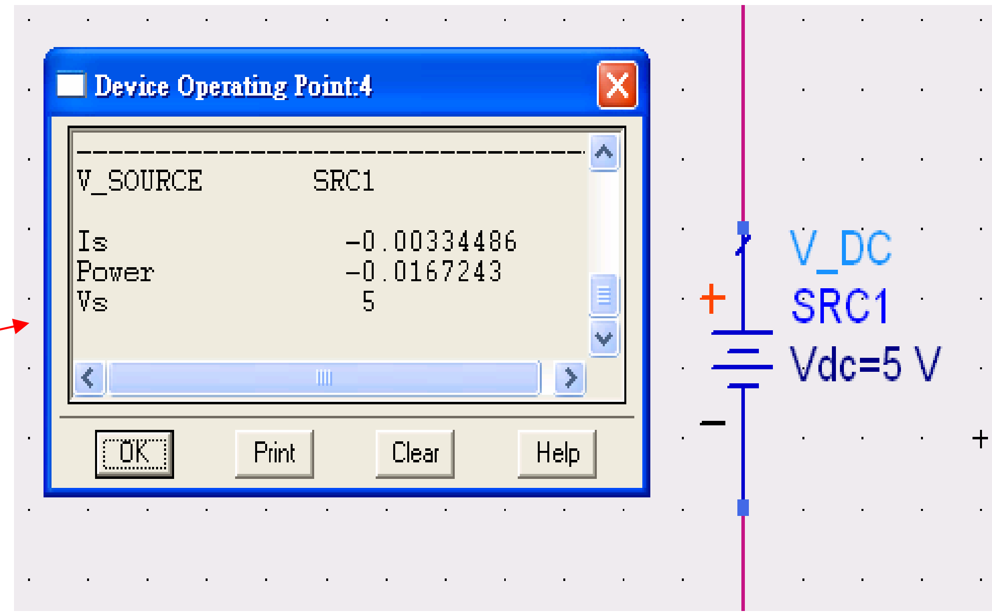
Sweep: allows you to sweep a parameter but it must be declared as a variable. Note the dialog entry automatically puts quotes on the controller (screen) entry.

VAR



Initialize Vce!!

DC Parameter



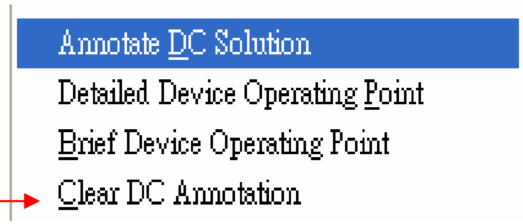
You can get V, I, and Power!

Available after simulation on schematic.

Schematic Annotation of DC values

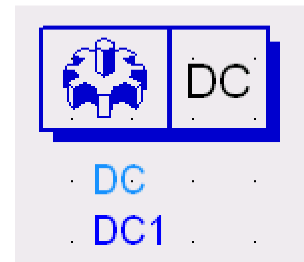
Immediately after DC simulation, click: Simulate > Annotate DC Solution.

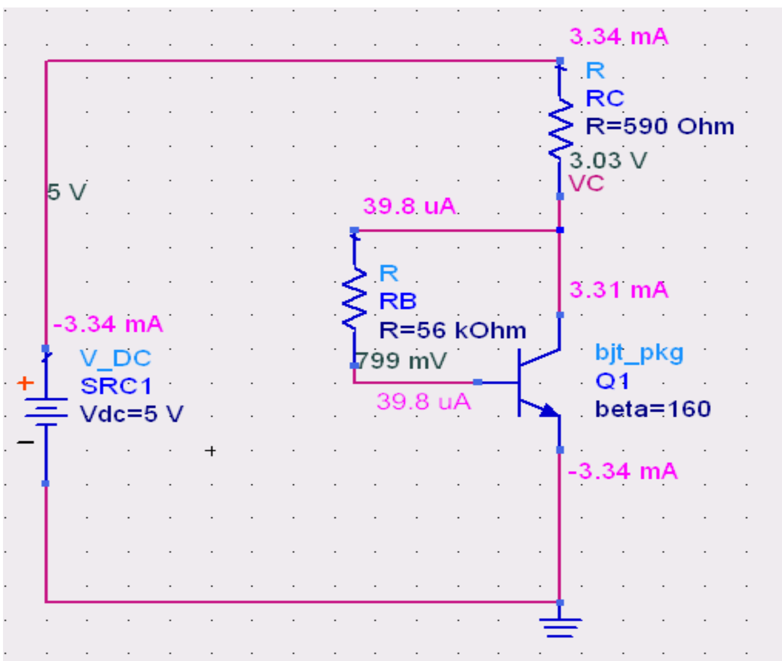
Simulate >



Clear it here

No controller settings necessary!

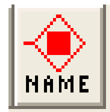




Minus sign used for current flowing out of a connection. Otherwise, current flows into a connection or device.

DC Simulation Controller is required in all simulations if you want DC annotation.

Wire/Pin Labels (node names) in schematic



To label a node, use the icon:

•Type in the name, point and click.

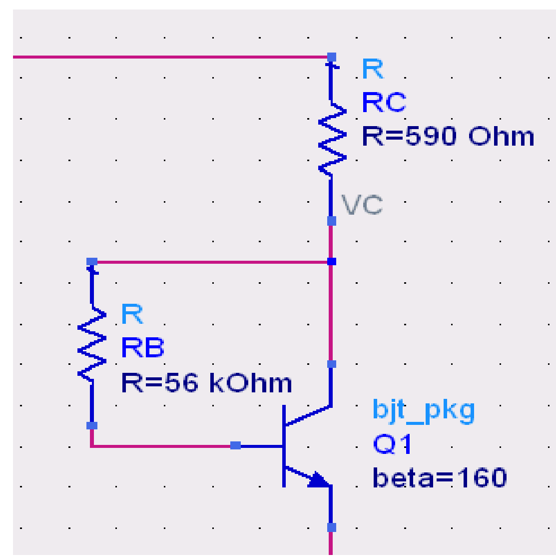
•You get node voltage in the dataset

•Use these in equations: dBm (Vout).

•Connect two pins by name – without a wire.

•Move and edit the label names (attributes).

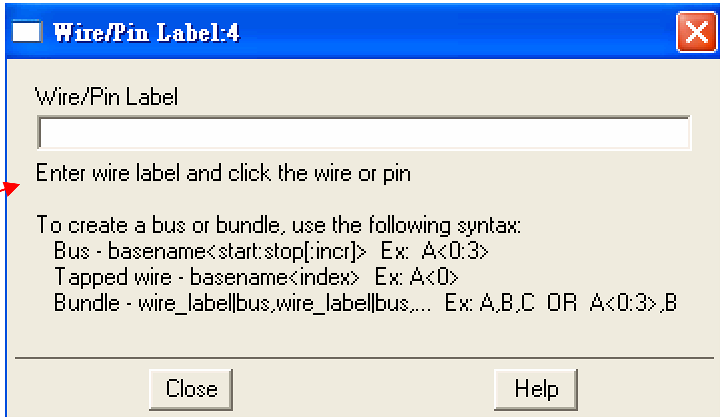
•You can also create busses.



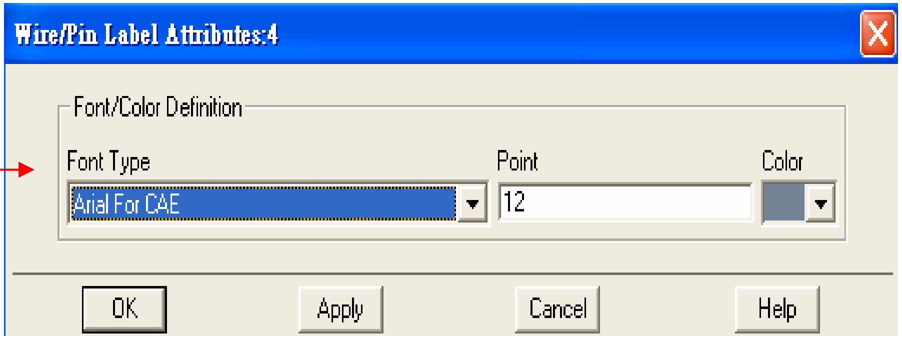
Wire/Pin Labels (node names) in schematic

To remove a label, use the icon:
With a blank (no name), click on the node.

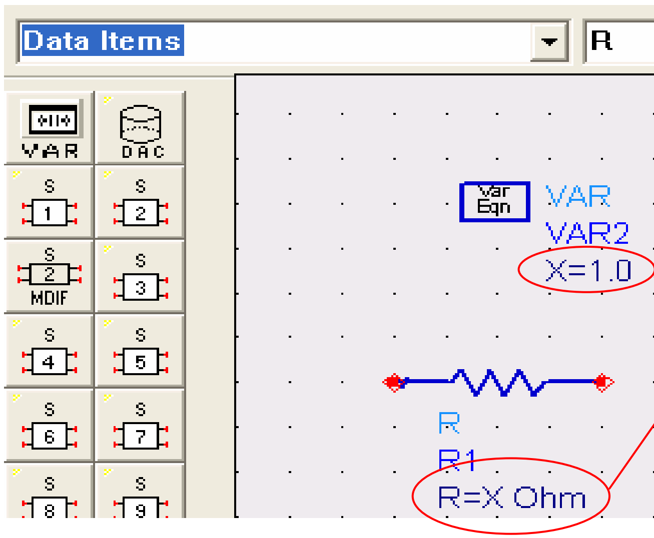
Or Edit > Wire/Pin Label > Remove Wire/Pin Label



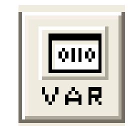
To edit the label, double click it, Or use the command: Edit > Wire/Pin Label Attributes

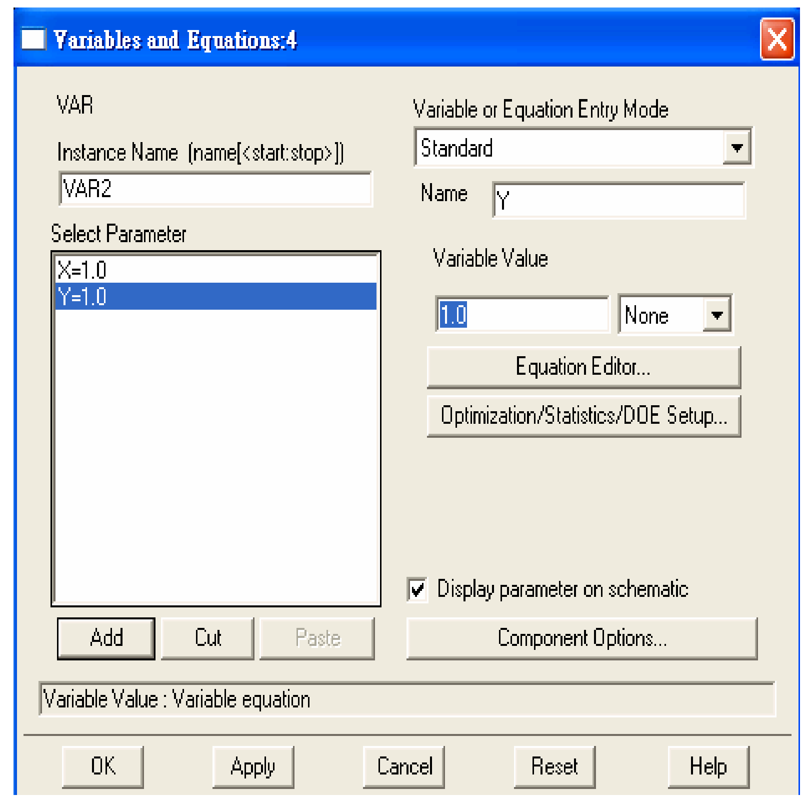


Variable Equations: VAR



兩邊至少要有一邊有單位

Click: 

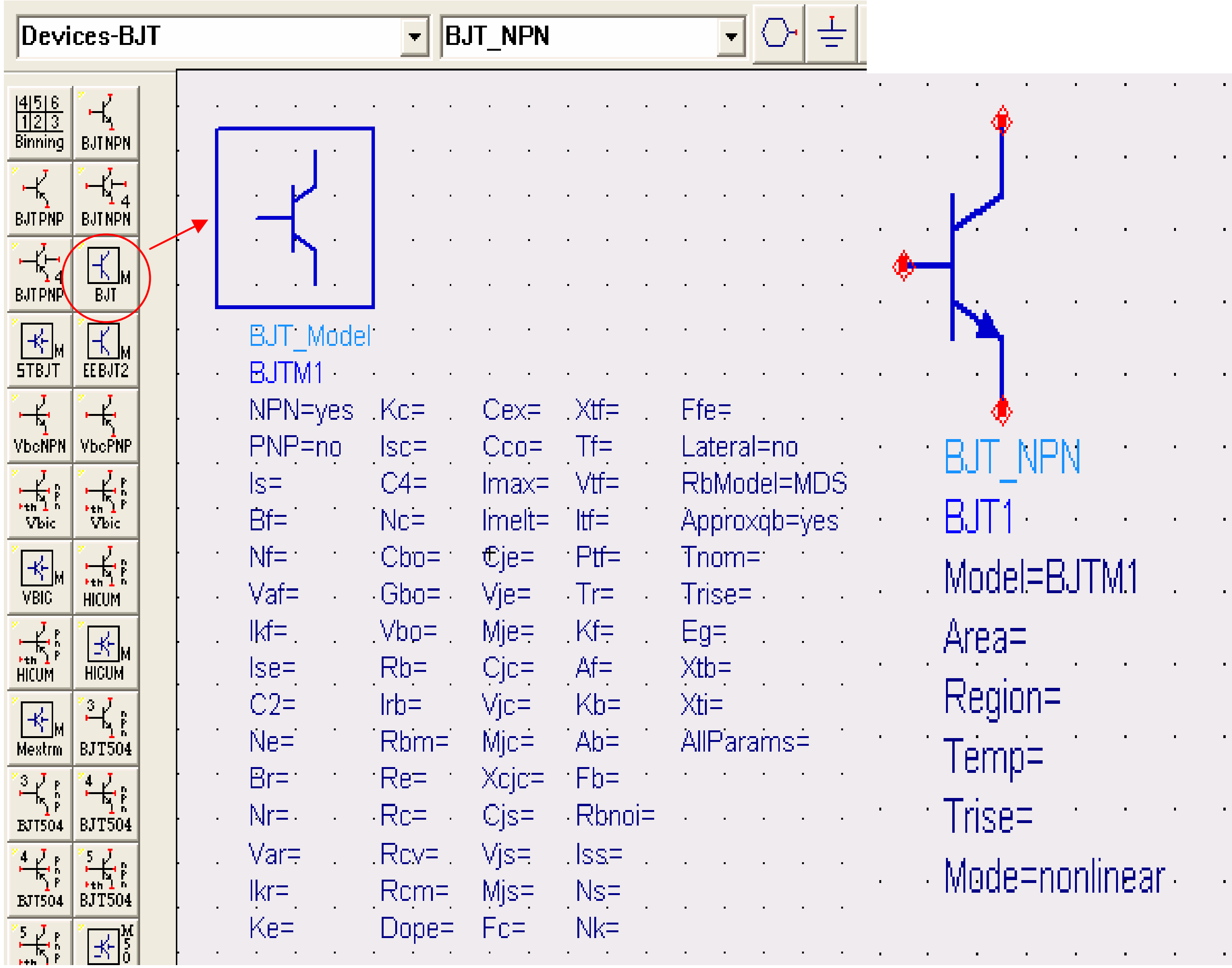


Component parameters can be assigned to a varable: VAR.

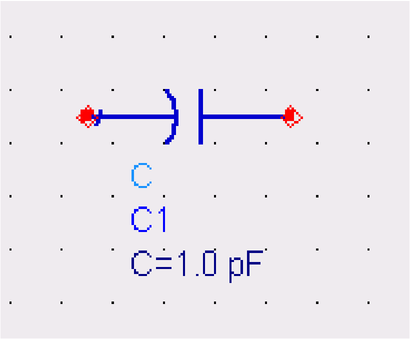
VARS can be used with optimization, parameter sweeps, and other applications!

Analog Models

Model Cards: use a built-in symbol and model card which lists all the parameters that you can modify. This example is a BJT model.



Symbols, units, names, case sensitivity



Example of on-screen control:

C (component name): changes the component
C1 (instance name): rename it
C= (parameter): a number (unit) or valid variable.

Case Sensitivity:

UNIX is always case sensitive:

PC is usually case sensitive:

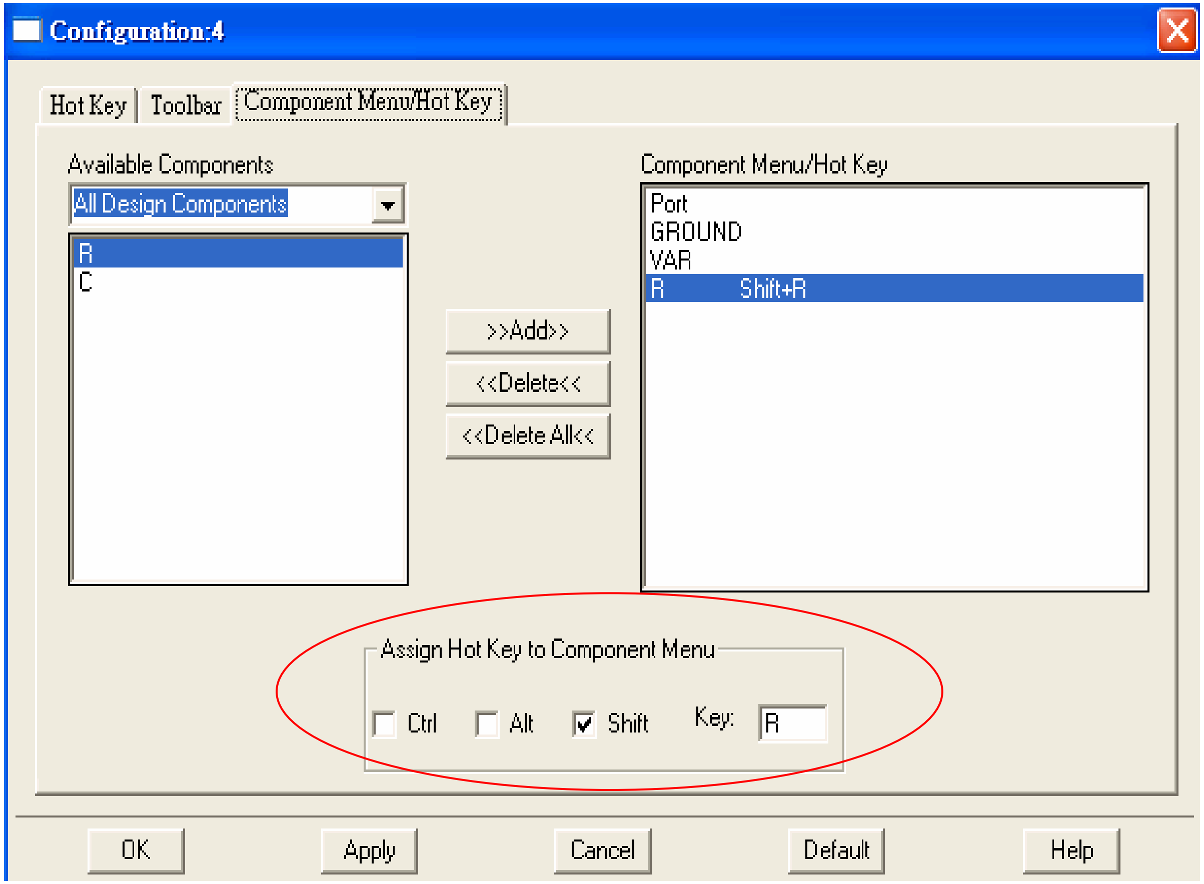
•For example, insert R – after the first insert, PC will recognize either r or R.

•But m=milli, M=mega, V=volts, and VARS are case sensitive all of the time!

Hot keys (key binding) for components

Tools > Hot Key / Toolbar Configuration > Component Menu / Hot Key.

In this tab, you set Hot Keys for components, library items, controllers, and sources.



ADS default command Hot Keys

Pre-configured keys:

F7 = simulate

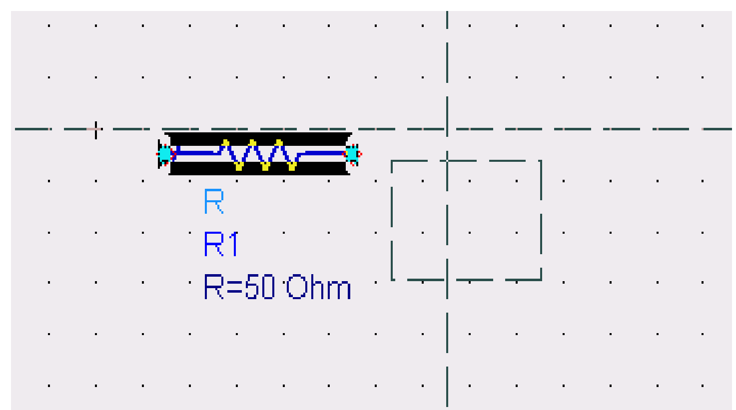
F5 = Move Component Text

Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V

If you don't like mouse clicks, HOT KEY your keyboard.

Its global for all projects.

Try this now: click the F5 key, select the component, move the cursor and the text will follow!



NOW: Set up a Hot Key in a new project

STEP 1: Tools > Hot Key / Toolbar Configuration

STEP 2: Set up a hot key for the Simulation setup → a. Select the command.

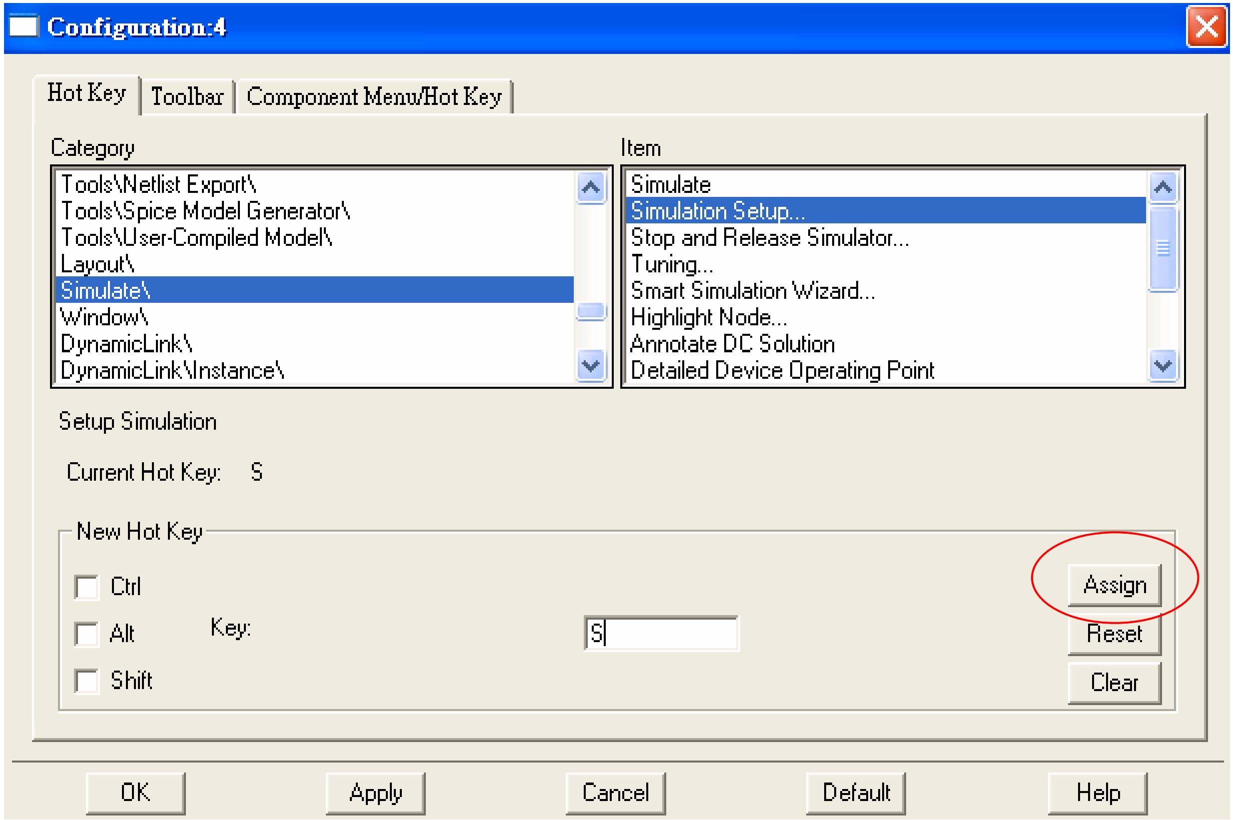
b. Type in the letter: S

(not case sensitive).

c. Click: Assign & Apply.

d. Press the S key to verify

it works.



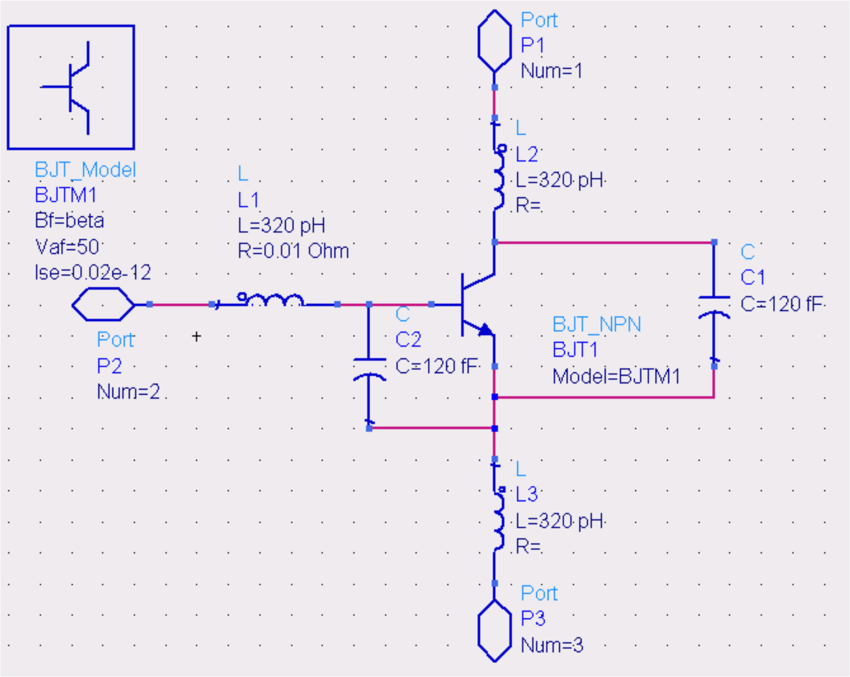
Viewing and creating a schematic symbol

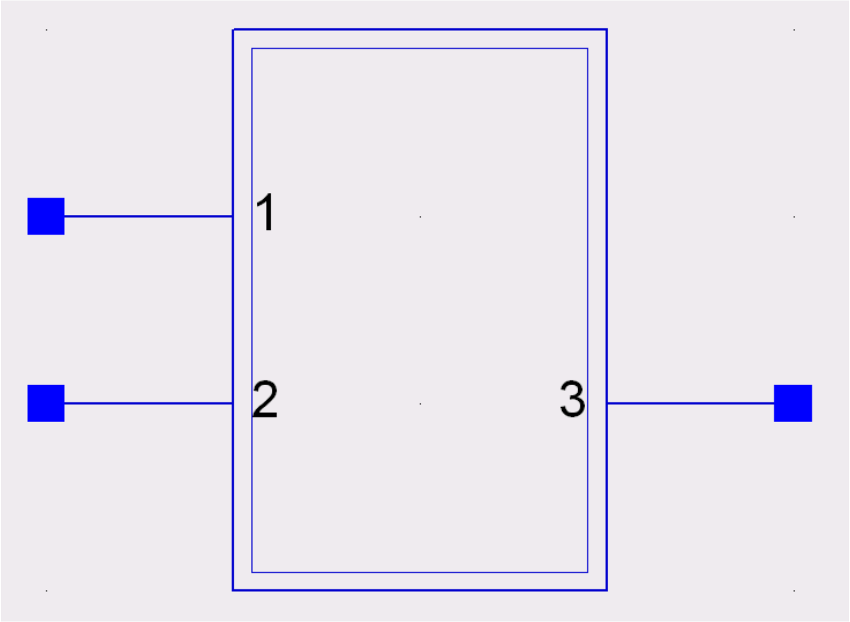
Schematic view

View > Create / Edit schematic Symbol

Symbol view

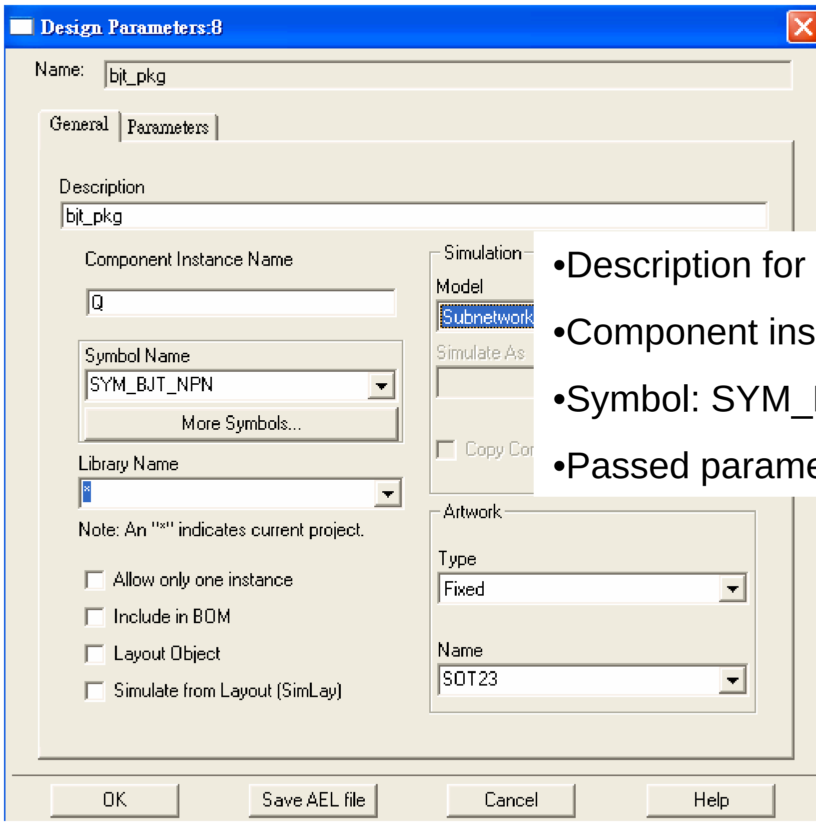
View > Create / Edit Schematic



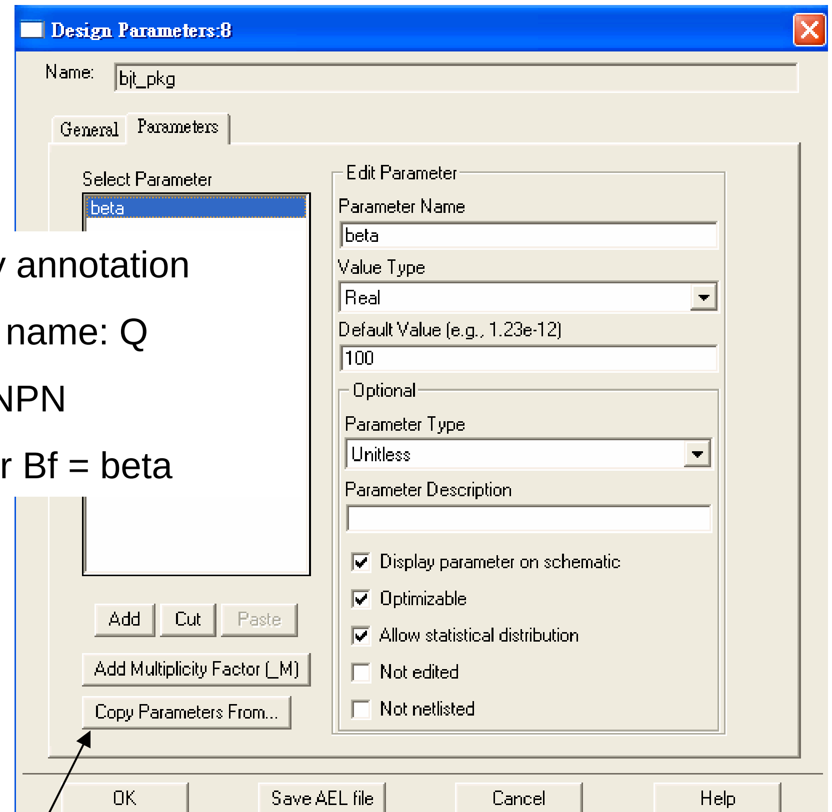


Design parameters for any schematic

Click: File > Design / Parameters to set parameters for your design:

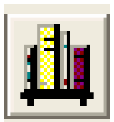


- •Description for library annotation
- •Component instance name: Q
- •Symbol: SYM_BJT_NPN
- •Passed parameter for Bf = beta

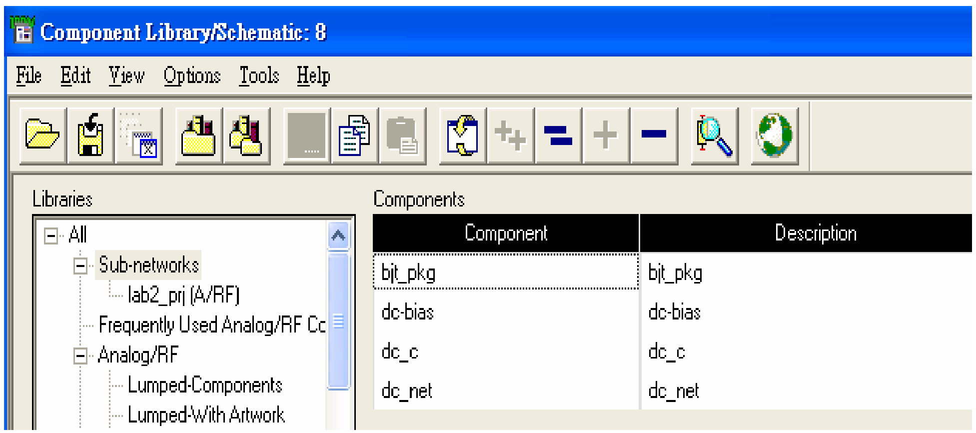


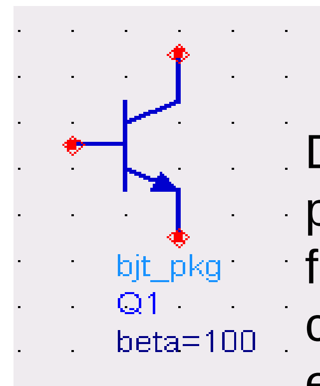
You can copy parameters from other library models.

Insert the model in a new schematic

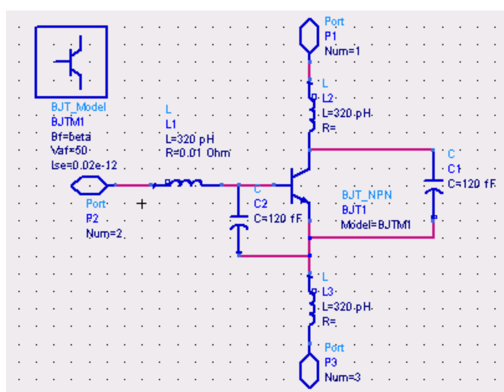


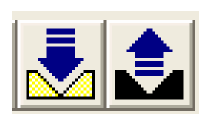
Insert the sub-circuit from the library





Design parameters follow the sub-circuit: Q1, beta, etc.





ICONS: Push into and Pop out of the hierarchy.

Course Topics

- 1:Circuit Simulation Fundamentals
- 2:DC Simulation and Circuit Modeling
- 3:AC Simulation and Tuning
- 4:S-Parameter Simulation and Optimization

AC Simulation

- You get linear small-signal response and you get Noise values:

  •DC analysis performed (unseen)

  •Nonlinear devices are linearized

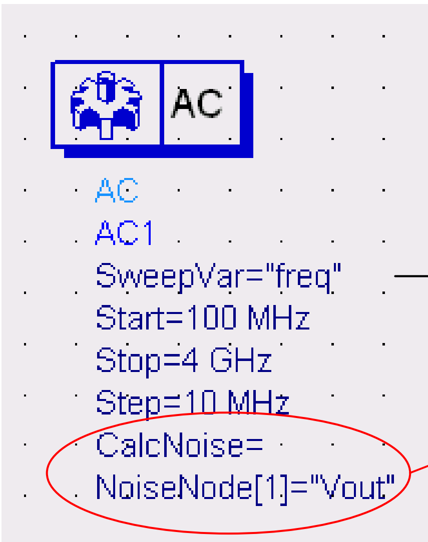
  •Kirchoff's Law satisfied: sum of node current = 0

  •Noise contributors defined and listed

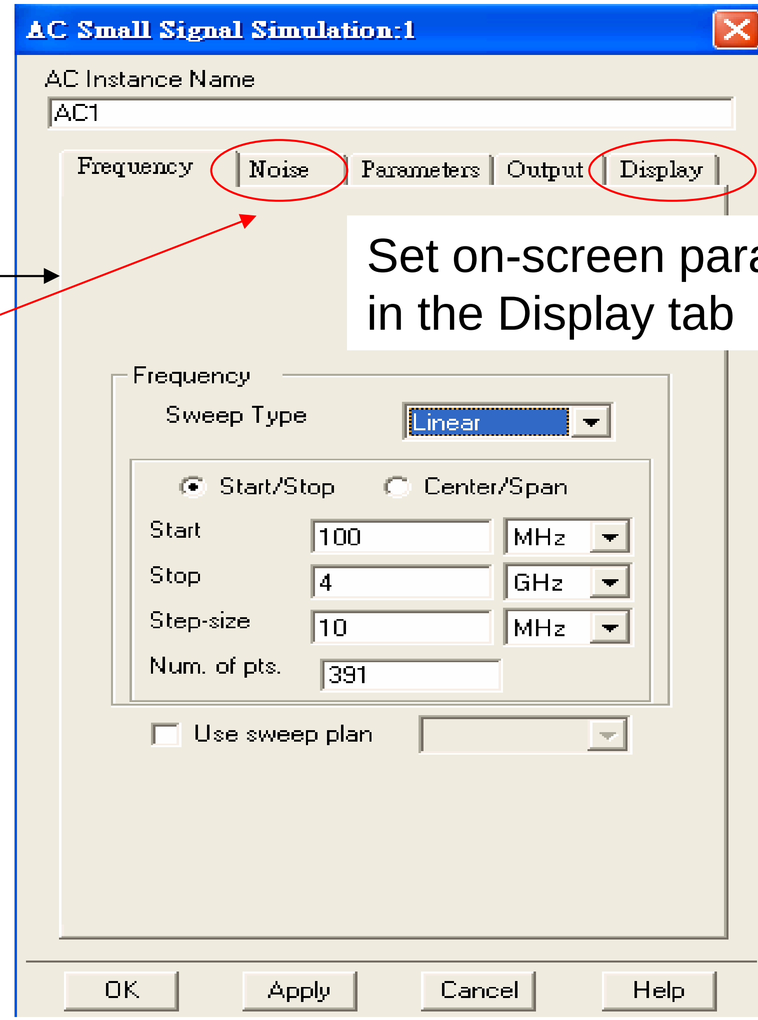
  •Budget analysis available (for named nodes)

  •Signal voltages are peak – noise voltages are RMS

AC Simulation Controller



Turn Noise on/off: yes/no

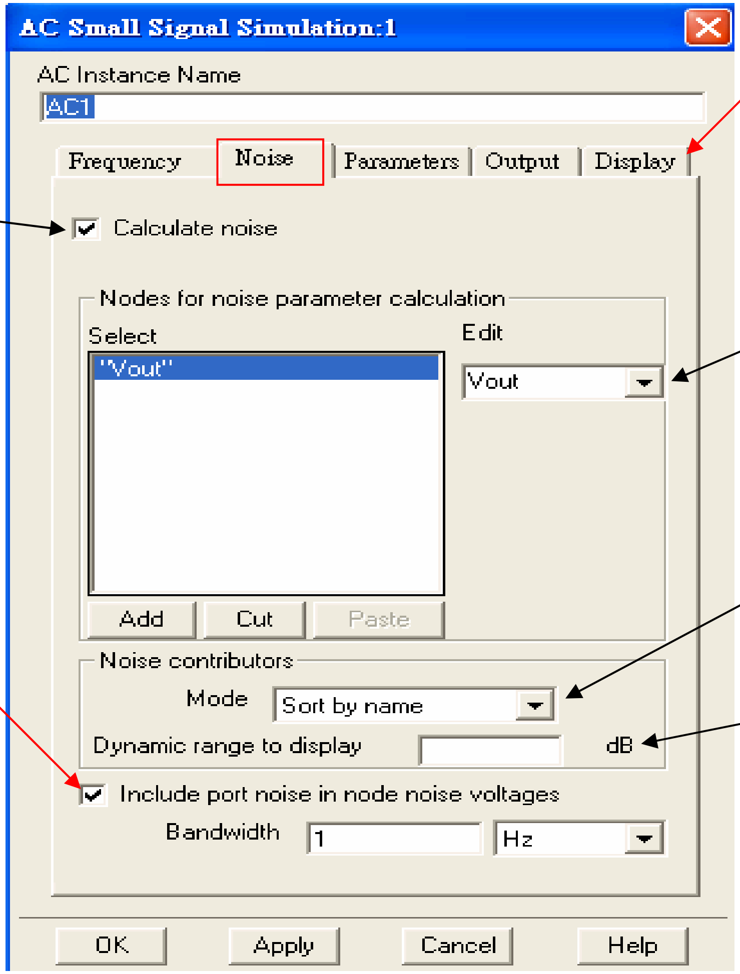


Set on-screen parameters in the Display tab

AC is a linear or small signal simulation and freq is defined in the cintroller not the source.

AC Noise calculations

Click here:



Again, use the Display tab to see your settings on-screen.

Nodes are Wire/Pin Labels.

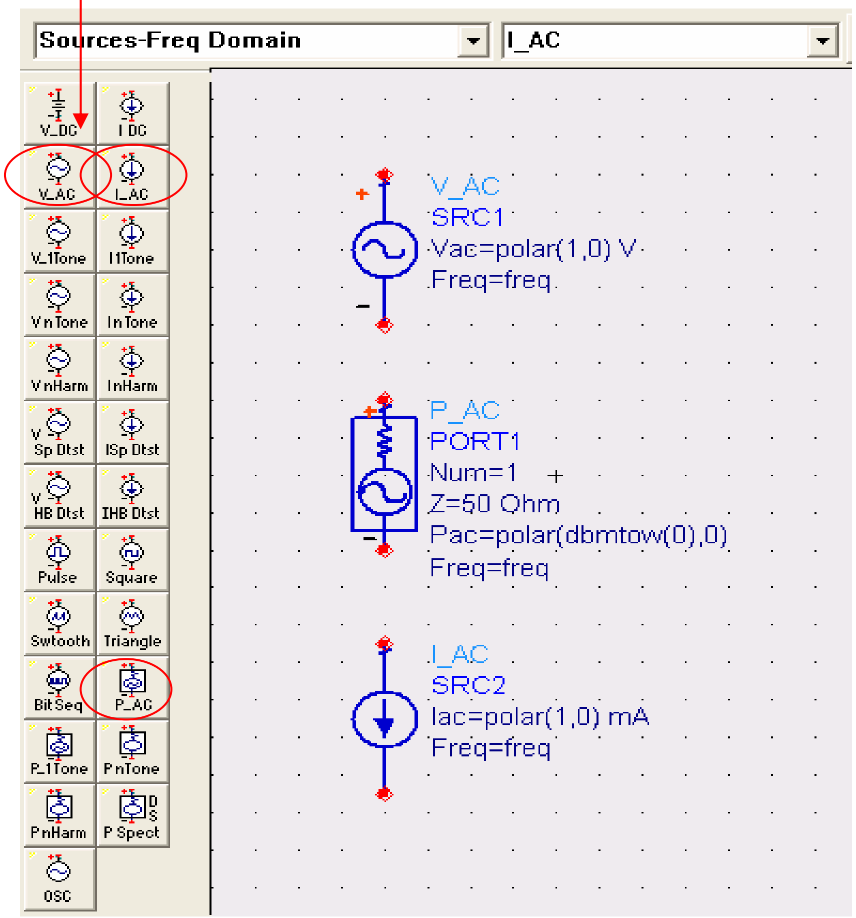
Sort by name or by value: in the dataset.

Blank gives you all contributors.

NOTE: Port Noise can be included in the simulation, but it does not apply to NF.

AC sources are for AC simulations

Three AC sources…



Source parameter definitions:

- **V_AC, P_AC, and I_AC are component names.**
- **SRC1, PORT1, and SRC2 are instance names which you can change.**
- **Vac=polar(1,0)V is the default value. The polar function can be removed.**
- **Freq=freq is a global variable – you set the start & stop values in the simulation controller.**
- **P sources are also ports (ok for S-parameter).**
- **Num=1 is the port number.**
- **I current sources can be used for other simulations to inject current.**
- **Arrow in symbol shows the direction of current flow.**

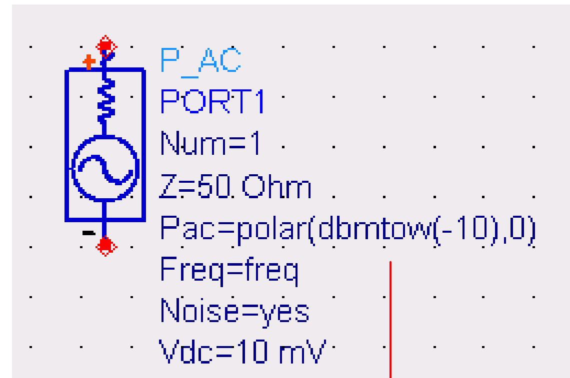
Setting AC source values

•POWER: The dbmtow function converts power in dbm to watts for the simulator.

•PHASE: The polar function specifies phase. By default, all sources are cosine waves. Use -90 for a sinewave.

•NOISE and Vdc: By default, noise is turned on for the P_AC source. Use Display tab/settings to make visible. Vdc 10 mV is an offset.

•Equations can also be used: P=1W, P=1+j*1W, P=complex(1,0), etc.



Simplify Pac by removing the polar function.

Pac=dbmtow(-10)

Review of ADS equation types

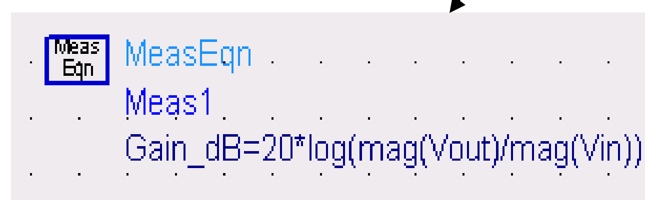
•VAR: pre-simulation

Use for initializing sweep variables or other settings. VARs are available in the dataset if you select them in the Output tab of the simulation controller.

Schematic

•MeasEqn: pre-simulation

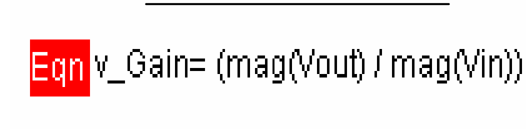
Use for calculations on schematic and are available in the dataset by default. Use node labels and functions.



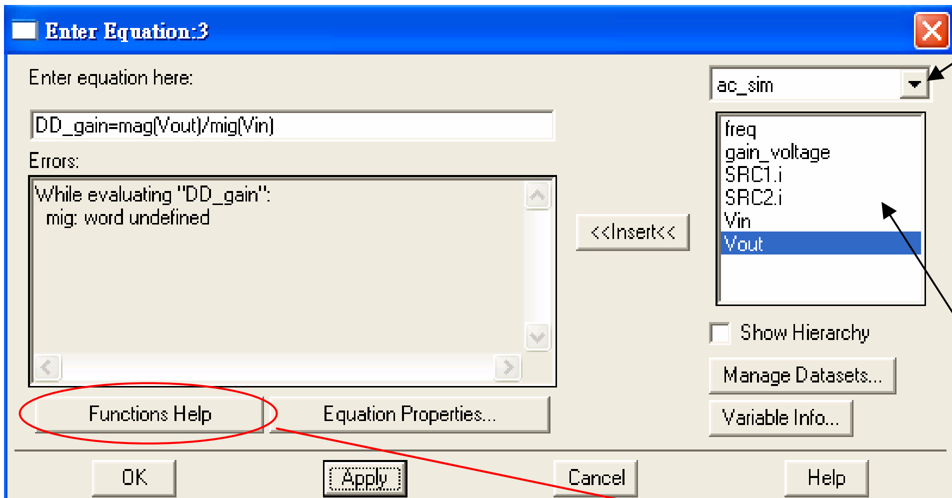
•Eqn; post-simulation

Use for calculations in the data display. Use node voltages, functions, and any dataset data.

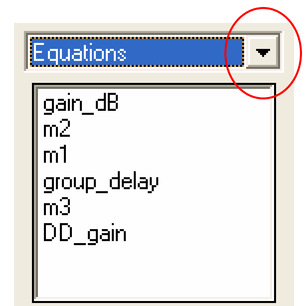
Data display



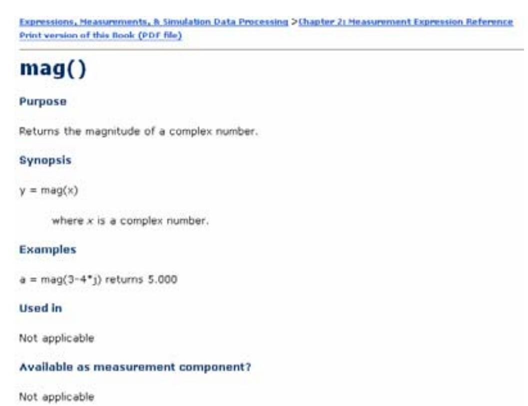
Review of Data Display equation editor



Click here for DDS equations:



Schematic MeasEqns appear in the dataset.



Expressions, Measurements, & Simulation Data Processing > Chapter 2: Measurement Expression Reference
Print version of this Book (PDF file)

mag()

Purpose

Returns the magnitude of a complex number.

Synopsis

y = mag(x)

where x is a complex number.

Examples

a = mag(3-4*j) returns 5.000

Used in

Not applicable

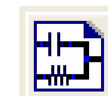
Available as measurement component?

Not applicable

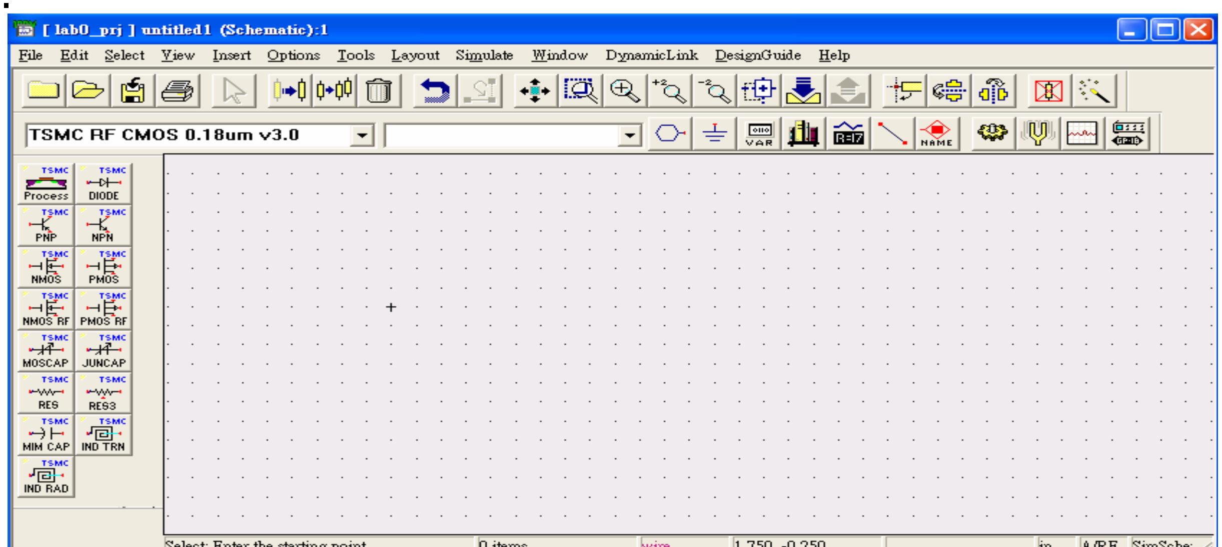
- Invalid equations are red.
- Valid equations are black.

LAB 1: Circuit Simulation Fundamentals

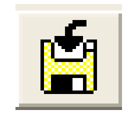
1. Start ADS on the computer.
2. Create a new Project. (type the name: lab1)
3. Create a low-pass filter design.



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.

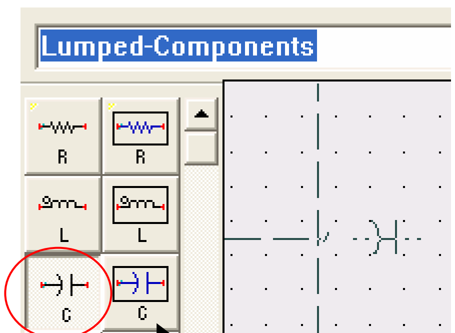


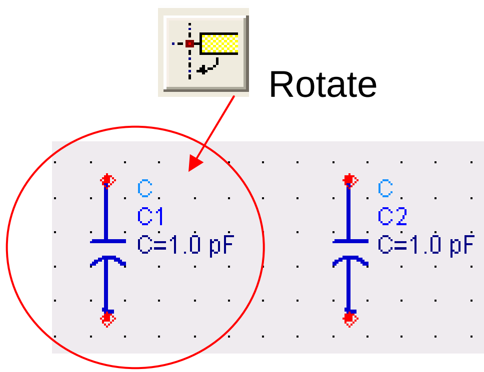
b. Save the schematic. Notice the top window border of the schematic shows the schematic name as untitled. Click the icon (shown here) and the Save Design As dialog will appear. Type in the name **lpf** and click **Save.** This will save it in the networks directory of lab1 project.



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.

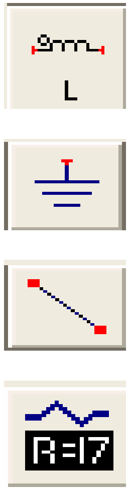
c. In the **Lumped Components** palette, select (click) the **capacitor C** shown here (not the C model). Then click the **Rotate By Increment** icon as needed for the correct orientation and then click to insert the capacitor as shown on the schematic. Next, insert another capacitor.





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

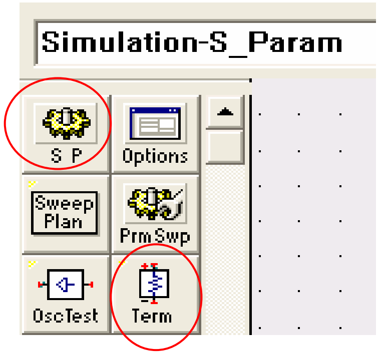
d. Continue creating the low-pass filter as shown by inserting the **inductor** and **grounds** (icons are shown here). Then **wire** the components together. This will give you practice with schematic capture. You can try using the copy, move and other icons or commands.

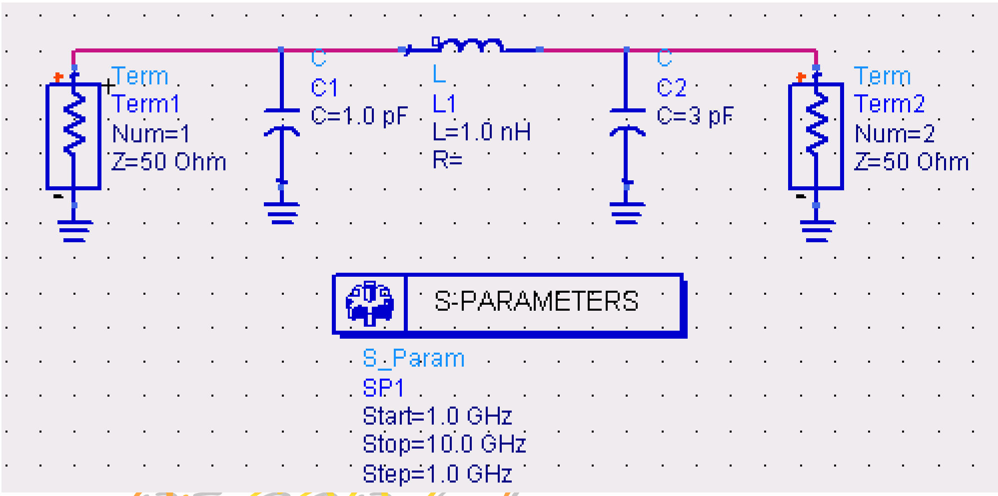


e. After the filter is built, edit the value of C2 to be 3 pico-farads. To do this, double click the capacitor symbol or select the capacitor and use the icon (shown here). When the dialog box appears – change the value: **C=3.0 pF**, click **Apply** and **OK.**

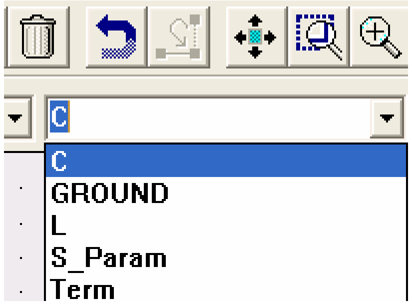
f. Next, select the **Simulation-S_Param** palette and insert the **S-parameter** simulation controller (gear icon). Use the ESC key to end the command.

g. Then insert the port terminations: **Term** Num=1 and **Term** num=2 shown here.



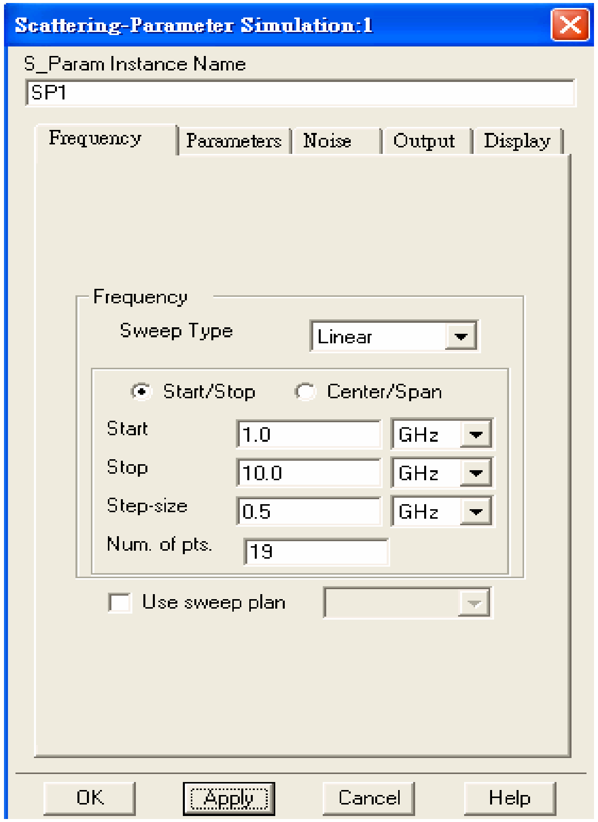


h. Use Component History: After the circuit is built, delete capacitor C2 and then reinsert it by typing or selecting (history) the capital letter **C** in the **Component History** field and press Enter. Next, edit the value directory on the schematic by highlighting the value and typing over it with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



4. Setup the S-param Simulation.

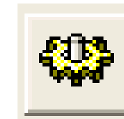
a. To setup the simulation, double click on the S-parameter simulation controller on the schematic. When the dialog box appears, change the **Step-size to 0.5 GHz** and click **Apply**. Notice how it updates the value on the screen . The OK button does the same thing as Apply and also dismisses the dialog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



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

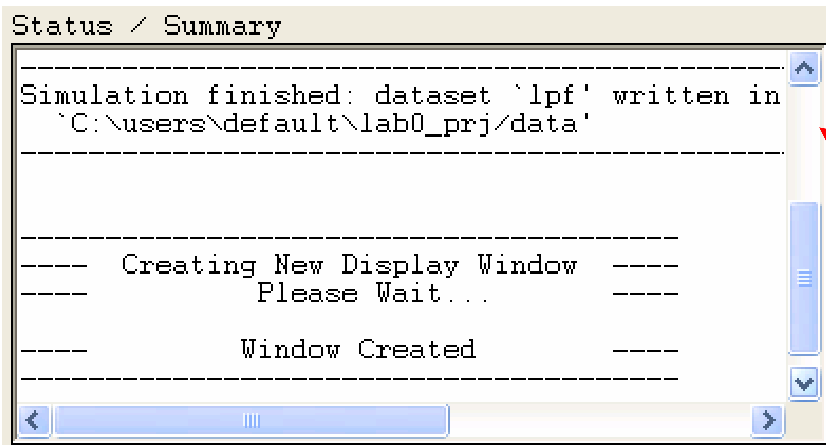
c. Click the OK button to dismiss the dialog box. You are now ready to simulate.

5. 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.

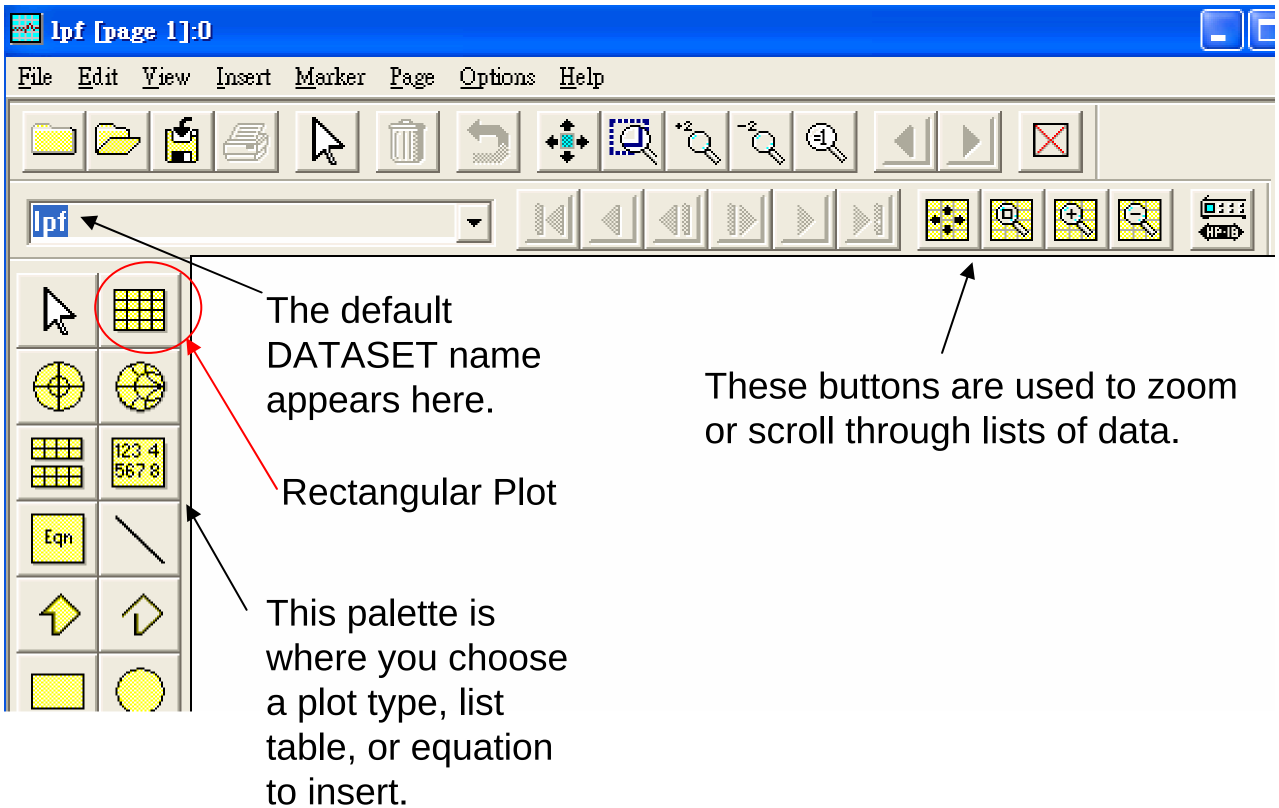
b. Next, look for the **Status window** to appear and you should see messages similar to the ones here, describing the results of the simulation, the writing of the dataset file, and the creation of a display window. If not, ask the instructor for help.



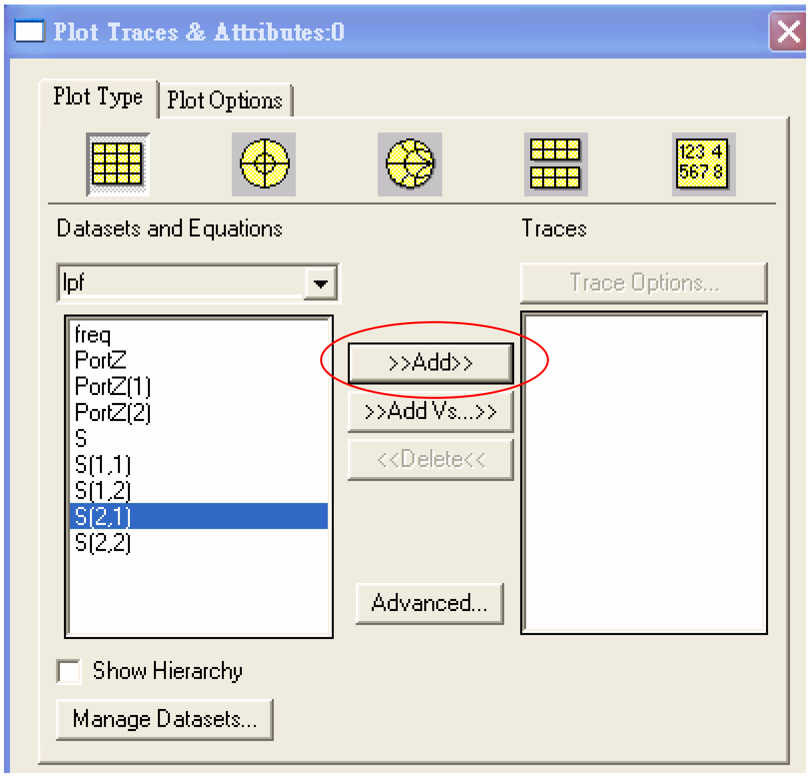
NOTE: If you scroll up, you will see more simulation information

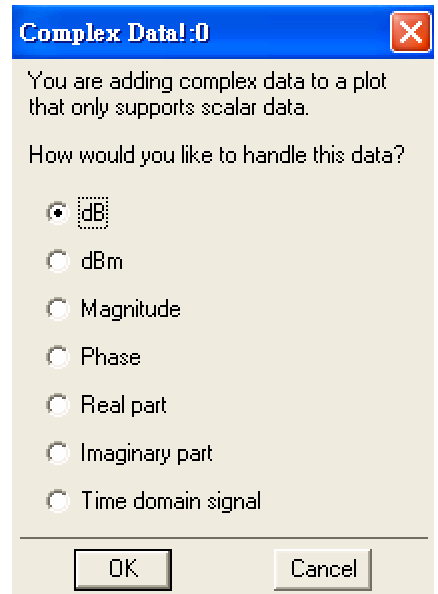
c. If no simulation errors occurred, close the Status window. You can always get the status window back using the schematic window command: **Window > Simulation Status.**

d. 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 page1 which is blank at this time. Examine the picture below – the next steps will show how to display the simulation data.



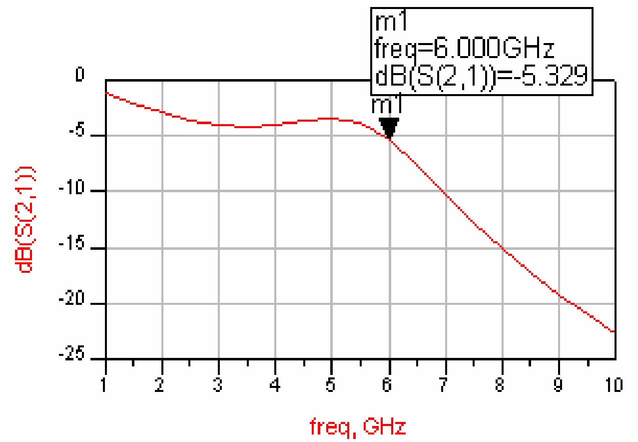
e. To create the plot, click on the **Rectangular Plot** icon and move the cursor (with outlined box) into the window and click. When the next dialog box appears, select the **S(2,1)** data and click the **Add** button. Then select **dB** as the format for the data. Click **OK** in both boxes.





f. The plot should show a reasonable low pass filter response.

g. Put a marker on the trace: Click the menu command: **Marker > New.** Select a point on the trace and click to insert the marker. Select the marker or the marker text and 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.

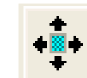


6. Save the Data Display Window.

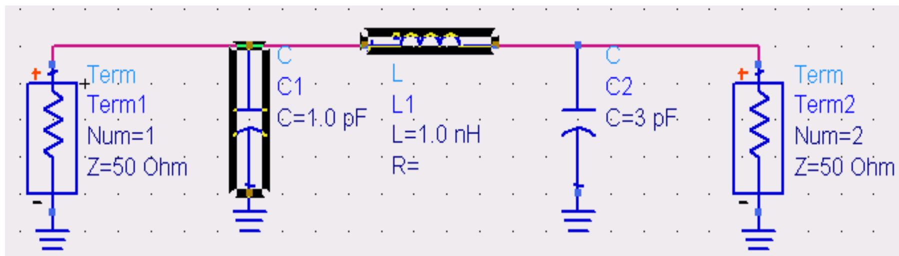
7. Tune the filter circuit.

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.

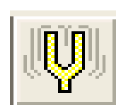
a. Position the Data Display and the Schematic windows so you can see them both on the screen. If necessary, re-size thewindows and use **View All.**



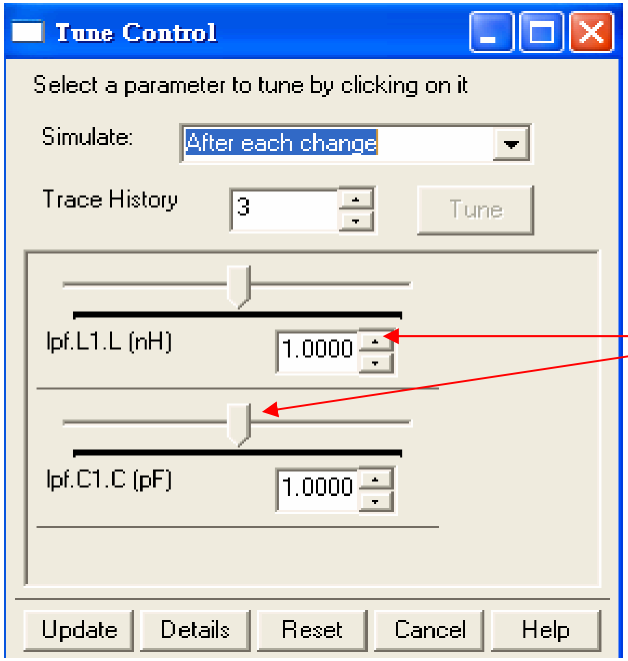
b. Now, in the **lpf** schematic, select both components **C1** and **L1** using the **SHIFT** or **Ctrl** key as shown here.



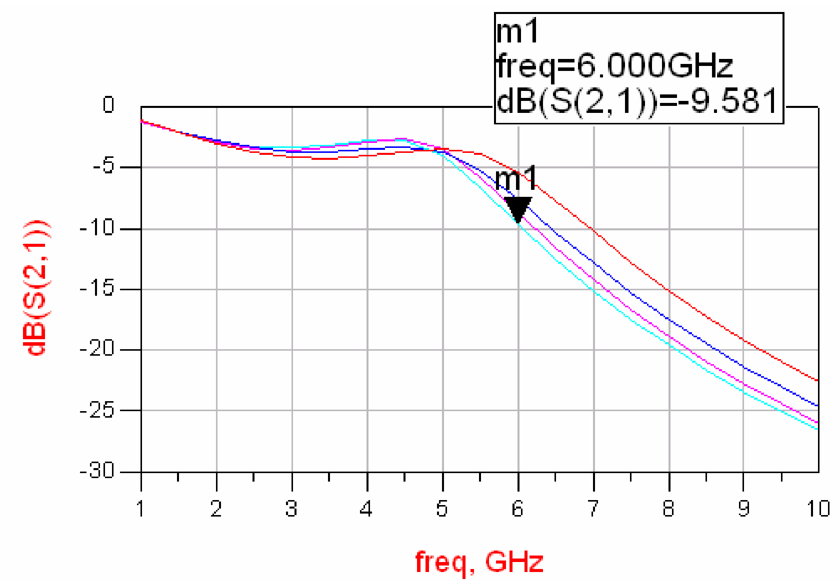
c. Now, start the tuner by clicking the command: **Simulate>Tuning** or click the Tune Parameters icon (shown here).



Immediately, the status (simulation) window will appear along with the Tune Control dialog box (shown here). Go ahead and tune the filter using the default settings and watch the updated traces appear in the Data Display.

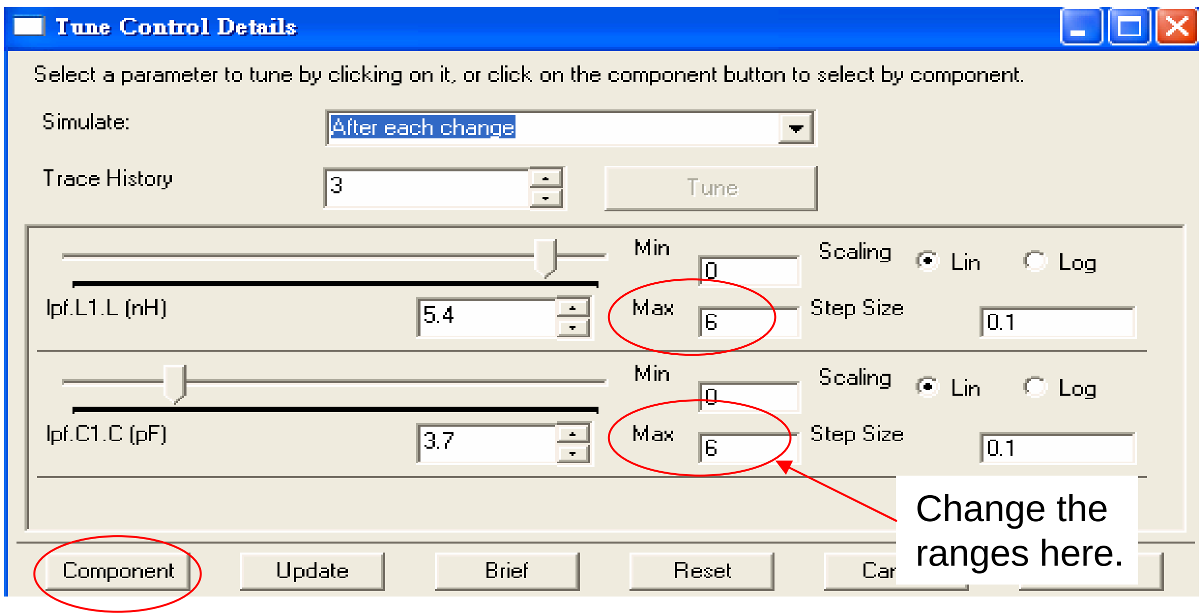


Move the slider or click on the buttons to tune values. Notice how the new traces appear on the s_data plot after each change.



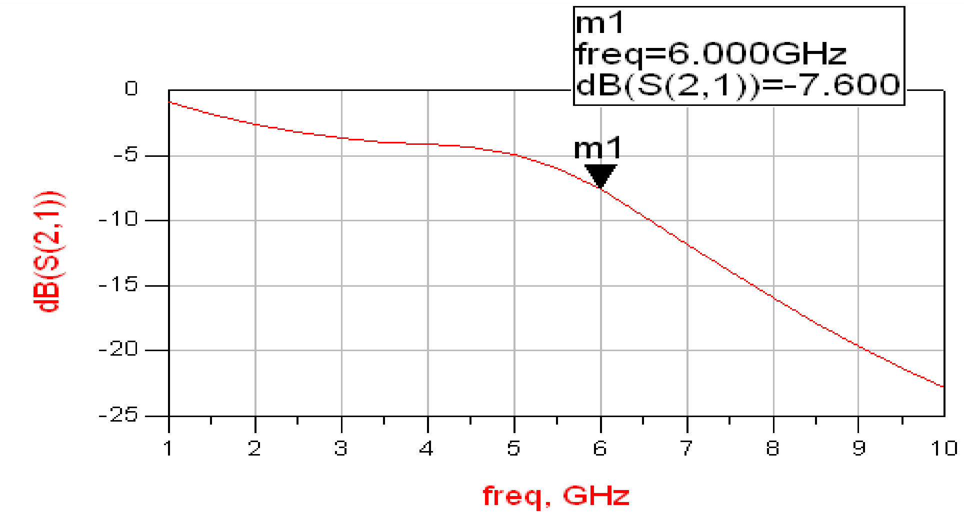
Each tuning creates another trace. The marker moves to the most recent simulated (tuned) trace which is thicker than the others. This trace is the s_data dataset which is changed each time you tune (simulate).

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



e. Continue tuning until you are satisfied with the results – click the **Update** button to have the C and L unit 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.

f. When you are satisified with the tuned response, simply click the **Cancel** button and the plot will contain the final tuned trace similar to the one shown here. It is not important to achieve any particular goal in this lab exercise but simply to learn how to use the Tuner.



g. Save the data display and the schematic using the **Save** icon. Then close both windows so that only the ADS Main window remains.

Lab2:DC simulations and Circuit Modeling

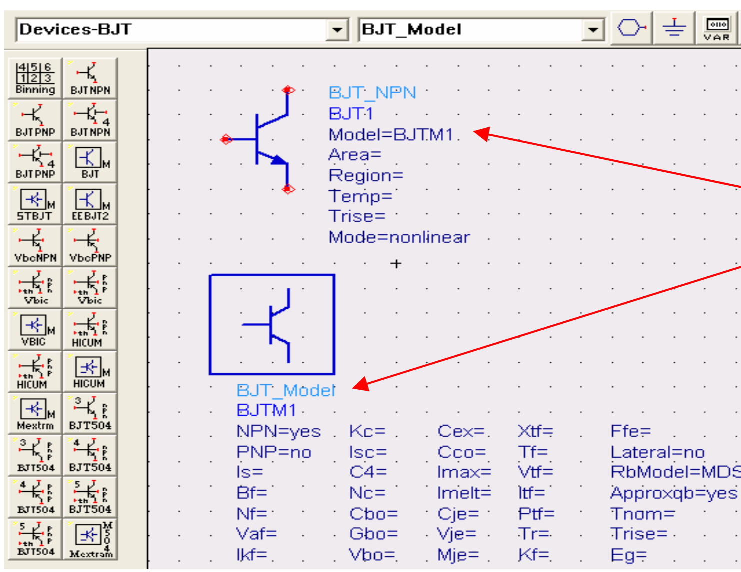
1. Create a new project: amp_1900.

a. Then, open a new schematic window and save it with the name: **bjt_pkj**.

2. Set up a generic BJT symbol and model card.

a. In the schematic window, select the palette: **Devices-BJT**. Select the BJT-NPN device shown here and insert it onto the schematic.

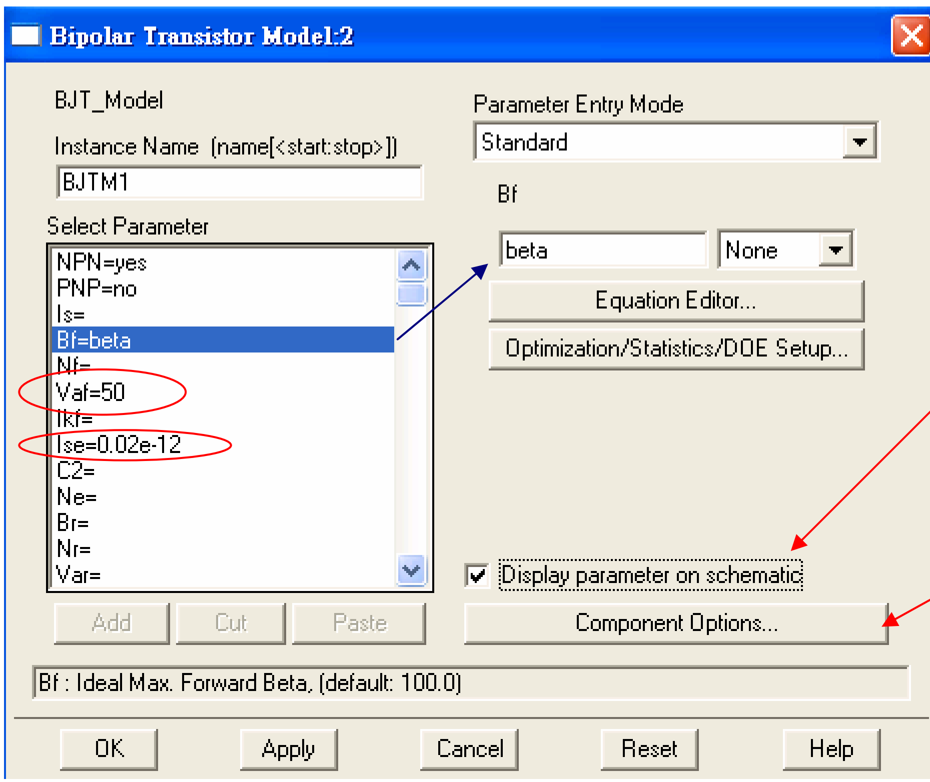
b. Insert the **BJT_Model** (model card) shown here.



NOTE: The BJT_NPN symbol shows Model = BJTM1. This means the symbol will use that specific model (model card) for simulation.

c. Double click on the **BJT_Model** card. When the dialog appears, click **Component Options** and in the next dialog, click **Clear All** for parameter visibility – then click **Apply**. This will remove the Gummel-Poon parameter list from the schematic. Keep this dialog open.

d. Next, in **BJT_Model** dialog, select the **Bf** parameter and type in the word **beta** as shown here. Also, click the small box: **Display parameter on schematic** for **Bf** only and then click **Apply.** Beta is now a parameter of this circuit - later on you will tune it like a variable.



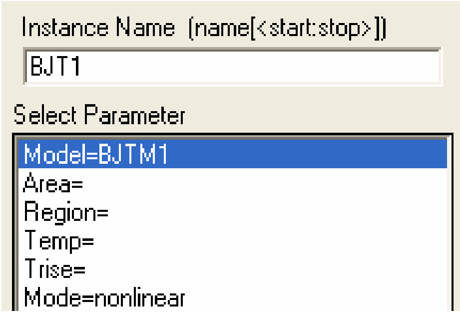
Click here to display an individual value.

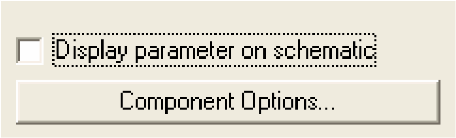
Use Component Options to clear all the displayed parameters.

e. Set **Vaf** (Forward Early Voltage) = **50** and display it.

f. Set **Ise** (E-B leakage) = **0.02e-12**, and display it also. Then close the dialog with **OK**. The device now has some more realistic parameters.

g. For the **BJT device**, remove the unwanted display parameters (Area, Region, Temp and Mode) by unchecking the box. This will make the schematic less crowded with parameters that you are not using.





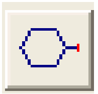
3. Add parasitics and connectors to the circuit.

The picture shown here is the completed sub-circuit with connectors and parasitics. Remember to use the **rotate icon** for orientation of the components as you insert them. Here are the steps:

a. Insert lumped L and C components: Insert three lead inductors of **320 pH** each and two junction capacitors of **120 fF** each. Be sure to use the correct units (pico and femto) or your circuit will not have the correct response. Tip: type L or C in component history to get the components onto your cursor without using the palette.

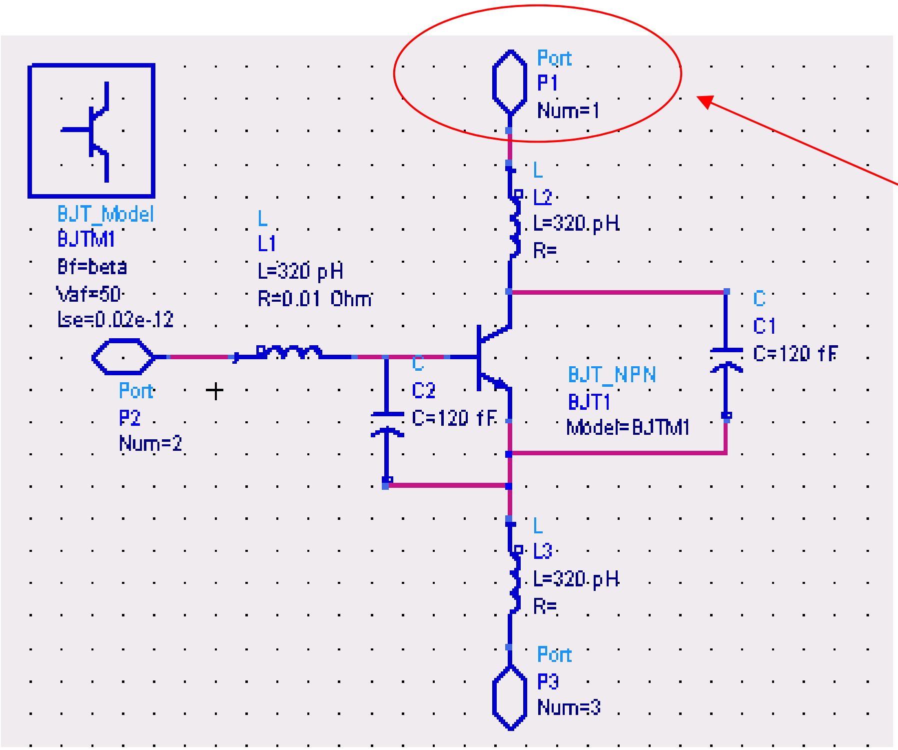
b. Add some resistance **R=0.01 ohms** to the base lead inductor and display the desired component values as shown.

c. Insert port connectors: Click the port connector icon (shown here) and **insert the connectors exactly in this order**: 1) collector, 2) base, 3) emitter. You must do this so that the connectors have the exact same pin configuration as he ADS BJT symbol.



d. Edit the port names as show here: change P1 to **C**, change P2 to **B**, and change P3 to **E**.

e. Clean up the schematic: Position the components so that the schematic looks organized – this is good practice. To move component text, press the **F5** key, **select the component**, use the cursor to position the text.



NOTE: you must number the ports (num=) exactly as shown or the device will not have the correct orientation for the symbol that will be used.

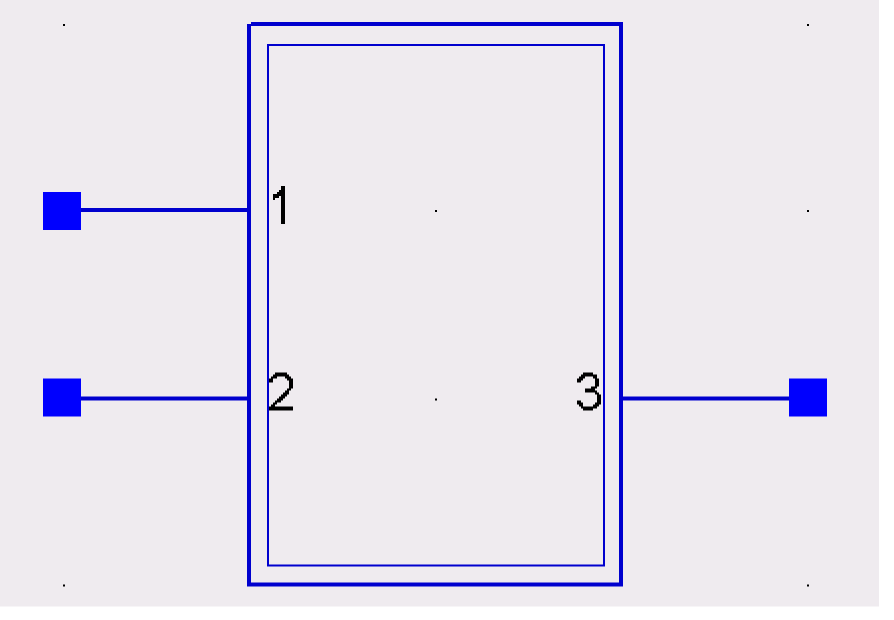
4. View the default symbol.

There are three ways to create a symbol for a circuit: 1) use the default symbol, 2) draw a symbol, or 3) use a built - in ADS symbol. For this lab you will use a built-in BJT symbol. The following steps show how to do this:

a. To see the default symbol, click: **View > Create/Edit Schematic Symbol**. When the dialog appears, click **OK** and the default symbol will appear.

b. Next, a box or rectangle with three ports is generated. This is the default symbol. However, delete this symbol using the commands: **Select > Select All**. Then click the **trash can** icon or delete key.

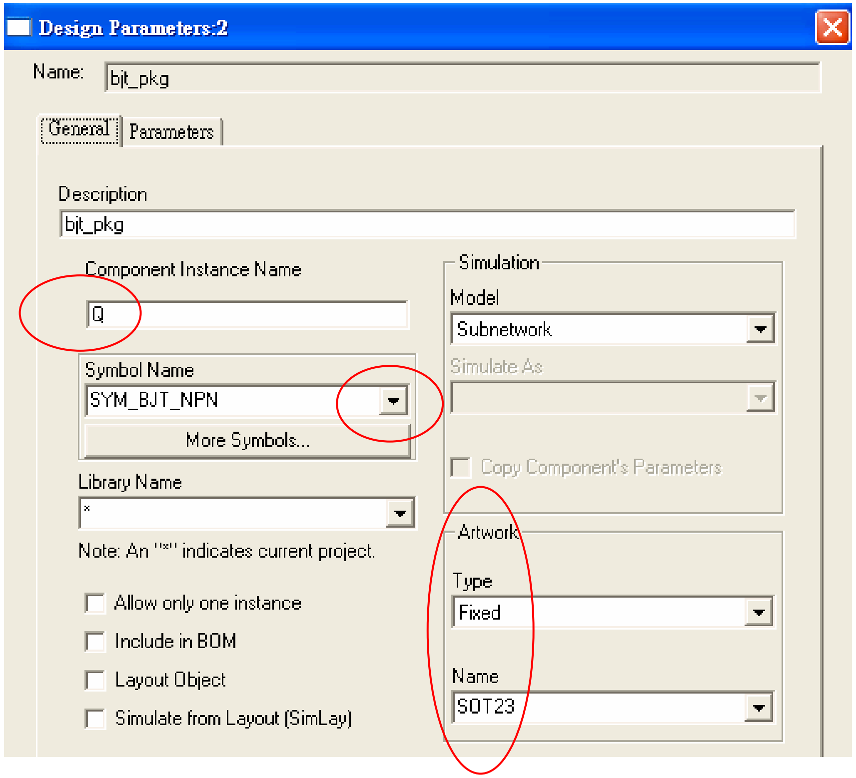
c. Return to the schematic – click: **View > Create/Edit schematic.**



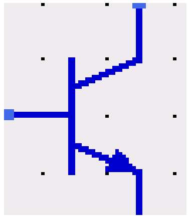
5. Set up the Design Parameters and built-in symbol.

a. Click **File > Desgin Parameters** and the dialog appears.

b. In the **General tab**, make theses changes: 1) change the Component Instance Name to **Q**, 2) change the Symbol Name to **SYM_BJT_NPN** by clicking the arrow and selecting it (this is the built-in symbol), 3) in the Artwork field, select **Fixed** and **SOT23** as shown here.



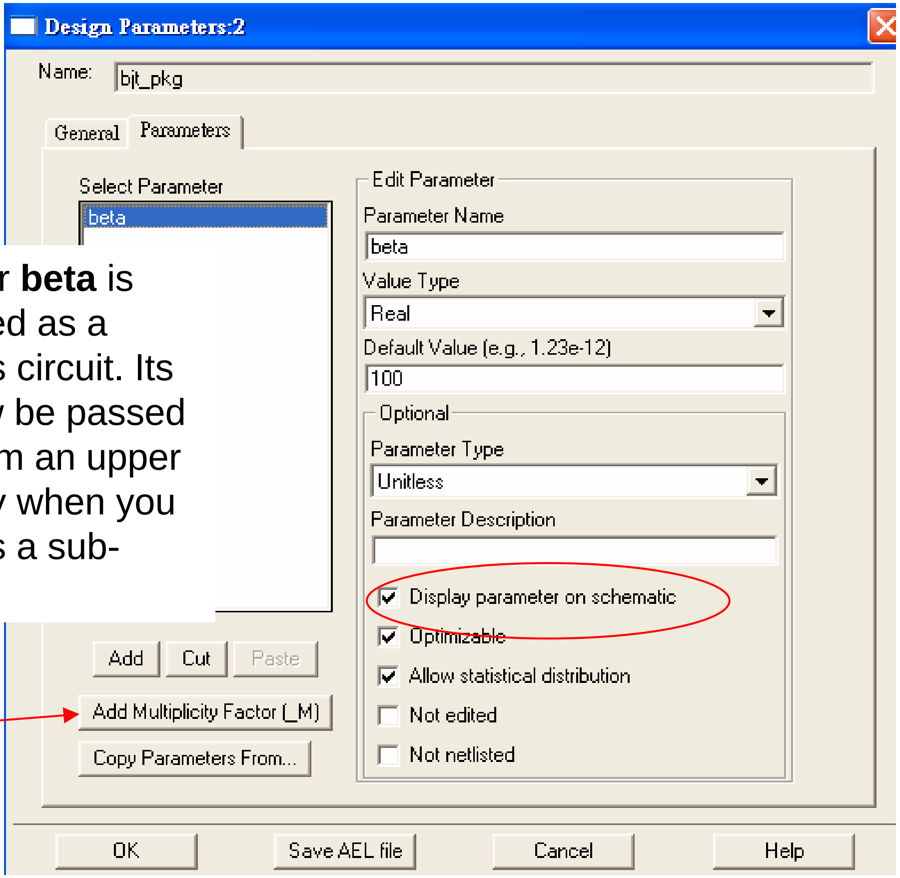
Built-in symbol:
SYM_BJT_NPN



c. Click **Save AEL File** to write these changes but do not close this dialog yet because you still need to set other parameters.

d. Go to the **Parameters tab**. In the Parameter Name area, type in **beta** and assign a default value of **100** by clicking the **Add** button. Be sure the box to **Display** the parameter is checked as shown here. Click the **OK** button to save the new definitions and dismiss the dialog.

e. In the schematic window, **Save** the design. In the next steps, you will see how the Design Parameters will be used.



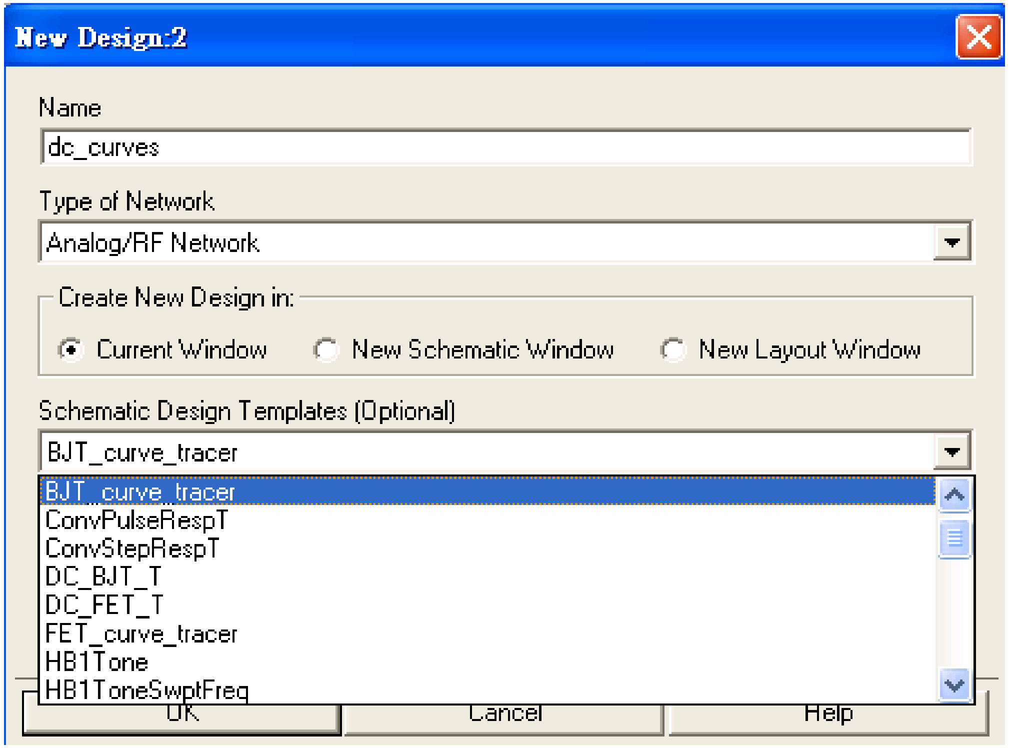
The parameter **beta** is now recognized as a variable of this circuit. Its value can now be passed (assigned) from an upper level hierarchy when you use bjt_pkg as a sub-circuit.

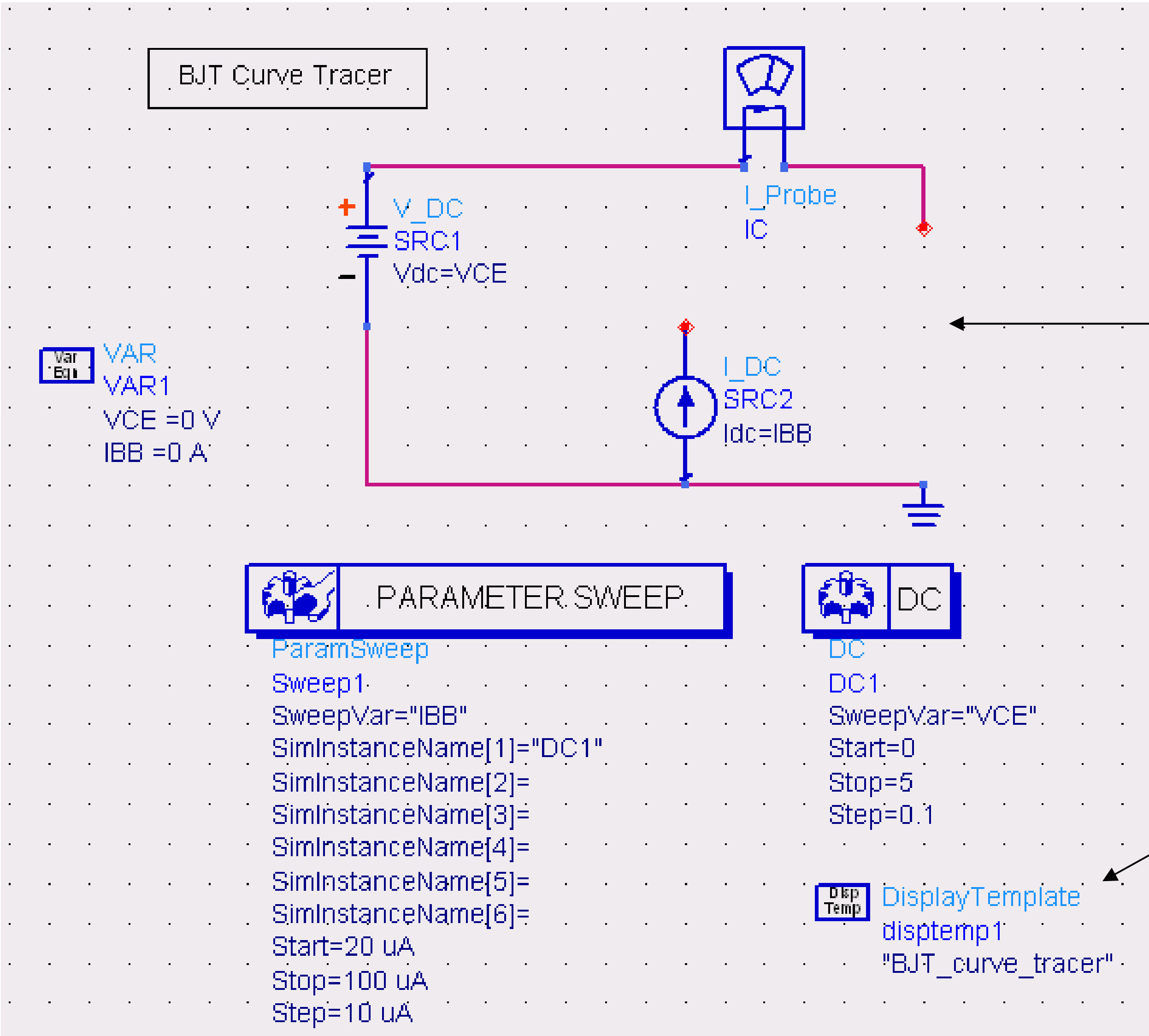
NOTES: multiplicity_M

You can define multiple components in parallel. You can also copy parameters from another device or file.

6. Use a curve tracer template to test the bjt_pkg sub-circuit.

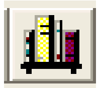
a. In the current schematic of the bjt_pkg, click **File > New Design**. When the dialog appears, type in the name: **dc_curves** and select the **BJT_curve_tracer** template as shown. Click **OK** and a new schematic will be created with the template, ready to insert bjt_pkg.



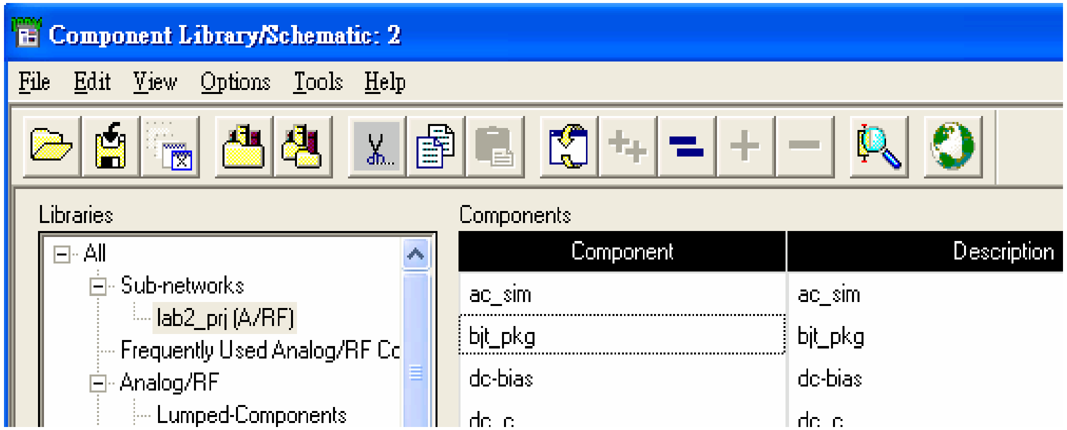


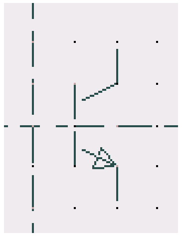
This is where you will connect the device (bjt_pkg) in the next few steps.

This is a data display template - it will automatically plot the curves.

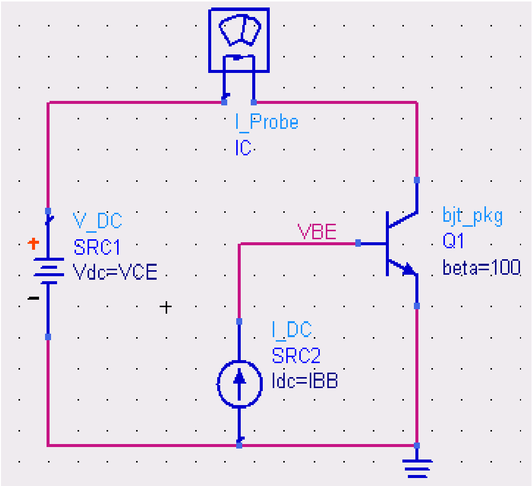
b. Save the design and then click the Component **Library** icon (shown here). 

c. When the dialog opens, select the **amp_1900** project as shown and click on the **bjt_pkg** sub-circuit. Insert it into the schematic as shown here. Every circuit that you build will be available in the project library as a sub-circuit.



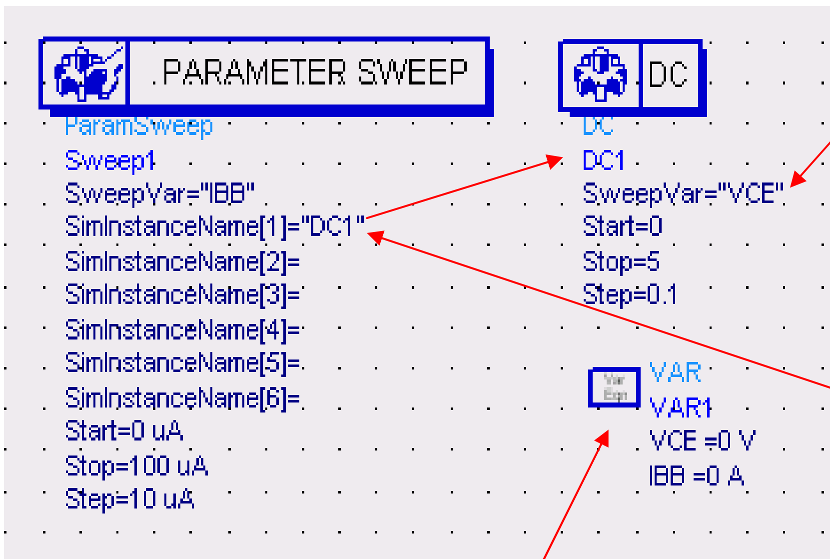


d. Connect the **bkt_pkg** component as shown. You may have to adjust the wires and text (F5) to make it look good. Also, you can now **close the library window** and save the dc_curves design again – it is good practice to save often.



7. Modify the template Parameter Sweep.

a. Change the Parameter Sweep **IBB** values to: 0 uA to 100 uA in 10 uA steps as shown here. Do not change the DC simulation controller default settings for sweeping VCE – they are OK. Notice that the VAR1 variables (VCE=0 and IBB=0) do not require modification because they are only required to initialize (declare) the variable for the simulator.



Only one variable can be swept in the controller.

Parameter sweep used for multiple variable sweeps. Note that DC1 is the name (SimInstance Name) of the simulation controller.

VarEqn is required to initialize variables.

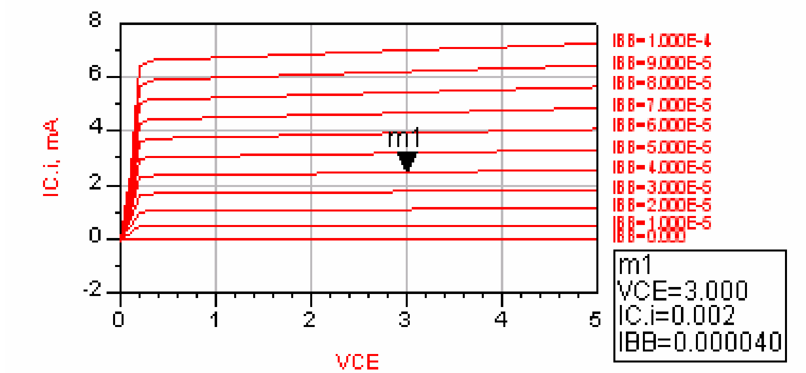
8. Simulate at beta=100 and 160.

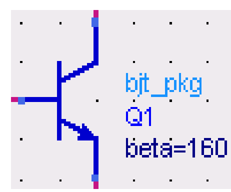
a. Simulate (F7) with **Beta = 100**. After the smulation is finished, the data display will appear with the curve tracer results (data display template). Try moving the marker and watch the updated values appear.

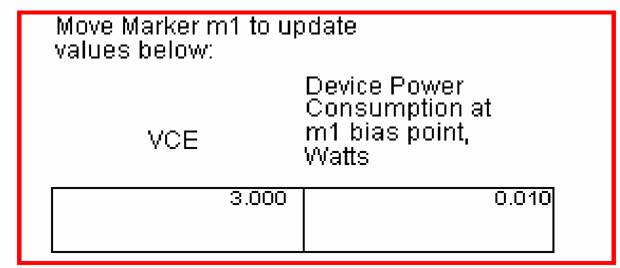
b. Simulate again **with Beta = 160** by changing the value directly on the schematic. You should see the updated values.

c. Verify the values for beta = 160 and VCE = 3V, where IBB=40 uA and IC=3 mA with about 10 mW of consumed power If not, check the design.

DC Curves at beta = 100







9. Open a new design and check all your files in the Main window.

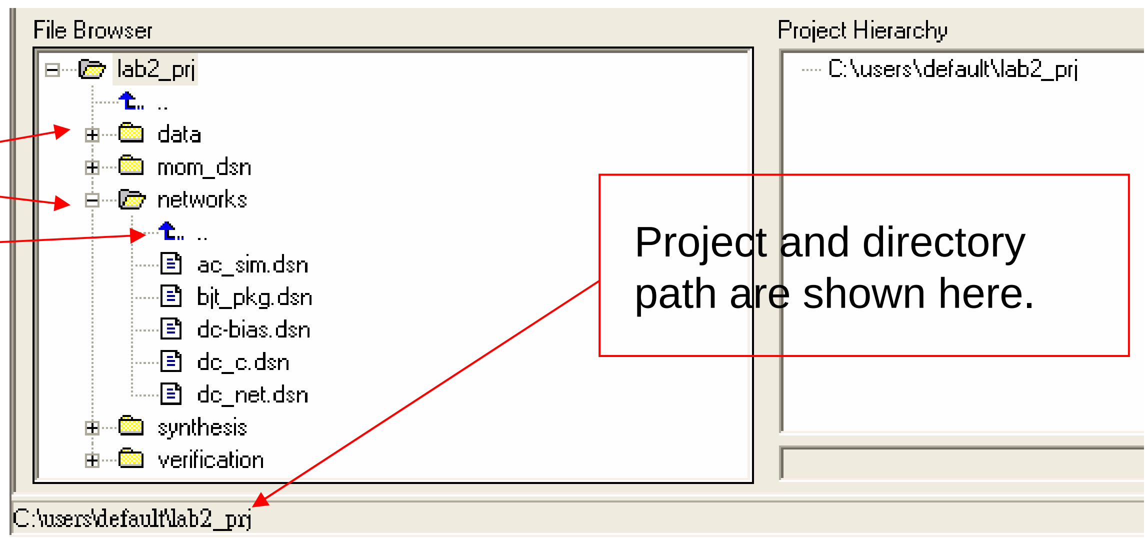
a. Save the current schematic: dc_curves. In the same window, create a new design (without a template) named: **dc_bias**. Then save the design by clicking on the **Save Current Design** icon (shown here) so that it is written into the ADS database.

b. Now, check the ADS Main Window: you should have 3 designs in the networks directly: bjt_pkg, dc_curves, and dc_bias. You may have to double-click on the file browser **networks** to refresh the browser.

c. In the **File Browser** area, click on the plus / minus boxes and the up arrow (or two dots). This allows you to see the files you have created in this project. Remember: **you can only work in one project at a time**, but you can copy files from other projects and bring them in.

Try clicking the plus (expand) and minus (contract) boxes.

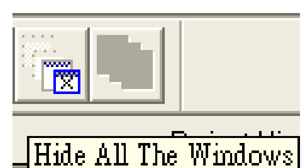
Try the up arrow.



Project and directory path are shown here.

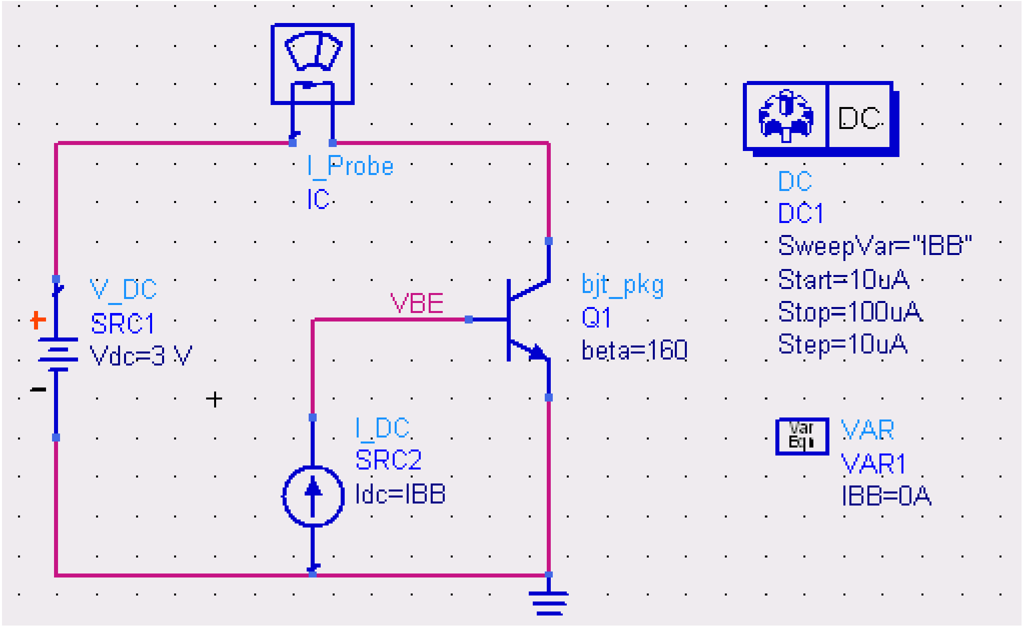
d. Finally, try the Show / Hide all windows feature. This is used for security or to find other open windows that are not ADS.

In this case, only the Main window remains.



10. Set up and simulate the dc bias parameter sweep.

NOTE on parameter sweeps: If only one variable is swept, you can use the Simulation controller (Sweep tab). However, if more than one parameter is swept, a Parameter sweep component is required as in the templates you have just used. In general, all simulation controllers allow you to sweep only one parameter (variable).



a. Insert the **bjt_pkg** using library icon or the component history. Now **push into** the bjt_pkg and click **File > Design Parameters**. Reset the beta parameter default to160, pop out and **delete** the bjt_pkg and **reinsert** it – beta is now 160 whenever you use the modeled circuit.

b. From the Probe components palette or component history, insert an **I_Probe** and rename it **IC** instead of I_Probe1 as shown here.

c. From the **Sources-Frequency domain palette** or using component history, insert a dc supply and current source and set their values as shown: **Vdc = 3V** and **Idc = IBB** as shown here.

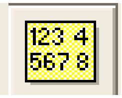
d. Wire the components together and add the ground (ground icon).

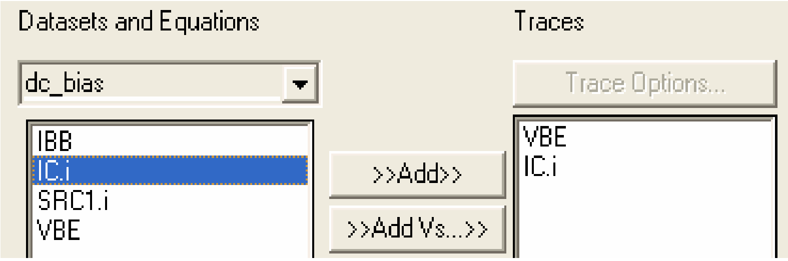
e. Insert a **DC simulation controller** or DC. Edit (double click) the controller and go to the **Sweep tab** and assign: **IBB**: **10uA** to **100uA** in **10uA** steps. Then go to the **Display tab** and check the settings to be displayed as shown. Then click **Apply** and **OK.**

f. Insert a **VAR** (click icon) variable equation. Use the cursor on the screen to set **IBB=0 A** to initialize (declare) the variable to be swept.

g. Insert a wire label **VBE** at the base. The voltages at that node will appear in the dataset for use in calculating bias resistor values.

h. Simulate and **plot the data**. When the data display opens, insert a list of **VBE** and **IC.i** only. Because you swept IBB to get these values, IBB will automatically be included.





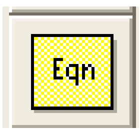
IBB	IC.i	VBE
1.000E-5	599.8uA	754.8mV
2.000E-5	1.430mA	777.1mV
3.000E-5	2.349mA	789.9mV
4.000E-5	3.325mA	798.8mV
5.000E-5	4.341mA	805.7mV
6.000E-5	5.389mA	811.3mV

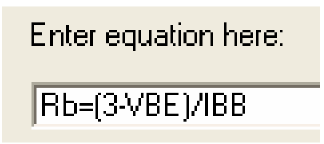
NOTE on results: As you can see, with 3 volts across the device, 40 uA of base current results in about 799mV across the base-emitter junction, with about 3.3 mA of collector current. If you want, draw a box around the values at 40 uA IBB.

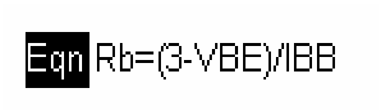
i. Save the design and data display.

11. Calculate bias values Rb and Rc for a grounded-emitter circuit.

a. In the data display, insert an equation and type: **Rb = (3 – VBE) / IBB.**

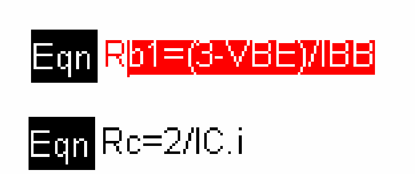




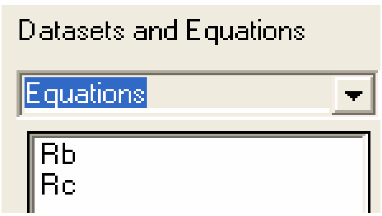


b. Select the Rb Eqn and use the keyboard **Ctrl C** and **Ctrl V** to copy/paste it – it will become Rb1.

c. Highlight the Rb1 equation as shown and change it to become Rc: **Rc = 2 / IC.i**. The total DC supply will be 5 volts. Therefore, with 3 volts VCE, 2 volts remain for the collector resistor.



d. Insert a new **List** and scroll down to the **Equations** menu (shown here), and add **Rb** and **Rc**. Then edit both column headings on the list with a bracketed **[3]** as shown. This references the 40uA IBB using its index value [3]. Index values begin at zero: 0, 1, 2, 3, etc. You can also use Plot Options to add a label to the list as shown:



IBB	Rb[]	Rc
1.000E-5	224518.366	3334.233
2.000E-5	111142.833	1398.882
3.000E-5	73669.698	851.344
4.000E-5	55029.027	601.500
5.000E-5	43886.103	460.687
6.000E-5	36479.132	371.123

Bias Resistor Values @ 40 uA IBB

Rb[3]	Rc[3]
55029.027	601.500

12. Set up the biased network.

Now that you have the calculated bias resistor values, you can test the bias network.

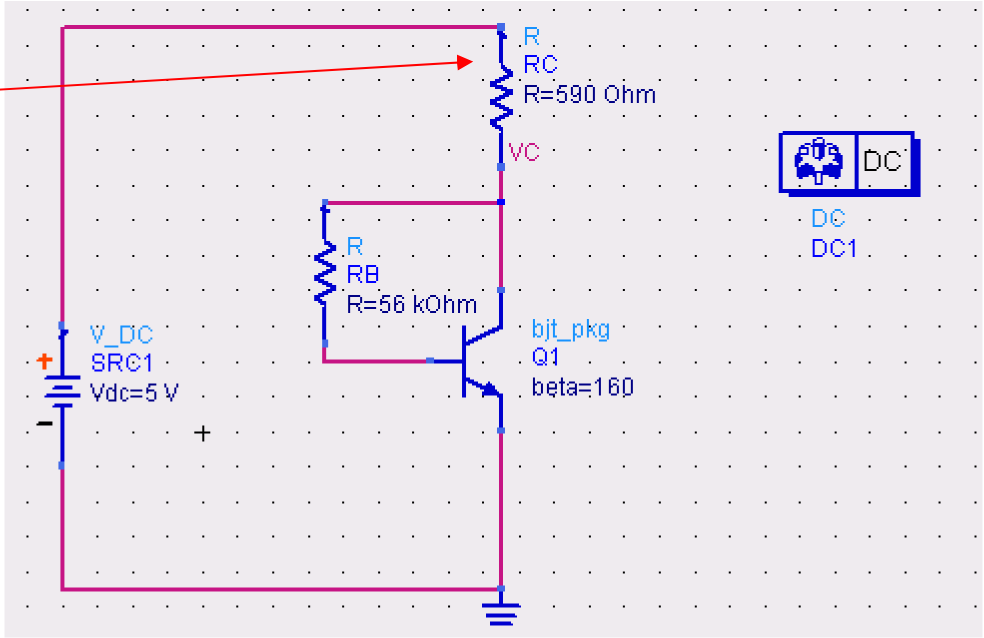
a. Save the design (dc_bias) with a new name: **dc_net**. By this time, you should know how to do this (File > Save Design As). Also, save and close the dc_bias data display.

b. Modify the design as shown. Begin by deleting the current source IBB, the I_Probe, and the Var Eqn.

c. Go to the **Lumped Components** palette or use component history or Hot Key **Ctrl R** to insert **resistors** for the base **(56 kOhm)** and collector **(590 Ohm)** as shown. You may need to use the rotate icon to insert it correctly.

d. Change the instance R names to **RC** and **RB** as shown.

Insert the cursor and type over to rename it to RC.



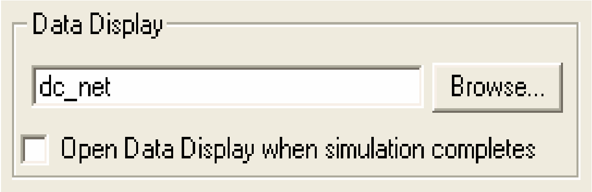
NOTE on components with artwork: Later on (after the last lab), you can easily and quickly change to lumped components with artwork by changing the component name – for example, change R to R_Pad1, C to C_Pad1, L to L_Pad1, etc. Then you can create a layout of the schematic. For now, use lumped without artwork.

e. Set the V_DC supply: **Vdc = 5V**. Wire the circuit and organize it.

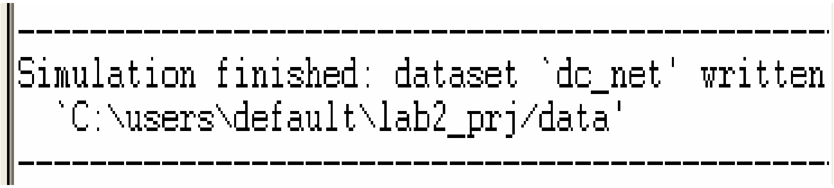
f. Delete the DC simulation controller and put a new one in its place – this is faster and more efficient than removing the sweep settings. Because there is no sweep, you do not have to set anything to check DC values.

13. Simulate and annotate the DC solution.

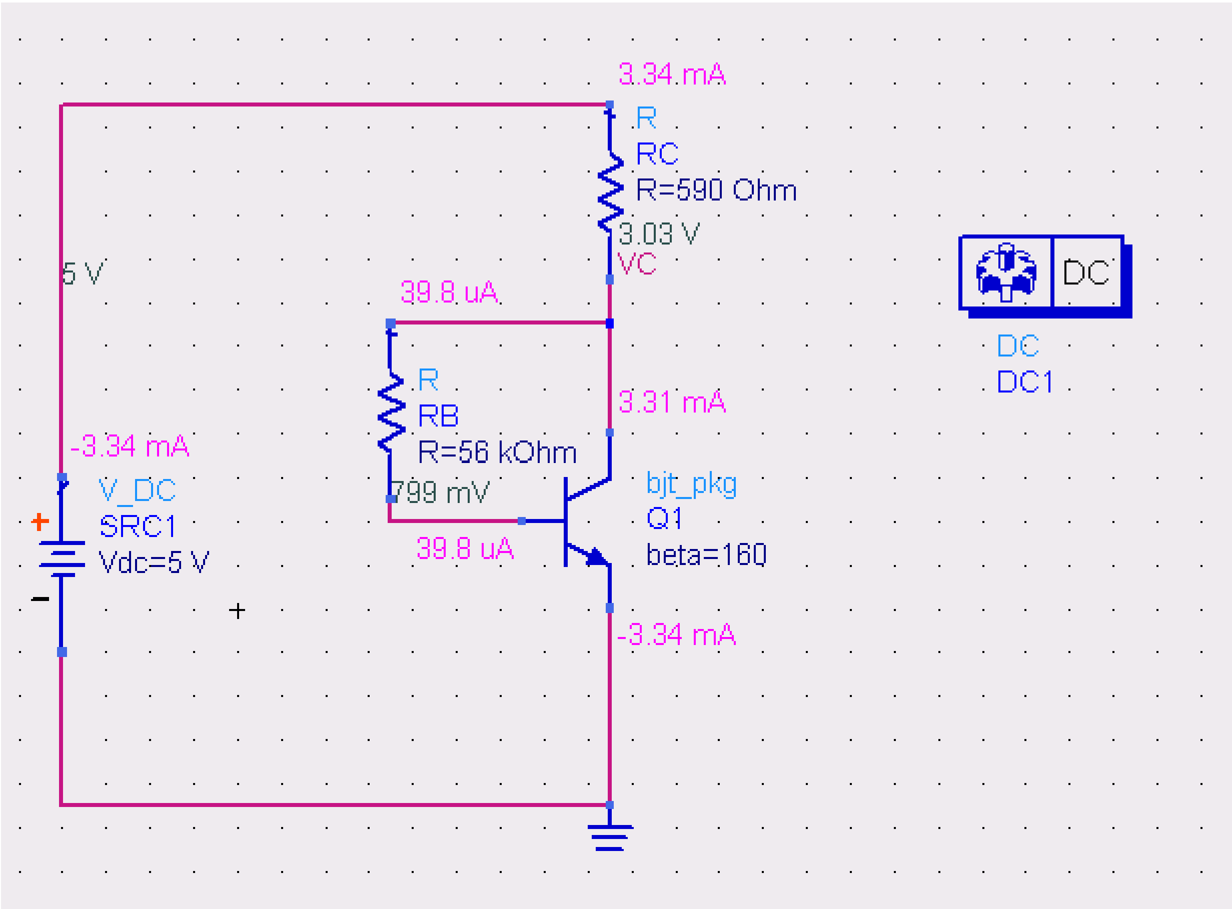
a. Use the **Simulate > Simulation setup** (or Hot Key "S" if you have set it) and **uncheck** the automatic open data display feature.



b. Click **Simulate > Simulate**, or on a PC try the **F7** keyboard key, to run the simulation. The dataset name will be the same as the schematic – this is the default. You can verify this by reading the status window.



c. Annotate the current and voltage by clicking on the menu command; **Simulate > Annotate DC Solution**. If necessary, move components or component text (F5 key) to clearly see the values of voltage and current. Be sure that you have the same values dhown here. If not, check your work, including the sub-circuit.



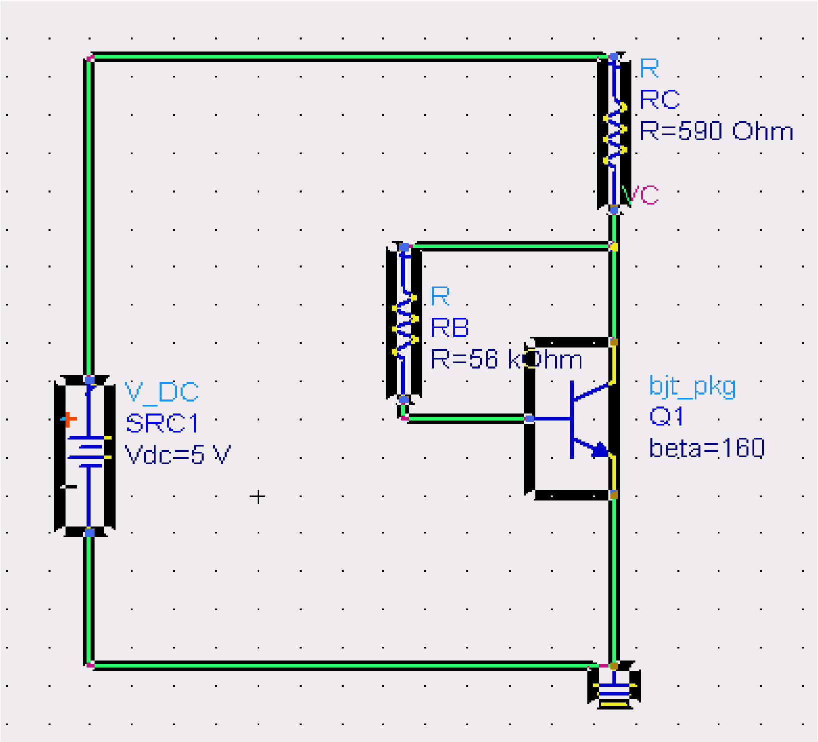
NOTE: To move pin/node names, point-click-drag.

d. Clear the annotation, click: **Simulate > Clear DC Annotation** and then **Save** all you work. Close all windows if nor doing the optional steps that follow.

Lab3: AC Simulations

1. Copy & Paste (Ctrl+C / Ctrl+V) from one design to another.

a. Open the last design (dc_net) and copy the circuit shown highlighted here by dragging the cursor around the area – this is known as rubber banding. With the items highlighted, copy then by using the keyboard keys **Ctrl+C** or the **Edit > Copy** command. Using Ctrl+C is preferred because it eliminates mouse clicks.

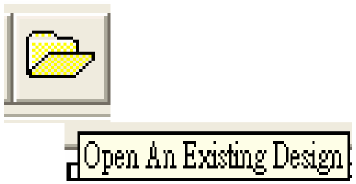


b. Use the **File > New Design** command to create a new schematic and name it: **ac_sim.** Then use **Ctrl+V** or use **Edit > Paste** and insert (ghost image) the copy by clicking into the new schematic.

c. Save the **ac_sim** design. You must save it or it will not be written to the database.

d. Click the command **Window > Designs Open**. This command gives you access to designs that are open in memory but not visible in a window or not saved in memory. When the dialog appears, select **dc_net** and click **Ok**. Then close dc_net design using **File > Close Design** (no need to save the changes).

e. In the empty schematic window, reopen the ac_sim design using the **File > Open Design icon.** This gives you a list of all the designs in the project. If a design is created but not saved initially, it will not be in this list and you will need to use the command Window > Designs Open to access it.



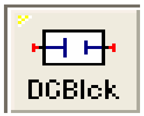
2. Modify the copied circuit and pin labels.

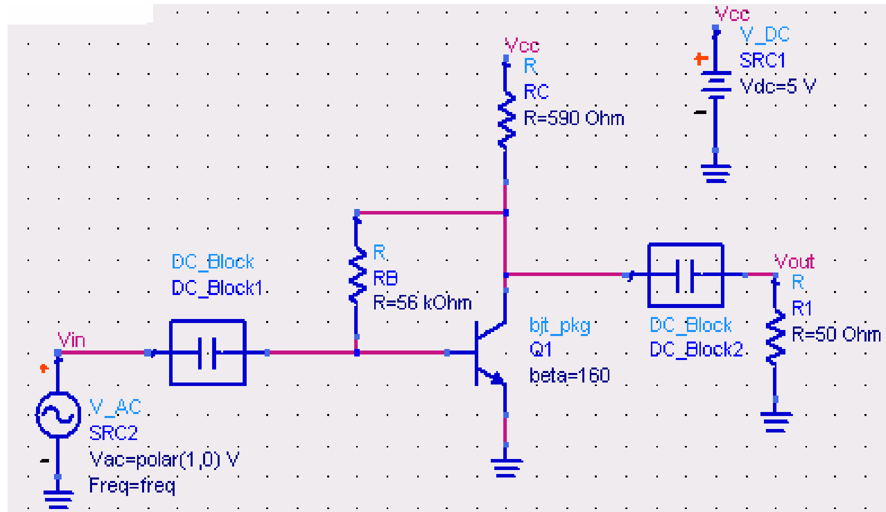
Delete wires, insert new components, and rewire as needed. The steps follow:

a. Disconnect the DC source and move it to the side with a ground.

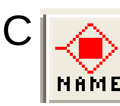
b. Insert two ideal **DC_Block** capacitors from the Lumped-Components palette or use component history.

c. Insert a **V_AC** source from the Sources-Freq Domain palette. Ground the source. Then add a **50 ohm** load resistor and ground to the output.

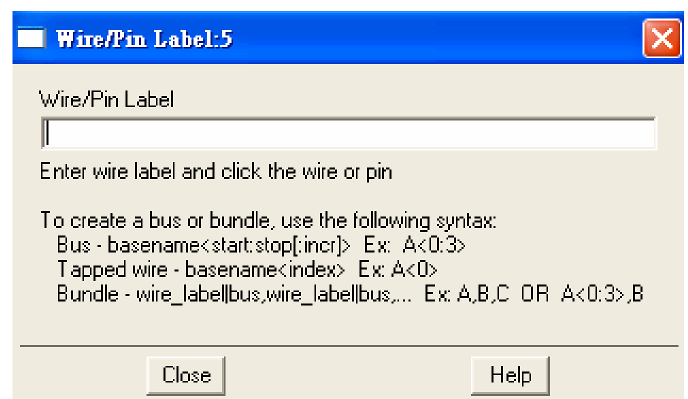




d. Modify the Pin/Wire (node) labels. Click the **Name** icon. Add **Vcc** as a label to both RC and the DC source. This will connect them electrically instead of a wire.

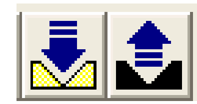


e. Add **Vin** and **Vout** as shown. Also, if you did any OPTIONAL steps in lab3, remove VC and VBE by clicking on those labels when the dialog is blank (shown here) or use the command: **Edit > Wire/Pin Label > Remove Wire/Pin Label.**



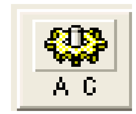
f. Verify that the circuit looks like the one shown here.

3. Push and pop to verify the sub circuit.



a. Select the bjt_pkg and **push** into the sub-circuit (use the icons) to check your sub circuit, and then **pop** out again.

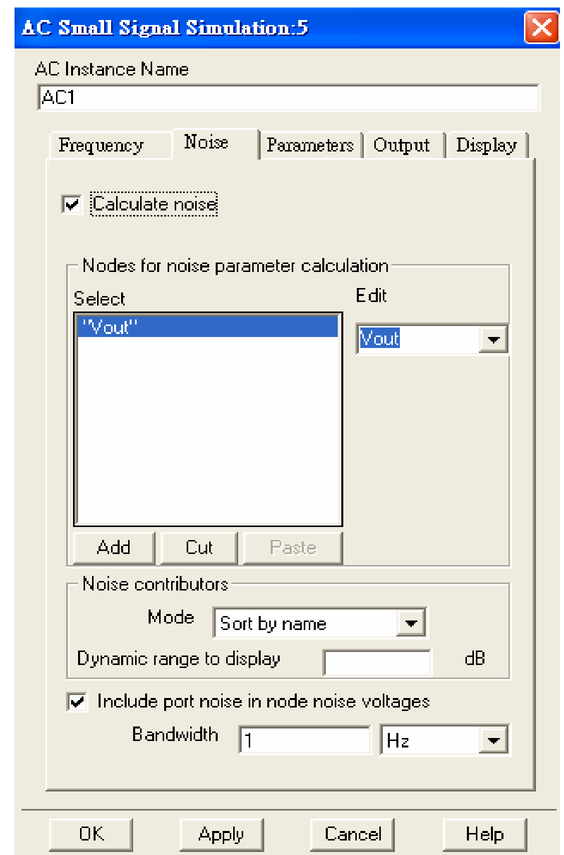
4. Set up an AC simulation with Noise.



a. Insert an AC Simulation controller. Then edit the start, stop, and step frequencies: 100 MHz to 4 GHz in 100 MHz steps.

b. In the Noise tab, check the box for **Calculate noise** and add the **Vout** node. Set the Mode to **Sort by Name** for each noise contributor. Sort by value is good for large circuits to see the largest contributors first. Also, all noise values will be simulated if a Dynamic range (threshold) is not set.

c. Turn on the **Display** for each of the parameters as shown here.

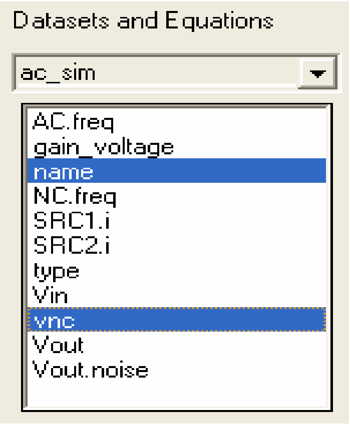


5. Simulate and list the noise data.

a. Simulate (F7).

b. In the data display, insert a **list** (icon) of **name** and **vnc** (voltage noise contributors) using the **Ctrl** key to select them both. As shown here, at each frequency, Q1.BJT1 is the total noise voltage for the device and is composed of: Q1.BJT1.ibe and Q1.BJT1.ice. However, these are not correlated voltages but have been added as noise powers: $(V_{total})^2 = (V_{ibe})^2 + (V_{ice})^2$. The total vnc is the same as Vout noise.

c. Save the schematic and data display.

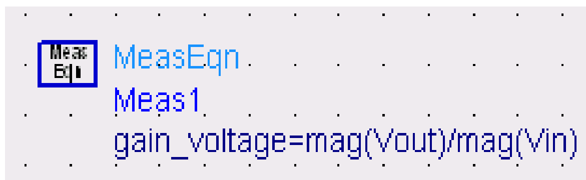


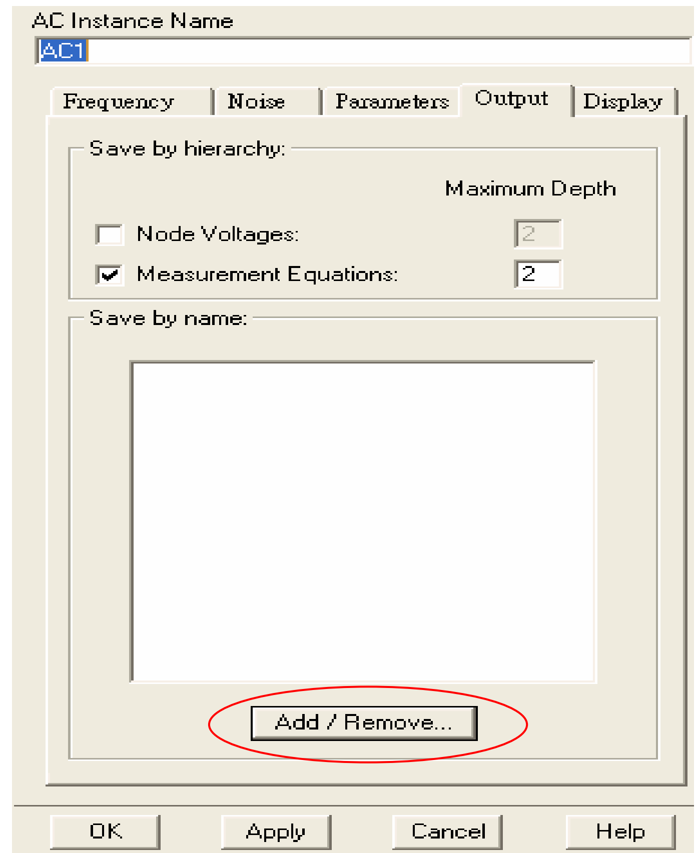
name	vnc
freq=100.0MHz	
Q1.BJT1	1.494nV
Q1.BJT1.ibe	8.479pV
Q1.BJT1.ice	1.494nV
Q1.L1	75.74pV
R1	833.3pV
RB	24.90pV
RC	242.6pV
_total	1.729nV
freq=110.0MHz	
Q1.BJT1	1.494nV
Q1.BJT1.ibe	9.325pV
Q1.BJT1.ice	1.494nV
Q1.L1	75.73pV
R1	833.3pV
RB	24.90pV
RC	242.6pV
_total	1.729nV
freq=120.0MHz	
Q1.BJT1	1.494nV
Q1.BJT1.ibe	10.17pV
Q1.BJT1.ice	1.494nV
Q1.L1	75.72pV
R1	833.2pV
RB	24.90pV
RC	242.6pV
_total	1.729nV
freq=130.0MHz	
Q1.BJT1	1.493nV
Q1.BJT1.ibe	11.02pV
Q1.BJT1.ice	1.493nV

6. Control the output of equations and node voltages.

a. In the ac_sim schematic, insert a **MeasEqn** from any simulation palette. Or, you can type in **MeasEqn** in component history.

b. Directly on the schematic screen, edit (type) the equation to compute voltage gain using the node (pin) labels Vin and Vout. Use the keyboard arrow key to move across the equal (=) sign.

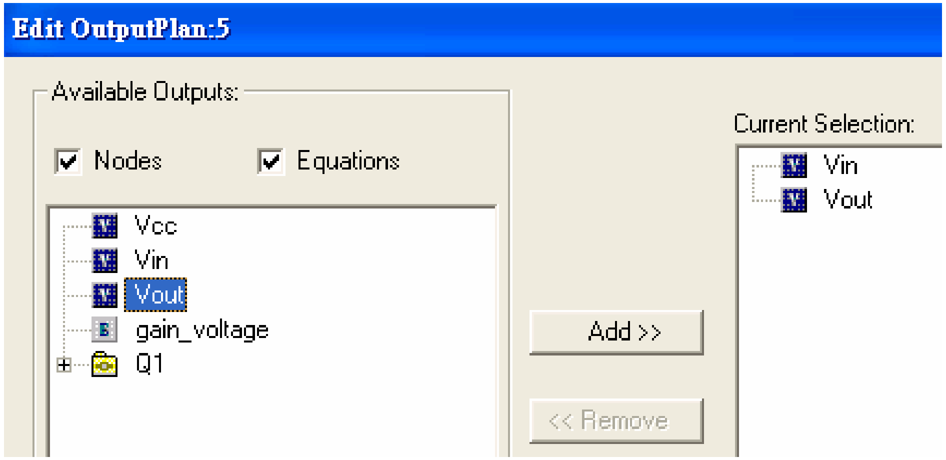


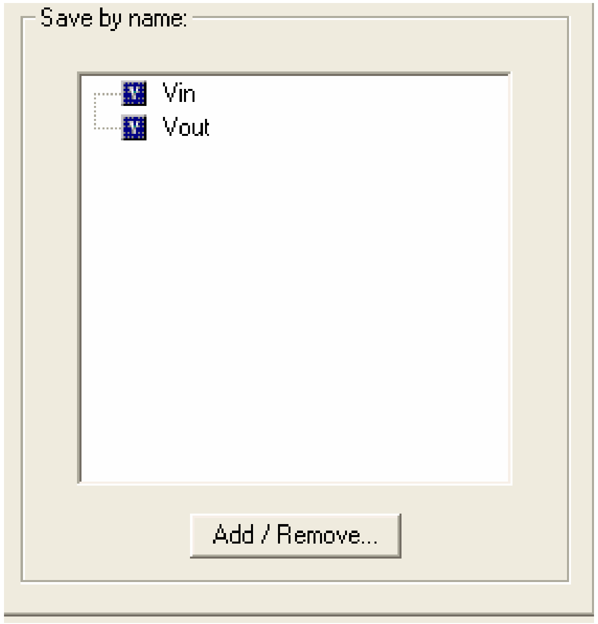


c. Edit the AC simulation controller and go to the **Output** tab. The default is for all labeled node voltages (pin/wire labels) and all Measurement equations to be reported in the dataset. You will change this in the next steps.

d. Uncheck the box for Node Voltages and click on the **Add/Remove** button.

e. Select **Vin** and **Vout** from the list of available outputs and **Add** them as shown here – then click **OK.** Only those node voltages will be written into the dataset after simulation and **Vcc will not**. This works for measuremnt equations also.





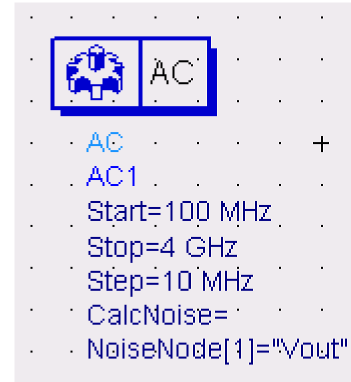
d. Click **OK** to dismiss the dialog – you are now ready to simulate.

NOTE on node name display: You can display the node names (Display tab – NodeName check box) but it is not necessary.

7. Simulate without noise

a. In the schematic, turn off the noise calculation by changing (typing) yes to **no** as shown here. This will save simulation time and memory, especially for large circuits. Of course, this will make your dataset list (name and vnc) invalid.

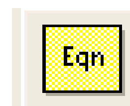
b. Save the schematic and **Simulate** (F7)



8. Write a data display equation using a measurement equation.

a. In the data display, **delete** the invalid noise listing.

b. Insert a data display equation (use the icon).

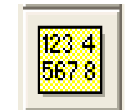


c. In the dialog, write an equation for the gain in dB as shown here. Notice that you are inserting the schematic measurement equation into your data display equation and click **OK**.

Eqn gain_dB=20*log(gain_voltage)

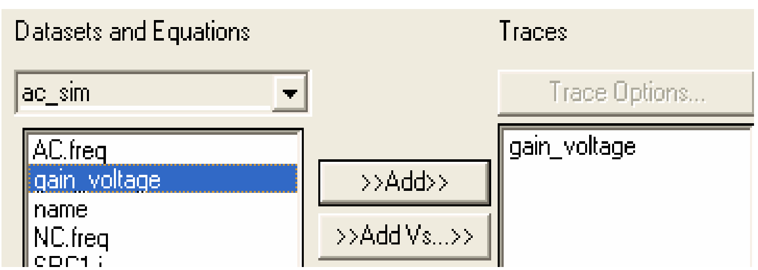
NOTE on equations – If the measurement equation for voltage gain was not already calculated, you would write the data display equation with all the required values, for example: gain_dB=20* log (mag(Vout)/mag(Vin)). However, because that voltage gain was already calculated, it is easier to simply insert it here.

9. Work with measurement and data display equations.

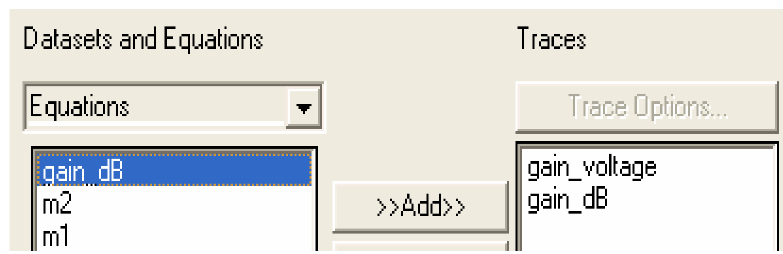


a. Insert a list of the measurement equation **gain_voltage** and the DDS equation you just wrote **gain_dB**. Again, schematic measurement equations are automatcially written into the dataset as shown here. But equations you write in the data display are not – they are accessed in the data display Equations memory. To display your dataset equation, gain_dB, click on the arrow box (shown here) and then select it and add it. Click **OK** and both equations will appear in the list.

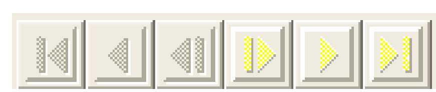
Measurement Equations listed here.



Data Display equations listed here.



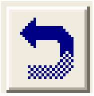
b. Scroll down the list to values around 1900 MHz, using the arrow buttons as shown.



c. Insert the cursor directly into the gain_voltage column and type in the **dB** function as shown. Then add **parentheses** so that it reads: **dB (gain_voltage).** This demonstrates the flexibility of the data display for operating (with ADS functions) directly on data and equations.

freq	dB(gain_voltage)	gain_dB
100.0MHz	15.419	15.419
110.0MHz	15.418	15.418
120.0MHz	15.417	15.417
130.0MHz	15.417	15.417
140.0MHz	15.416	15.416
150.0MHz	15.415	15.415
160.0MHz	15.414	15.414
170.0MHz	15.413	15.413
180.0MHz	15.412	15.412
190.0MHz	15.410	15.410
200.0MHz	15.409	15.409
210.0MHz	15.408	15.408
220.0MHz	15.406	15.406
230.0MHz	15.405	15.405

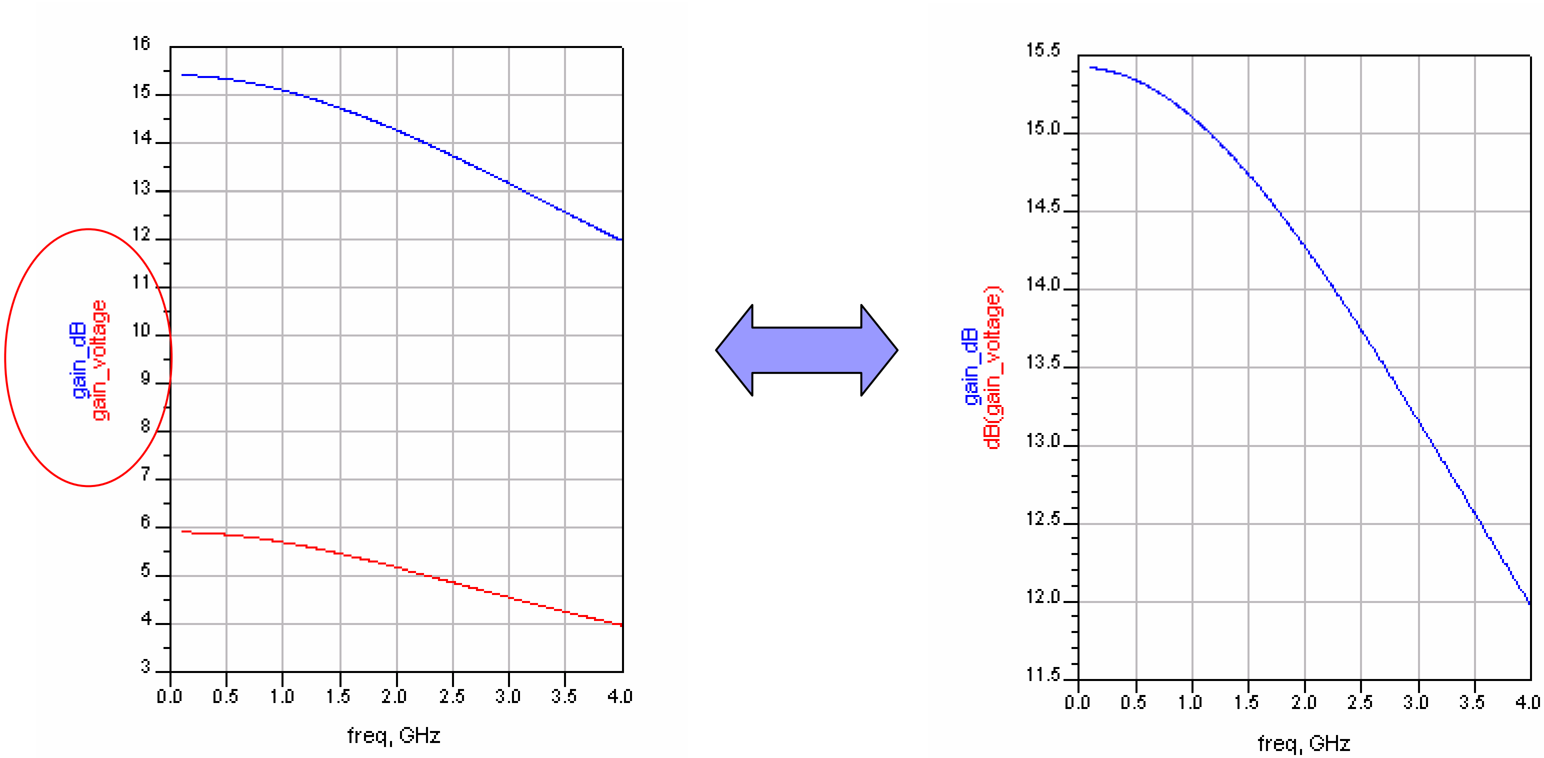
d. Click the data display **Undo** command to return to remove the dB function.



e. Edit the list (double click) and change it to a **rectangular plot** by selecting the icon.

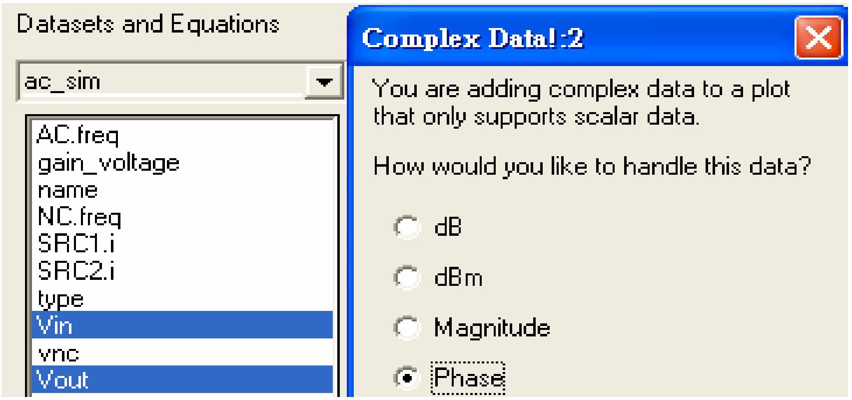
f. Insert the cursor directly onto the Y-axis label and change gain_voltage to dB(gain_voltage) similar to the way you did in the list. Then undo it. Again, this shows the power of functions and the data display.

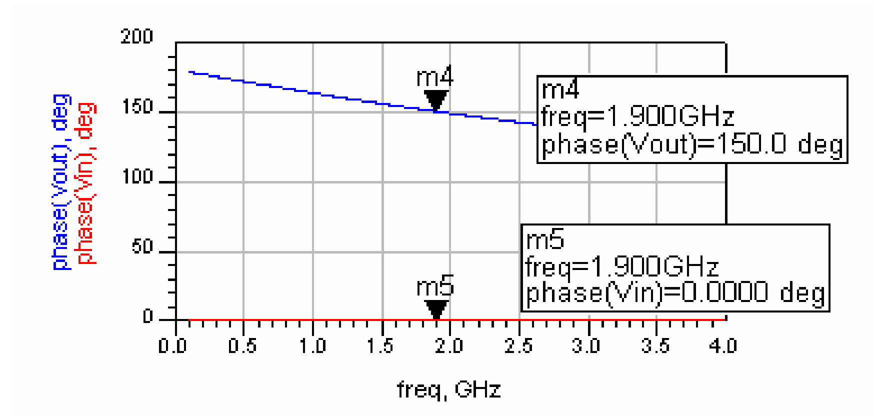
NOTE on dB values – Converting the AC analysis voltage to dB is not the same as S-parameter analysis in dB that uses power (V and I) and also has a 50 ohm source Z.



10. Plot the phase and group delay for the ac analysis data.

a. Insert a rectangular plot of the **phase** of **Vin** and **Vout** – put markers on 1900 MHz (type in the value). The phase is not 180 degrees due to the bjt_pkg parasitics. Move the markers and see the phase closer to180 at lower frequencies. You may want to **Hot Key** the new marker command using the DDS Options > Hot Key similar to schematic.

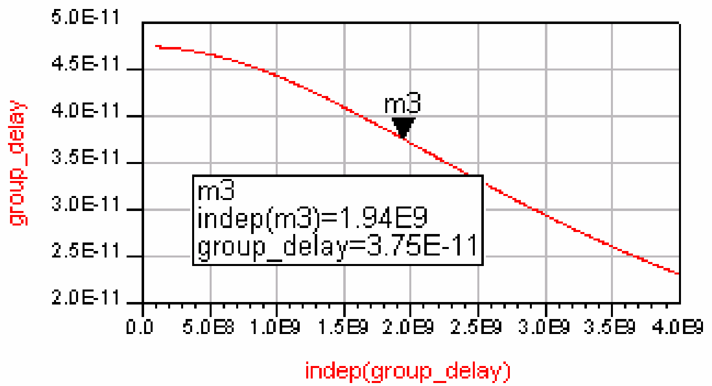




b. Insert a new **equation** to calculate **group delay**. As shown here, use the phase of Vout and the **diff** function then **plot** the equation. The **diff** function calculates the difference between points on the slope. The minus sign gives the result in decreasing value. Place a marker on the trace and notice that it will be on either side of 1900 MHz (+/- 50 MHz) because of the *diff* function.

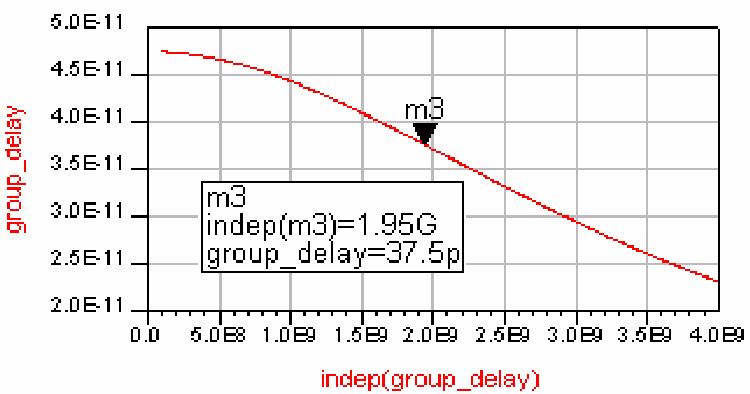
Eqn group_delay=(1/360)*(-diff(phase(Vout)))

c. Go back to the schematic, change the **step** size to **10 MHz**, **simulate** again and watch the plot update.



d. Edit (double click) the **marker**. In the **readout** tab, set Format to **Engineering** with **2** significant digits as shown here. Notice the marker value changes to pico (pico-seconds) and the independent value resolves to 1.90 GHz.

e. OPTIONAL-Try grouping the group delay equation and the plot so they stay together when you move them. Use the Shift key and select the plot and the equation. Then click: **Edit > Group**. They should now move together in the data display.



Course Topics

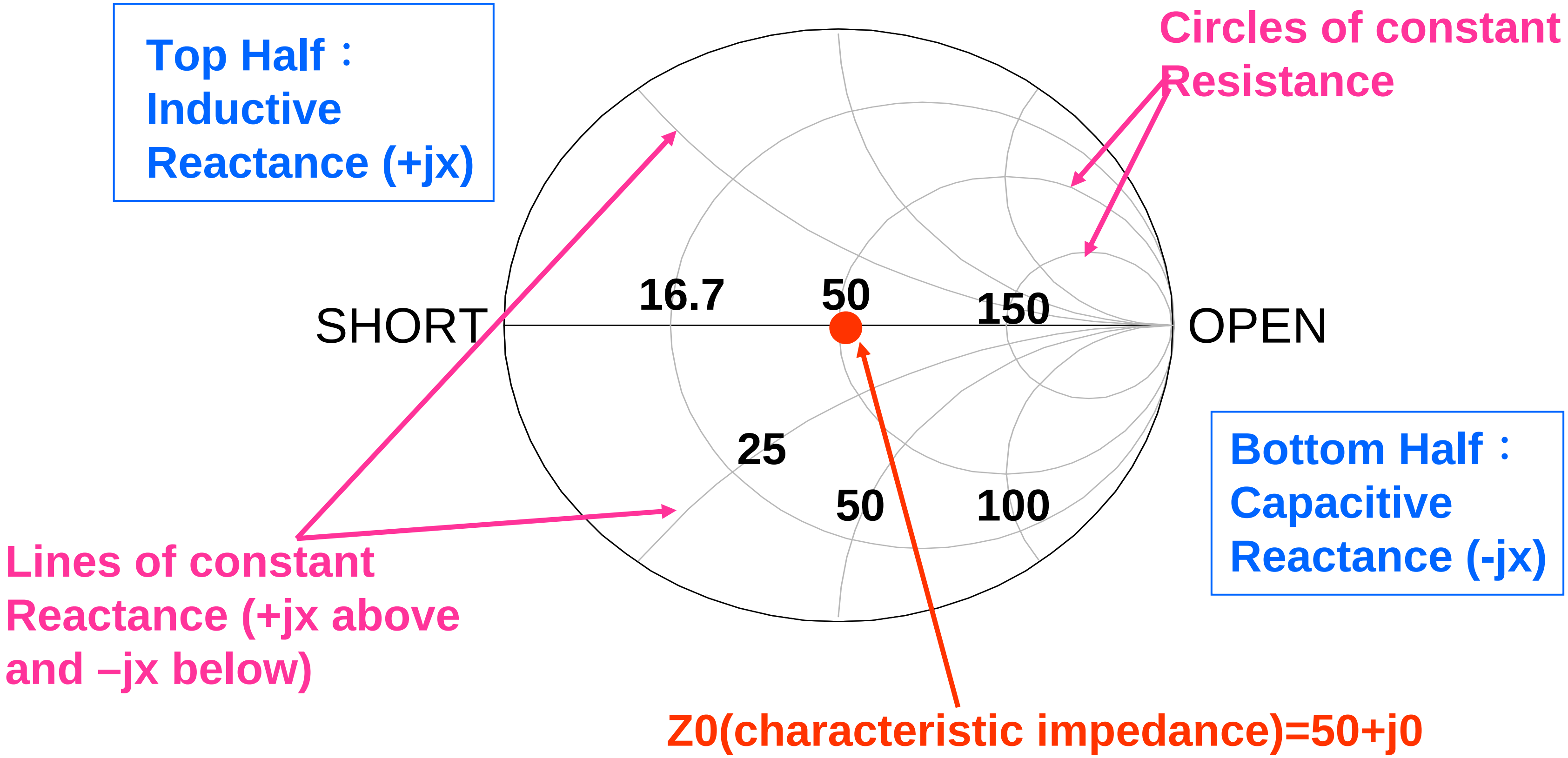
- 1:Circuit Simulation Fundamentals
- 2:DC Simulation and Circuit Modeling
- 3:AC Simulation and Tuning
- 4:S-Parameter Simulation and Optimization

S-parameters

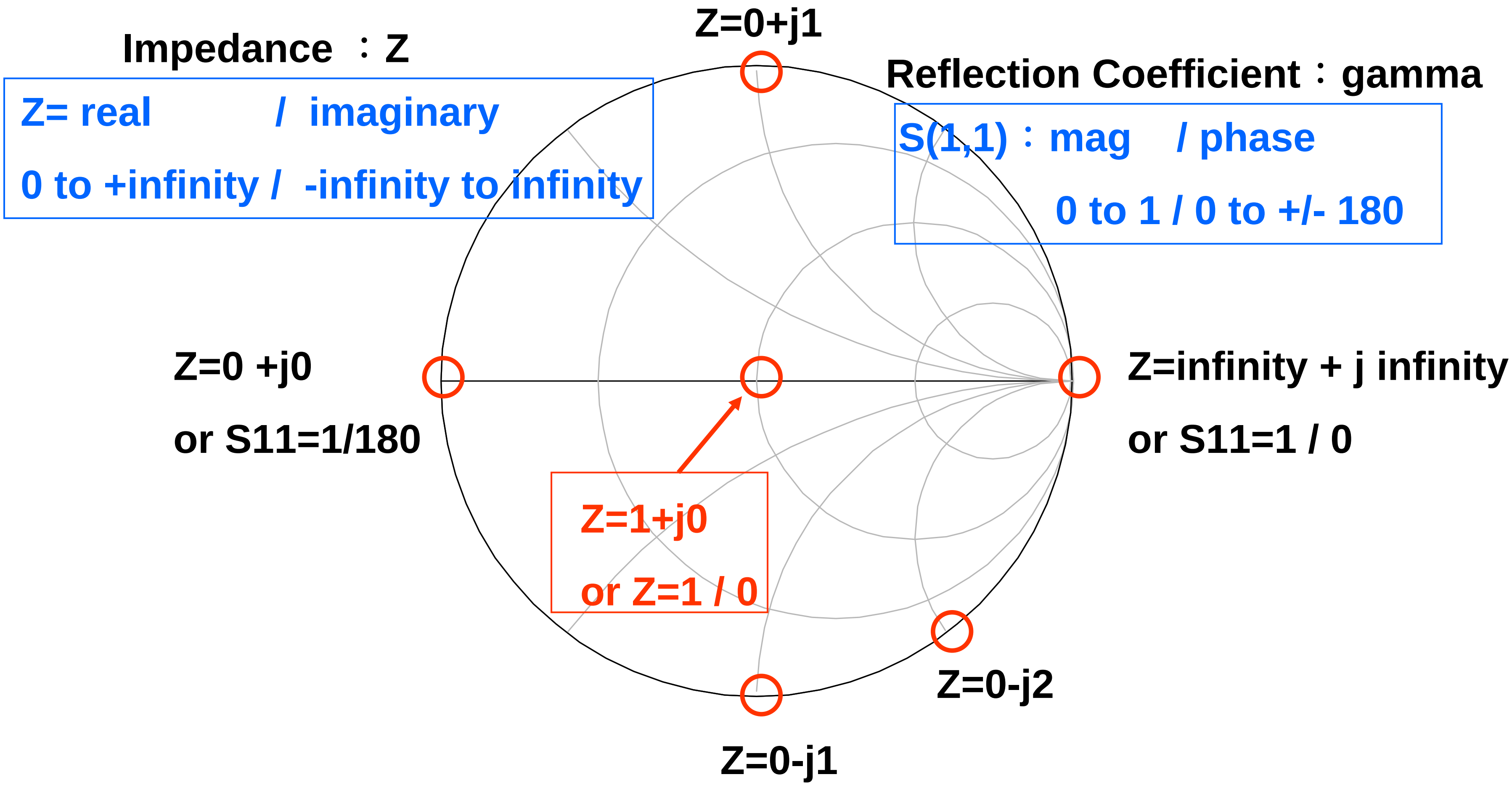
- Usually given in dB as 20 log of the voltage ratios of the waves at the ports：incident, reflected, or transmitted
- S11-Forward Reflection (input match – impedance)
- S22-Reverse Reflection (output match –impedance)
- S21-Forward Transmission (gain or loss)
- S12-Reverse Transmission (leakage or isolation)
- S11 and S22 are best viewed on a Smith chart
- S21 and S12 are easier to understand and simply plotted

The impedance Smith Chart

- **This is an impedance chart transformed from rectangular Z. Normalized to 50 ohms, the center=R50+J0 or Zo(perfect match).**

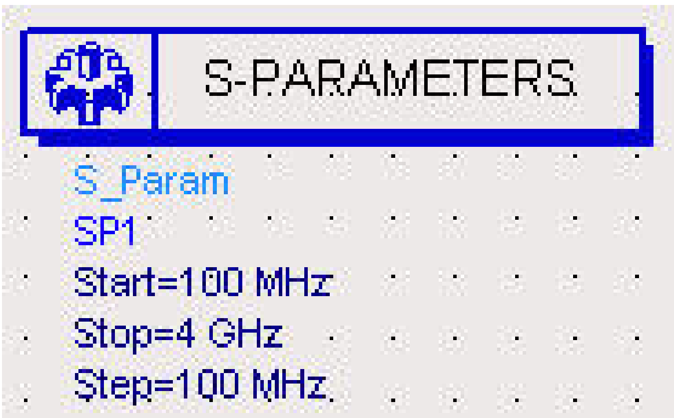


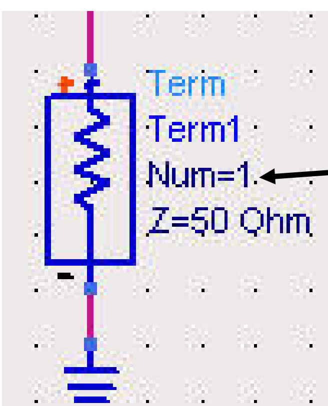
The Smith chart in ADS Data Display



S-Parameter Simulation Controller

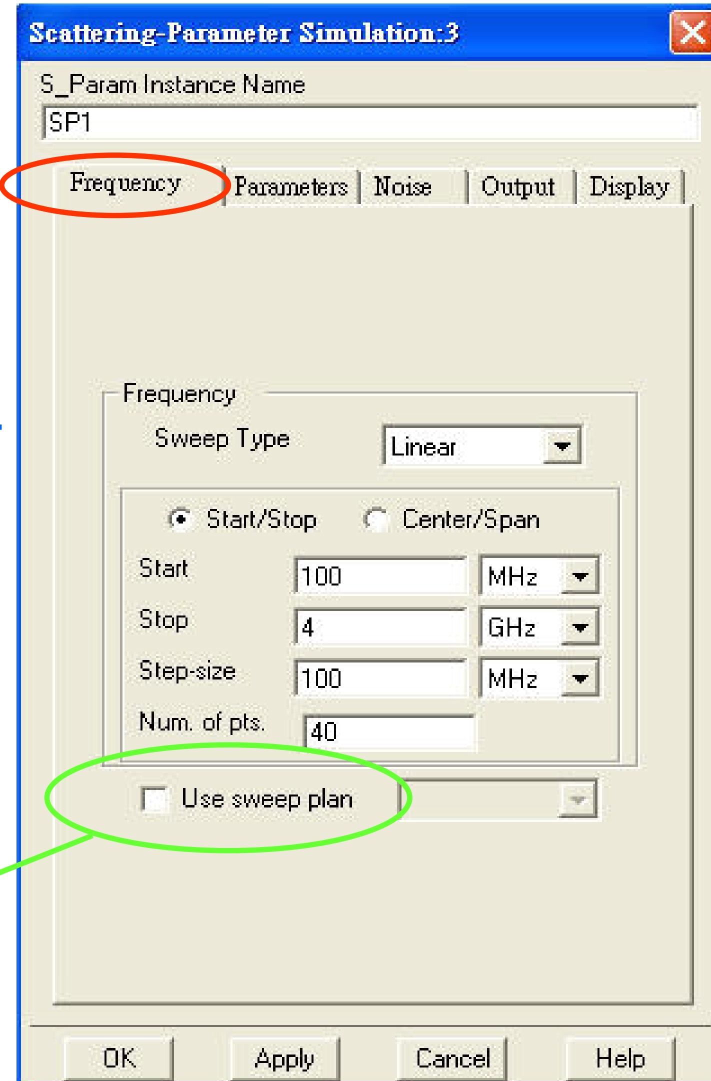
Default sweep variable = **freq**





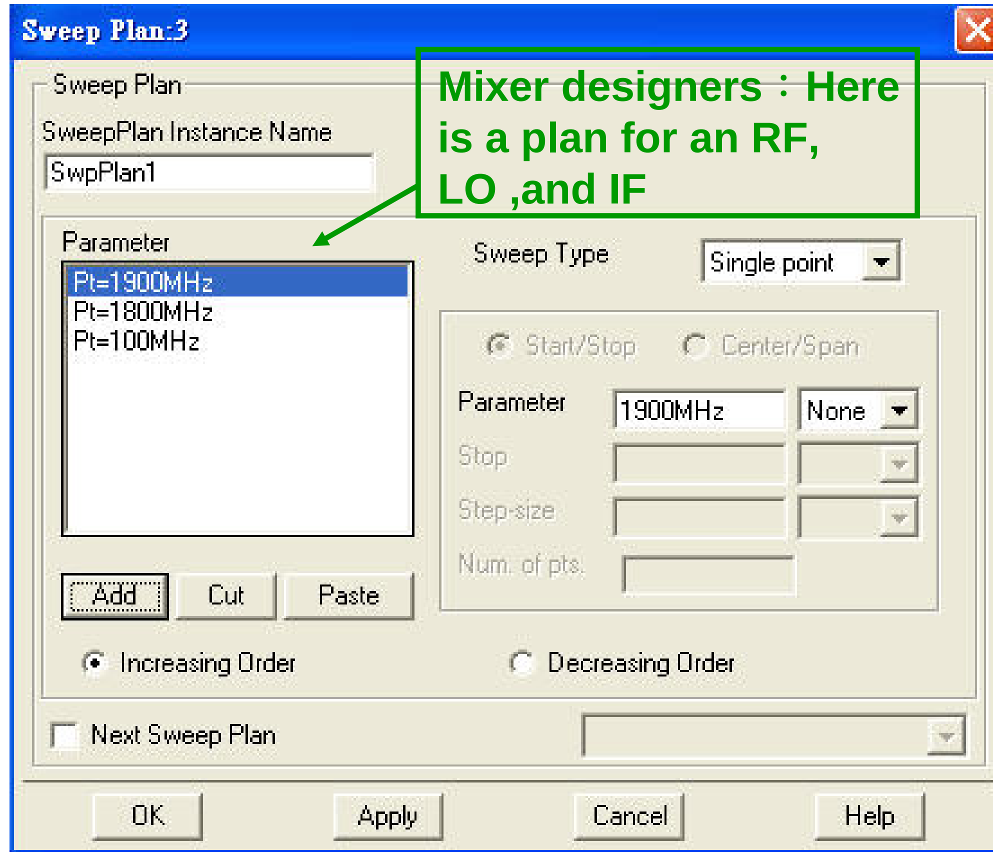
The simulator requires a port termination Num=__

Next slide



Sweep Plan with S-parameter simulations

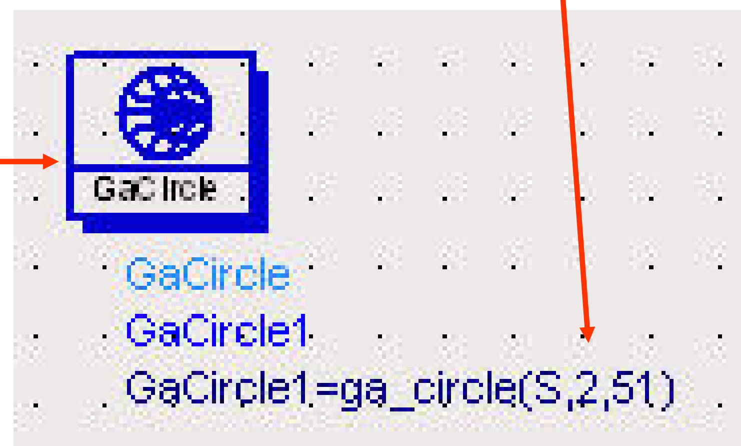
Sweep Plan is for sweeping **FREQ**. Otherwise, use a **Parameter Sweep** for variables(Vcc, pwr, etc.)



Here is a sweep within a sweep

S-Parameter measurement equations

All simulation palettes have specific measurement equations – you set the arguments if necessary. Here, S is the matrix, 2 is the value in dB, and 51 point used to draw the circle

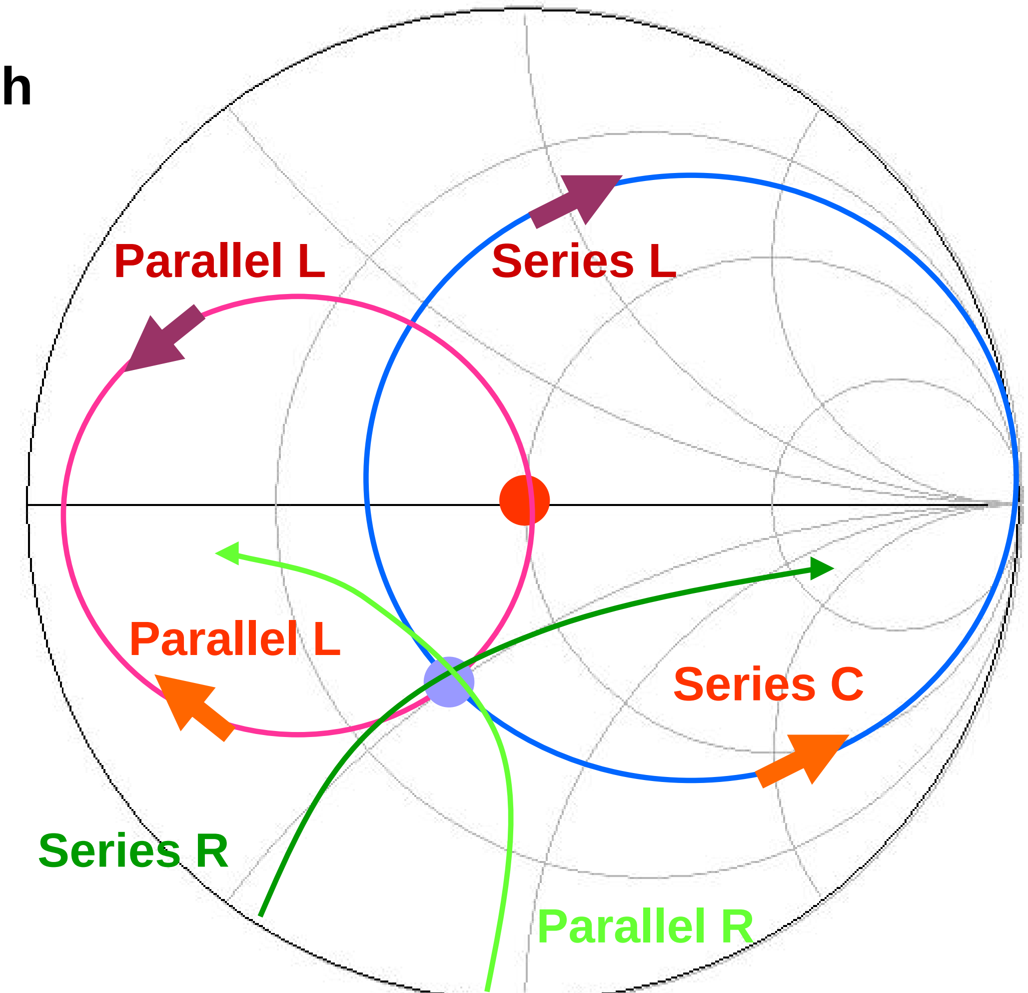


Stability circle

Noise circle

Matching Networks

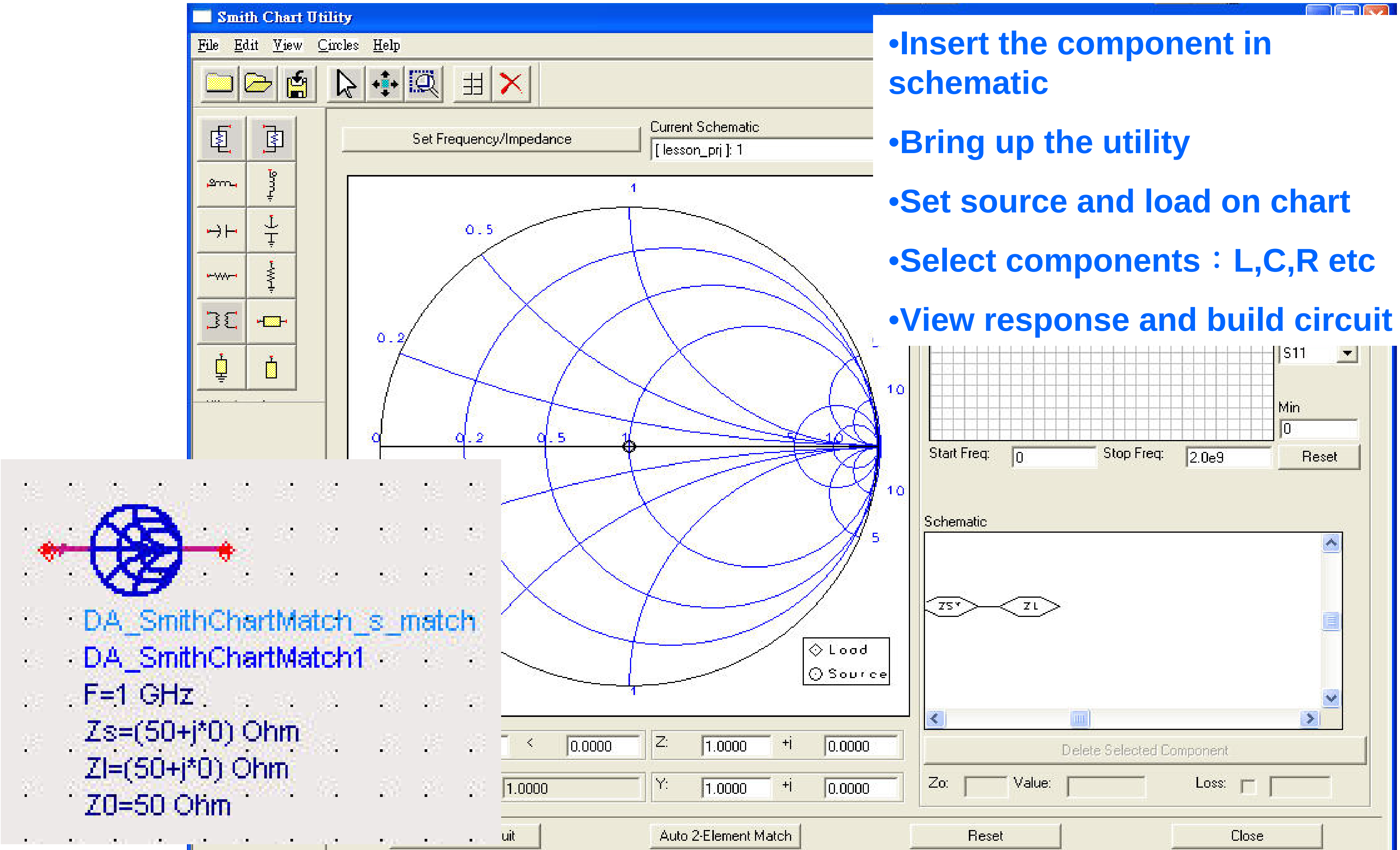
Matching means：Moving toward the center of the Smith Chart



Add Series of Parallel components

Adjust the value to move toward open , short , L , C , or center of chart

Design Guide – Smith Chart Utility for matching



- **Insert the component in schematic**
- **Bring up the utility**
- **Set source and load on chart**
- **Select components：L,C,R etc**
- **View response and build circuit**

ADS Optimization Basics

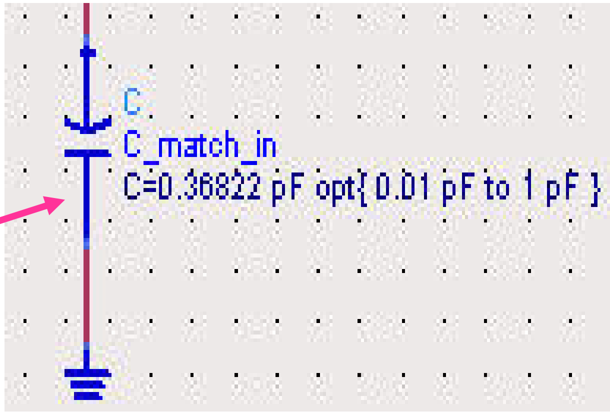
- Definition：Optimization is a simulation that tries to achieve a performance goal.
- Start with a simulation that gives you results
- Set up the optimization which includes：
  - An optimizer type and search method
  - A specific goal or specification to be met
  - Enable components or parameters to be adjusted

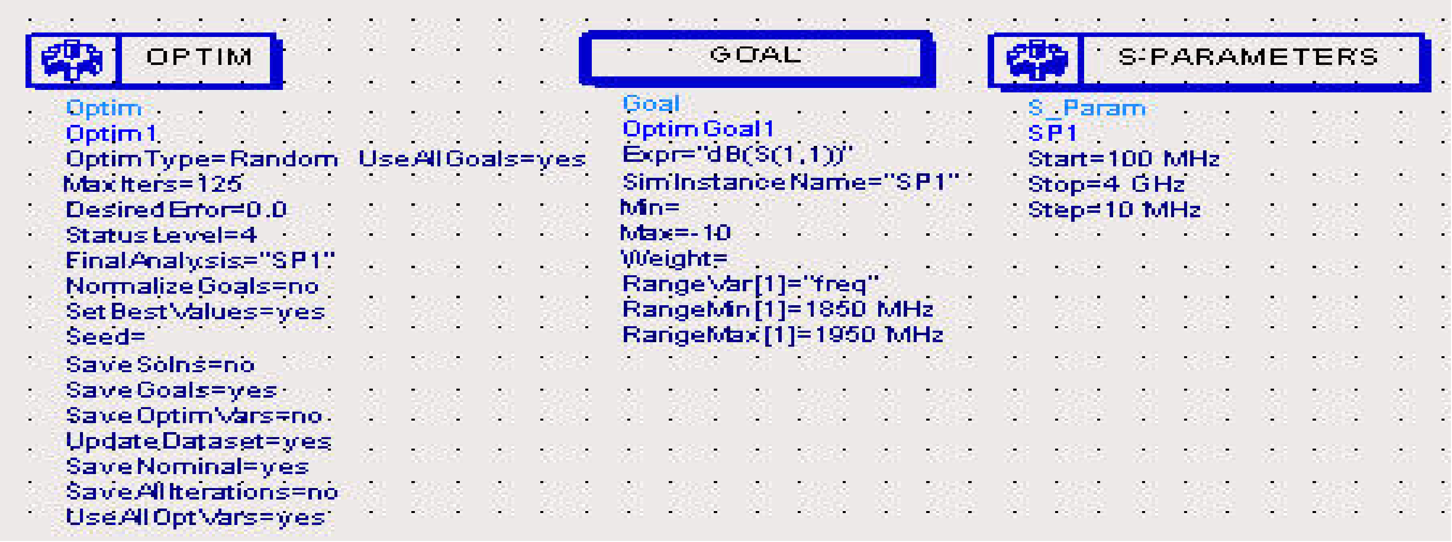
Four elements for Optimization setup

1-Optim controller：set the type

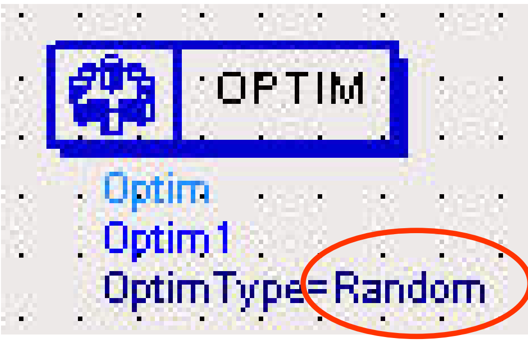
2-Goal statement：use valid measurement equation or data set expression

3-Enable component





Error Function Formulation(1)



Optimizer Type determines Error Form

Error Function Formulation	Optimizers
Least-squares	Gradient , Quasi-Newton , Random , Discrete , Genetic
Minimax	Gradient Minimax , Minimax , Random Minimax
Least P^{th}	Least P^{th}
Worst-case	Random Maximizer

Error Function Formulation(2)

- Least Squares：Each residual is squared and all terms are then summed. The sum of the squares is averaged over frequency
- Least Pth：The Least Pth EF formulation is similar to L2 , except that instead of squaring the residuals , it raises them to the Pth power with P=2 , 4 , 6 etc
- Minimax：attempts to minimize the largest of the residuals . This tends to result in equal ripple responses
- Worst case：minimizes the reciprocal of the least squares error function. This effectively maximizes the error function. The goal is to find a worst typical response for a given set of parameters

Goals and Error Function

- **The goals are minimum or maxi mum target values**
- **The error function is base on the goal**
- **The weighting factor prioritizes multiple goals**

Error function is defined as a summation of residuals. A residual r_i may be defined as：

$$r_i = W_i \, |m_i - S_i|$$

Si is the simulated ith response

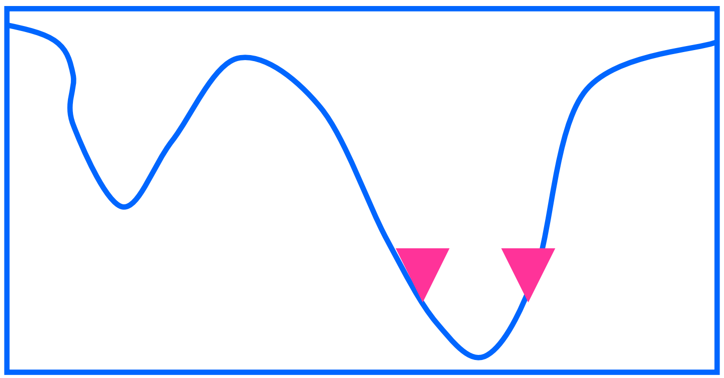
Mi is the desired response for the ith measurement

Wi is the weighting factor for multiple goals：higher number is greater

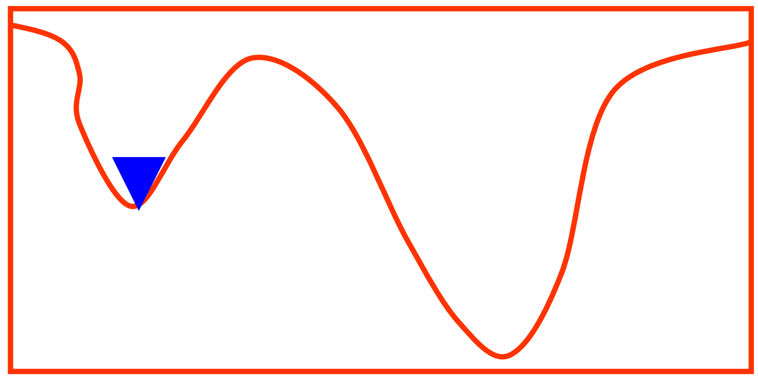
✓Simulations continue until the maximum iterations is reached or the reeor function(summation of the residuals)reaches zero (same as 10dB)

Search Method examples

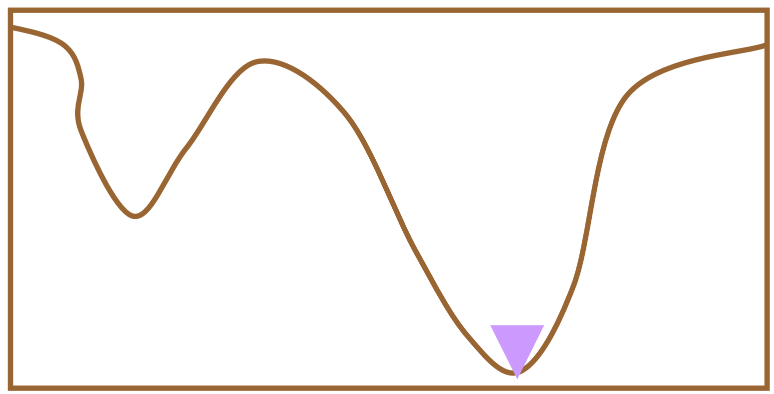
Random analysis often get you close to the goal(minimum error function)



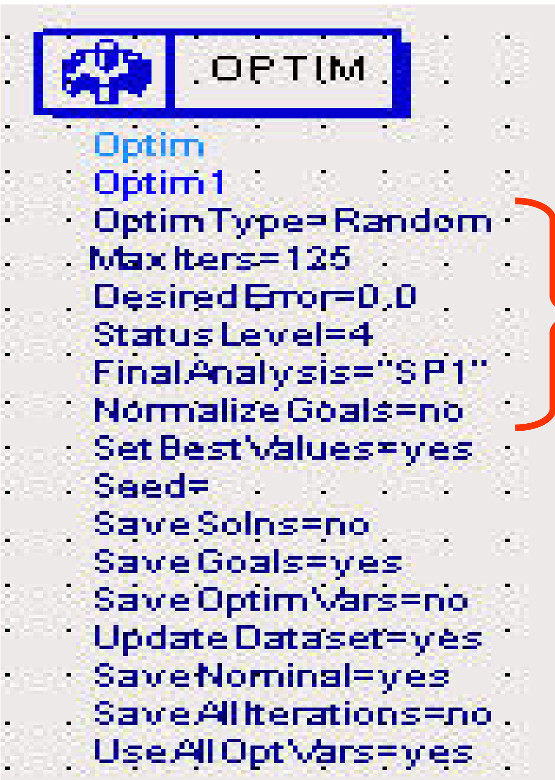
Gradient analysis may get stuck in a local minimum(not optimal error function)



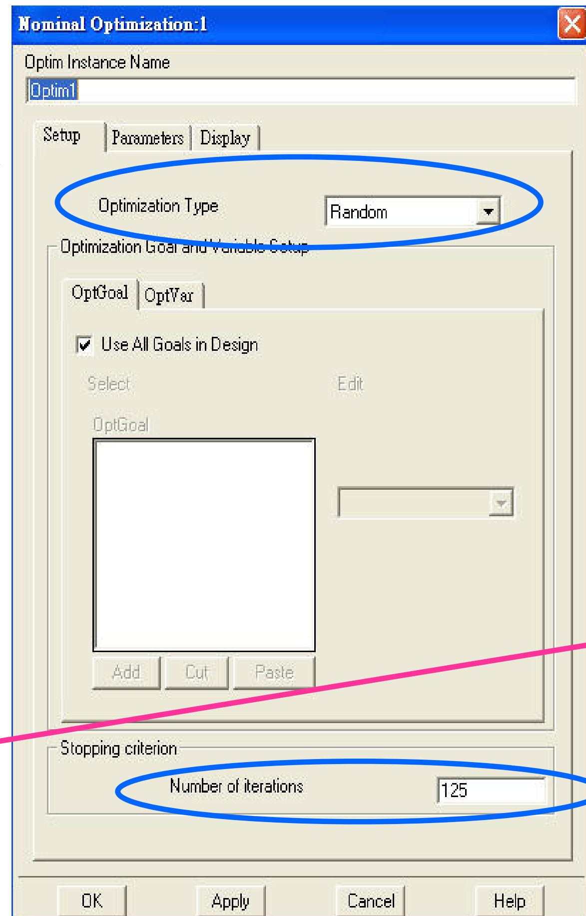
Using both RANDOM and GRADIENT can reach the desired goal or in some cases , a hybrid type such as Genetic



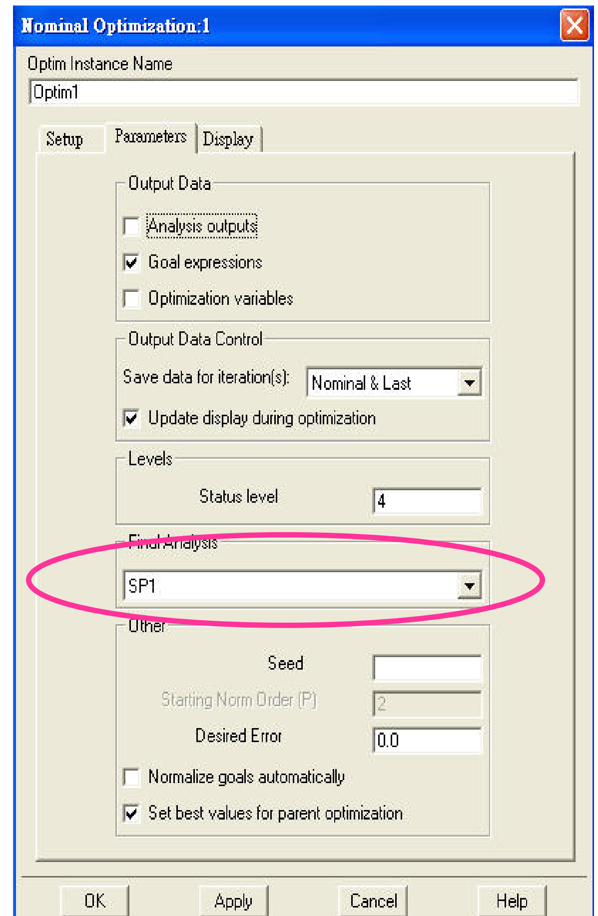
Optimization Controller setup



- **Setup tab：Select type and set iterations.**



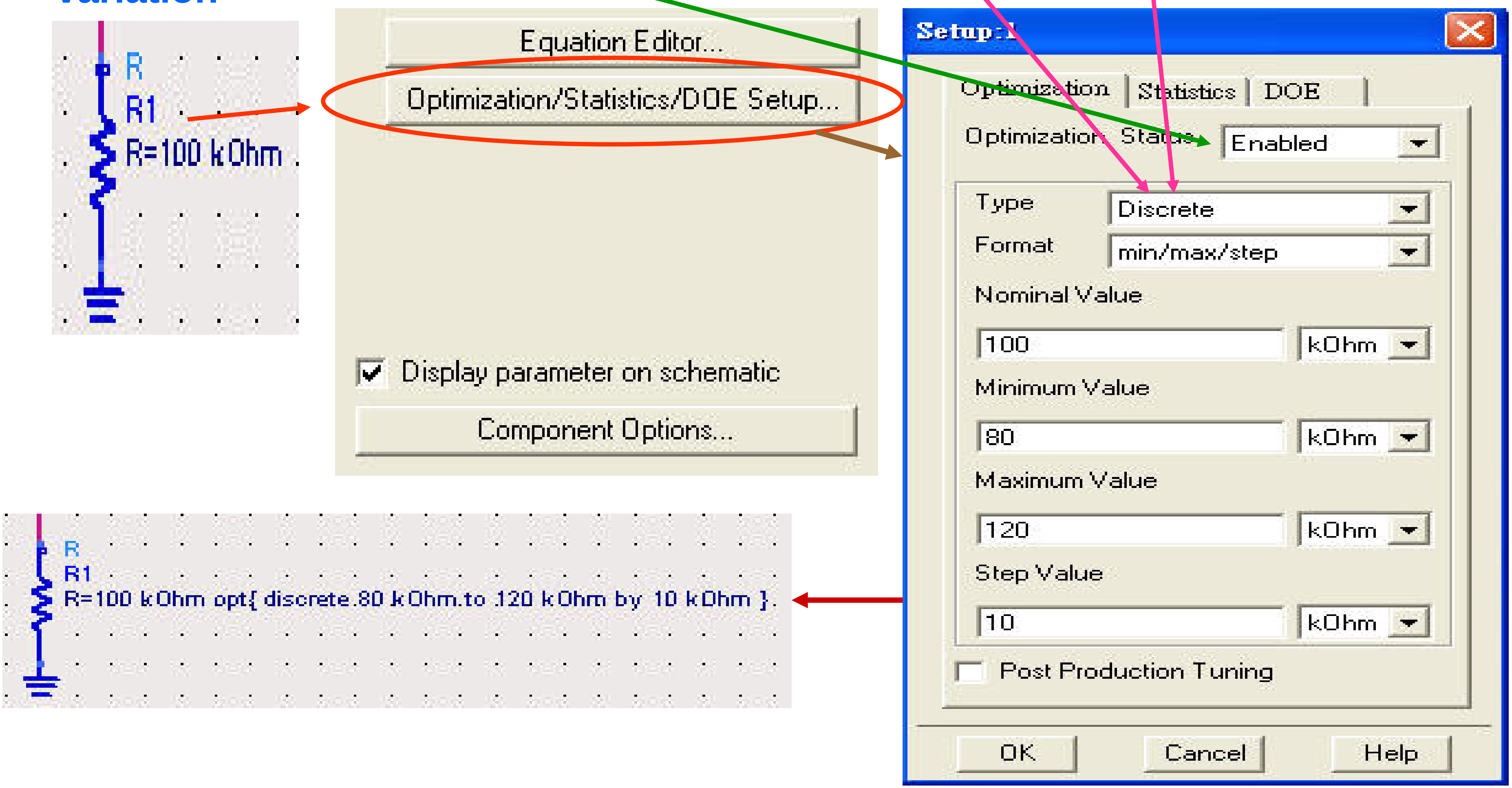
- **Parameters tab：type, iterations, etc.**



- **Final Analysis：change to SimInstanceName(SP1)**

Enabling components for Opt or Stats (yield)

Edit (double click) – Enable and specify continuous or discrete (stepped) variation



Lab 4 : S-parameter Simulations and Optimization

➢ **The circuit you build will be used as the lower level sub-circuit.**

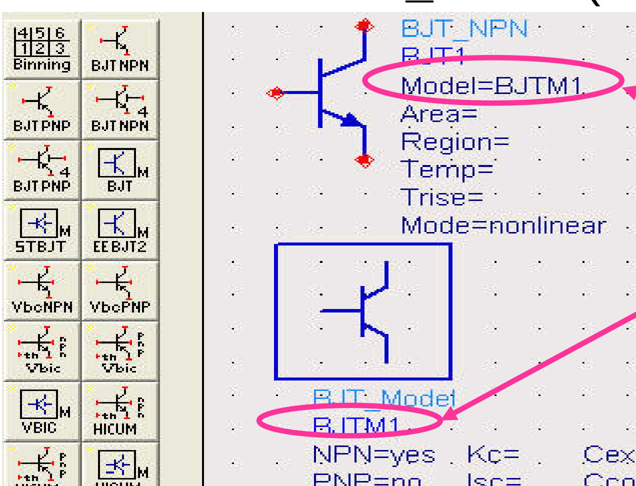
✓ **Create a new project S_parameter_Simulation_and_Optimization**

a. Open a new schematic window and save it with the name : **bjt_pkg**

✓ **Set up a generic BJT symbol and model card**

a. In the schematic window , select the palette : **Devices-BJT**. Select the **BJT- NPN** device shown here and insert it onto the schematic.

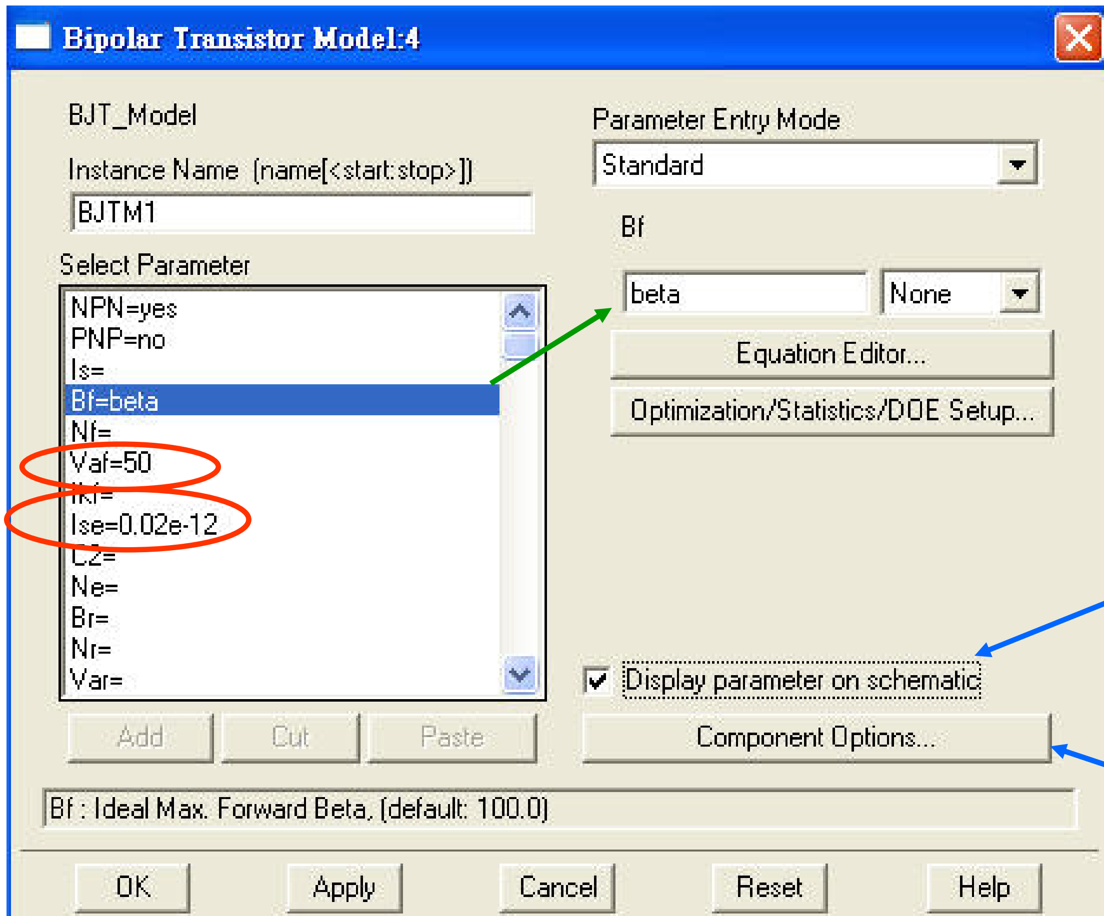
b. Insert the **BJT_Model** (model card)shown here.



NOTE : The BJT_NPN symbol shows Model=BJTM1.This means the symbol will use that specific model (model card) for simulation

Lab 4 : S-parameter Simulations and Optimization

c. Next , in **BJT_Model** dialog , select the **Bf** parameter and type in the word **beta** as shown here. Also , click the small box : **Display parameter on schematic** for **Bf** only and then click **Apply**. Beta is now a parameter of this circuit – later on you will tune it like a variable.



d. Set **Vaf** (Forward Early Voltage) = **50** and display it

e. Set **Ise** (E-B leakage) = **0.02e-12** , and display it also. Then close the dialog with **OK**. The device now has some more realistic parameters

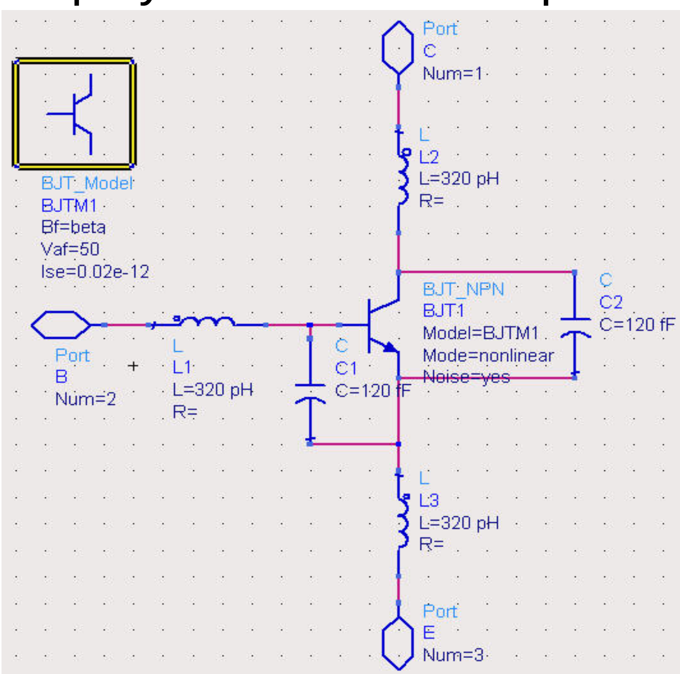
Click here to display an individual value

Use Component Options to clear all the displayed parameters

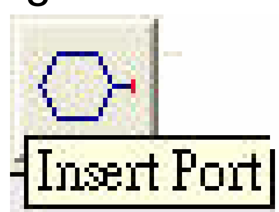
✓Add parasitic and connectors to the circuit

a. Insert lumped L and C components：Insert three lead inductors of **320 pH** each and two junction capacitors of **120fF** each. Besure to use the correct units (pico and femto) or your circuit will not have the correct response.

b. Add some resistance R=0.01 ohms to the base lead inductor and display the desired component values as shown



c. Insert port connectors：Clock the port connector icon (shown here) and **insert the connectors exactly in this order**：1)collector, 2)base, 3)emitter. You must do this so that the connectors have the exact same pin configuration as the ADS BJT symbol.



d. Edit the port names as show here：change **P1 to C** , **P2 to B** and **P3 to E**

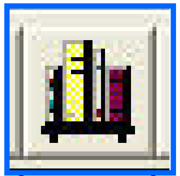
Lab 4：S-parameter Simulations and Optimization

➢ This lab you will learn how to simulate S-parameter

✓Set up the simulation and circuit with ideal components

a. Open a new schematic window and save it with the name：**s_params**

- Click the Component Library icon shown here →



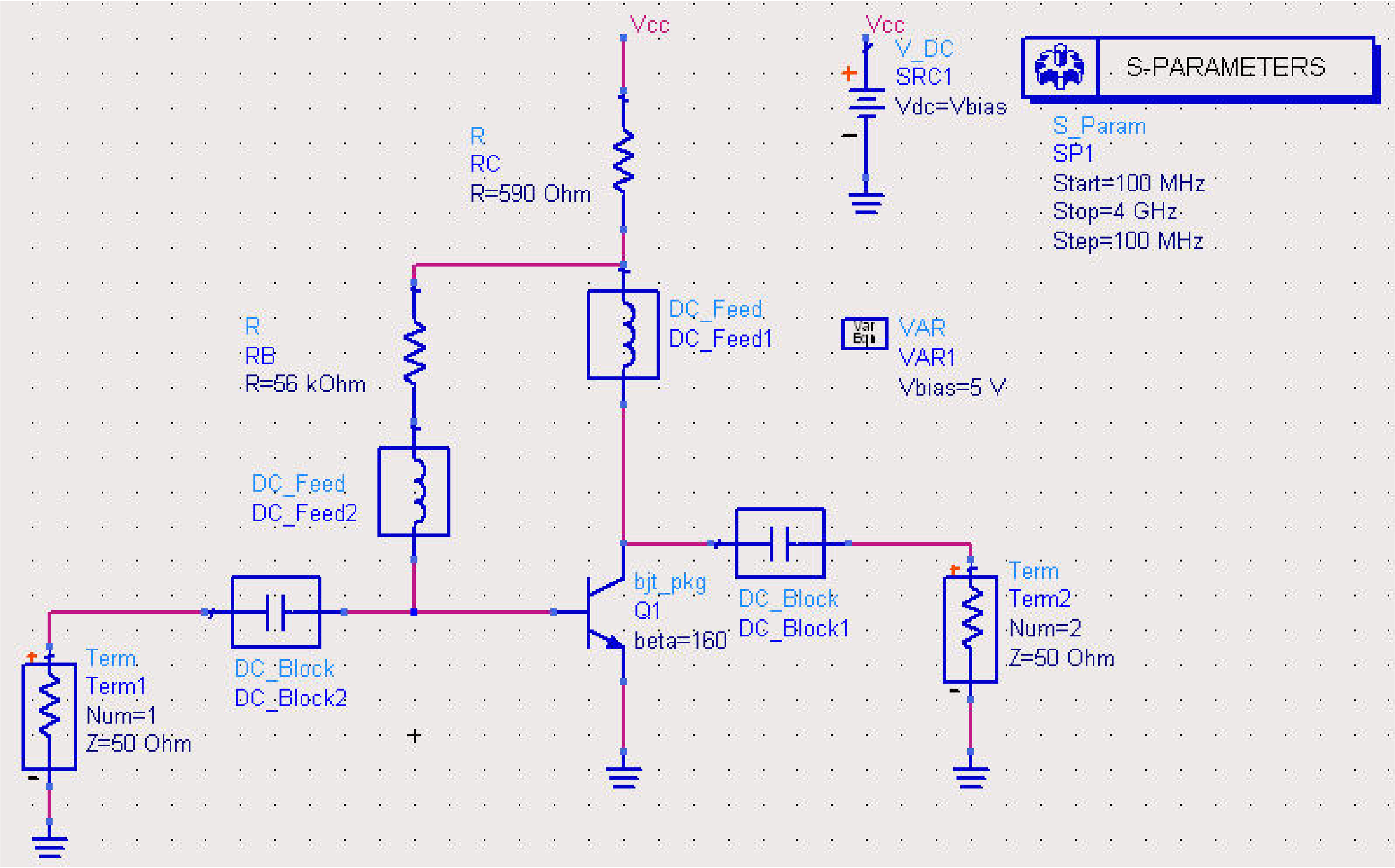
- When the dialog opens , click on the **bjt_pkg** sub-circuit. Insert in into the schematic.
- Insert terminations (**Term**) from the S-parameter palette
- From the Lumped Components palette , insert two ideal inductors：**DC_Feed** to keep the RF out of the DC path
- Insert two ideal **DC_Block** capacitors

b. Insert an **S-Parameter** simulation controller and set：**Start=100 MHz** , **Stop=4 GHz** , and **Step=100MHz**

c. Save the design

Note：your circuit must like the next slide

Lab 4 : S-parameter Simulations and Optimization



Lab 4 : S-parameter Simulations and Optimization

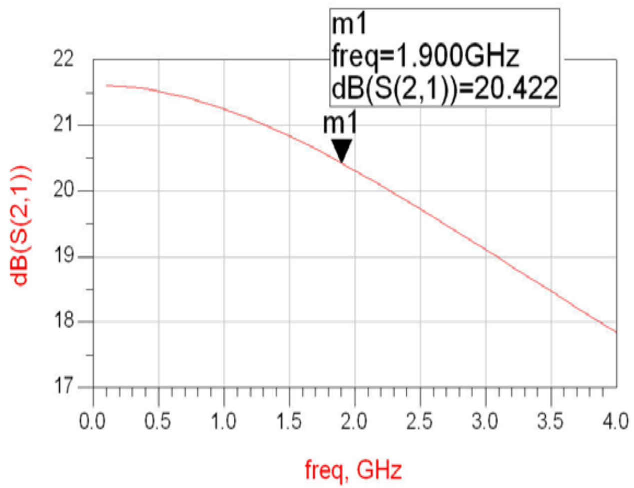
✓Simulate and plot data with marker readout modifications

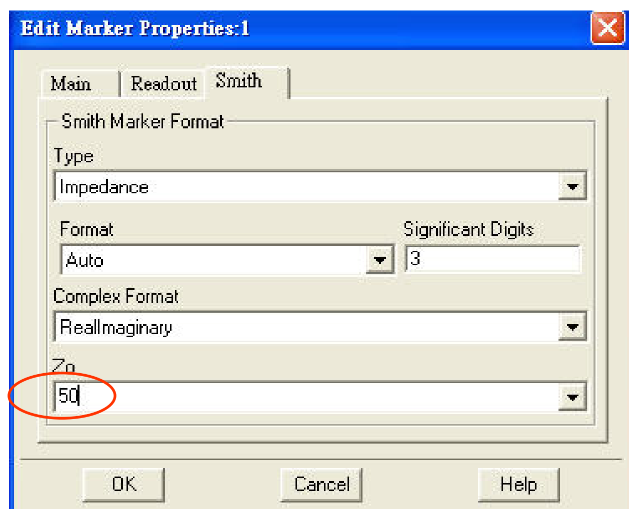
a. Be sure the name of the dataset is ： **s_params** and then **simulate**

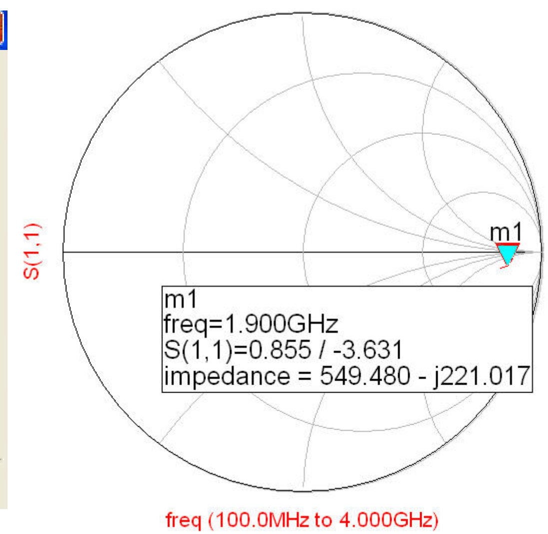
b. When the simulation is finished , insert a rectangular plot of **S21 (dB)** . Insert a marker on 1900 MHz and verify that the gain is about 20 dB

c. Insert a Smith chart of **S11** and place a marker on 1900 MHz. to move the marker , select the readout and use the arrow keys

d. Edit the **marker readout** (double click). Go to the **Smith** tab and change **Zo** to **50** as shown. Click OK and the marker will now read the value in ohms , referenced to 50 ohms





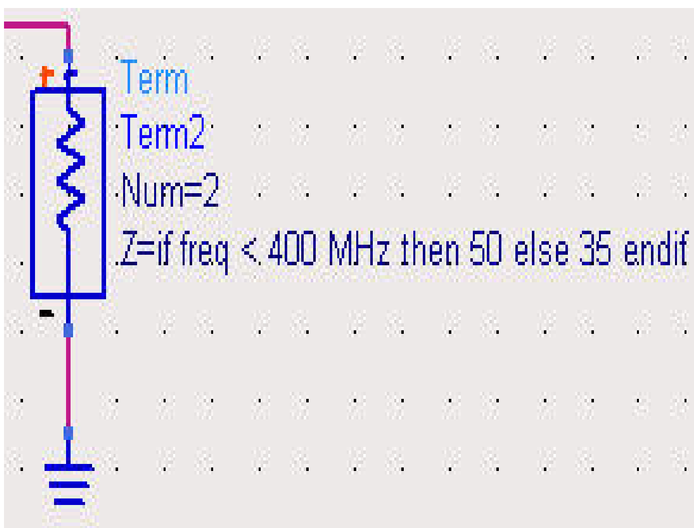


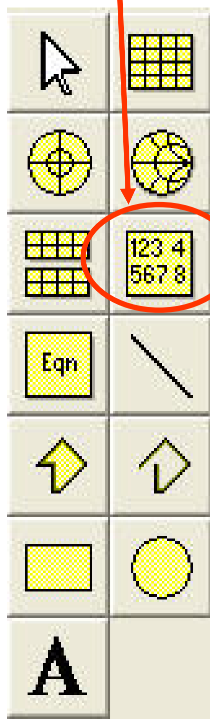
✓Write an equation to vary the Term port impedance.

a. In schematic , **write an equation** for port 2 Term Z to be 35 ohms above 400 MHz ： **Z = if freq < 400 MHz then 50 else 35 endif**.

b. Simulate and then insert a list of **PortZ(2).** Verify the Z is 35 Ohms above 400 MHz

c. Reset the value of port 2 Term to 50 ohms：**Z = 50 Ohm**





freq	PortZ(2)
100.0MHz	50.000 / 0.000
200.0MHz	50.000 / 0.000
300.0MHz	50.000 / 0.000
400.0MHz	35.000 / 0.000
500.0MHz	35.000 / 0.000
600.0MHz	35.000 / 0.000
700.0MHz	35.000 / 0.000
800.0MHz	35.000 / 0.000
900.0MHz	35.000 / 0.000
1.000GHz	35.000 / 0.000
1.100GHz	35.000 / 0.000
1.200GHz	35.000 / 0.000
1.300GHz	35.000 / 0.000
1.400GHz	35.000 / 0.000

- **The transmission and reflection characteristics of the biased circuit show about 20 dB of gain but with a mismatch to 50 ohms at the input. Also , the DC feeds and blocks are ideal and need to be replaced with real values.**

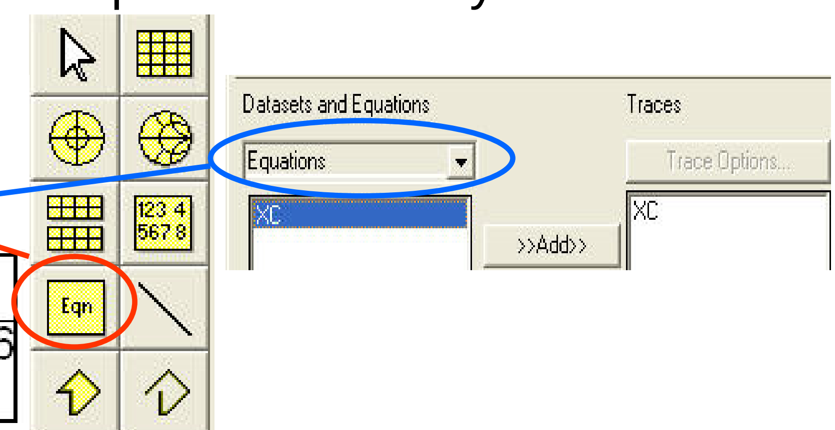
  - ✓ **Calculate L and C values in the data display**

a. In data display , write an equation , **XC** , for the capacitive reactance of **10 pF** at 1900 MHz. Then **list** equation XC as shown here. If desired , title the list using **Plot Options**. With this low reactance , 10pF will be the blocking capacitor values.

b. Change the value of the capacitor in the equation and verify that XC is automatically updated in the list.

Eqn XC= -1 / (2*pi*1900*10e-12)

XC
-8.377E6



c. Create a table for range of inductor values and reactances. **L_val** is a range of swept values form 1 nano to 200 in 10 nano steps. In ADS , the syntax of two colons is a wild card (all values) and can also be used to indicatea range as shown here. The square brackets are used to generate the sweep. After writing the equations and listing them as shown here , scroll through the list. As the inductor value increases , the reactance at 1.9 GHz increases. Therefore , a value of 120 nH should be enough for the DC feed (RF choke)

NOTE：The XL equation will be red (invalid) until L_val is written

```
Eqn XL= 2*pi*1900M*L_val
Eqn L_val = [1n :: 10n :: 200n]
```

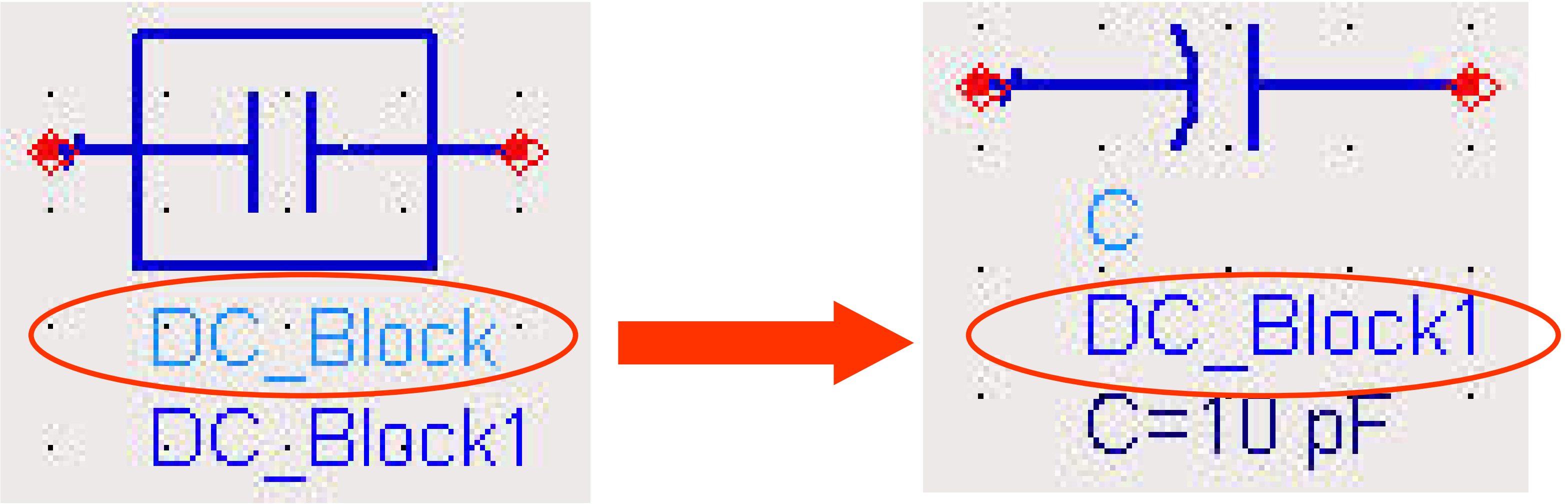
XL	L_val
11.938	1.000E-9
131.319	1.100E-8
250.699	2.100E-8
370.080	3.100E-8
489.460	4.100E-8
608.841	5.100E-8
728.221	6.100E-8
847.602	7.100E-8
966.982	8.100E-8
1086.363	9.100E-8
1205.743	1.010E-7
1325.124	1.110E-7
1444.504	1.210E-7
1563.885	1.310E-7
1683.265	1.410E-7
1802.646	1.510E-7
1922.026	1.610E-7
2041.407	1.710E-7
2160.787	1.810E-7
2280.168	1.910E-7

d. Save the current data display and the schematic

✓Replace L and C with calculated values and simulate

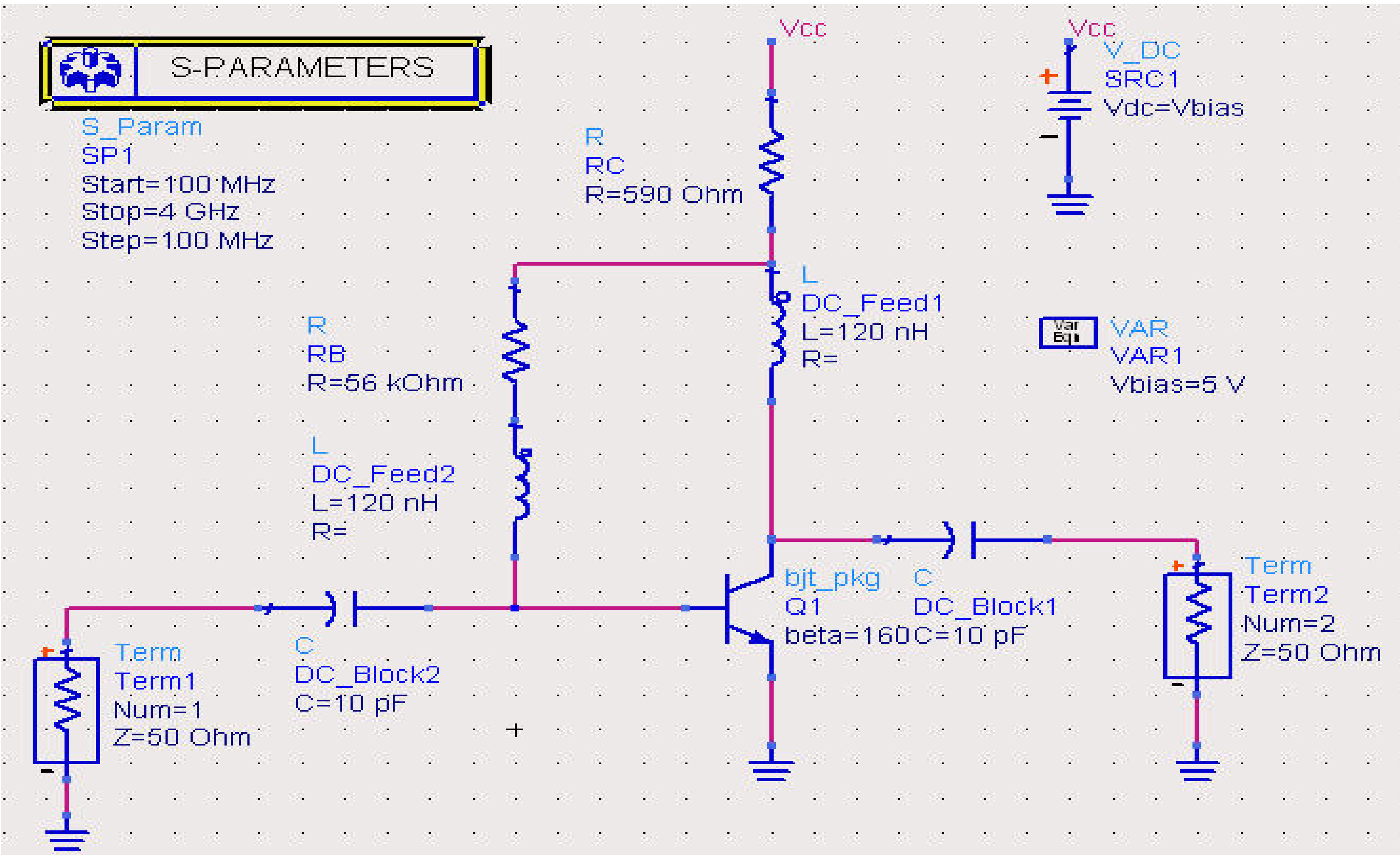
a. Save the schematic with a new name ：**s_match**

b. Change the component name (DC_Block) of both blocking capacitors to **C** and they will automatically become lumped capacitors as shown here. Assign the value for each **C=10pF**

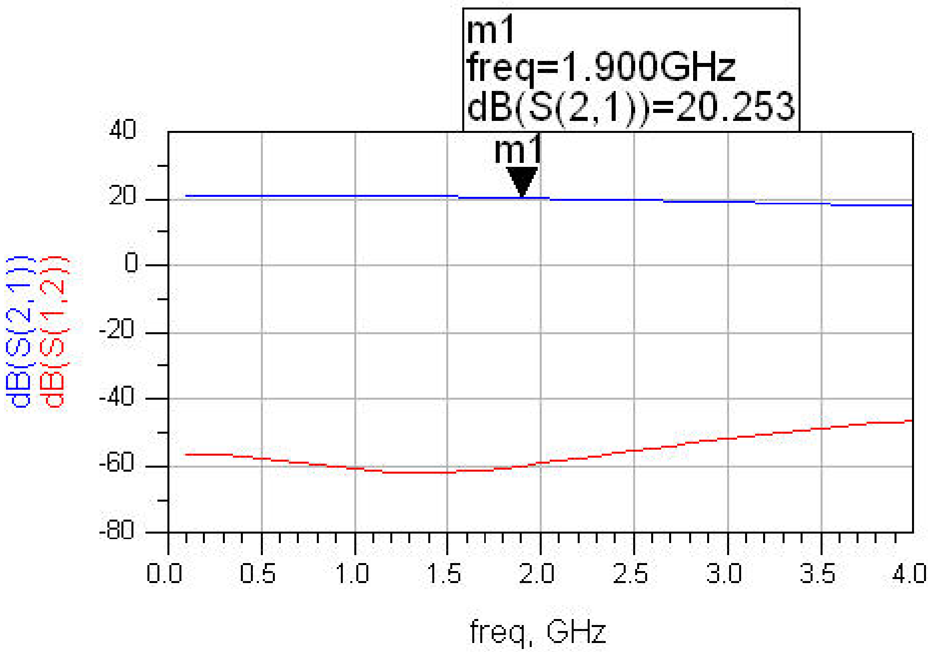


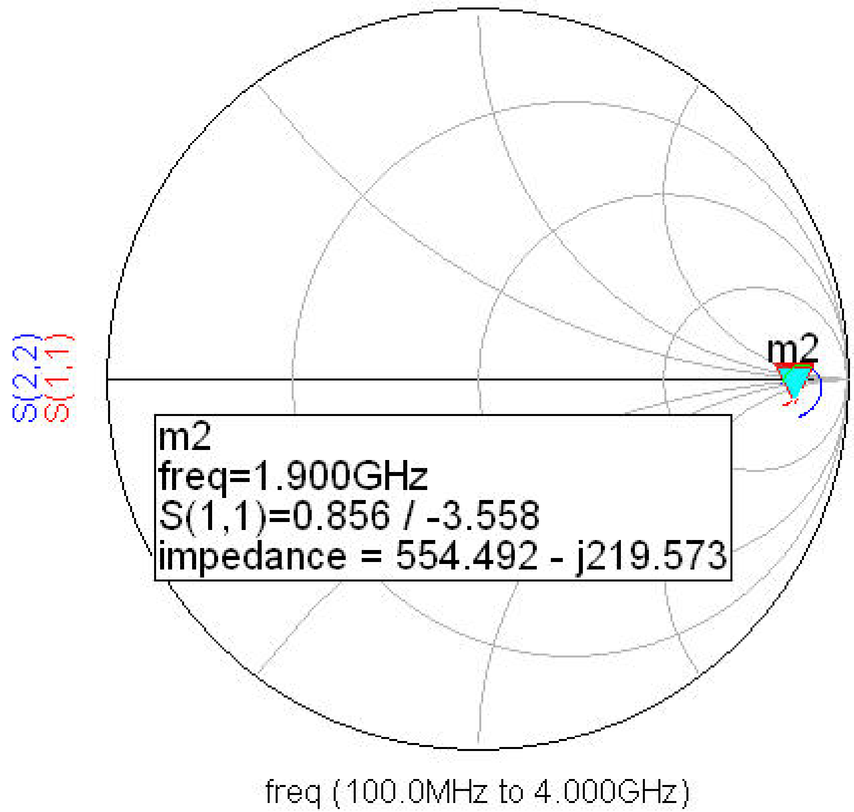
c. Change the **ideal inductors (DC_Feed)** in the same manner and set **L= 120 nH** for each. According to the XL and L_val table , the reactance at 1900 MHz is about 1.5k , which is reasonable at this point in the design.

d. The schematic should now look like the one shown here. Check you values and then **Simulate**



e. In the data display , plot the transmission (S12 and S21) and reflection (S11 and S22) data with markers as shown here. Notice the gain stays relatively flat , step is to create an input matching network.

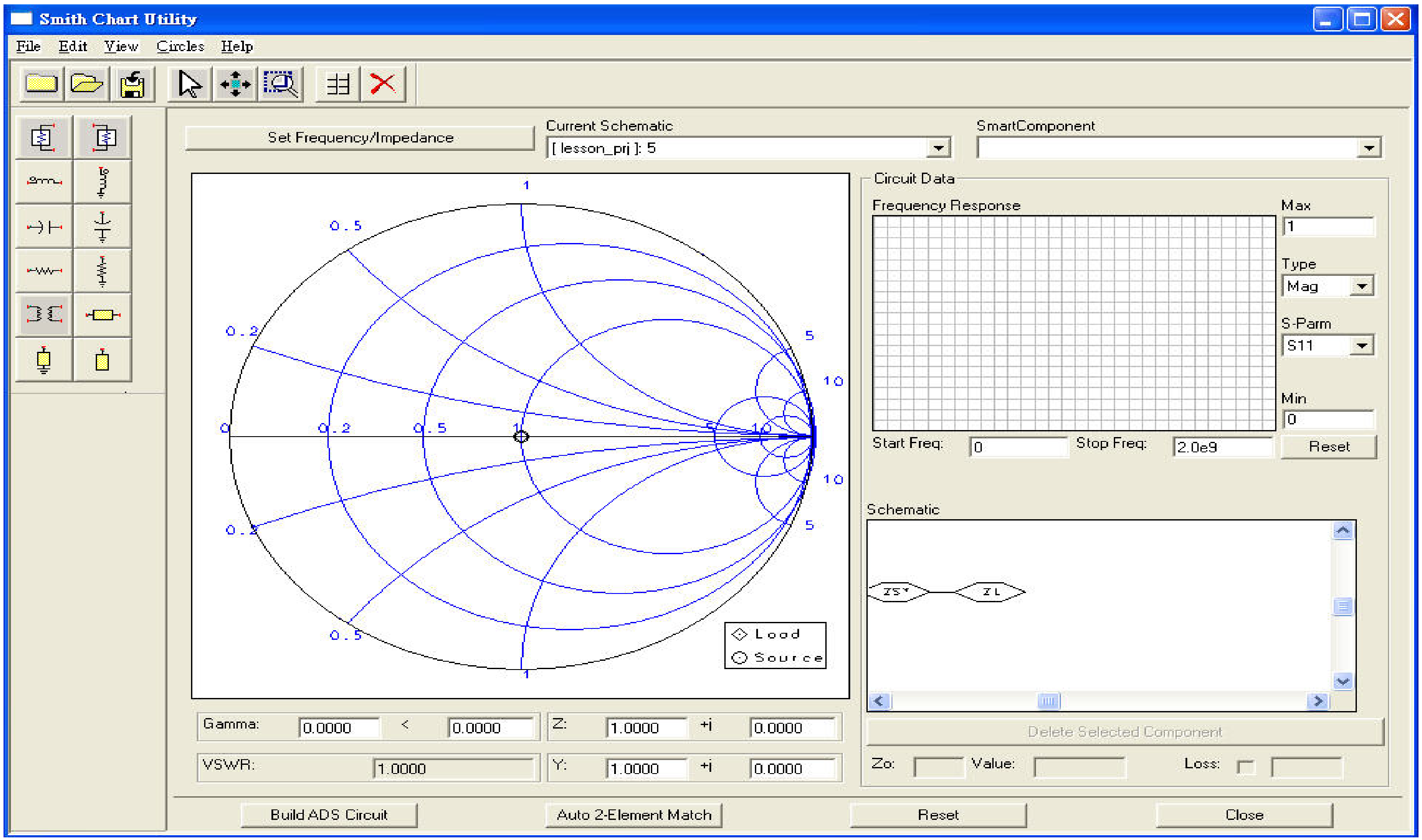




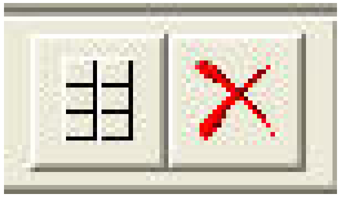
Lab 4 : S-parameter Simulations and Optimization

✓**Use the Smith Chart utility to build a simple matching network.**

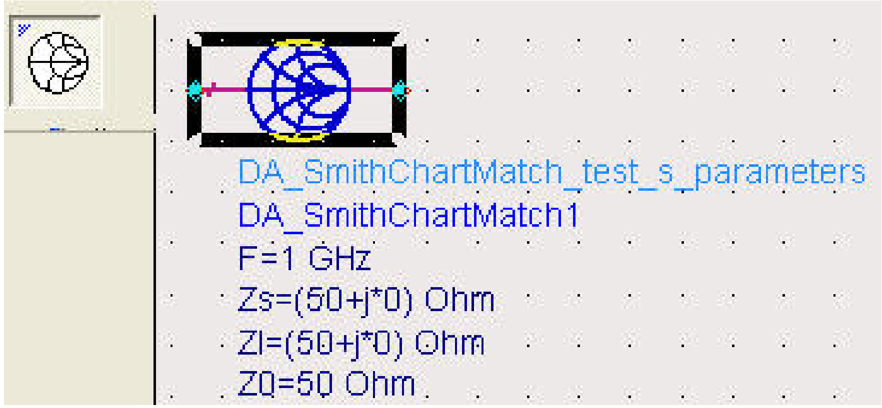
a. In the current schematic , click on the commands : **DesignGuide>Filt**er and then choose the Smith Chart Control Window.



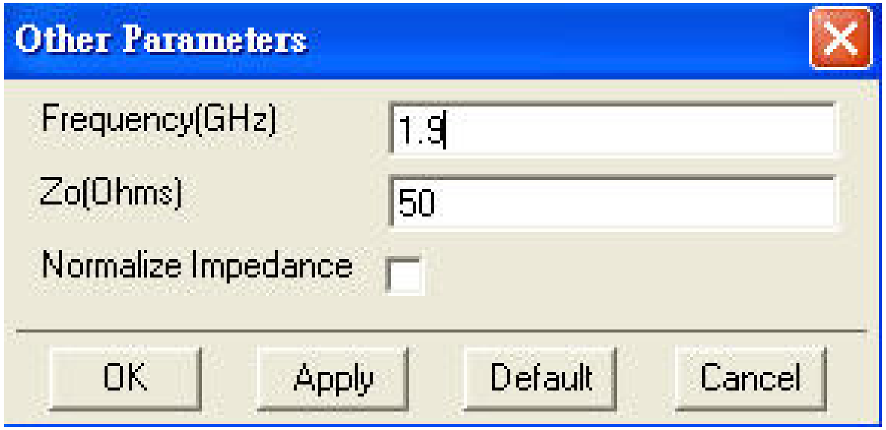
b. Near the top of the Smith Chart control window , click the Palette icon shown here. This will bring up the Smith Chart palette on your schematic.



c. Insert the Smith Chart Matching Network component (also know as a Smart Component) onto the schematic near the input of the amplifier – no need to connect it – but it is required . Also , click OK when a message dialog appears.



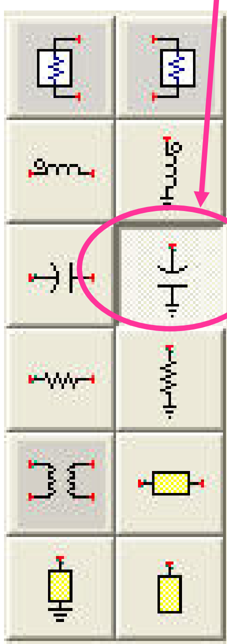
d. Go back to the Smith Chart control window and click the **Set Frequency / Impedance** button. Then set the frequency to 1.9 GHz and turn off the normalization. Click Ok

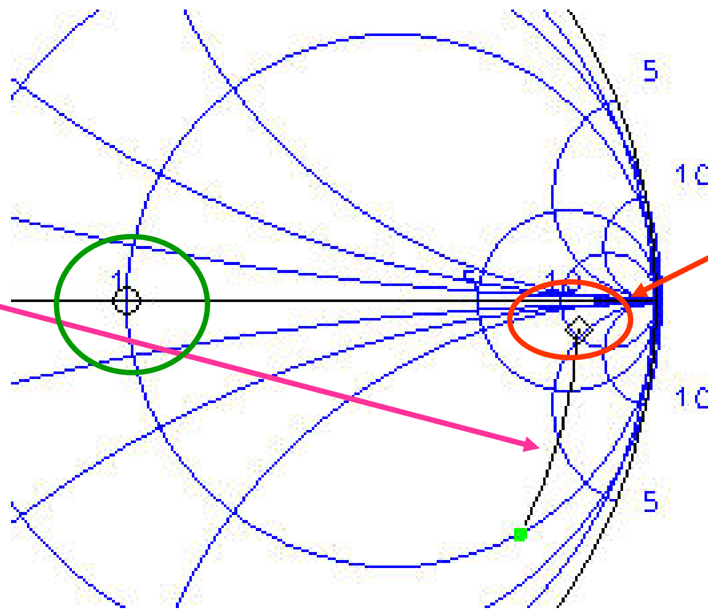


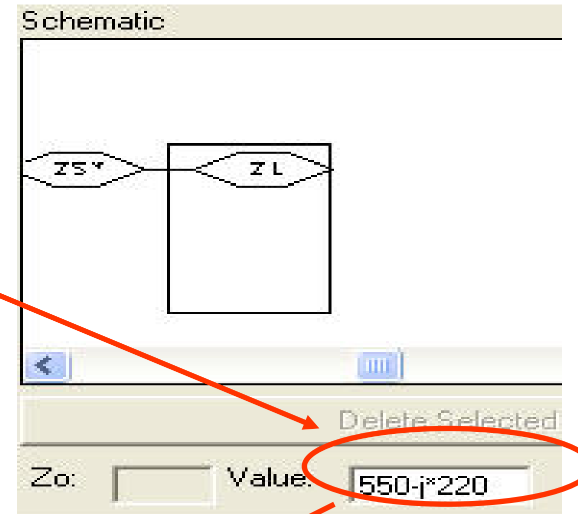
Lab 4 : S-parameter Simulations and Optimization

e. IN the lower right corner of the window , select the ZL component and type in the impedance looking into the amplifier from the last simulation (S11≈550-j*220)

f. Notice that the load symbol on the smith chart has relocated as shown here. Next , select the **shunt capacitor** form the palette and move the cursor on the Smith chart. When you get to the 50Ohm circle of constant resistance , click to stop , as shown here

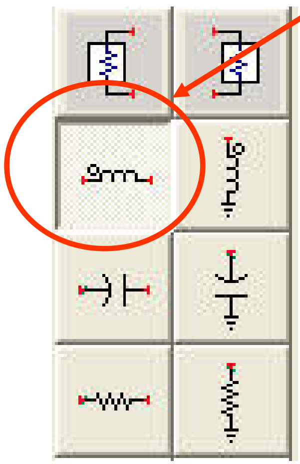


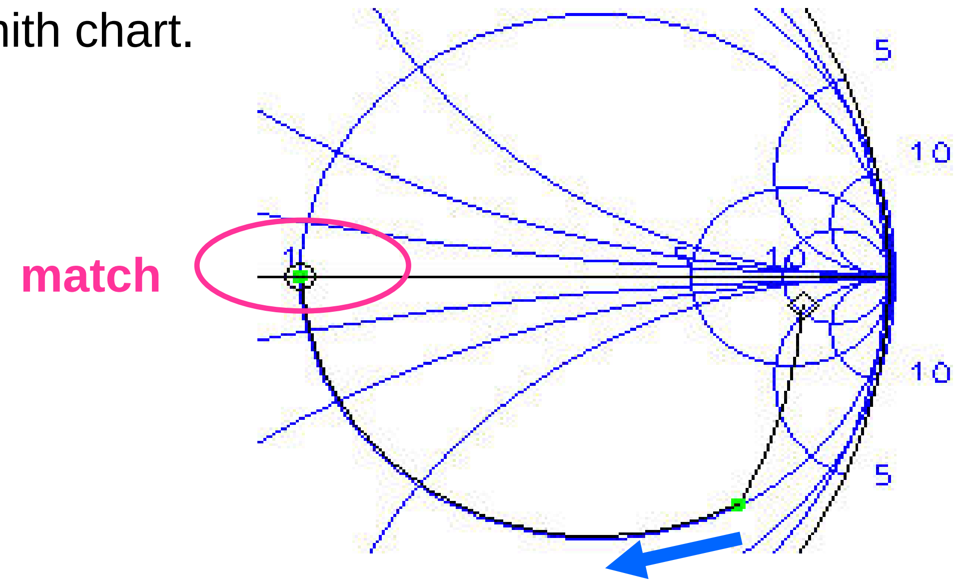




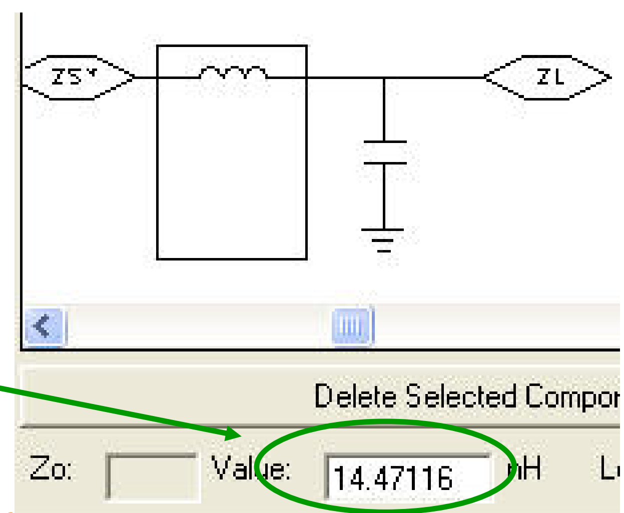
Load symbol is at 550-j220. Source symbol is at 50+j0

g. Next , select the series inductor and move the cursor along the circle until you reach the center of the Smith chart.





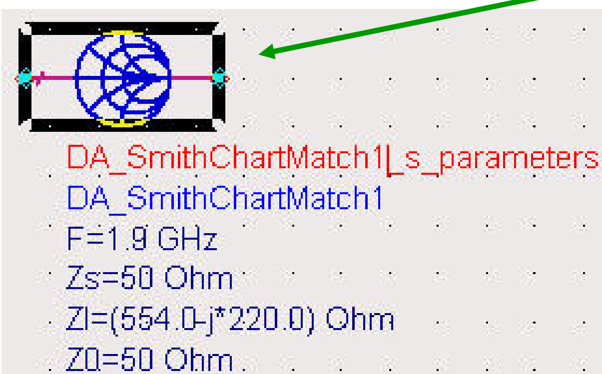
h. Move the cursor into the lower right corner of the window and click on each of the components in the Schematic as shown here. You will see the value for the inductor and capacitor : approximately L≈14 nH and C ≈ 400 fF or 0.4pF.

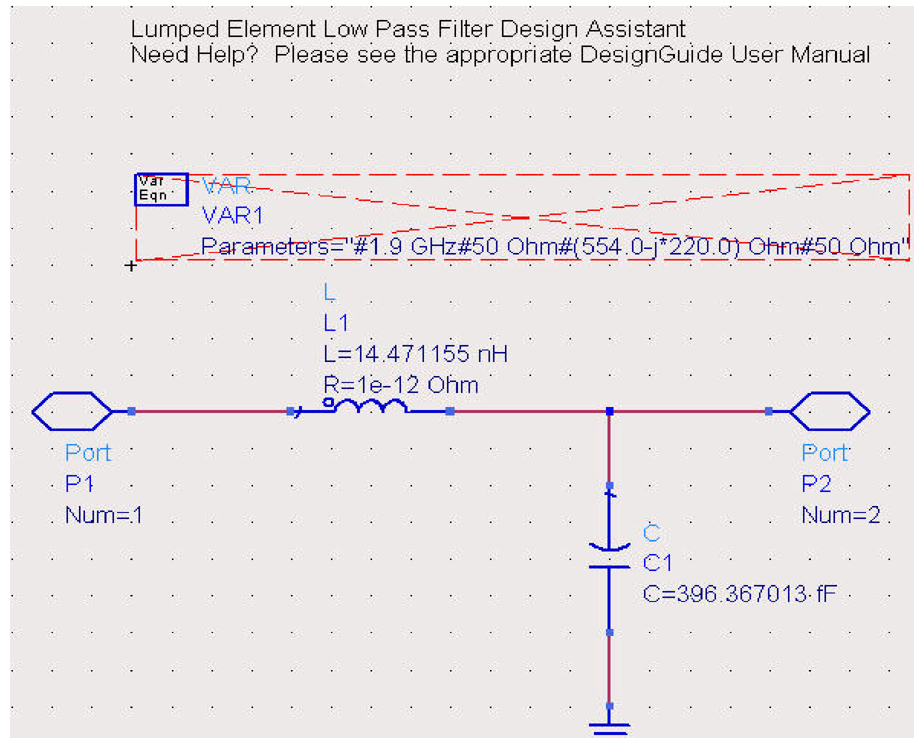


Lab 4 : S-parameter Simulations and Optimization

j. To have the DesignGuide build the circuit , click the button on the bottom of the window : **Build ADS Circuit** . Click OK to any messages that appear.

k. On the schematic , push into the Smith Chart component and you should see the network as shown below :

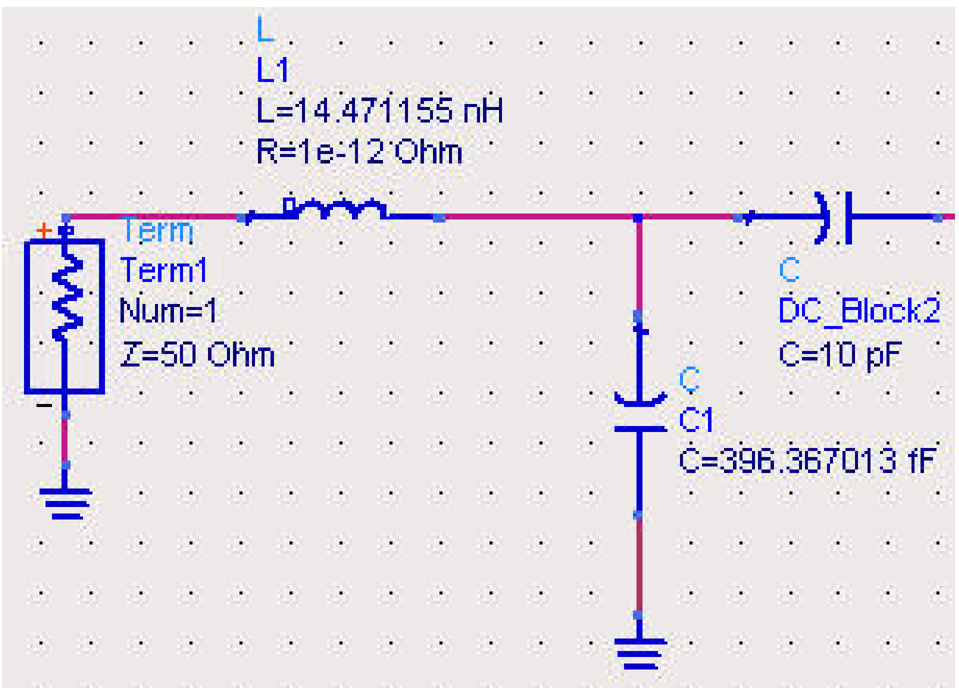




Lab 4 : S-parameter Simulations and Optimization

✓**Now it is time to use the matching network with the amplifier. You could use the component by connecting it to the amplifier input. However , because you will be using the optimizer , it is better to have the L-C components on the schematic.**

l. Either copy / paste the L-C and ground onto the amplifier or simply insert and L and C on your schematic. Then set the values to **L=14.3 nH** and **C=0.4PF**. The amplifier input should now be as shown here.

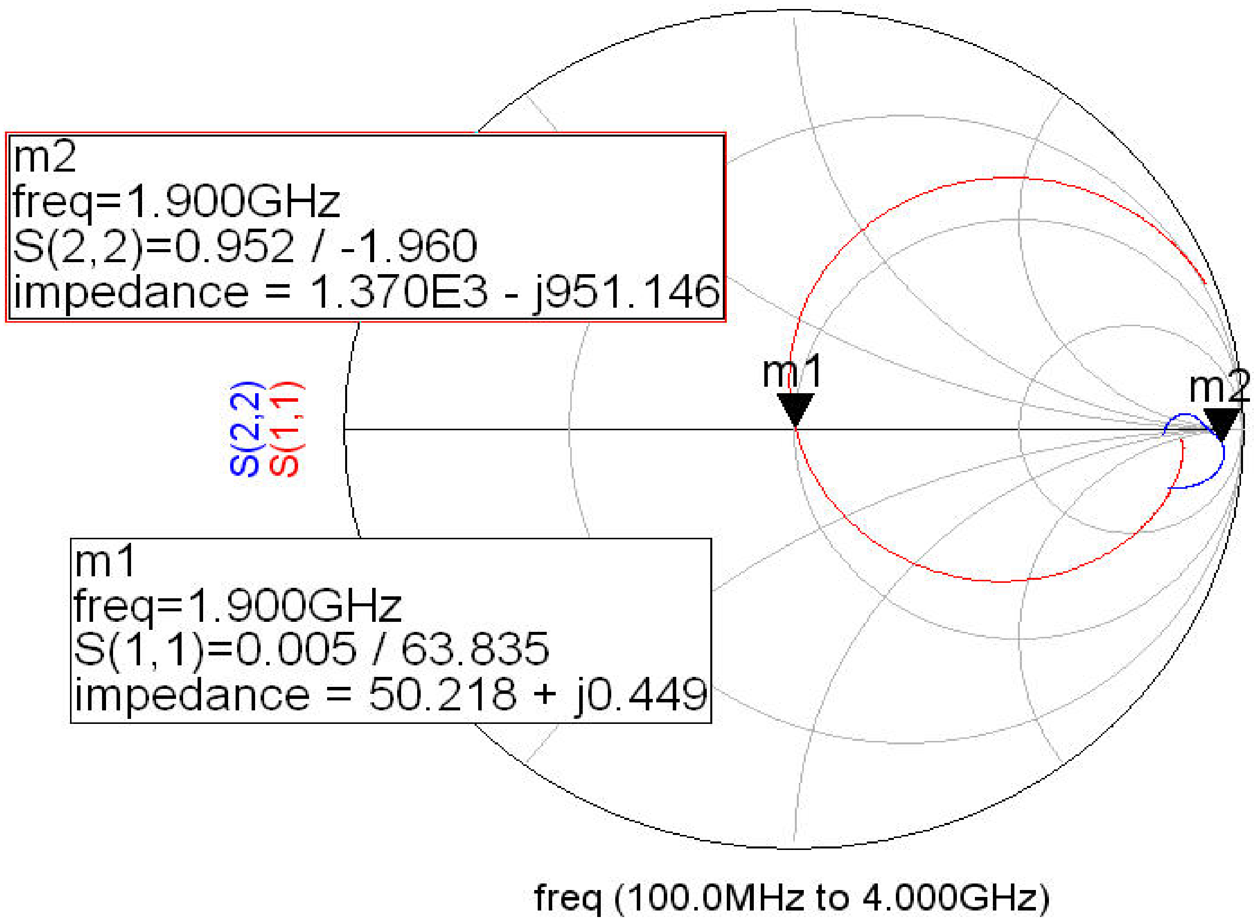


m. Delete the Smith Chart component from the schematic and **close** the Smith Chart utility window

n. Simulate with the input network

p. Set the simulation step size to **10 MHz** , and **simulate**

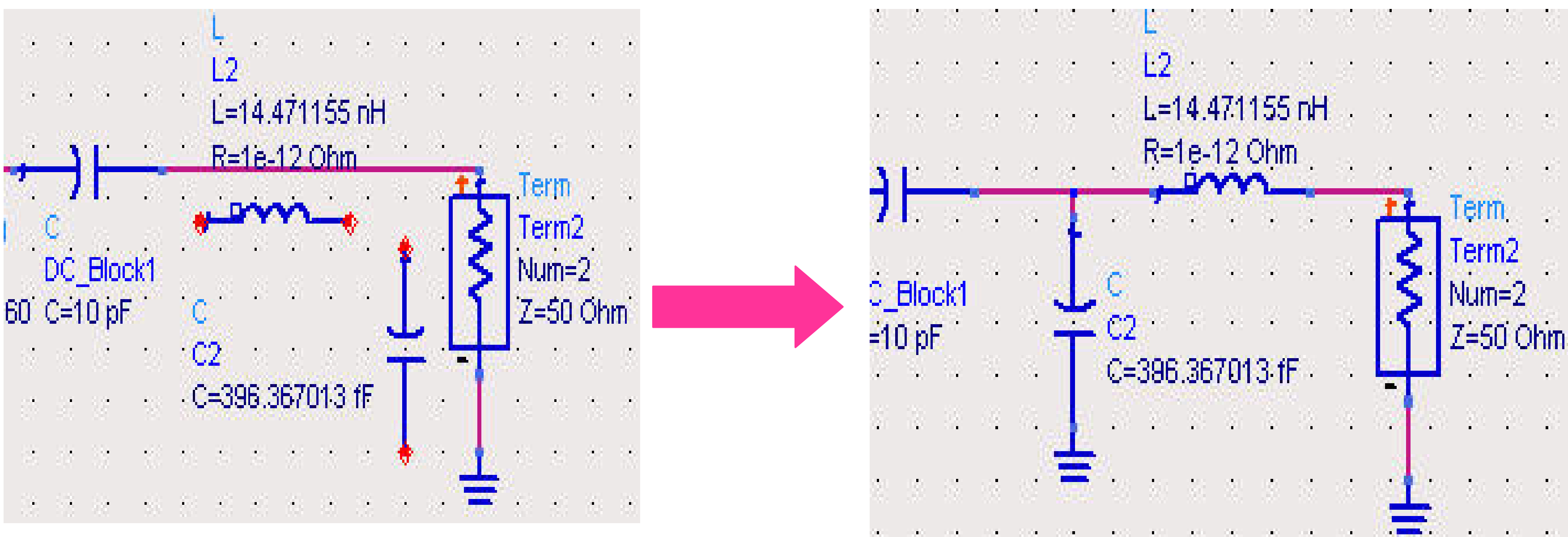
q. When the simulation is complete , add S-22 to the Smith chart. Place a marker on S-22 at 1.9 GHz. The result show S-11 is good but S-22 is not match as shown here



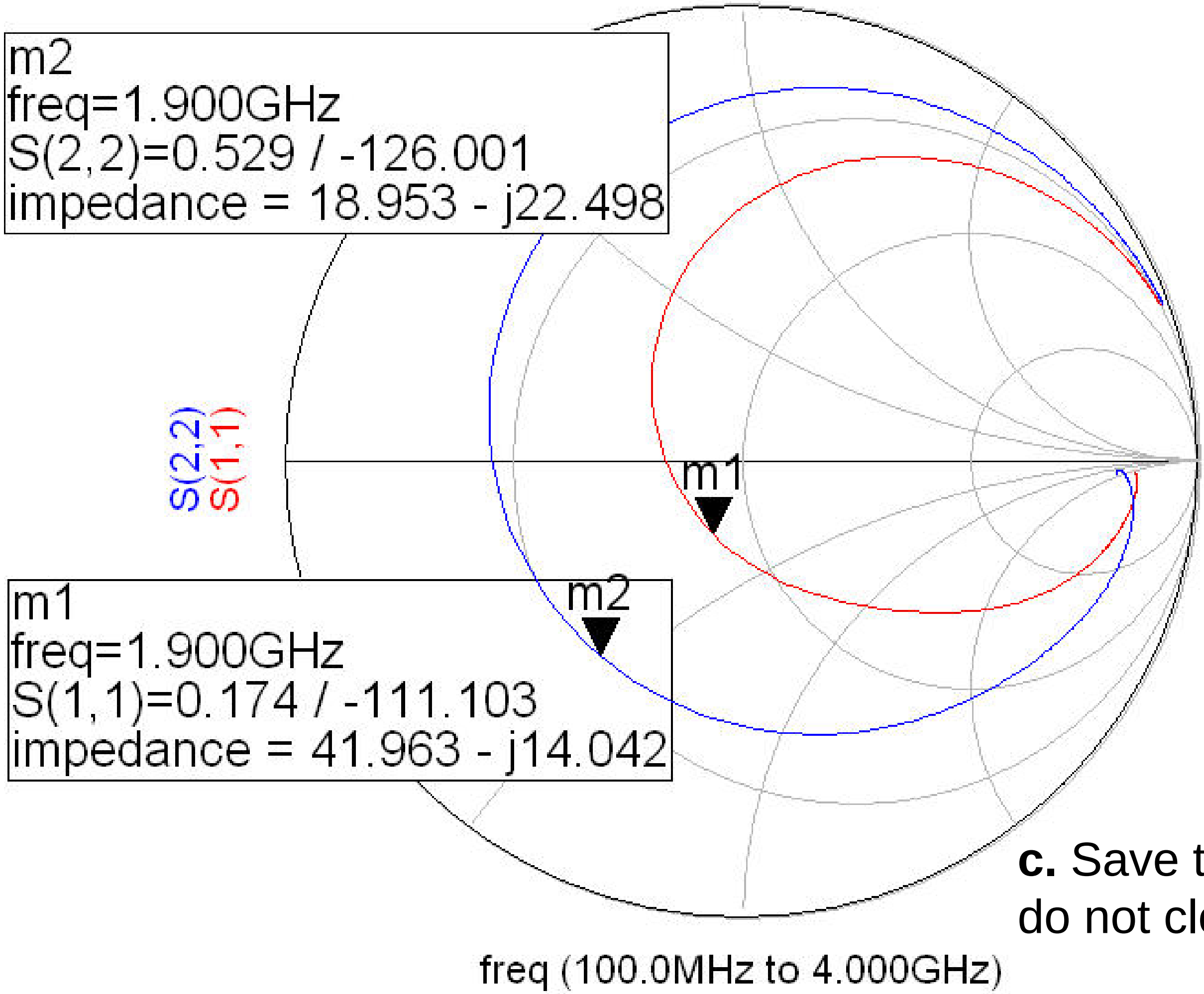
➢**Because S-22 is similar to the unmatched impedance of the input , it is reasonable to put a similar topology on the output and simulate the response.**

✓**Add output matching components.**

a. Select the input L-C network and use the **copy icon** (shown here) to make a copy to the components. Then place them near the output , with a ground on the capacitor. Delete the wires and insert them as shown at the output.



b. Simulate and check the response. Your data should be similar to the results shown here where S22 is now closer to 50 ohms. However , S11 has shifted , as you should expect. Set the S-22 marker readout edited to Smith Zo=50



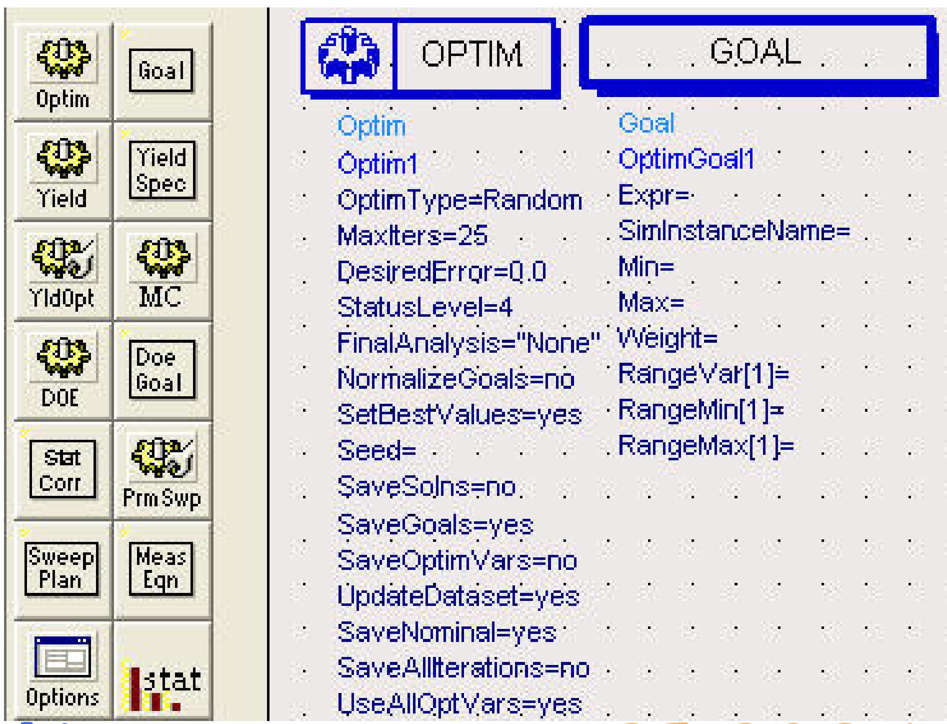
c. Save the design but do not close the window

✓Set up an Optimization controller and Goals.

a. Save the s_match schematic design with a new name：**s_opt**

b. Go to the **Optim / Stat / Yield** palette and insert an **optimization controller** and one **goal** as shown here

c. Edit the goal by double clicking. In the dialog box , type in the following settings and clock **Apply** after each one and OK when done



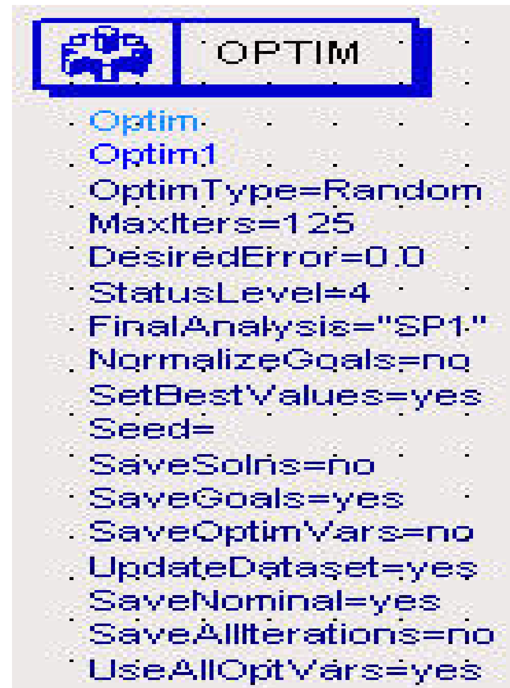
- **Expr：dB (S(1,1))**
- **SimInstanceName：SP1**
- **Max= -10 (S11 must be at least -10 dB to achieve the goal)**
- **RangeVar=freq Range Min=1850 MHz RangeMax：1950 MHz**

Lab 4 : S-parameter Simulations and Optimization

d. Copy the S11 goal – select it and use the copy icon

e. On screen , change the goal expression to **“dB(S(2,2))”** as shown here. Now , you have two goals for the input and output match.

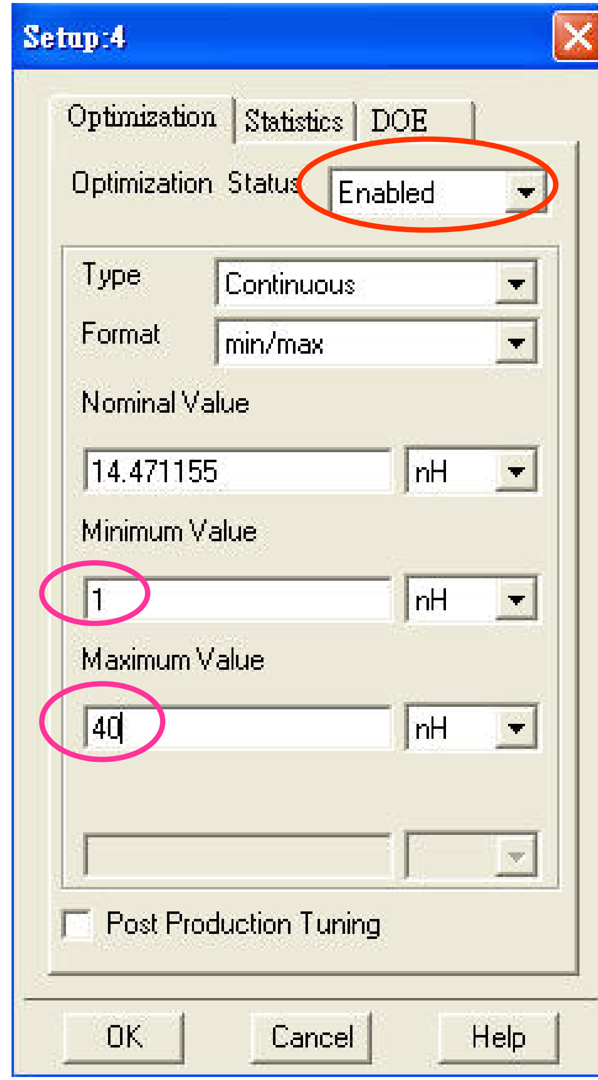
f. Set up the OPTIM controller. For this lab exercise , most of the default settings can remain , controller and set the **MaxIter = 125** and set **the FinalAnalysis = “SP1”**. These settings mean that the optimizer will run for up to 125 iterations to achieve the goals. The Normalize goals setting means that all goals will have equal weighting. Also , a final analysis is automatically run with the last values so that you can plot the results without running another simulation.



Lab 4 : S-parameter Simulations and Optimization

✓ **Enable the components to be optimized**

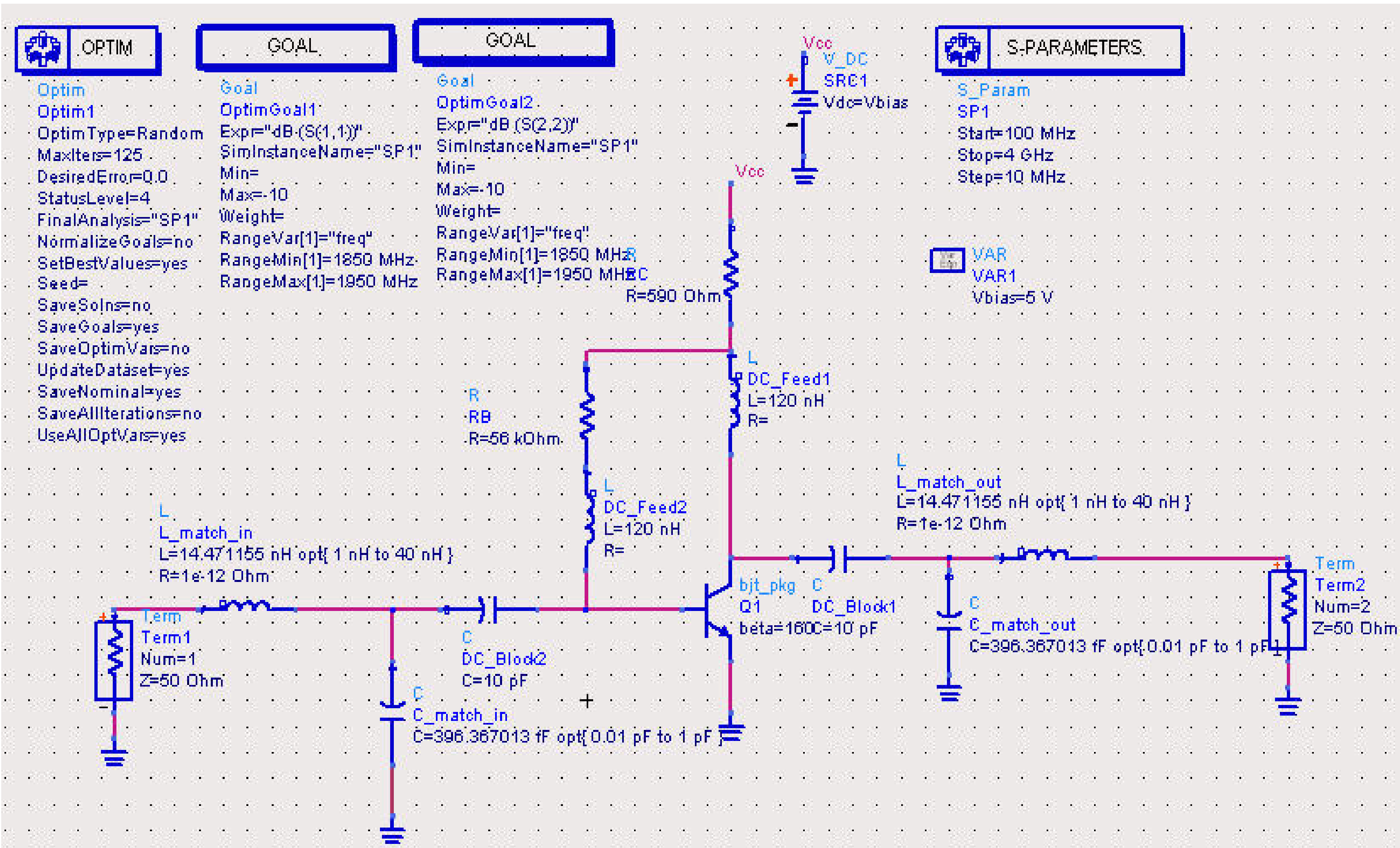
a. Edit (double click) the inductor **L_match_in**. When the dialog appears , click the **Optimization/Statistics/DOE Setup** button. In the **Optimization tab** , set the inductor to be **Enabled** as shown and type in the continuous range from **1 nH** to **40 nH** as shown here. Click **OK** and the component text will show the opt function and range.



b. Go ahead and Enable the other three matching components as shown. Edit each one using the dialog box or you can type directly on-screen using the opt function and curly braces for the range. Also , use the F5 key to move component text as need :

Lab 4 : S-parameter Simulations and Optimization

c. Check the circuit as shown here and then **Simulate** and watch the status window.



Lab 4 : S-parameter Simulations and Optimization

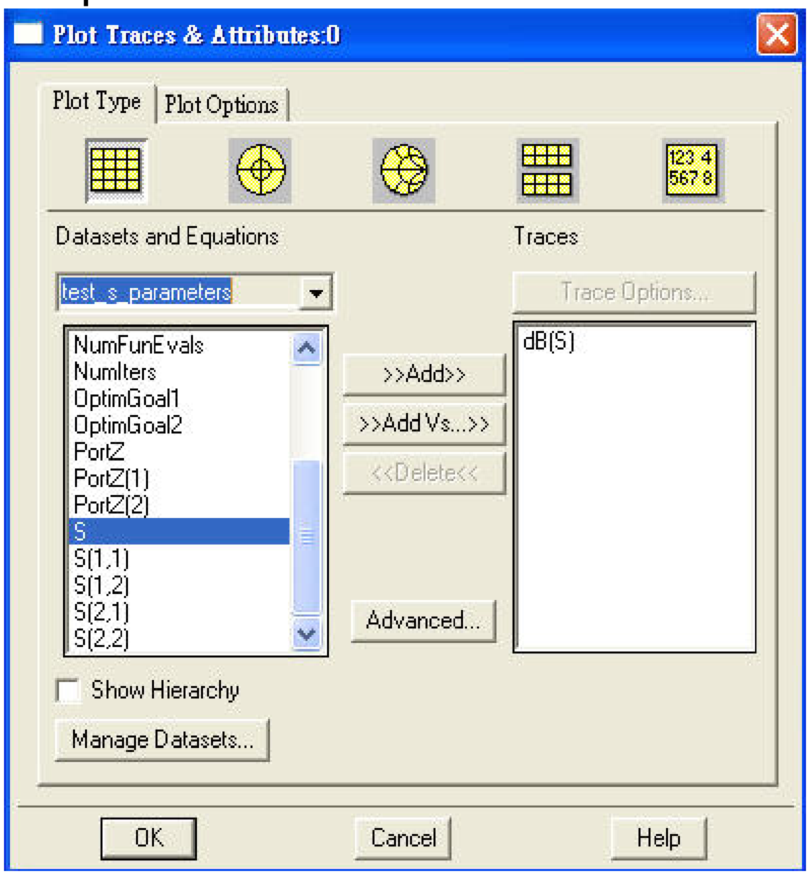
d. The status window reports progress. If the goals are met , the EF (error function) = 0. A successful iteration occurs if the **EF moves closer to zero**. With EF = 0 (or close in some cases) , the next step is to update component values and plot the results. If your EF is not zero , check the schematic and try it again.

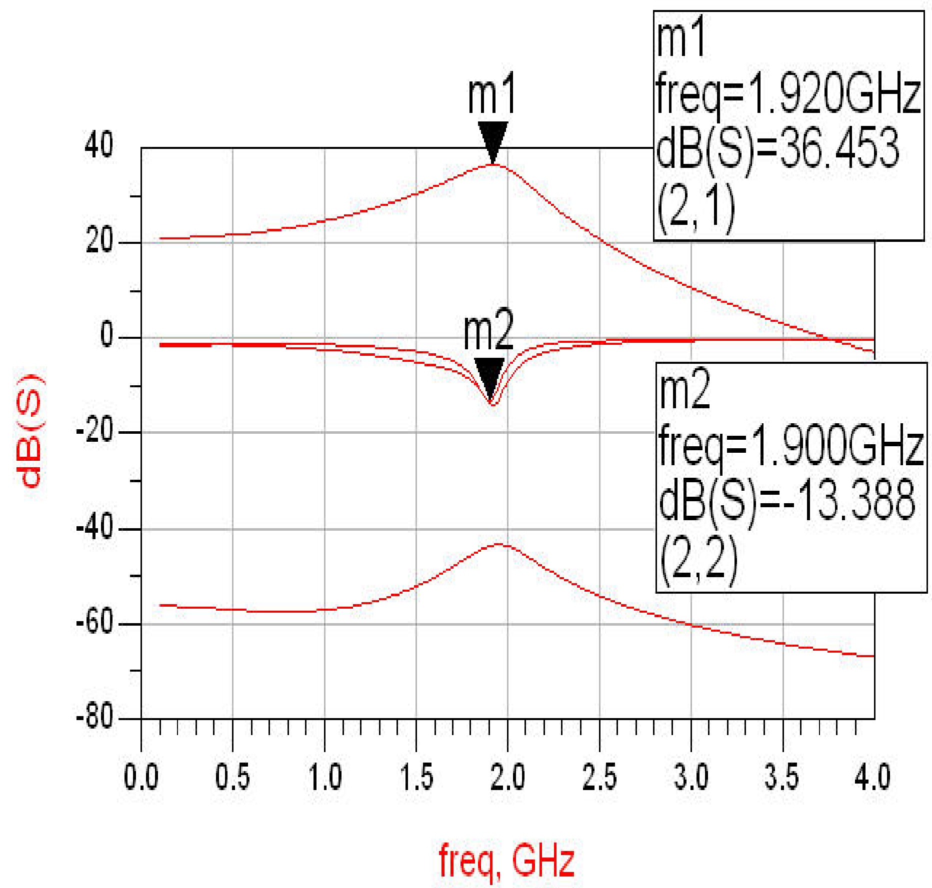
Status / Summary

```
Iteration/Trial #65:
  CurrentEF: 0
  Optimization variables:
    L_match_in.L = 16.4709e-09
    C_match_in.C = 359.506e-15
    L_match_out.L = 22.3167e-09
    C_match_out.C = 274.762e-15

SP Optim1[1].FinalAnalysis1[1].SP1[1] <(GEMX netlist)>    :
.........................................................

Resource usage:
    Total stopwatch time: 9.73 seconds.

----------------------------------------------------------
Simulation finished: dataset `test_s_parameters' written :
  `C:\users\default\lesson_prj/data'
```

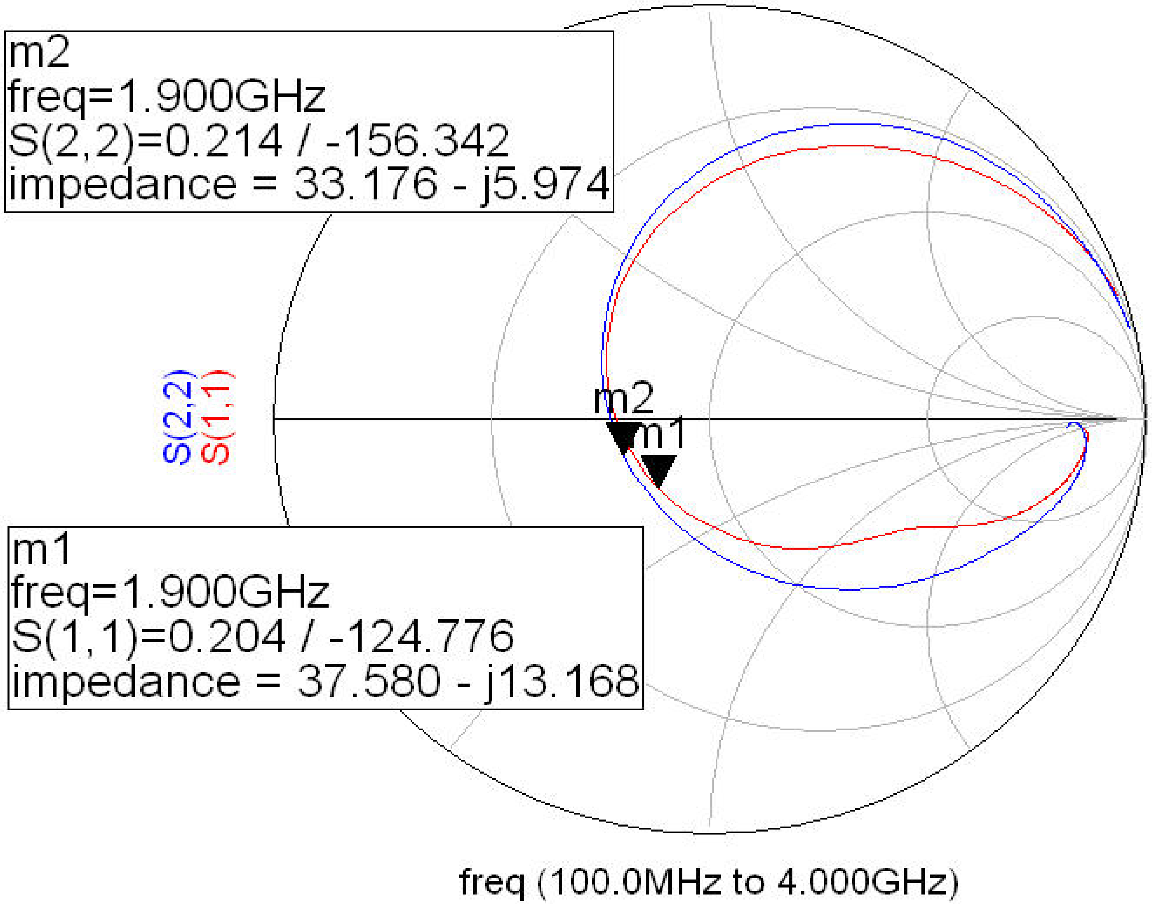
✓ Plot the results

a. In the data display , insert a rectangular plot. Then , as shown here , add the complete S matrix in dB to see all four S parameters. This way you can quickly verify the results. Your values may differ but the goals from the optimization should be met





b. Plot the impedance **S11** and **S22** on a Smith chart. Change the marker readout Smith tab to Z0=50. As you will see , the impedance is not close enough to 50 ohms. But first , you will update the schematic with the values from the optimization.



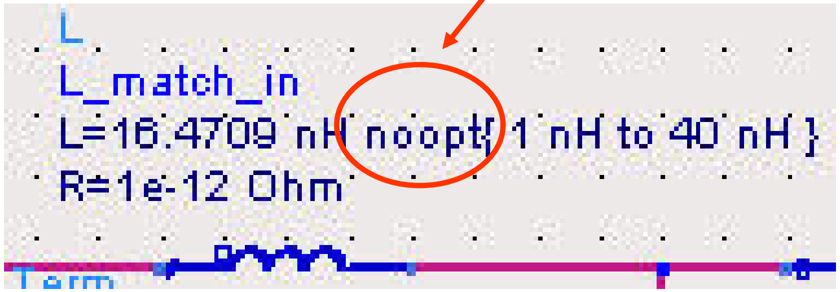
Lab 4 : S-parameter Simulations and Optimization

✓ **Update optimized values and disable the opt fucntion.**

a. Click the command：**Simulate > Update Optimization Values**. The enabled components should now have the final values as the nominal values. For example , the input inductor may look like the one shown here – your values may vary a little because of the random mode and no seeding.

b. Disable a component. Edit **L-match_in** inductor. Then click the **Optimization/Statistics/DOE** Setup button. Select Disabled as shown here nad click OK. Notice that the component function changes from opt to noopt. This means the component will not be used in an optimization. You can also disable a component by inserting the cursor on – screen and typing no infront of opt function to make it noopt – try it.

c. Save the s_opt schematic. In the next set of steps , you will set up a final matched circuit.



Lab 4 : S-parameter Simulations and Optimization

✓ **a.** Save the s_opt schematic as ：**s_final**.

✓ **b. Delete** the optimization controller and goals.

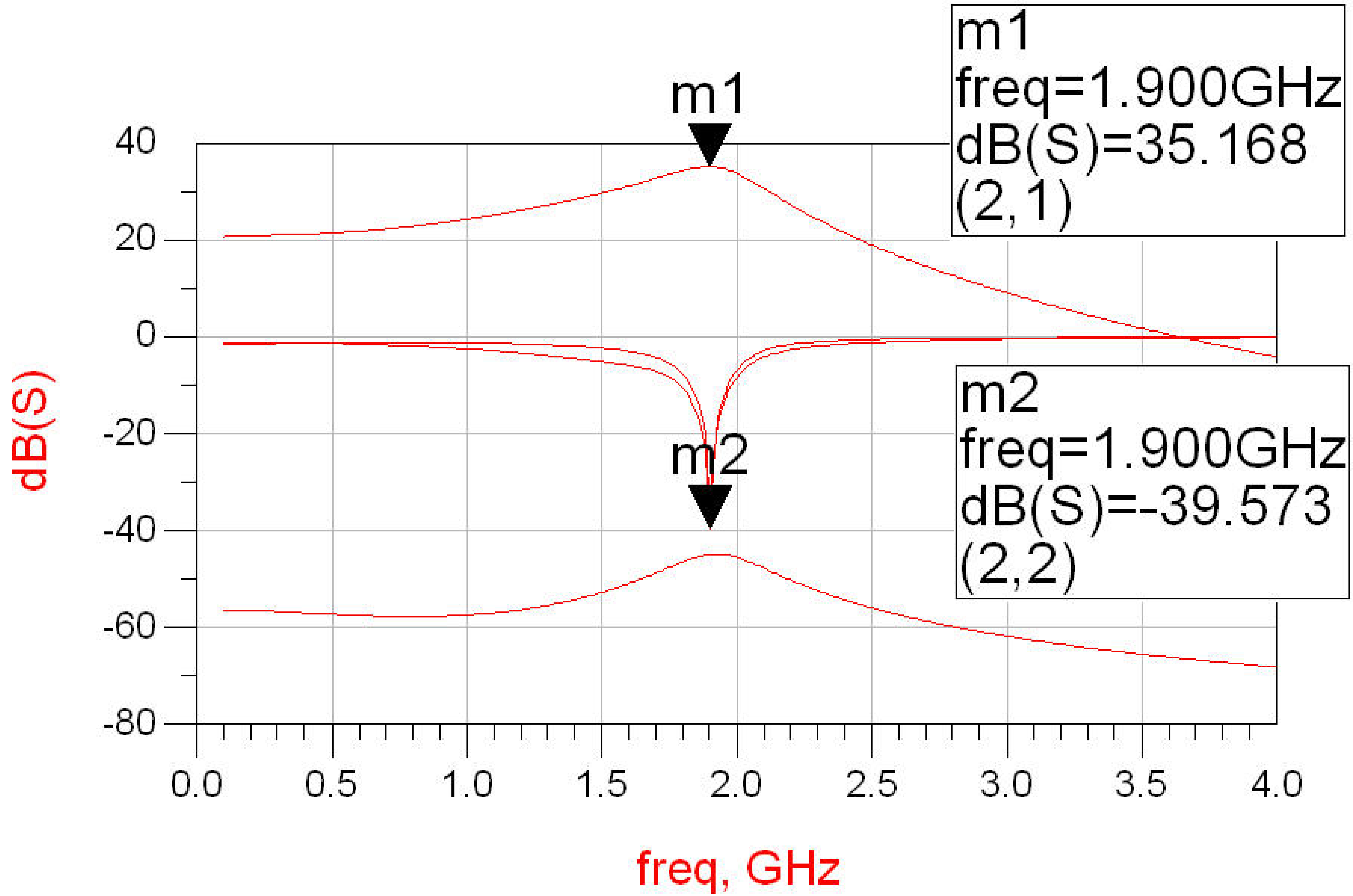
✓ **c.** modify the four L and C matching component values , adding resistance to the inductors as shown here. This will result in a good match and will be used for the remainder of the lab exercises. Go ahead and change the values the values by typing directly on –screen as shown here ：

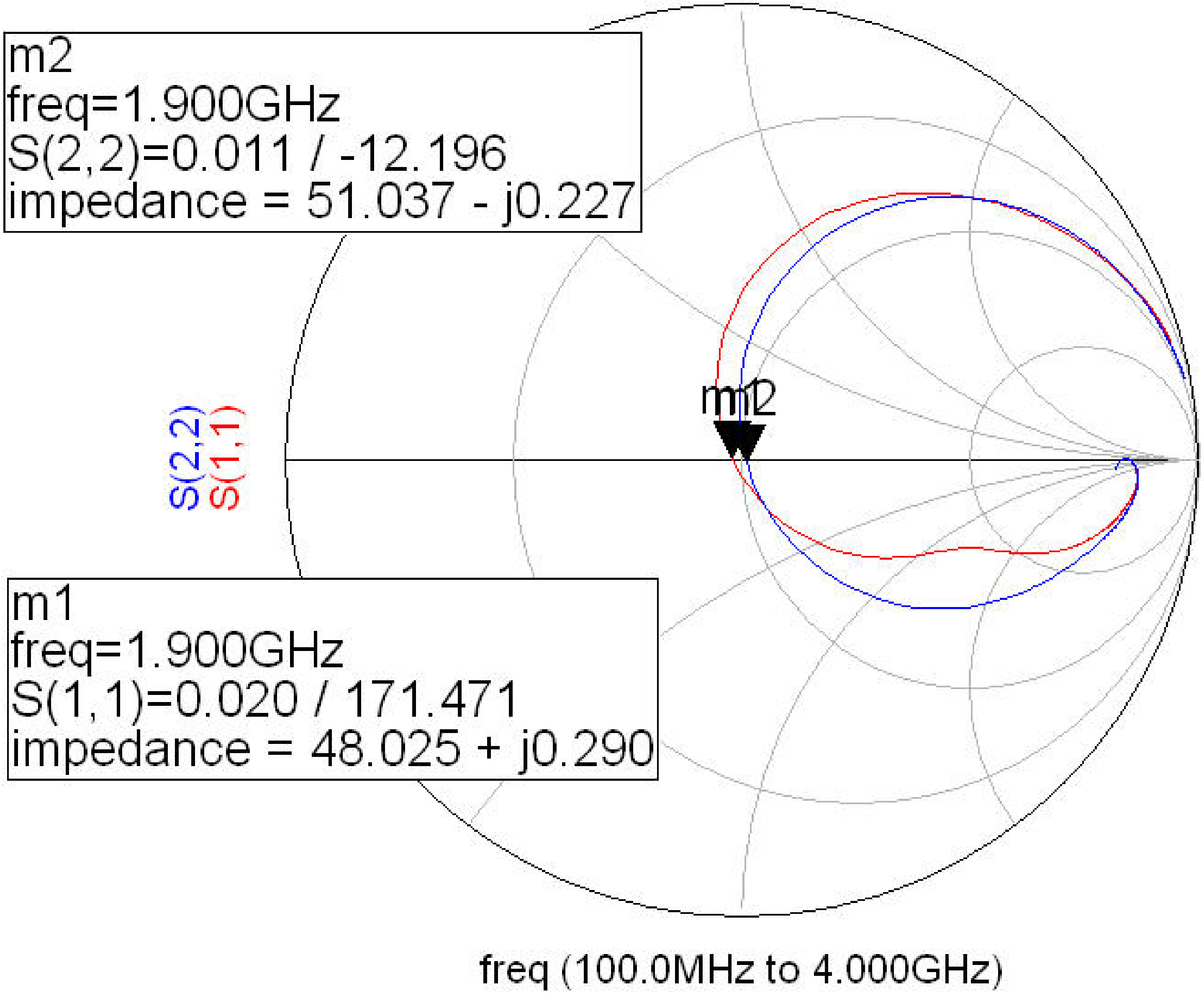
L_match_in = 18.3 nH & R = 12 Ohm **L_match_out = 27.1 nH & R = 6 Ohm**

C_match_in = 0.35pF **C_match_out = 0.22 pF**

✓ **d.** With the new final component values and **Simulate**

✓ **e.** When the data display opens , plot the entrie Smatrix by selecting S in the dataset. Also plot the S11 and S22 on the Smith chart to verify the match is close to 50 ohms at 1900 MHz. With these results.


m1
freq=1.900GHz
dB(S)=35.168
(2,1)
m2
freq=1.900GHz
dB(S)=-39.573
(2,2)
m1
m2
40
20
0
-20
-40
-60
-80
dB(S)
0.0
0.5
1.0
1.5
2.0
2.5
3.0
3.5
4.0
freq, GHz


m2
freq=1.900GHz
S(2,2)=0.011 / -12.196
impedance = 51.037 - j0.227
m1
freq=1.900GHz
S(1,1)=0.020 / 171.471
impedance = 48.025 + j0.290
m12
S(2,2)
S(1,1)
freq (100.0MHz to 4.000GHz)