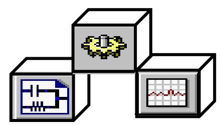
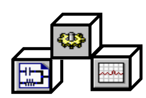
Topic 1:

ADS and Circuit Simulation Fundamentals



Here is ADS Simplified: 3 steps



STEP 1: design capture

Insert circuit & system components and set up the simulation.

STEP 2: Simulate

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.

Layout / Momentum.

User Variables, Licenses, Directories

Just in case you need to know:

**$HOME (UNIX variable) or %HOME% (PC variable)
is usually where you run and use ADS.**

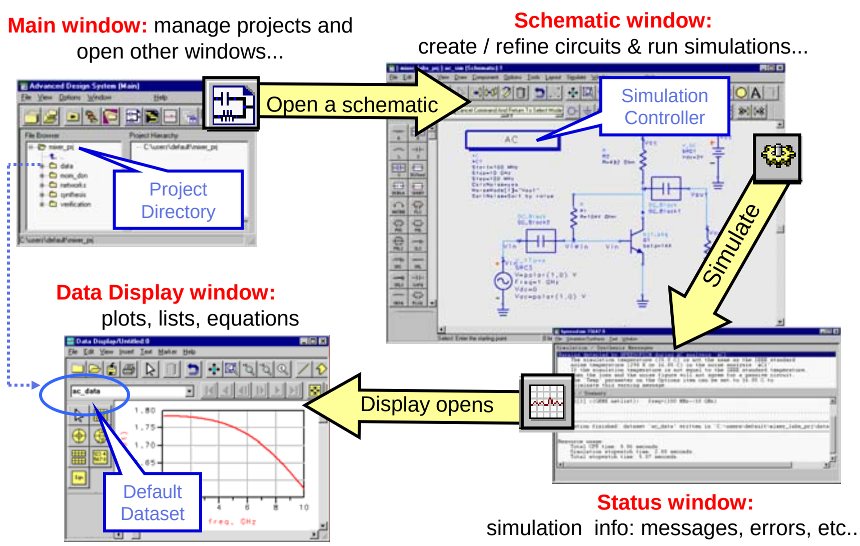
**$HPEESOF_DIR (UNIX variable) or %HPEESOF_DIR% (PC variable)
points to where you get ADS.**

**LM_LICENSE_FILE points to the license file
(default is: LM_LICENSE_FILE = $HPEESOF_DIR/licenses/license.dat.).**

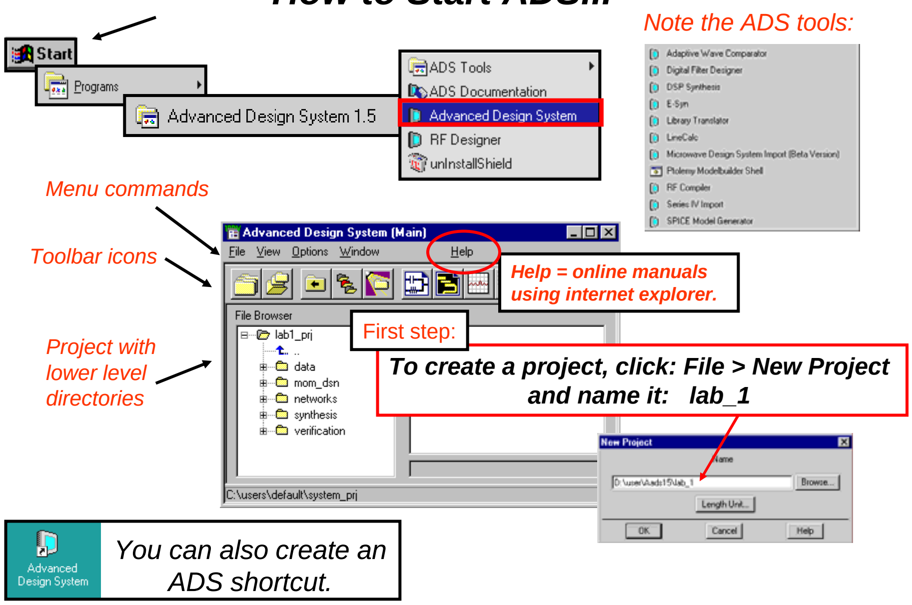
NOTE: *When you install ADS, you will be prompted where to load ADS (HPEESOF_DIR) and where you want to run ADS (PC: C:\users\default).*

For this class (US laptops), you should be working in either:
D:\user\ads15
or on the HP 3000- C:\user\ads15

ADS Windows: Main, Schematic, Status, Data Display



How to Start ADS...



ADS Project Directory Structure and Files

Project *for example: lab_1_prj*

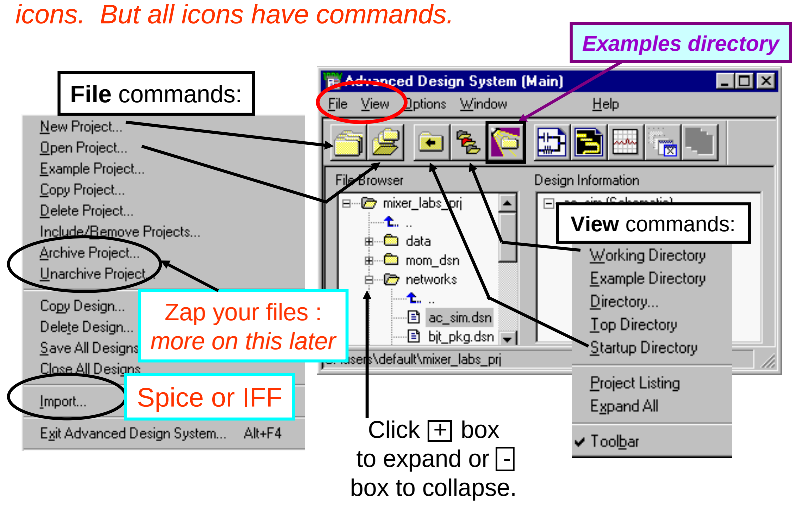
- preferences & ADS netlist.log
- *filename* .dds **Data Display files** (windows to display data)
- **data**
  - *filename*.ds **Dataset files** (simulation data)
- **networks**
  - *filename*.dsn **Schematic & Layout files**
  - *filename*. **ael** (application extension language)
  - *filename*. **atf** (compiled ael)
- mom_dsn (Momentum only)
- synthesis (used for E-Syn & DSP)
- verification (used for DRC)

Not required for most simulations (can be removed)

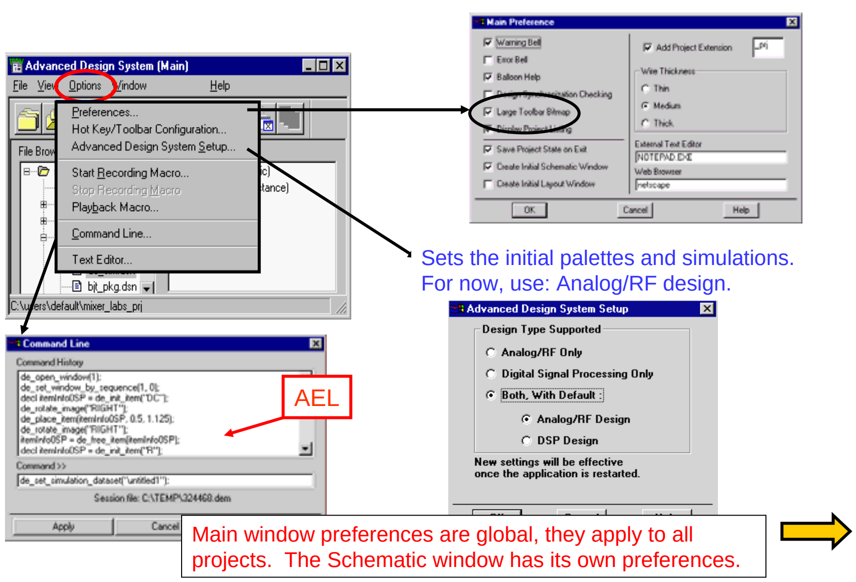
Automatically created by ADS

Main window: File , View, and more...

Use icons or commands. However, not all commands have icons. But all icons have commands.



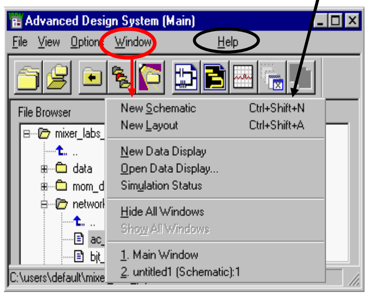
Main window: Options

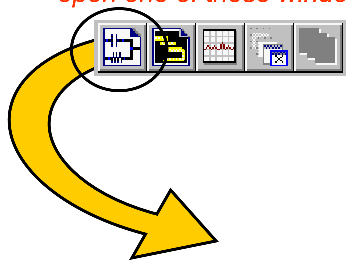


Main window: Window commands...

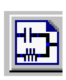
Notice the default Hot Keys

If you are in a project, you can open one of these windows:





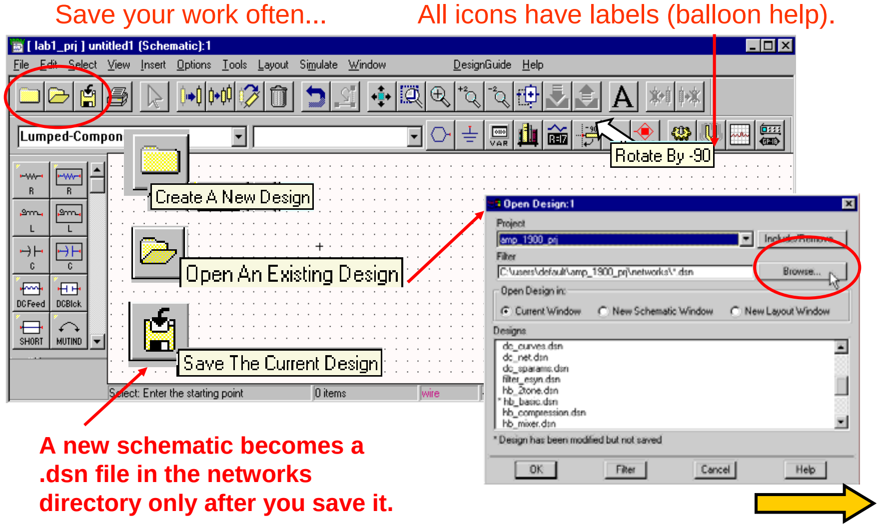
So, let's open the Schematic window...



Schematic *window*

Save your work often...

All icons have labels (balloon help).

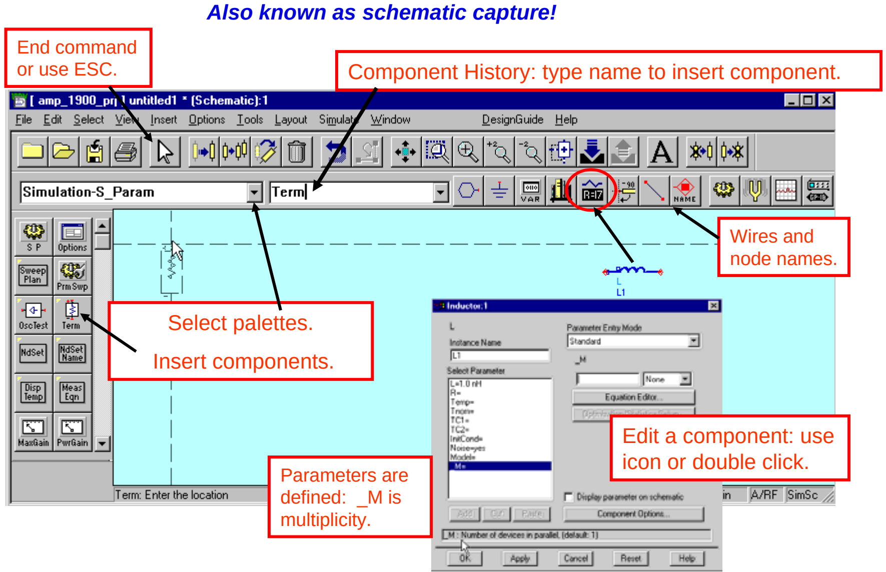


A new schematic becomes a .dsn file in the networks directory only after you save it.

Also – use Window > Open Designs

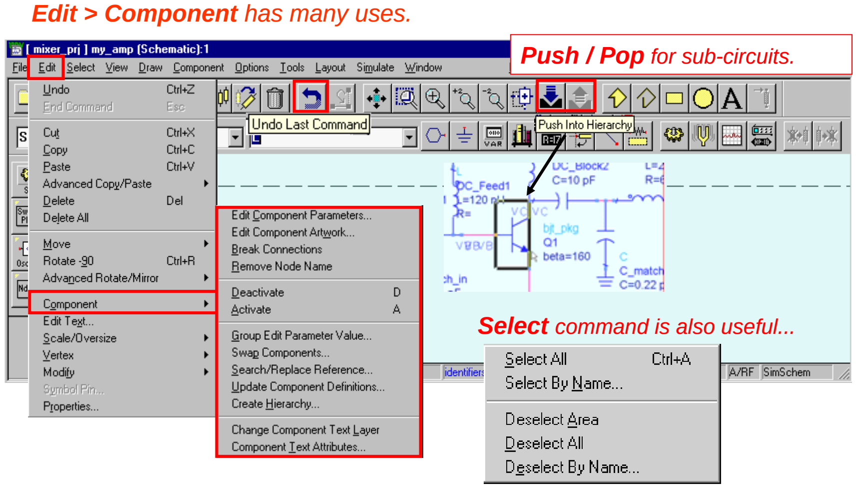
Inserting and Editing components

Also known as schematic capture!

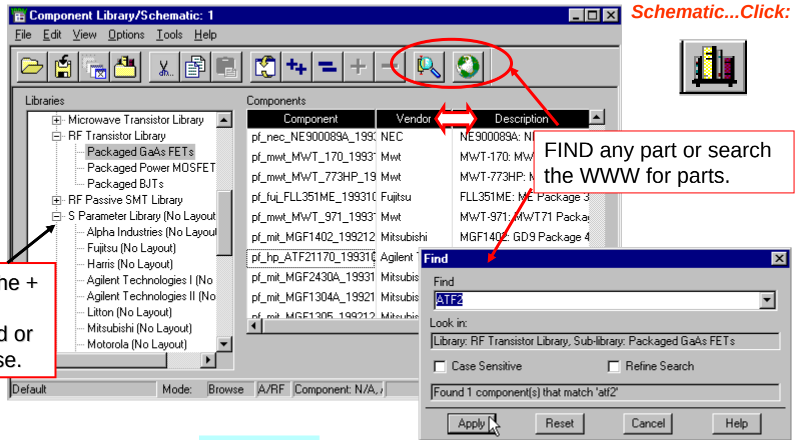


Editing commands and icons

***Edit > Component** has many uses.*



Library vendor parts + all your circuits

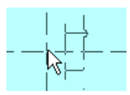


Schematic...Click:

FIND any part or search the WWW for parts.

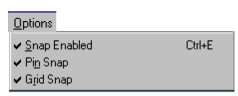
Click the + or - to expand or collapse.

Select the part and it is attached to your cursor, ready to insert.

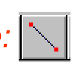


Wiring and Moving components

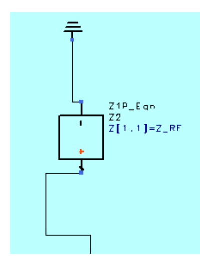
Tips for wiring!



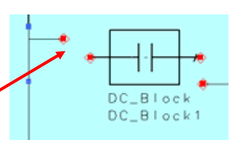
Click to wire:



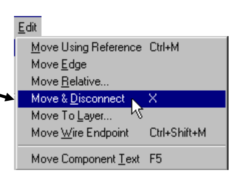
- Use the wire or connect pins = wired
- Point and click to snap to grid
- Drag a wired component and it stays wired.
- Wire colors are in Options > Layers



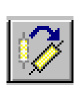
Red pin = not connected

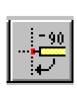


Edit wired components:



Two rotation icons:

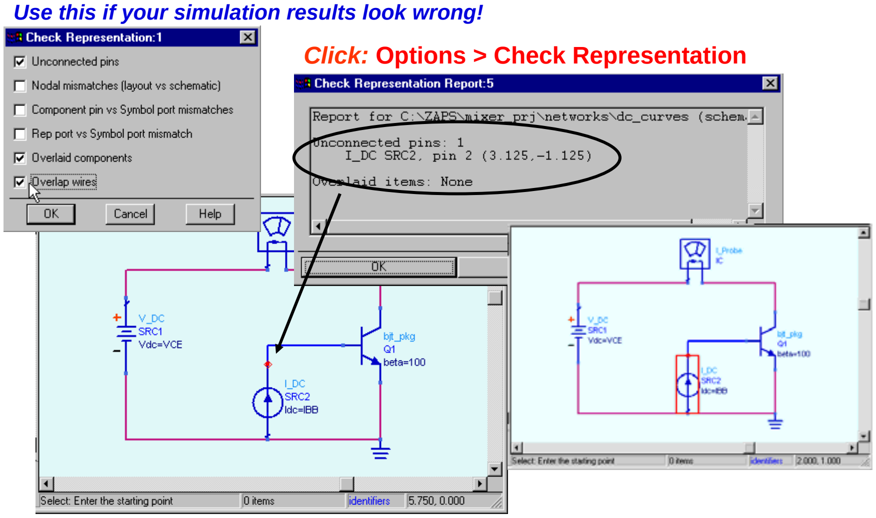




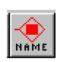
Check Representation for errors

Use this if your simulation results look wrong!

Click: **Options > Check Representation**

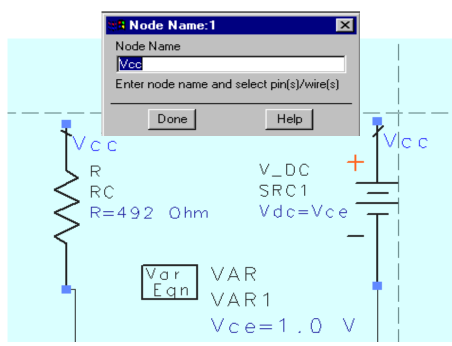


Node Names in Schematics

Click the icon: 

- Node names result in simulation ***data*** (node voltage) in the dataset.
- Node names can also connect two points without using a wire.
- Global nodes can connect node names in sub-circuits:

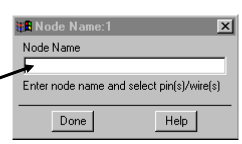
Insert > Global Node



To remove a node name:

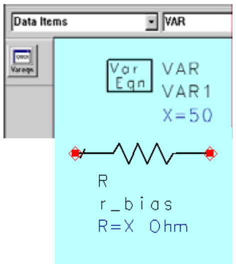
Edit > Component > Remove Node Name

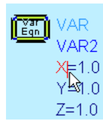
or ***use the icon with a blank name.***



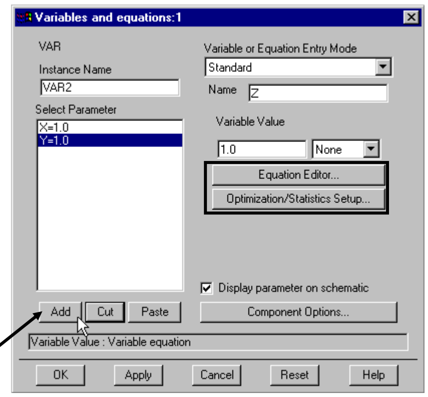
Variable Equations: VAR

Click: 





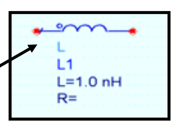
TIP: Add dummy (X,Y,Z) variables and then edit on-screen.

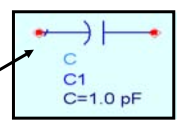


VAR is a declaration (initialization). Component parameters can be assigned a value using a variable equation.

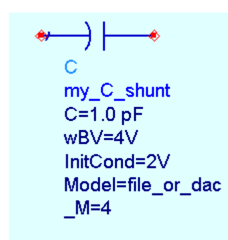
Variables can be used with optimization, parameter sweeps, and many ...

Symbols, units, and names

Circle for mutual inductance: 

Slash for pin# 1 (layout): 

Example of on-screen control:



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

QUIZ: Is this valid?	C coupling_c C = x	*Answer:* ________

Units and case sensitivity

- General Rule #1: UNIX is always case sensitive.
- General Rule #2: PCs are case sensitive, except for:

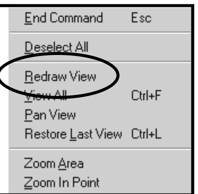
– Inserting some components: R or r is OK, because after the first insert, PC will recognize either one.

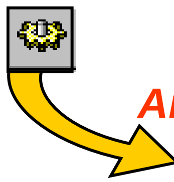
However, some rules apply to both, for example:

m = milli, M = Mega, V = volts

& <u>variable names</u> are always case sensitive!

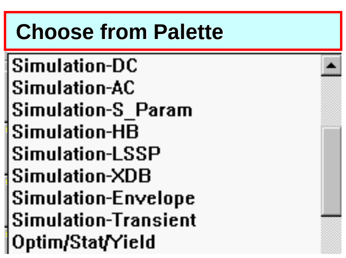
By the way, right mouse button has some commands:





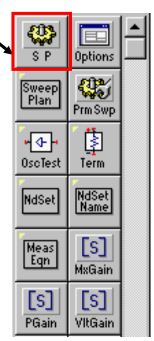
And now, let's Simulate...

First, select the Simulation Controller

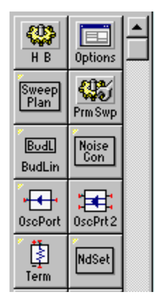


Click the gear to insert the controller.

S-Parameter



HB



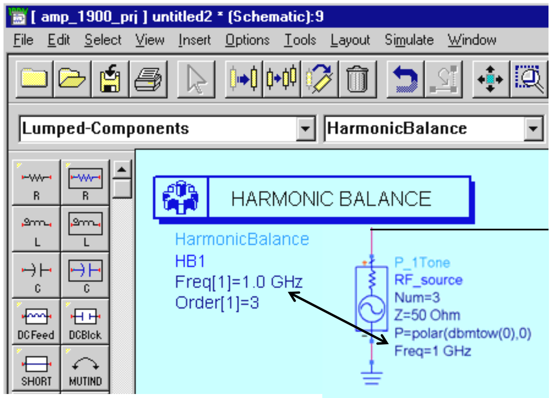
DC

NOTE: all palettes have many of the same components (Prm Swp) and specific Meas Eqns.

Set up the Simulation

Simulation requires setting the simulation controller.

*NOTE asterisk means schematic is not yet saved.



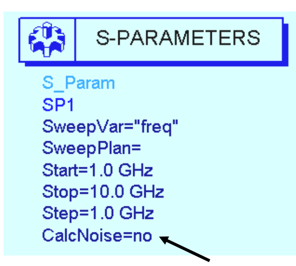
Some simulations, like HB, require more setup.

DC requires no setup in most cases.

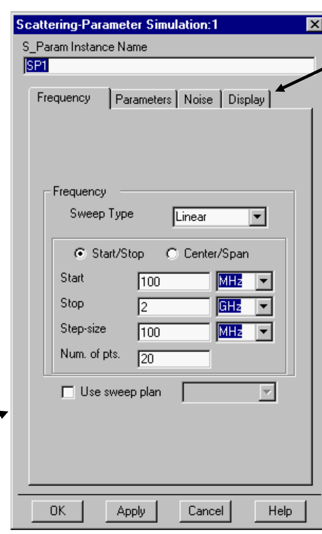
S-param requires Terms.

Edit on-screen or double click for dialog box.

Example dialog for Controller



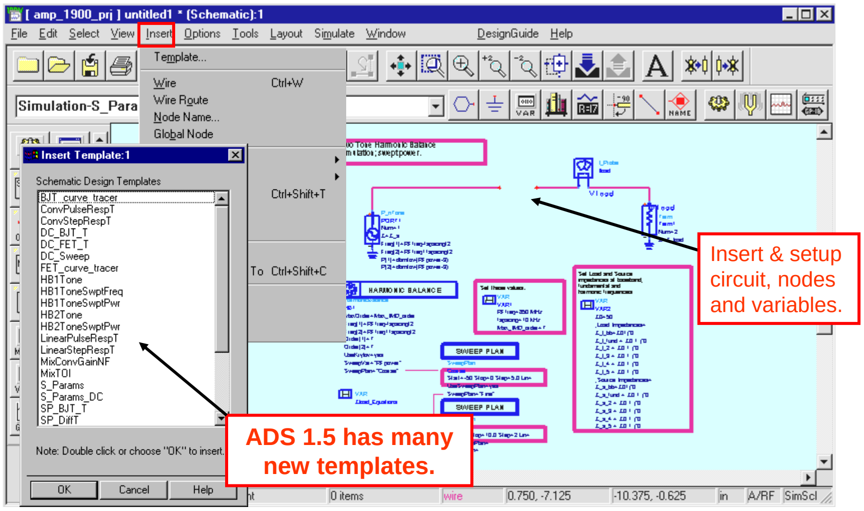
*Edit **on-screen** if the parameter is displayed or use the dialog box.*



*Display the parameters: **Display tab** lists all the settings you can show on-screen.*

You will get lots of practice in the labs.

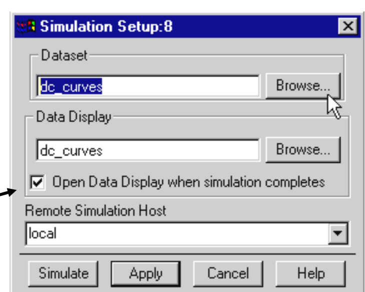
Templates for simulation



*Specifying a **Dataset** and **Data Display***

***<u>Before you simulate:</u>** you can name the dataset. If not, the default dataset = schematic name or the last dataset named.*

Click: **Simulate > Setup:**



Uncheck this box if desired.

NOTE: Dataset files (.ds) are in the DATA directory.
Data Displays (.dds) are in the PROJECT directory & read datasets.

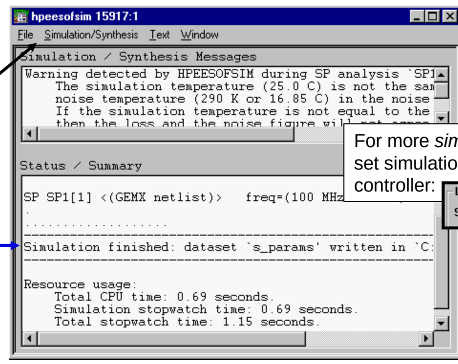
To run the simulation, click F7 key or click the gear...

During Simulation: Status Window appears!

One way to stop a simulation, click:

Simulation/Synthesis > Stop Simulation

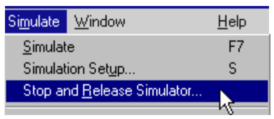
Or click: Window > Simulation Status

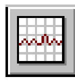


For more *sim* info, set simulation controller:

A successful simulation results in a ***dataset:***

To stop a simulation from the schematic window.

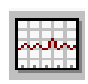


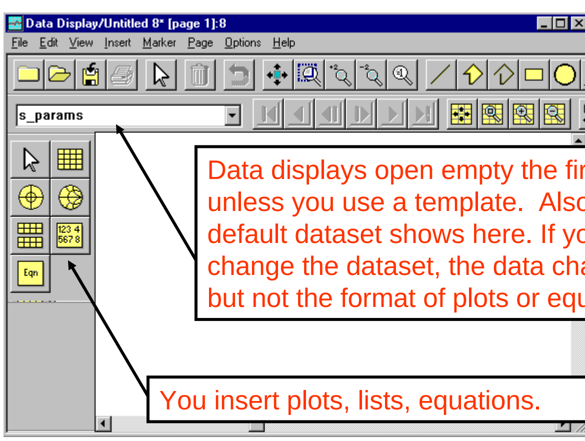


When finished, the Data Display opens...

Data Display window

Also, open Data Displays from schematic or the Main window:

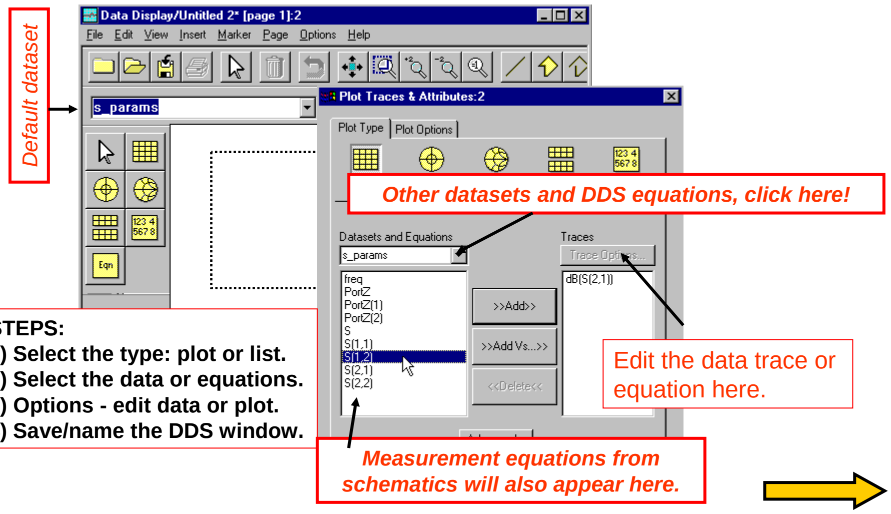




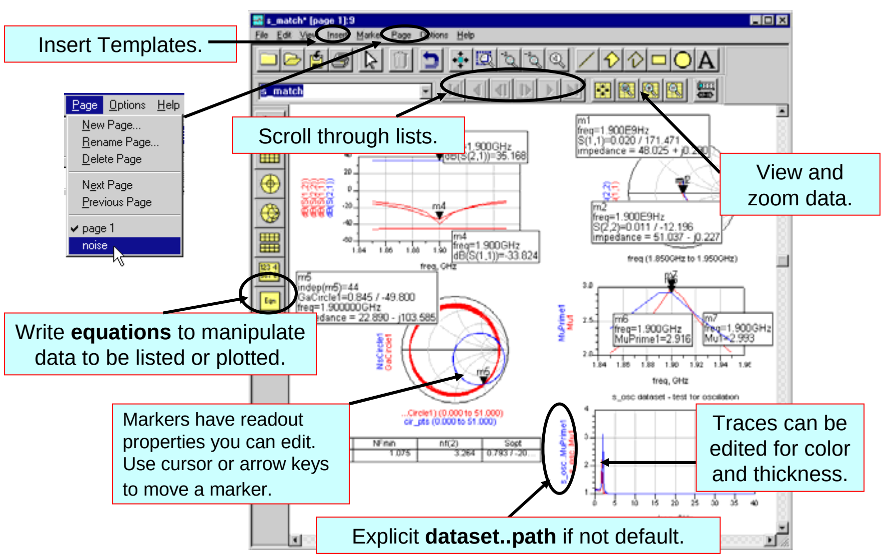
Data displays open empty the first time, unless you use a template. Also, the default dataset shows here. If you change the dataset, the data changes, but not the format of plots or equations.

You insert plots, lists, equations.

List or plot your simulation data



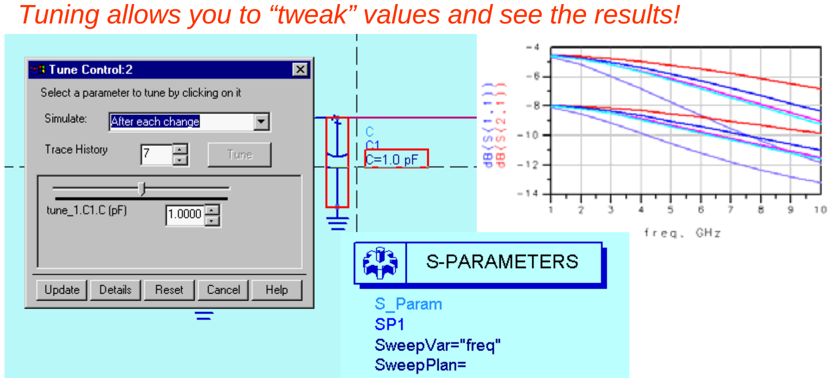
Data Displays are powerful...



Tune mode is simulation!

Simulate > Tuning...

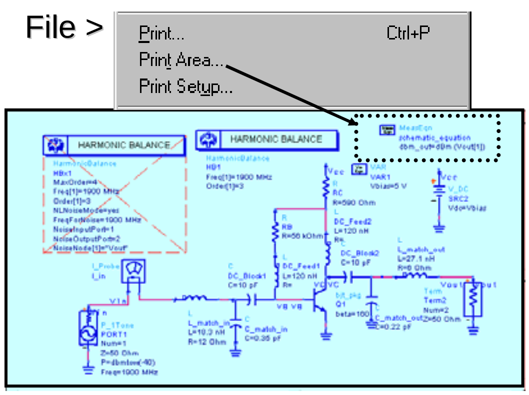
Tuning allows you to "tweak" values and see the results!



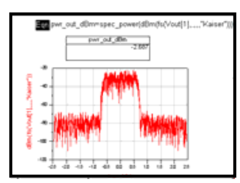
You will do this in the lab in just a few minutes...

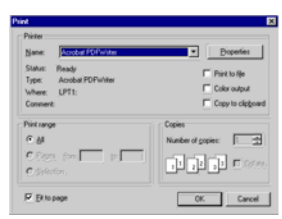
Printing data or schematics

File >



For Data, you can print selected plots, lists, equations.





Print Setup - select the printer, style, etc.

NOTE on ending ADS processes

If your computer is locked up or if there is any other problem (Data Display), you can safely stop some processes:

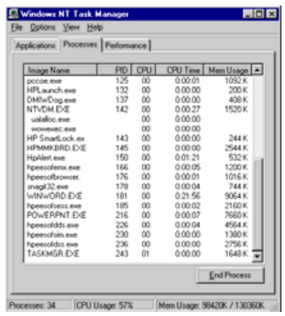
hpeesofde.exe - closes the ADS program (same as exit)
Hpeesofsim.exe - stops the simulation
hpeesofdds.exe - end to stop the data display server
hpeesofdss.exe - end to stop the dataset server

These 2 work together!

UNIX users kill processes - PC users end processes

PC task manager - NT: ctrl-alt-delete

In a UNIX window, use: ps -ef | tail and you can kill (xxx) a processes.



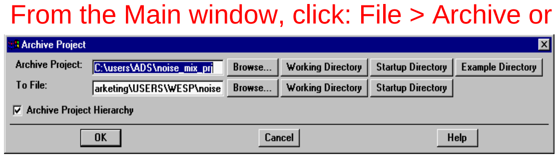
AVOID killing these:

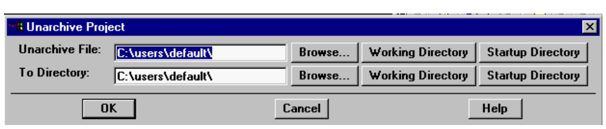
hpeesofsess.exe
hpeesofbrowser.exe
hpeesofemx.exe

Killing these will require re-starting ADS.

*To mail or send ADS projects: **ZAP***

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





NOTE: Archive files become .ZAP files (like .ZIP files). They can include all networks, data, and display files (entire project).

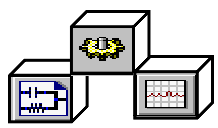
When you leave this class, archive the projects to C:\temp and then copy the ZAP to the A drive. Be sure to delete the datasets or it will not fit on a floppy disk (not necessary if you e-mail a zap file).

For any ADS problems, call support and send the ZAP file. In the United States call: 1 - 800 473-3763.

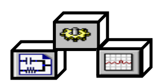
What the lab is about ...

Lab 1:

Circuit Simulation Fundamentals



Learn Circuit Simulation Basics



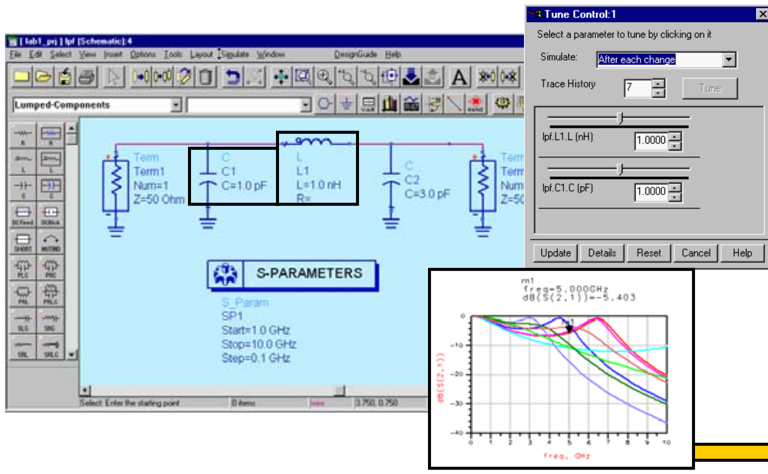
- Create a project and schematic
- Build a low-pass filter
- Set up and run an S-parameter simulation
- Display the results and Tune the filter
- Copy an example RFIC amplifier
- Insert a library part and simulate with a Template
- Simulate an RFIC amp with Harmonic Balance
- Display the results and write equations

***NOTE*: Lab 1 can be skipped if students already know the basic operation of ADS: schematic capture, simple simulation, basics of plotting data.**
Also, it's OK to make mistakes…you won't need the work you do in this lab for any of the other exercises.
But starting in Lab 2, you will!

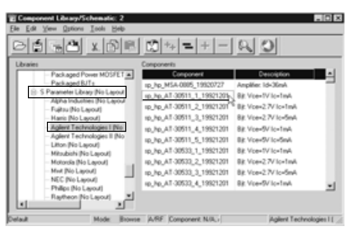
STEPS IN THE LAB:

Build and simulate a low-pass filter

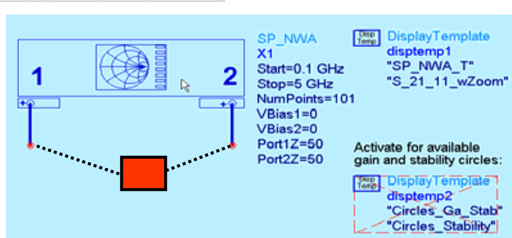
After setting up the S-parameter simulation, tune the filter.



Also…library device simulation with template

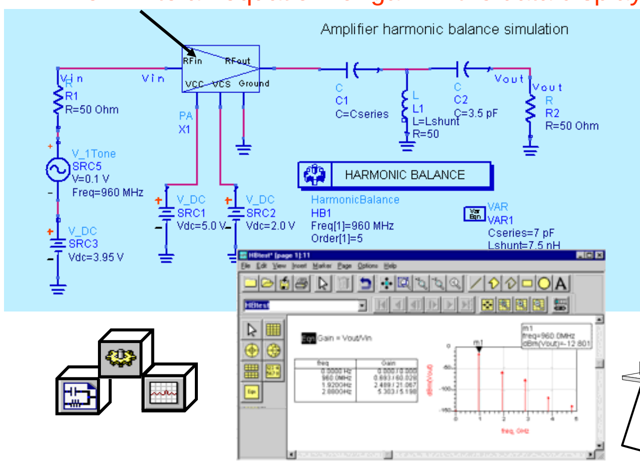


Insert a library device & simulation template: Get S-parameters for all swept bias points!



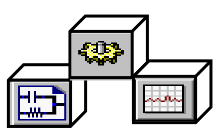
Last, example RFIC with Harmonic Balance

You will name the node Vin, check the sub-circuit, and simulate.
Then write an equation for gain in the data display.

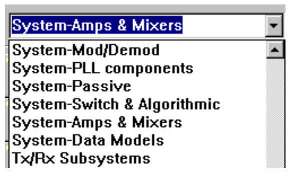


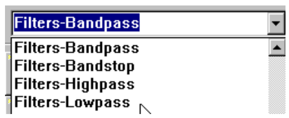
Topic 2:

System Design Fundamentals



System component libraries and fundamentals:

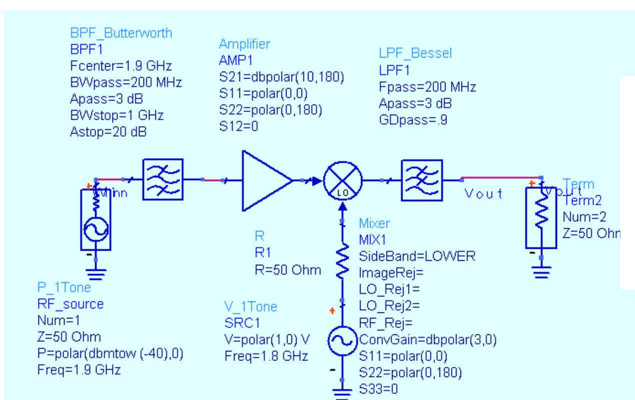




- System design is at the higher level.
- System models are *behavioral*: equation based describing node I and V, table based, etc.
- System components can be integrated with circuit components.
- IMPORTANT - System simulation and data display are the same as for *circuit*.

Typical system design using ...behavioral models

What are behavioral models?

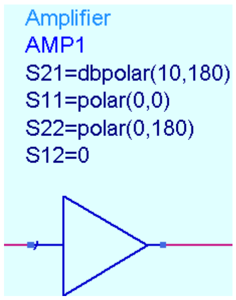


Behavioral models are described by equations.

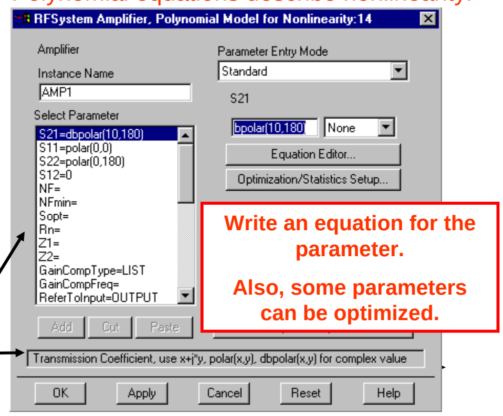
(Similar to SDD-time domain and FDD-frequency domain.)

Typical system model: Amplifier

Behavioral model



Polynomial equations describe nonlinearity:

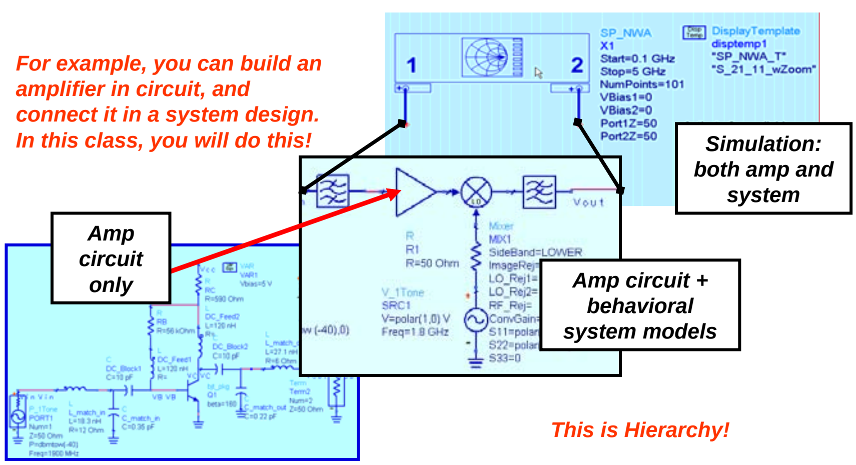


You specify the behavior:

Simulation is the same

for circuit and system (behavioral):

For example, you can build an amplifier in circuit, and connect it in a system design. In this class, you will do this!

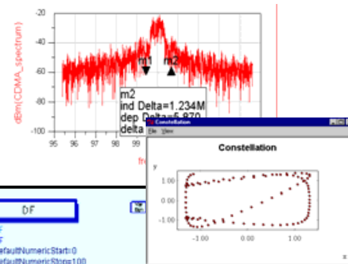


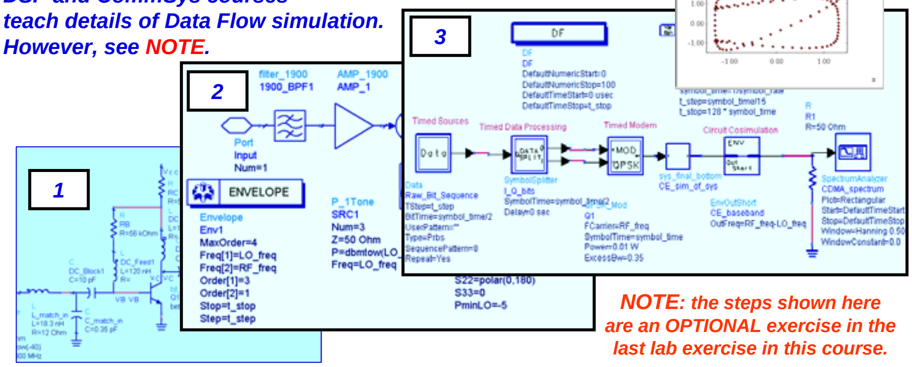
Data Flow simulation

Data Flow simulation (Ptolemy) is 3 levels here:

1 - Circuit design
2 - System designs with Envelope or Transient
3 - Data Flow (Ptolemy): bits, sinks, TK plots

DSP and CommSys courses teach details of Data Flow simulation. However, see NOTE.



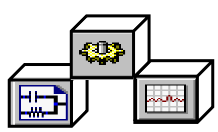


NOTE: the steps shown here are an OPTIONAL exercise in the last lab exercise in this course.

What the lab is about ...

Lab 2:

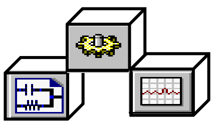
System Design Fundamentals



Steps in the Design Process

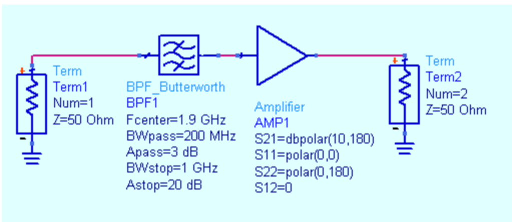
You are here:

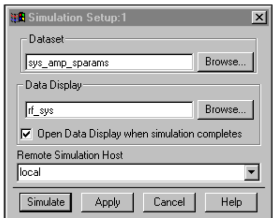
- ***Design the rf_sys behavioral model receiver***
- ***Test conversion, budget gain, spectrum, etc.***
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*

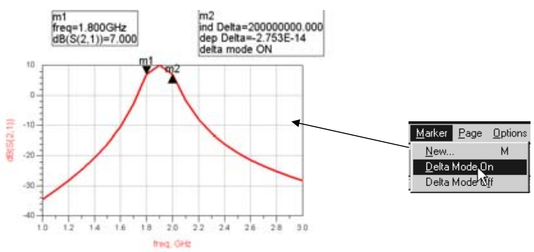


Amp and filter simulation: S-parameters

S-21 measurement verifies bandpass and gain:

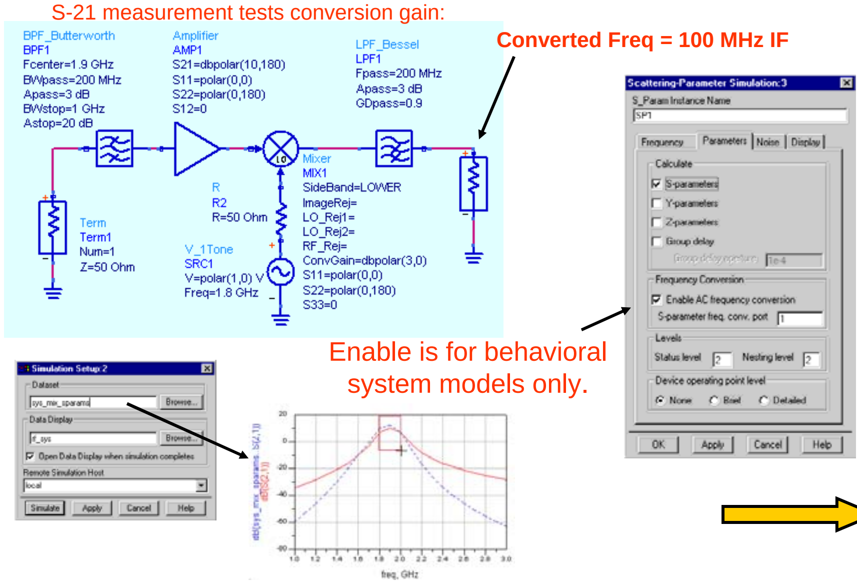






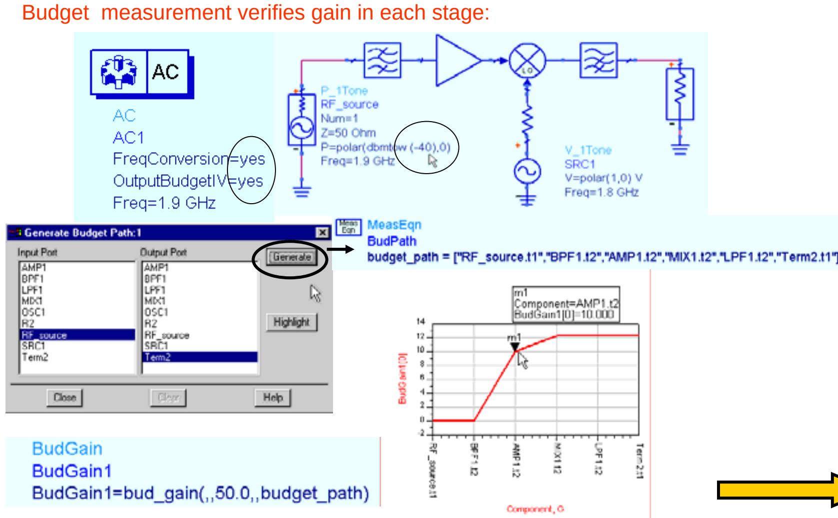
Receiver simulation: S-parameters

S-21 measurement tests conversion gain:



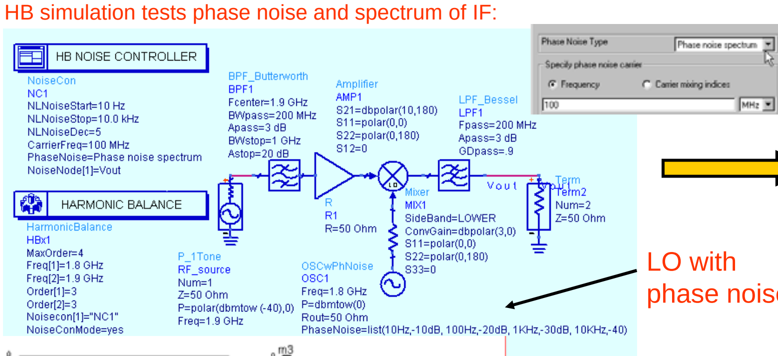
AC simulation with Budget Gain

Budget measurement verifies gain in each stage:

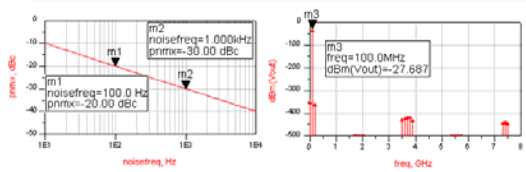


HB simulation with Noise Controller

HB simulation tests phase noise and spectrum of IF:



LO with phase noise

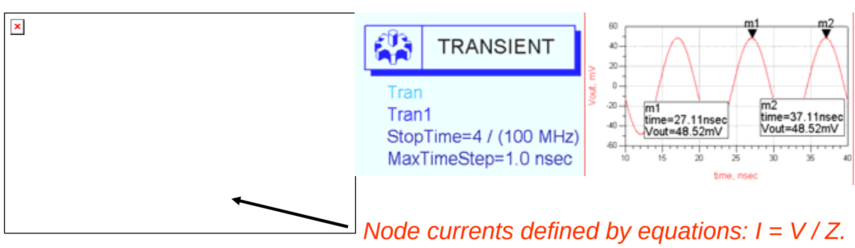


Plot spectrum and phase noise.

Phase Noise NOTE: HB FFT oversample parameter can be increased until answer is constant!

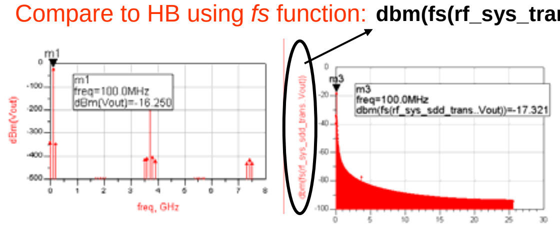
Transient simulation of an SDD

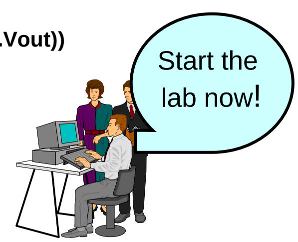
Time domain signal displayed and transformed into the frequency domain:



Node currents defined by equations: I = V / Z.

Compare to HB using *fs* function: **dbm(fs(rf_sys_trans..Vout))**





Class Exercise

(after lab 2)

HOT KEYS and Schematic Preferences

Efficient ADS Techniques

Would you like to customize the system to be more efficient for the way you work?
Here are some things you can do:

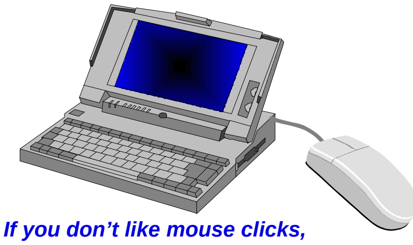
- Moveable Toolbar: PC only - put it below
- Tear Off Menus: UNIX only - keep some on screen
- Options > Preferences: set these to your liking
- Templates - learn to use them or set up your own
- Examples - familiarize yourself with them
- HOT KEYS - Options > Menu / Toolbar Configuration

Pre-set schematic Hot Keys

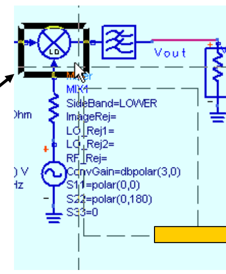
***Pre-configured keys*:**

F7 = Simulate
F5 = Move Component Text

Ctrl+R = Rotate 90
Ctrl+M = Move
Ctrl+C = Copy
Ctrl+Z = Undo
plus more...



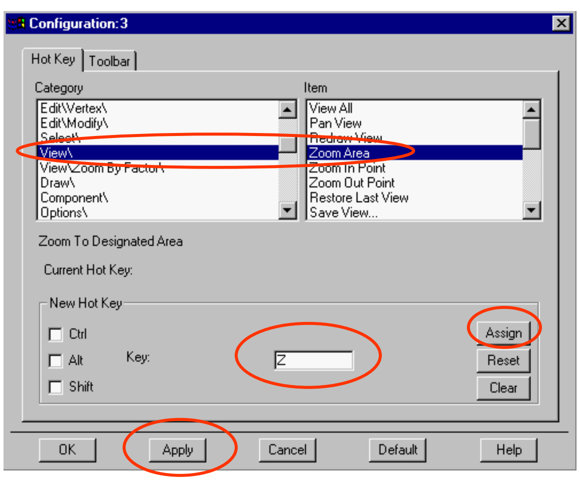
If you don't like mouse clicks, HOT KEY your keyboard. Its global for all projects



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

Set the *View > Zoom* schematic *HOT KEY*

Click: **Options > Hot Key / Toolbar Configurations...**



Follow these steps to set Zoom Area command:

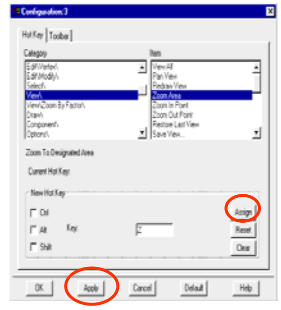
1. Select the command
2. Type in a letter: z (not case sensitive)
3. Click: **Assign**
4. Click: **Apply**
5. Now,try the Z hot key to verify it works.

Next, set these ***Hot Keys***

Options > Hot Key / Toolbar Configurations...

S = Simulate > Setup
A = Activate
D = Deactivate
X = Edit > Move > Move & Disconnect

and any others you want ...

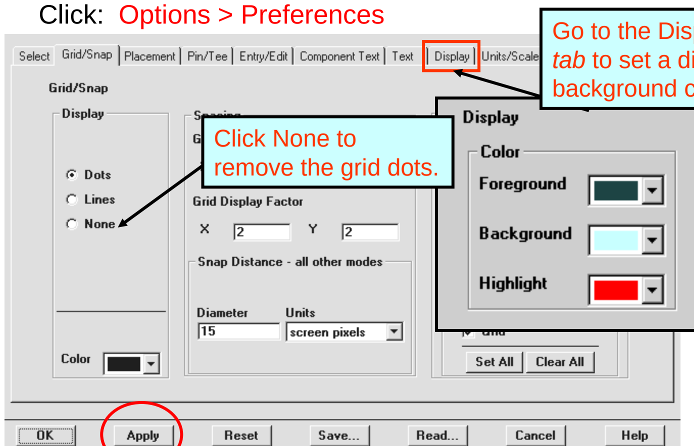


You will be able to use these hotkeys for all the labs in this project.

When everyone has finished, continue

If desired, set Schematic Preferences

Click: Options > Preferences



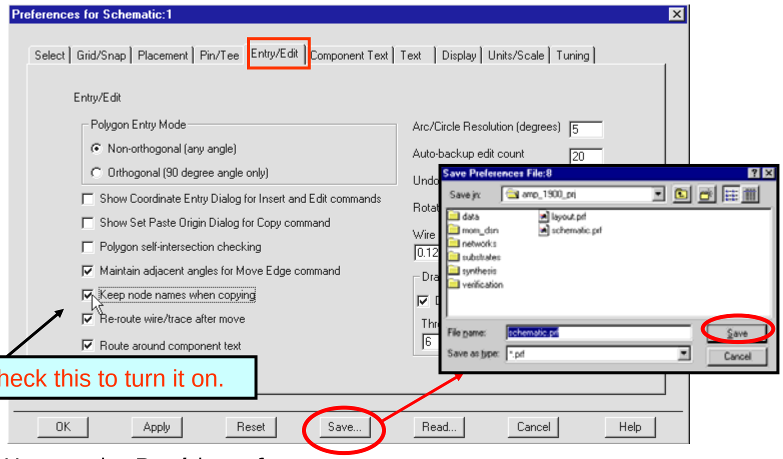
Go to the Display *tab* to set a different background color.

Click None to remove the grid dots.

NOTE: Set wire color in - Options > Layers.

Save the schematic preferences

...and the settings will then apply to all schematics in your project.



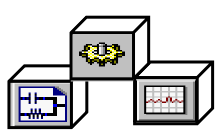
Check this to turn it on.

You can also ***Read*** the ***.prf*** file into other projects.

★ End of class exercise.

Topic 3:

DC Simulations and sub-circuit modeling



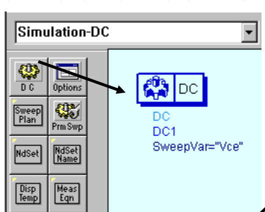
DC Simulation

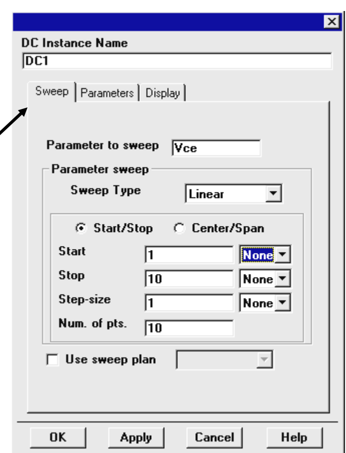
You get steady-state DC voltages and currents according to Ohm's Law: V= IR

- 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
- Convergence simulator algorithms (modes) can be set

DC simulation controller

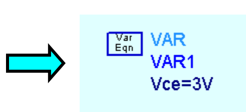
Simulation Controller and Editor (dialog box)



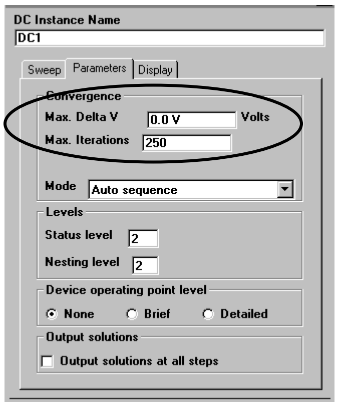


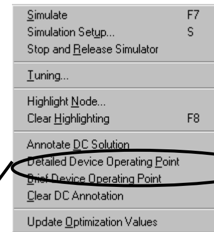
Sweep: allows you to sweep a parameter but it must be defined as a variable. Note the dialog entry automatically puts quotes on the controller (screen) entry.

VAR

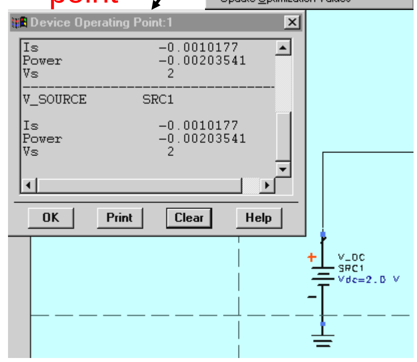


...more on DC





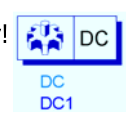
Example: device op point



Convergence: increase V or iterations or change mode if you don't converge.

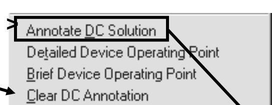
Back Annotation

No settings necessary!

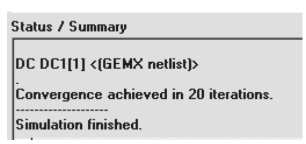


Immediately after DC simulation, currents and voltages are available. Click: **Simulate > Annotate DC Solution**.

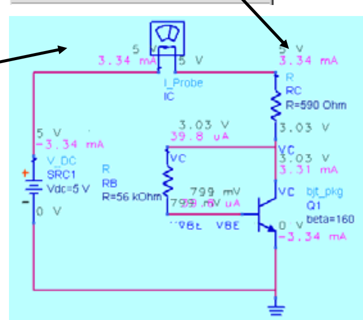
Simulate >



Clear it here



Current Probe = use values of current in the dataset.



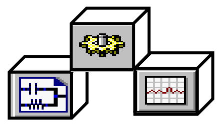
Minus sign used for current flowing out of a connection. Otherwise, current flows into a connection or device.

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

What the lab is about ...

Lab 3:

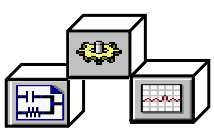
DC Simulations and modeling the sub-circuit



Steps in the Design Process

You are here:

- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- ***Start amp_1900 design – subckt parasitics***
- ***Simulate amp DC conditions & bias network***
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*



Start with some specifications...

***AMP** with max gain & low noise:*

Available voltage: 5 volts
Device: Generic BJT (Gummel-Poon)
Collector current: 3.25 mA
Frequency: RF = 1900 MHz
Gain: >10 dB (or much more with this model)
50 ohm match input/output

Filters: also, build 1900 MHz BPF for the input and a LPF for the IF output

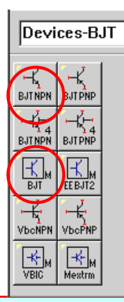
Later on, test the AMP for TOI, distortion, noise, compression,
GSM & CDMA modulation response, and more in labs 3 through 9.

***YOUR JOB**: Build, test, and refine the circuits to meet specifications.*

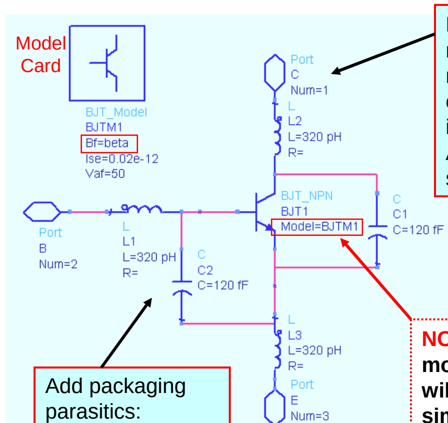
Start with a sub-circuit model...

Model the device with package parasitics

Create a sub-circuit to model your components:



Insert a device and a model: Gummel-Poon BJTM1. Bf will be a passed parameter. Vaf is changed as shown.

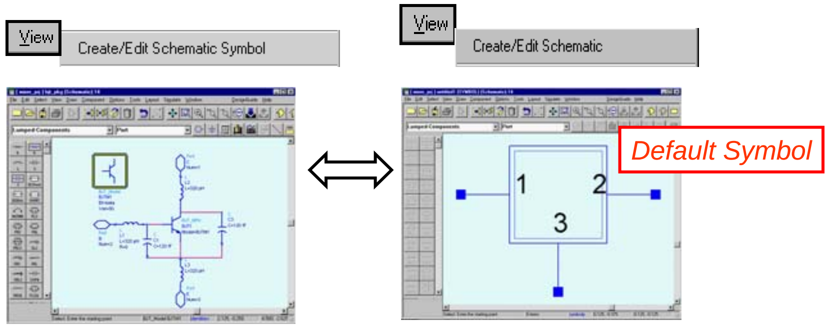


Port connector numbers: **Num=** must be set in specific order as shown - this is necessary to use ADS built in transistor symbol.

Add packaging parasitics:
L in pico: pH
C in femto: fF

NOTE: BJTM1 is the model card name that will be used for simulation. Library devices do not require this mapping.

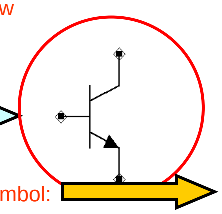
Select a symbol for your sub-circuit



Schematic view

Symbol view

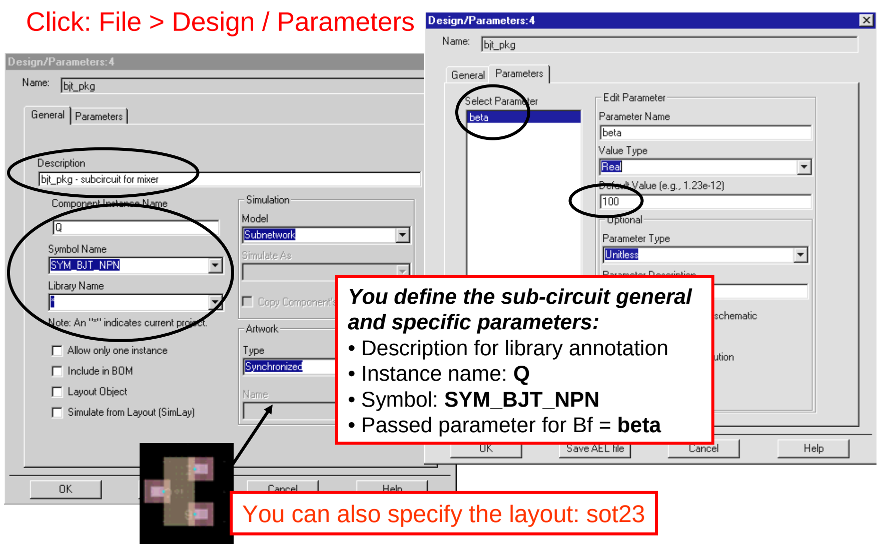
To use a built-in ADS symbol that looks like an NPN BJT use **File > Design Parameters**.
Or, draw your own symbol!



Its easy to use a built-in symbol:

Define the sub-circuit parameters: 2 tabs

Click: File > Design / Parameters

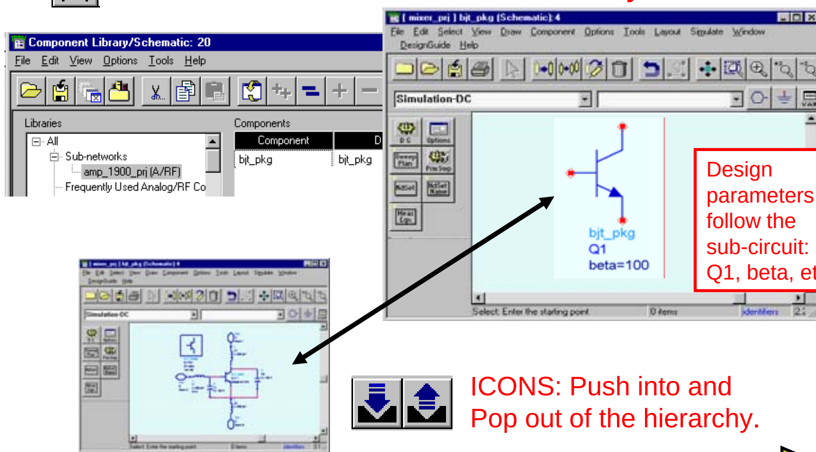


You define the sub-circuit general and specific parameters:
- Description for library annotation
- Instance name: **Q**
- Symbol: **SYM_BJT_NPN**
- Passed parameter for Bf = **beta**

You can also specify the layout: sot23

Insert the model in a new schematic

Insert the sub-circuit from the library.

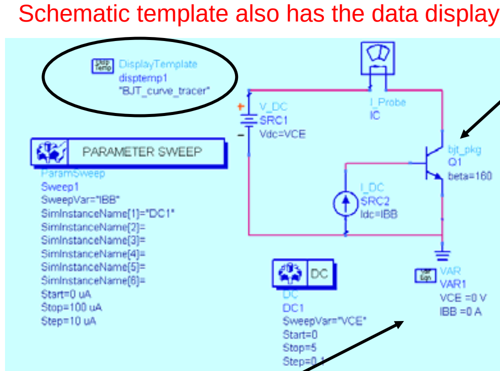


Design parameters follow the sub-circuit: Q1, beta, etc.

ICONS: Push into and Pop out of the hierarchy.

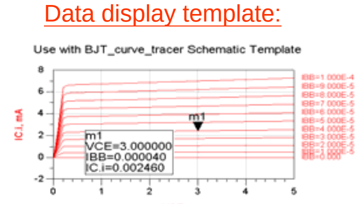
Set up a DC curve sweep with a ***template***

Schematic template also has the data display template:



Your model (**bjt_pkg**) with annotation and passed parameter: beta.

Data display template:

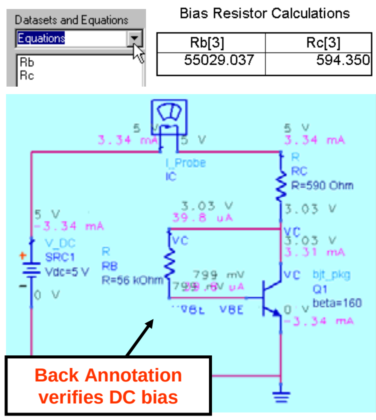


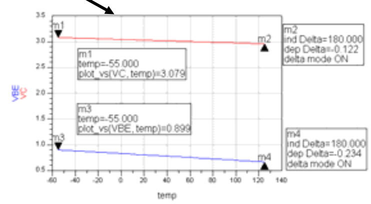
Initialized **VARs:**
VCE = 0 V
IBB = 0 A

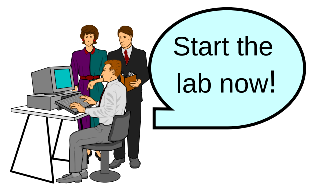
NOTE: DC controller sweeps the X-axis and the Parameter Sweep, sweeps the Y-axis.

Finally, calculate and test the bias network

Calculate resistor values, verify the DC specs for Ib and Ic, and sweep temperature!

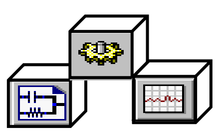






Topic 4:

AC Simulation and Tuning Parameters

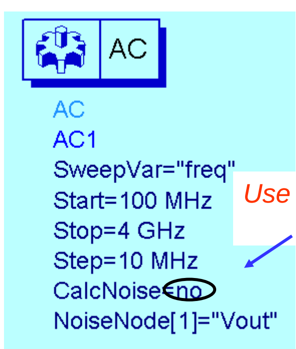


AC Simulation

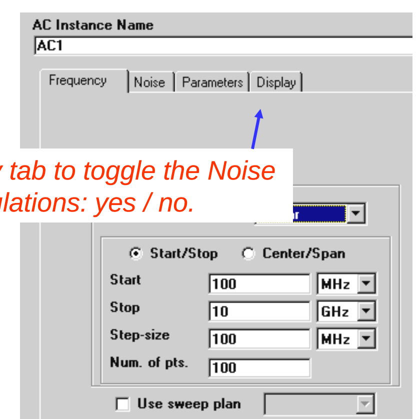
You get linear small-signal response and you get Noise values:

- 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



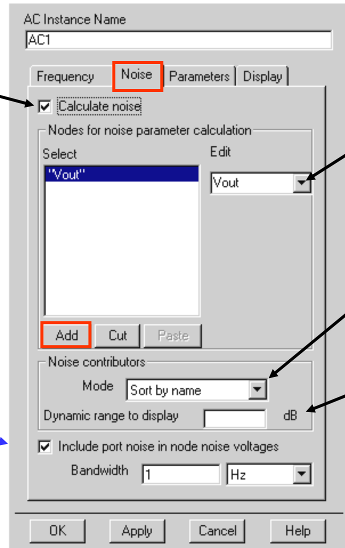
Use Display tab to toggle the Noise calculations: yes / no.



AC is a linear or small signal simulation and ***freq*** is usually set in the controller not the source.

Turn on and set up Noise calculation

Click here:



Your Node Names

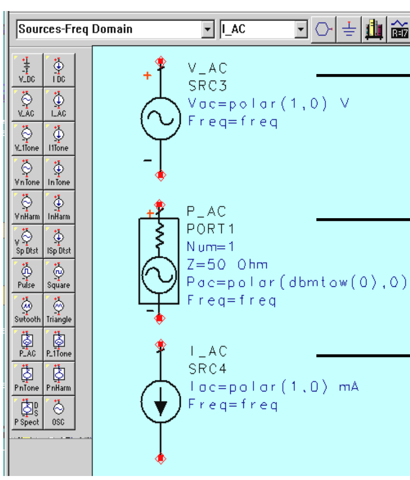
Sort by name or by value: in the dataset

Blank gives you all contributors

NOTE: Port Noise can be included in the simulation, but it does not apply to NF. Also, Port noise is turned on/off in the sources.

Use specific sources for specific simulations

Use AC sources for AC simulations!



Source parameter definitions:

- **V_AC** is the component name
- **SRC3** is the instance name (you can change this)
- **Vac = polar (1,0)V** is the default)
- **Freq =** ***freq*** is a global variable - you set the start & stop values in the simulation controller

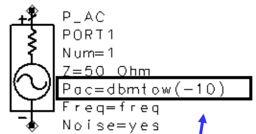
- **P_AC** = is the component name
- **PORT1** is the instance name (OK to change)
- **Num=1** is the port number (use for S-parameters)
- **Pac = polar (dbmtow(0),0)** dbmtow is a function
- you enter the dbm value (*see note)

- **I_AC** is the component
- **Iac=polar (1,0) mA** is the default
- Arrow is the direction of current flow.

More on AC source settings...

Setting AC source values

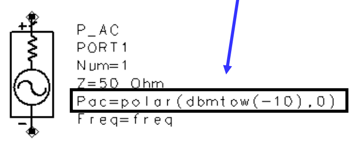
POWER SETTINGS: The dbmtow function converts power in dbm to watts for the simulator.



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

Simplify by removing **polar**

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 (superposition).



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

Review of ADS Equations

- **VAR**: pre-simulation -
  Use for initializing sweep variables or other settings. VARs are not available in the dataset unless the OutVar function is used (later labs).
- **MeasEqn**: pre-simulation -
  Use for calculations to be available in the dataset (can use node names and functions).
- **Eqn:** post-simulation -
  Use for calculations in the data display (can include node voltages, functions, and any other dataset data).

VAR
VAR1
Vbias=2.0 V

schematic

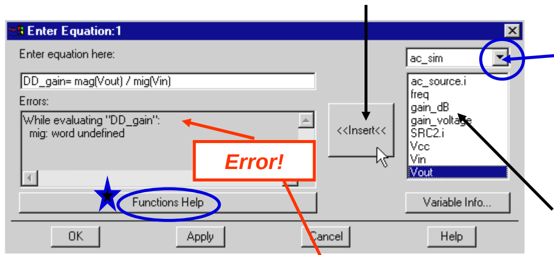
MeasEqn
meas1
Gain_dB=20*log(mag(Vout)/mag(Vin))

data display

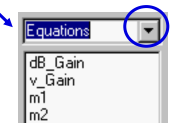
Eqn v_Gain= (mag(Vout) / mag(Vin))

Review the Data Display equation editor

Insert button gives full path (dataset..) if not the default.



Click here for your DDS equations:



Gain_dB is a <u>schematic</u> MeasEqn.

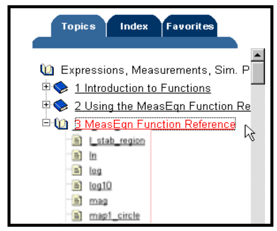
Invalid equations are red:

Eqn DD_gain= mag(Vout) / mig(Vin)

Valid equations are black:

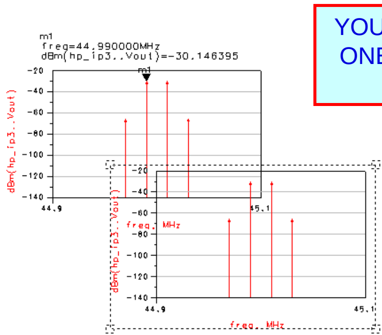
Eqn DD_gain= mag(Vout) / mag(Vin)

★ Click ***Functions Help*** for on-line manuals:



***TIP* for copy/paste in the Data Display**

- Keyboard keys: **Ctrl C** copies to the buffer
- Keyboard keys: **Ctrl V** pastes from the buffer



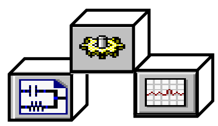
YOU CAN COPY A PLOT FROM ONE DATA DISPLAY WINDOW TO ANOTHER ALSO!

UNIX users probably know this but it works for ADS on the PC also. Try copying a plot or equation now!

What the lab is about ...

Lab 4:

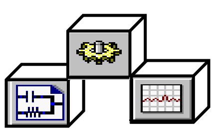
AC Simulations and Tuning Parameters



Steps in the Design Process

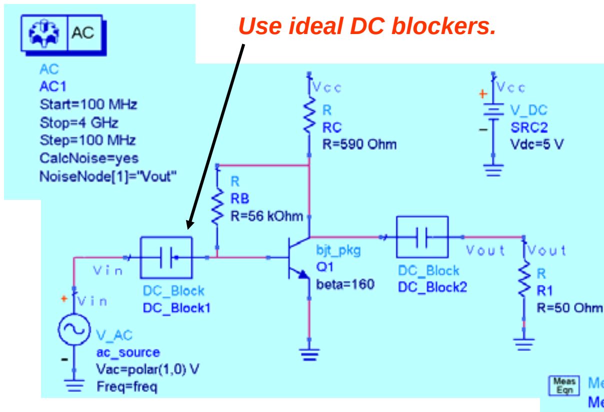
You are here: → (at "Simulate amp AC response - verify gain")

- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- ***Simulate amp AC response - verify gain***
- ***Test amp noise contributions – tune parameters***
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*



Set up the circuit & simulate with Noise

Use ideal DC blockers.

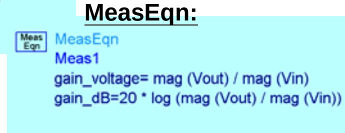


Vcc is a node name.

Vin and **Vout** nodes provide data (noise and voltage) for equations.

V_AC source voltage = 1 V, cosine wave.

MeasEqn:



NOTE: Freq is a global variable.
Here, Freq is controlled by the source.
Use freq=freq, freq=10 MHz, or a variable: freq=F_RF.

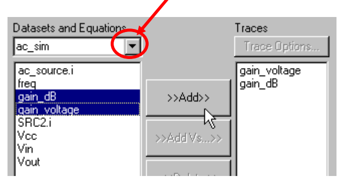
Simulation results...

Two schematic MeasEqns: *One Data Display eqn:*

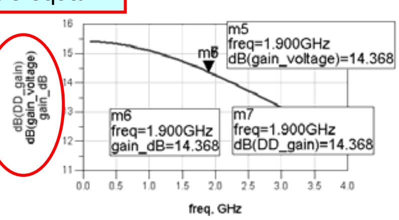
freq	gain_dB	gain_voltage	DD_gain
1.500GHz	14.736	5.455	5.455
1.600GHz	14.649	5.401	5.401
1.700GHz	14.559	5.345	5.345
1.800GHz	14.465	5.287	5.287
1.900GHz	14.368	5.229	5.229
2.000GHz	14.268	5.169	5.169
2.100GHz	14.165	5.108	5.108
2.200GHz	14.061	5.047	5.047

Eqn dB_Gain=20*log(mag(Vout) / mag(Vin))

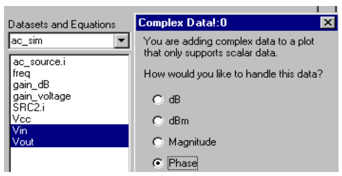
Write the same equation in Data Display as you did in schematic. Then put it in a list.



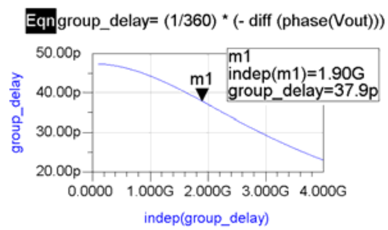
Edit the traces on-screen: all are equal.

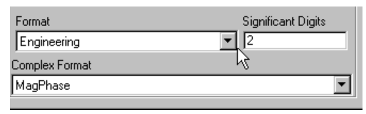


More AC results...

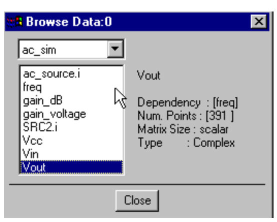


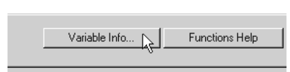
Calculate group delay with an equation using Phase data. Also, control marker readout formats.

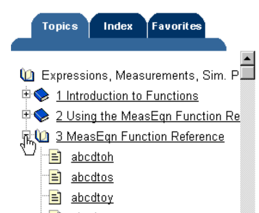




*Use the **what** function…what?*







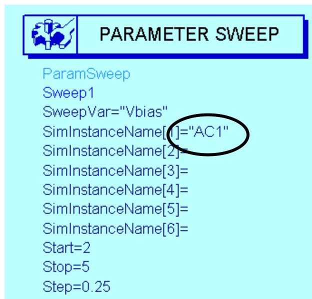
Examine the data using the **what** function and the **Variable Info**.

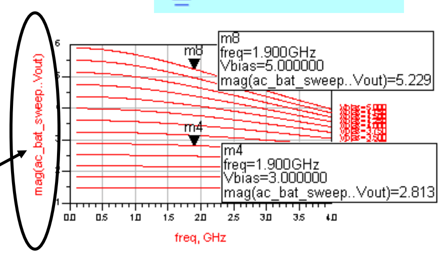
what (Vout)
Dependency : [freq]
Num. Points : [391]
Matrix Size : scalar
Type : Complex

freq	what (Vout)
100.0MHz	5.901 / 178.294
200.0MHz	5.895 / 176.590
300.0MHz	5.883 / 174.891
400.0MHz	5.868 / 173.199

Sweep battery voltage

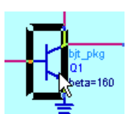
For an **AC** simulation, use a **parameter sweep** of a dc value which is assigned to a **VAR.**



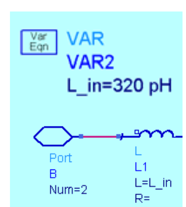


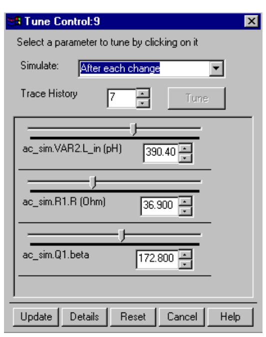
Note the explicit dataset path..

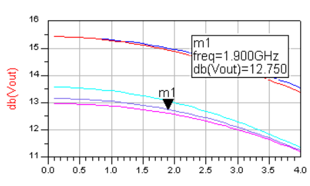
OPTIONAL - tune sub-circuit parameters



Set up a **variable** in the sub-circuit and tune several parameters from the top level.

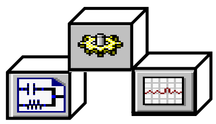






Topic 5:

S-parameter Simulation and Optimization



S-parameters are Power Ratios

(voltage ratios squared)

S-parameter ratios: *S out / S in*

These are best viewed on a Smith chart (next slides).

- S11 - Forward Reflection (input match - impedance)
- S22 - Reverse Reflection (output match - impedance)
- S21 - Forward Transmission (gain or loss)
- S12 - Reverse Transmission (isolation)

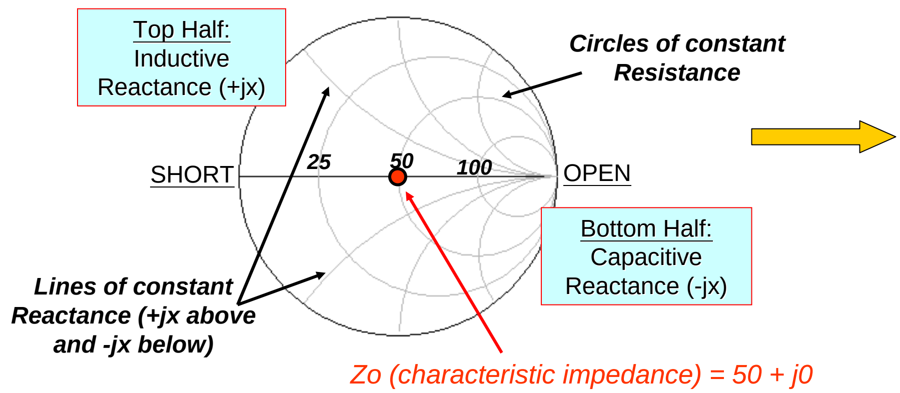
These are easier to understand and simply plotted.

Results of an S-Parameter Simulation in ADS

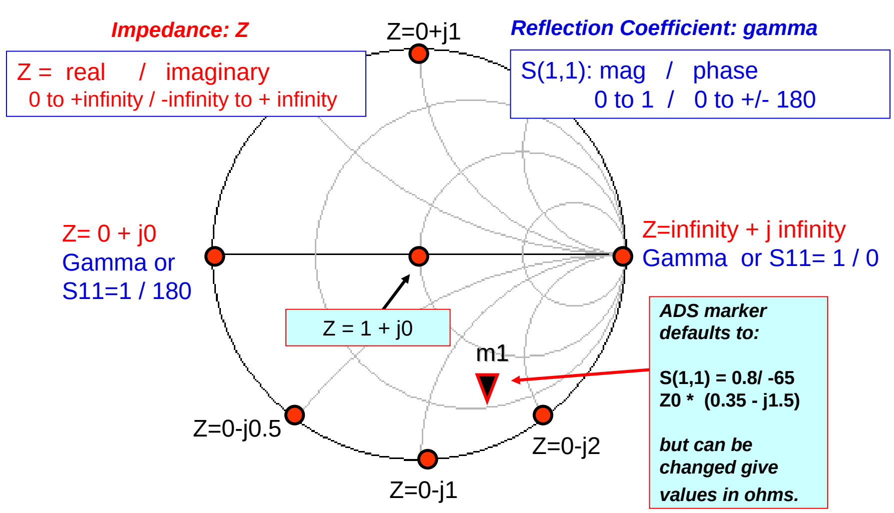
- Read the complex reflection coefficient (Gamma)
- Change the marker readout for Zo
- Read all four S-parameters
- Smith chart plot: use for impedance matching (S11 and S22)
- Similar to Network Analyzer measurements

The Impedance Smith Chart simplified...

This is an impedance chart transformed from rectangular Z.
Normalized to 50 ohms, the center = R50+J0 or Zo (perfect match).
For S11 or S22 (two-port), you get the **complex impedance**.



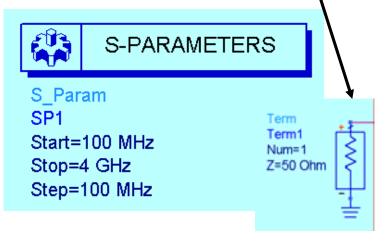
The Smith chart in ADS Data Display

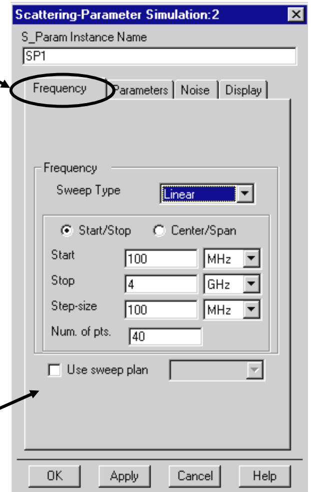


S-Parameter Simulation Controller

Default sweep = ***Freq***

The simulator requires port termination: Num = 1

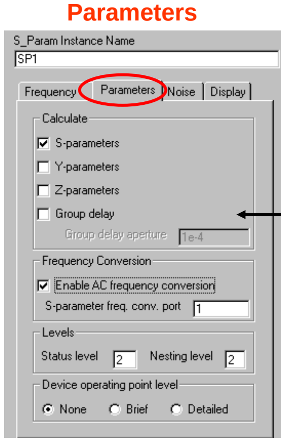




Sweep plan can also be used (see next slide). Either way, simulation data (S matrix) will be for the specified range and points.

Other S-Parameter controller tabs

Parameters

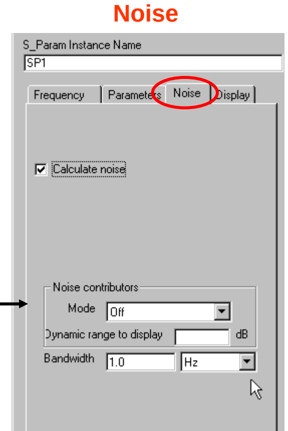


Calculate other parameters.

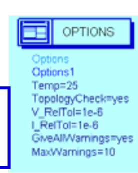
Enable Frequency Conversion for a mixer.

Calculate SS noise - same as in the AC simulation.

Noise

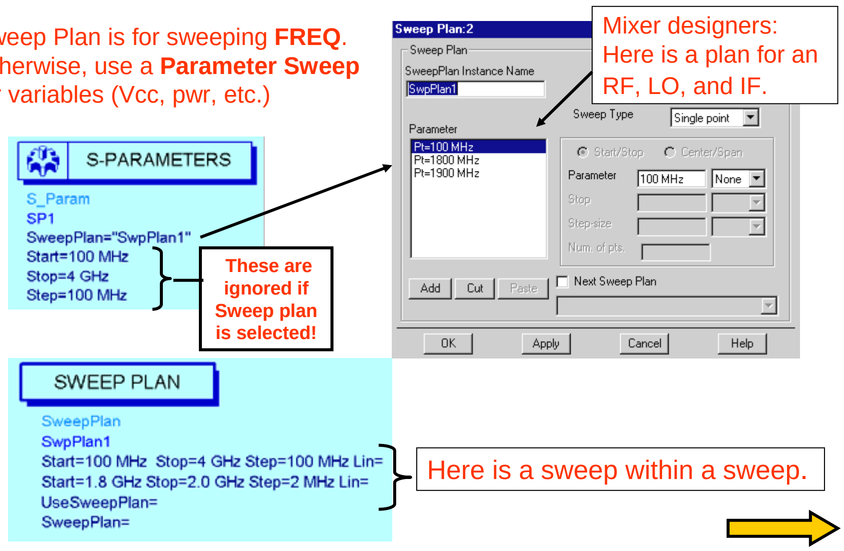


NOTE: Insert the Options controller for a noise simulation = no error message on temperature.



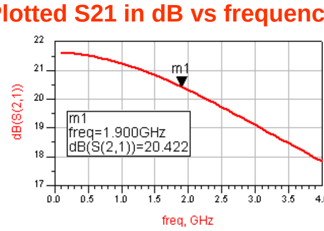
Sweep Plan with S-parameter simulations

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

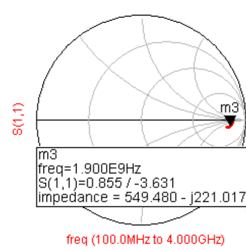


Typical S-parameter plots: ADS data display

Plotted S21 in dB vs frequency



Plotted S11 on a Smith Chart: note marker readout.

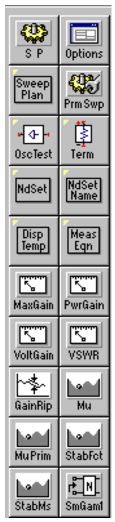


Note marker readout is x50.

Complete S-matrix with port impedance

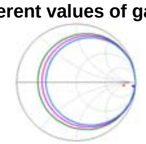
freq	S				freq	PortZ	
	S(1,1)	S(1,2)	S(2,1)	S(2,2)		PortZ(1)	PortZ(2)
100.0M...	0.879 / ...	2.392E-...	12.036 /...	0.994 / ...	100.0M...	50.000 /...	50.000 /...
200.0M...	0.879 / ...	7.978E-...	12.020 /...	0.993 / ...	200.0M...	50.000 /...	50.000 /...
300.0M...	0.878 / ...	1.729E-...	11.994 /...	0.992 / ...	300.0M...	50.000 /...	50.000 /...
400.0M...	0.877 / ...	2.556E-...	10.019 /...	0.994 / ...	400.0M...	50.000 /...	35.000 /...
500.0M...	0.877 / ...	4.007E-...	9.982 / ...	0.993 / ...	500.0M...	50.000 /...	35.000 /...
600.0M...	0.876 / ...	5.818E-...	9.939 / ...	0.991 / ...	600.0M...	50.000 /...	35.000 /...
700.0M...	0.875 / ...	8.013E-...	9.888 / ...	0.990 / ...	700.0M...	50.000 /...	35.000 /...

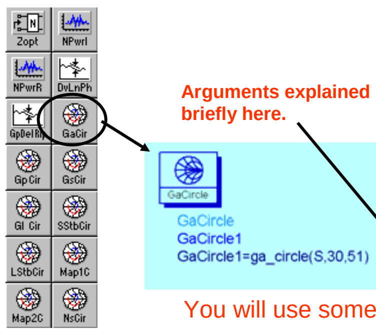
S-Parameter measurement equations



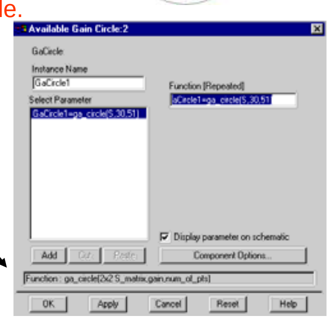
All simulation palettes have specific measurement equations. You insert the measurement equation and set the arguments if necessary. Here, S is the matrix, 30 is the value in dB, and 51 points used to draw the circle.

Example: 3 circles for 3 different values of gain.





Arguments explained briefly here.

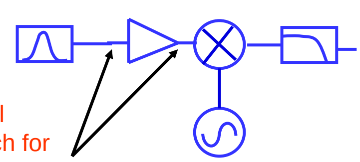


You will use some of these in the labs...

Creating Matching Networks

- Various topologies can be used: L, C, R
- Avoid unwanted oscillations (L-C series/parallel)
- Yield can be a factor in topology (sensitivity)
- Use the fewest components (cost + efficient)
- Sweep or tune component values to see S-parameters
- Optimization: use to meet S-parameter specs (goals)

In the lab, you will optimize the match for the amplifier.



NOTE: For a mixer, match S11 @ RF and S22 @ IF.

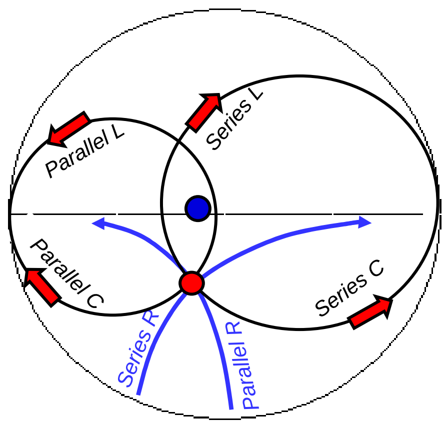
Matching means:

Moving toward the center of the Smith Chart!

Add Series or Parallel (shunt) components.

You will do this in the lab.

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



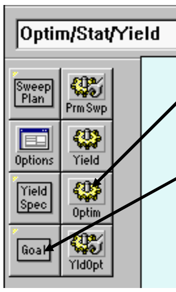
ADS Optimization Basics

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:
  - A search method.
  - A specific goal or specification to be met.
  - Enabled components or parameters to be adjusted.

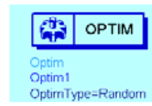
NOTE: **ADS** has both continuous and discrete optimization. Yield analysis or a yield optimization is also available.

Optimization palette: Controller and Goals

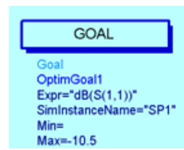


Four steps for optimization:

Optim *controller: set the type, etc.*



Goal *statement: use valid measurement equation or dataset expression.*



Enable *component (opt).*

Simulation *controller.*

Type of Optimization:
In general, we recommend using Random first, then using Gradient. Also, first tune and then optimize.

Many Types are Available

Random & Gradient are often used together…

Optimizer	Description
Random	Random search method with least-squares error function
Random Minimax	Random search method with minimax error function
Gradient	Gradient search method with least-squares error function
Gradient Minimax	Gradient search method with minimax error function
Quasi-Newton	Quasi-Newton search method with least-squares error function
Least P^{th}	Quasi-Newton search method with least Pth error function
Minimax	Two-stage, Guass-Newton/Quasi-Newton method with minimax error function
Random Max	Random search method with procedure to internally negate the error functions to get error function maximization (worst case analysis)
Discrete	Discrete optimization, provided there is at least one discrete valued optimization parameter in the design.
Genetic	Direct search method using evolving parameter sets

NOTE: See manual for details (***minimax*** function works well for filters).

Goals and Error Function

- The goals are minimum or maximum target values.
- The error function is based on the goal(s).
- The weighting factor prioritizes multiple goals.

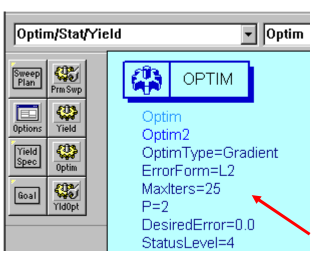
Error function is defined as a summation of residuals.
A residual r_i may be defined as:

$$r_i = W_i \mid m_i - s_i \mid$$

s_i is the simulated ith response (example: S21= 9.5dB)
m_i is the desired response for the **i**th measurement (example: S21=10dB)
W_i is the weighting factor for multiple goals: higher number is greater.

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

More about the Error Function...



Least Squares: Each residual is squared and all terms are then summed. The sum of the squares is averaged over frequency.

Choice of optimizer determines these settings.

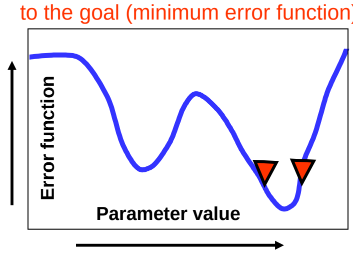
Least Pth: The Least Pth error function formulation is similar to the previous one, except that instead of squaring the residuals, it raises them to the Pth power with P=2 , 4 ,6 etc.

Minimax: attempts to minimize the largest of the residuals. This tends to result in equal ripple responses .

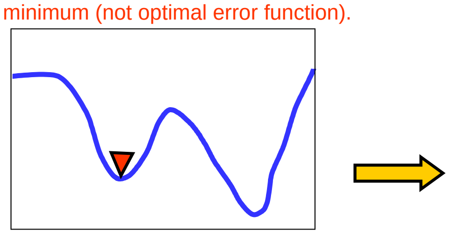
Worst case: minimizes the reciprocal of the least squares error function. This has the effect of maximizing the error function. The goal is to find a worst typical response for a given set of parameters.

Using a combined approach...

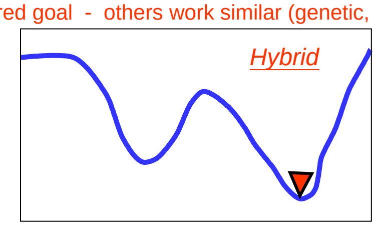
Random analysis often gets you close to the goal (minimum error function).



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



Using both RANDOM and GRADIENT can reach the desired goal - others work similar (genetic, etc.).

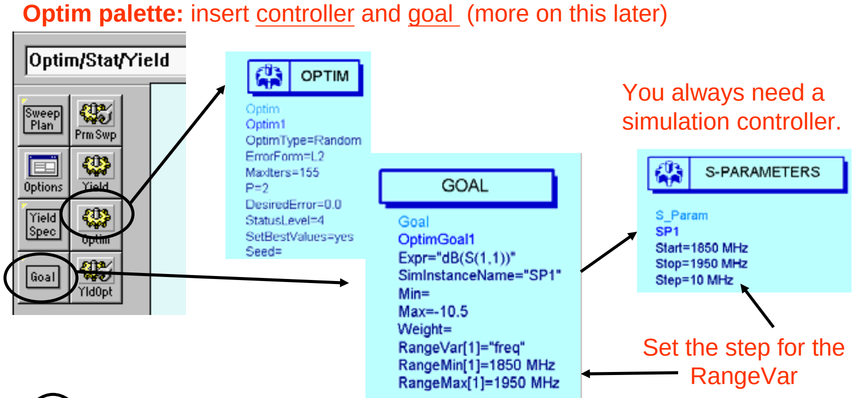


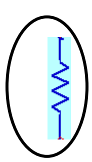
NOTE: Random is not totally random. It uses an adaptation that helps it move closer to the goal.

You will do this in the lab!

ADS Optimization Setup

Optim palette: insert controller and goal (more on this later)

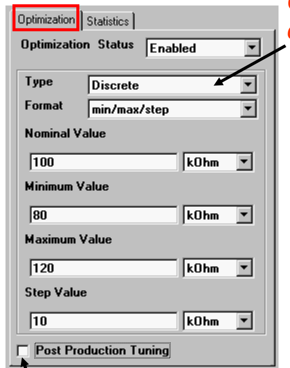




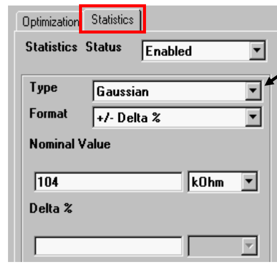
Edit the component, then click  to **enable** for optimization.

Details on next slide:

Enabling components for opt or stats (yield)



Once enabled, you can specify a continuous or discrete (stepped) variation..



Gaussian, Uniform, or discrete. Results will be viewable in the data display.

NOTE: If discrete values are not realistic, use file based: DAC

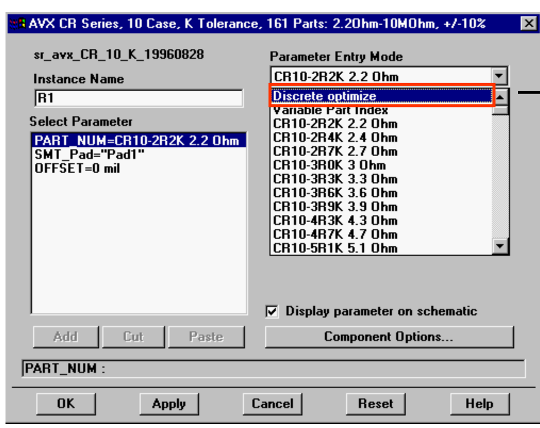
PPT is an optimization within a Yield Analysis only. Allows value to be shifted to achieve goal.

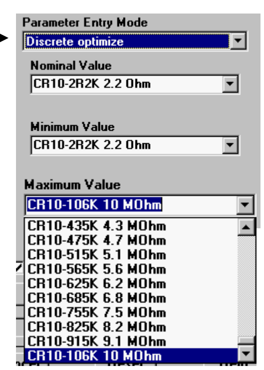
noopt = disabled (after optimization):

R=96.527 kOhm noopt{ discrete 80 kOhm to 100 kOhm by 10 kOhm }

Discrete Optimization for Library Parts

Inserted library part with listed range of values (like a DAC)

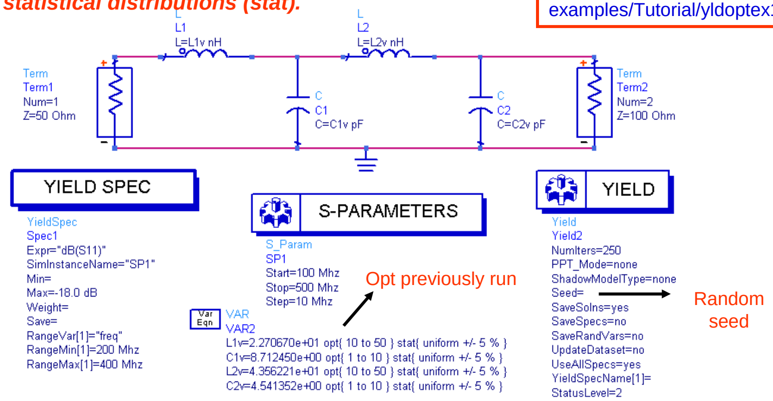




Yield Analysis: % meeting specs!

Example: 200-400 MHz (50-to-100 ohm) Impedance Transformer. Simulation tests the % of circuits to meet spec using component values with defined statistical distributions (stat).

NOTE: Optimize yield results by changing the nominal values until you get maximum (near 100%). See: examples/Tutorial/yldoptex1_prj



As an extra exercise: **Step 1- Copy: *examples / Tutorial / yldex1_prj***

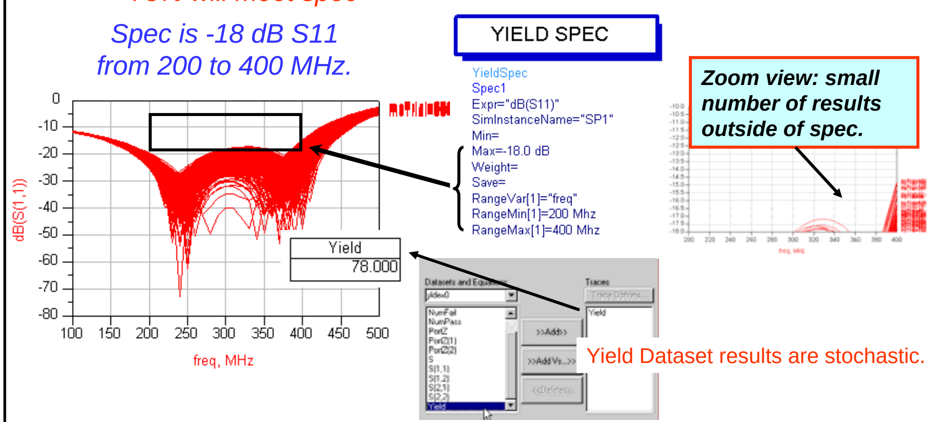
Yield Analysis Results (data)

Step 2 - Run the simulation once. Then change the specs and resimulate. Or, try it on your own circuit!

Refer to this and the previous slide to setup your own Yield analysis after the class.

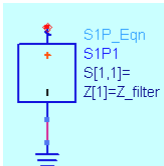
78% will meet spec

Spec is -18 dB S11 from 200 to 400 MHz.



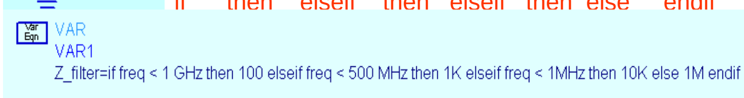
Additional information:
frequency sensitive components!

Z-PORT: equation describes changing Z with changing freq

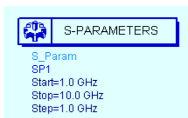


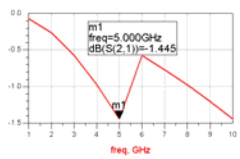
Note the ADS syntax for Z increasing as freq (global variable in ADS) decreases:

if then elseif then elseif then else endif



CAPQ and INDQ: equation describes changing L or C with freq

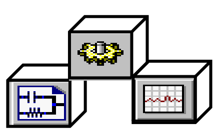




What the lab is about ...

Lab 5:

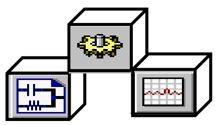
S-parameter Simulation and Optimization



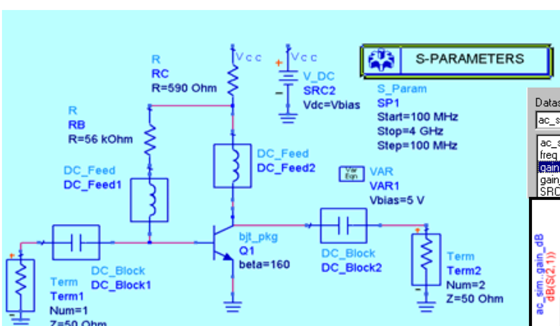
Steps in the Design Process

You are here:

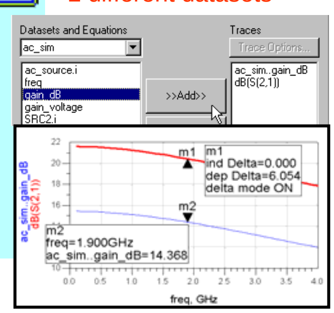
- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- ***Simulate amp S-parameter response***
- ***Define amp matching topology and tune input***
- ***Optimize the amp in & out matching networks***
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*



First Step: Simulate with ideal components



2 different datasets



•Plot the data and compare to ac_sim data.
•Change Term Z and list the S matrix.

freq	S				freq	PortZ	
	S(1,1)	S(1,2)	S(2,1)	S(2,2)		PortZ(1)	PortZ(2)
100.0M...	0.879 / ...	2.392E-...	12.036 /...	0.994 / ...	100.0M...	50.000 /...	50.000 /...
200.0M...	0.879 / ...	7.978E-...	12.020 /...	0.993 / ...	200.0M...	50.000 /...	50.000 /...
300.0M...	0.878 / ...	1.729E-...	11.994 /...	0.992 / ...	300.0M...	50.000 /...	50.000 /...
400.0M...	0.877 / ...	2.556E-...	10.019 /...	0.994 / ...	400.0M...	50.000 /...	35.000 /...
500.0M...	0.877 / ...	4.007E-...	9.982 / ...	0.993 / ...	500.0M...	50.000 /...	35.000 /...
600.0M...	0.876 / ...	5.818E-...	9.939 / ...	0.991 / ...	600.0M...	50.000 /...	35.000 /...
700.0M...	0.875 / ...	8.013E-...	9.888 / ...	0.990 / ...	700.0M...	50.000 /...	35.000 /...

Calculate C and L values and re-simulate

Reactance of 10 pF at 1.9 GHz and a list of L values:

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

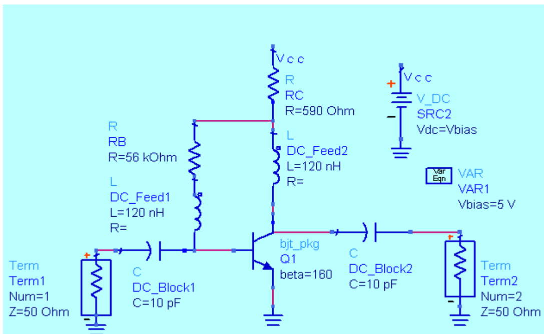
Cap value: reactance at 1900 MHz

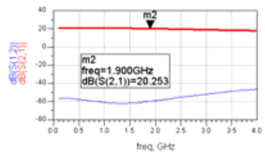
Xc
-8.377

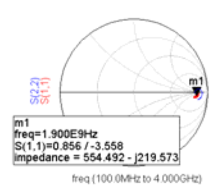
Eqn XL= 2 * pi * 1900M * L_val

Eqn L_val = [1n :: 10n :: 200n]

L_val	XL
7.100E-8	847.602
8.100E-8	966.982
9.100E-8	1086.363
1.010E-7	1205.743
1.110E-7	1325.124
1.210E-7	1444.504
1.310E-7	1563.885

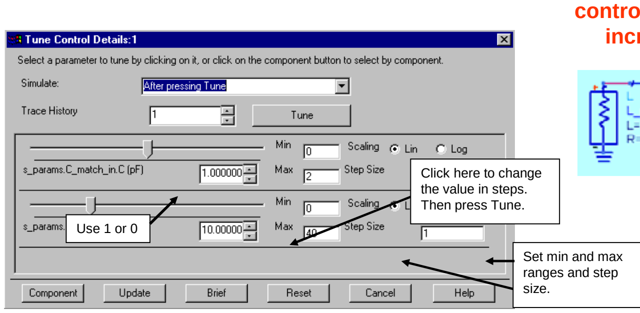


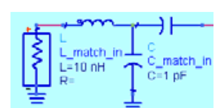


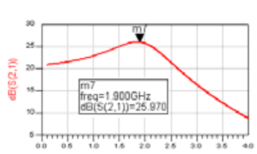


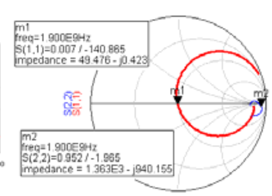
Tune the input match

Use detailed Tune control with small increments!

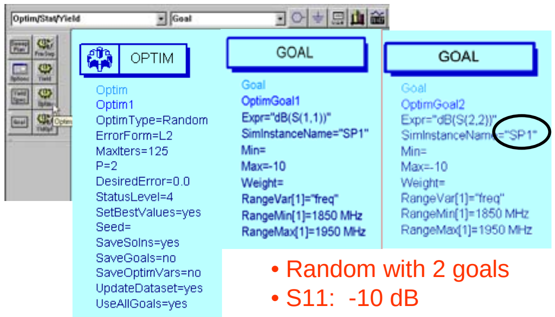


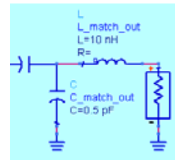




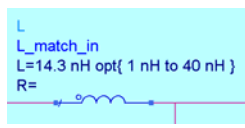


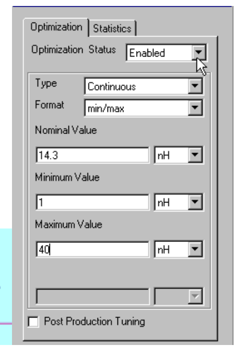
Optimize the output & input match





- Random with 2 goals
- S11: -10 dB
- S22: -10 dB

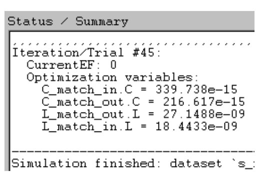


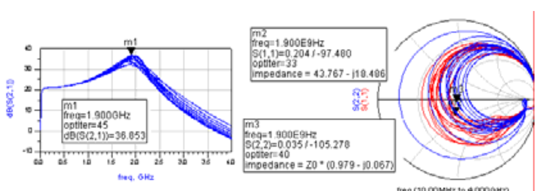


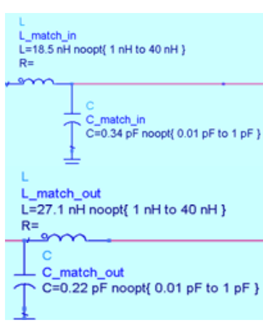
REVIEW of 4 steps:

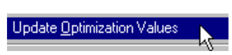
- ***Set up the OPTIM controller***
- ***Set up the GOALS***
- ***Enable the components***
- ***Setup the simulation***

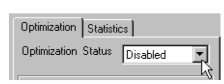
Display opt data, update and simulate

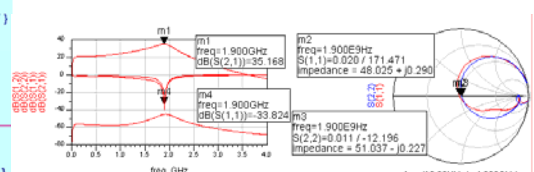








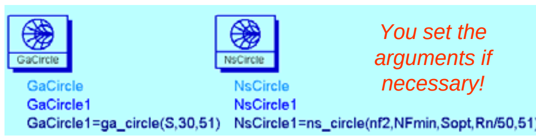


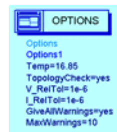


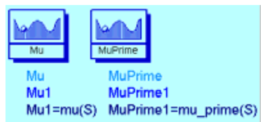
Final values look good!

Next, use built-in measurements

S-parameter simulation with gain and noise circles, and stability.
These are built-in measurement equations from the simulation palette.

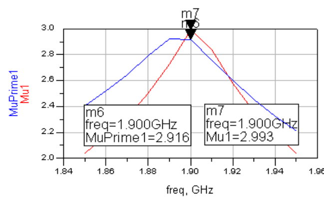


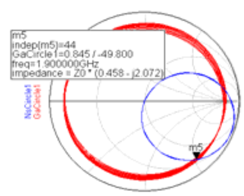




NOTE: Listed results from noise turned on in the simulation controller.

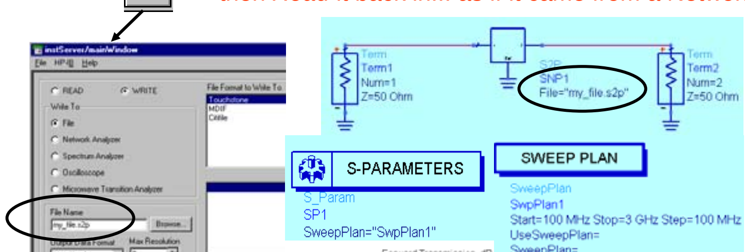
freq	NFmin	nf(2)	Sopt
1.900GHz	1.075	3.264	0.793 / -20.468

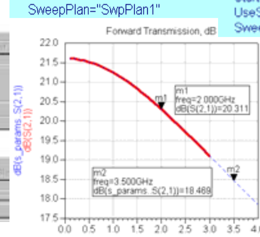




Last step: Read / Write data files

Write an ADS “S” dataset as a Touchstone file,
then *Read* it back in... as if it came from a Network Analyzer!

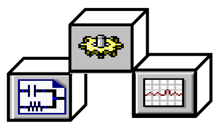




Use a Sweep Plan and compare the data.

Topic 6:

E-Syn, Momentum, Transient and the DAC

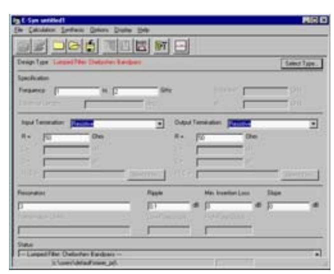


Using E-Syn

Quick S-parameter simulation

What does E-Syn do?
It makes it easy to create FILTERS and Matching Networks.

E-Syn user interface is a little different than the ADS interface. Also, you could use a Filter Design Guide instead of E-Syn. This may appear in future releases of ADS!



• You specify the TYPE of design, the RESPONSE, and the BAND.

• SYNTHESIZE the design and you get a selection of topologies and values.

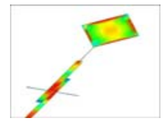
• ANALYZE the design and plot the response in the ADS data display. Optimize if desired.

Using **Momentum**

MOM engine gives S-parameter results

What is Momentum? ***E-M (electro-magnetic) solver using Method of Moments technique and Green's functions to compute the current in planar structures, including vias and the coupling between surfaces.***

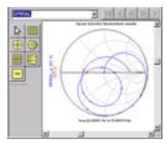
Antenna patterns!

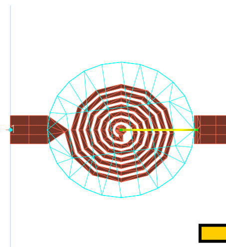


Why use Momentum?
- You have no accurate model for a passive layout.
- You want to know the coupling effects between structures.
- You want to optimize the layout real-estate, performance, etc.
- Your other structure simulator takes too long to simulate!
- You want to use the results in ADS simulations.

Example spiral meshed as a "strip" geometry.

Hole in ground plane is meshed as a "slot", which is more efficient than meshing the entire ground plane.





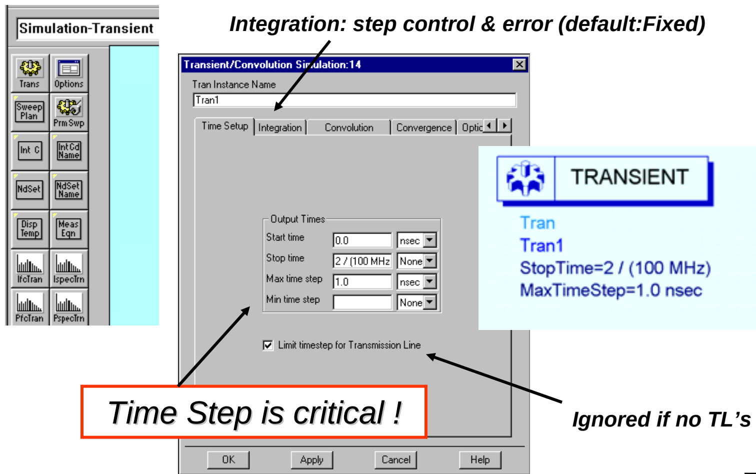
Transient simulation

- *Analysis performed in the Time Domain*
- *Use any Source*
- *Solutions use Newton_Raphson iterations*
- *You get Amplitude vs. Time*
- *Time Domain data can be transformed: FS*

NOTE on Convolution:
Frequency domain models (microstrip) can be brought into the time domain and converted to the time domain - then convolved with the time-domain input signal to obtain the time-domain output signal. The convolution tab in the transient simulator allows you to define methods and settings.

Transient simulation controller

Integration: step control & error (default:Fixed)

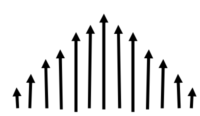


Time Step is critical !

Ignored if no TL's

Setting the Transient Time Step

Start time: sampling begins
Stop time: sampling ends
Time step: sampling rate



Sample @ 2 x BW

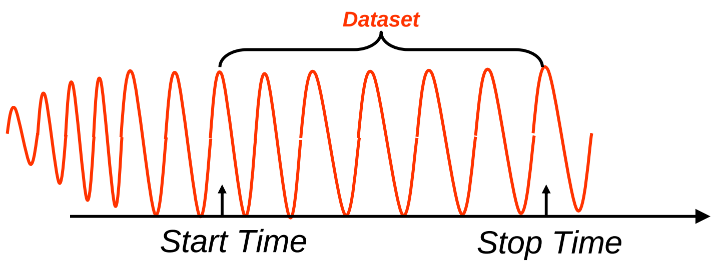
Use the Nyquist rule: Sample at 2 x or more the rate of the highest frequency of interest:

To sample the fundamental (1900 MHz) plus harmonics, you must calculate @ 2 x (rate of highest harmonic desired).

1 / (2 x 15 x 1900MHz) = 17.54 picoseconds.

Setting the Transient Stop Time

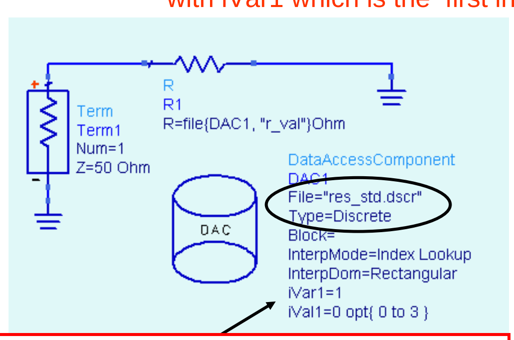
For many circuits: stop time should allow for ***periodic - settling****.*

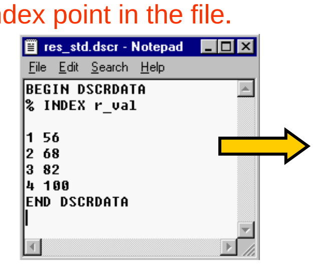


NOTE: Transient analysis can be tricky. Sampling before a circuit reaches steady state will not give correct results when transformed into the frequency domain. Also, you must use a time step that is a multiple of the frequencies of interest or the results will not be correct.

Using a DAC (data access component)

For optimization, R1 is assigned to a file which is read by the DAC. "res_std.dscr" contains an index and a list of values (r_val). In the DAC, variable iVal1 is enabled over the range of indexed values beginning with iVar1 which is the first index point in the file.

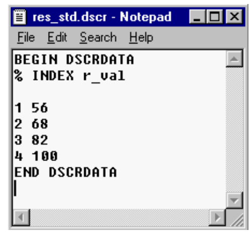




NOTE: iVar1 always = 1 (index at first column).
However, iVal1 always starts at 0 (same as ADS data).

The file must be in the DATA directory and must be in correct format. The circuit simulation manual has information on file formats such as DSCR..

DAC Optimization: setup and results




```
BEGIN DSCRDATA
% INDEX r_val

1 56
2 68
3 82
4 100
END DSCRDATA
```

freq	my_dbs11	iVal1
1.000GHz	-24.943	0.000

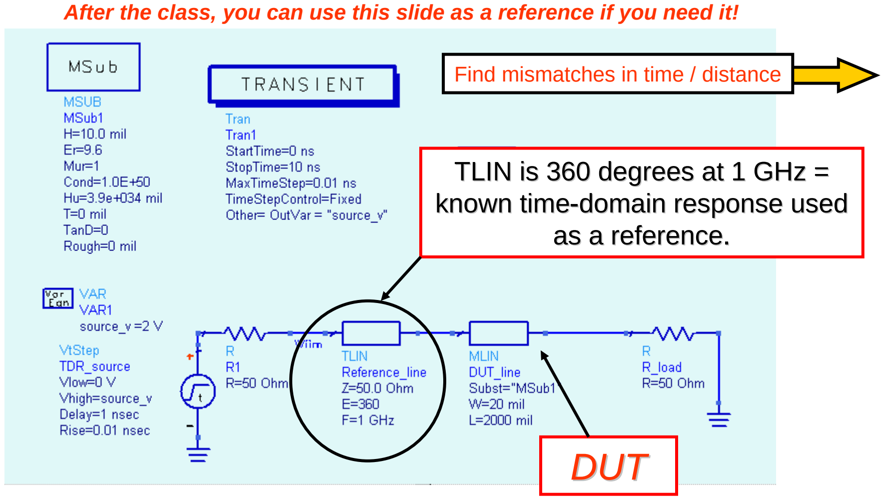
Save values = no. Only the MeasEqn is sent to the dataset with the final iVal1 which is 0 = 56 ohms which gives the least reflection. Also, click Simulate > Update Optimization Values to update the iVal in the DAC.

DataAccessComponent
DAC1
File="res_std.dscr"
Type=Discrete
Block=
InterpMode=Index Lookup
InterpDom=Rectangular
iVar1=1
iVal1=0.000000e+000 opt{ discrete 0 to 3 by 1 }

EXTRA NOTE: TDR Setup

REFERENCE SLIDE ONLY

After the class, you can use this slide as a reference if you need it!



EXTRA NOTE: TDR Data Display

REFERENCE SLIDE ONLY

Eqn Er = 9.6

Eqn velocity = 3e8 / sqrt(Er)

Eqn distance_mtrs = (velocity * (indep(m2) - indep(m1))) / 2

Eqn distance_mils= distance_mtrs * (1e5/2.54)

Eqn bw=0.35/0.1e-9

bw
3.500E9

After the class, you can use this slide as a reference if you need it!

Eqn z_line = Zo * ((1 + rho) / (1 - rho))

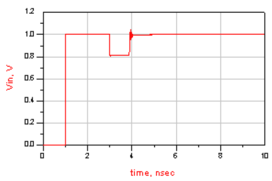
Eqn Zo = 50

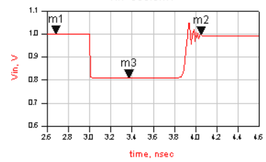
Eqn rho = (m3 - ref_v) / (ref_v)

Eqn ref_v = source_v [1]/ 2

rho	z_line	ref_v
-0.192	33.916	1.000

distance_mtrs	distance_mils
0.066	2611.214





Results:
mismatch
delay time
rho
BW
VSWR
mtrs to mils

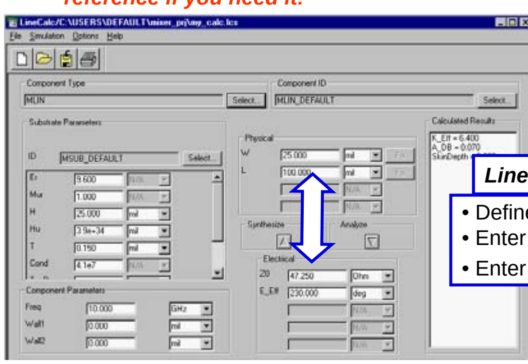
Eqn VSWR = (1 + mag (rho)) / (1 - mag (rho))

VSWR
1.474

EXTRA NOTE on LineCalc and Model Composer

After the class, you can use this slide as a reference if you need it!

REFERENCE SLIDE ONLY



LineCalc is still available!

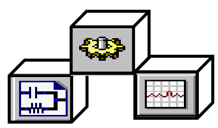
- Define the substrate
- Enter L & W to get electrical
- Enter electrical to get L & W

ALSO - New add-on for ADS 1.5: MODEL COMPOSER. Generate your own passive library models (parameterized) for simulation. You get circuit simulation speed with EM simulation accuracy (Momentum) for microstrip components - more types in the future!

What the lab is about ...

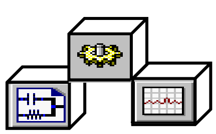
Lab 6:

Filters: E-Syn, Momentum,
Transient Simulation
and the *DAC*



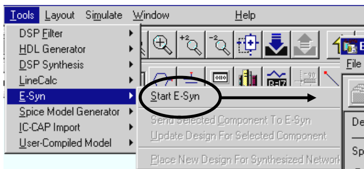
Steps in the Design Process

You are here:



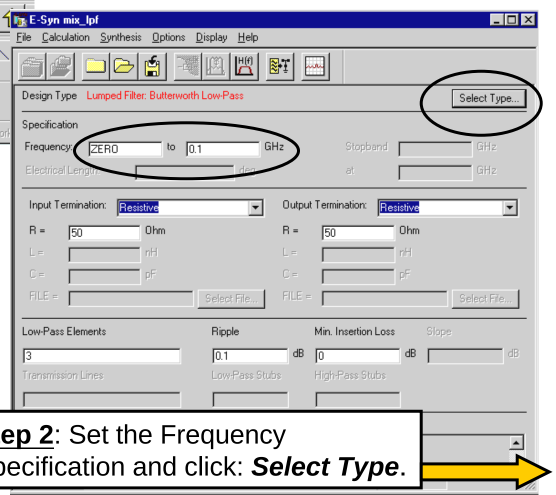
- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- ***Filter design – lumped 200MHz LPF - use E-Syn***
- ***Filter design – microstrip 1900 MHz BPF***
- ***Transient and Momentum filter analysis***
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*

E-Syn - 200 MHz LPF design



Step 1:
Start E-Syn in the ADS schematic window:

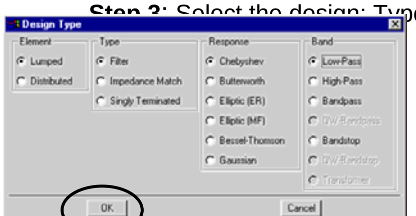
Tools > Start E-Syn



Step 2: Set the Frequency Specification and click: ***Select Type***.

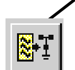
E-Syn - 200 MHz LPF (continued)

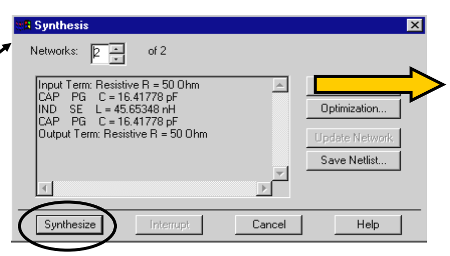
Step 3: Select the design: Type, Response, and Band.



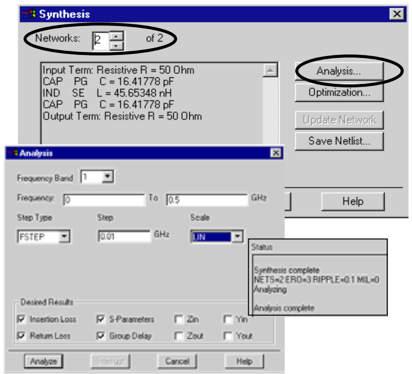
At first, window will be blank. Therefore, click the ***Synthesize*** button to calculate topology & component values as shown here.

Step 4: Go back to E-Syn main menu and click the ***Synthesis*** icon:



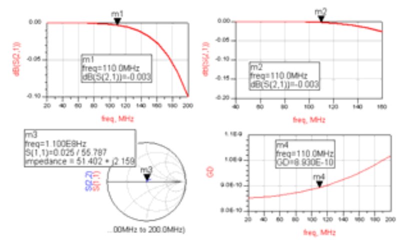


***E-Syn -** 200 MHz LPF (continued)*

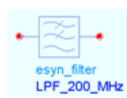


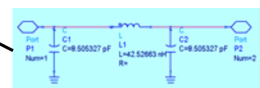
Step 5: Select the Network you want to use and then click ***Analysis***.

Step 6: Set up the simulation and ***Analyze***. The results will appear in the ADS Data Display window.



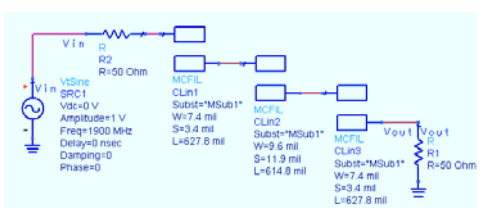
Result of design (Butterworth) to be used in Final simulation.

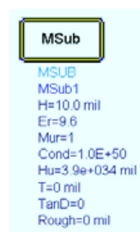


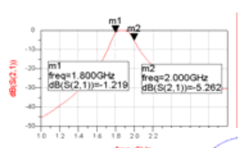


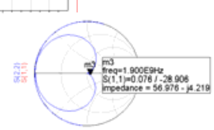
Transient simulation of 1900 MHz BPF

Microstrip coupled line filter with substrate and VtSine source.







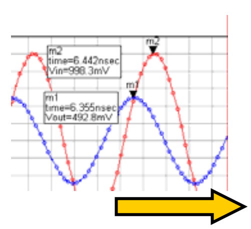


Eqn marker_val = indep (m2) - indep (m1)

Eqn marker_freq = 1 / marker_val

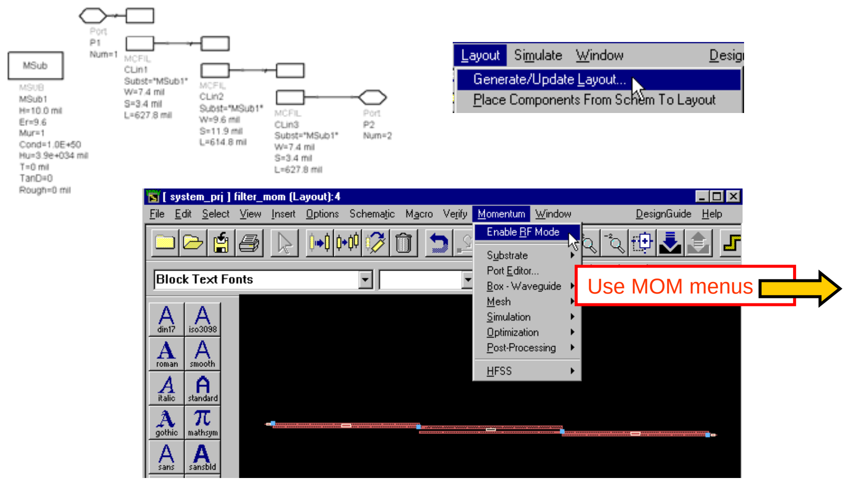
Also calculate delay:

marker_val
8.772E-11

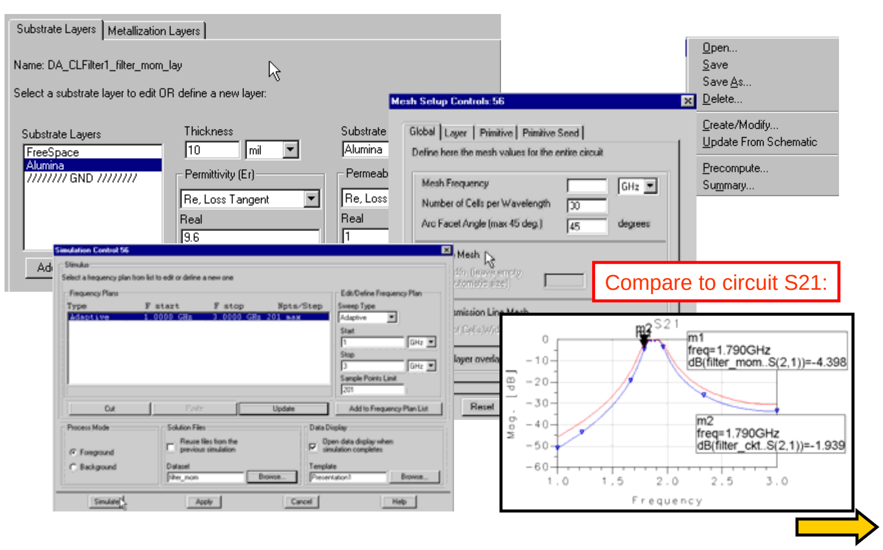


Generate the Layout of 1900 MHz BPF

Remove controllers and Terms - insert Port connectors



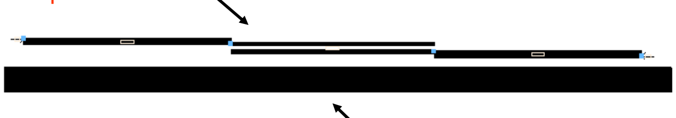
MOM: substrate, mesh, simulation



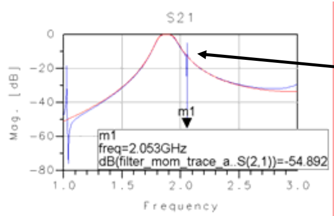
MOM - calculate coupling effects

Use the same setup and add a rectangle:

Coupled Line Filter

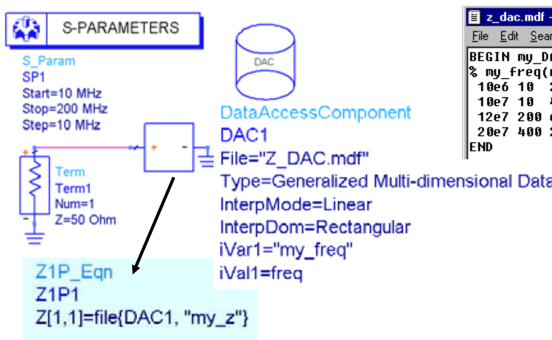


...draw a trace in close proximity

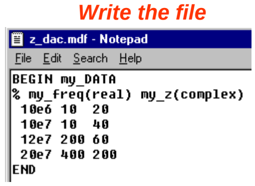


Momentum simulation shows resonance from coupling.

OPTIONAL - DAC exercise



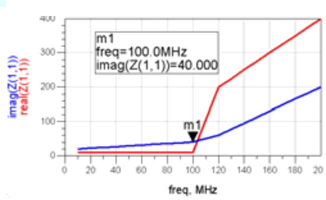
Write the file


z_dac.mdf - Notepad

File Edit Search Help

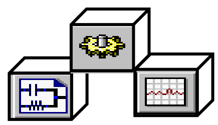
```
BEGIN my_DATA
% my_freq(real) my_z(complex)
 10e6 10  20
 10e7 10  40
 12e7 200 60
 20e7 400 200
END
```

After setting S-parameter controller to calculate Z, results show results.



Topic 7:

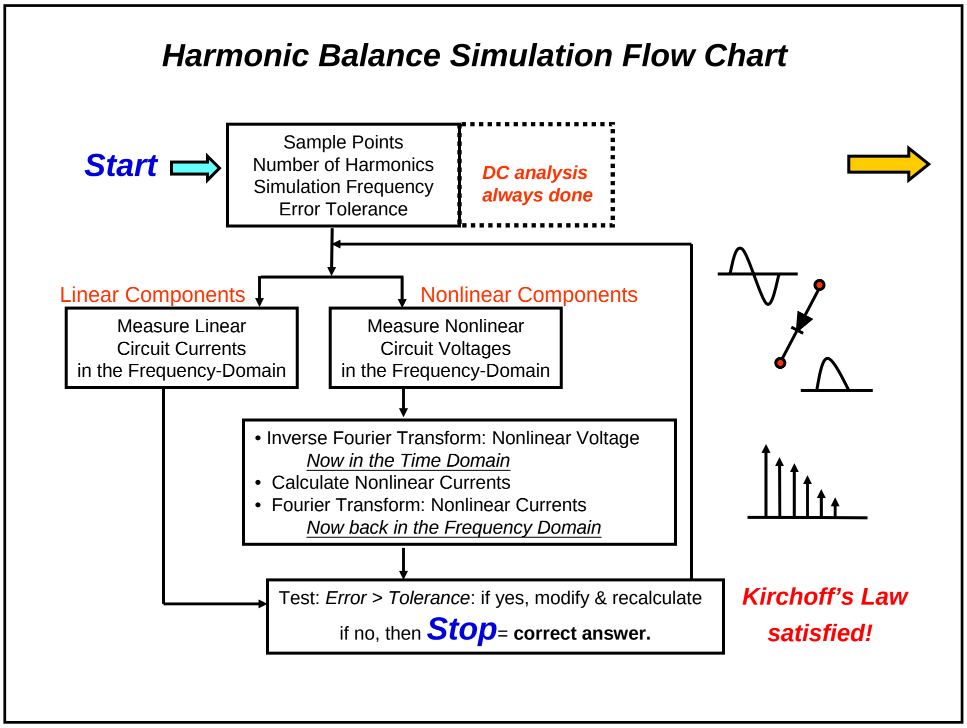
Harmonic Balance

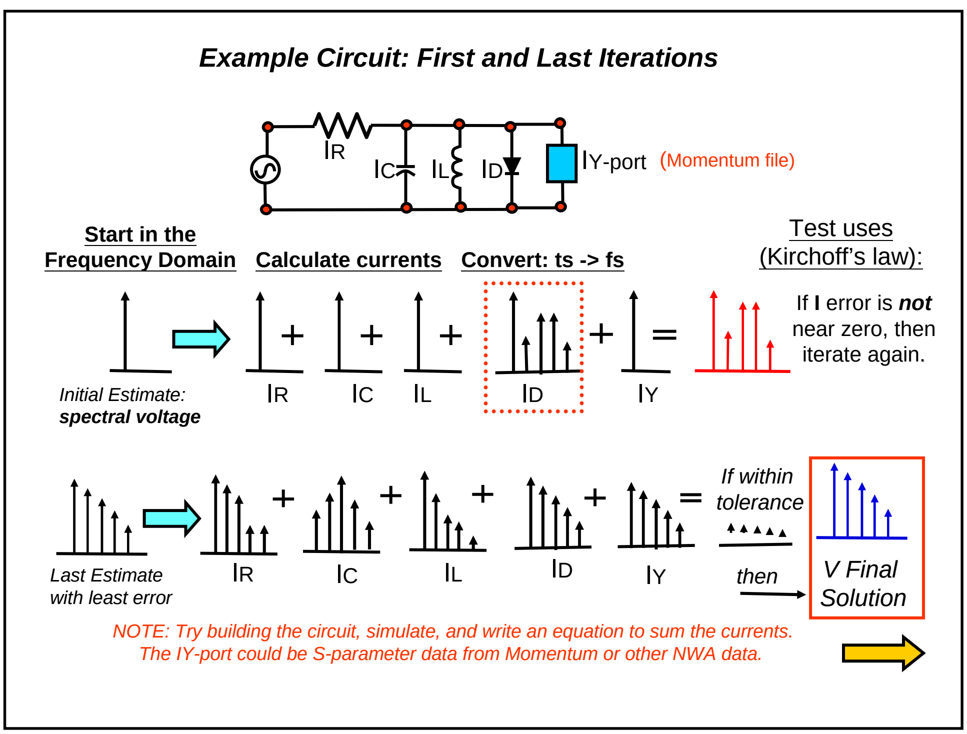


Harmonic Balance Simulation

Analyze circuits with Linear and Non-linear components:

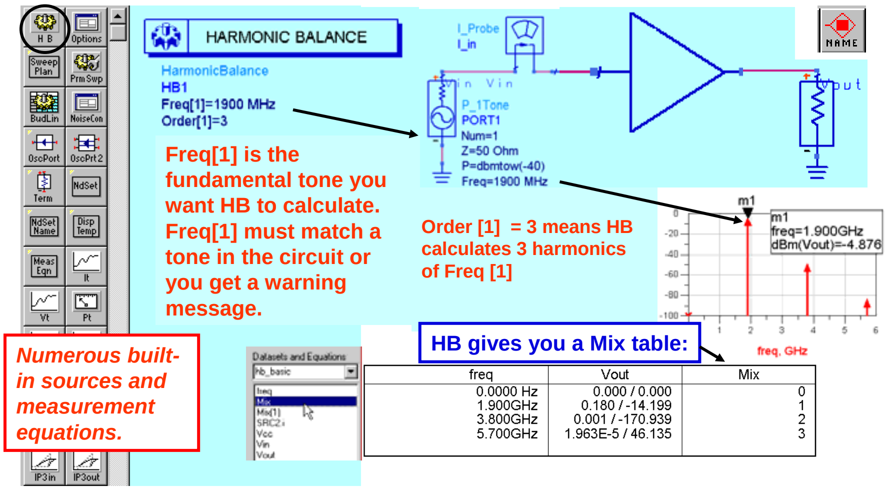
- You define the tones, harmonics, and power levels
- You get the spectrum: Amplitude vs. Frequency
- Data can be transformed to time domain (ts function)
- Solutions use Newton-Raphson technique
- Krylov subspace method also available (large circuits)
- Use only Frequency domain sources
- Similar to Spectrum Analyzer


Harmonic Balance Simulation Flow Chart
Start
Sample Points
Number of Harmonics
Simulation Frequency
Error Tolerance
DC analysis always done
Linear Components
Nonlinear Components
Measure Linear Circuit Currents in the Frequency-Domain
Measure Nonlinear Circuit Voltages in the Frequency-Domain
• Inverse Fourier Transform: Nonlinear Voltage
Now in the Time Domain
• Calculate Nonlinear Currents
• Fourier Transform: Nonlinear Currents
Now back in the Frequency Domain
Test: Error > Tolerance: if yes, modify & recalculate
if no, then Stop = correct answer.
Kirchoff's Law satisfied!


Example Circuit: First and Last Iterations
IR
IC
IL
ID
IY-port
(Momentum file)
Start in the Frequency Domain
Calculate currents
Convert: ts -> fs
Test uses (Kirchoff's law):
Initial Estimate: spectral voltage
IR
IC
IL
ID
IY
If I error is not near zero, then iterate again.
Last Estimate with least error
IR
IC
IL
ID
IY
If within tolerance
then
V Final Solution
NOTE: Try building the circuit, simulate, and write an equation to sum the currents.
The IY-port could be S-parameter data from Momentum or other NWA data.

Basic 1 Tone HB simulation setup

Basic HB controller and source setup gives you spectral tones:

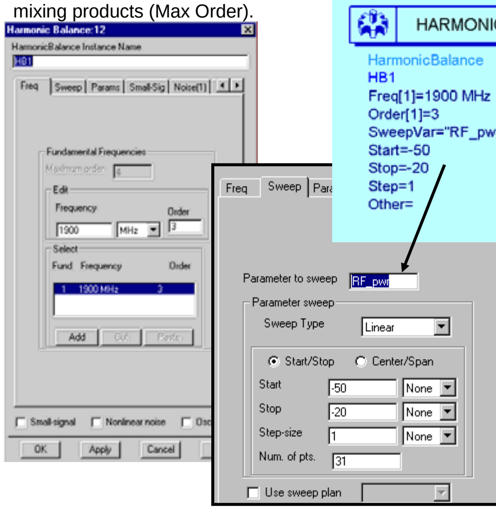


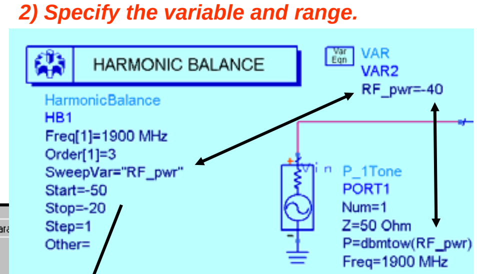
freq	Vout	Mix
0.0000 Hz	0.000 / 0.000	0
1.900GHz	0.180 / -14.199	1
3.800GHz	0.001 / -170.939	2
5.700GHz	1.963E-5 / 46.135	3

Swept variables in Harmonic Balance

HB Freq tab: specify tones (Freq), harmonics (Order), and mixing products (Max Order).

1) Initialize the VAR to sweep.
2) Specify the variable and range.

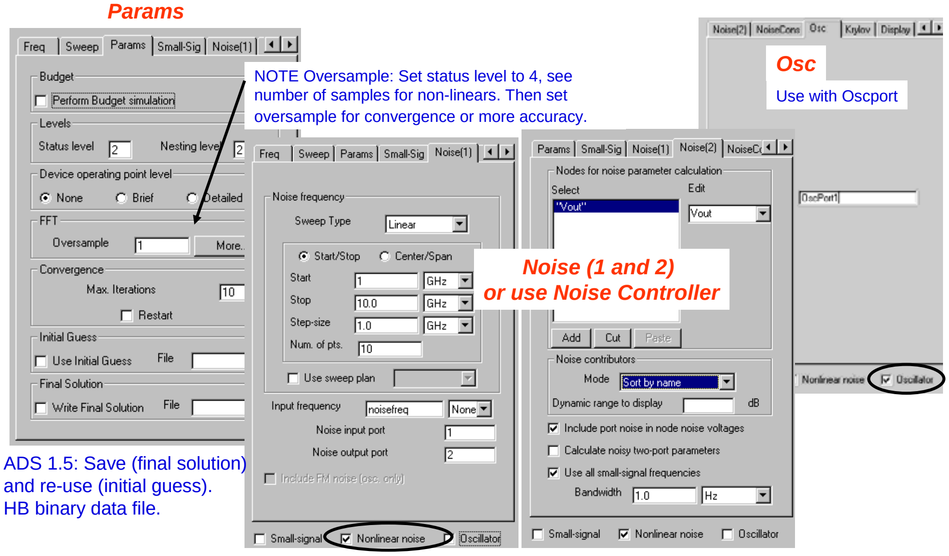




3) Be sure the VAR, the source, and simulation controller all have the same information.

NOTE: Swept variables always go to the dataset.

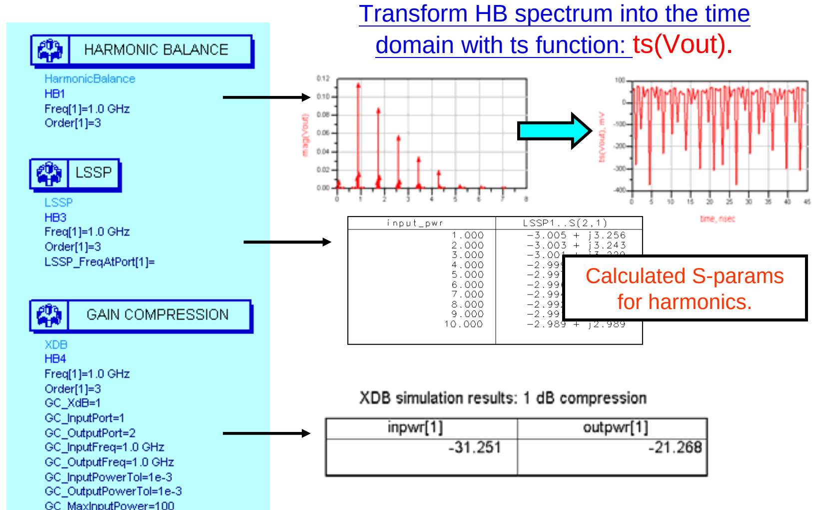
Other settings (tabs) in Harmonic Balance



ADS 1.5: Save (final solution) and re-use (initial guess). HB binary data file.

Related harmonic balance controllers ...

Transform HB spectrum into the time domain with ts function: ts(Vout).

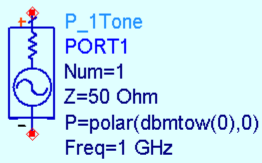


You will use HB and XDB in the lab!

Types of Power Sources for HB

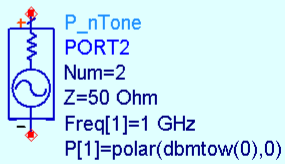
Default power function for these sources is **polar**,

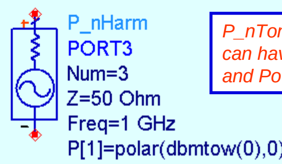
but you can simplify it on the screen as: dmbtow(0)
Therefore, dbmtow(0) *is the same as* polar(dbmtow(0),0)



Notice that these sources are also **ports** (OK for S-param analysis).

Also, they can be considered ***noiseless*** like sources in a measurement system.





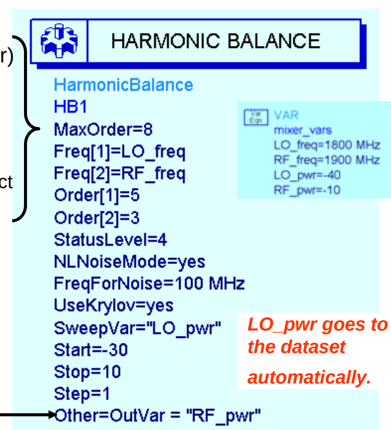
P_nTone and P_nHarm can have multiple Freqs and Power.

Example: HB simulation setup for mixer with swept LO power

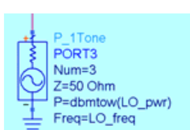
Mixer example:

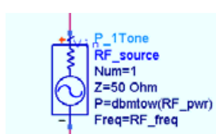
Freq [1] fundamental tone (most power: LO for mixer)
Freq [2] fundamental tone (RF for mixer)
Order [1] number of harmonics for Freq [1]: LO.
Order [2] number of harmonics for Freq [2]: RF.
MaxOrder = mixing products, depends on Order[n].
NOTE: Here if MaxOrder = 9 , you won't get 9th order product because Order[1] and [2] only go up to the 8th order.

Other settings: for status info, noise, and Krylov simulator (used in lab).



LO_pwr goes to the dataset automatically.





Do not this: Freq = LO_freq MHz or MHz units will multiply.

To get any other VAR to the dataset (RF_pwr) requires using this syntax:

OutVar ="variable name"

data

Example data: use mix function on *Mix* table

DC term=0. Freq, harmonics [order], and products [max order] are indexed:

***Mixer example**: Max order=8*

LO order=5 RF order=3

freq	Mix(1) (LO)	Mix(2) (RF)
0.0000 Hz	0	0
100.0MHz	-1	1
200.0MHz	-2	2
300.0MHz	-3	3
1.500GHz	4	-3
1.600GHz	3	-2
1.700GHz	2	-1
1.800GHz	1	0
1.900GHz	0	1
2.000GHz	-1	2
2.100GHz	-2	3
3.300GHz	5	-3

LO: Freq[1]=1800 MHz RF: Freq[2]=1900 MHz

To get dBm of IF (100 MHz) at Vout, use the *mix* function:

MeasEqn
IF_100MHz_output
dbm_out=dBm (mix(Vout,{-1,1}))

Arguments in parentheses ( ) and curly braces {generate the matrix }, required for *Mix table*.

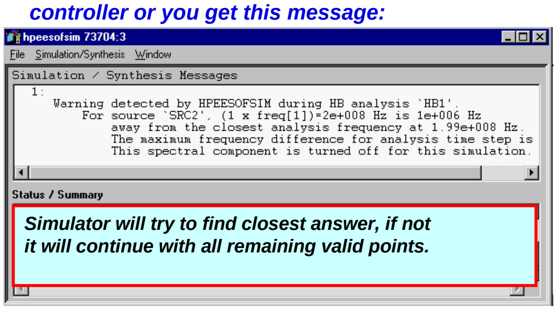
8th order term uses +5th & -3th, but not: -3th & +5th (+5th of RF does not exist).

QUIZ: Can you use this equation **dBm(Vout[1])** for this data? Is it valid?

*Answer: **YES** - if no other dependencies exist - it's the same as:* **dBm(mix(Vout,{-1,1}))**

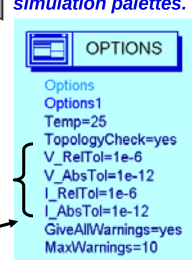
Harmonic Balance convergence & errors

Freq [x] in each source must match Freq [x] in the controller or you get this message:



NOISE TEMP error for all noise simulations: Set Temp=16.85 to eliminate any error message.

OPTIONS controller is in all simulation palettes.

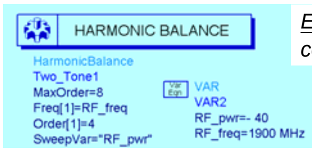


HB convergence error message:
"cannot sweep to desired level"
or "arc length continuation error"
To solve these problems, either loosen the V and I tolerances in the options controller by ten times (for example, set: I_AbsTol= e-11), or reduce the step size for power or frequency sweeps.

TIP about "quotes", brackets, braces, etc.

QUOTES:

- Only when editing on the screen for string value parameters, if necessary.
- When in doubt, double click and use the dialog boxes.



Exceptions - Swept variables are always in quotes and controller names ("HB1") in opt goals.

In dialog only: @ stops quotes when not needed.

If you see 2 quotes ""X"", remove one set!

Parentheses, Brackets, Curly braces:

(parentheses for function arguments)
[brackets for one, two, or three dimensional data]:
{curly braces for vectors and the mix function}:

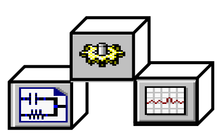
Double colon is a wildcard in ADS:

Examples: **dBm(Vout [1])** **dBm(mix(Vout,{-1,1}))** **mag(Vout [1:: 6]**

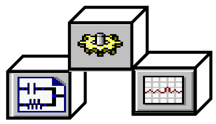
What the lab is about ...

Lab 7:

Harmonic Balance Simulations

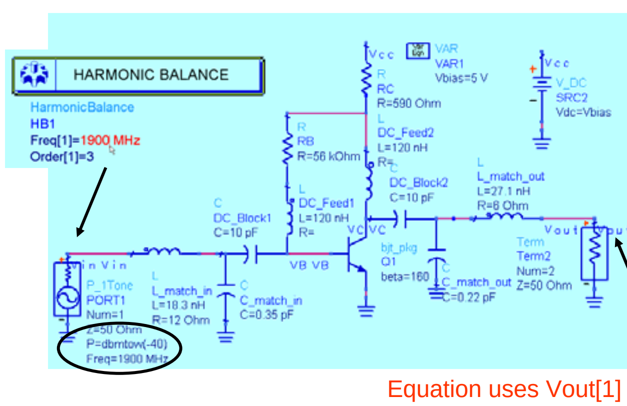


Steps in the Design Process



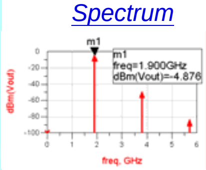
- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB* ← ***You are here:***
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- ***Final CDMA system test CE with fancy DDS***
- ***Co-simulation of behavioral system***

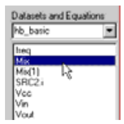
First, HB for spectrum and Meas Eqn



Equation uses Vout[1]

Spectrum





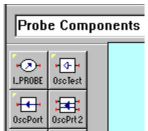
Dataset values

Dataset contains node voltages and Mix table

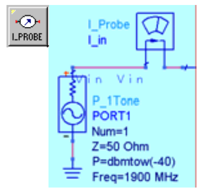
freq	Vout	Mix
0.0000 Hz	0.000 / 0.000	0
1.900GHz	0.180 / -14.199	1
3.800GHz	0.001 / -170.939	2
5.700GHz	1.963E-5 / 46.135	3

Next, simulate Power Delivered and Zin

Probe Components palette



NOTE for Oscillator testing: Use OscTest to determine if oscillation exists (S-param). Use OscPort to determine the frequency of oscillation (HB). See examples!



Rename the probe: I_in

Data display equations calculate power using voltage Vin and current I_in.

Eqn P_del_dBm = 10*log (0.5*real (Vin[1]*conj(I_in.i[1])))+30

NOTE: 0.5 is for 1/2 peak value and +30 give dBm (ref to 0.001 W)

Z_in is calculated at 1900 MHz which is index value [1]. Also, Z_in is used in dBm argument instead of default of 50.

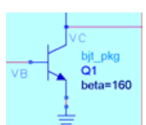
Eqn Z_in = Vin[1] / I_in.i[1]

Z_in
47.619 / 0.686

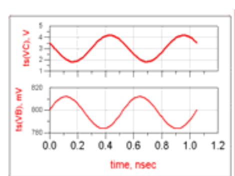
dBm(Vin[1],Z_in)	dBm(Vin[1])	P_del_dBm
-40.003	-40.214	-40.003

HB Phase, ts, and Gain Compression

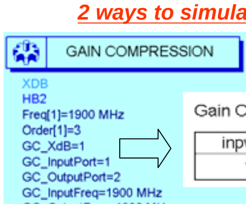
Use ts function on HB data at device nodes



Verify inversion of signal:

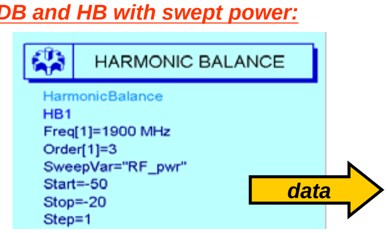


2 ways to simulate gain compression XDB and HB with swept power:



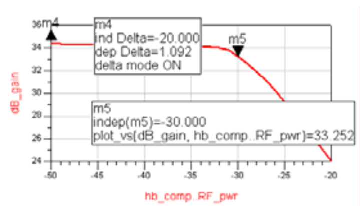
Gain Compression at 1900 MHz

inpwr[1]	outpwr[1]
-30.159	3.189



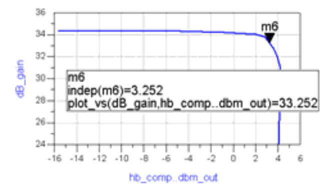
HB swept power compression data

Eqn dB_gain = hb_comp..dbm_out - hb_comp..RF_pwr



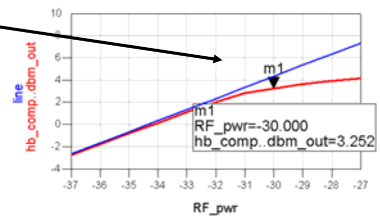
Plot_vs function

Plot_vs(dB_gain,hb_comp..dbm_out)



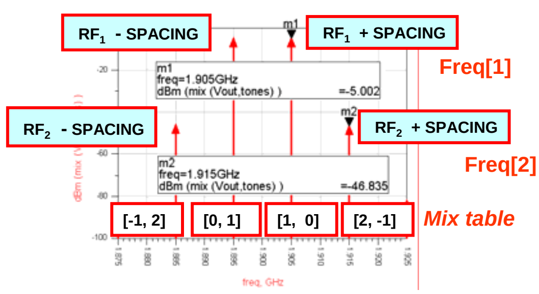
Creating a line: nonlinear to linear

Eqn line = hb_comp..RF_pwr + dB_gain [0]

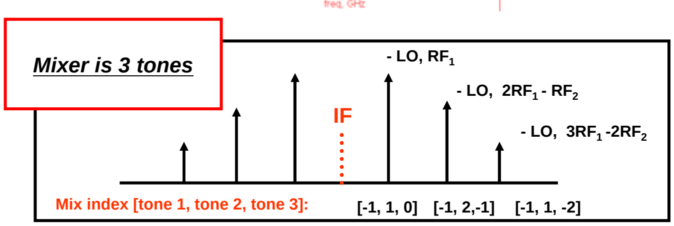


Simulating closely spaced tones...

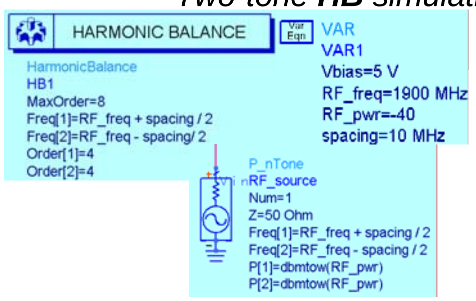
AMPS use 2 tones, such as RF +/- spacing (VAR)



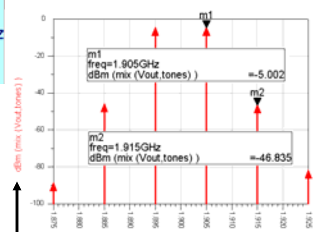
Lab setup and data



*Two-tone **HB** simulation and data*

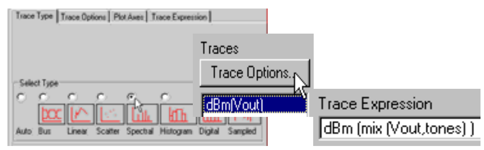


Spacing @ 10 MHz =
1.895 and 1.905 GHz



Eqn tones=[{1,0},{0,1},{2,-1},{1,-2}]

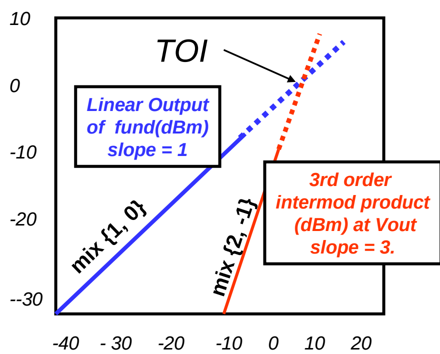
Use [brackets to generate a matrix] and { curly braces to vector the data from Mix table}



TOI or IP3 Measurement

When the input power drives the non-linear device into saturation or distortion, third order products near the desired frequency can become large. The point at which 3rd order products intercept the linear rise in output power is the intercept point TOI or IP3.

Output Power (dBm)



Input Power (dBm)

A measurement equation is used to get the answer...

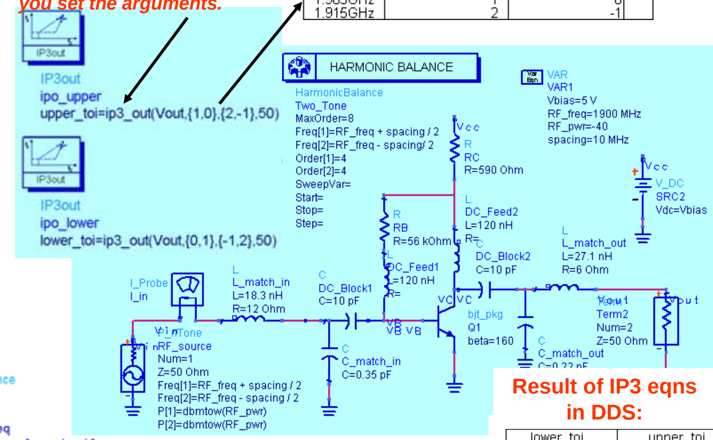
Lab setup and data

NOTE for mixer 3 tones use:
mix{-1,1,0}
mix {-1, 2, -1}

IP3 or TOI s...

Built-in measurements use functions - you set the arguments.

1.885GHz	-1	2
1.895GHz	0	1
1.905GHz	1	0
1.915GHz	2	-1



Result of IP3 eqns in DDS:

lower_toi	upper_toi
15.679	15.914

HarmonicBalance
HB1
MaxOrder=10
Freq[1]=LO_freq
Freq[2]=RF_freq + f_spacing / 2
Freq[3]=RF_freq - f_spacing / 2
Order[1]=7
Order[2]=3
Order[3]=3

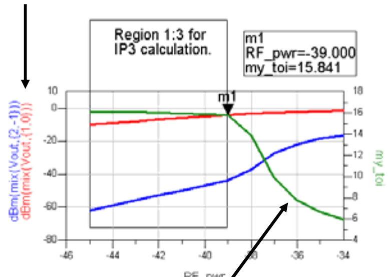
NOTE: Mixers use this setup for 3 tone TOI.

Compare swept values to values in the TOI measurement range:

OPTIONAL: Sweep RF pwr vs TOI

HARMONIC BALANCE
HarmonicBalance
Two_Tone
MaxOrder=8
Freq[1]=RF_freq + spacing / 2
Freq[2]=RF_freq - spacing/ 2
Order[1]=4
Order[2]=4
SweepVar="RF_pwr"
Start=-45
Stop=-30
Step=1

Swept values used for IP3



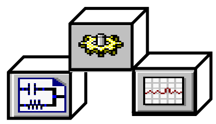
Eqn my_toi = ip3_out(Vout,{1,0},{2,-1},50)

The Eqn, ***my_toi***, is on the right Y axis.
When RF_pwr is greater than 39dBm, RF and third order slopes are no longer 1:3, so IP3 Eqn becomes invalid.

Start the lab now!

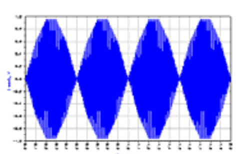
Topic 8:

Circuit Envelope Simulation



What is Circuit Envelope ?

- Time samples the modulation envelope (not carrier)
- ***Compute the spectrum at each time sample***
- ***Output a time-varying spectrum***
- ***Use equations on the data***
- ***Faster than HB or Spice in many cases***
- ***Integrates with System Simulation & HP Ptolemy***

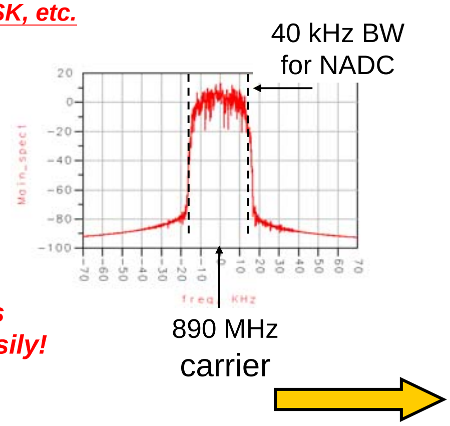


Use realistic Signals with CE

GSM, CDMA, GMSK, pi/4DQPSK, QPSK, etc.

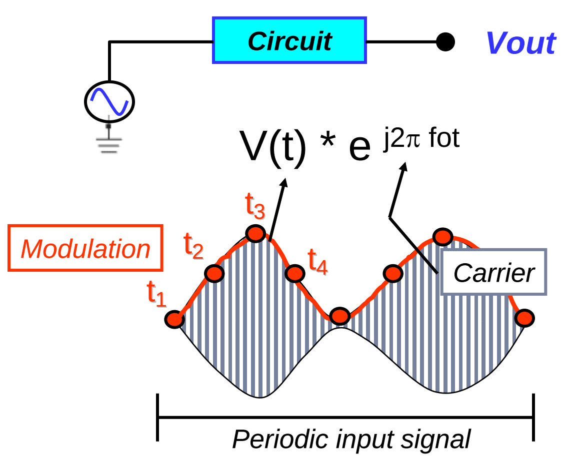
Simulations include:

- Adjacent Channel Power Ratio
- Noise Power Ratio
- Error Vector Magnitude
- Power Added Efficiency
- Bit Error Rate



2-tone tests and linearized models do not predict this behavior as easily!

Circuit Envelope Technology:

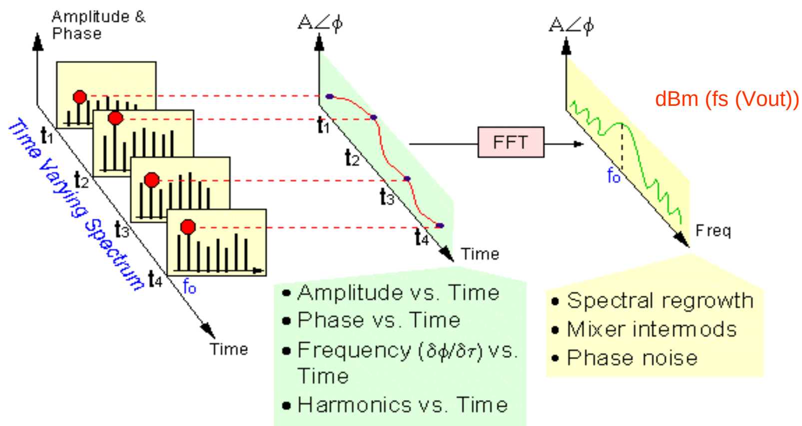


Time sample the envelope and then performs Harmonic Balance on the samples!

NOTE: V(t) can be complex - am or fm or pm

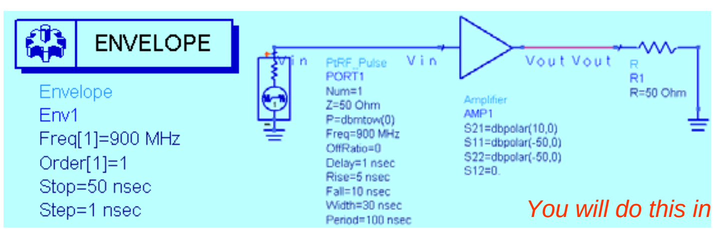
…more on CE Technology

Captures time and frequency impairments:



CE example: AMP with RF pulse

ONE TONE

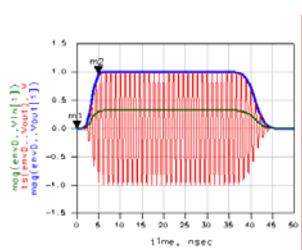


You will do this in the lab!

Step time is critical for sampling the envelope: rise, fall, and modulation rate. Therefore, Step (sample time) is NOT the same as Transient.

- ***mag of Vin [1]*: envelope**
- ***ts of Vout*: signal**
- ***mag of Vout [1]*: envelope**

…where [1] is the carrier Freq[1].

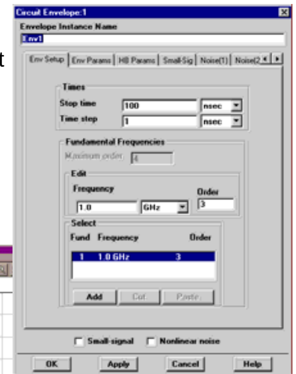


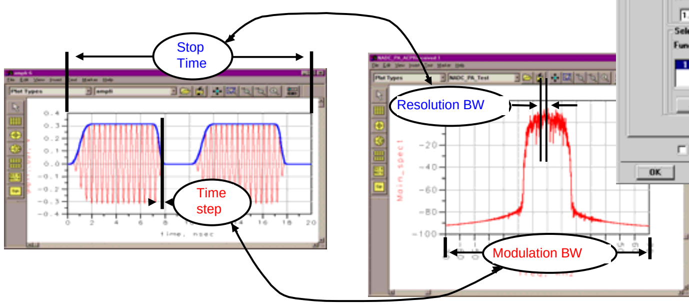
Env Setup *tab in dialog*

(Reference slide for one tone with 3 harmonics)

Stop time
- Determines resolution bandwidth of output spectrum
- Large enough to resolve spectral components of interest

Time step
- Determines bandwidth of Circuit Envelope simulation
- Small enough to capture highest modulation frequency



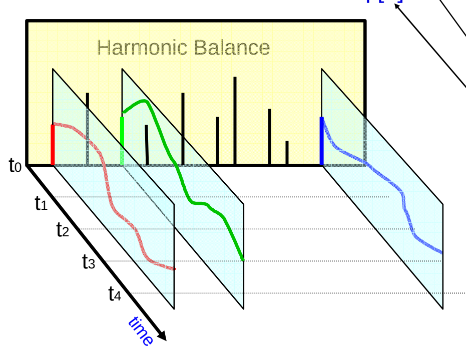


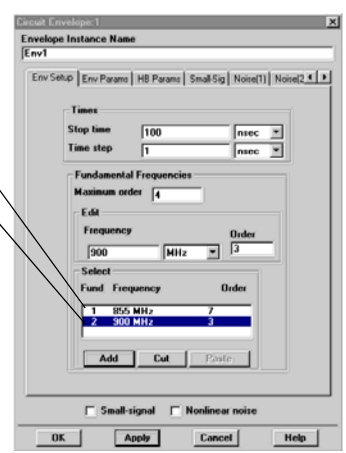
more

ENV Steup *tab (continued)*

(Reference slide for multiple tones: mixer)

7 Harmonics of Fundamental: Freq [1]
3 Harmonics of Fundamental: Freq [2]



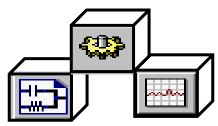


Multiple tone simulation requires more data display

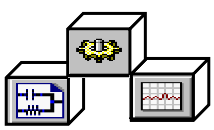
What the lab is about ...

Lab 8:

Circuit Envelope Simulations



Steps in the Design Process

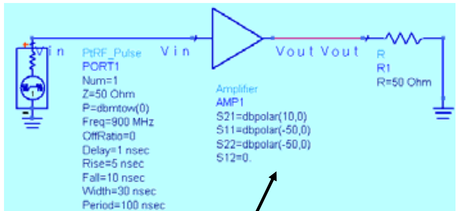


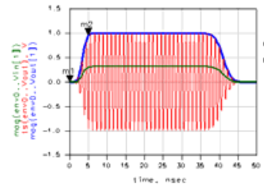
You are here:

- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- ***CE basics for spectrum and baseband***
- ***CE for amp_1900 with GSM source***
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*

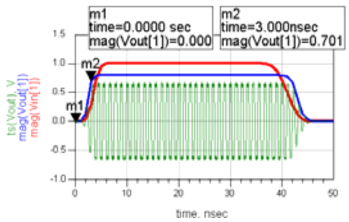
First, simulate an RF pulse

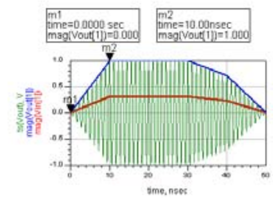
Use a behavioral amp and different time steps:



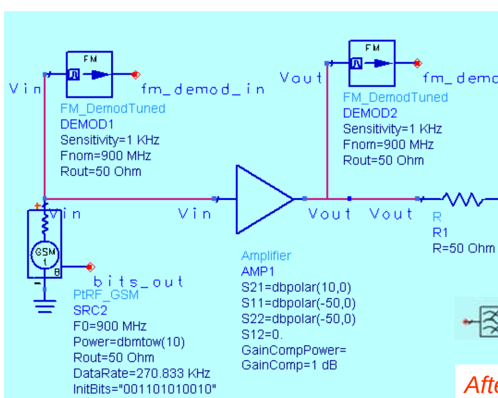


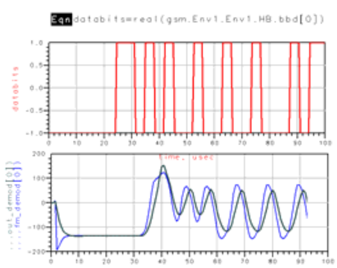
Add distortion: get odd harmonics out-of-phase!

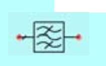




Next, phase distortion bits (baseband)

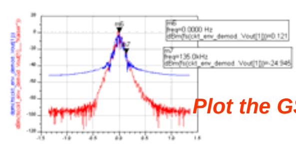






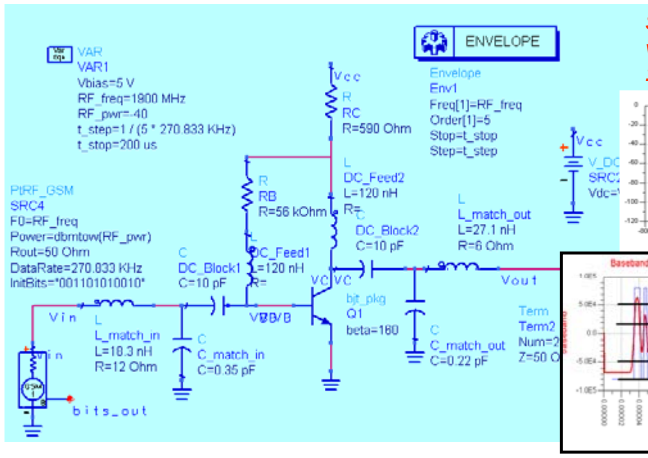
Look at the bit stream: ***bits_out*** *node and* ***Vout****.*

After simulating the response of the amplifier, insert a filter at ***Vin*** *to alter the phase response.*

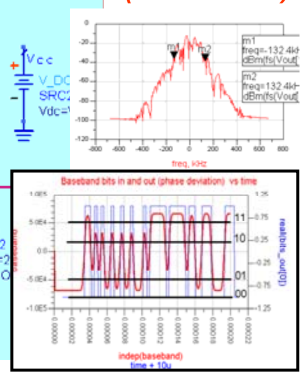


Plot the GSM BW spectrum with windowing.

GSM source on your amp_1900



Simulation with variables set t_step = 1 / (5X BW of GSM)



Verify baseband integrity using an equation that is like a demodulator:

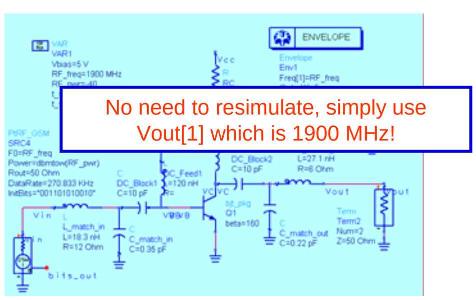
Eqn baseband=diff (unwrap (phase (Vout [1])) / 360)

Optional - channel power calculation

On a new page in DDS, write two equations:
limits **defines the bandwidth and**
channel_pwr **calculates power in the channel.**

Eqn limits= {-(270KHz / 2), (270KHz / 2)}

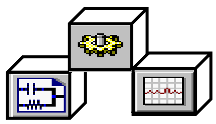
Eqn channel_pwr=10*log(channel_power_vr (Vout[1], 50, limits, "Kaiser"))+30



channel_pwr
-4.886

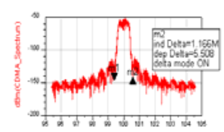
Topic 9:

Final Circuit / System Simulation and Co-Simulation



What is the final topic in this class?

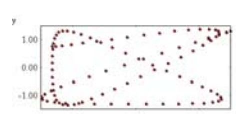
• Simulation of your amp_1900 and filters in the receiver system to verify analog performance.



Gain (S-21), HB with swept LO, and CE with a CDMA source

The co-simulation is optional if you have time - otherwise, take the CommSys class.

• Co-simulation: simulation of the entire system using digital circuits and analog circuits together.



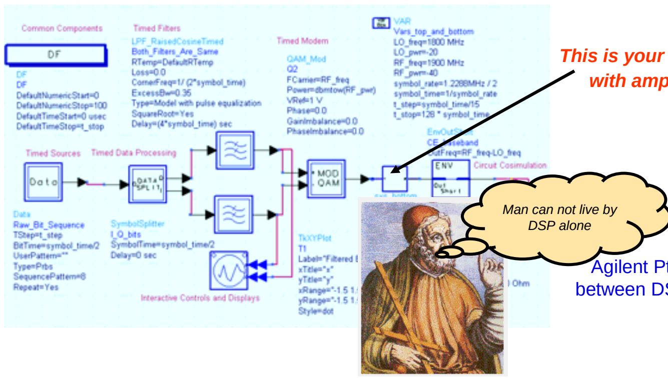
Constellation, spectrum, etc...

A little on co-simulation before starting the lab...

What is a co-simulation?

It is your amp_1900 and filters in the receiver system at the bottom level. On the top level is a Data Flow simulation.

Co-simulation is digital & RF/analog circuits simulating together.



This is your rf_system with amp_1900.

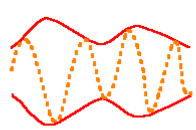
Agilent Ptolemy: a bridge between DSP and RF/Analog worlds

What is Ptolemy ?

Agilent Ptolemy is a Timed Synchronous Data Flow Simulator

- TSDF is a unique Agilent EEsof Innovation
- Agilent Ptolemy adds timed elements
  - parameters on signals are t, I, Q, Fc (rf carrie

A key advantage for real world



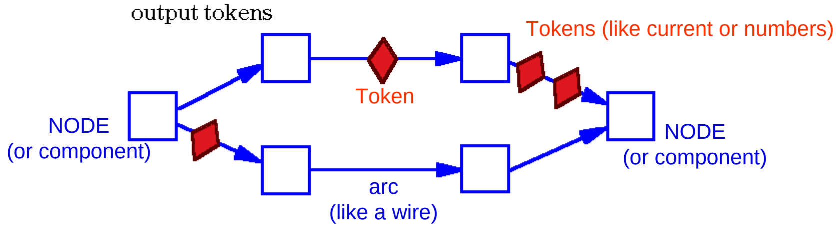
mixed signal design (DSP / analog / RF)

- Benefits:
  - easy to add real RF effects on signals
  - more efficient simulations
  - more accurate modeling of RF effects

Agilent Ptolemy is the data flow simulator used in the DSP schematic window...

What is Data Flow?

- Dataflow is ....
  - Each arc has FIFOs
  - A node maps input tokens onto output tokens
  - A set of firing rules specify when a node runs
  - A firing consumes input tokens and produces output tokens

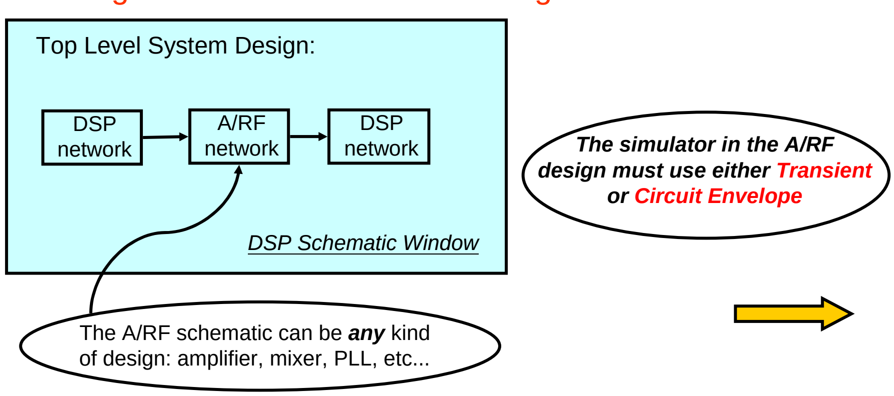


Tokens can also be "time stamped" - then they become samples. Now you can simulate time and frequency domain impairments such as multi-path and fading ...

What is Co-simulation?

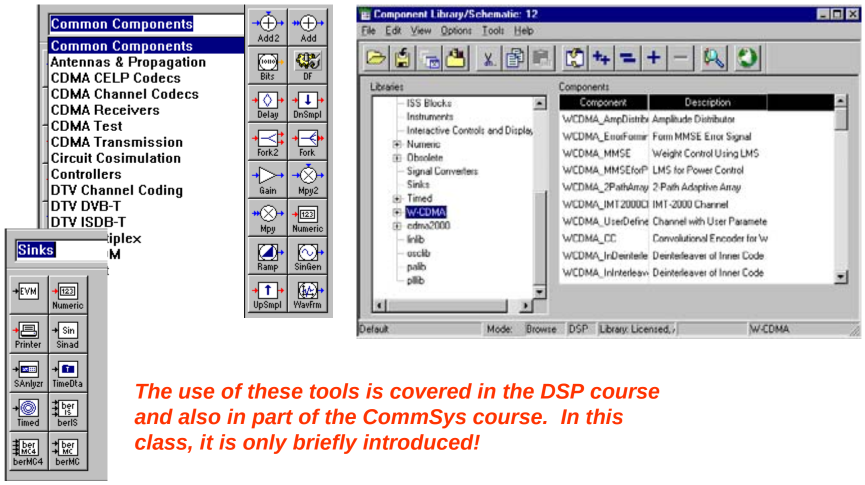
- ***Integrated Circuit, System Simulation & HP Ptolemy***

Co-simulation is simulating an A/RF schematic design with a DSP schematic design.



The DSP components library…

Many palette selections and additional libraries:

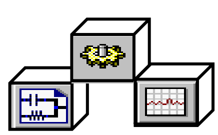


The use of these tools is covered in the DSP course and also in part of the CommSys course. In this class, it is only briefly introduced!

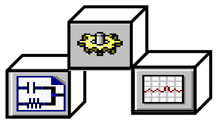
What the lab is about ...

Lab 9:

Final System Simulation using:
amp_1900 and filters



Steps in the Design Process

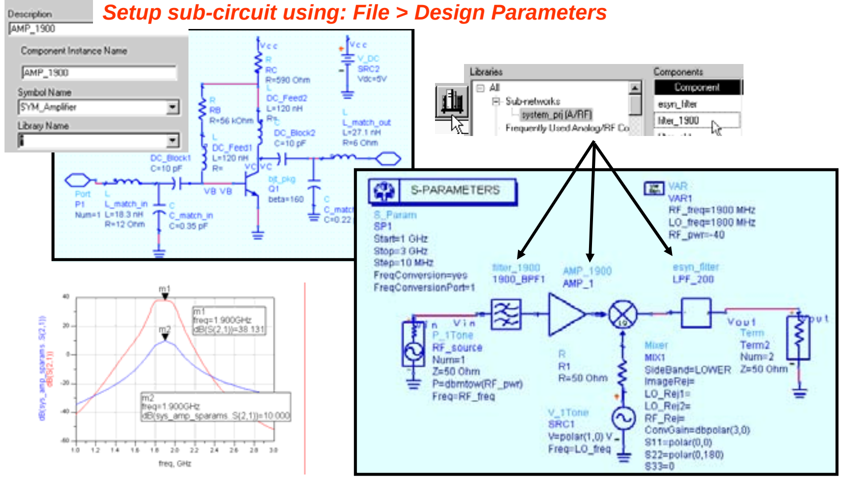


- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- *Start amp_1900 design – subckt parasitics*
- *Simulate amp DC conditions & bias network*
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- ***Final CDMA system test CE with fancy DDS***
- ***Co-simulation of behavioral system***

You are here:

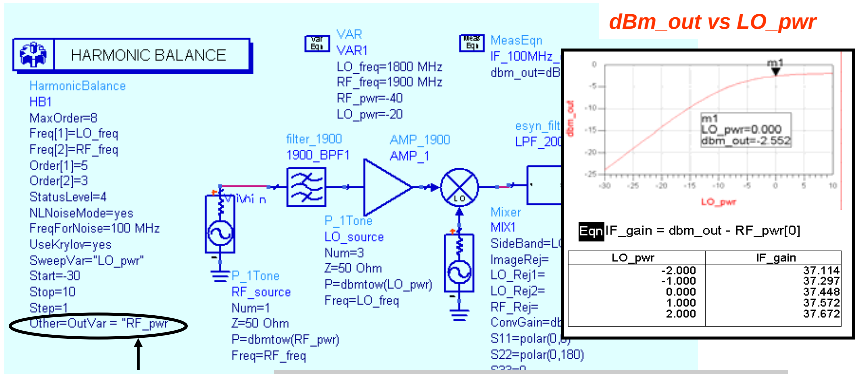
First, system setup with S-params

Setup sub-circuit using: File > Design Parameters

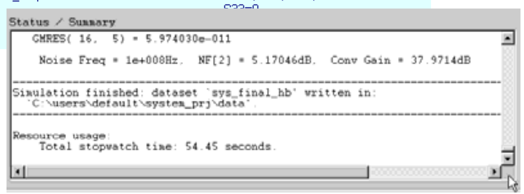


First, simulate system S21 with frequency conversion to see gain!

Next, HB simulation with swept LO

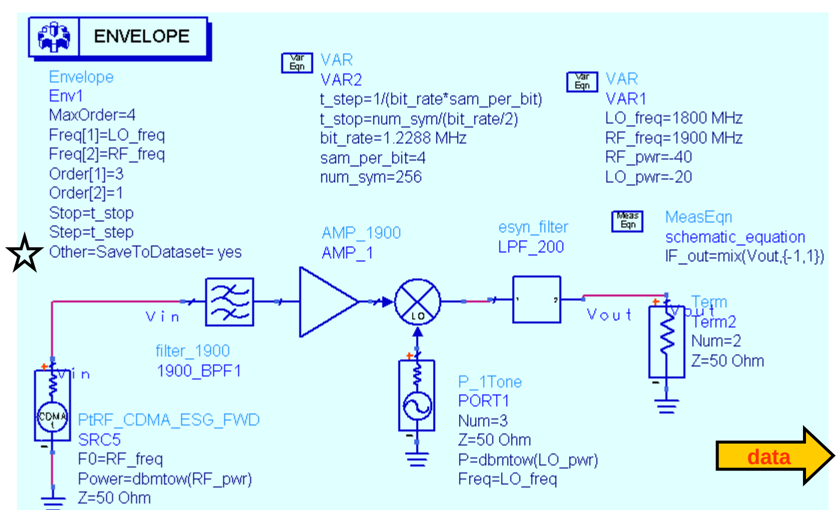


OutVar = *"variable" to dataset*



Noise measurement gives NF and conversion gain when status level = 4.

Final system CE simulation with CDMA

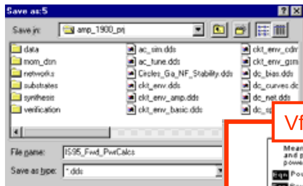


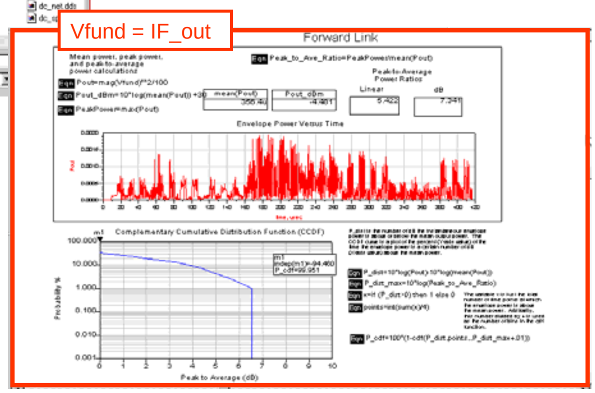
☆ **NOTE: Other = SaveToDataset = no** It means only the MeasEqn **IF_out** goes to dataset.

Power calculations from an example DDS

Use your CE data:

1. ***Open the example DDS file from the Tutorials/ModSources /IS95_Fwd_PwrCalcs***
2. ***Save it in your system_prj.***
3. ***Change the dataset and MeasEqn***



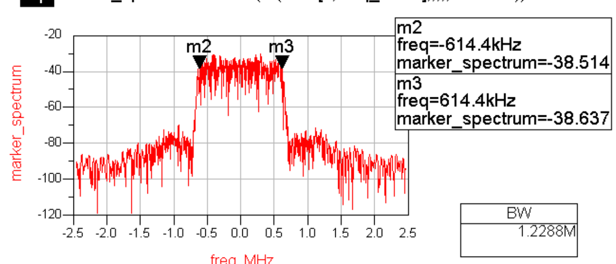


Simply change the eqn name to Vfund=IF_out and all the data fills in the plots and equations values!

Fancy Data Display for CDMA spectrum

Set up a marker slider using equations: m1 slides to a frequency and the spectrum of that carrier frequency is displayed: RF, LO, IF, etc.

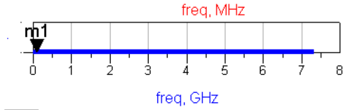
Eqn marker_spectrum = dBm(fs(Vout[::,freq_index],,,,,"Kaiser"))



BW
1.2288M

Eqn BW = indep (m3) - indep (m2)

m1 is at the IF



Eqn marker_freq = freq [0,::]

Eqn freq_index=find_index (marker_freq,m1)

m1
freq=100.0MHz
marker_freq=10.000E7

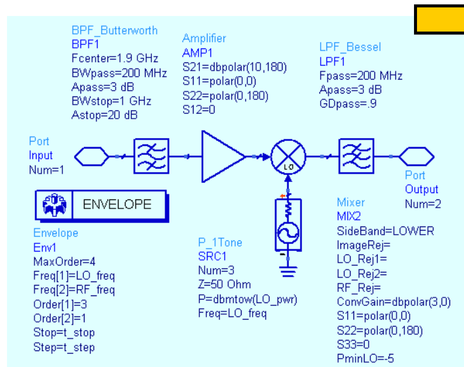
NOTE: If you finish this step in the lab, you have achieved all the goals for this class!

OPTIONAL - Co-simulation of rf_sys

This process requires several steps:

First step: Modify the system to become a sub-circuit (bottom CE simulation level) as shown here. Co-simulation requires a Transient or CE simulation setup.

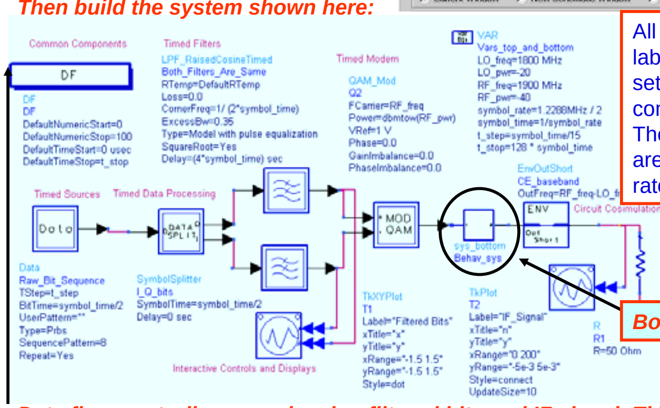
In this class, it is easier and faster to set up and run co-simulation using the behavioral system. The Extra exercise shows even more co-simulations you can try after the course!



Co-simulation continued...

Next step: Open a top level DSP network so that the Ptolemy / DSP palettes become available in schematic.

Then build the system shown here:

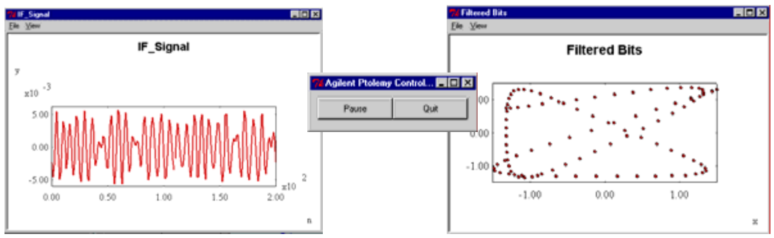


All the steps are in the lab, including the settings for the data components, filters, etc. The t_step and t_stop are now set for symbol rate and time.

Bottom level system.

Data flow controller runs showing filtered bits and IF signal: TkXY plots

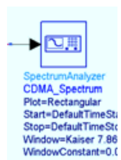
Data Flow simulation - TK plots are active!

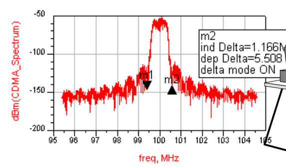


Quit the DF simulation and connect a SpectrumAnalyzer sink to collect the data. Results of this co-simulation show spectrum of the behavioral system. To use amp_1900 and your filters, replace them in the system and setup a new simulation (requires more time).

Start the lab now!

Spectrum Analyzer sink:





By the way... RF Board and RFIC Design Flow Integration Solutions from Agilent EEsof EDA

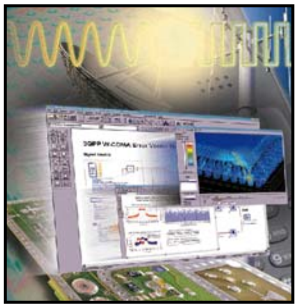
- Software and Services available to integrate ADS into existing design flow based on:
  - Schematic transfer via IFF
  - Layout transfer via IFF
  - Schematic sharing via Dynamic Link
- For more info, contact your Agilent EEsof Sales Representative, or visit:

cadence | Intercept Technology Inc. | ZUKEN REDAC | Mentor Graphics | Supermax E-CAD | VIEWLOGIC | VERIBEST, INC. The EDA Systems Company

RFIC Design Flow:
http://www.tm.agilent.com/tmo/hpeesof/products/ads/prod600a.html

RF Board Design Flow:
http://www.tm.agilent.com/tmo/hpeesof/products/ads/prod600b.html

End of the Course



Before you leave, please fill-out the course evaluation…thanks!

Look for us on the WWW for more ADS training and applications.

Goodbye and see you next time!
www.agilent.com
Agilent EEsof EDA - Customer Education