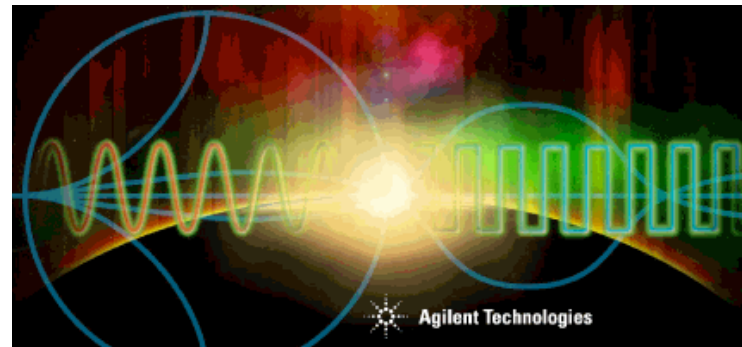# Advanced Design System – 2009

Fundamentals

Intensive hands-on training…prerequisite for all other ADS courses.
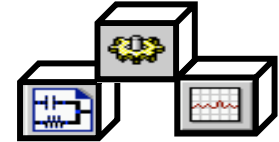
Course Part Number N3211A/B

from Agilent EEsof EDA Customer Education

Workbook part number: E8900-90672

Introductions & Logistics:

- Instructor: _____

- **Students:** Introduce yourself, your job,
  your experience with CAE tools,
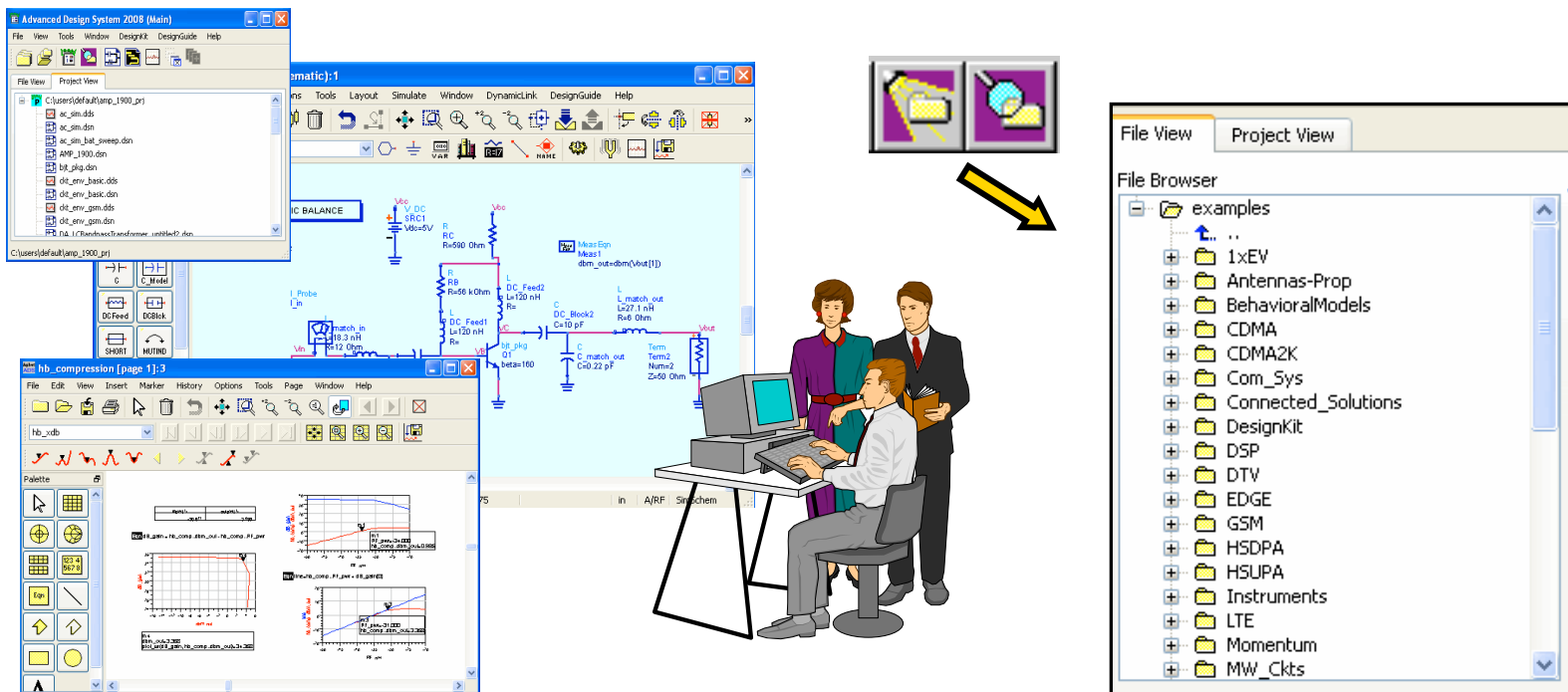 and your expectations for this class.

What is the value of this class?

You practice using ADS

With the aid of an instructor, you get hands-on experience.
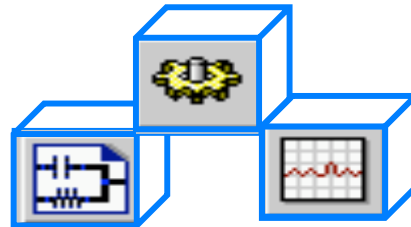
You gain expertise

You learn the same methods used by ADS experts in the examples directory.

Topic 1:

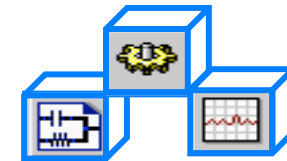# ADS Basics and Circuit Simulation Fundamentals

Topics and measurements in this class:

Built-in example designs
Tuning feature
System behavioral models
Amp, filter, and mixer measurements
Measurement equations
DC simulations: curves, bias, sweeps
Models, parameters, libraries
AC simulations: gain, group delay, noise
Use of DesignGuides: filter and smith chart
S-parameters: matching & optimization
Momentum, E-Syn, DAC
Transient & Harmonic Balance
2-tone simulations such as TOI (IP3)
Noise figure, power and distortion
Circuit Envelope simulations
Modulated sources: GSM & CDMA
Ptolemy simulation
and more…

Applies to:

RFIC, RF board,
uWave circuits,
and Systems.

Schedule of Topics and Lab exercises

Approximately 3 topics per day:

Lab 1: user interface, schematic capture, simulation basics.

|	
Day 1	1 . Circuit Simulation Fundamentals
	2. System Design Fundamentals
	3. DC Simulation and circuit modeling
Day 2	4. AC Simulation and noise contributions
	5. S-Parameters, Matching and Optimization
	6. Filters: DesignGuide,Transient, Momentum
Day 3	7. Harmonic Balance Simulations
	8. Circuit Envelope Simulation
	9. Final Circuit & System simulations

Labs 2 - 9: receiver system with amplifier and filter circuits.

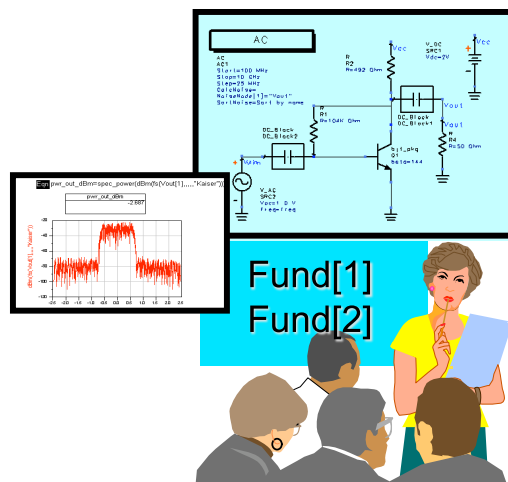GOAL: Complete all the labs - you will be busy!

Extra: The instructor may have class exercises or demos, if applicable. Also, if time allows, ask for information on other topics, circuit simulations, Design Guides, or examples.
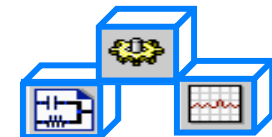
How this class is structured:

| Lecture: 25 % | Lab Exercises: 75 % |

Lecture: 25 %

Instructor describes the theory and the lab exercise using overhead slides, demos, etc.

Lab Exercises: 75 %

All exercises build on each other. **Optional** steps are not required and are for fast-paced students. **Extra Exercises** are provided for future practice.

Fund[1]
Fund[2]
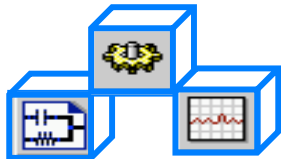
Average lecture time: 30 min (+/- 10 min).

Average lab time: about 1.5 hrs (+/- 15 min).

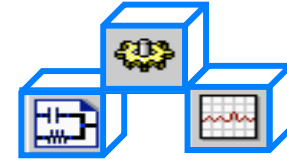This arrow means: there's more about this topic on the next slide.

Logistics and Materials:

- Class hours: _____

- Food, facilities, phone: _____

- WORKBOOK
  - Contains: slides, lab exercises, appendix.

- About the COMPUTERS
  - Login: _____
  - Password: _____

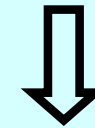**It's time to get started...** ⟹

Here is ADS Simplified: 3 steps

STEP 1: design capture

Insert circuit & system components and set up the simulation.

STEP 2: Simulation

Netlist is automatically sent to the simulator.

Simulation results (data) are written to a *dataset*.

STEP 3: display the results

Plot or list data & write equations.

Layout / Momentum.

ADS Windows: Main, Schematic, Status, Data Display

Main window: manage projects, open or copy designs, and get examples...

Schematic window: capture and modify circuits, setup or run simulations...

Schematic setup

Project Directory

Simulation Controller

Simulate

Data Display window: plot and list data, also write equations...

Data Display

Default Dataset

Resource usage:
   Total stopwatch time: 3.47 seconds.

Simulation finished: dataset `ac_sim` written in:
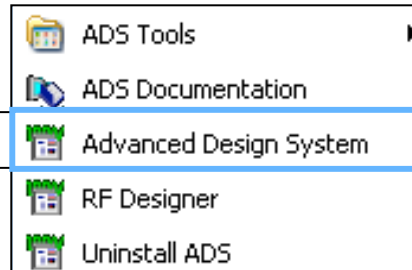   `C:\users\default\amp_1900_prj/data`

Simulation Status window:
simulation info: messages, errors, etc...

Start ADS and create a project…

start

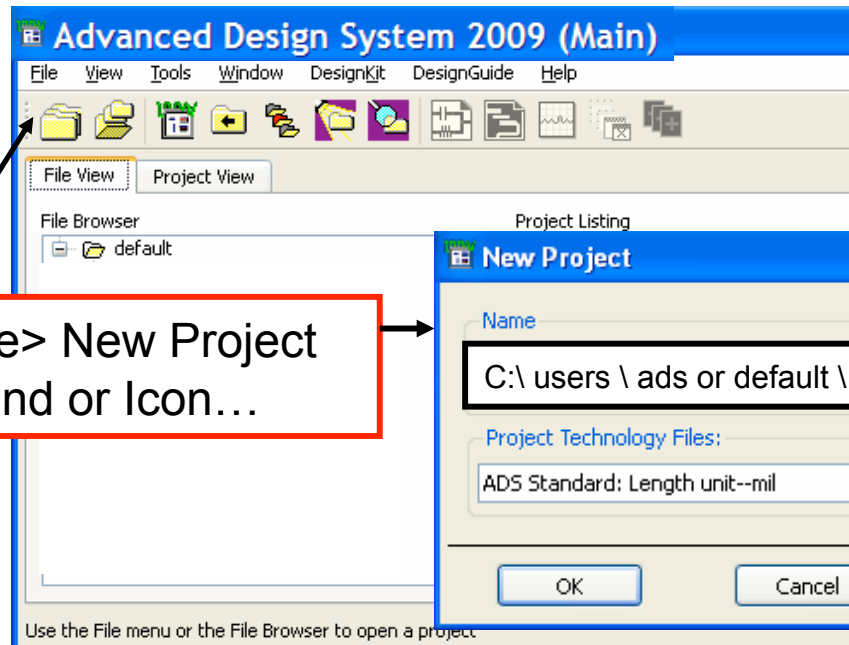All Programs ▶

Advanced Design System 2009

| ADS Tools ▶ |
| ADS Documentation |
| **Advanced Design System** |
| RF Designer |
| Uninstall ADS |

Advanced Design System

Or, use a desktop shortcut.

NOTE: UNIX users use a script!

Getting Started dialog...

Getting Started with ADS

Search the Knowledge Center [   ] Go

Agilent Technologies

Projects
Create a new project
Open an existing project
Open an example project
Open a recently used project

Help Center
ADS Quick Start
Agilent EEsof Web
Applications
Customer newsletter archive
Knowledge center
Manuals
Technical support
Technical support contacts
Vendor libraries for ADS
Videos
Web-based training
What's New

Close

ADS Main window:

Use menu commands or Icons →

Advanced Design System 2009 (Main)

File   View   Tools   Window   DesignKit   DesignGuide   Help

File View   Project View

File Browser   Project Listing

default

Use File> New Project command or Icon…

New Project

Name

C:\ users \ ads or default \ **lab1**   Browse...

Project Technology Files:

ADS Standard: Length unit--mil ▾   View Details...

OK   Cancel   Help

Use the File menu or the File Browser to open a project
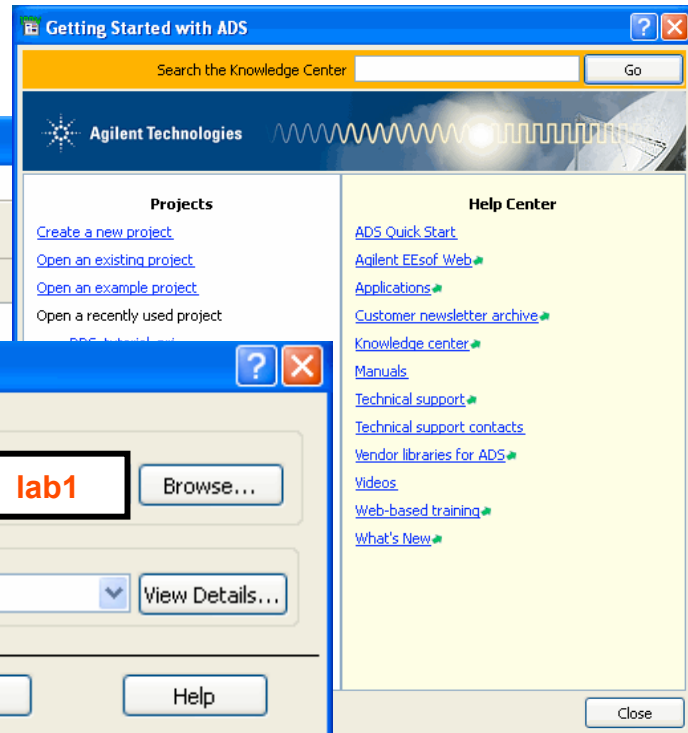
ADS Main window Tools…

Click Tools > Preferences:

For this course, these will be turned off in the lab exercise. You can try them later!
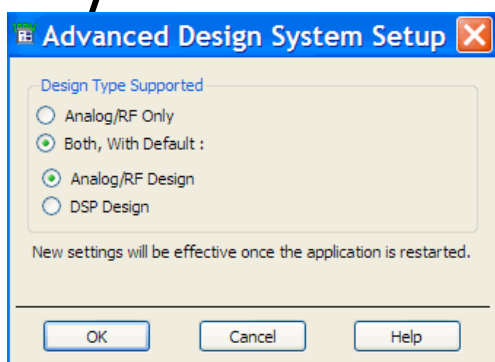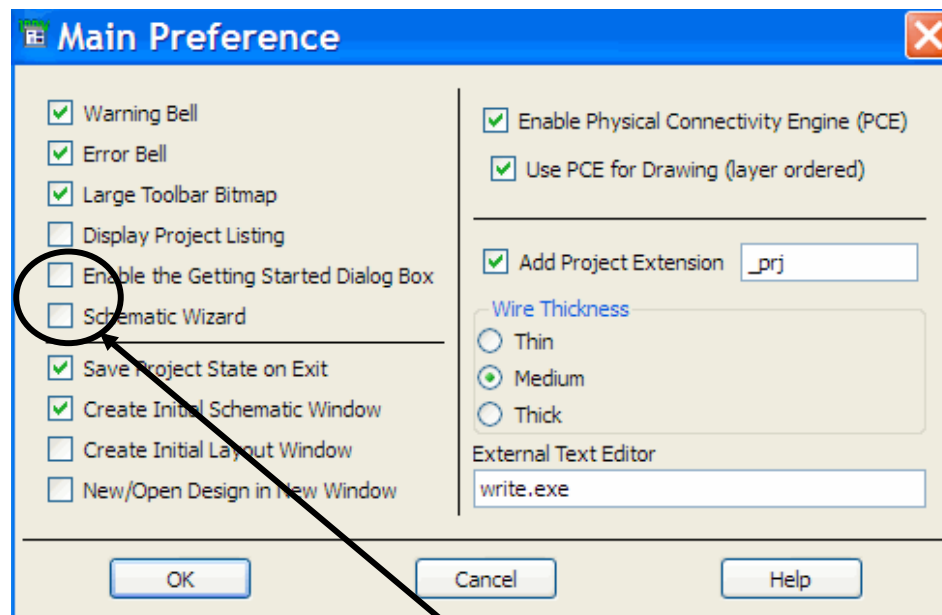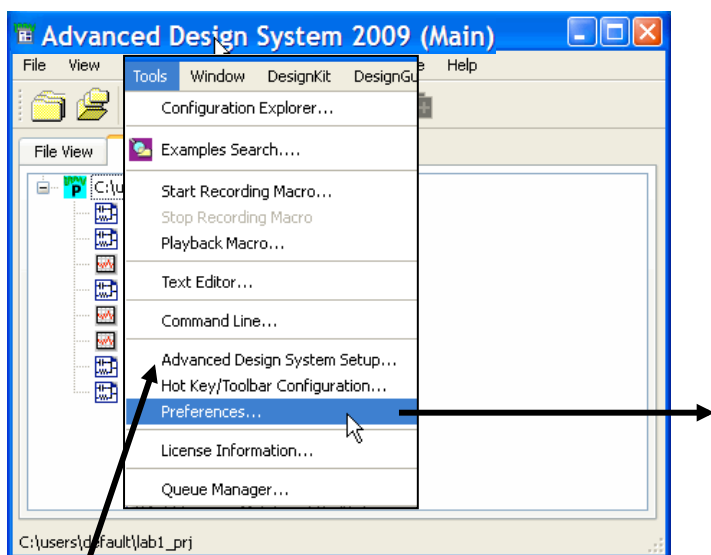
Setup is for setting the schematic availability of either Analog or DSP components or both.

Next, inside the project..

Files and folders are automatically created for new projects and a blank schematic window opens!
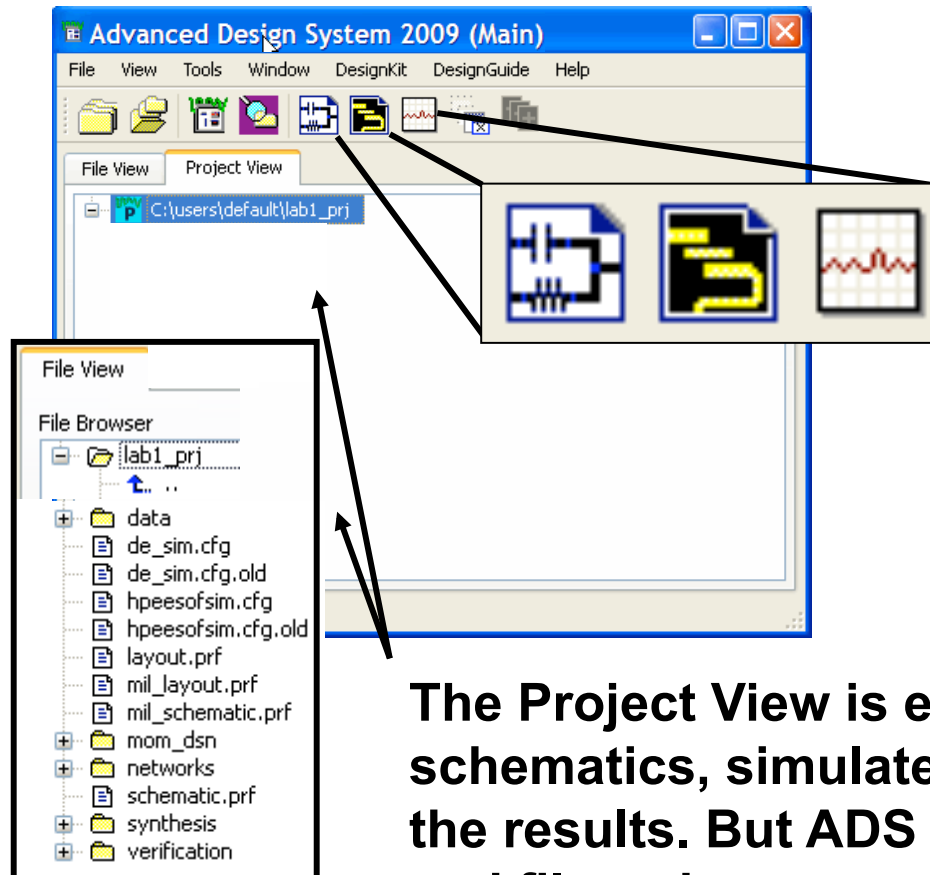
When you are in a project, these icons become active so that you can open the windows.

Next, the Schematic window...

The Project View is empty until you create the schematics, simulate to produce data, and display the results. But ADS automatically builds folders and files when you create a project.

Schematic window...

Use Options > Preferences to change grid, color, etc.

Use Window > Open Designs for unsaved designs.

Use icons to create, open, and save designs.

Rotate By Increment

Rotates the selected component or shape based on the current Rotation Increment (angle) setting.

Zooming and View all icons.
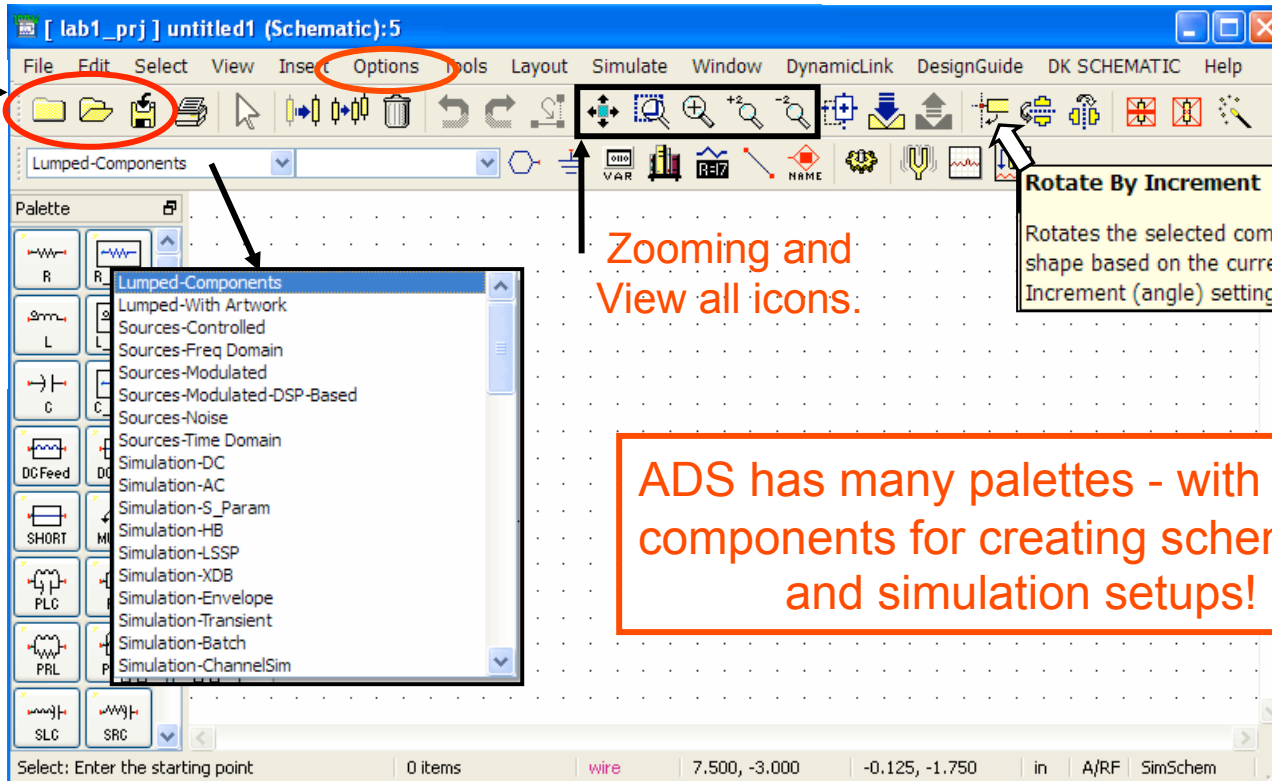
All icons have balloon help.

ADS has many palettes - with many components for creating schematics and simulation setups!

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

More on schematic!

Inserting and editing components

End command or use ESC.

Component History: **type the name = get the component !**

Activate / Deactivate toggle icons...

Edit components to see and modify parameter values.

The cursor shows cross-hairs and an image of the component you are inserting.

Parameters are defined: _M is multiplicity.

More...

Some useful schematic commands

You will use these and others in the lab exercises!

Next, the Library

Push / Pop for sub-circuits.

Hot keys are labeled.

Select command is also useful...

Edit > Component has many uses.

Library vendor parts + all your circuits
Search in any column – including URLs

Component Library

Analog/RF Libraries
  Projects
  Analog Parts Library (No L...
  Analog/RF
  Block Text Fonts
  Board Demo Design Kit
  Frequently Used DSP Com...
  HF Diode Library
  Measurement Based SMT ...
  Microwave Transistor Library
  RF Passive SMT Library
  RF Transistor Library
    Packaged BJTs
    Packaged GaAs FETs
    Packaged Power MOSF...
  S Parameter Library (No L...

Download Libraries...

Search	Search	Search	Search
Component ▲	Description	Library	Vendor
pb_hp_AT41486_19931202	AT41486: AVNK86 Pack...	RF_Transistor...	Agilent Techn...	L
pb_hp_AT41511_19931202	AT41511: SOT143 Pack...	RF_Transistor...	Agilent Techn...	L
pb_hp_AT41511_19950125	AT41511: SOT143 AT...	RF_Transistor...	Agilent Tech...
pb_hp_AT41533_19950125	AT41533:		
pb_hp_AT41586_19931202	AT41586:		
pb_hp_AT41586_19950125	AT41586:		
pb_hp_AT42010_19920721	AT42010:		
pb_hp_AT42010_19921101	AT42010,		
pb_hp_AT42035_19911003	AT42035,		
pb_hp_AT42035_19920721	AT42035:		
pb_hp_AT42035_19921101	AT42035:		
pb_hp_AT42070_19920721	AT42070:		
pb_hp_AT42070_19921101	AT42070,		
pb_hp_AT42070_19931202	AT42070: AVNK70 Pack...	RF_Transistor...	Agilent Techn...	L

- To **insert a component**, drag and drop the component name onto a design window.
- To **search any column**, type text into the Search field above the column.
- To **hide or show a column**, right click on any column name and check or uncheck the column.
- To **rearrange columns**, drag the column name left or right.
- To **resize columns**, drag the seperator between column names.

Select the part and it is attached to your cursor, ready to insert.
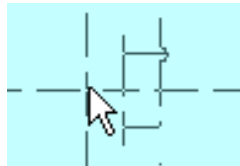
One more slide on schematics...

Check your Schematic for errors

Click: Tools > Check Representation

Before or after simulation!

Unconnected wire

Next, setting up a simulation...

First step: insert a **Simulation Controller**

Select the palette:

Simulation-S_Param

- Simulation-DC
- Simulation-AC
- **Simulation-S_Param**
- Simulation-HB
- Simulation-LSSP
- Simulation-XDB
- Simulation-Envelope
- Simulation-Transient
- Simulation-Batch
- Simulation-ChannelSim
- Simulation-Instrument
- Simulation-Budget
- Simulation-Sequencing
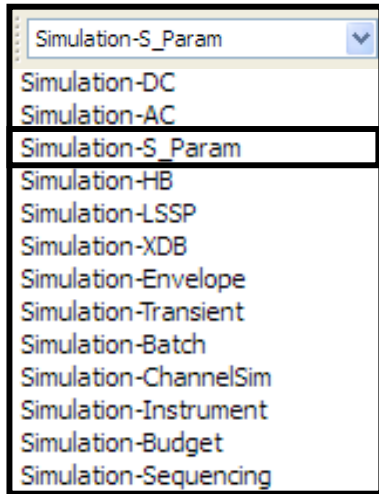
Other Palette items: Sweep Plan, Options, Meas Eqn. Some items (measurements and components) are simulation specific.

*NOTE asterisk means schematic is not yet saved.

[lab1_prj] lpf * (Schematic):1

File   Edit   Select   View   Insert   Options   Too

Simulation-S_Param

Palette

- S P
- Lab
- Sweep Plan
- Prm Swp
- Options
- Term
- RefNet
- RefNet
- OscTest
- Linear Net
- NdSet
- NdSet Name
- Disp Temp
- Meas Eqn
- MaxGain
- PwrGain
- VoltGain
- VSWR

Click the gear and insert the controller.

S-PARAMETERS

S_Param
SP1
Start=1.0 GHz
Stop=10.0 GHz
Step=0.5 GHz

L
L1
L=1.0 nH
R=

Term
Term1
Num=1
Z=50 Ohm

C
C1
C=1.0 pF

C
C2
C=3.0 pF

Term
Term2
Num=2
Z=50 Ohm

Select: Enter the starting point   0 items   wire   6.500, -1.375   2.250, -2.750   in   A/RF   SimSchem

Next, editing the controller!

Next, edit the Simulation parameters...

S-PARAMETERS

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

By default, **freq** is the swept variable here!

Edit **on-screen** if the parameter is <u>displayed</u> or use the dialog box.

You will learn to use these in the lab exercises!

Scattering-Parameter Simulation:3

S_Param Instance Name
SP1

| Frequency | Parameters | Noise | Output | Display |

Frequency
- Sweep Type — Linear
- ⦿ Start/Stop ◯ Center/Span
- Start — 1.0 — GHz
- Stop — 10.0 — GHz
- Step-size — 1.0 — GHz
- Num. of pts. — 10
- ☐ Use sweep plan

OK   Apply   Cancel   Help
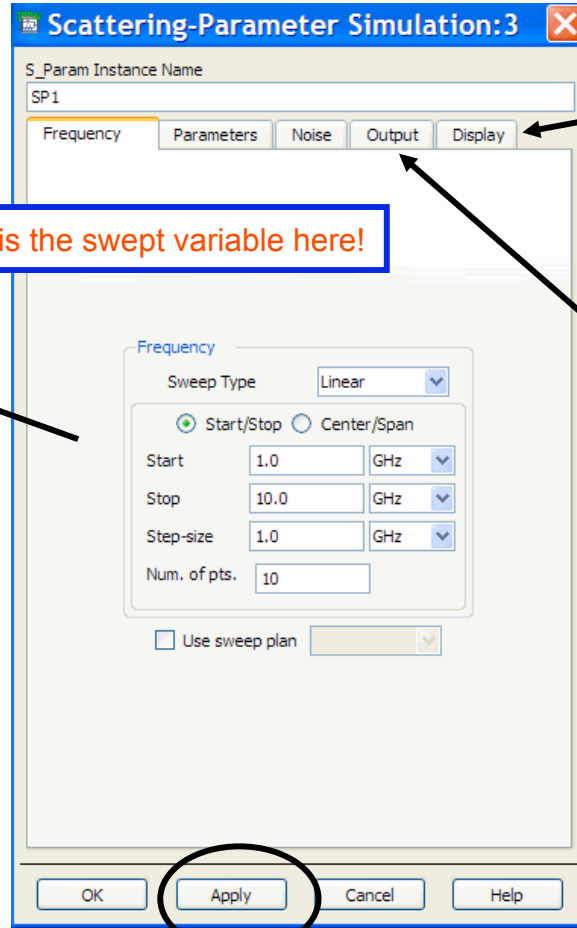
Display tab lists all the settings for on-screen display.

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

Almost ready to simulate...

Running the simulation...

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

Dataset files (.ds) are in the DATA directory.

Data Display windows (.dds) are in the PROJECT directory.
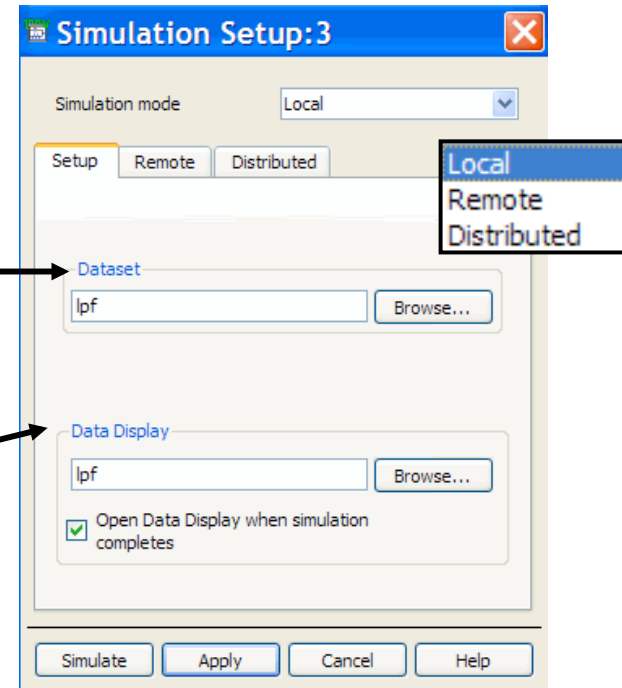
To simulate: use F7 key, click Simulate, or click the gear icon on the schematic.

NOTE: Servers can allow parallel simulation.

…and the simulation runs!

Simulation information: Status Window

One way to stop a simulation, click:

Simulation/Synthesis > Stop Simulation

Or, use schematic window simulate command:

Simulate		
Simulate	F7
Simulation Setup...	S
Simulation Variables Setup...	
Stop and Release Simulator...	
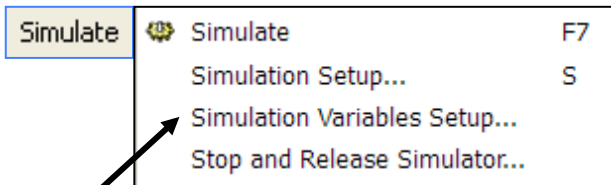
Simulation Variables list (tuning)

The simulation results: **dataset** (.ds file) written into the data directory.

Opens when simulating or click: Window > Simulation Status

hpeesofsim 4:0

File Simulation/Synthesis Text Window

Simulation / Synthesis Messages

Status / Summary

```
hpeesofsim (*)
Copyright Agilent Technologies

SP SP1[1] <C:\users\default\lab1_prj\networks\lpf>    freq=(1 GHz->10
...........................
Resource usage:
    Total stopwatch time: 0.97 seconds.
------------------------------------------------------------------------
Simulation finished: dataset `lpf' written in:
  `C:\users\default\lab1_prj/data'
------------------------------------------------------------------------
```
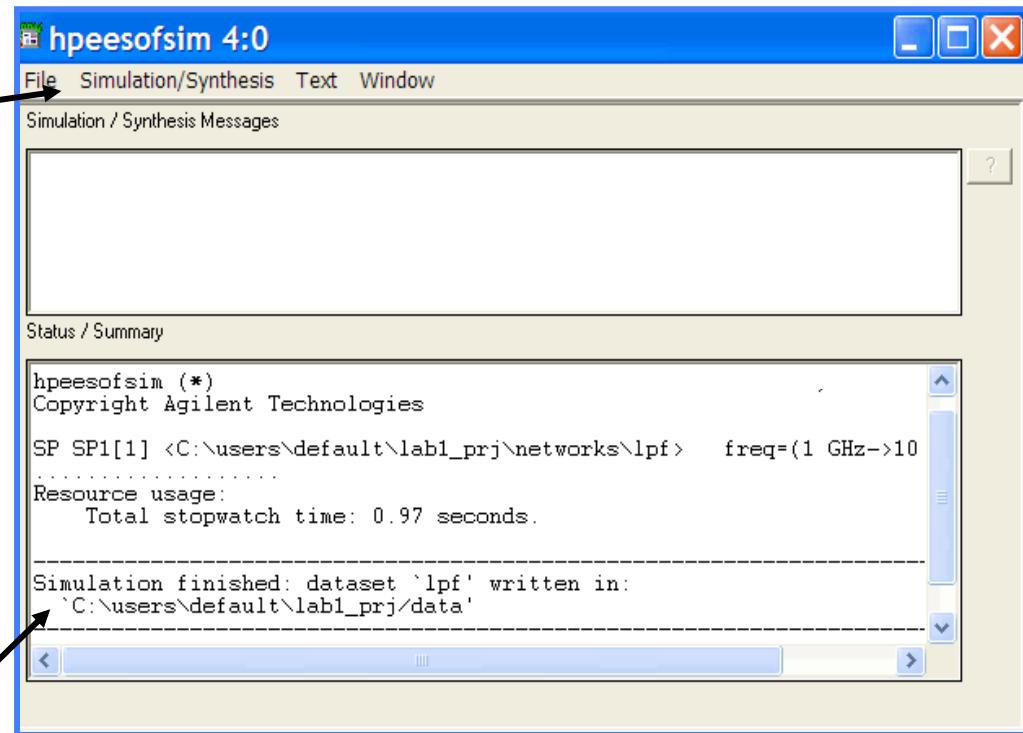
When finished, the Data Display opens…
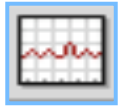
Data Display (DDS) window

It automatically opens or you can open this window

from any Schematic or the ADS Main window:

Default Dataset

Marker Toolbar: peaks and valleys

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

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

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

NOTE: Use the Options menu for control of DDS features - turn on/off toolbar icons, set Hot Keys, etc.

Next, plotting data...

List or plot the simulation data...

QUIZ:
What is this?

Answer:

_____ _____

lpf [page 1]:0

File   Edit   View   Insert   Marker   History   Options   Tools   Page   Window   Help

lpf

Palette

Plot Traces & Attributes:0

Plot Type | Plot Options

Datasets and Equations

lpf

freq
PortZ
PortZ(1)
PortZ(2)
S
S(1,1)
S(1,2)
S(2,1)
S(2,2)

Show Hierarchy

Manage Datasets...

Traces

Trace Options...

dB(S(2,1))

>>Add>>

>>Add Vs...>>

<<Delete<<

Advanced

OK   Cancel   Help
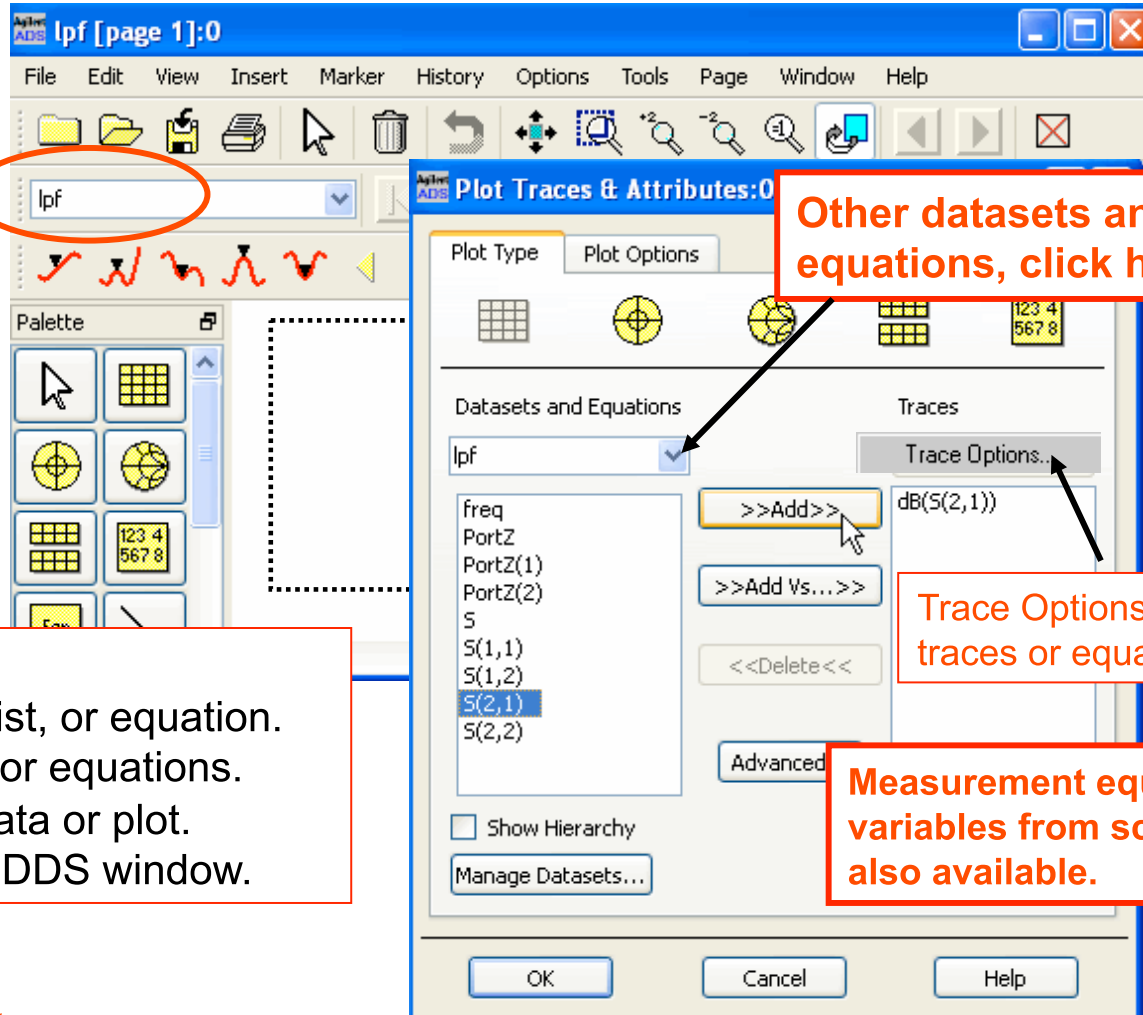
Other datasets and DDS equations, click here!

Trace Options: edit data traces or equations here.

Measurement equations and variables from schematic are also available.

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

Answer: default dataset

You control all your simulation data

Here is S(2,1) and more - the default DDS name is the same as the schematic!

Page tabs (see bottom).

Also, use mouse wheel to zoom in/ out.

Type in marker freq to move it.

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

Plots, **Traces**, and **Markers** can be edited.

Trace label: explicit **dataset...path** if not the default dataset.

Write **equations** using ADS functions to operate on data.

Next, Tuning…

TUNING PARAMETERS: tune mode = simulation!

  Simulate > Tuning...

Select parameters, tune, and watch the results!
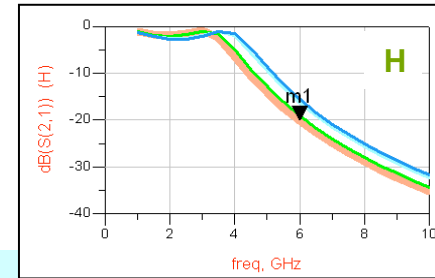
H: means the trace history is turned on.
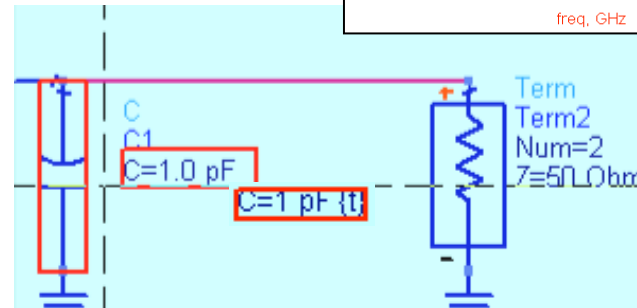
List of available simulation variables: Simulate > Simulation Variables Setup:

Next, file views …

Views of an ADS Project Directory

Project   **for example: lab1_prj**

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

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

networks directory contains **.dsn** files (designs)
These are layouts and <u>schematics with simulation setups</u>.

OTHER folders / files
mom_dsn (Momentum)
substrates (Momentum)
synthesis (used for E-Syn & DSP)
verification (used for DRC)
notebook
verilog

These folders are not required unless using the specific tool or model, etc.

The prf, cfg, log and some other files can be ignored.
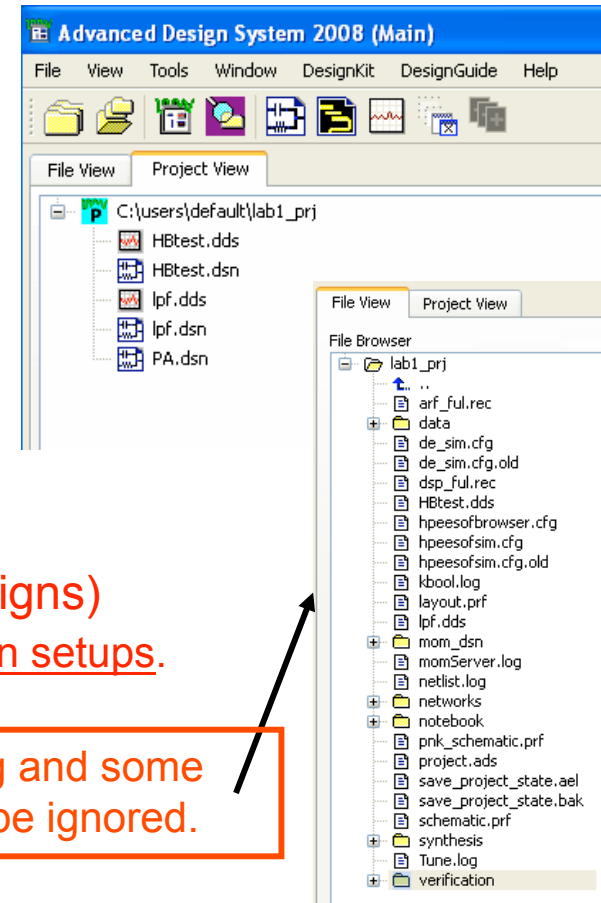
To delete ADS files,
Use: File > Delete ___

ZAP your projects for e-mail or disk

From the Main window, click: File > Archive or Unarchive
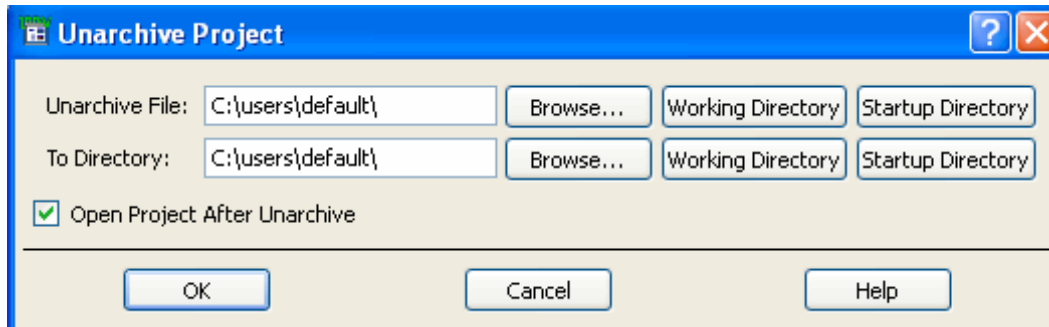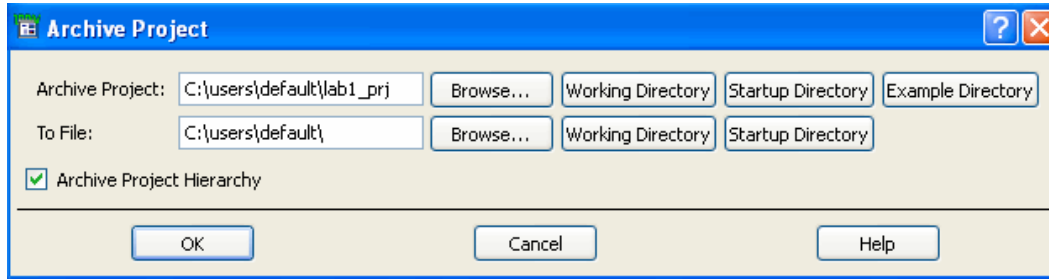
NOTE: Archive files become .ZAP files (like .ZIP files). They can include all networks, data, and display files (entire project

At the end of this course, archive your projects to C:\temp and then copy the ZAP files to your floppy or CD drive. Be sure to delete the datasets if using a floppy disk - not necessary if copy to a CD or if you are e-mailing a zap file.

ADS technical support in the United States call: 1 - 800 - 473-3763.
Or, on the Web: http://eesof.tm.agilent.com/support/supp101.html

ADS has some powerful features:

- Templates
- Examples
- Design Guides
- Smart Simulation Wizard
- Test Labs and Test Benches
- Link to GENESYS
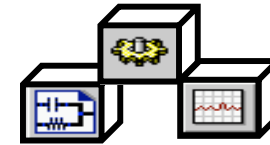
**The next few slides briefly shows these features.
Then you will start the first lab exercise!**

Templates for simulation and data display

You can also make your own templates:
File > Save Design as Template...

Insert your device or circuit and simulate. But you need to verify settings: nodes, bias, variables, parameters...

Data Display also has templates!

Examples and Example Search...
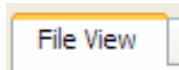
Main window File View icons
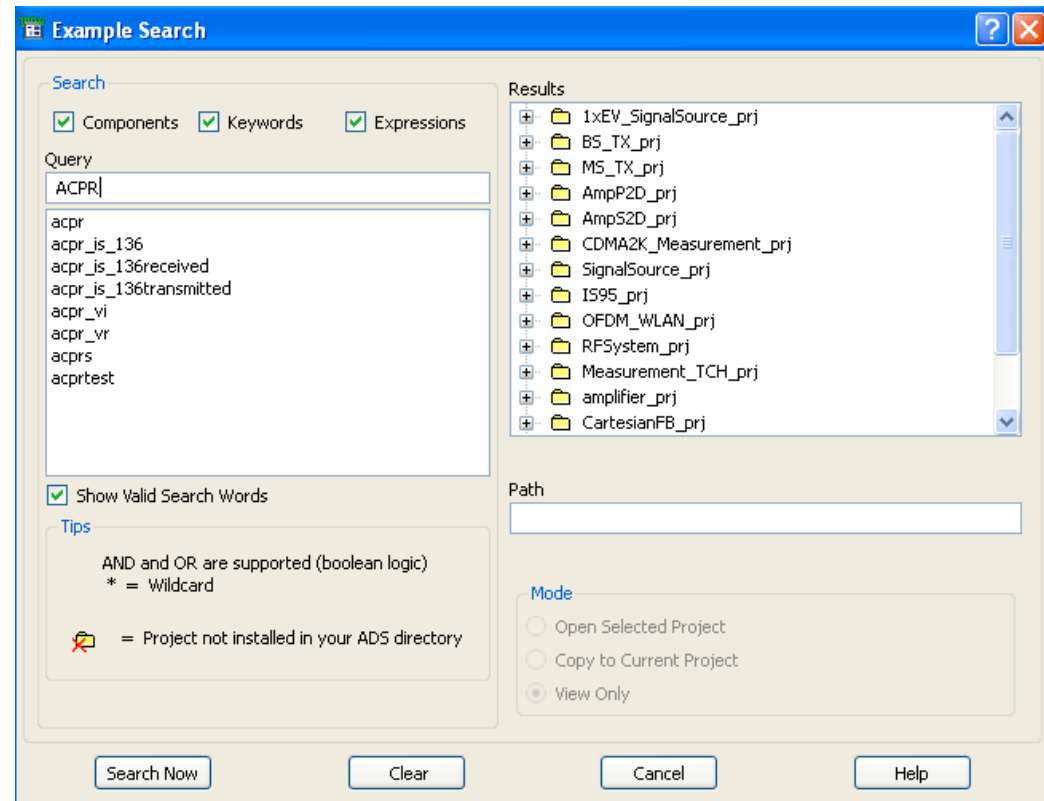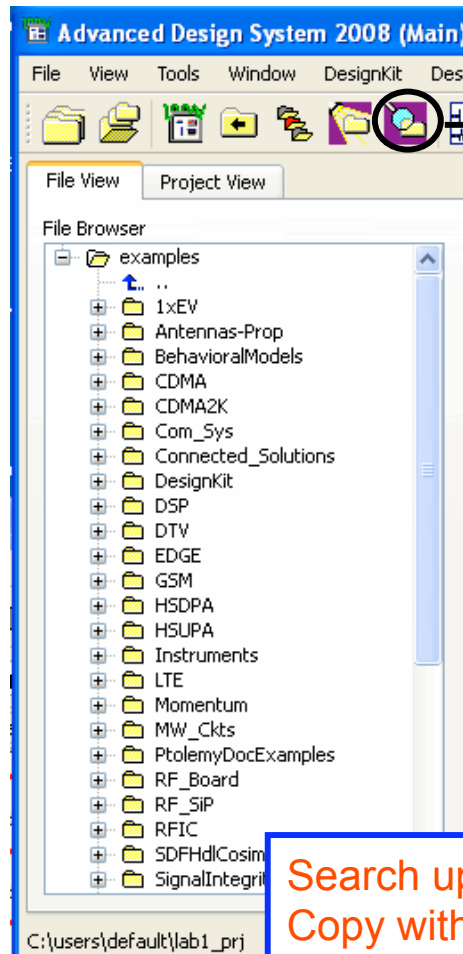
Search up to four words (implies AND) but do not combine AND and OR.
Copy with hierarchy or view (no hierarchy). Also, copy Data Displays!

The KC: Knowledge Center

Supported customers receive a Username and Password. Search the Knowledge Center by topic …

You get access to more documents, downloads, and the latest examples…

Design Guides: design + simulation

- **Filter DG** is a synthesis tool (E-syn replacement).
- **Smith Chart** utility can be for matching.
- Other DGs are available - some are like templates.
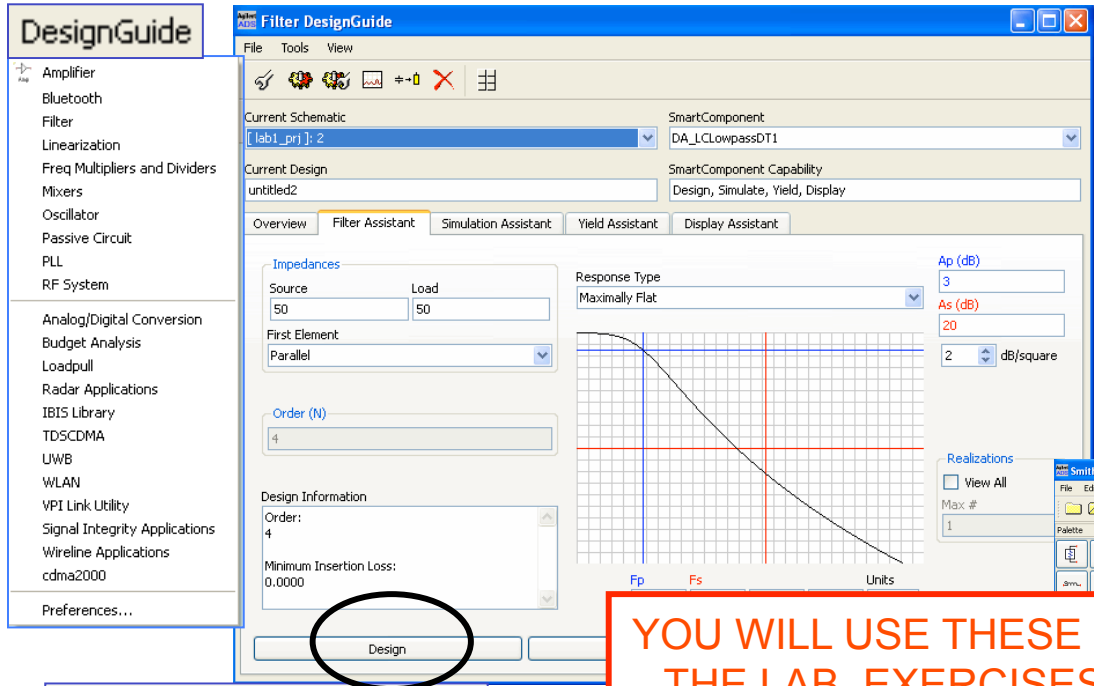
YOU WILL USE THESE IN THE LAB EXERCISES!
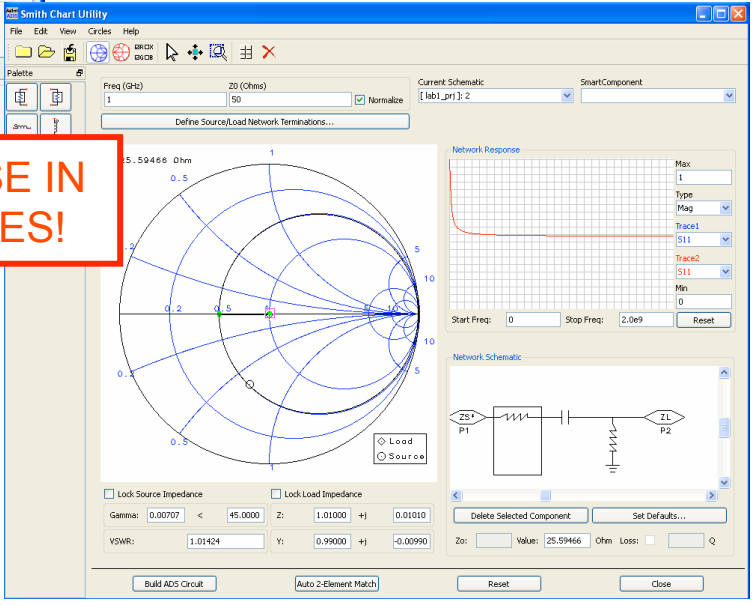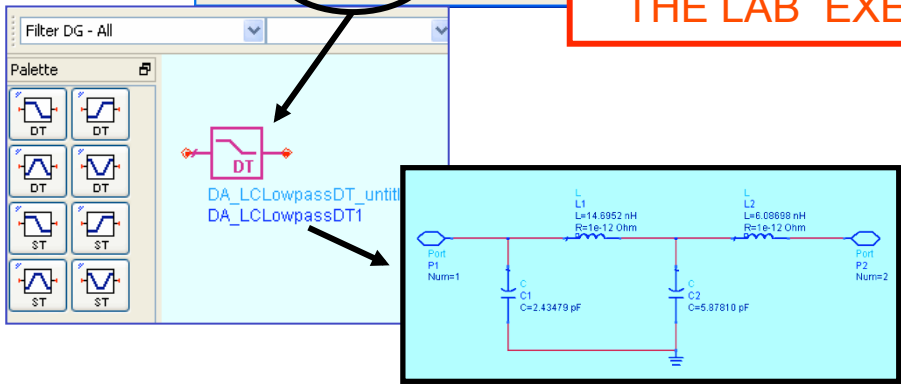
Smart Simulation Wizard

Schematic Icon

Predefined simulation setups for devices or circuits you build.
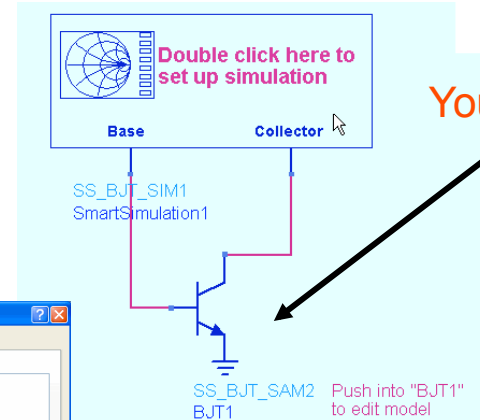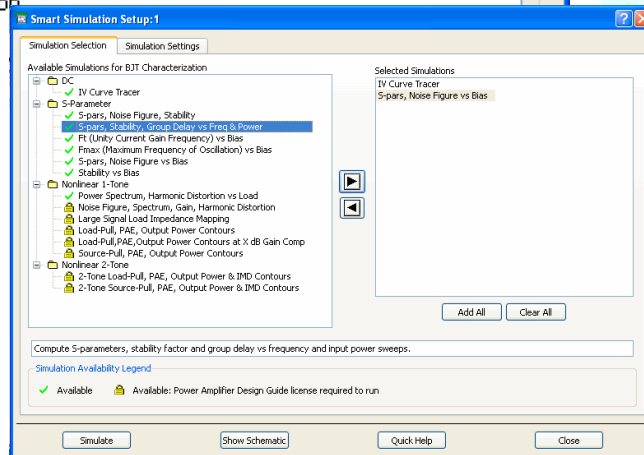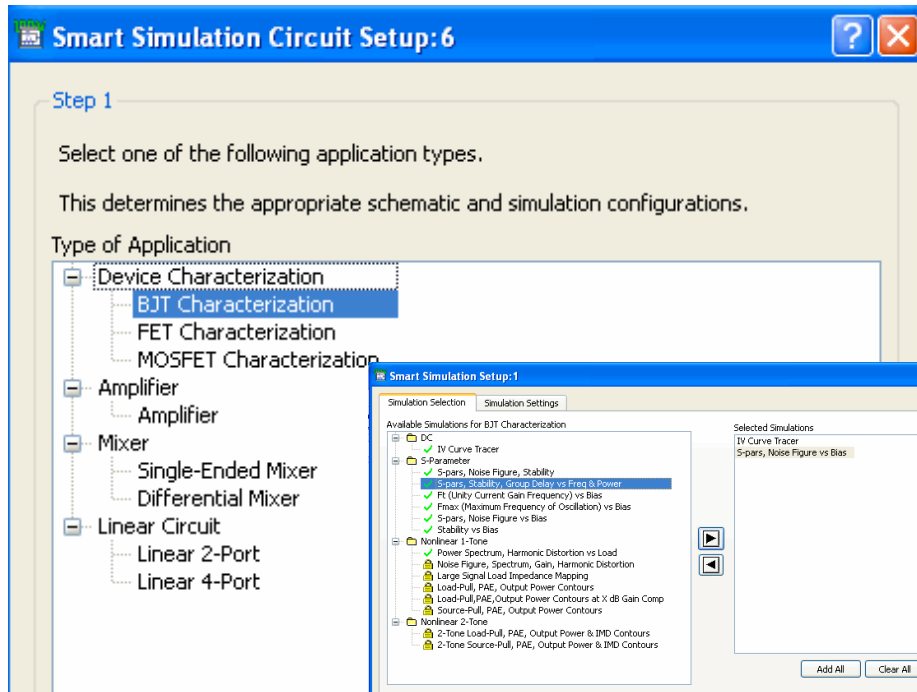
Your device or circuit goes here.

Data is automatically plotted in predefined Data Display template:
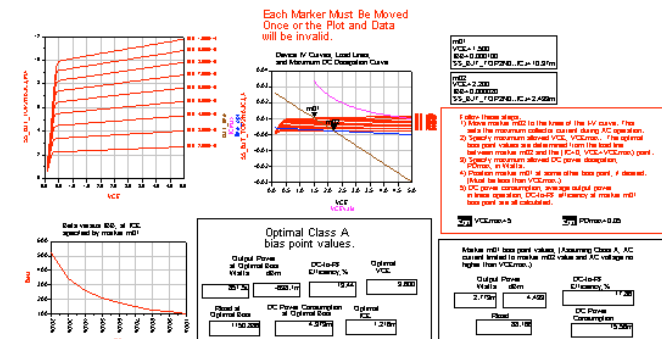
Simulation setups or Design Guide setups (license required). Type in the values to be passed into the setup. Simulate and the results are automatically plotted in data display templates.

Test Labs and Test Benches

For S-parameter analysis, you save simulation time and memory! ADS now contains examples of using these.

NOTE: Series IV users may remember this concept.

S_ParamTestLab
TestLab1
Start=0.1 GHz
Stop=10 GHz
Step=0.1 GHz
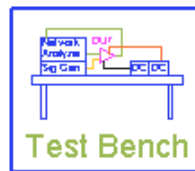
TEST LAB: simulator is flagged to use Test Benches for analysis. **TEST BENCHES** are separate circuits with separate port assignments – **no need to put all circuits on one schematic with consecutive port numbers** for multi-stage designs.

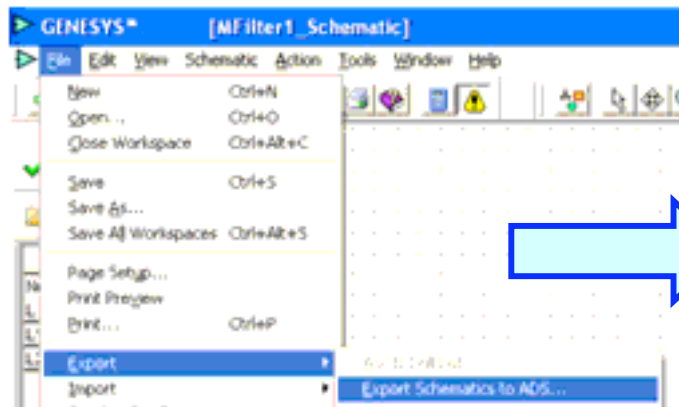**For example,** set up an optimization to create a simulation model equal to the measured S-parameters.

Measured_tb
TB1

Modeled_tb
TB2

More on this later: in the S-parameter analysis section.
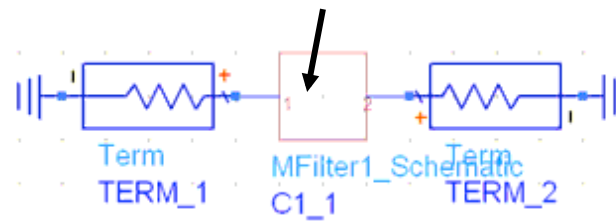
ADS link with GENESYS…

Use ADS with GENESYS or Spectrasys – some rules apply.
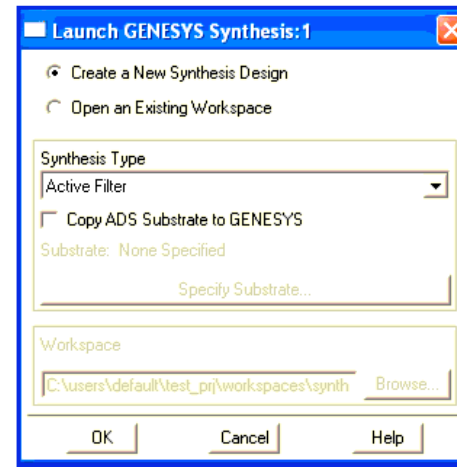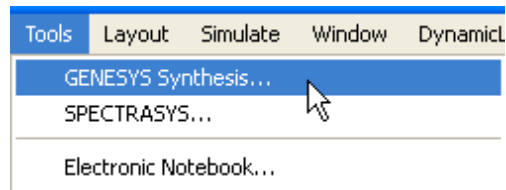
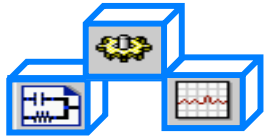Purpose: transfer GENESYS schematics to ADS for analysis.

GENESYS circuit:

Now, the lab exercise!

NOTE: ADS prj and GENESYS workspace must be opened on PC at the same time. Also, models not mapped in both will be reported.
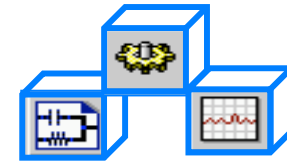
What the lab is about ...

Lab 1:

Circuit Simulation Fundamentals

Learn Circuit Simulation Basics

- Create a new project: lab1
- Build a low-pass filter in the schematic
- S-parameter simulation
- Display the results: S21 transmission
- Tune the filter response

- Create another schematic in lab1 project
- Copy an example RFIC amplifier into lab1 project
- Setup the Vin and Vout node voltage names
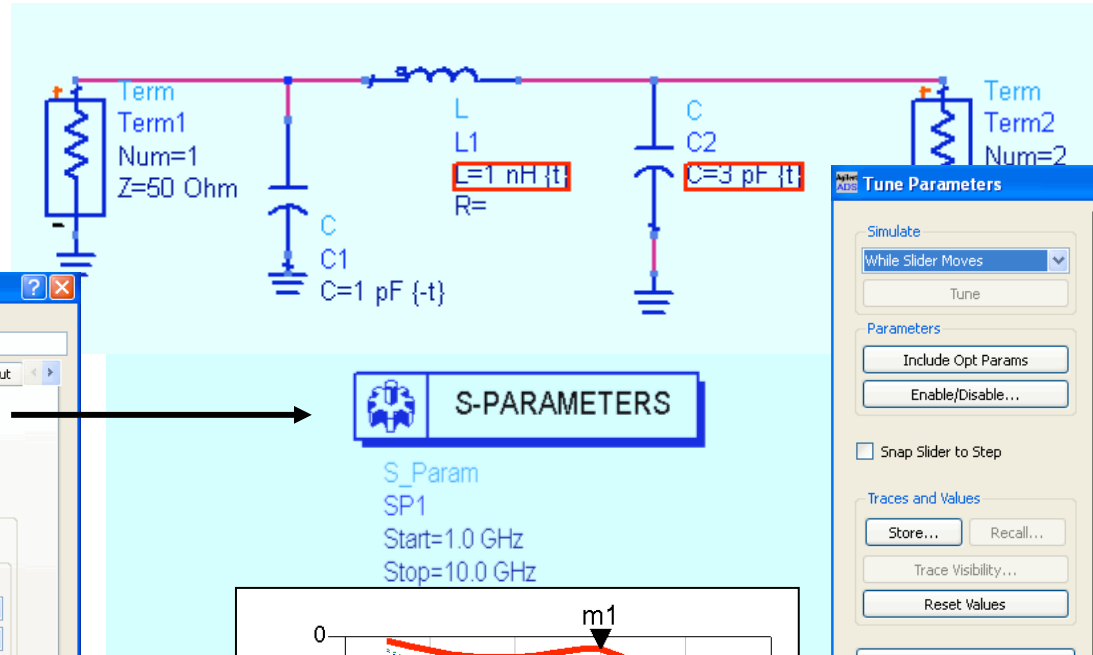- Perform a Harmonic Balance simulation
- Display the spectrum and gain equation results

NOTE: Lab 1 can be skipped if you already know the basic operation of ADS. Or, try the extra exercises at the end of the lab.

MORE ON THE LAB:

Low-pass filter: S-parameter simulation

Build the filter, simulate and tune the filter parameters...

SET UP THE
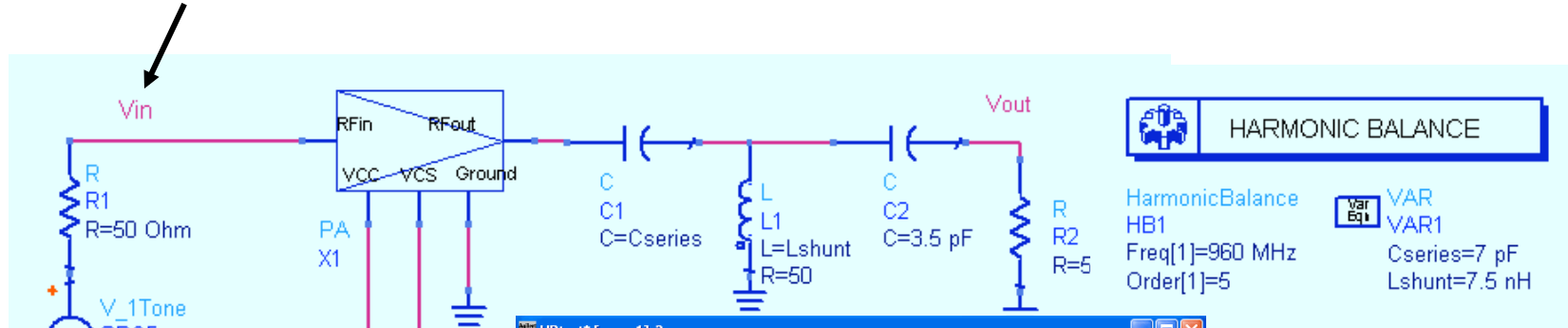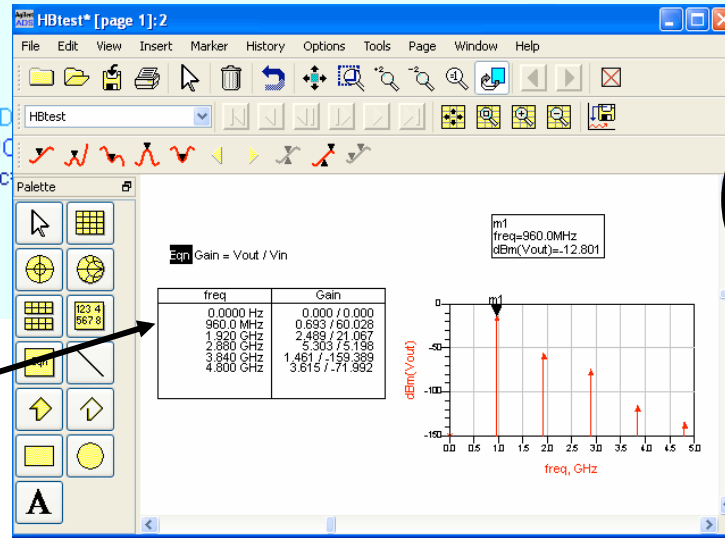SIMULATION:

RFIC example: Harmonic Balance simulation

With **Vin** and **Vout** labeled, check the sub-circuit, and simulate.

Write and list the gain equation.

Start the lab now!

NOTE: If you finish early, try the lab Extra Exercises!