

# Ansoft Designer Training



# Agenda

- ◆ Overview of Ansoft Designer GUI
- ◆ Using interface
  - ◆ Circuit Design
    - ◆ Building schematic
    - ◆ Analysis circuit
    - ◆ Tuning
    - ◆ Optimization
    - ◆ Statistical analysis
    - ◆ Layout basic
    - ◆ Building Hierarchy
  - ◆ LNA Design
    - ◆ Input/output Matching – Smith tool
    - ◆ Nonlinear analysis
      - ◆ RF 1 tone
      - ◆ DC analysis
      - ◆ RF 2 tones
      - ◆ modulation
  - ◆ Load-Pull analysis
  - ◆ Oscillator Analysis
    - ◆ Transient Analysis
    - ◆ Harmonic balance and Phase noise
- ◆ Field Solver Design basic
  - ◆ Create stack-up
  - ◆ Drawing geometry
  - ◆ Parameterized geometry
  - ◆ Analysis
  - ◆ Co-simulation
    - ◆ Use field solver simulation in circuit design
    - ◆ Tuning Field solver design
  - ◆ Planar EM Antenna Design

# Overview of Ansoft designer GUI



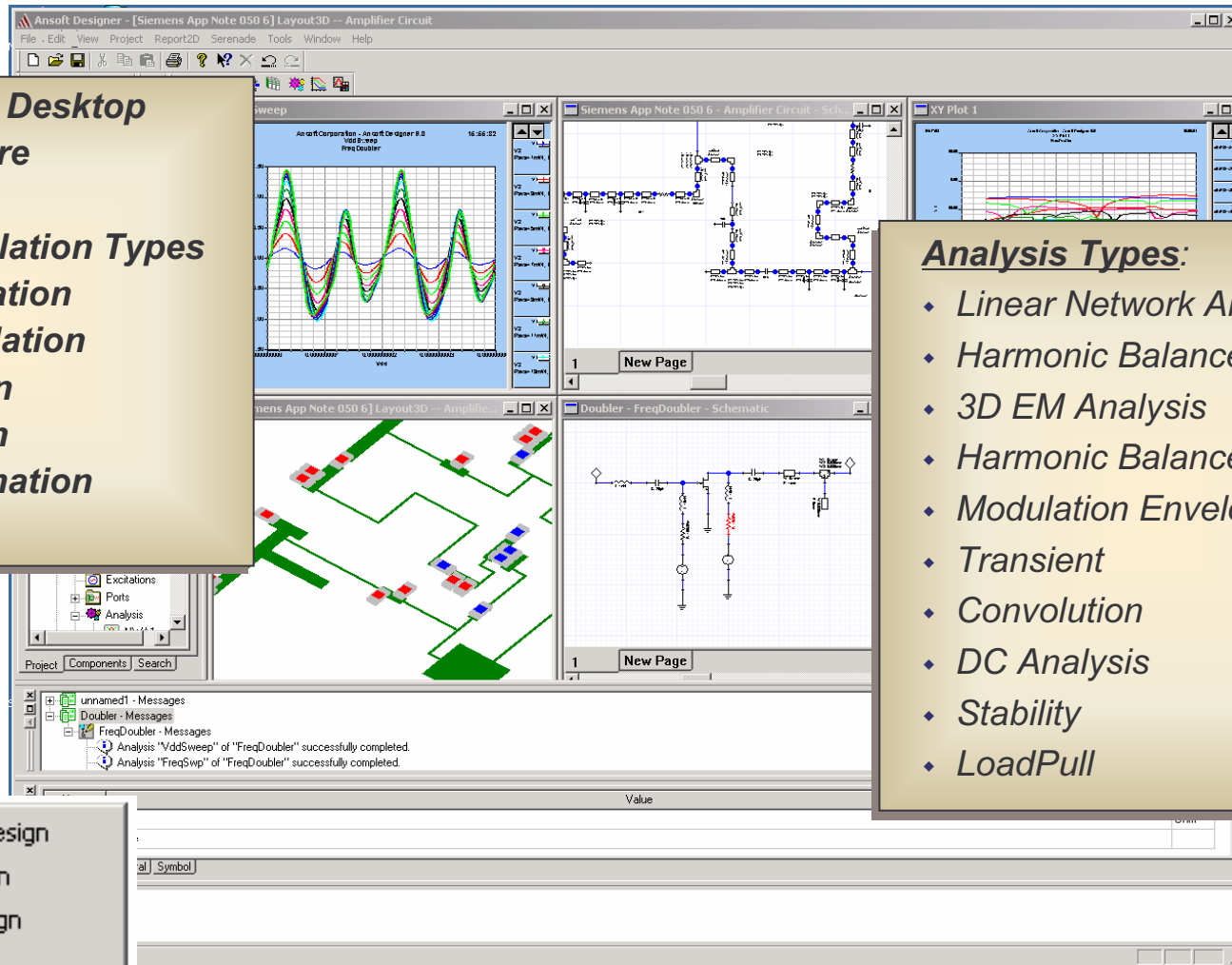
# Ansoft Designer Overview

- ◆ *Multi Window Desktop*
- ◆ *Design Capture*
- ◆ *Layout*
- ◆ *Multiple Simulation Types*
- ◆ *Circuit Simulation*
- ◆ *System Simulation*
- ◆ *EM Simulation*
- ◆ *Co-Simulation*
- ◆ *Design Automation*
- ◆ *Synthesis*

## *Analysis Types:*

- ◆ *Linear Network Analysis*
- ◆ *Harmonic Balance*
- ◆ *3D EM Analysis*
- ◆ *Harmonic Balance Oscillator*
- ◆ *Modulation Envelope*
- ◆ *Transient*
- ◆ *Convolution*
- ◆ *DC Analysis*
- ◆ *Stability*
- ◆ *LoadPull*

- ◆ *Insert Planar EM Design*
- ◆ *Insert Circuit Design*
- ◆ *Insert System Design*
- ◆ *Insert Filter Design*
- ◆ *Insert Documentation File*



ANSOFT CORPORATION

# Ansoft Designer Windows

The screenshot displays the Ansoft Designer software interface for a BandPass filter design. The interface is divided into several key windows:

- Property Window:** Located on the left, it shows a table of parameters for the design. The table is as follows:

Name	Value	Unit
W	w1	
S	s1	
P	p1	
SUB	Duroid5880	
TRL	TRL	
CoSimSta...	Layout stackup	
CoSimulator	CurrentProduct	
Status	Active	
- Project Manager:** Also on the left, it shows a hierarchical tree view of the project structure, including folders for BandPass, Data, Excitations, Ports, Analysis, Results, and Definitions.
- 3D Layout Viewer:** The top-left pane shows a 3D perspective view of the filter layout, with a coordinate system (X, Y, Z) visible.
- Schematic Editor:** The top-right pane shows the circuit schematic, featuring various components like ports, wave sources, and transmission lines.
- Layout Editor:** The bottom-left pane shows a top-down 2D layout view of the filter components, with ports labeled Port1 and Port2.
- Results Window:** The bottom-right pane displays an XY Plot showing the filter's frequency response. The x-axis is labeled 'F [GHz]' and ranges from 4.00 to 6.00. The y-axis is labeled 'Y1' and ranges from -40.00 to 0.00. Two curves are plotted: a red curve representing the magnitude response and a blue curve representing the phase response.
- Message Window:** Located at the bottom left, it shows the status 'Ready'.
- Progress Window:** Located at the bottom right, it shows the status 'Ready'.

At the bottom of the image, the text 'ANSOFT CORPORATION' is visible, along with a stylized graphic of a computer monitor and keyboard.

# Project Manager Window

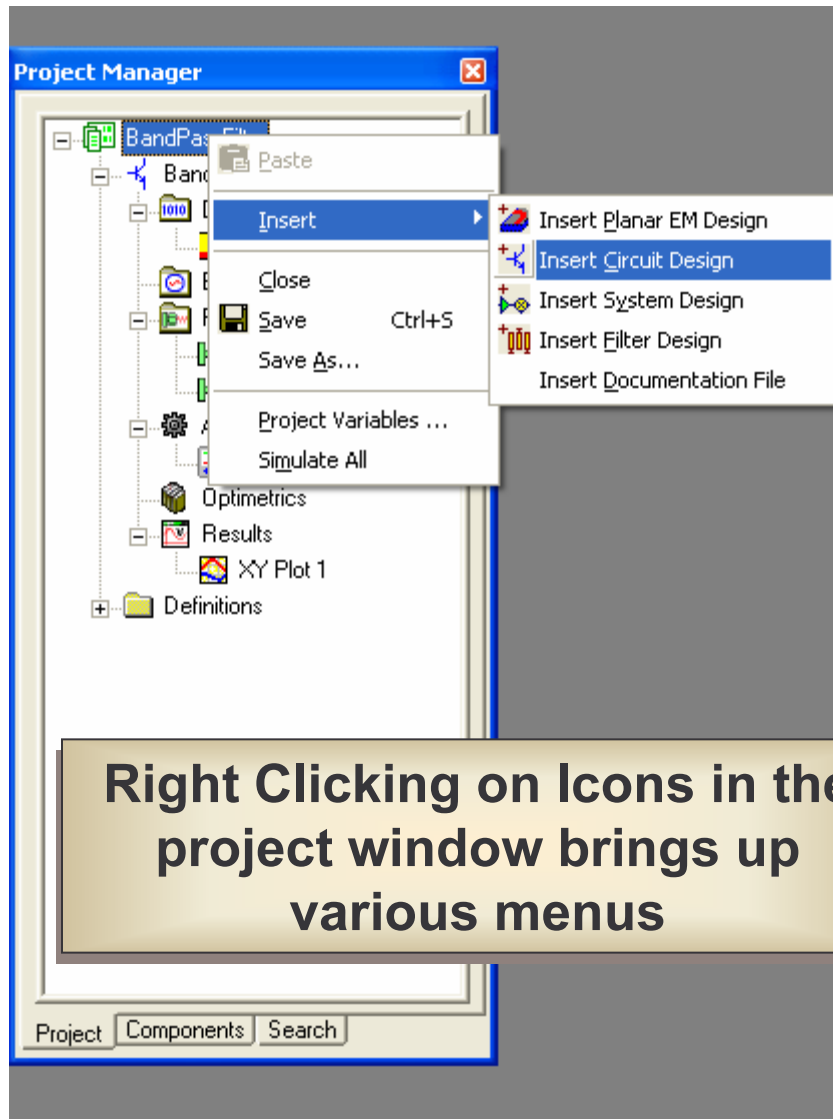
The screenshot shows the Project Manager window with a tree view of project components. Callout boxes provide instructions for interacting with the tree:

- Click on the +/- to expand or collapse the tree**: Points to the minus sign icon next to the BandPassFilter\* folder.
- Double click on the design icon to open the schematic editor**: Points to the design icon (a blue 'X') next to the Export\_Filter1\* folder.
- Double click on the substrate or analysis icon to open the Definition dialog for those objects**: Points to the substrate icon (a yellow and red square) next to the Alumina component.
- Double click on the graph icon to display results**: Points to the graph icon (a red and yellow square) next to the XY Plot 1 component.
- Tabs for project, Components, and Search**: Points to the 'Components' tab at the bottom of the window.

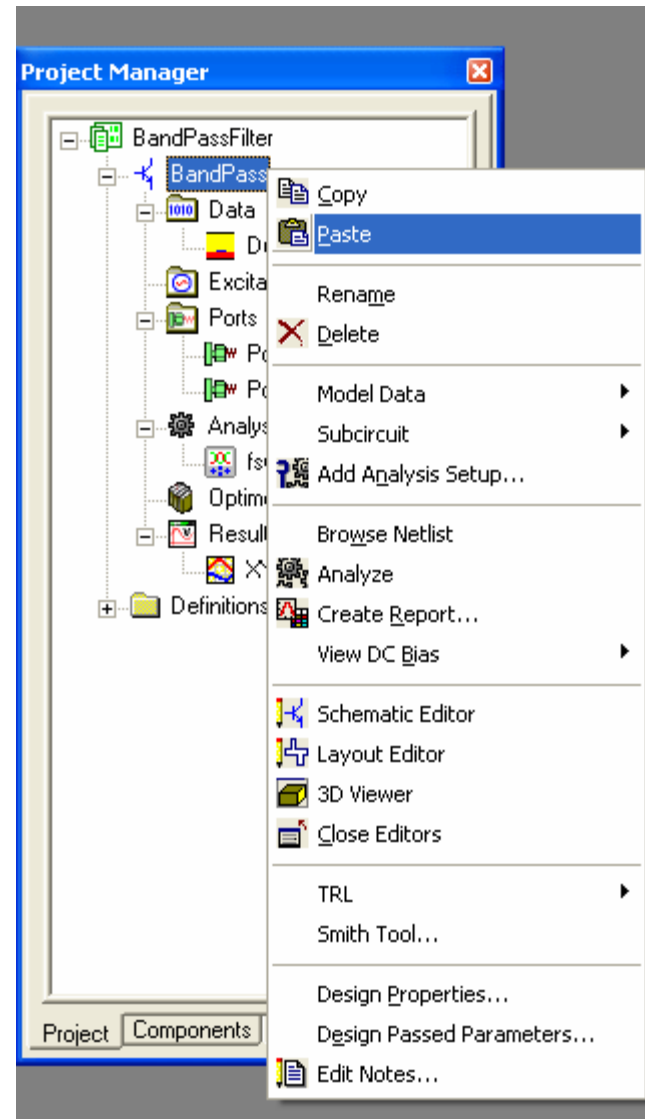
The tree view includes the following items:

- BandPassFilter\* (expanded)
  - Export\_Filter1\* (expanded)
    - Data (expanded)
      - Alumina (selected)
    - Excitations
  - Ports (expanded)
    - Port1
    - Port2
  - Analysis (expanded)
    - NWA1
  - Optimetrics
  - Results (expanded)
    - XY Plot 1
  - ToolObjects
  - Definitions

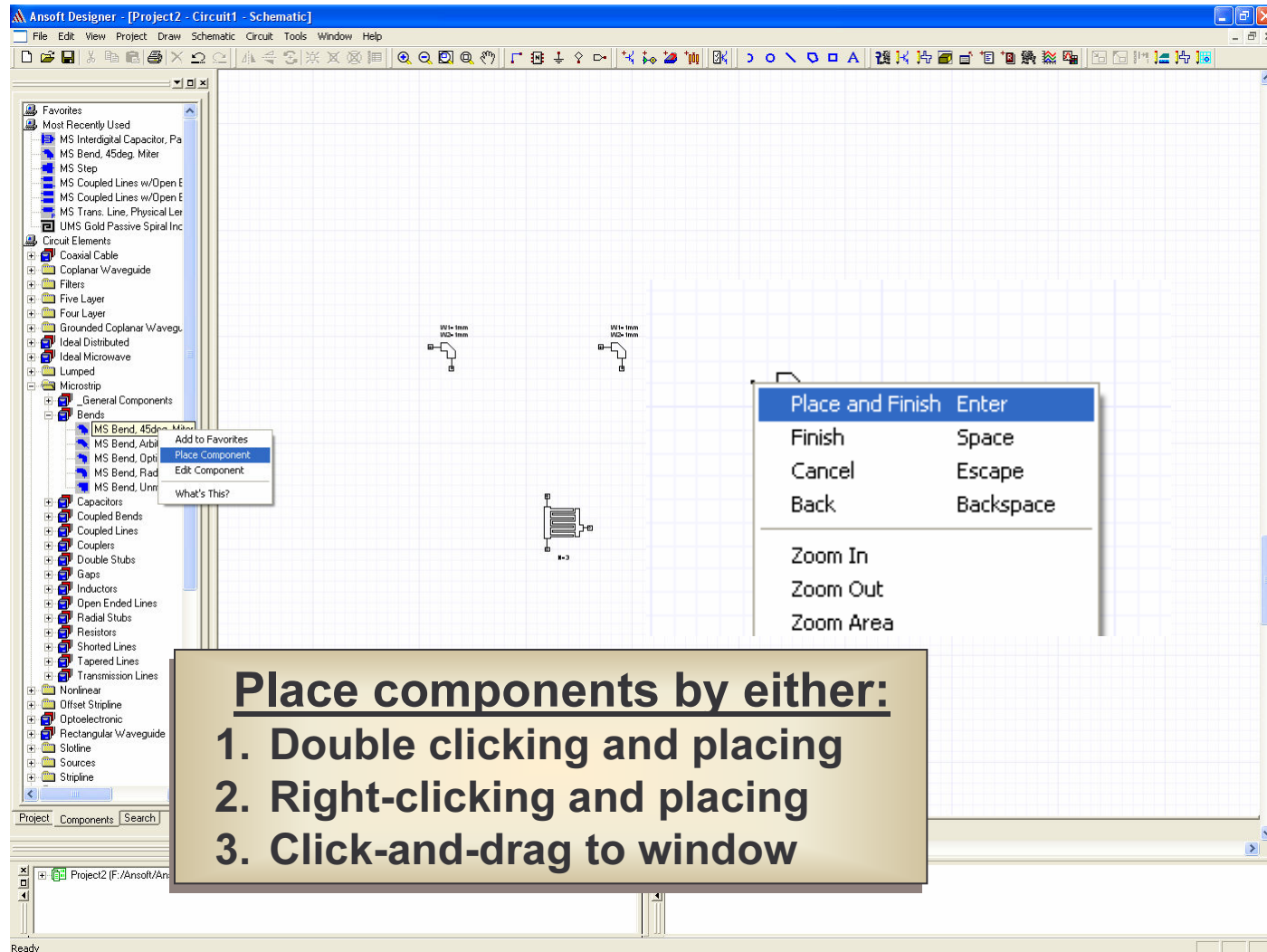
# Right Click Pop Up Menu



**Right Clicking on Icons in the project window brings up various menus**



# Component Tab





# Search Tab

The screenshot shows the Ansoft Designer Project Manager interface. The search tab is active, displaying search results for the query "ms\*trl". The results list includes MS4MLTRL, MS4TRL, MS5MLTRL, MSTRL, MSTRL\_Ref, MSTRLE, MSTRLE\_Ref, and MSTRLED. The MS4MLTRL component is selected. The interface also shows the search criteria, search and insert buttons, and a list of libraries to search in. A blue oval highlights the search tab area. A blue arrow points from the search tab to the right, where a schematic diagram of a bandpass filter is shown. The schematic includes two input ports labeled 1 and 2, and two output ports. The filter is represented by a rectangular block with a blue line connecting the input and output ports. The filter parameters are listed as W1=w1, W2=w2, W=w2, S=s2, and P=p2. The status bar at the bottom indicates that the analysis "tsweep" of "BandPass" was successfully completed.

Search by name, type, or partial names

# Property Window

Property windows contains tabs which address different types of properties, such as General, Symbol, Variables, etc.

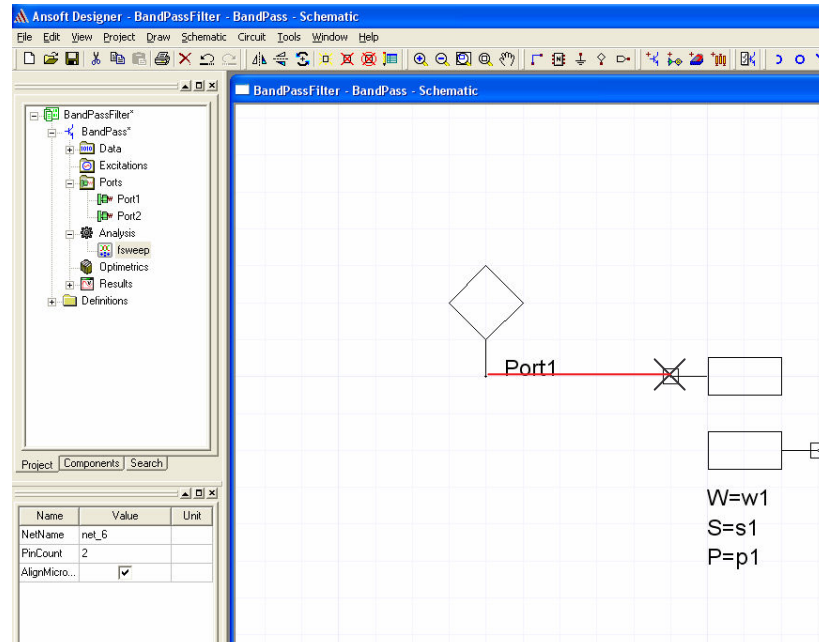
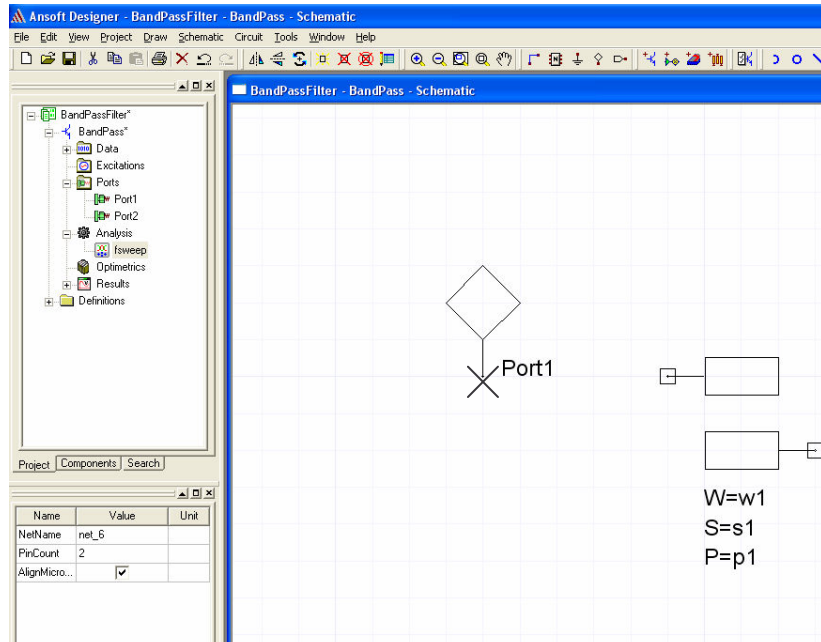
The screenshot shows the Ansoft Designer interface with a schematic containing a component labeled 'Port1'. A 'Properties' dialog box is open, displaying the 'Value' tab. The dialog box contains a table with the following data:

Name	Value	Unit	Description	Override
W	1	mm	Conductor width	<input type="checkbox"/>
S	1	mm	Conductor spacing	<input type="checkbox"/>
P	10	mm	Physical length	<input type="checkbox"/>
SUB	FR4		Substrate name	<input checked="" type="checkbox"/>
TRL	TRL		TRL properties	<input checked="" type="checkbox"/>
CoSimulator	Circuit			<input type="checkbox"/>
CoSimStackup	Layout stackup			<input type="checkbox"/>
Status	Active			<input type="checkbox"/>

Below the table, there is a 'Show Hidden' checkbox and 'OK' and 'Cancel' buttons. The background schematic shows a diamond-shaped component labeled 'Port1' connected to two rectangular components. Text next to the components indicates 'W=1mm', 'S=1mm', and 'P=10mm'. The software interface includes a menu bar, a toolbar, a project tree on the left, and a parameter table at the bottom left.

The property window is dockable, or can be brought up by double-clicking on a component

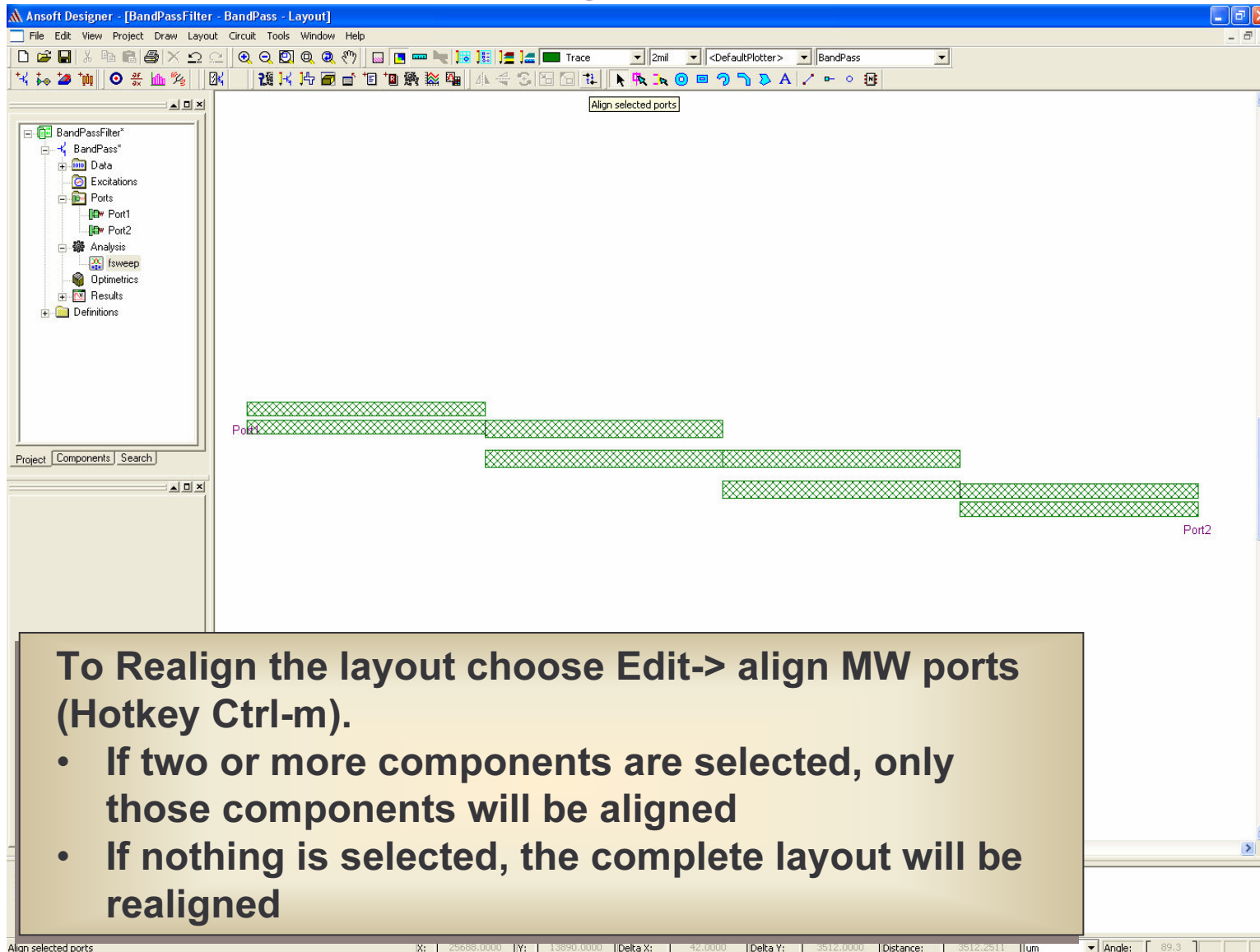
# Schematic Window



## To wire a component:

1. Move mouse to hover over a node
2. The cursor changes to an "X"
3. Click to make connection
4. Move cursor to another node (or a wire)
5. When the cursor is over another connection point, it will revert back to an "X"
6. Click to make connection

# Layout Editor



The screenshot shows the Ansoft Designer interface for a BandPassFilter layout. The main workspace displays a circuit layout with several green rectangular components. Two ports, Port1 and Port2, are highlighted with a green cross-hatch pattern. A text box in the center of the workspace reads "Align selected ports". The left sidebar shows a project tree with folders for Data, Excitations, Ports, Analysis, Optimetrics, Results, and Definitions. The bottom status bar displays coordinates and dimensions for the selected components.

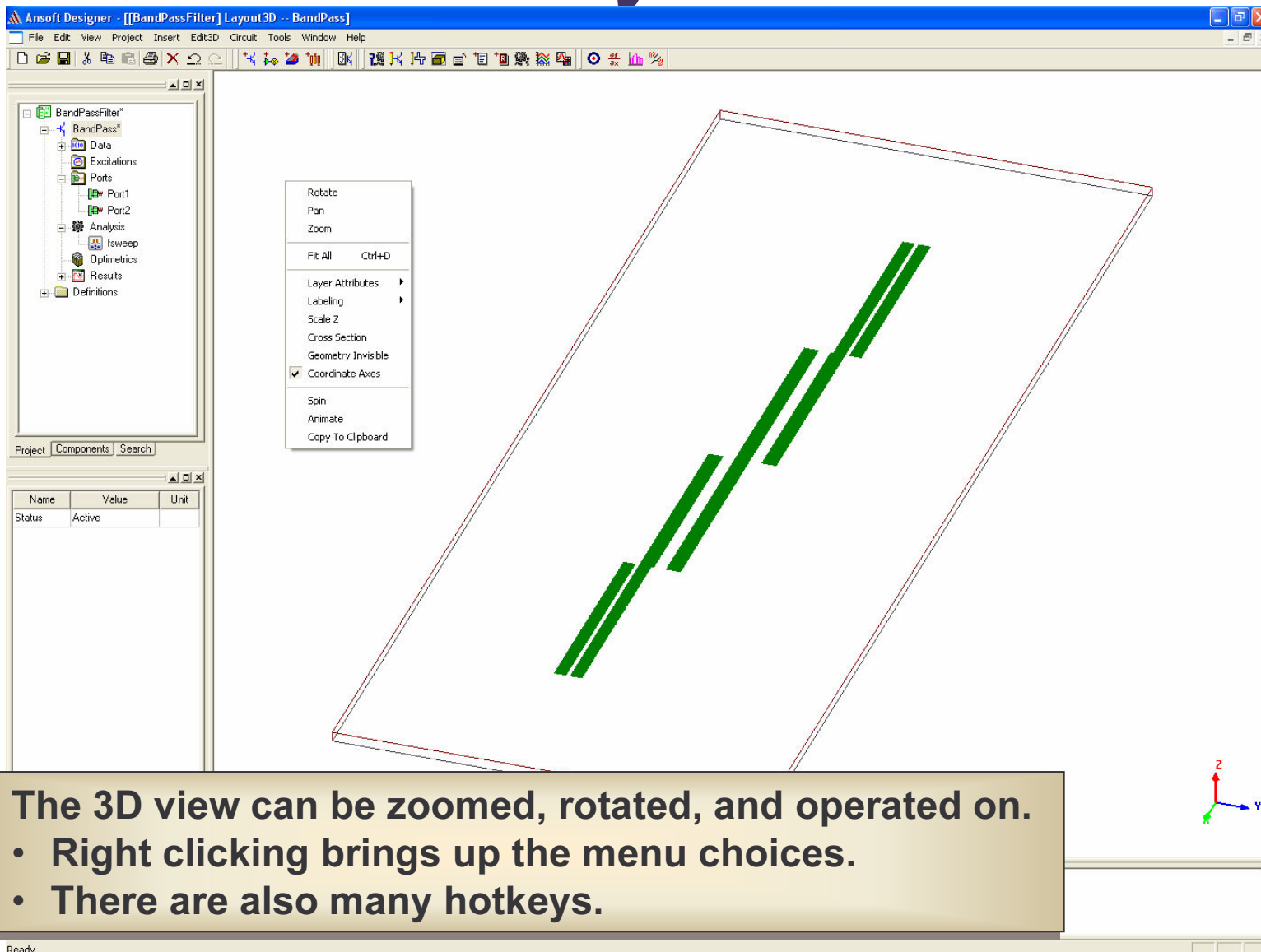
**To Realign the layout choose Edit-> align MW ports (Hotkey Ctrl-m).**

- If two or more components are selected, only those components will be aligned
- If nothing is selected, the complete layout will be realigned

Align selected ports

X: 25688.0000 | Y: 13890.0000 | Delta X: 42.0000 | Delta Y: 3512.0000 | Distance: 3512.2511 | um | Angle: 89.3

# 3D Layout Viewer



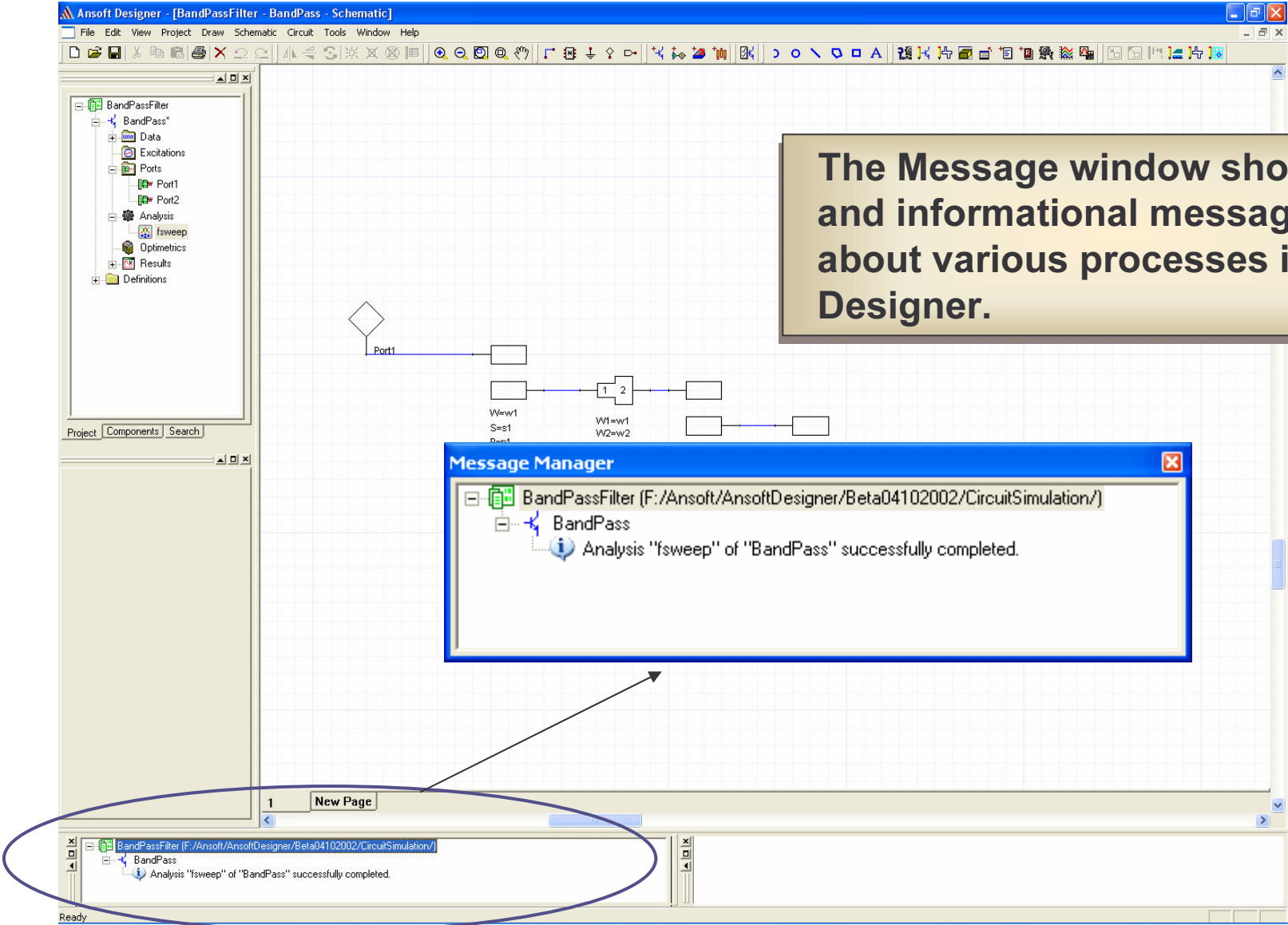
# Progress Window

The screenshot displays the Ansoft Designer interface. The main workspace shows a schematic diagram of a bandpass filter. The circuit starts with a diamond-shaped component labeled 'Port1'. This is followed by a rectangular component with parameters  $W=w1$ ,  $S=s1$ , and  $P=p1$ . The signal then passes through a two-port component with ports labeled '1' and '2' and parameters  $W1=w1$  and  $W2=w2$ . This is followed by another rectangular component with parameters  $W=w2$ ,  $S=s2$ , and  $P=p2$ . The circuit continues through another two-port component with ports labeled '2' and '1' and parameters  $W1=w1$  and  $W2=w2$ . Finally, it ends with a rectangular component with parameters  $W=w1$ ,  $S=s1$ , and  $P=p1$ , which is connected to a diamond-shaped component labeled 'Port2'. A 'Progress' window is overlaid on the schematic, titled 'BandPass - fsweep on Local Machine - RUNNING'. It features a red progress bar and the text '22 of 100', with an 'Abort' button below it. The left sidebar shows a project tree with folders for 'Data', 'Excitations', 'Ports', 'Analysis', 'fsweep', 'Dplometrics', 'Results', and 'Definitions'. The bottom status bar indicates 'Ready'.

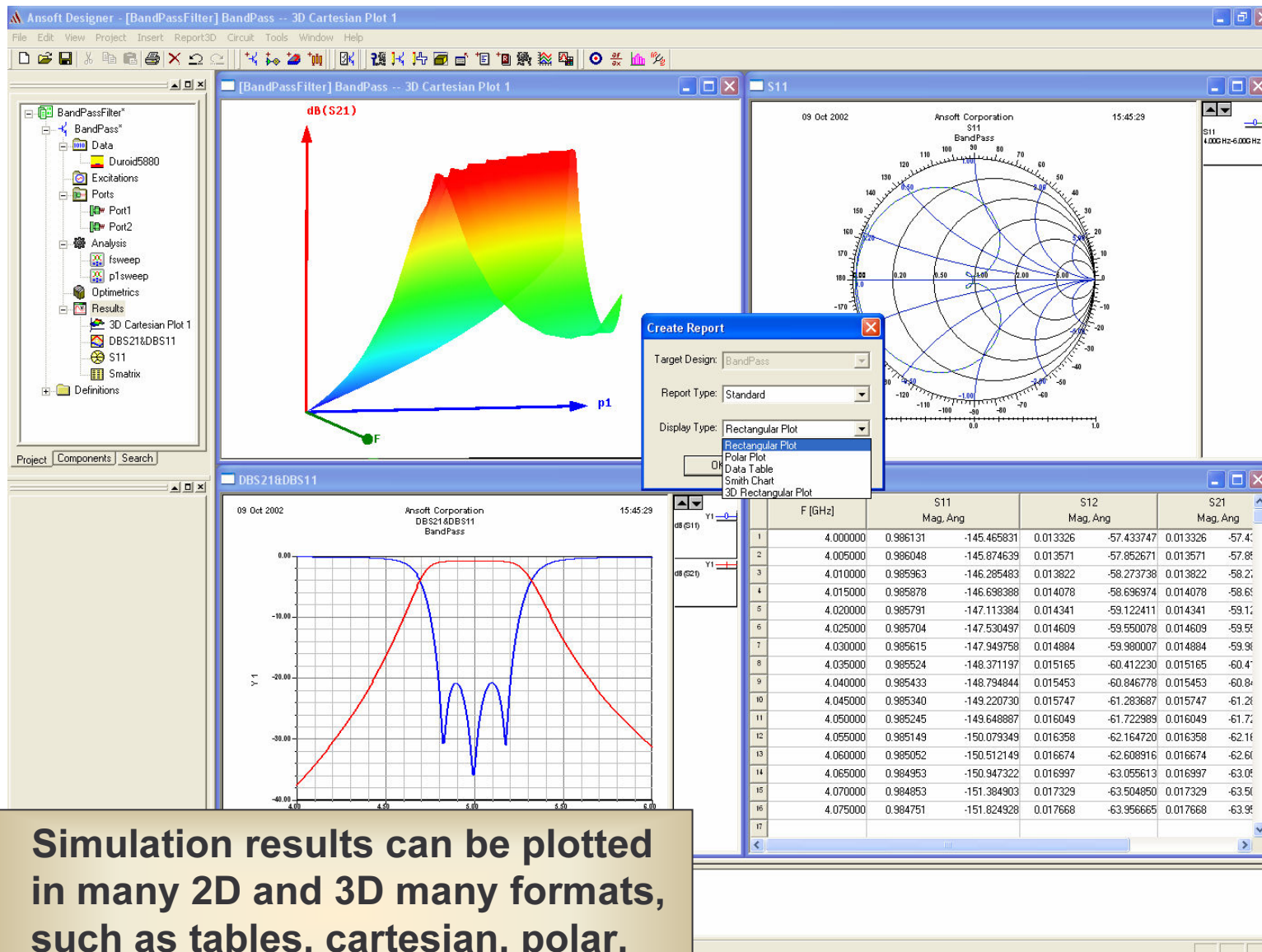
**The progress window is also a detachable window. This window tells you the status of a simulation. It comes up automatically when a simulation is performed**

# Message Window

The Message window shows error and informational messages about various processes in Designer.



# Results Window

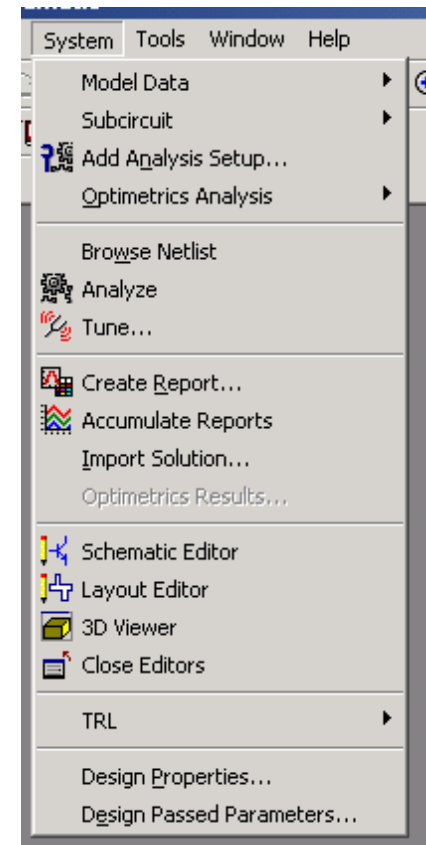
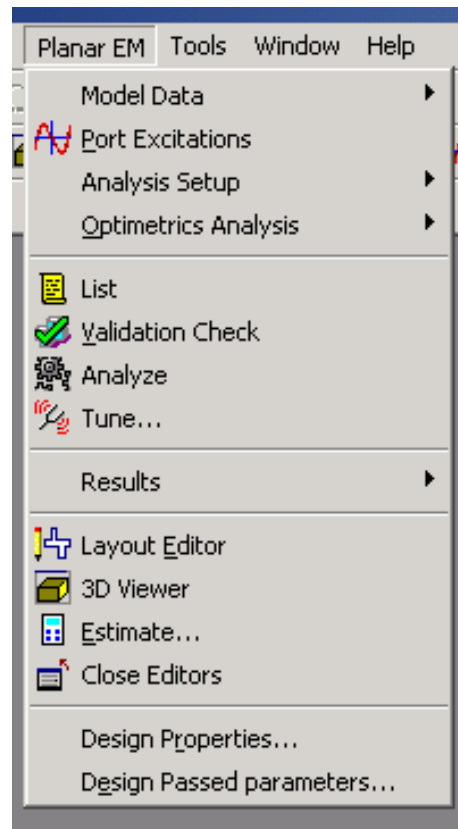
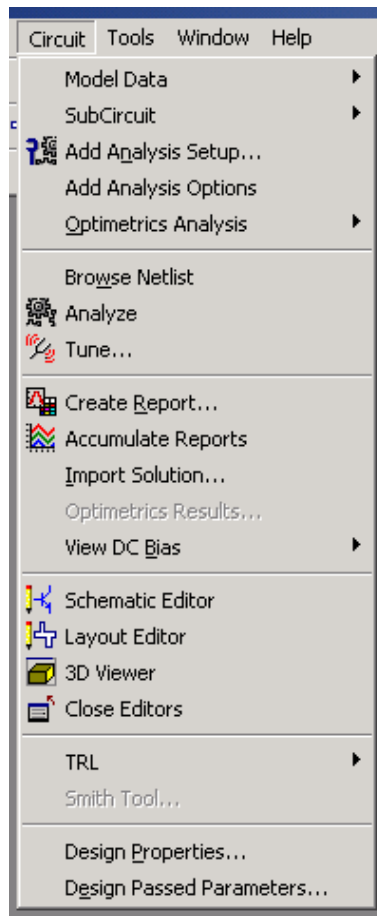


Simulation results can be plotted in many 2D and 3D many formats, such as tables, cartesian, polar, smith, and others.



# Dynamic Menus

Menus change dynamically depending on which Design window is highlighted (Circuit, Planar EM or System)

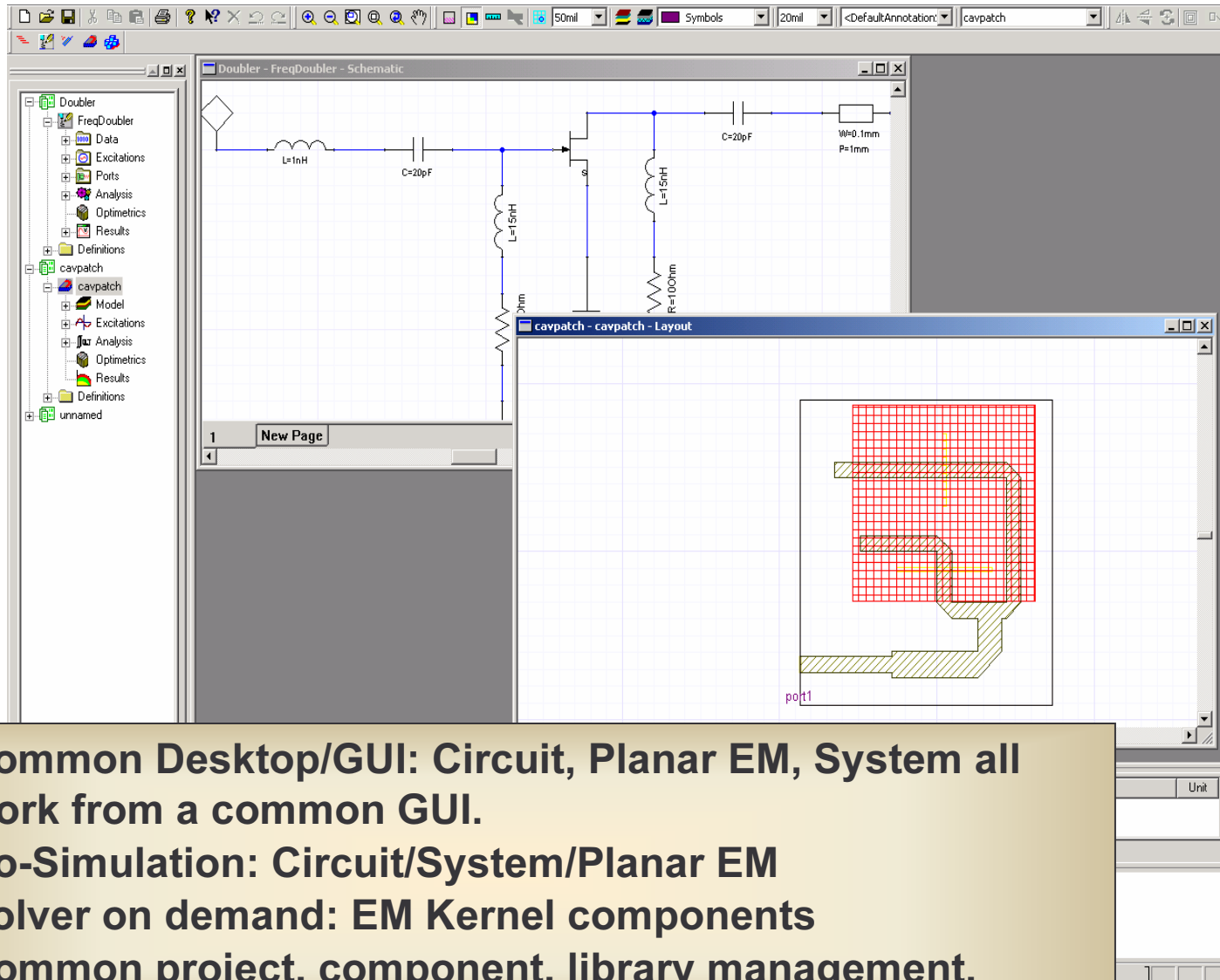


# Schematic/Layout Integration

The screenshot displays the Ansoft Designer interface with two main windows: 'Sch\_Edit\_Test - SerenadeCircuit1 - Schematic' and 'deCircuit1 - Layout'. The schematic window shows a circuit diagram with two ports, Port1 and Port2, connected by a transmission line. The transmission line is labeled with dimensions: W1=1mm, W2=1mm, W=1mm, and P=10mm. The layout window shows a green grid representing the physical layout of the circuit, with a blue arrow indicating the connection between the schematic and the layout. The interface includes a menu bar, a toolbar, and a project tree on the left side.

- ◆ One to One Component Correspondence
- ◆ User can work directly in Layout
- ◆ Components placed in schematic automatically appear in Layout and vice-versa
- ◆ Property edits in Layout are reflected in Schematic

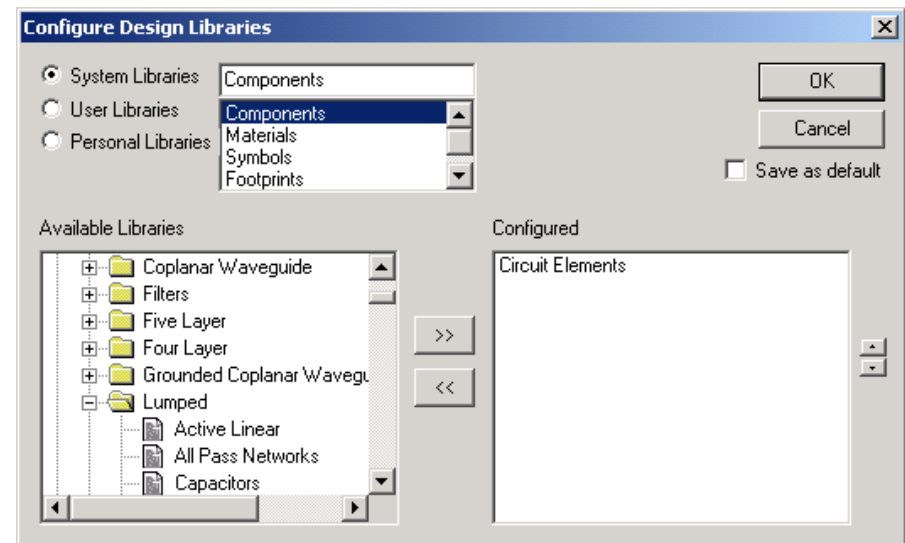
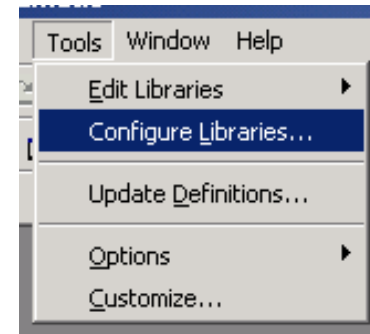
# Simulation Tool Integration



- ◆ **Common Desktop/GUI: Circuit, Planar EM, System all work from a common GUI.**
- ◆ **Co-Simulation: Circuit/System/Planar EM**
- ◆ **Solver on demand: EM Kernel components**
- ◆ **Common project, component, library management.**

# Design Automation Component Libraries

- ◆ *Components, Materials, Symbols etc. are organized into libraries.*
- ◆ *Libraries are stored in*  
*Ansoft\Designer\syslib*  
*Ansoft\Designer\userlib*  
*Ansoft\Designer\PersonalLib*
- ◆ *Specific Libraries are “configured” for each project making the Components, Materials etc. available for use in that project.*



# Ansoft Designer File Names

- ♦ *Important File Extensions for Ansoft Designer:*
  - ♦ *.adsn*      *Project File*
  - ♦ *.aclb*      *Component Lib*
  - ♦ *aflb*        *Footprint Lib*
  - ♦ *.asty*      *Technology File*
  - ♦ *.aslb*      *Symbol Lib*
  - ♦ *.asol*      *Solution Data File*
  - ♦ *.amat*      *Material Lib*

Every Project created is saved on disk as an *.adsn* file (AnsoftDesign file)

Ansoft Designer automatically creates a File Folder named "*ProjectName.results*" to hold the results files, netlist etc. for the project.

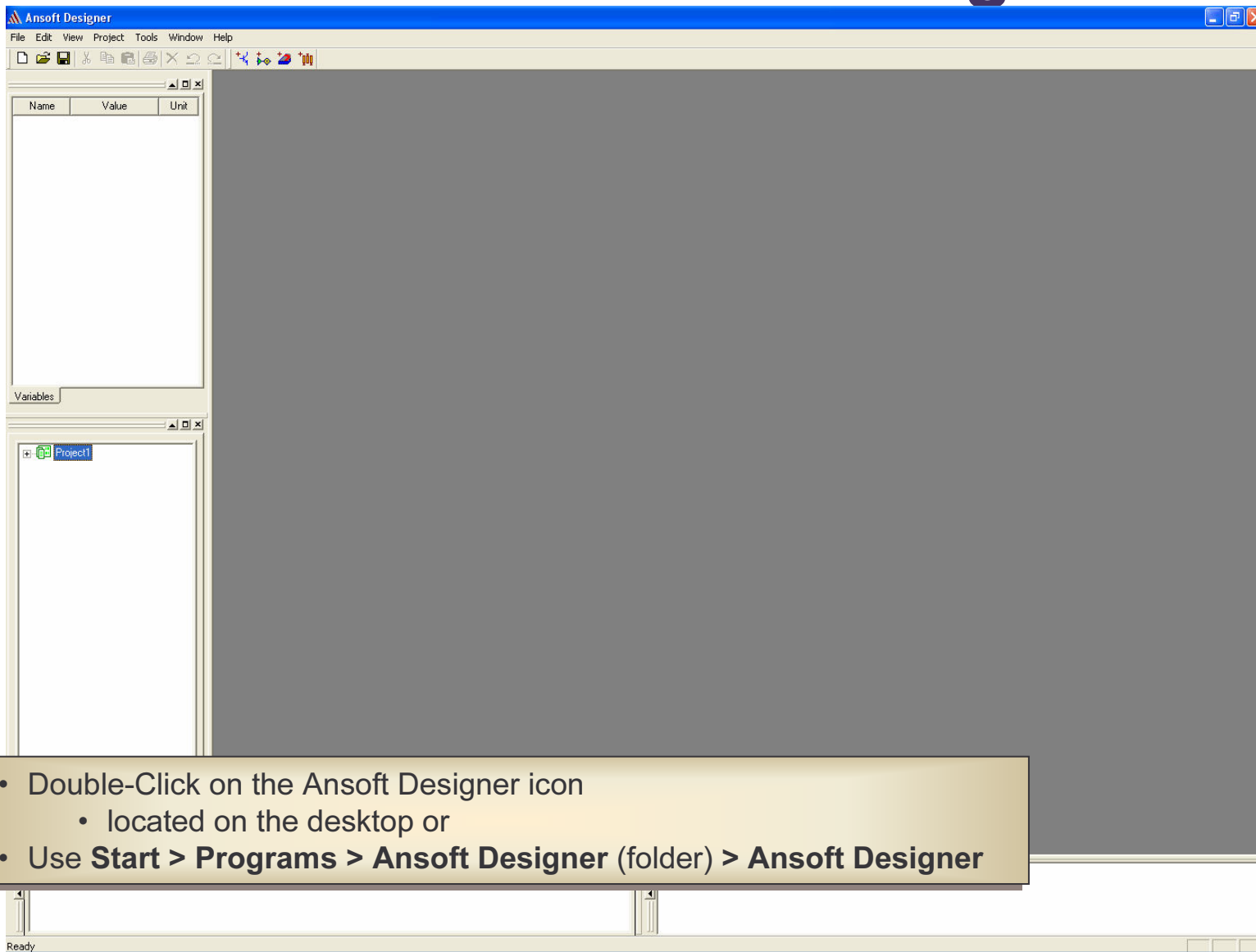
# Exercise 1: Using The Interface

*Building a Low Pass Filter*

# Exercice overview

- Open Designer
- Insert Circuit Design
  - Select Technology file
  - View data loaded by technology file
  - Save technology file
- Insert components
  - Move copy and paste
  - 6 Microstrip transmission lines
  - 2 smc capacitors Philips library
  - 2 grounds
  - 2 Microstrip Tees
  - 1 smc inductor toko library
  - Add ports
- Set substrate parameters
  - Rename circuit
- Define variables From component or project tree
  - Wline = 0.8mm
  - Lline = 1mm
  - Cvalue = 10 (pf)
- Add analysis setup
  - Select Linear
  - Start 0.1Ghz Stop 3Ghz Step 0.01Ghz and click Add
  - Run
- Create Report
  - Add traces
  - Edit graph
- Create parametric Sweep
  - Step Cvalue from 2 to 12 step 2
- Create Report
  - Plot S21 (Cartesian)
  - Plot s21 (3D)
- Tune
  - Set Cvalue for tuning
  - Set l parameter for tuning (inductor)
  - Tune (real time)
- Optimization
  - Set L and CValue for optimization
  - Set parameters to optimize
  - db(s21) at 1GHz = -3 Weight 10
  - Db(s11) from .5Ghz to 1ghz <= -30 Weight=1
- Statistical Analysis
  - Define Parameters
  - Set both C and L for uniform distribution
- View results
- View Data and Histogram
- Layout

# Load Ansoft Designer

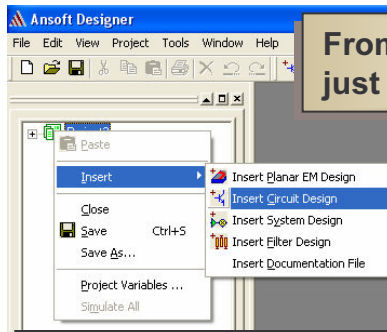


- Double-Click on the Ansoft Designer icon
  - located on the desktop or
- Use **Start > Programs > Ansoft Designer (folder) > Ansoft Designer**

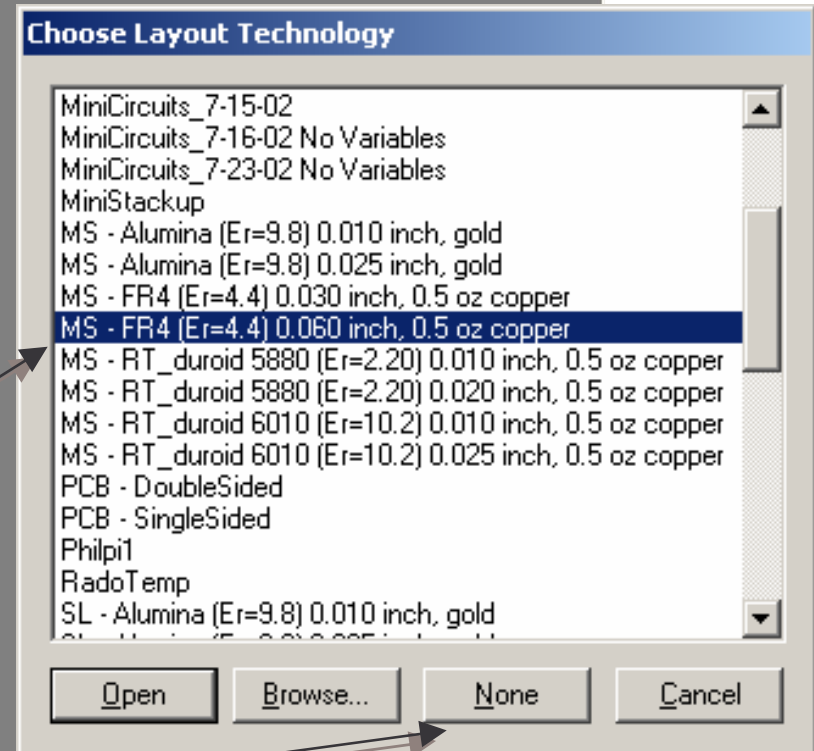


# Insert Circuit Design (Select Technology File)

From a general way of working when you don't know what to do on a specific folder, just click right on it and the list of command available for this folder will be prompted.



- Right - Click on project folder
  - Select Insert -> Circuit Design



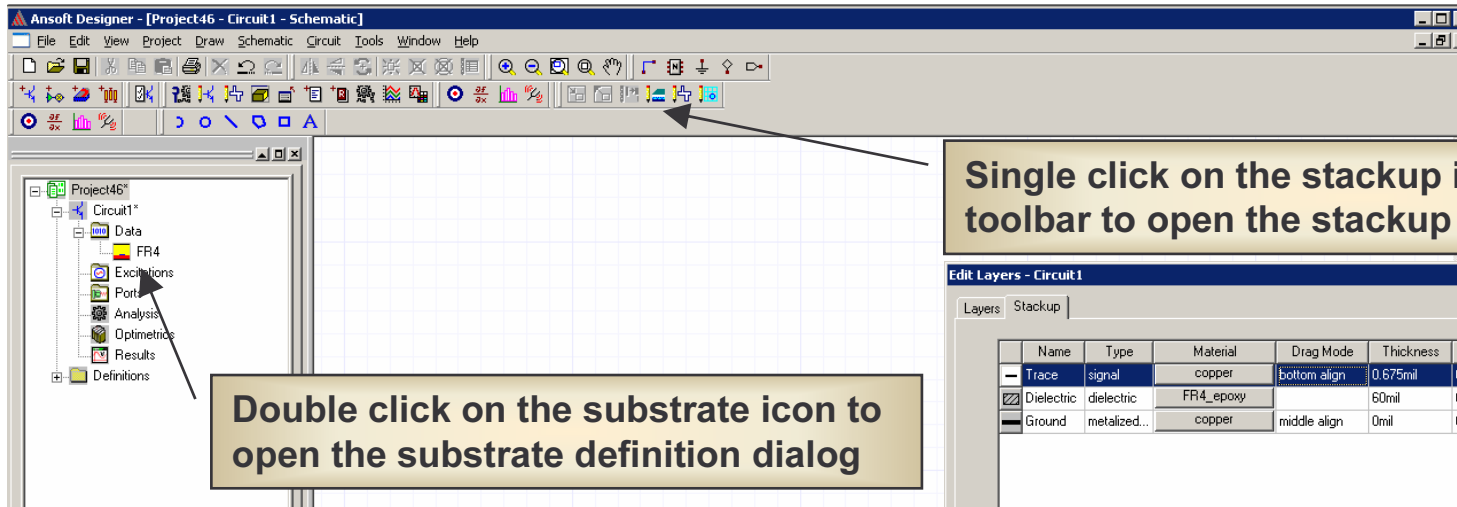
- The "Choose Layout Technology" appears
- Select FR4 .060in
- Click "Open"

Note that if you don't want the set of definitions that come with a technology file, you can choose the **None** button. This is useful for basic concept designs without any manufacturing or substrate information.

# Technology file

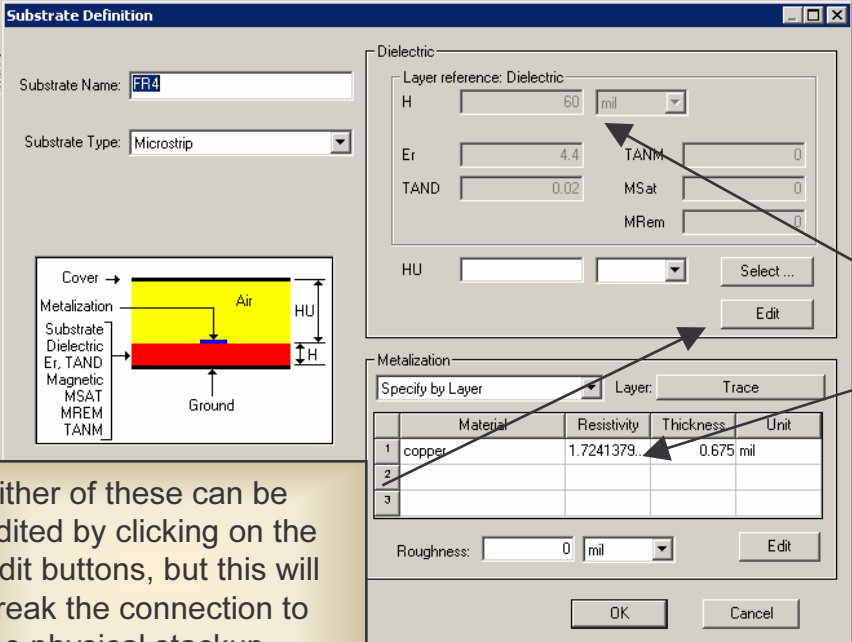
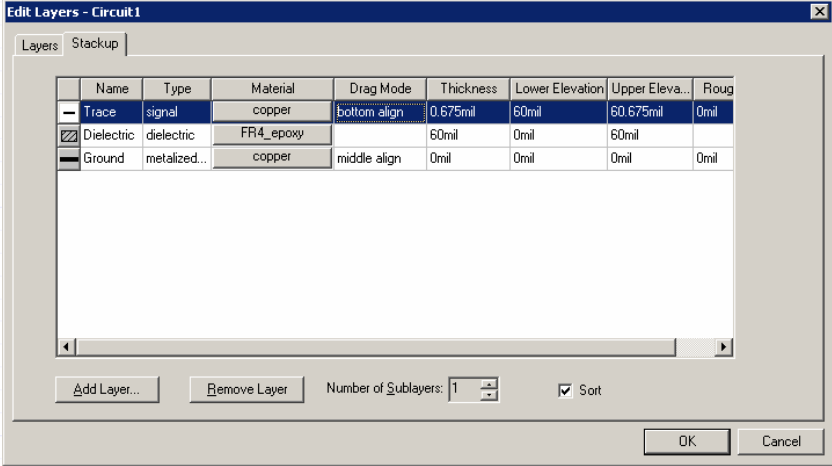
- ◆ A Technology File initializes a design with a set of data to avoid repeated entry of commonly used data.
- ◆ This data can consist of layers and stackup information for layout, configured libraries of components, and substrate definition(s) for circuit analysis.
- ◆ Users and foundries can customize Technology Files for their own manufacturing process and simulation models.
- ◆ A “technology” file contains stackup, substrate information and list of libraries to load.
- ◆ These information can be created, saved, copied and changed to suit the user. Designer has many standard technology files to choose from. They range from simple substrates such as single layered alumina, to complex multi-layered stackups. The user can also create their own simple or complex technology files and stackups, or modify existing ones.

# View Data Loaded by Technology file



Single click on the stackup icon in the toolbar to open the stackup dialog

Double click on the substrate icon to open the substrate definition dialog

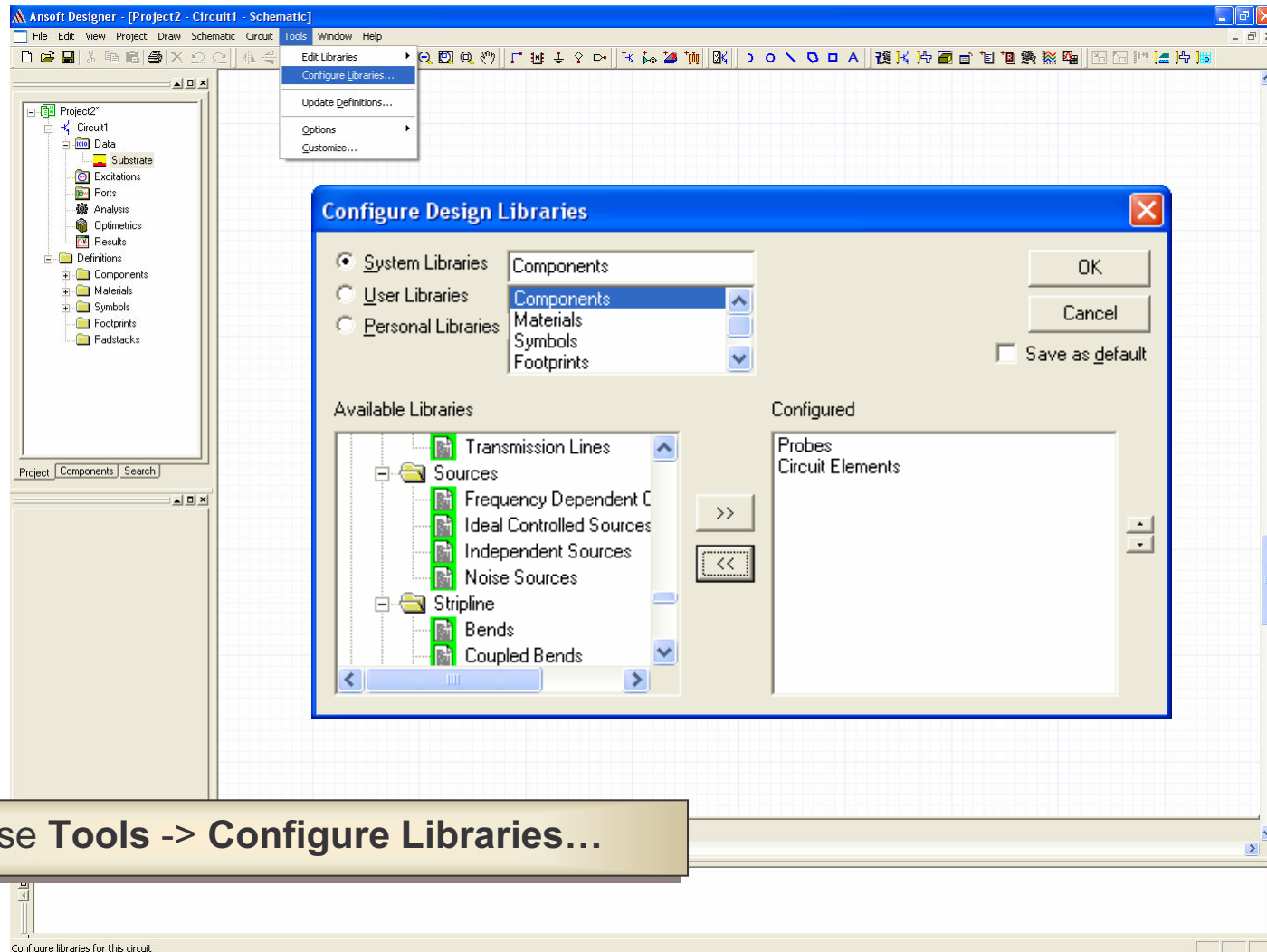


Either of these can be edited by clicking on the Edit buttons, but this will break the connection to the physical stackup.

Note that the dielectric information is disabled. This indicates it is referencing a layer in the stackup.

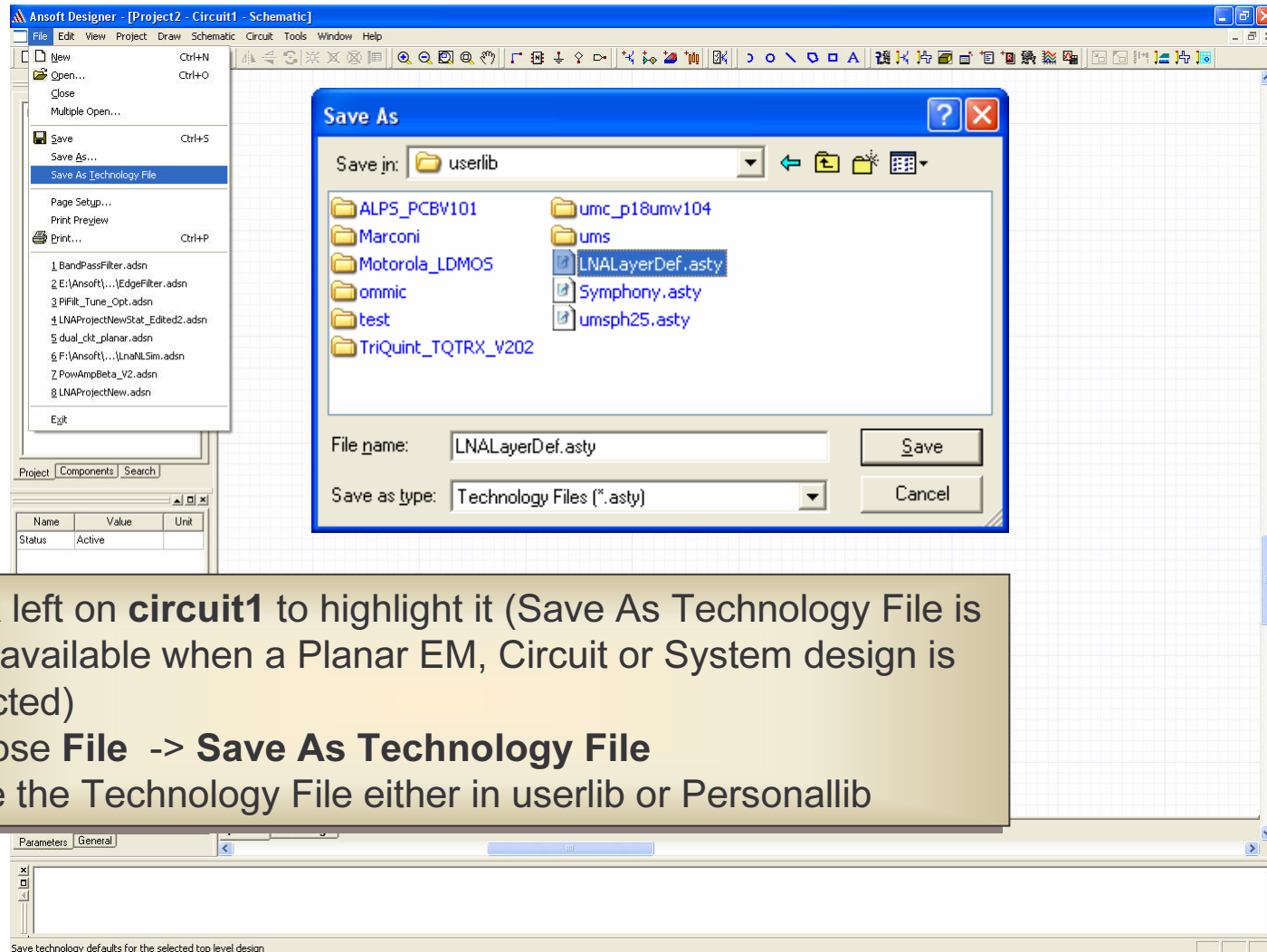
The metallization is also referencing the Trace layer.

# View Loaded Libraries



- Choose **Tools** -> **Configure Libraries...**

# Save Technology File



- Click left on **circuit1** to highlight it (Save As Technology File is only available when a Planar EM, Circuit or System design is selected)
- Choose **File** -> **Save As Technology File**
- Save the Technology File either in userlib or Personallib

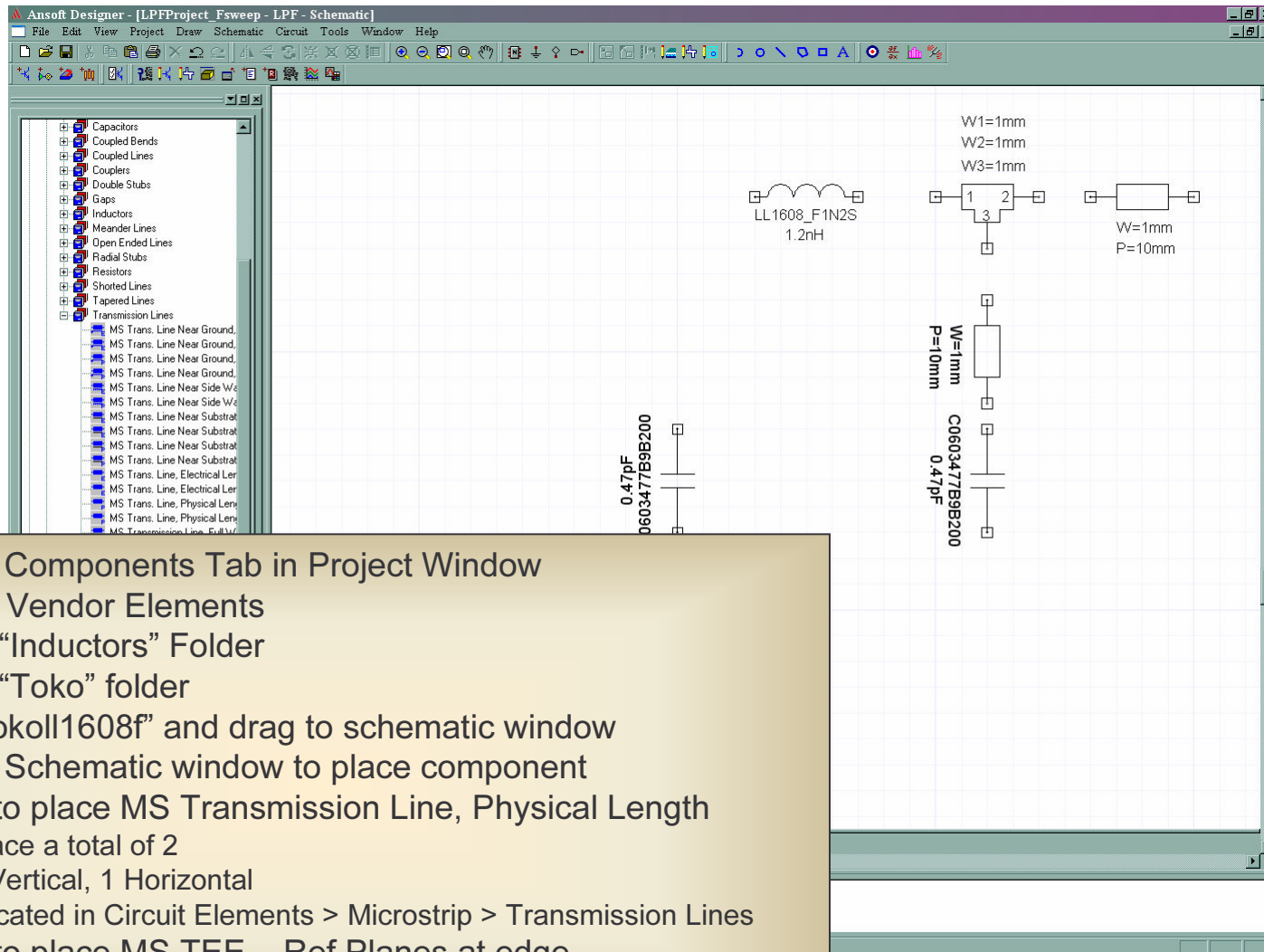
# Insert Component (Vendor Library Capacitors)

The screenshot shows the Ansoft Designer interface with a schematic window and two component libraries. The schematic window displays a capacitor component with the model name C060347789E200 and a value of 0.47pF. The component libraries show the navigation path: Vendor Elements > Capacitors > Philips > philips\_smc\_0603.

Name	Value	Unit
Model	C060347789E200	
DeviceLibr...	philips_smc_0603.lib	
C	0.47	pF
VComp	Choose Model	
Status	Active	

- Click on Components Tab in Project Window
- Scroll to Vendor Elements
- Expand “Capacitors” Folder
- Expand “Philips” Folder
- Click “Philips\_smc\_0603” and drag to schematic window
- Hit the “R” key to rotate Capacitor to vertical Position Shown
- Click on Schematic window to place component
- Move cursor to another area click again
  - This places a second capacitor

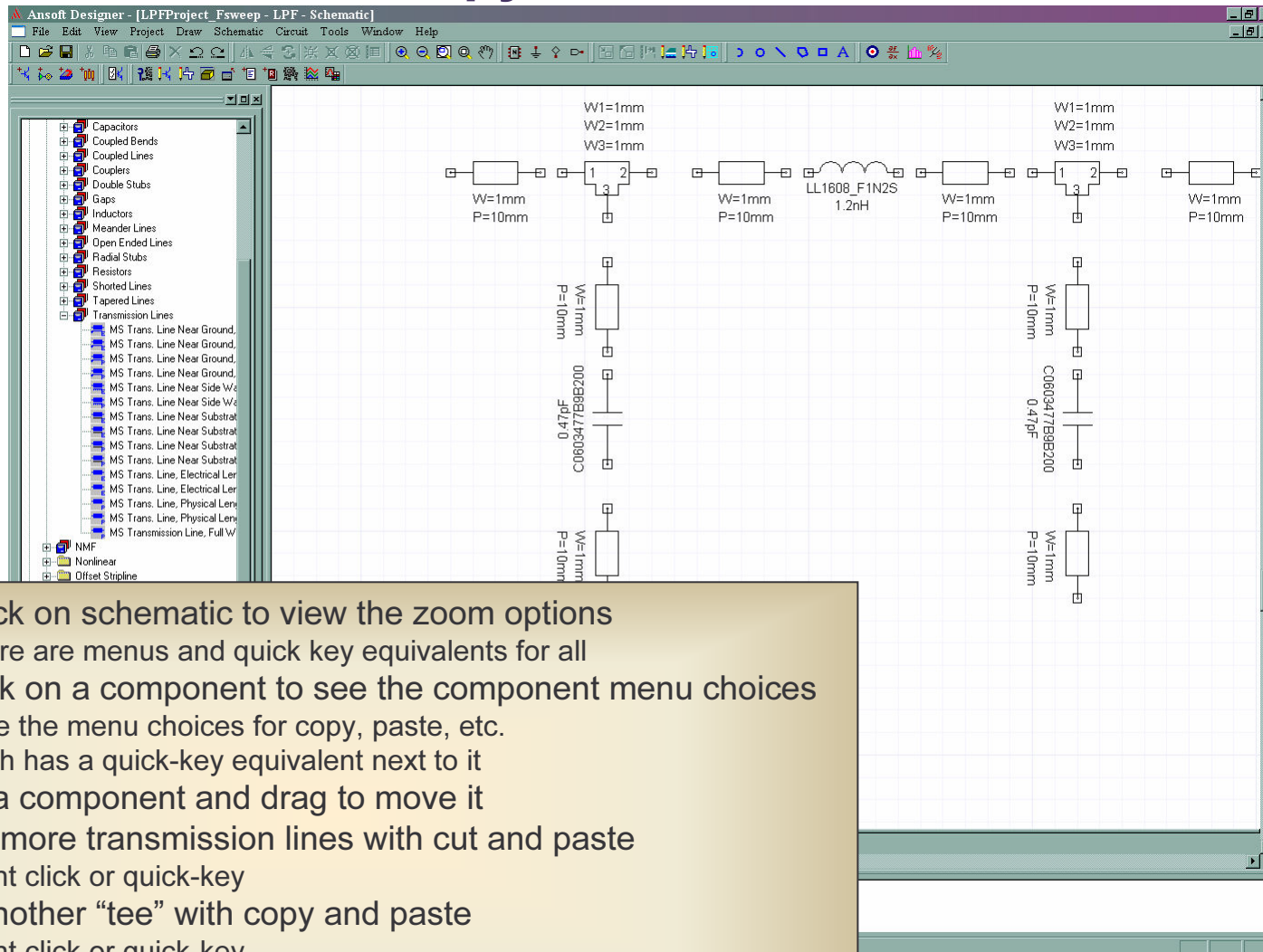
# Insert Remaining Components (Vendor Library Inductor, Transmission Lines)



- Click on Components Tab in Project Window
- Scroll to Vendor Elements
- Expand “Inductors” Folder
- Expand “Toko” folder
- Click “Tokoll1608f” and drag to schematic window
- Click on Schematic window to place component
- Repeat to place MS Transmission Line, Physical Length
  - Place a total of 2
  - 1 Vertical, 1 Horizontal
  - Located in Circuit Elements > Microstrip > Transmission Lines
- Repeat to place MS TEE – Ref Planes at edge
  - Located in Circuit Elements > Microstrip > \_General Components

# Viewing Window

## Copy, Paste & Move



- Right-Click on schematic to view the zoom options
  - There are menus and quick key equivalents for all
- Right click on a component to see the component menu choices
  - Note the menu choices for copy, paste, etc.
  - Each has a quick-key equivalent next to it
- Click on a component and drag to move it
- Create 6 more transmission lines with cut and paste
  - Right click or quick-key
- Create another “tee” with copy and paste
  - Right click or quick-key
- Rotate & move the transmission lines to match schematic shown
  - Right click or quick-key



# Wiring Components

Place two grounds by clicking on the ground icon

Cursor turns into an "X" when you move the mouse over a pin

Connect components as shown on next slide

- Place cursor over a component pin
  - Cursor becomes an "x" for the wiring tool
- Click on pin
- Move cursor to pin you want to connect
  - You will see a "blue" wire
- Click on that pin
- Move a component so that its pin lies directly over another pin
  - This also connects the two component pins

# Add Ports

**Place two ports by clicking on the Port Icon**

**Double clicking on the port brings up the port dialog box**

**Ports also appear in project tree. Double-click to open properties box**

**At this point, rename the circuit and save the project**

- Click right on circuit1
- select rename
- enter LPF and hit return

**Save the project**

- Right click on project folder and select save.
- Enter LPFProject in field File Name
- click Save

**Port Definition**

Port  
Port name: Port1  
Port number: 1  
Symbol:  Interconnect  Microwave Port

Termination  
 Simple termination: Re: 50 Im: 0 Impedance  
 One-port data: Edit... Create New...

Source Definition  
Source type: Power  
Sources:  

Enable	Name	Type	Modulation	Noise

  
Add... Edit... Delete

Load Pull Tuner and Reference Node  
Load Pull Tuner: <none> Edit... Create New...  
Reference Node: Ground

OK Cancel

# Component Properties

The screenshot shows the Ansoft Designer interface with a circuit schematic. A central component is highlighted in red. A Properties dialog box is open, showing a table of parameters. A dockable properties window is also visible, showing a list of parameters for the selected component.

**Properties Dialog Box Table:**

Name	Value	Unit	Description	Override
W1	1	mm	Conductor width on node 1	<input checked="" type="checkbox"/>
W2	1	mm	Conductor width on node 2	<input checked="" type="checkbox"/>
W3	1	mm	Conductor width on node 3	<input checked="" type="checkbox"/>
NSUM	3		Number of higher order modes	<input type="checkbox"/>
SUB	FR4		Substrate name	<input checked="" type="checkbox"/>
CoSimulator	Circuit			<input type="checkbox"/>
CoSimStackup	Layout stackup			<input type="checkbox"/>
CoSimDeembedRa...	3			<input type="checkbox"/>

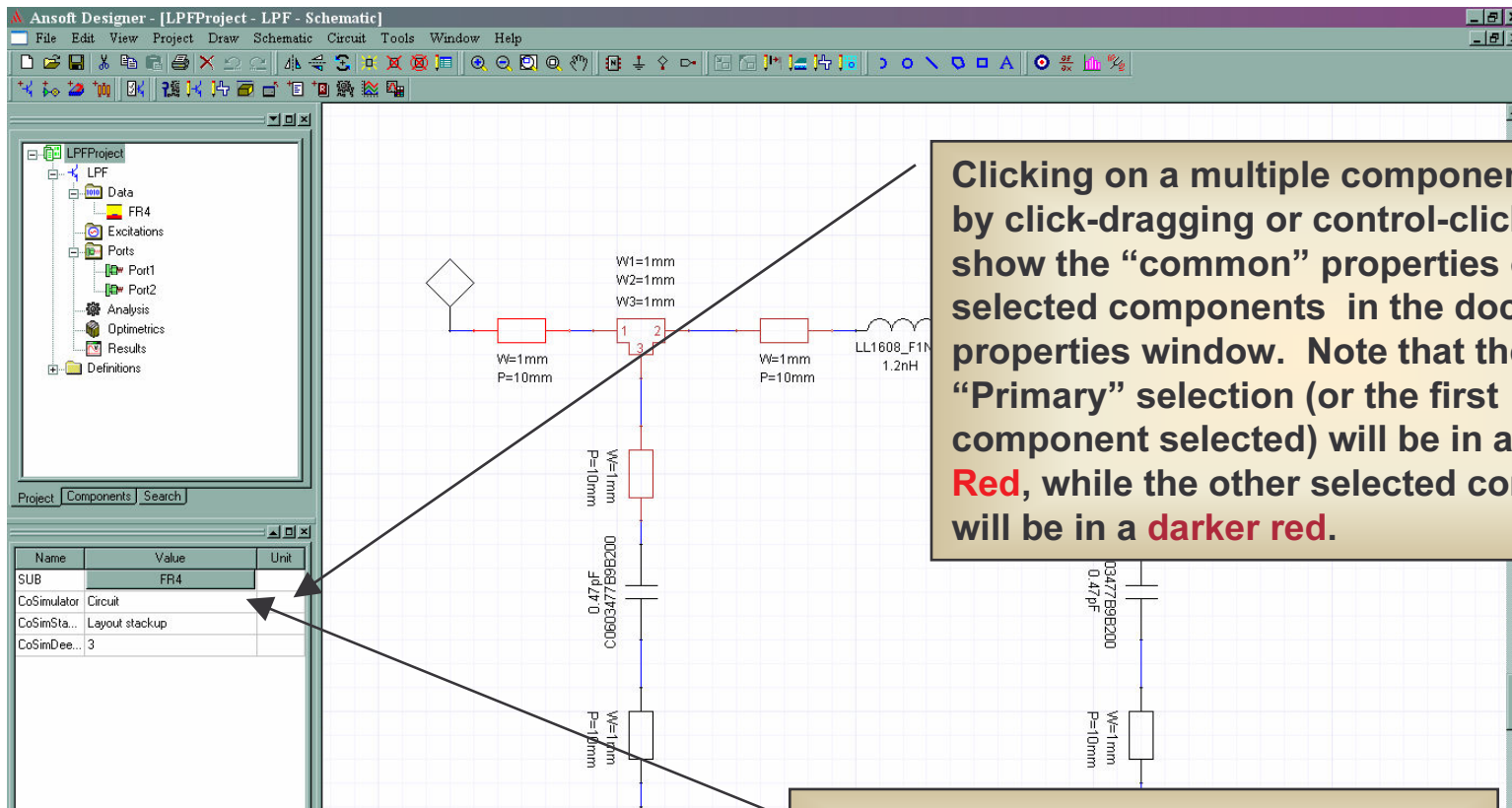
**Dockable Properties Window Table:**

Name	Value	Unit
W1	1	mm
W2	1	mm
W3	1	mm
NSUM	3	
SUB	FR4	
CoSimulator	Circuit	
CoSimSta...	Layout stackup	
CoSimDee...	3	
Status	Active	
Info	MSTEE	

**Annotations:**

- Multiple tabs are available which show different sets of properties (pointing to the 'General' and 'Symbol' tabs in the dockable window).
- Clicking on a single component outlines the component in Red and shows that component's properties in the dockable properties window (pointing to the red component and the dockable window).
- Double-Clicking on a component outlines brings up the Properties Dialog Box, which is just an "expanded" version of the dockable properties window (pointing to the Properties dialog box).

# “Multiple” Component Properties



Clicking on a multiple component (either by click-dragging or control-click) will show the “common” properties of the selected components in the dockable properties window. Note that the “Primary” selection (or the first component selected) will be in a brighter Red, while the other selected components will be in a darker red.

The user can also edit on the schematic, modifying parameters value directly by typing on the schematic

In this schematic, the selected components are 3 MS transmission lines and 1 MS TEE. Substrate is a common parameter. This allows the user to quickly change the “common” values of many components at once.

# About Vendor Components

The inductor and Capacitors are elements form the vendor library. These components have predefined properties, such as footprints (discussed later) and parameters.

**Properties**

Parameter Values | General | Symbol | Property Displays

Value     Optimization     Tuning     Sensitivity     Statistics

Name	Value	Unit	Description
Model	LL1608_F1N2S		
DeviceLibraryName	tokoll1608f.lib		
L	1.2	nH	
VComp	Choose Model		
Status	Active		

Clicking on this value brings up the possible vendor models shown on the right. Select 10nh for the inductor and 10pf for the capacitance

**Model List**

Model	L	Tolerance
LL1608_F1N2S	1.200nH	0.30nH
LL1608_F1N5S	1.500nH	0.30nH
LL1608_F1N8S	1.800nH	0.30nH
LL1608_F2N2S	2.200nH	0.30nH
LL1608_F2N7S	2.700nH	0.30nH
LL1608_F3N3S	3.300nH	0.30nH
LL1608_F3N3K	3.300nH	10.00%
LL1608_F3N3M	3.300nH	20.00%
LL1608_F3N9S	3.900nH	0.30nH
LL1608_F3N9K	3.900nH	10.00%
LL1608_F3N9M	3.900nH	20.00%
LL1608_F4N7S	4.700nH	0.30nH
LL1608_F4N7K	4.700nH	10.00%
LL1608_F4N7M	4.700nH	20.00%
LL1608_F5N6S	5.600nH	0.30nH
LL1608_F5N6K	5.600nH	10.00%
LL1608_F5N6M	5.600nH	20.00%
LL1608_F6N8J	6.800nH	5.00%
LL1608_F6N8K	6.800nH	10.00%

OK

Double click on this inductor to bring up this properties dialog box

# Defining Variables

## Component Selection Methods

Click on the value field of the inductor. The cursor will change to an insertion cursor, allowing the user to type in a new value. Type in 10nh

Add Variable to LPF

Name: wline

Value: 0.8mm

Parameter Default

Local Variable

Project Variable

OK Cancel

Name	Value	Unit
W	wline	mm
P	10	mm
SUB	FR4	
TRL	TRL	
CoSimulator	Circuit	
CoSimSta...	Layout stackup	
CoSimDee...	3	

Multiple select ALL the MS transmission lines. In the dockable properties window, type wline for the value of width, w. Since wline has not been previously defined, the dialog box to the right will appear, prompting the user to enter a value for wline. Enter 0.8mm and click the radio button for Local Variable. Click ok. Type lline for p, Enter 1mm and click the radio button for Local Variable.

# Defining Variables

## Circuit Selection method

Right click on the LPF icon in the project window. This brings up the shown menu. Choose Design Properties...

We want to add a new parameter, click add...

Name	Value	Unit	Description	Read-only	Hidden
wline	0.8	mm		<input type="checkbox"/>	<input type="checkbox"/>
lline	1	mm		<input type="checkbox"/>	<input type="checkbox"/>
Cvalue	10	pF		<input type="checkbox"/>	<input type="checkbox"/>

The Add Property dialog appears. Type Cvalue for the name and 10pf for the value. Click OK

Enter initial value into Value field. This should be a number, variable, or expression. Referenced project variables should be prefixed with a '\$'. Examples: 22.4pF, \$C1, 2\*cos(\$x).

# Variable Assignment

- ◆ Three different types of **Variables**:

- ◆ **Local Variables**

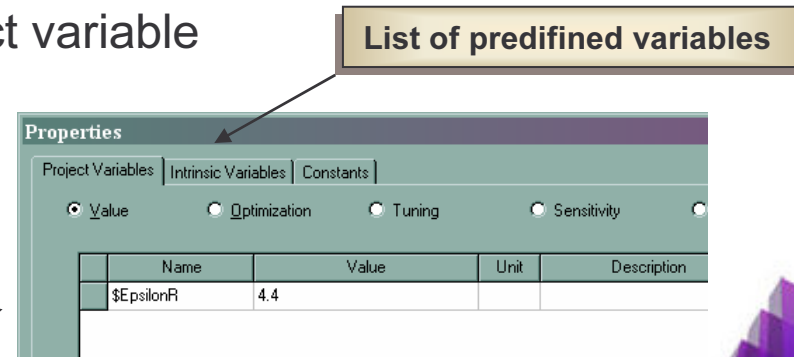
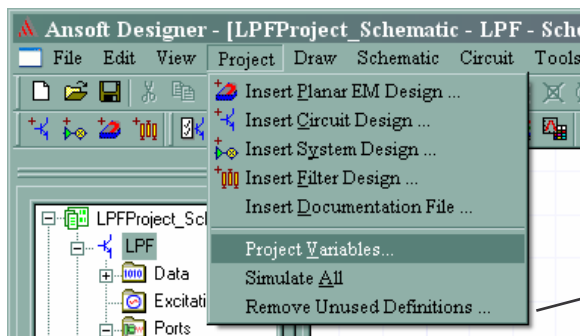
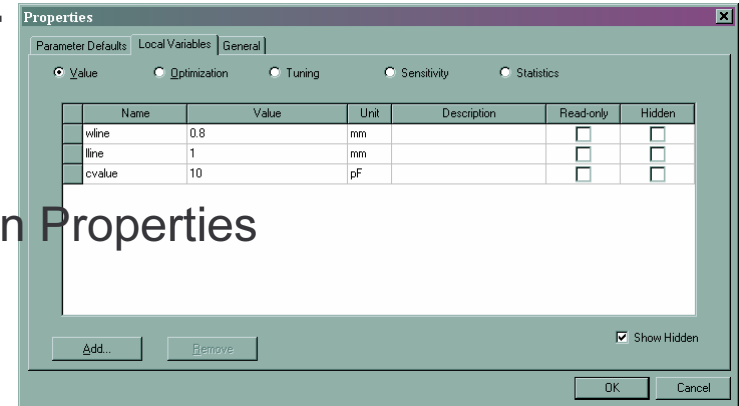
- ◆ Entered directly in parameter field
    - ◆ RM Click on Design and select Design Properties

- ◆ **Definition Parameters**

- ◆ Passed parameters for a Design
    - ◆ Entered in same manner as Local Variables

- ◆ **Project Variables**

- ◆ Global parameters selected from Project menu
    - ◆ \$ added to designate project variable

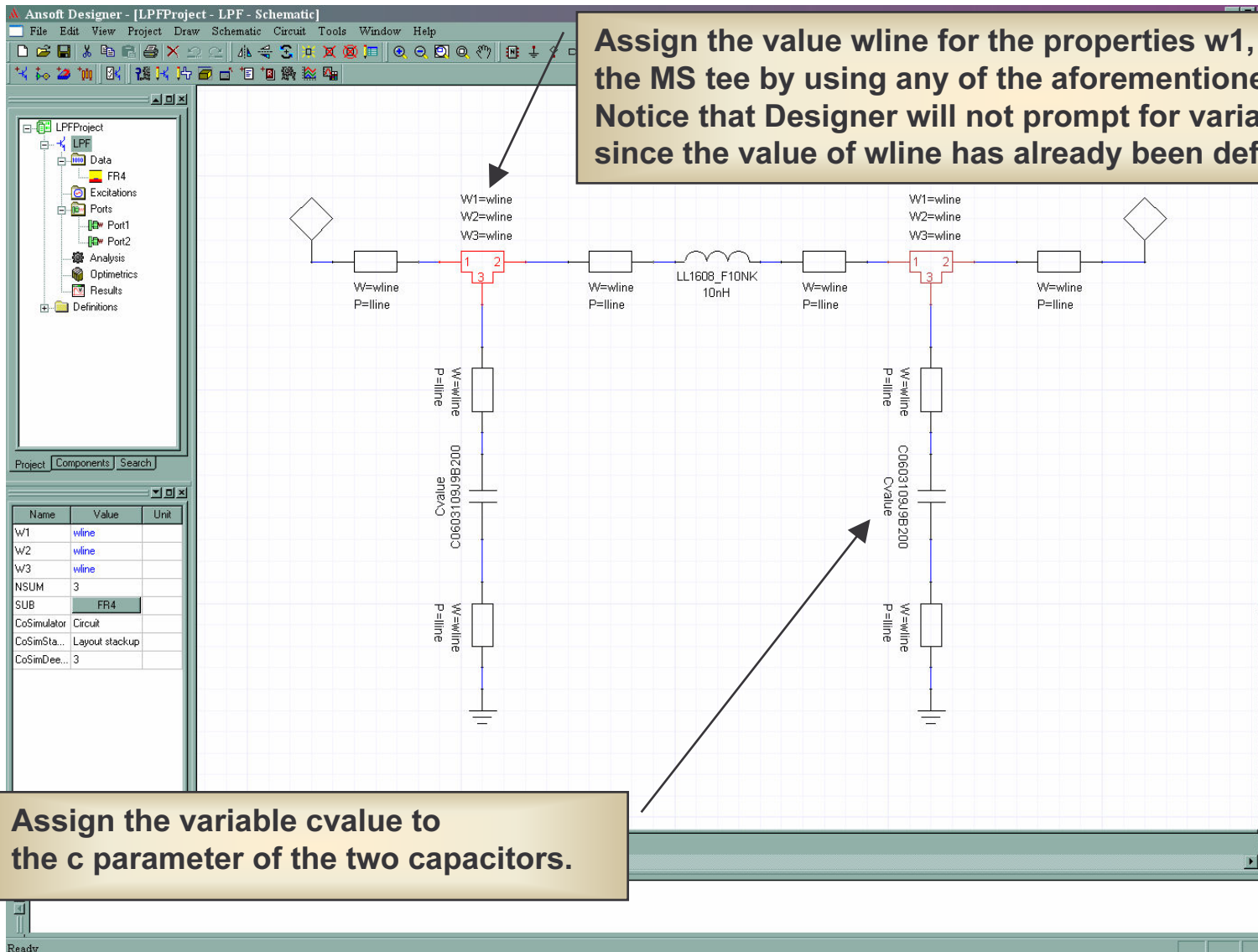




# Defining Variables

## Final Variable Assignment

Assign the value wline for the properties w1, w2 and w3 in the MS tee by using any of the aforementioned methods. Notice that Designer will not prompt for variable names since the value of wline has already been defined.



Assign the variable cvalue to the c parameter of the two capacitors.

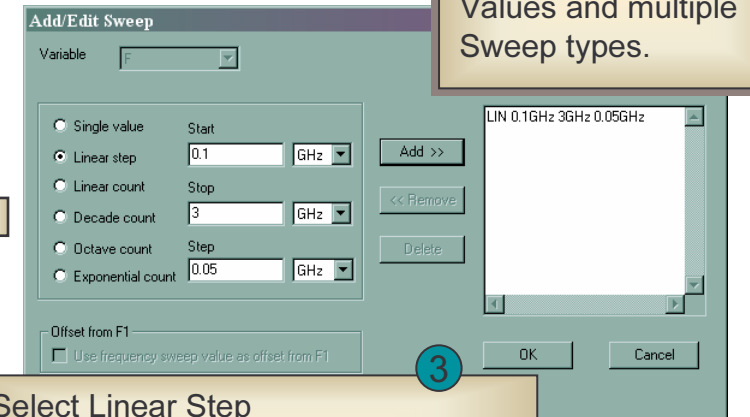
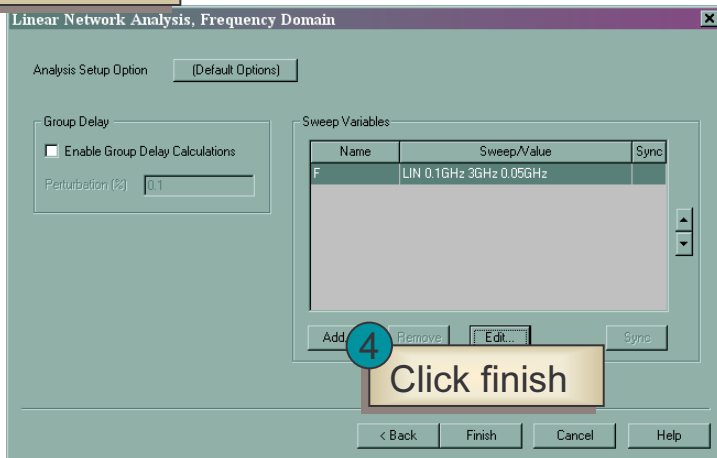
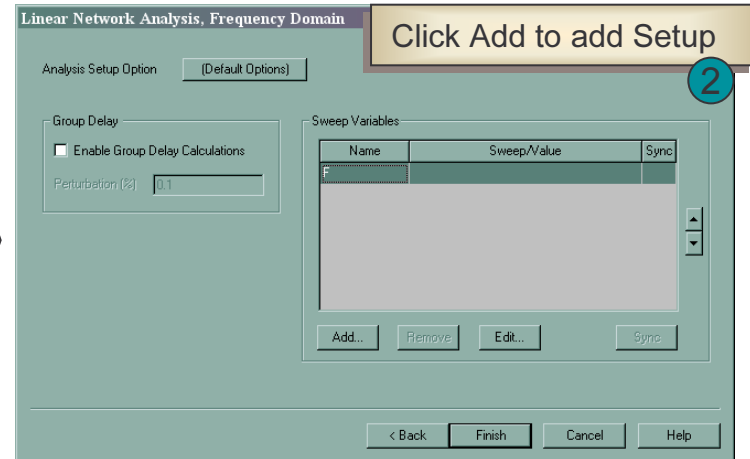
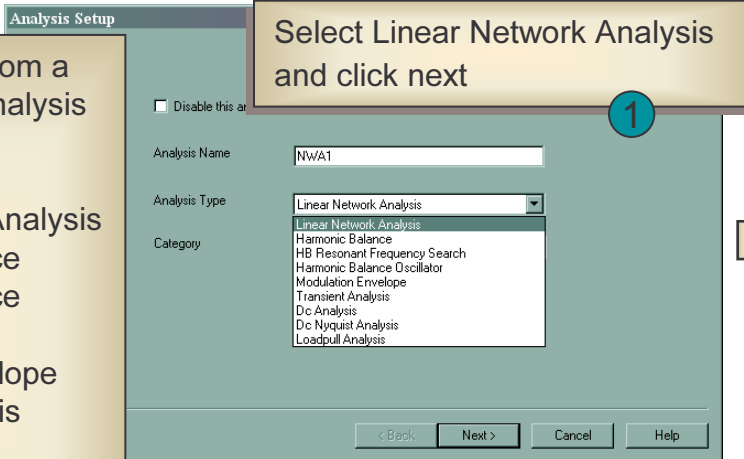
# Add Analysis Setup

The screenshot displays the Ansoft Designer interface for a project named 'LPFFProject'. The main workspace shows a schematic diagram of a low-pass filter. The circuit consists of two ports, a central inductor labeled 'LL1808\_F10NK' with a value of '10nH', and two parallel branches. Each branch contains a capacitor labeled 'C0603109J09B200' and a resistor. The schematic is annotated with parameters such as 'W=wline' and 'P=liline'. A context menu is open over the 'Analysis' folder in the project tree, with the option 'Add Analysis Setup...' selected. A text box at the bottom of the screenshot provides instructions: 'The schematic is completed we have to define the Analysis setup. Click right on Analysis, select Add Analysis Setup'.

# Set Analysis Setup

You can select from a list of different Analysis Setup :

- Linear Network Analysis
- Harmonic Balance
- Harmonic Balance
- Oscillator
- Modulation Envelope
- Transient Analysis
- DC Analysis
- DC Nyquist Analysis
- Load Pull Analysis



The Add/Edit Sweep dialog enables the Setup of Single Values and multiple Sweep types.

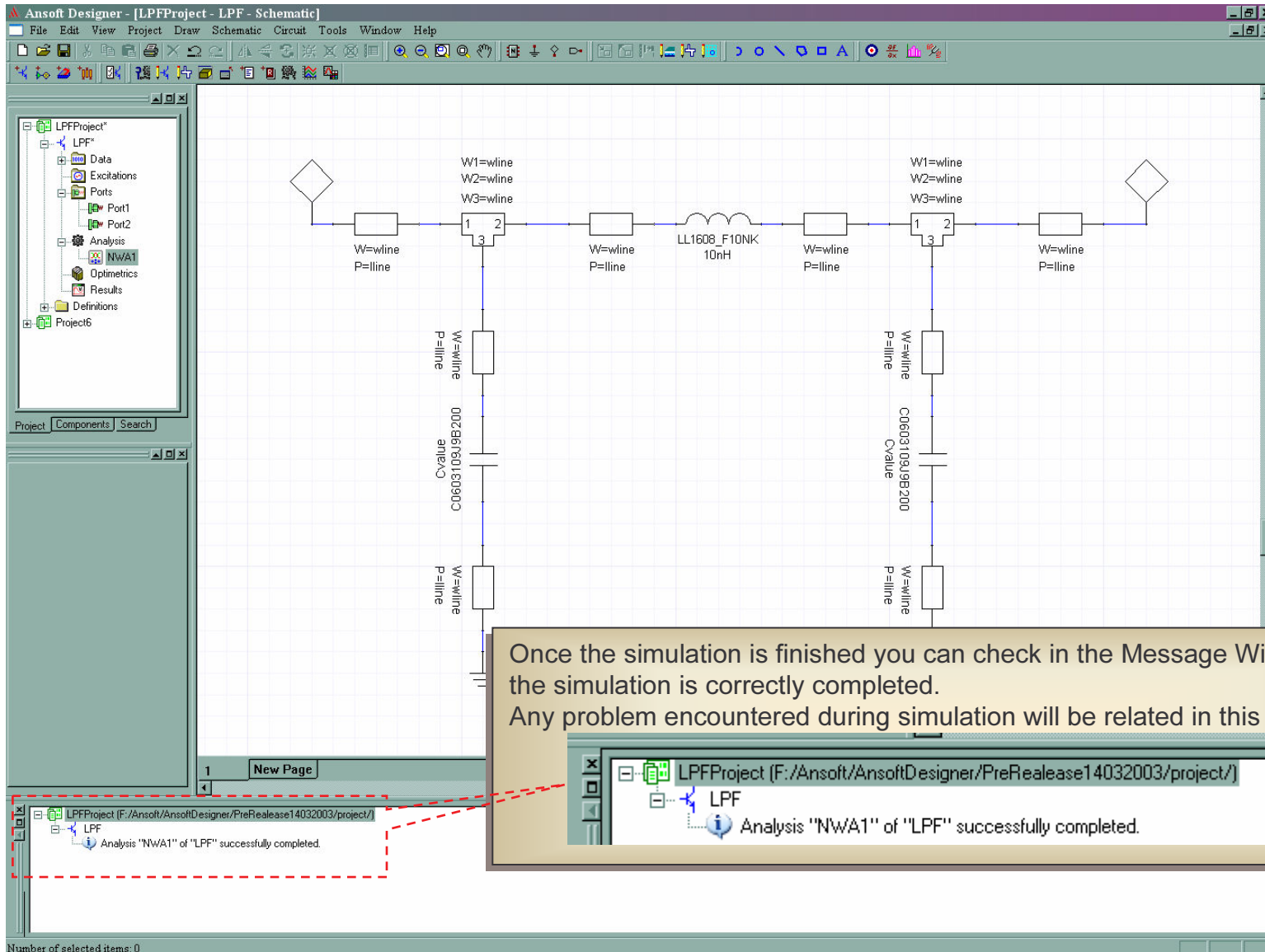
Hit CTRL+S to save the project

# Run Analysis

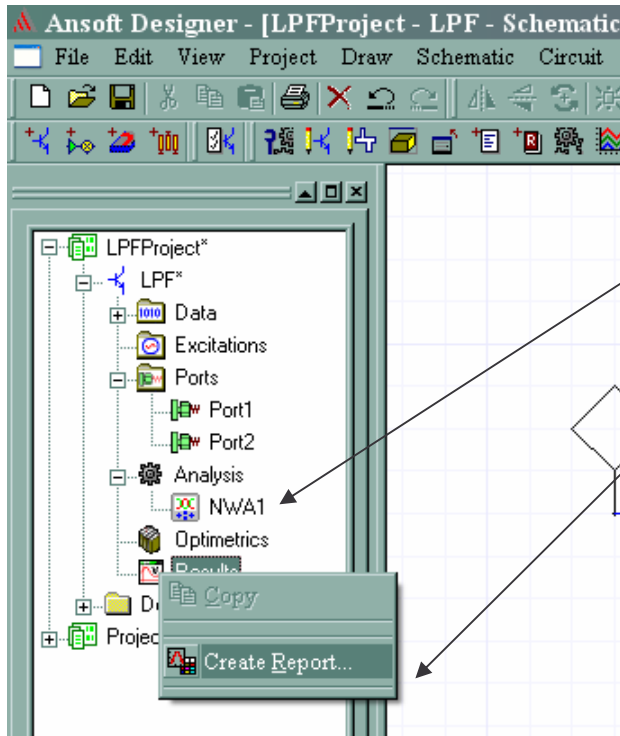
The screenshot displays the Ansoft Designer interface for a project named 'LPFFProject - LPF - Schematic'. The main workspace shows a circuit schematic with a central component labeled '1' with pins 1, 2, and 3. It is connected to various components including wave sources (W=wline), ports (P=line), and a capacitor (C0003109109B200) with a value of 0.0003109109B200. A context menu is open over the 'NWA1' analysis folder in the left-hand project tree, with 'Analyze NWA1' selected. A 'Progress' dialog box is overlaid on the schematic, titled 'LPF - NWA1 on Local Machine - RUNNING'. It features a red progress bar and the text '42 of 100', with an 'Abort' button. A text box at the bottom of the screenshot provides instructions: 'Analysis setup name NWA1 is added under the Analysis Folder. Click right on NWA1 icon and select Analysis NWA1. The progress bar shows the status of the analysis'.

Analysis setup name NWA1 is added under the Analysis Folder.  
Click right on NWA1 icon and select Analysis NWA1.  
The progress bar shows the status of the analysis

# Simulation Successfully Completed

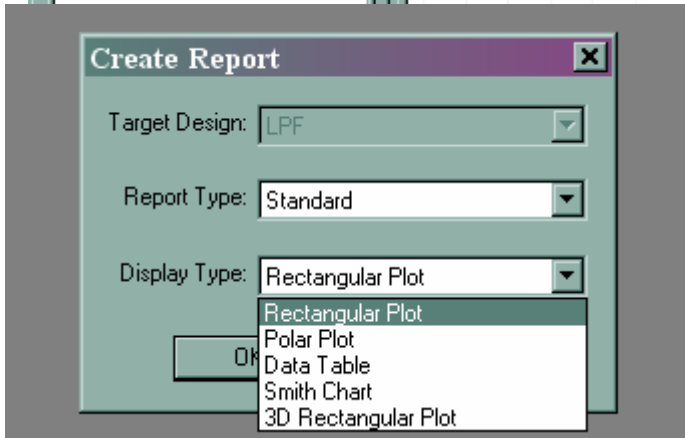


# Plotting Results: Create Report



To initiate plotting results:  
Right click on Results in the Project Manager Window

Then:  
Select Create Report



The Create Report Dialog allows the generation of Rectangular, Polar, Table, Smith Charts, and 3D Plots.

For this analysis we'll create a Rectangular Plot  
Select OK and the Report Editor Appears

# Plotting Results: Add Traces

**To create a Plot from the Report Editor:**

Click on S-parameters

Click on S11 and S21

Click on dB

Select Add Trace

Select Done

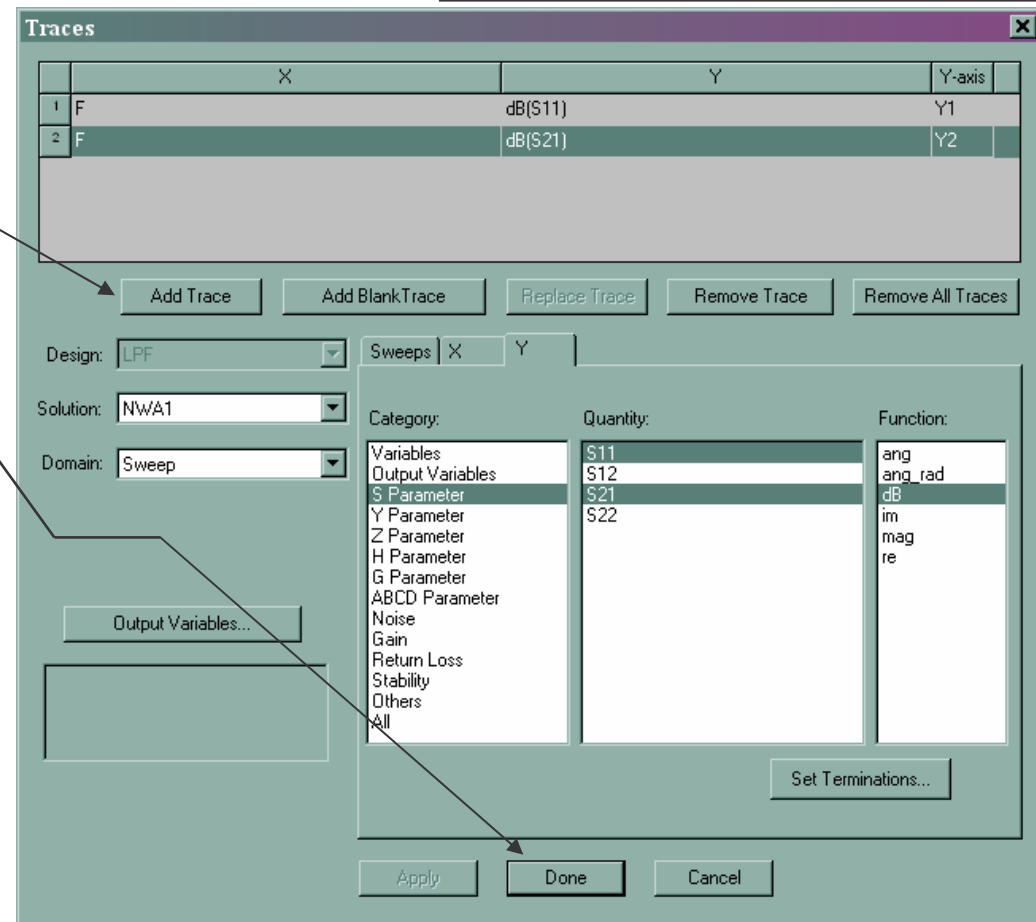
**Y-Axis** – select the Y-axis which to plot the trace against (maximum of 4 available)

**Design** – if the project has multiple top-level designs, select the desired design

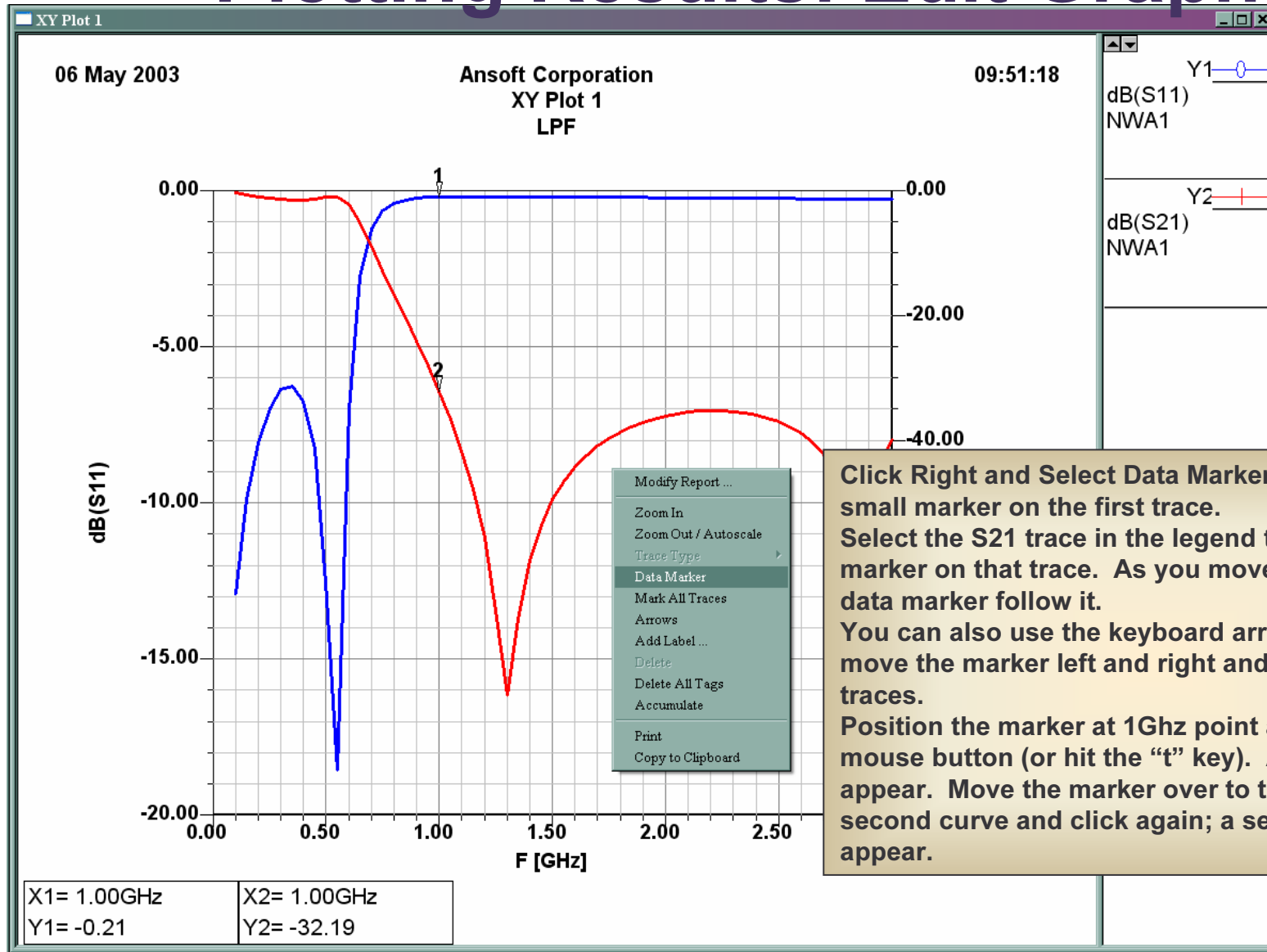
**Solution** – if the design has multiple analysis setups, select the desired analysis

**Domain** – different domains are available depending on the solution, including frequency domain, time domain, and sweep domain

**Sweeps Tab** – allows you to determine the sweep variables and their order to be used



# Plotting Results: Edit Graph

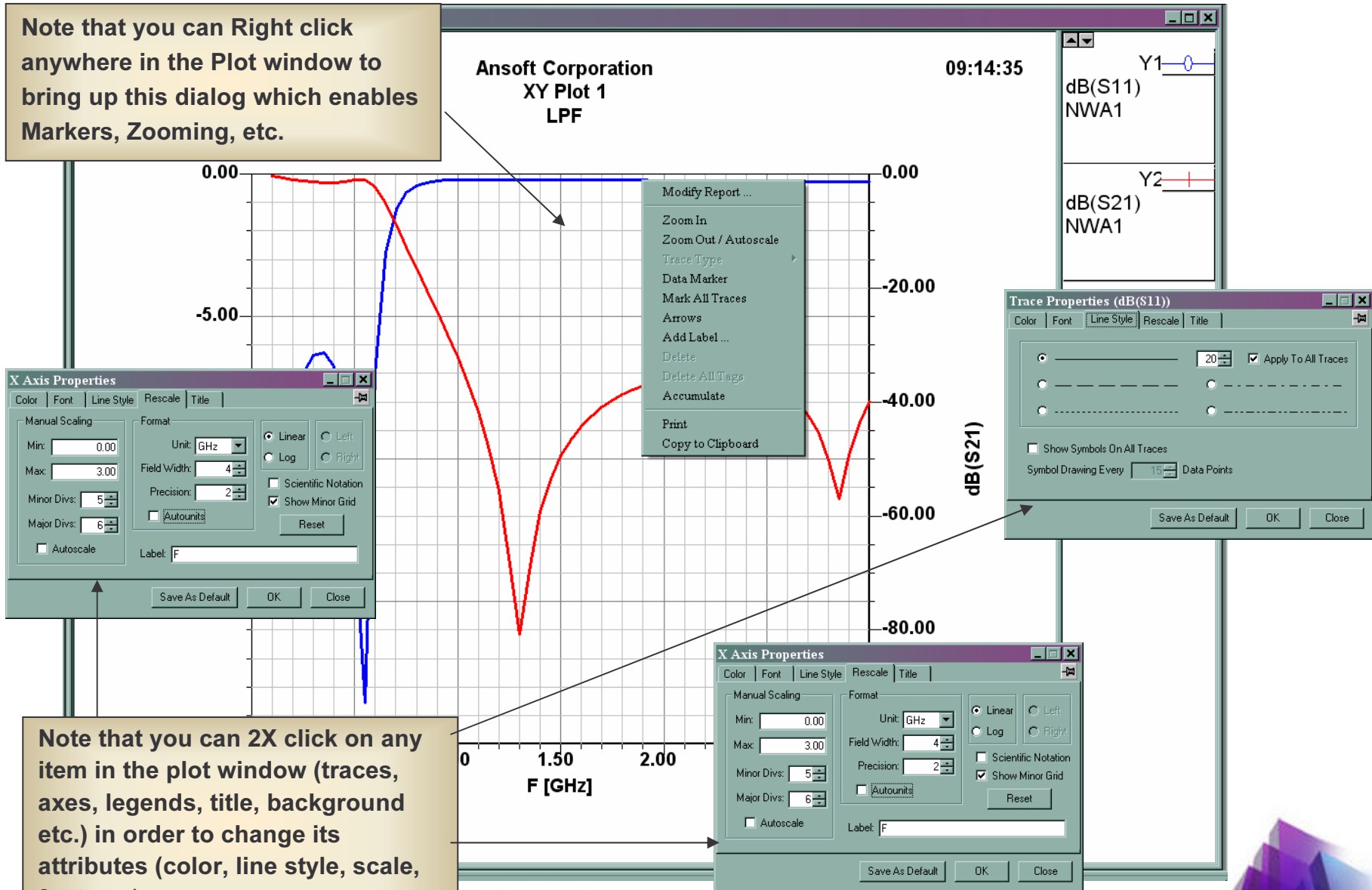


Click Right and Select Data Marker, you'll see a small marker on the first trace. Select the S21 trace in the legend to put the data marker on that trace. As you move the cursor, the data marker follow it. You can also use the keyboard arrow keys to move the marker left and right and between traces. Position the marker at 1Ghz point and click the mouse button (or hit the "t" key). A tag will appear. Move the marker over to the second second curve and click again; a second tag will appear.



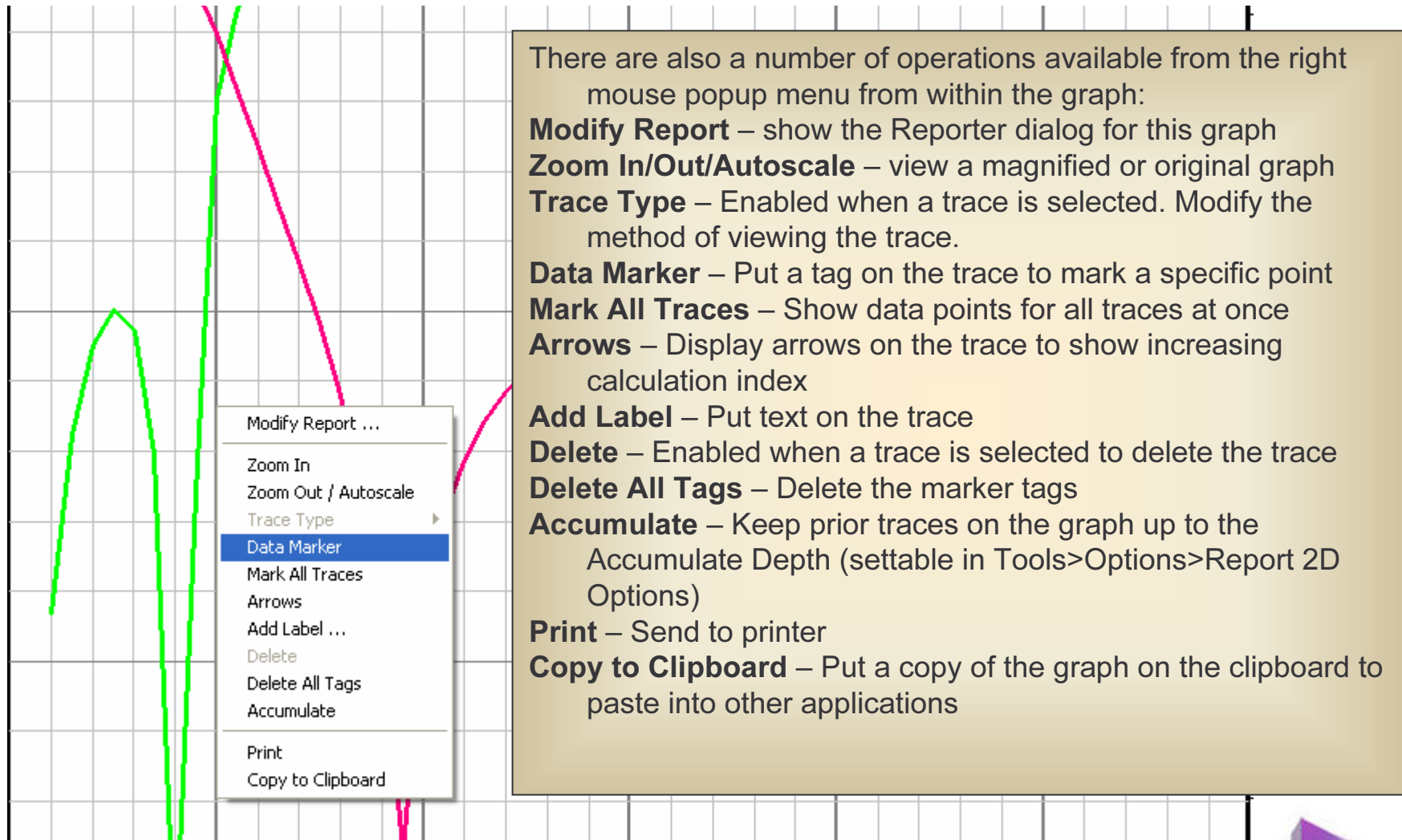
# Plotting Results: Edit Graph

Note that you can Right click anywhere in the Plot window to bring up this dialog which enables Markers, Zooming, etc.



Note that you can 2X click on any item in the plot window (traces, axes, legends, title, background etc.) in order to change its attributes (color, line style, scale, font etc.)

# Plotting Results: Edit Graph



# Parametric Sweep

In Ansoft Designer you can use any variable to define a sweep. We will sweep Cval. Click right on Analysis, select Add Analysis Setup, select Linear Network Analysis, click next.

You can notice that for a same circuit one can define multiple analysis setup. Edit F and define linear step sweep from 0.1Ghz to 3Ghz by step of 0.05Ghz, click ok.

Name	Sweep/Value	Sync
F	LIN 0.1GHz 3GHz 0.05GHz	

Select Cvalue and define a linear step from 2pf to 12pf by step of 2pf. Click ok and finish.

Click Add, Click the arrow close to variable field, all the defined variable can be selected.

Hit CTRL+S to save the project

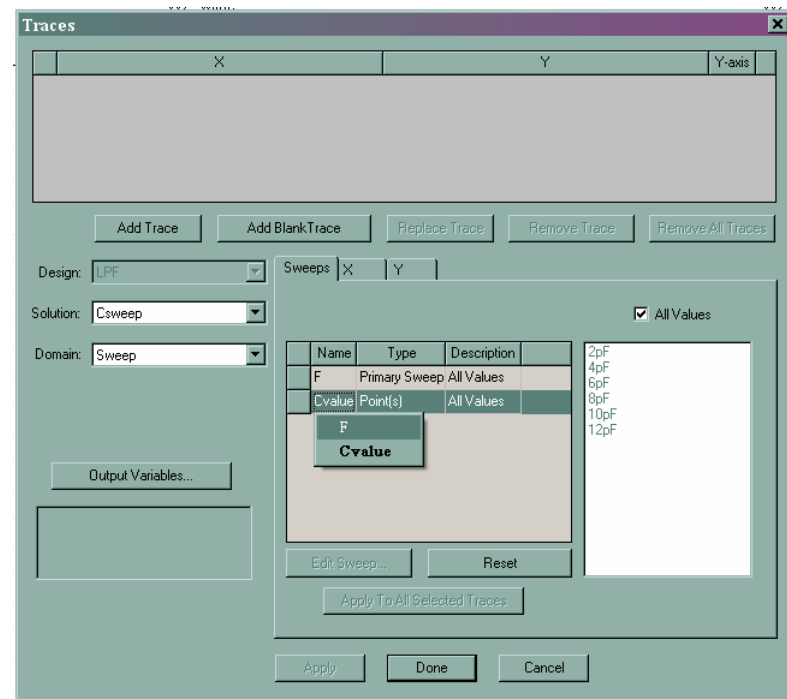
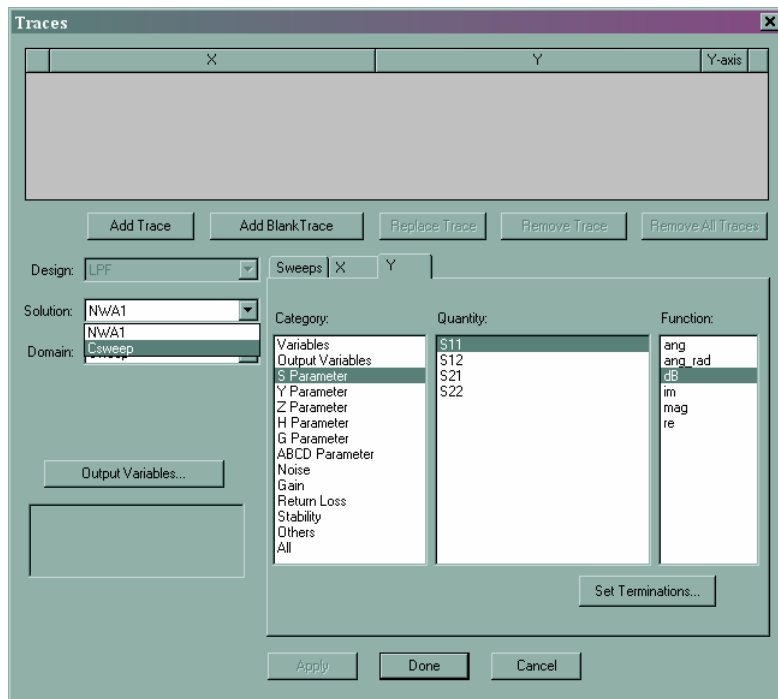
# Run Analysis Csweep

The screenshot displays the Ansoft Designer interface for a project named 'LPFProject'. The main window shows a schematic of a Low Pass Filter (LPF) circuit. The circuit includes two diamond-shaped ports, several rectangular components labeled 'W=wline' and 'P=liline', and a central inductor component labeled 'LL1608\_F10NK 10nH'. A capacitor component labeled 'C0603109J9B200' is connected to the bottom of the circuit. A context menu is open over the 'Analysis' folder in the left-hand tree, with 'Analyze Csweep' selected. A 'Progress' dialog box is overlaid on the schematic, showing the text 'LPF - Csweep on Local Machine - RUNNING' and a progress bar indicating '36 of 100' completion. An 'Abort' button is visible in the dialog box. A text box at the bottom left of the screenshot provides instructions: 'Click right on Csweep and select Analysis Csweep. You can notice that if you click right on Analysis and select Start Analysis you will run successively NWA1 and Csweep analysis.'

# Plotting Results

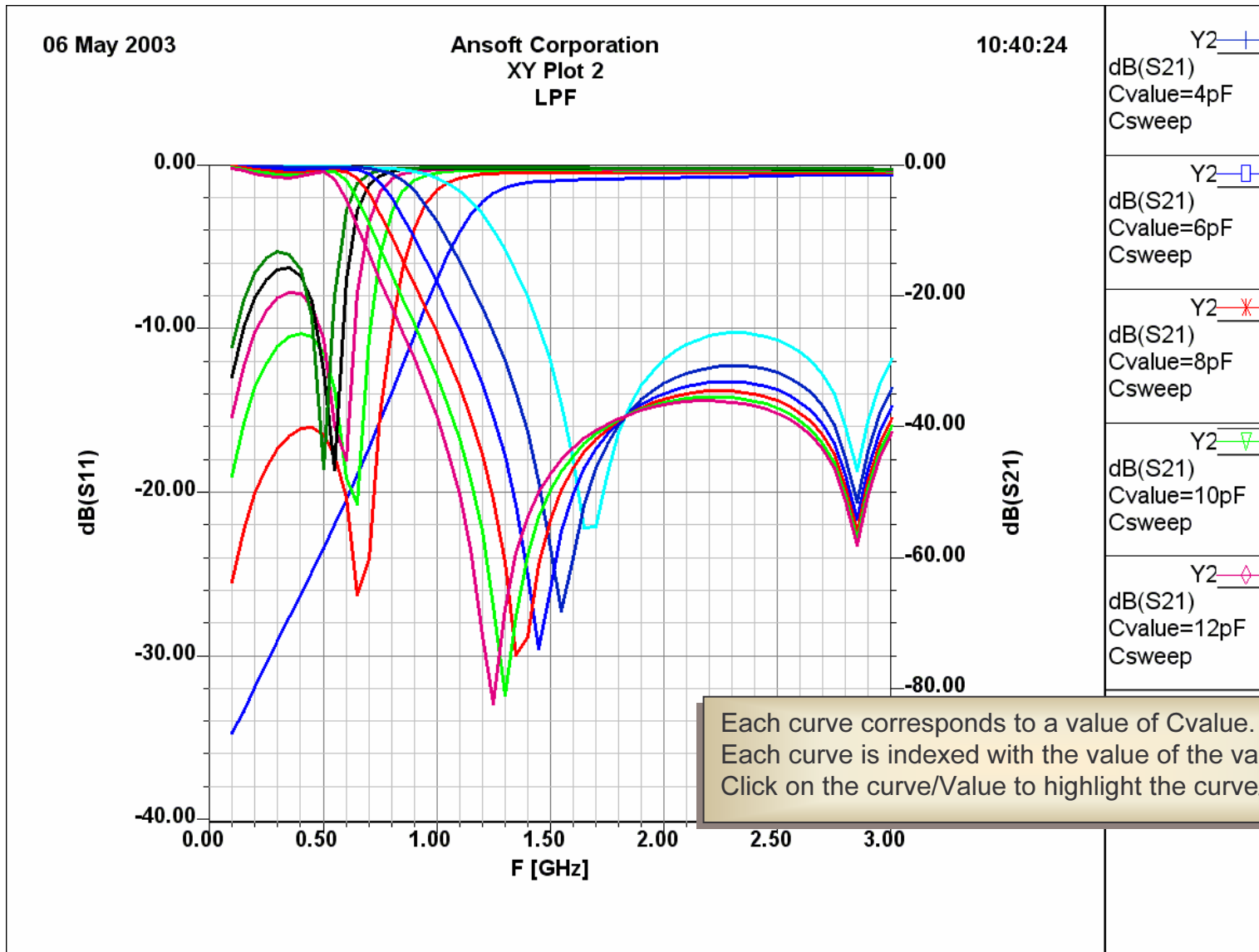
Click right on Results folder select create report.  
Click the arrow of field solution and select Cvalsweep, hit the sweep Tab.

Clicking on one of the sweep variable shows the swept values, you can select all the value, one value or several value.  
Use the shift and CTRL key to select multiple value.



Clicking on the name of the variable you can change the sweep order.  
Hit Tab Y and add S21 in DB, Click Done

# Plotting Results: Edit Graph



# Plotting Results: 3D Plot

The screenshot shows the Ansoft Designer interface with a 3D Cartesian plot. The plot displays a surface representing the relationship between three variables: dB(S21) (vertical axis, red arrow), Cvalue (horizontal axis, blue arrow), and F (depth axis, green arrow). The surface is colored with a gradient from red (high values) to blue (low values). Two dialog boxes are open: 'Create Report' and 'Traces'.

**Create Report Dialog:**

- Target Design: LPF
- Report Type: Standard
- Display Type: 3D Rectangular Plot

**Traces Dialog:**

Name	Type	Description	Value
F	Primary Sweep	All Values	2pF, 4pF, 6pF, 8pF, 10pF, 12pF
Cvalue	Secondary S...	All Values	

**Properties Panel (Bottom Left):**

Name	Value	Unit
Name	3D Cartesian ...	
Scale Min.	-81.8986	
Scale Max.	-0.144173	
Spectrum	Rainbow	
Type	Spectrum	
Graph	Trace_1	
Graph Name	Trace_1	
IsoType	Tone	
Filled	<input checked="" type="checkbox"/>	
Addgrid	<input type="checkbox"/>	
Outline	<input type="checkbox"/>	
SmoothSh...	<input checked="" type="checkbox"/>	

It is possible to create 3D plot to simultaneously view two parameters sweep.  
Click right on results, select 3D Rectangular plot, hit Tab Z add S21 in DB. Click done.

# Tuning

The screenshot shows the Ansoft Designer interface with a circuit schematic. The schematic includes a diamond-shaped component, a rectangular component labeled 'W=wline P=line', an inductor labeled 'L 160', and a capacitor labeled 'C0603109G9B200 W=wline'. A context menu is open over the capacitor, showing options like 'Cut', 'Copy', 'Paste', 'Delete', 'Rotate', 'Flip about X', 'Flip about Y', 'Push Down', 'Pop Up', 'Activate', 'Deactivate (Open)', 'Deactivate (Short)', 'Zoom In', 'Zoom Out', 'Zoom Area', 'Fit Drawing', and 'Redraw'. The 'Properties...' dialog box is also open, showing the 'Tune' tab. The 'Tune' tab has a table with the following data:

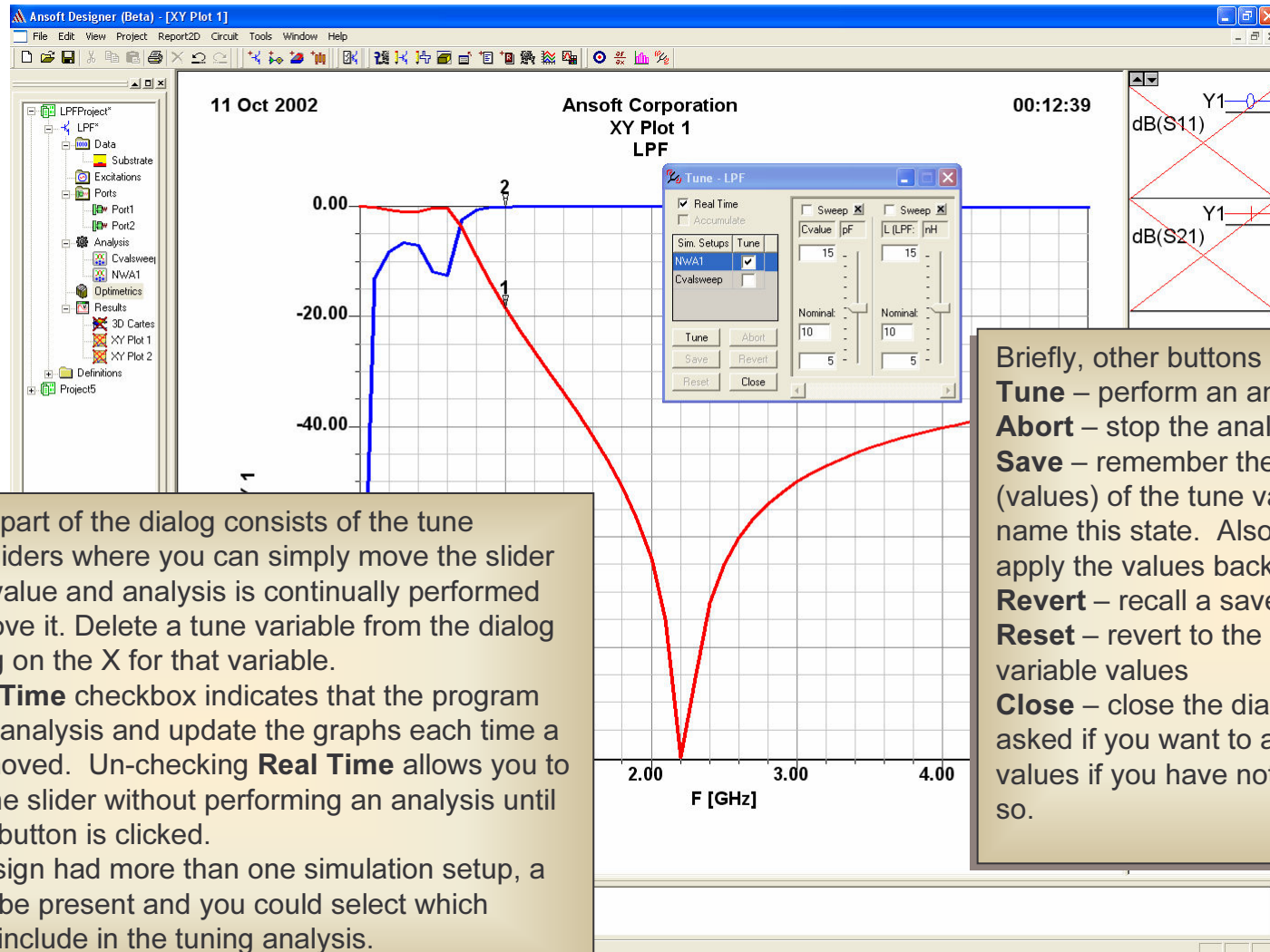
Name	Include	Min	Unit	Max	Unit	Step	Unit
wline	<input type="checkbox"/>	0.4	mm	1.2	mm	0.08	mm
lline	<input type="checkbox"/>	0.5	mm	1.5	mm	0.1	mm
cvalue	<input checked="" type="checkbox"/>	5	pF	15	pF	1	pF

At the bottom of the Properties dialog, there are 'Add...' and 'Remove' buttons, a 'Show Hidden' checkbox, and 'OK' and 'Cancel' buttons.

Tuning provides an interactive means of changing a design's variables or component values and viewing the results immediately. Any variable or parameter can be swept. In this example, we will define variable Cvalue and parameter L of the inductor to be tunable. You can select any of the variables to include in tuning. Select the checkbox for Cvalue. The minimum and maximum values are automatically set to be 1/2 and 1 1/2 times the nominal value. You can change them, if desired. Next select **Circuit > Tune** and you'll see the tuning dialog or click right on Optimetrics and select tuning



# Tuning



The main part of the dialog consists of the tune variable sliders where you can simply move the slider to a new value and analysis is continually performed as you move it. Delete a tune variable from the dialog by clicking on the X for that variable.

The **Real Time** checkbox indicates that the program will do an analysis and update the graphs each time a slider is moved. Un-checking **Real Time** allows you to position the slider without performing an analysis until the **Tune** button is clicked.

If your design had more than one simulation setup, a list would be present and you could select which setups to include in the tuning analysis.

Briefly, other buttons include:

**Tune** – perform an analysis

**Abort** – stop the analysis

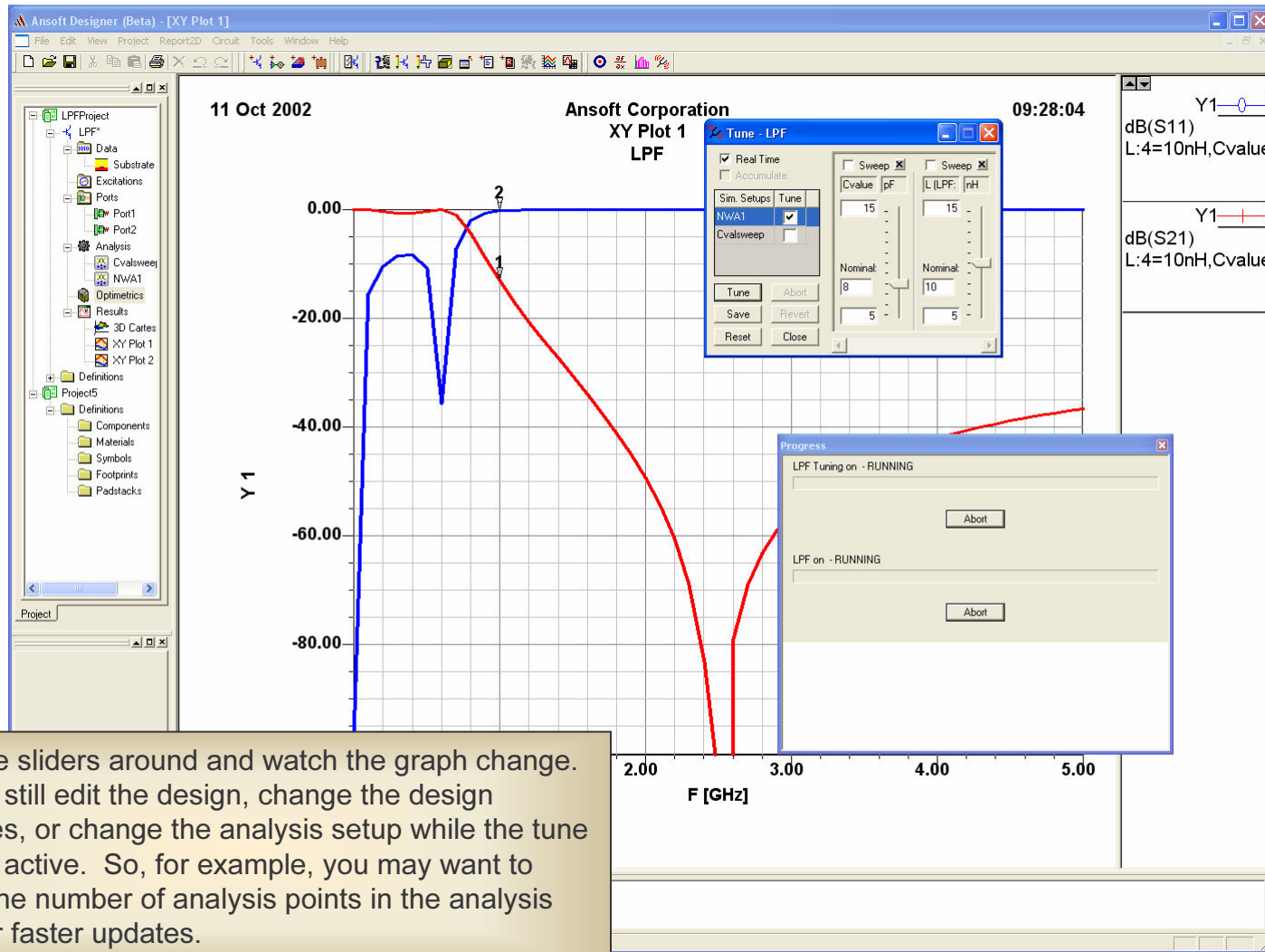
**Save** – remember the current state (values) of the tune variables and name this state. Also, optionally apply the values back to the design.

**Revert** – recall a saved state

**Reset** – revert to the original tune variable values

**Close** – close the dialog. You will be asked if you want to apply the tuned values if you have not already done so.

# Tuning: Real Time



Move the sliders around and watch the graph change. You can still edit the design, change the design properties, or change the analysis setup while the tune dialog is active. So, for example, you may want to reduce the number of analysis points in the analysis setup for faster updates.

# Tuning: Accumulate

The screenshot shows the Ansoft Designer interface for a project named 'LPPProject\*'. The main window displays a plot of the filter's response, with the y-axis labeled 'dB' ranging from -100.00 to 0.00 and the x-axis labeled 'F' (Frequency) ranging from 0.00 to 1.00. Multiple colored curves represent different parameter variations. Overlaid windows include:

- Tune - LPF**: A dialog box with 'Accumulate' checked and 'Sweep' options for 'cvalue' (pF) and 'L (LPF:)' (nH). The 'cvalue' is set to 15 and 'L (LPF:)' is set to 15. Buttons for 'Tune', 'Save', 'Reset', 'Close', 'Abort', and 'Revert' are visible.
- Apply Tuned Variation**: A dialog box with a table of parameter values:

L (LPF:5)	cvalue
10nH	13pF
10nH	14pF
10nH	15pF
10nH	5pF
10nH	6pF
10nH	7pF
10nH	8pF

- Progress**: Two windows showing 'LPP Tuning on - RUNNING' and 'LPP on - RUNNING' with 'Abort' buttons.

On the right side, a list of plots is visible, including dB(S11) and dB(S21) for various components like NWA1 and L:5=10nH, cval.

You can also setup a sweep of a tune variable by clicking on the Sweep checkbox.

# Tuning: Multiple Analysis Setup

The screenshot displays the Ansoft Designer interface for a project named 'DBS218S11'. The main window shows a plot of the filter's magnitude response. A 'Tune - LPF' dialog box is open, showing simulation settings for 'Real Time' and 'Sweep' (set to 15). An 'Apply Tuned Variation' dialog box is also open, listing inductor values from 10nH to 8nH. A 'Progress' window indicates that the LPF tuning is running. A text box at the bottom right of the interface says 'Hit CTRL+S to save the project'.

Delete Cvalue tune variable from the dialog by clicking on the X for that variable.  
 Select NWA1 and CswEEP to tune multiple sweep.  
 Move the sliders around and watch the graphs change.  
 Close the Tune window.

You can either select a value from the tune or none.  
 For none Click Don't Apply

Hit CTRL+S to save the project

# Optimization: Define optimizable parameters

The screenshot shows the Ansoft Designer interface with a circuit schematic. The schematic includes components like inductors (LL1608\_F10NK 10nH) and capacitors (C338C9B200) connected to ports. Two Properties dialog boxes are open, showing optimization settings for parameters 'cvalue' and 'L'.

**Text Box 1 (Top Left):** To run optimization we have to define the variable, parameters to be optimizable and to define goals. We will select the capacitors and inductor to be optimizable.

**Text Box 2 (Top Right):** Double click on the Inductor symbol, then select the Optimization view and click the Include button to ensure that L is used in the optimization. Set Min to 2nh and Max to 18nh

**Properties Dialog 1 (Bottom Left):** Shows optimization settings for 'cvalue'.

Name	Include	Min	Unit	Max	Unit
wline	<input type="checkbox"/>	0.4	mm	1.2	mm
lline	<input type="checkbox"/>	0.5	mm	1.5	mm
cvalue	<input checked="" type="checkbox"/>	2	pF	18	pF

**Properties Dialog 2 (Bottom Right):** Shows optimization settings for 'L'.

Name	Include	Min	Unit	Max	Unit
Model	<input type="checkbox"/>				
DeviceLibraryName	<input type="checkbox"/>				
L	<input checked="" type="checkbox"/>	2	nH	18	nH
VComp	<input type="checkbox"/>				
Status	<input type="checkbox"/>				

**Text Box 3 (Bottom Left):** As capacitors value are variable click right on LPF design and select Design Properties. Then select the Optimization view and click the Include button to ensure that Cvalue is used in the optimization. Set Min to 2pf and Max to 18pf.

# Optimization: Setup Optimization

The next step is to set up a goal for the Optimization.  
Right click on Optimetrics in the Project Manager and then select **Add > Optimization**  
This will bring up the **Setup Optimization Analysis** dialog.

The screenshot shows the Ansoft Designer interface with a circuit diagram and the Setup Optimization dialog box. The circuit diagram includes a central inductor labeled LL1608\_F10NJ 10nH, connected to various components labeled with variables like W1=wline, W2=wline, W3=wline, and W=wline, P=line. The Setup Optimization dialog box is open, showing the Optimizer set to Random, Max. No. of Iterations set to 100, and an empty Cost Function table.

**Setup Optimization**

Goals | Variables | General

Optimizer: Random  Randomize Seed

Max. No. of Iterations: 100

Cost Function

Solution	Calculation	Calc. Range	Condition	Goal	Weight
----------	-------------	-------------	-----------	------	--------

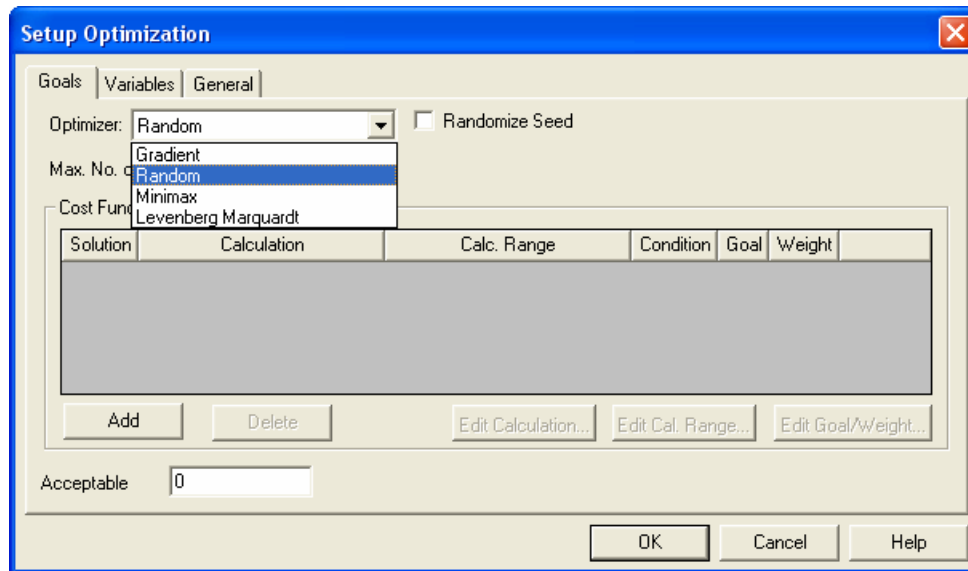
Add Delete Edit Calculation... Edit Cal. Range... Edit Goal/Weight...

Acceptable 0

OK Cancel Help

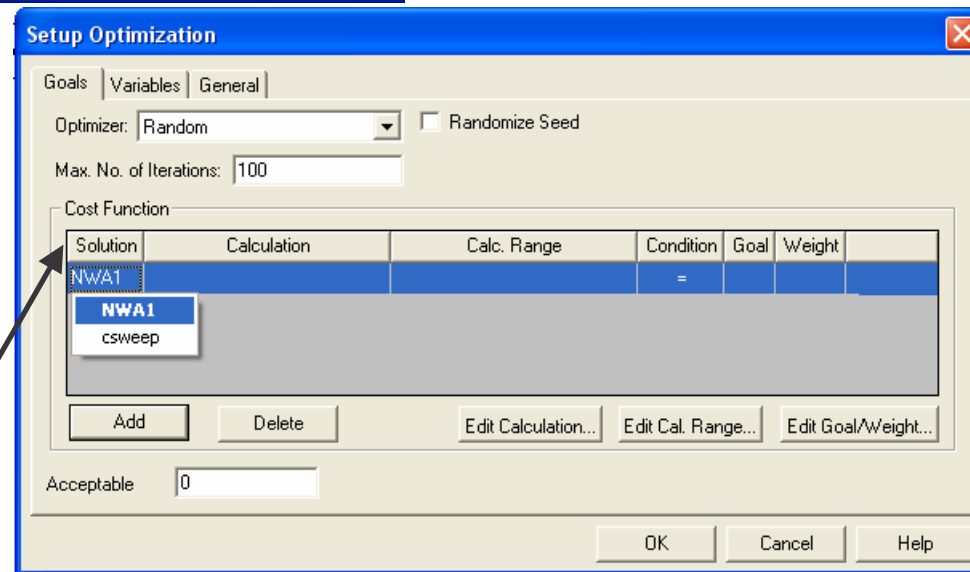
Add an optimization setup to the design.

# Optimization: Select Solution



Optimizer allows to select from the 4<sup>th</sup> different algorithms.

Select Random  
Select Max. No of iterations 100

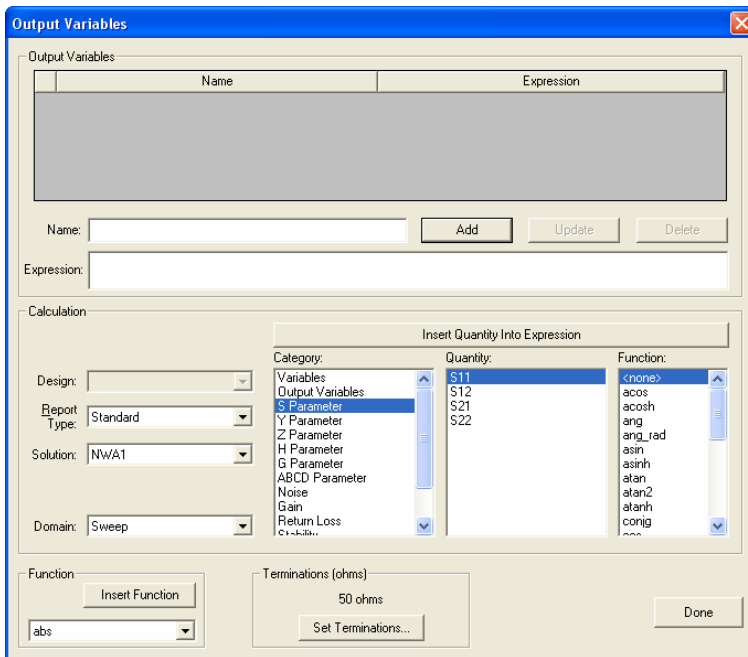
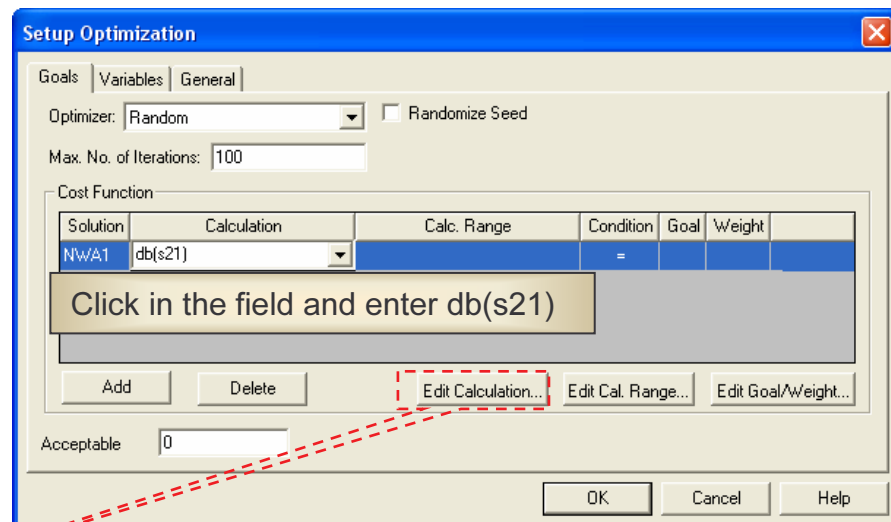
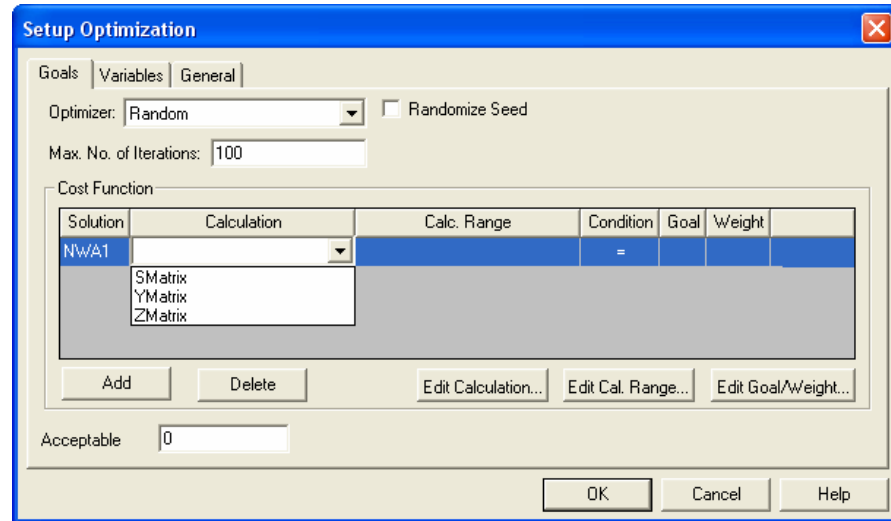


Clicking on **Solution** gives the user a dropdown dialog of all applicable Analyses.  
Click Add and Select **NWA1**

# Optimization: Define Calculation

**Calculation** allows to define the measurements to be optimized. It could be measurements, equations or complete S, Y, Z matrix

Ansoft Designer lets the user create custom expressions for plotting and optimizing.





# Optimization: Define Calculation Range

**Setup Optimization**

Goals | Variables | General

Optimizer: Random

Max. No. of Iterations: 100

Cost Function

Solution	Calculation	Calc. Range	Condition	Goal	Weight
NWA1	db(s21)	f(From 100MHz to 3GHz)	=		

Buttons: Add, Delete, Edit Calculation..., **Edit Cal. Range...**, Edit Goal/Weight...

Acceptable: 0

OK Cancel Help

**Calc. Range can be single value or band width. Default is the frequency range of Analysis setup**

**Edit Calculation Range**

Edit Range

Design: F

Range  Single Value

Value: 1GHz

Buttons: Add, Update, Delete, OK, Cancel

Variable	Range
F	Single value at 1GHz

**Select single Value,  
Value = 1GHz  
Click Update and Ok**

**Setup Optimization**

Goals | Variables | General

Optimizer: Random  Randomize Seed

Max. No. of Iterations: 100

Cost Function

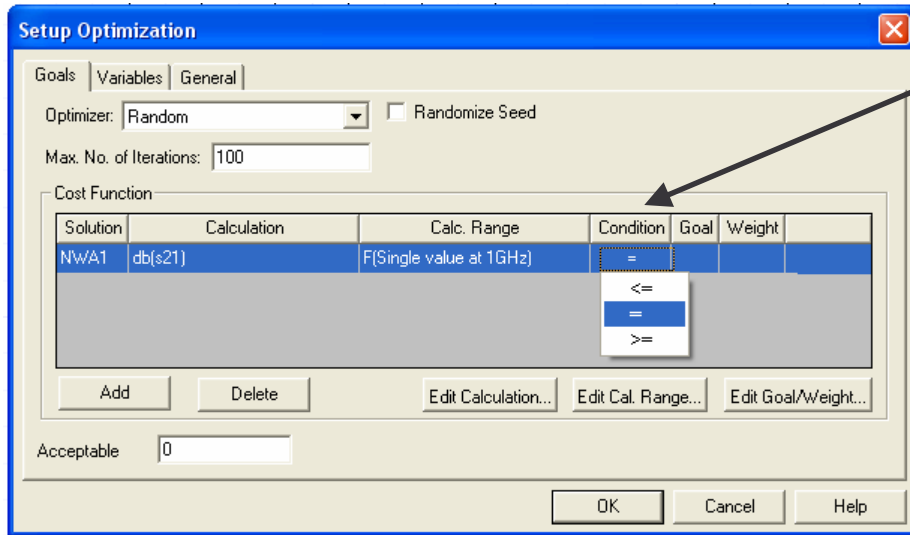
Solution	Calculation	Calc. Range	Condition	Goal	Weight
NWA1	db(s21)	f(Single value at 1GHz)	=		

Buttons: Add, Delete, Edit Calculation..., Edit Cal. Range..., Edit Goal/Weight...

Acceptable: 0

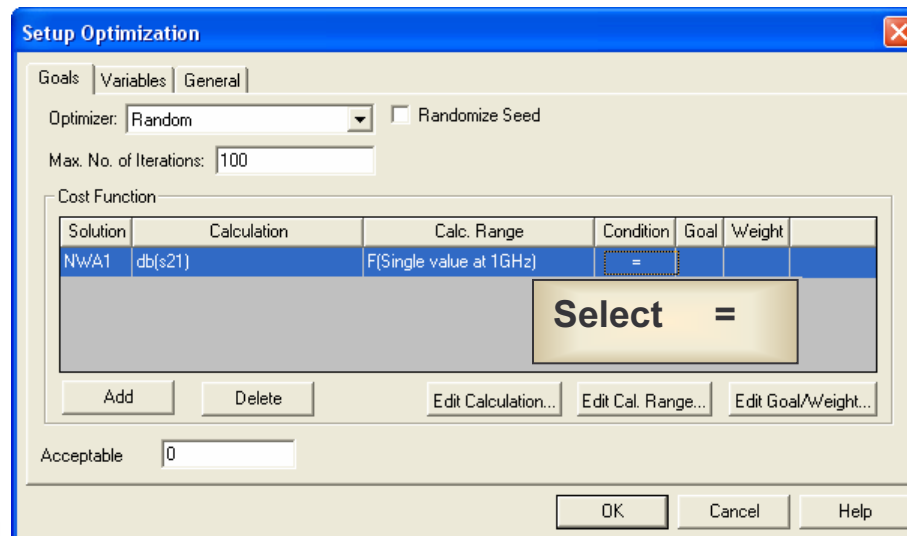
OK Cancel Help

# Optimization: Select Condition

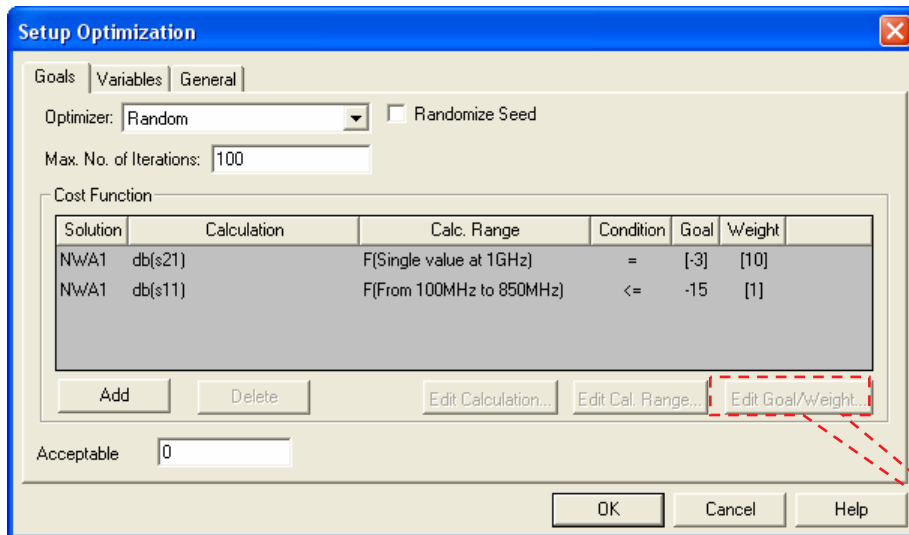


Clicking on **Condition** gives the a dropdown dialog of all equalities  
Condition can be

Equal : =  
Less Than or Equal: <=  
Greater than or Equal >=



# Optimization Setup: Goal, Weight



Enter -3 as the Goal and 10 as the weight

Enter an additional goal to be

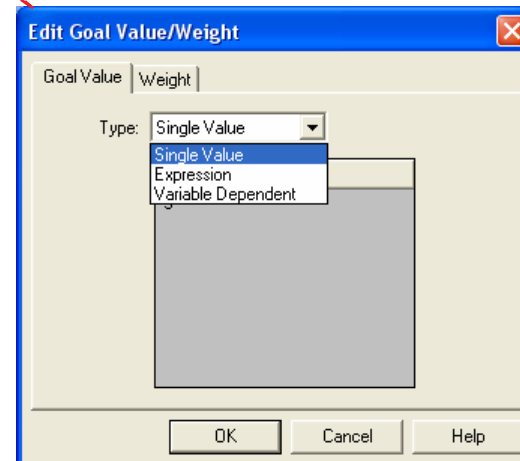
NWA1 db(s11) F (from 100MHz to 850MHz) <= -15 (1)

In this case the contribution of the s21 goal on the whole error is multiply by 10 due to weight setting.

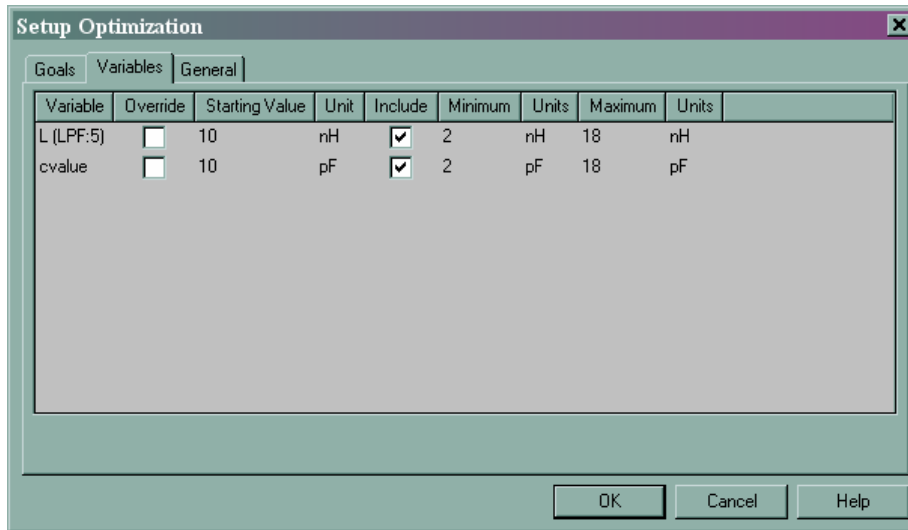
The Goal can be defined as :

Single numerical value,  
Expression (equation, sub-circuit name , S parameter file),  
Variable Dependent (for parametric sweep)

Weight allows to increase the contribution of a specific goal to the whole error function when multiple goals are defined

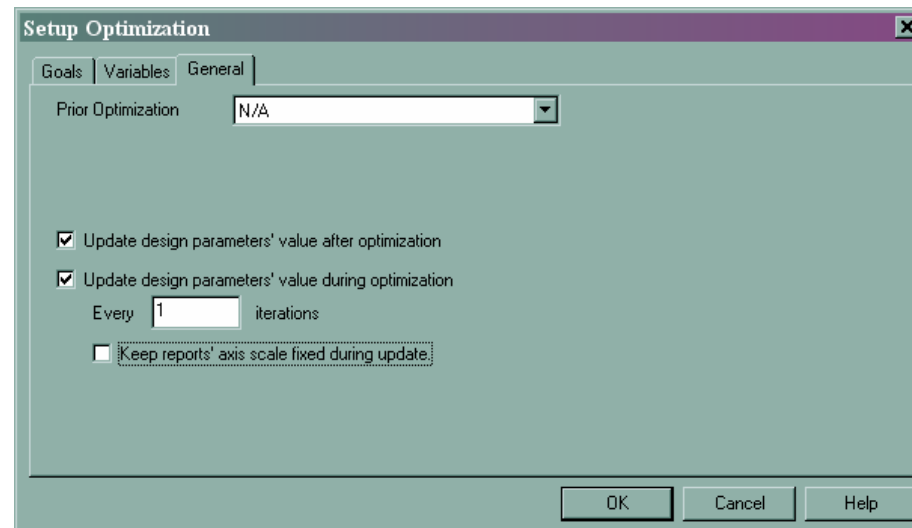


# Optimization Setup: Tab Variable and General



Hit the Variable tab.  
This allows to change the minimum and maximum values or to exclude one or more parameters from the optimization

Hit the General tab  
This is used to define the strategy of updating the design parameters during and after optimization



# Running Optimization

The screenshot displays the Ansoft Designer interface for a project named 'LPFProject'. The main workspace shows a circuit schematic for a low-pass filter (LPF). The circuit includes two ports, a series inductor labeled 'LL1608\_F10NJ 5.002nH', and several transmission line components. A context menu is open over the 'OptimetricSetup1' component, with the 'Analyze' option selected. A 'Progress' dialog box is overlaid on the schematic, indicating that the optimization is running on the local machine. The progress bar shows the process is in the 'Optimizing' phase. A text box at the bottom left provides instructions on how to initiate the optimization.

Click right on Optimetricsetup1 and select Analysis.  
The progress window appears showing the optimization progress.

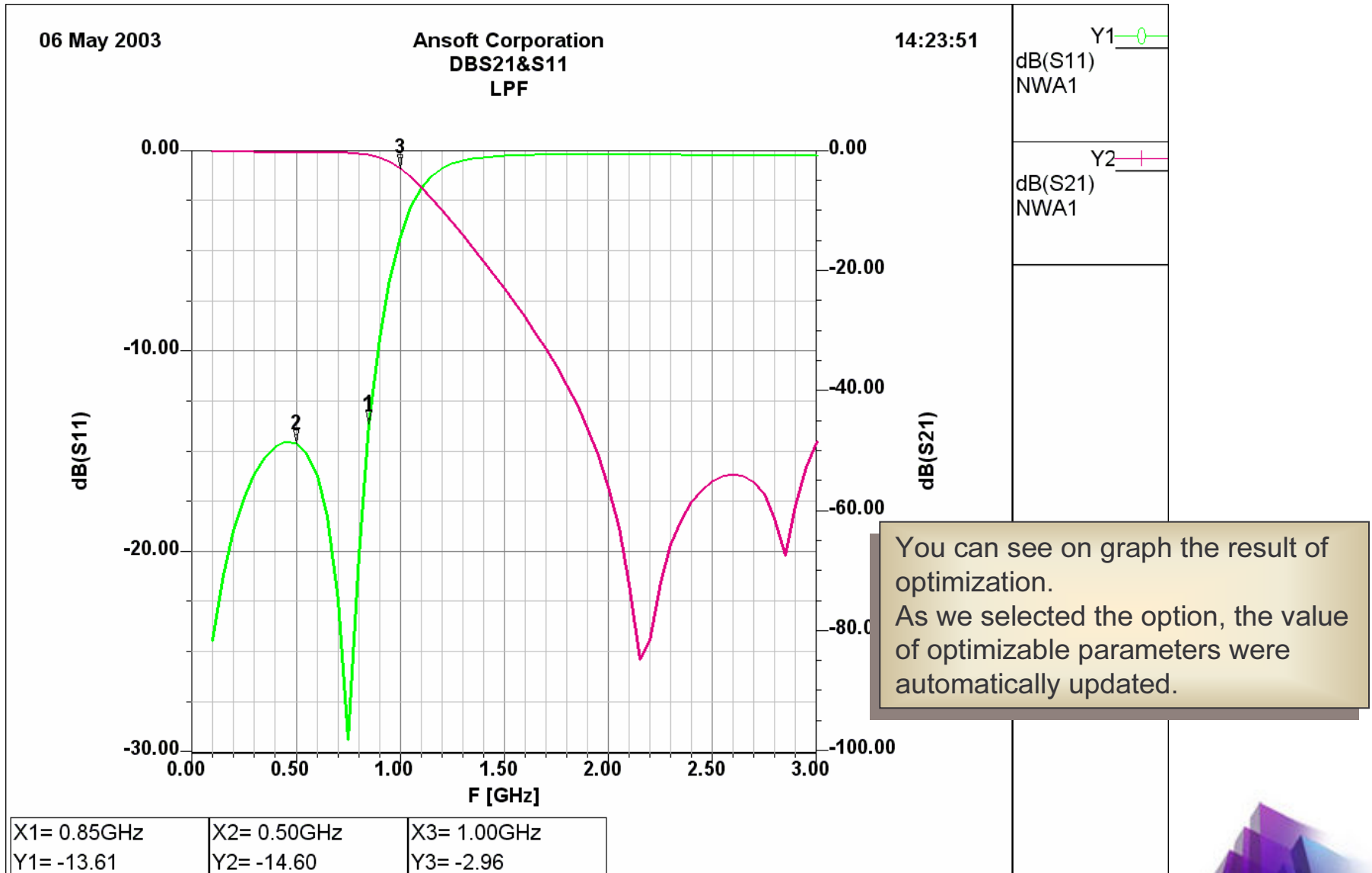
# View Optimization Results

Click right on optimization and select View Analysis Result

A window is displayed with the value of the cost function and the corresponding values of optimizable parameters

L (LPF:5)	cvalue	Cost
10nH	10pF	4694.3
7.83823nH	12.4373pF	2665
9.22918nH	2.02389pF	40.268
11.5908nH	3.14432pF	7.784
11.0134nH	3.75586pF	7.2944
12.6389nH	2.31328pF	5.2163
11.43nH	3.01103pF	4.448
10.8141nH	3.03262pF	3.1596
10.9126nH	3.13258pF	2.89
10.549nH	3.19279pF	2.2796
10.6583nH	3.03948pF	2.125
9.67299nH	3.60657pF	0.9135
10.185nH	3.5114pF	0.86884
8.90731nH	3.74865pF	0.71571
9.73655nH	3.4788pF	0.3869
9.01392nH	3.82951pF	0.36201
9.39699nH	3.61625pF	0.24439
8.64782nH	3.99155pF	0.19334
8.51866nH	4.04347pF	0.18998
8.90449nH	3.88327pF	0.185
8.69041nH	3.97493pF	0.18035
8.71397nH	3.95782pF	0.17545
8.71795nH	3.98357pF	0.16612
8.71675nH	3.99169pF	0.16548
8.71915nH	3.97545pF	0.16548

# View Optimization Result on Graph



# Set the parameters for Inductor

The results of optimization set L value of inductor to 8.72nH. Click on Choose model and select the normalized value closest To this value. Select 8.2nh 10% and click ok.

Model	L	Tolerance
LL1608_F8N2J	8.200nH	5.00%
LL1608_F8N2K	8.200nH	10.00%
LL1608_F8N2M	8.200nH	20.00%
LL1608_F10NJ	10.000nH	5.00%
LL1608_F10NK	10.000nH	10.00%
LL1608_F10NM	10.000nH	20.00%
LL1608_F12NJ	12.000nH	5.00%
LL1608_F12NK	12.000nH	10.00%

Select tolerance to 10% automatically defines tolerance for statistical analysis to 10%. All vendor library components using tolerance in their definition are automatically included in the statistics. You can verify it checking statistic window.

Name	Include	Distribution	Distribution Criteria
Model	<input type="checkbox"/>		
DeviceLibraryName	<input type="checkbox"/>		
L	<input checked="" type="checkbox"/>	Uniform	Tolerance = 10.00%
VComp	<input type="checkbox"/>		
Status	<input type="checkbox"/>		

Number of selected items: 1



# Set the parameters for Capacitor

**Select 3.9pf tolerance 0.25pf**

Model	C	Tolerance
C0603278D9B200	2.70pF	0.50pF
C0603338B9B200	3.30pF	0.10pF
C0603338D9B200	3.30pF	0.25pF
C0603338D9B200	3.30pF	0.50pF
C0603398B9B200	3.90pF	0.10pF
C0603398C9B200	3.90pF	0.25pF
C0603398D9B200	3.90pF	0.50pF
C0603478B9B200	4.70pF	0.10pF

**Properties**

Name	Include	Distribution	Distribution Criteria
Model	<input type="checkbox"/>		
DeviceLibraryName	<input type="checkbox"/>		
C	<input checked="" type="checkbox"/>	Uniform	Tolerance = 0.25pF
VComp	<input type="checkbox"/>		
Status	<input type="checkbox"/>		

**Uniform Distribution**

Tolerance: 0.25 pF

**Param Values**

Name	Value	Unit
Model	C0603109...	
DeviceLibr...	philips_sm...	
C	cvalue	
VComp	Choose Mo...	

Once C parameter is set you can double click on one of the capacitor and check the statistic window. If needed you can choice between Uniform and Gaussian distribution. Click on tolerance button to change the tolerance

Cvalue found by optimization=3.97. Select the two capacitors and click Choose Model . You will change in one action both capacitor C parameter

# Statistical Analysis: Define goals

The Variables tab allows to change parameter settings or exclude parameter

Hit Edit Calc. Range and select 1GHz

Enter db(s21) in Calculation field

Enter iteration number 50. Click Add and select NWA1 for Solution

Variable	Override	Starting Value	Unit	Include	Distribution	Dist. Criteria
C (LPPF:3)	<input type="checkbox"/>	3.3	pF	<input checked="" type="checkbox"/>	Uniform	Tolerance = 0.25pF
L (LPPF:4)	<input type="checkbox"/>	3.3	pF	<input checked="" type="checkbox"/>	Uniform	Tolerance = 0.25pF
L (LPPF:5)	<input type="checkbox"/>	8.2	nH	<input checked="" type="checkbox"/>	Uniform	Tolerance = 10.00%

Solution	Calculation	Calculation Range
NWA1	db(s21)	[Single value at 100MHz]

Variable	Range
F	Single value at 1GHz

# Statistical Analysis: Define goals

Maximum Iterations: 50

Solution	Calculation	Calculation Range
NWA1	db(s21)	F(Single value at 1GHz)
NWA1	db(s11)	F(Single value at 100MHz)
NWA1	db(s11)	F(Single value at 450MHz)
NWA1	db(s11)	F(Single value at 850MHz)

Buttons: Add, Delete, Edit Calculation..., Edit Cal. Range..., OK, Cancel, Help

Enter statistical goals as shown above

# Run Statistical Analysis

The screenshot displays the Ansoft Designer interface for a project named 'DBS218S11'. The main window shows a plot of dB(S11) versus frequency (MHz) from 0.00 to 2.00. The plot features multiple overlapping curves in various colors, representing statistical distributions of the results. A callout box with a black border and white background points to the 'StatisticalSet1' icon in the left-hand project tree, containing the text: 'Right-click on Statisticalsetup1 icon and choose analyze to run analysis'. A 'Progress' dialog box is open in the foreground, showing two progress bars: 'LPF Statistical Analysis on Local Machine - RUNNING' and 'LPF on Local Machine - RUNNING'. The dialog also displays the text 'Solving: //LPF.3//C=3.488627277pF //LPF.4//C=3.488627277pF' and '1 of 100'. On the right side of the plot area, there are several data points listed, including dB(S21) and dB(S11) values for different components like NWA1 and C:3. A date '28 Jan 2003' is visible in the top left corner of the plot area. At the bottom left, a text box contains the following text: 'The Statistical Analysis is running and the plot automatically updated. When Statistical Analysis is finished you can look at the distribution of the results.'

28 Jan 2003

Right-click on Statisticalsetup1 icon and choose analyze to run analysis

dB(S11)

dB(S21)

Y2

dB(S21) NWA1

Y1

dB(S11) C:3=3.3pF,C:4=

Y2

dB(S21) C:3=3.3pF,C:4=

Y1

dB(S11) C:3=3.18644825

Y2

dB(S21) C:3=3.18644825

Y1

dB(S11) C:3=3.31583147

Progress

LPF Statistical Analysis on Local Machine - RUNNING

Solving: //LPF.3//C=3.488627277pF //LPF.4//C=3.488627277pF

Abort

LPF on Local Machine - RUNNING

1 of 100

Abort

The Statistical Analysis is running and the plot automatically updated. When Statistical Analysis is finished you can look at the distribution of the results.

# View Analysis Result: Data Table

The screenshot shows the Ansoft Designer interface with the 'Post Analysis Display Dialog' window open. The dialog displays a table of analysis results for various components. A callout box with a black border and white background points to the 'StatisticalSetup1' icon in the left-hand tree view, containing the following text:

**Right-click on Statisticalsetup1 icon and choose View Results to see a data table or histogram plot**

C (LPF:3)	C (LPF:4)	L (LPF:5)	db(s21) F(Single value at 1GHz)	db(s11) F(Single value at 100MHz)	db(s11) F(Single value at 450MHz)	db(s11) F(Single value at 850MHz)
3.3pF	3.3pF	8.2nH	-1.2315	-27.151	-17.015	-22.642
3.186448256pF	3.186448256pF	7.827550279nH	-0.94825	-27.408	-16.997	-28.032
3.315831477pF	3.315831477pF	8.251927244nH	-1.2766	-27.116	-17.02	-22.008
3.471826838pF	3.471826838pF	8.763592029nH	-1.7977	-26.778	-17.097	-17.051
3.242617573pF	3.242617573pF	8.011785638nH	-1.0796	-27.28	-17.002	-25.207
3.45130314pF	3.45130314pF	8.6962743nH	-1.7212	-26.822	-17.083	-17.597
3.524578082pF	3.524578082pF	8.936616108nH	-2.0053	-26.667	-17.135	-15.756
3.425774407pF	3.425774407pF	8.612540056nH	-1.6294	-26.876	-17.068	-18.313
3.292469558pF	3.292469558pF	8.17530015nH	-1.2105	-27.168	-17.013	-22.955
3.167664113pF	3.167664113pF	7.765938292nH	-0.90803	-27.452	-16.996	-28.938
3.109541612pF	3.109541612pF	7.575296487nH	-0.79484	-27.587	-16.999	-30.521
3.479471725pF	3.479471725pF	8.788667257nH	-1.8268	-26.762	-17.102	-16.854
3.354483169pF	3.354483169pF	8.378704794nH	-1.3928	-27.031	-17.034	-20.579
3.0618717pF	3.0618717pF	7.418939177nH	-0.71416	-27.7	-17.007	-29.601
3.14665212pF	3.14665212pF	7.697018952nH	-0.86517	-27.5	-16.996	-29.799
3.123656423pF	3.123656423pF	7.621593066nH	-0.8208	-27.554	-16.998	-30.398
3.201997436pF	3.201997436pF	7.8789551592nH	-0.98293	-27.373	-16.997	-27.243
3.353582873pF	3.353582873pF	7.375751823nH	-1.39	-27.033	-17.033	-20.61
3.470072024pF	3.470072024pF	8.793836238nH	-1.7911	-26.782	-17.095	-17.097
3.120787683pF	3.120787683pF	7.612186599nH	-0.81545	-27.561	-16.998	-30.439
3.306767479pF	3.306767479pF	8.222197333nH	-1.2506	-27.136	-17.017	-22.368
3.154800562pF	3.154800562pF	7.723745842nH	-0.88153	-27.481	-16.996	-29.492
3.54426252pF	3.54426252pF	9.001181066nH	-2.0058	-26.626	-17.151	-15.308
3.378653829pF	3.378653829pF	8.457984558nH	-1.4697	-26.978	-17.044	-19.762
3.078107547pF	3.078107547pF	7.472192755nH	-0.74046	-27.661	-17.004	-30.105
3.289936522pF	3.289936522pF	8.166991791nH	-1.2035	-27.174	-17.012	-23.062
3.281040376pF	3.281040376pF	8.137812433nH	-1.1793	-27.194	-17.01	-23.443
3.319676809pF	3.319676809pF	8.264539933nH	-1.2878	-27.108	-17.021	-21.858
3.548397778pF	3.548397778pF	9.014744713nH	-2.1041	-26.617	-17.155	-15.216
3.295948668pF	3.295948668pF	8.186711631nH	-1.2202	-27.16	-17.014	-22.81
3.085187841pF	3.085187841pF	7.49541612nH	-0.7523	-27.645	-17.003	-30.273
3.225145726pF	3.225145726pF	7.954477981nH	-1.0369	-27.32		
3.226061281pF	3.226061281pF	7.957481002nH	-1.0391	-27.318		

# View Analysis Result: Histogram

The screenshot shows the Ansoft Designer interface with the Post Analysis Display Dialog open. The dialog is titled "Post Analysis Display Dialog" and contains a "StatisticalSetup1" section. The "View" section is set to "Plot", and the "Num. of bins on X axis" is set to 10. The plot area shows a histogram with the following data series selected in the legend:

- db(s11) F(Single value at 100MHz)
- db(s11) F(Single value at 450MHz)
- db(s11) F(Single value at 850MHz)
- db(s21) F(Single value at 1GHz)

The histogram shows the "Number of outcomes" on the y-axis (ranging from 0 to 8) and the "db(s11) F(Single value at 100MHz)" on the x-axis (ranging from -27.80 to -26.60). The histogram bars are blue and have a height of 5 for most bins, with some variations.

Callout boxes provide instructions:

- "Select Plot to see histogram" points to the "Plot" radio button in the "View" section.
- "Select the number of bins" points to the "Num. of bins on X axis" input field.
- "Select which result to view" points to the legend area where results are listed.

Name	Value	Unit
Name	StatisticalS...	
Max Iters	50	
Prior Para...	N/A	

**End of Exercise 1**



# Layout Basics

ANSOFT CORPORATION





# View Layout Window

**Click on the layout icon to see the layout view**

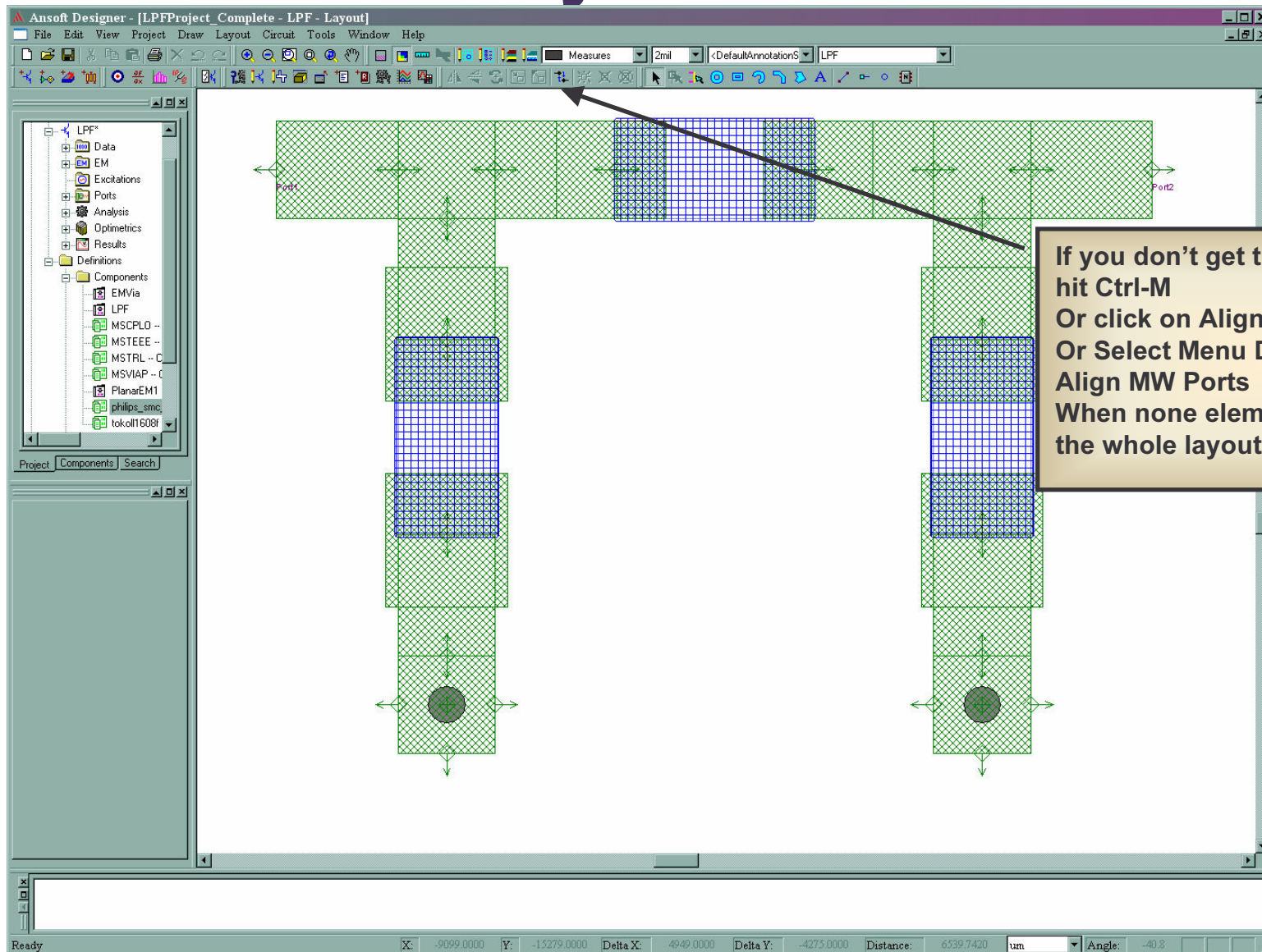
**Alternatively, you can right-click on the circuit icon and choose layout**

**Replace ground elements by MS Via Pad**

**MS Via Pad**

- MS Air Bridge
- MS Bond Pad
- MS Compensated TEE Junc
- MS Cross Junction
- MS Crossover, Q
- MS Crossover, Tand
- MS Dielectric Resonator, Ba
- MS Dielectric Resonator, Ba
- MS Slit in Line
- MS Step
- MS Tee - Ref. Plane at Cent
- MS Tee - Ref. Planes at Edg
- MS Transformer
- MS Via
- MS Via Pad
- MS Via Pad, w/Ref Node
- MS Via, w/Ref Node
- MS Wire
- MS Wrap Around Ground
- MS Y Junction

# Layout Window



# Connection points and wires

The screenshot shows the Ansoft Designer interface for a PCB layout. The main workspace displays a green grid with a blue footprint and a wire. The 'Edit Layers - LPF' dialog is open, showing a table of layers. The 'Settings' dialog is also open, showing drawing extent and connection point settings.

Layers	Stackup	Name	Type	Color	Pattern	
		Measures	measures	Black	Black	neither
		Rats	rat	Blue	Black	neither
		Errors	error	Red	Black	neither

**Settings**

Drawing Extent:

Left: -100000um Top: 100000um

Right: 100000um Bottom: -100000um

Fit to Entities

Symbol Scale Factor: 2

Default Hole Size: 25mil

AutoPosition MW/ Comp.

Draw Connection Points

Advanced Settings

OK Cancel

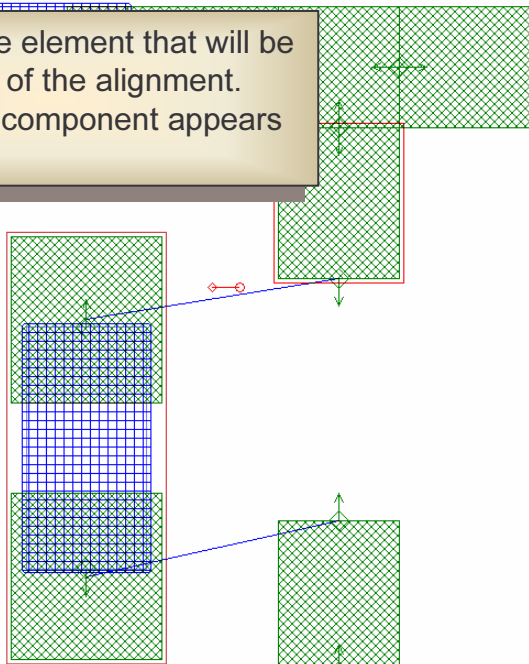
**When you select a footprint and move it, if this footprint is connecting the connection is drawn as a wire on layer rats.**

**The connection points are visible. To make it non visible:  
Select Menu Layout  
Select Settings  
Uncheck Draw Connection Points**

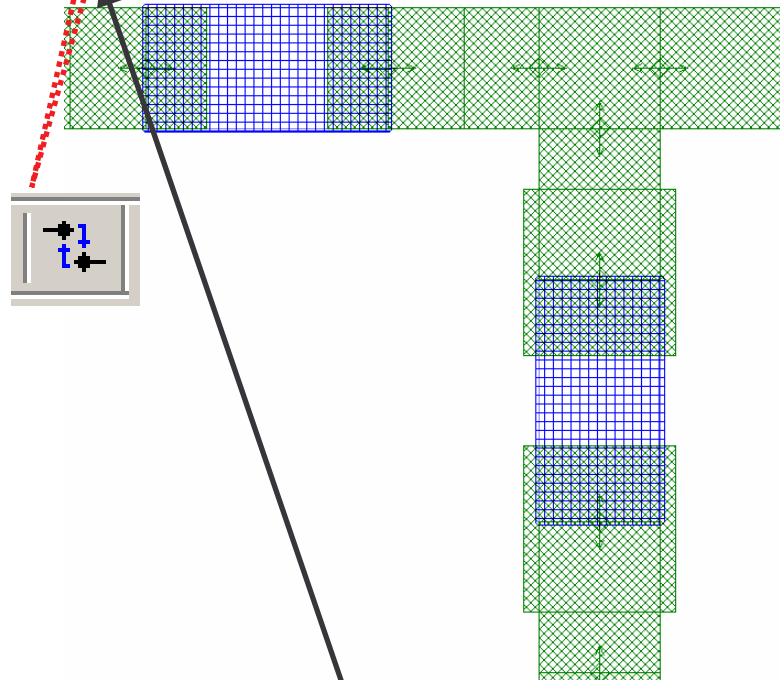
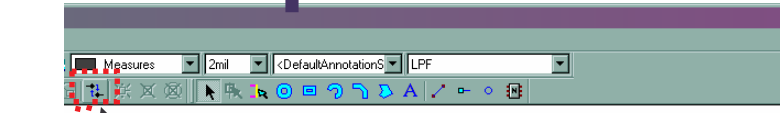
# Align Footprint



Select first the element that will be the reference of the alignment. The selected component appears in red.



Using Ctrl + Left Mouse button Select the second element to aligned. This second element appears in low red.



Click on the Align Microwave Port to icon to align the Footprint (Hotkey Ctl-m)

You can apply any positioning modification to footprint **move, rotate, mirror** and then use the Align Microwave Port function to align them. Any positioning modification are save in the footprint properties.



13059.7481 Delta X: 0.0000 Delta Y: 0.0000 Distance:



0.0000 um Angle:



# Layout Footprint Properties

The image shows the ANSOFT software interface for editing layout footprint properties. It features three main windows: 'Select Definition', 'Parameter Values', and 'Layer Mapping'. The 'Select Definition' window lists various footprint types like '0603\_RF' and 'CPLD'. The 'Parameter Values' window shows settings for 'CurrentFootprint' and 'Location'. The 'Layer Mapping' window allows mapping footprint layers to circuit board layers.

**Select Definition**

Name	Loca...	Origin	Pin Count	Footprint Image
0603_RF	Project	Footprints	2	
0603_RF	Project	Footprints	2	
CPLD	Project	Distributed Footprints	2	
TEEE	Project	Distributed Footprints	3	
TRL	Project	Distributed Footprints	2	
VIAP	Project	Distributed Footprints	4	
0302	SysLibrary	Footprints	2	
0302_RF	SysLibrary	Footprints	2	
0402	SysLibrary	Footprints		
0402_RF	SysLibrary	Footprints		
04021	SysLibrary	Footprints		
04021_RF	SysLibrary	Footprints		
0404	SysLibrary	Footprints		

**Parameter Values**

Name	Value	Unit	Description
LayerMapping	Show...		
Edit layer mapping	Edit...		
CurrentFootprint	0603_RF		
PlacementLayer	Assembly		
Location	-7.6, -13.8	mm	
Angle	270	deg	
Flipped	<input type="checkbox"/>		
Scaling	1		

**Layer Mapping**

Parent Layer	Local Layer
Measures	Measures
Flats	Flats
Errors	Errors
Symbols	Symbols
Assembly	Assembly Top
Silkscreen	< none >
Trace	Top
Dielectric	< none >
Ground	< none >

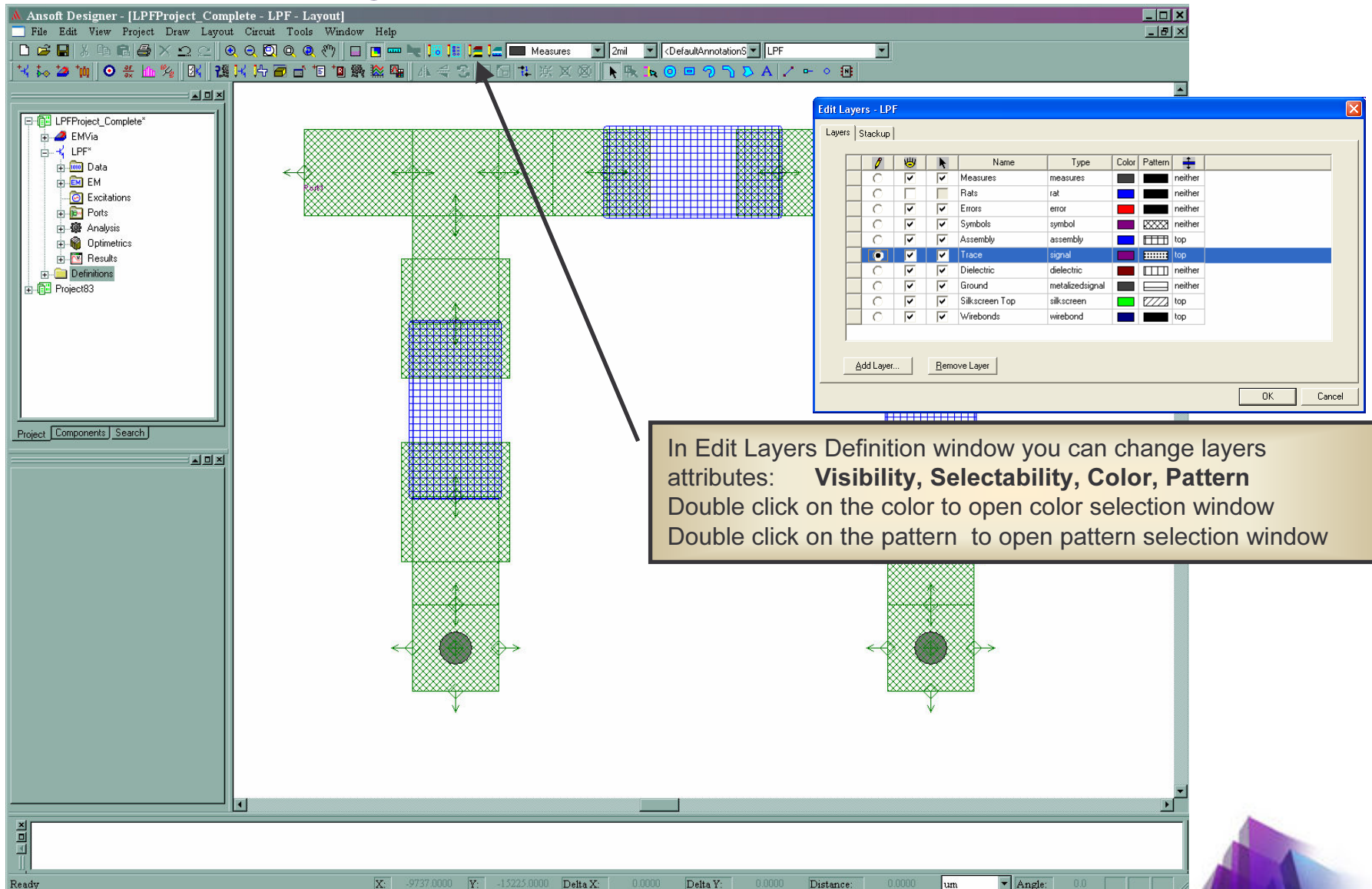
**Callout 1:** Click on **Current Footprint** value to open the footprint library window. Then select the new footprint you want to use.

**Callout 2:** **Location** = x,y coordinate of the footprint on layout. You can use it to move the footprint from a specific value. It is allowed to use +/-.  
Example: -7.6mm+1mm,13.8 will move the footprint of 1mm on the x axis  
**Angle** changing this value will rotate the footprint  
Flipped when checked indicates that the footprint was flipped.  
**Scaling** multiply all dimension by scaling value

**Callout 3:** **Edit layer mapping** allows to change the default mapping between Footprint layers and circuit layers

ANSOFT CORPORATION

# Layout Window Option



The screenshot shows the Ansoft Designer interface with a layout window displaying a grid. A dialog box titled "Edit Layers - LPF" is open, showing a table of layer attributes. The table has columns for Name, Type, Color, and Pattern. The "Trace" layer is selected.

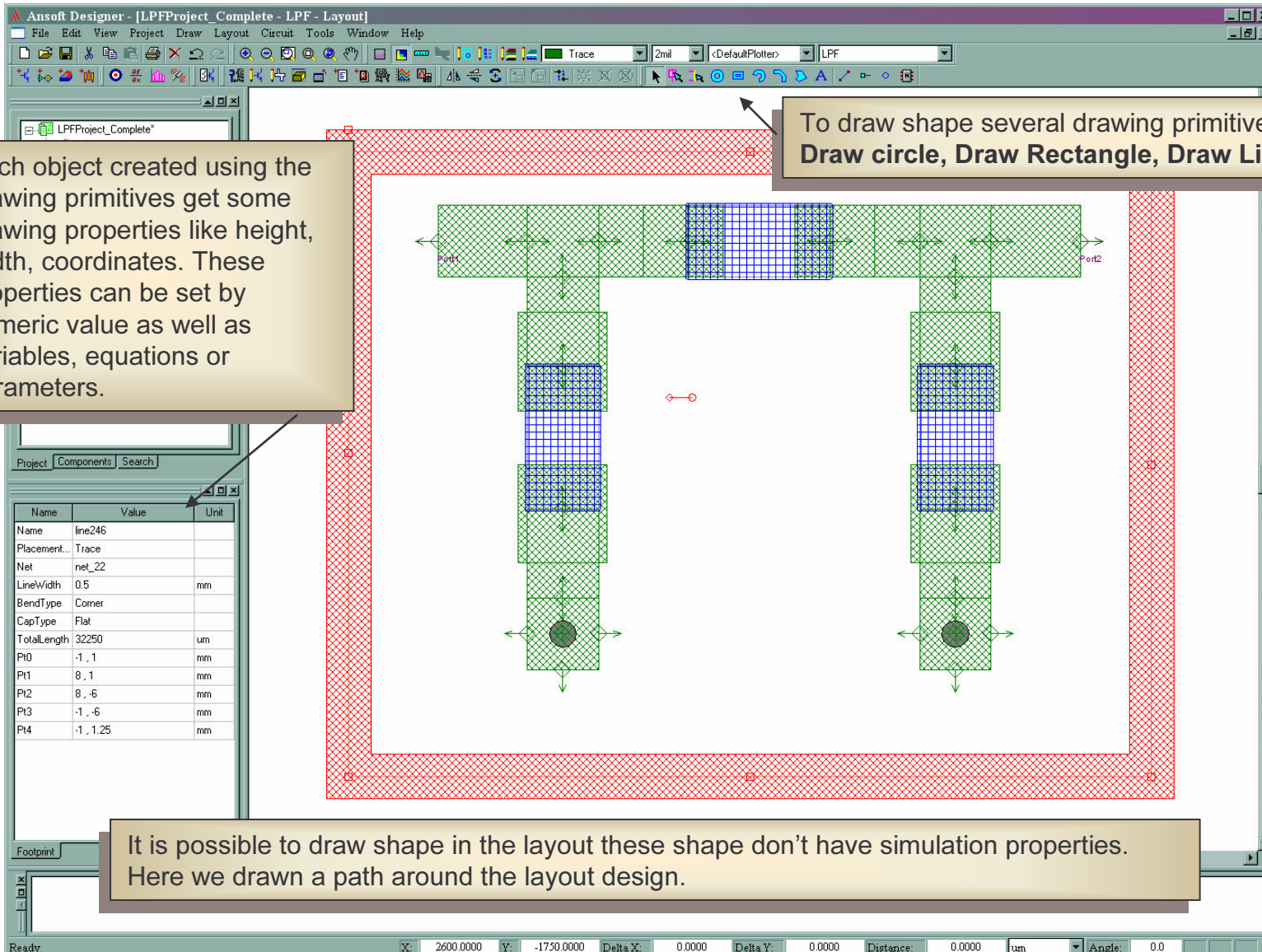
	Name	Type	Color	Pattern	
<input type="checkbox"/>	Measures	measures		neither	
<input type="checkbox"/>	Rats	rat		neither	
<input type="checkbox"/>	Errors	error		neither	
<input type="checkbox"/>	Symbols	symbol		neither	
<input type="checkbox"/>	Assembly	assembly		top	
<input checked="" type="checkbox"/>	Trace	signal		top	
<input type="checkbox"/>	Dielectric	dielectric		neither	
<input type="checkbox"/>	Ground	metalizedsignal		neither	
<input type="checkbox"/>	Silkscreen Top	silkscreen		top	
<input type="checkbox"/>	Wirebonds	wirebond		top	

In Edit Layers Definition window you can change layers attributes: **Visibility, Selectability, Color, Pattern**  
Double click on the color to open color selection window  
Double click on the pattern to open pattern selection window

# Add Layout only element

Each object created using the drawing primitives get some drawing properties like height, width, coordinates. These properties can be set by numeric value as well as variables, equations or parameters.

To draw shape several drawing primitives are available :  
**Draw circle, Draw Rectangle, Draw Line, Draw Polygon**



It is possible to draw shape in the layout these shape don't have simulation properties. Here we drawn a path around the layout design.

# Drawing Primitives

**Draw Rectangle**  
The properties of the rectangle are layer, Center, Width, Height, Angle

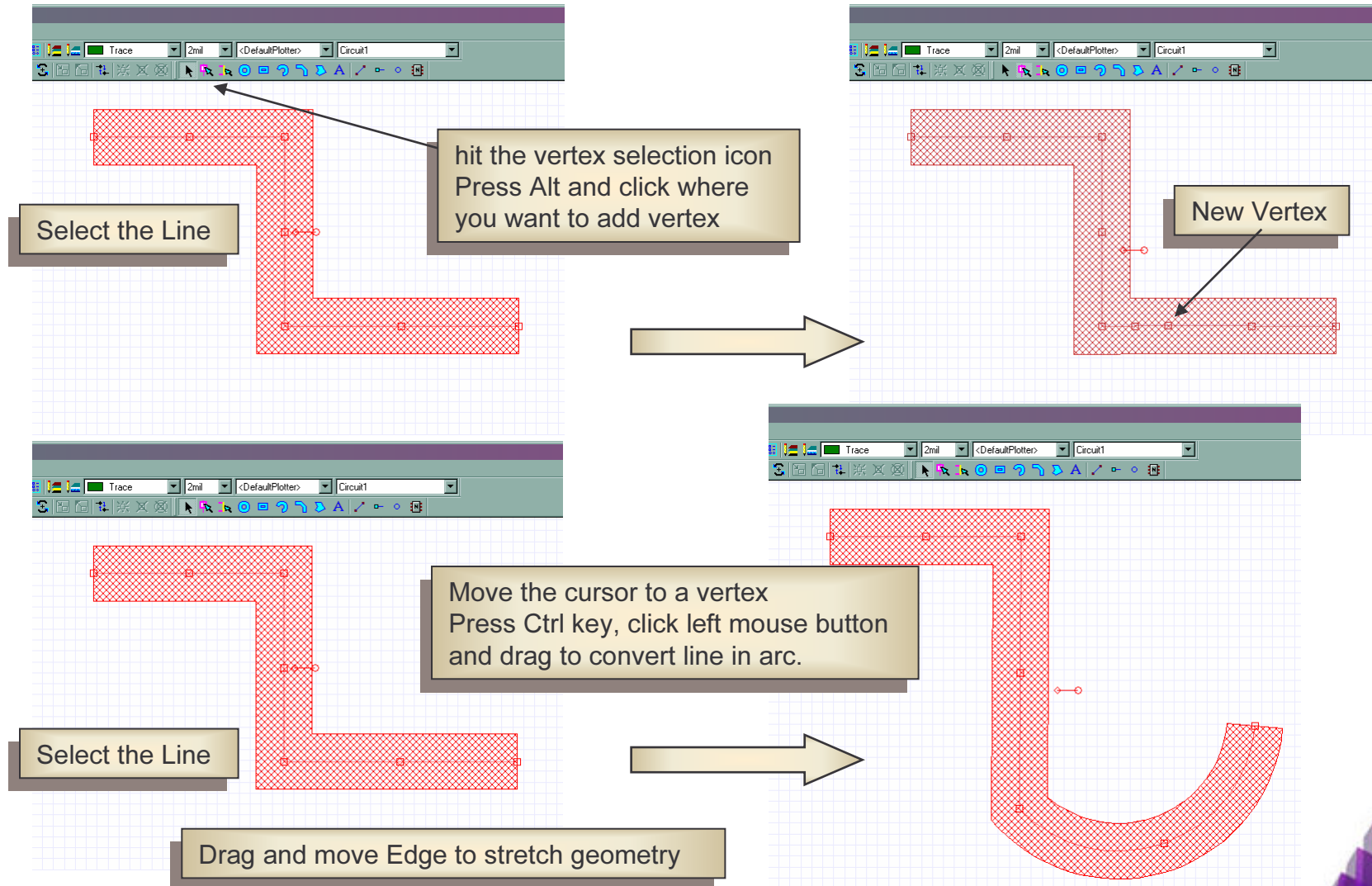
**Draw Line**  
The properties of the Line are layer, LineWidth, (x,y) coordinate of vertex, Angle, Bend Type, Cap Type

**Draw Circle**  
The properties of the Circle are layer, Center, Radius

**Draw Polygon**  
The properties of the Polygon are layer, (x,y) coordinate of vertex, Angle



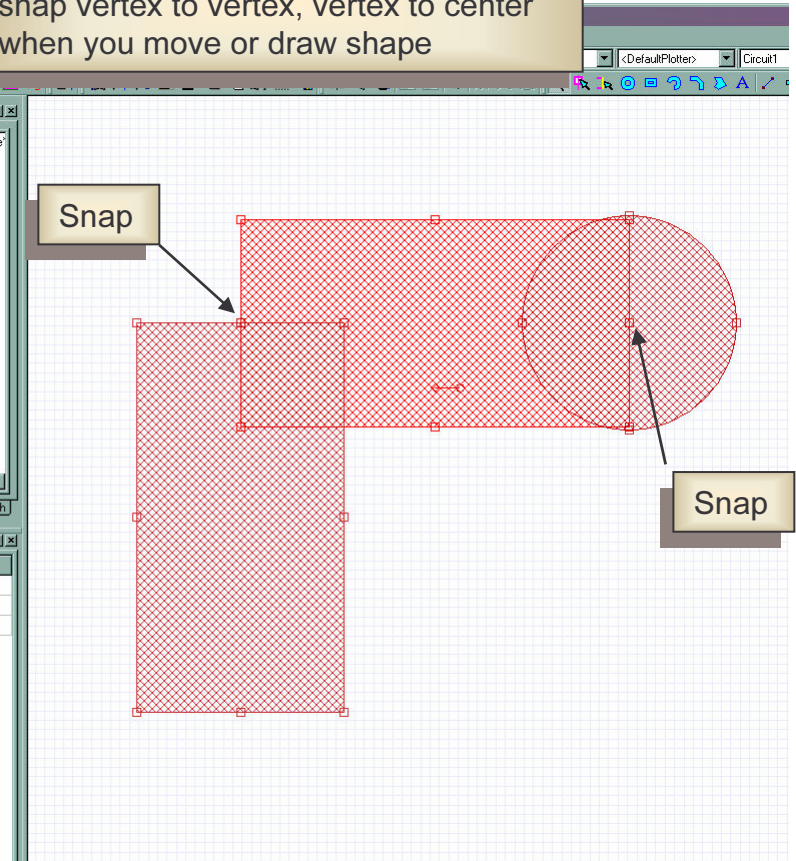
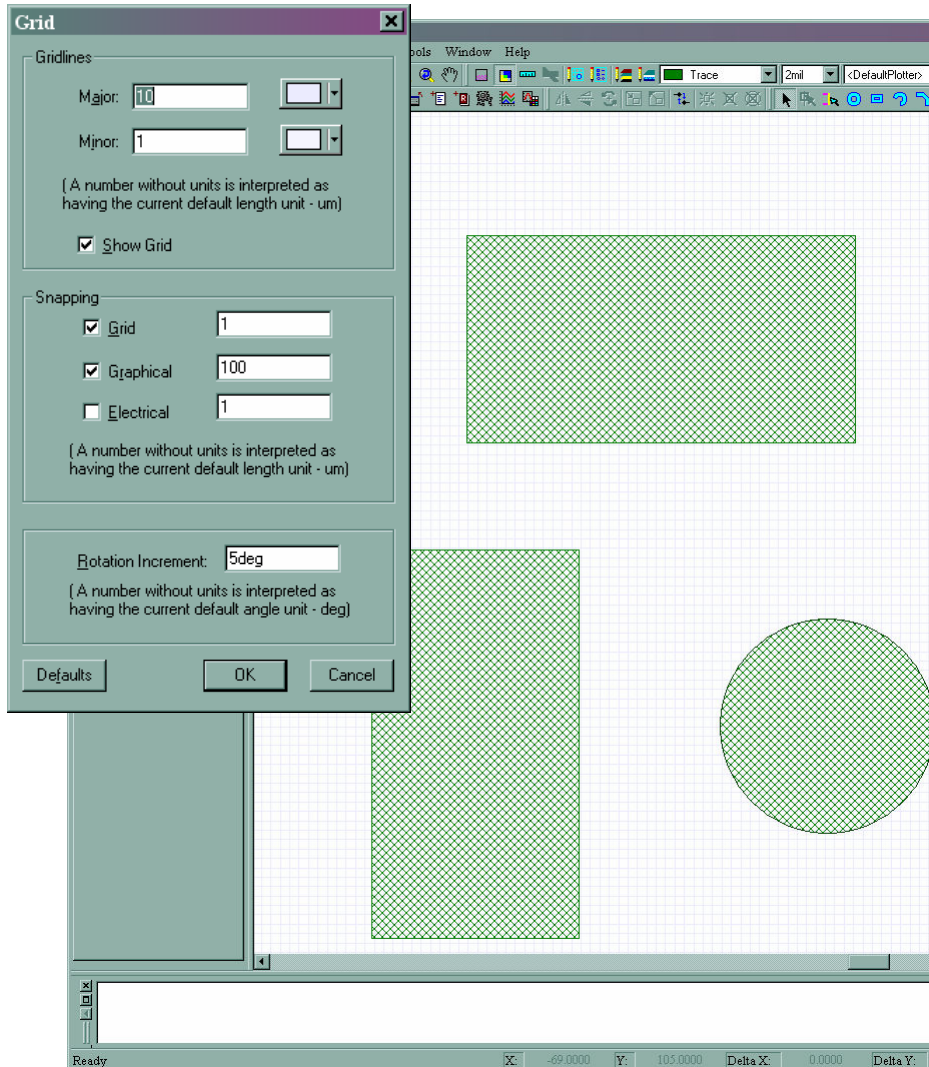
# Specific Operation on Primitive



# Snapping

**Snap to grid:** snap the shape on the grid

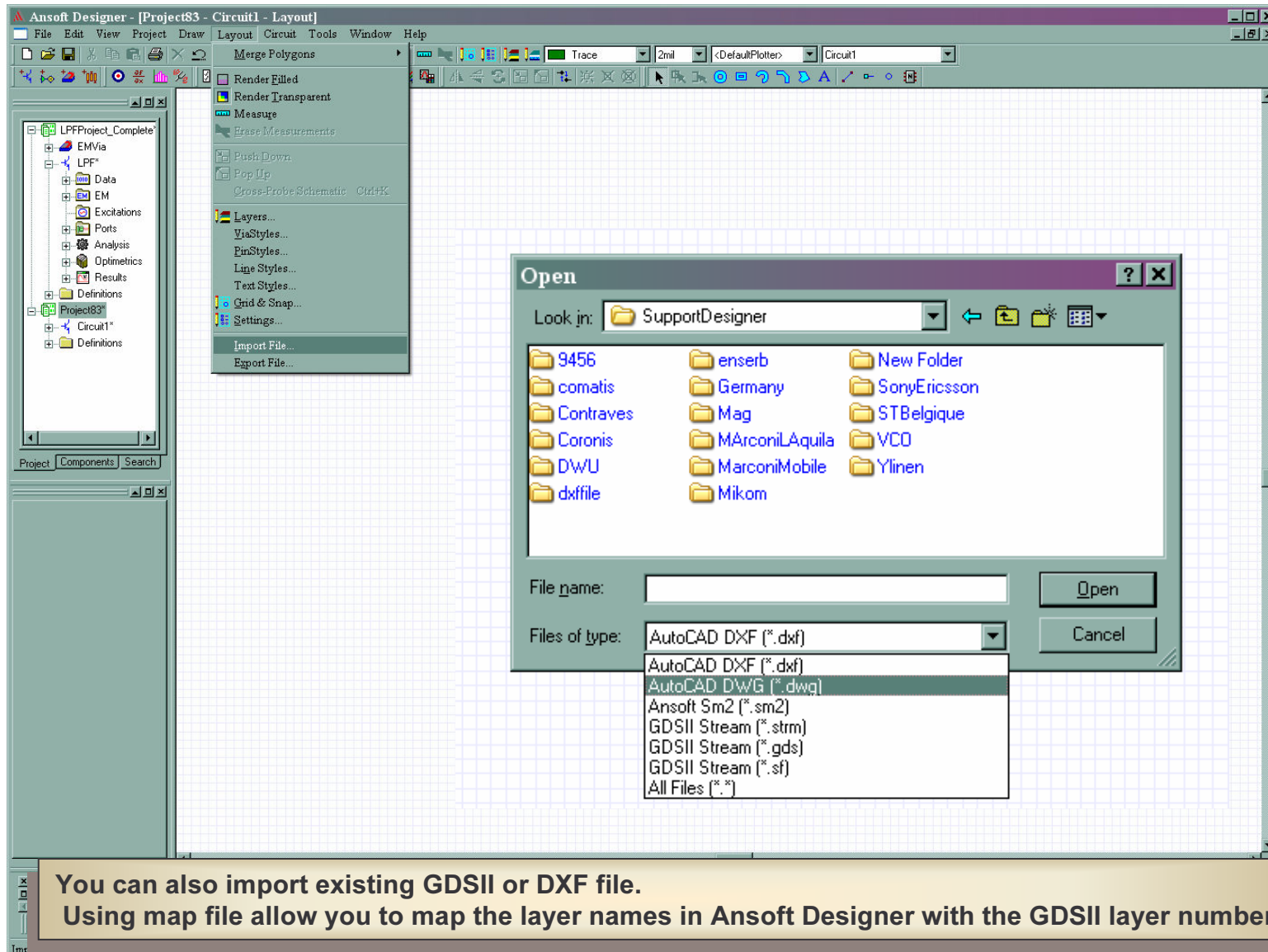
**Graphical snap:** snap vertex to vertex, vertex to center when you move or draw shape



# Boolean Operations

The screenshot displays the Ansoft Designer interface with two windows. The left window shows a project tree and a workspace with two overlapping green polygons: a rectangle and a semi-circle. A text box at the bottom left of this window states: "Boolean operation allows union, intersection and subtract." The right window shows the same workspace after a Boolean operation. A red hatched polygon represents the result of a 'union' operation between the two shapes. A context menu is open over the red polygon, listing 'Union', 'Intersection', and 'Subtract'. A text box with an arrow pointing to the red polygon explains: "Subtract Subtract is order selected dependant". Another text box with an arrow pointing to the red polygon states: "union".

# Import DXF and GDSII Files



The screenshot displays the Ansoft Designer interface with the 'Open' dialog box open. The dialog shows the 'SupportDesigner' directory containing various folders. The 'Files of type' dropdown is set to 'AutoCAD DXF (\*.dxf)'. The 'File name' field is empty. The 'Open' and 'Cancel' buttons are visible.

**You can also import existing GDSII or DXF file.**  
Using map file allow you to map the layer names in Ansoft Designer with the GDSII layer number

# Edit and Create Footprint

**Select the Technology File (use existing stackup)**

**Select Edit Definitions**

**Add a new footprint or use An existing one**

**Choose Layout Technology File**

**Get Name**

Enter the name for this new Footprint

Footprint

OK Cancel

Name	Location	Origin	Pin Count	Footprint Image
TEEC	SysLibrary	Distributed Fo...	3	
TEECPN	SysLibrary	Distributed Fo...	3	
TEEE	Project		3	
TEEE	SysLibrary	Distributed Fo...	3	
TEECPN	SysLibrary	Distributed Fo...	3	
TO-220D_2_RF	SysLibrary	Footprints	2	

MS - Alumina (Er=9.8) 0.010 inch, gold  
MS - Alumina (Er=9.8) 0.025 inch, gold  
MS - FR4 (Er=4.4) 0.030 inch, 0.5 oz copper  
MS - FR4 (Er=4.4) 0.060 inch, 0.5 oz copper  
MS - RT\_duroid 5880 (Er=2.20) 0.010 inch, 0.5 oz copper  
MS - RT\_duroid 5880 (Er=2.20) 0.020 inch, 0.5 oz copper  
MS - RT\_duroid 6010 (Er=10.2) 0.010 inch, 0.5 oz copper  
MS - RT\_duroid 6010 (Er=10.2) 0.025 inch, 0.5 oz copper  
PCB - DoubleSided  
PCB - SingleSided  
SL - Alumina (Er=9.8) 0.010 inch, gold  
SL - Alumina (Er=9.8) 0.025 inch, gold  
SL - FR4 (Er=4.4) 0.030 inch, 0.5 oz copper  
SL - FR4 (Er=4.4) 0.060 inch, 0.5 oz copper  
SL - RT\_duroid 5880 (Er=2.20) 0.010 inch, 0.5 oz copper  
SL - RT\_duroid 5880 (Er=2.20) 0.020 inch, 0.5 oz copper

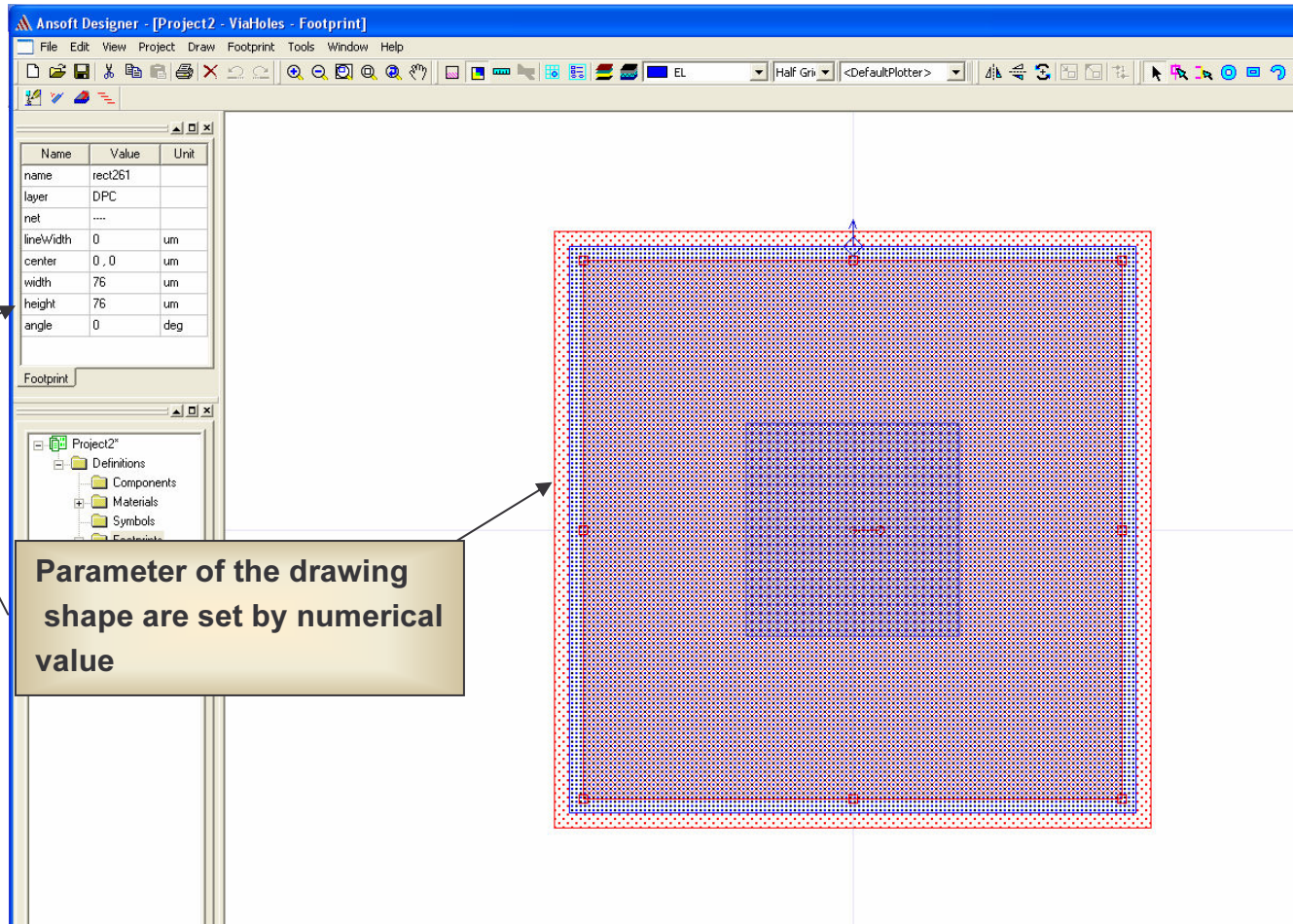
Open Browse... None Cancel

Project Components Search

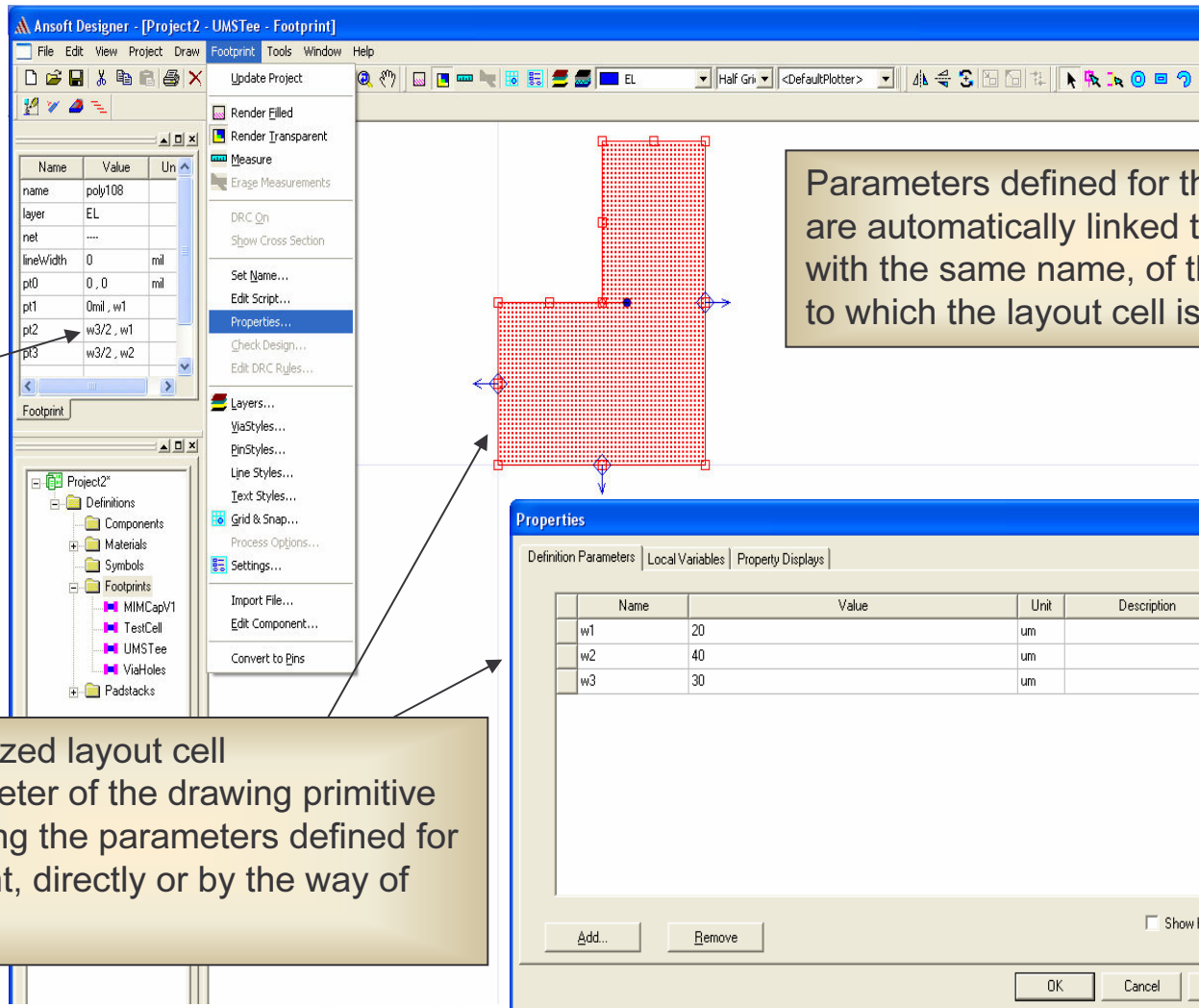
LPPProject (F:/Ansoft/AnsoftDesigner/Wew10102002Beta/project/)

X: 4.5000 Y: 0.8000 Delta X: 0.0000 Delta Y: 0.0000 Distance: 0.0000 mm Angle: 0.0

# Fixed Footprint



# Parameterized Layout Cell



Parameters defined for the layout cell are automatically linked to parameters, with the same name, of the component to which the layout cell is associated.

Parameterized layout cell  
The parameter of the drawing primitive are set using the parameters defined for the footprint, directly or by the way of equation.

# Scripted Layout Cell

The image displays three windows from the Ansoft Designer software:

- Ansoft Designer - [Project2 - UMSRectSpirale - Footprint]:** Shows a 2D layout of a rectangular spiral inductor footprint with a central bridge. The spiral is drawn with blue lines, and the bridge is highlighted in green.
- Edit Script:** A window for editing the footprint's script. It is set to use JavaScript. The script defines parameters for the footprint's dimensions and geometry, including a bridge and a spiral inductor.
- Properties:** A window showing the definition parameters for the footprint cell.

**Scripted parameterized layout cell.** the footprint is generated using a script which can be written in JavaScript or VBScript. Using a script allows to create complex parameterized layout cells like transistor, rectangular spiral inductor. Moreover a script allows to check minimum and maximum dimensions provided design rules checking at the component level. The edit script window allow the user to create, modify the script and see the effect of the modification on the layout cell just by clicking apply no need to compile or reload after modification.

Name	Value	Unit	Description
W	10		Width of line 5, 10, 15, 20 in um
L	4.6		Inductor value in pH
ANGout	0	deg	Output bridge angle
H	20		Width of input pad in um



# Exercise 2: Creating A Footprint

- ◆ Create a footprint
  - ◆ Set grid
  - ◆ Define parameters
  - ◆ Draw and parameterize shape
  - ◆ Add ports
  - ◆ Save to library
- ◆ User Exercise
  - ◆ Create new circuit and insert into previously created circuit
  - ◆ Edit symbol

# Create a simple Footprint

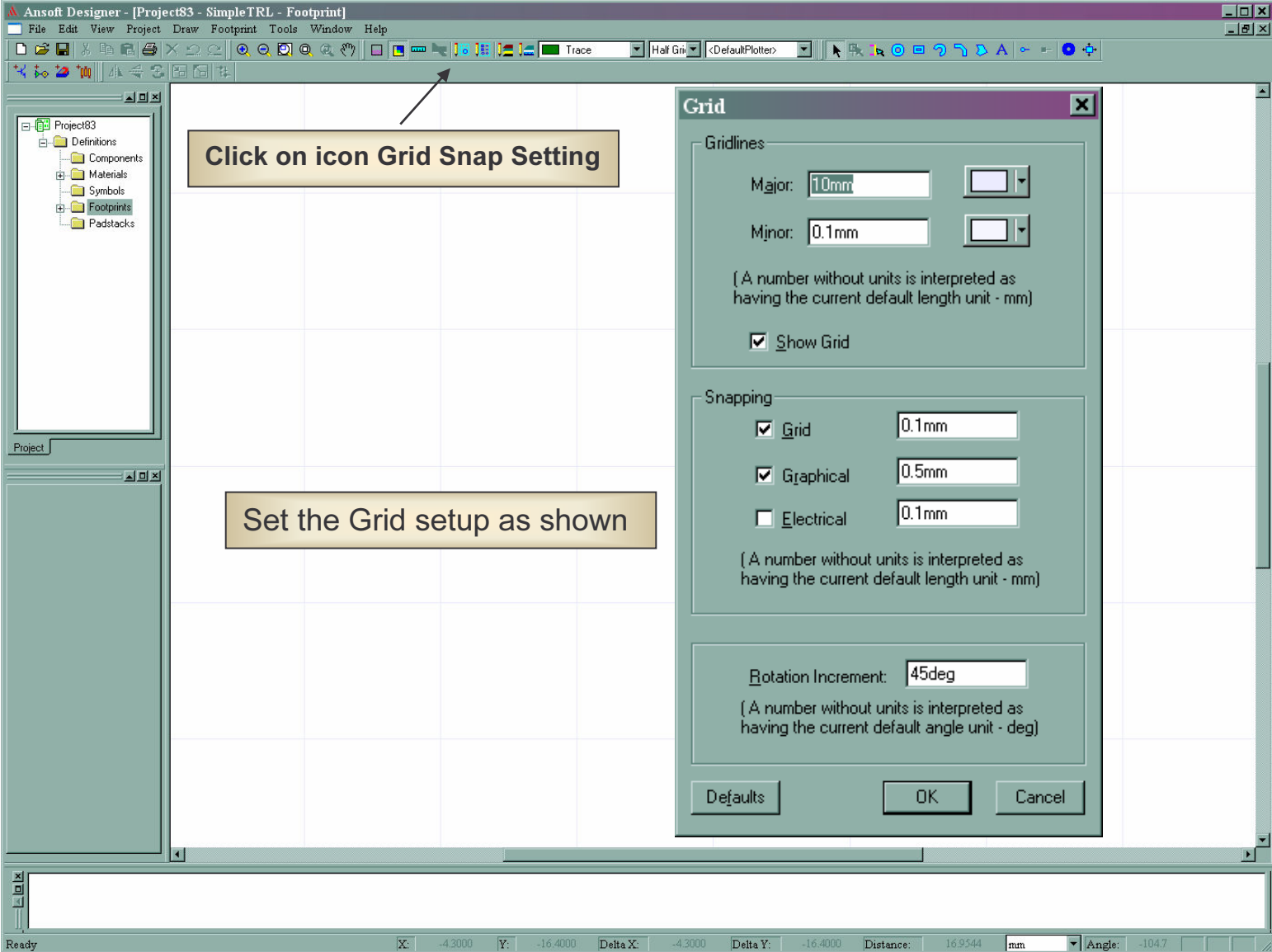
The screenshot shows the Ansoft Designer interface with several windows and callouts illustrating the steps to create a simple footprint:

- 1 Select Edit Definitions:** A callout points to the 'Edit Definitions...' button in the left-hand project tree.
- 2 Click Add Footprint:** A callout points to the 'Add Footprint...' button in the 'Edit Libraries' window.
- 3 Enter SimpleTRL and Click OK:** A callout points to the 'Get Name' dialog box where 'SimpleTRL' is entered.
- 4 Select the Technology As shown and Click open:** A callout points to the 'Choose Layout Technology' dialog box, where a technology is selected from the list.
- 5 Click Edit Footprint:** A callout points to the 'Edit Footprint' button in the 'Edit Libraries' window.

The 'Edit Libraries' window shows a table of footprint definitions:

Name	Location	Origin	Pin Count	Footprint Image
RectSimple	PersonalLibrary	Marconi 0		
SimpleTRL	Project		0	

# Set the Grid



Click on icon Grid Snap Setting

Set the Grid setup as shown

# Define the parameters

**Click properties** 1

**Enter : Name W  
Value 2mm  
Select Variable  
Click OK** 3

**Add Property**

Name   Variable  Checkbox  Netlist  Text  Number  Separator  
 Value  Menu  File Name  VPoint  Point

Value

Enter initial value into Value field. This should be a number, variable, or expression. Referenced project variables should be prefixed with a '\$'. Examples: 22.4pF, \$C1, 2\*cos(\$x).

OK Cancel

**Properties**

Parameter Defaults | Local Variables | Property Displays

Name	Value	Unit	Description
w	2	mm	
p	6	mm	

Add... Remove Show Hidden

OK Cancel

**Add another parameter  
P with Value 6mm** 4

**Select Definition Parameters  
Tab and click Add** 2

# Draw Shape

The screenshot shows the Ansoft Designer interface with the following components and annotations:

- Callout 1:** "Select Trace as the drawing layer" points to the "Trace" option in the layer selection dropdown menu.
- Callout 2:** "Click on Draw line" points to the "Draw Line" icon in the top toolbar.
- Callout 3:** "Click on Left" points to the left endpoint of a red hatched trace line.
- Callout 4:** "Drag the mouse" points to the middle of the red hatched trace line.
- Callout 5:** "Click on Left escape" points to the right endpoint of the red hatched trace line.

The Properties panel on the left shows the following details for the selected trace:

Name	Value	Unit
Name	line102	
Placement...	Trace	
Net	....	
LineWidth	2000	um
BendType	Mitered	
CapType	Extended	
TotalLength	52.9	mm
Pt0	-26.2, ...	mm
Pt1	24.7, ...	mm

The status bar at the bottom displays: Ready X: -0.7500 Y: -0.3000 Delta X: 0.0000 Delta Y: 0.0000 Distance: 0.0000 mm Angle: 0.0

# Parameterized the Shape

**Set BendType to Corner  
Set CapType to Flat 1**

**Use the Definition Parameters  
To set the properties.  
lineWidth = w  
Pt0 = 0,0  
Pt1 = p,0 2**

**Changing the value  
Of w and p will 3  
Change the shape.**

Name	Value	Unit
Name	line102	
Placement...	Trace	
Net	----	
LineWidth	w	
BendType	Corner	
CapType	Flat	
TotalLength	6	mm
Pt0	0,0	mm
Pt1	p,0	

Name	Value	Unit	Description
Name	line102		
PlacementLayer	Trace		
Net	----		
LineWidth	w		
BendType	Corner		
CapType	Flat		
TotalLength	6	mm	
Pt0	0,0	mm	
Pt1	p,0		

# Add Connection Ports

The image displays two side-by-side screenshots of the Ansoft Designer software interface, illustrating the steps to add connection ports to a footprint. Both screenshots show a project named 'Project83' with a footprint selected. A green hatched rectangular footprint is centered on a grid. A red double-headed arrow indicates the port's location and angle.

**Left Screenshot (Step 1 & 2):** A port is being added to the left edge of the footprint. The 'Project' table shows the following properties:

Name	Value	Unit
Type	Pin	
Clearance	0	mm
Net	---	
Location	0,0	mm
Angle	180	deg
Padstack	NoPad SMT East	
LayerMap...	Show...	
Edit layer ...	Edit...	
Name	1	

**Right Screenshot (Step 3 & 4):** A port is being added to the right edge of the footprint. The 'Project' table shows the following properties:

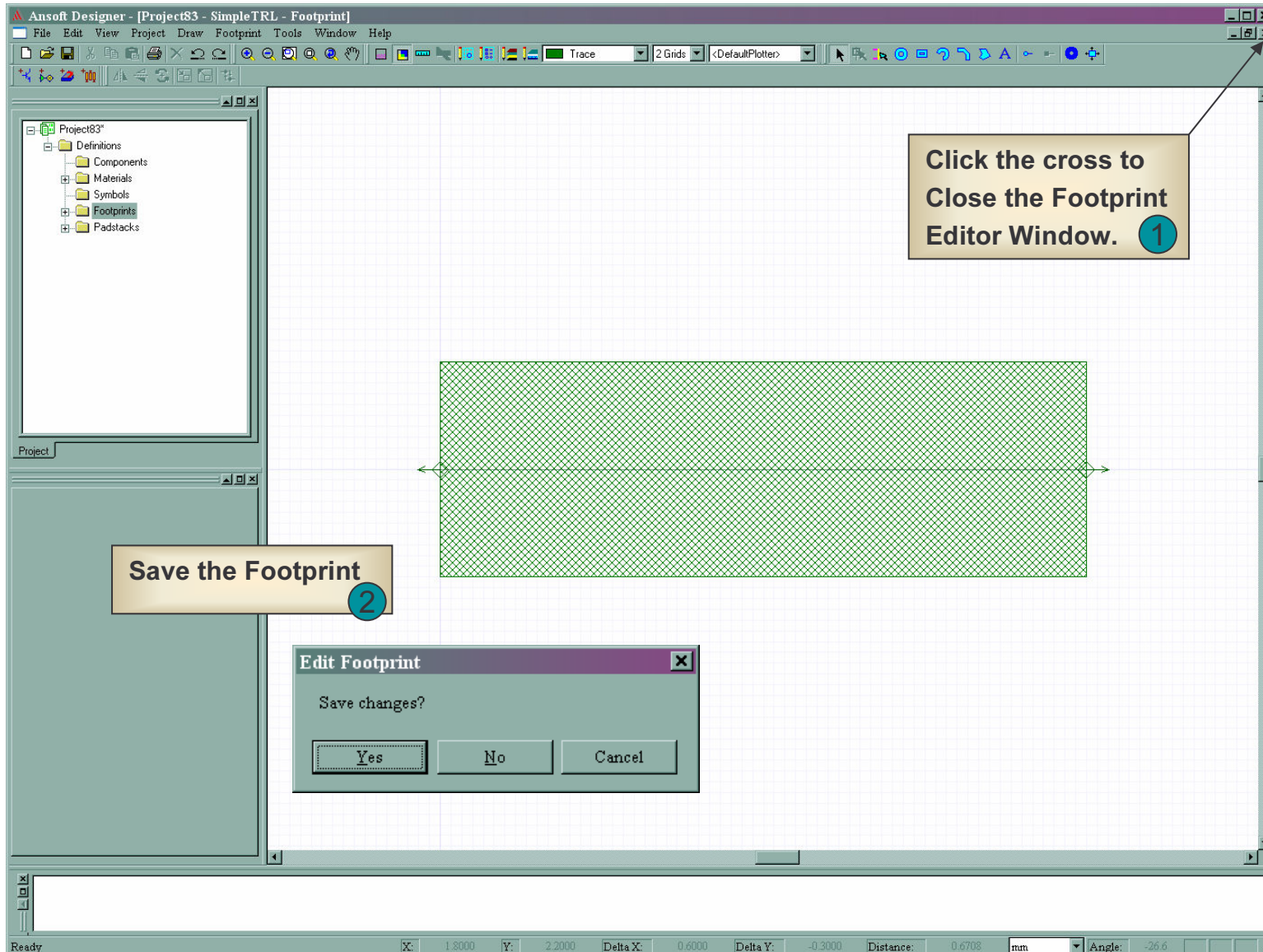
Name	Value	Unit
Type	Pin	
Clearance	0	mm
Net	---	
Location	p,0	deg
Angle	0	deg
Padstack	NoPad SMT East	
LayerMap...	Show...	
Edit layer ...	Edit...	
Name	2	

**Footprint Draw Toolbar:** A central toolbar labeled 'Footprint Draw' contains various drawing tools including a mouse cursor, selection tools, and drawing aids.

**Instructional Callouts:**

- 1** Insert connection Ports To the left edge of the footprint
- 2** Set the Port Angle to 180
- 3** Insert connection Ports To the right edge of the footprint
- 4** Set the location To p,0

# Save The Layout Cell to Library





# Export The Layout Cell to Library

**Select Edit Definition 1**

**Select SimpleTRL 2**

**Click Export Footprint Library 2**

**Enter the name of the file and Click Save**

**You can select PersonalLib or Userlib as the location where to Save the file**

Name	Location	Origin	Pin Count
RectSimple	PersonalLibrary	Marconi 0	
SimpleTRL	Project		2

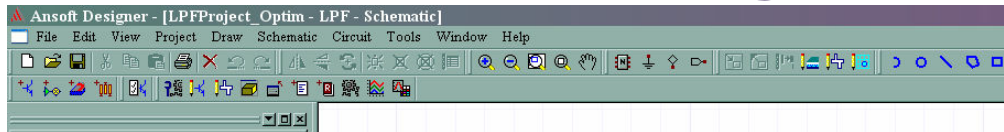
Save in: PersonalLib

File name: SimpleLib.afib

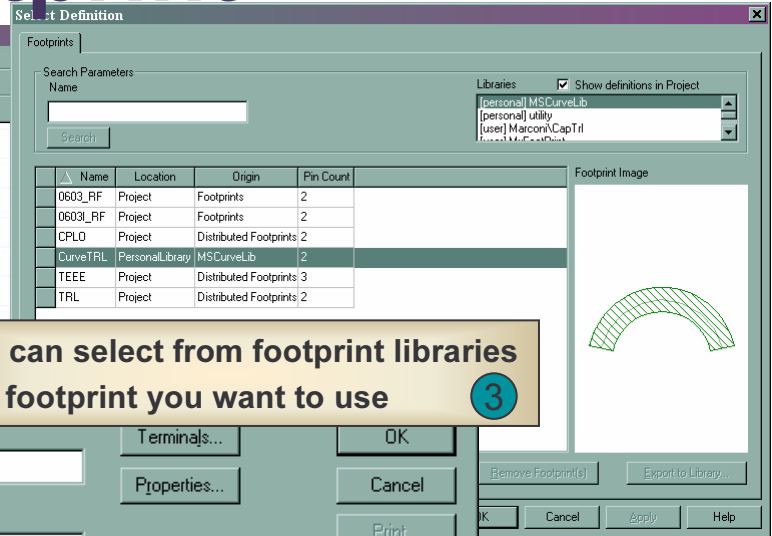
Save as type: Ansoft Footprint Libraries (\*.afib)

PersonalLib UserLib

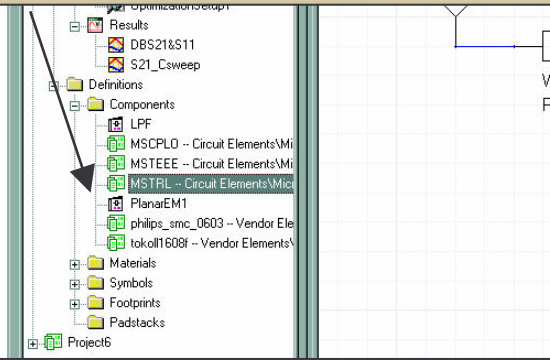
# Using Footprint



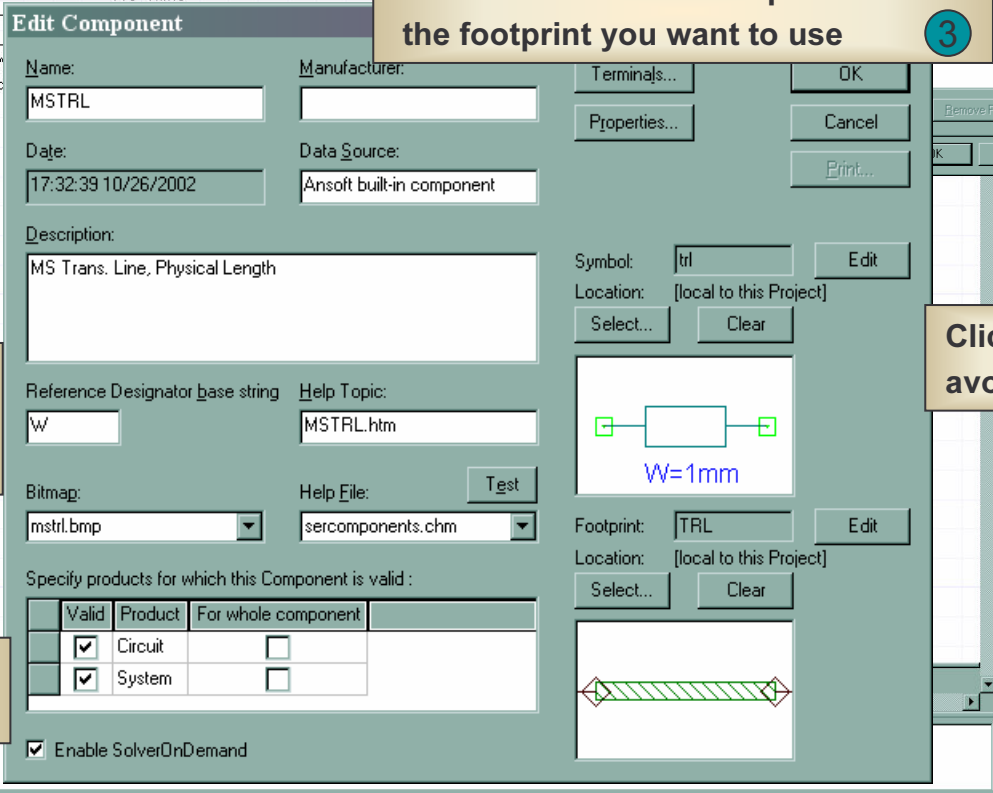
From Project window expand folder Definitions/Components. Double click on MSTRL to open The edit Component window **1**



You can select from footprint libraries the footprint you want to use **3**



The Edit Component window allows to select Symbol and Footprint to associate to the component **2**



Click Cancel to avoid modification

Click select above footprint view To select a new footprint.



# User Exercise

- ◆ Using Subcircuits and Symbols
  - ◆ Inserting Subcircuits
  - ◆ Changing Symbols



# Insert Another Circuit Design

Open LPFProject previously created.  
Click right on LPFProject and select Insert Circuit Design.  
Select MS-FR4(4.4) .060in as the Technology File.

1

Build the schematic as shown above.

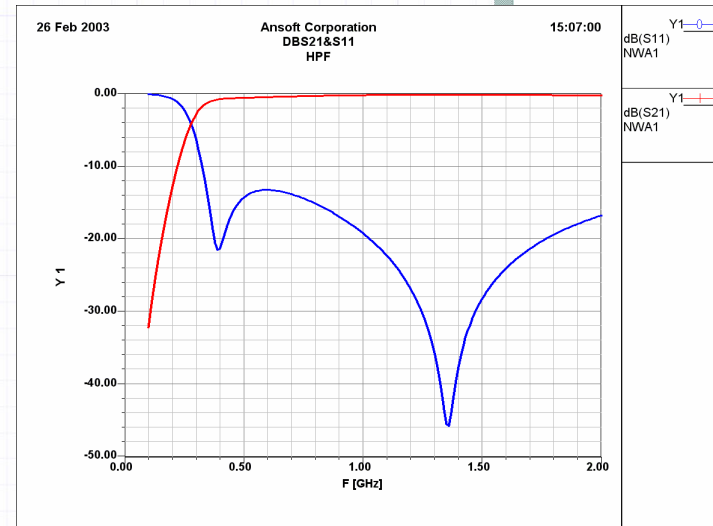
Tee  $w1=w2=w3=wline$   
Trl  $w=wline, p=lline$   
Cap Philips\_smc\_0603, 6.8nh, Tol 0.25pf  
Ind Tokoll1608f, 18nh, Tol 10%

3

Add Analysis Setup Linear Sweep from  
0.1GHz to 5GHz by step of  
0.01GHz.

4

Analyze the circuit.



Create a report with S21 and S11 in DB,  
rename it DBS21&S11.  
Change the Xscale to 0.1GHz-2GHz.  
Save the project

5

Rename the inserted circuit HPF.  
Define Definition Parameters as shown:  
 $Wline=0.8mm, Lline=1mm$

2

Properties

Parameter Defaults | Local Variables | General

Value  Optimization  Tuning  Sensitivity  Statistics

Name	Value	Unit	Description	Read-only	Hidden
ModelName	X			<input type="checkbox"/>	<input checked="" type="checkbox"/>
wline	0.8	mm		<input type="checkbox"/>	<input type="checkbox"/>
lline	1	mm		<input type="checkbox"/>	<input type="checkbox"/>

Buttons: Add... Remove Show Hidden OK Cancel

# Edit The Symbol and Modify It

You can draw arc, circle, line, polygon, rectangle and add text.

Click the cross to close the Symbol editor widow

Port1

Port2

Desktop

Save changes?

Yes No

Modify the symbol as shown using arc. Close and save the symbol. Save the project.

Each time a Planar EM, Circuit or System Design is created, a symbol, with the same name as the design, is created. To edit this symbol expand the Definitions folders, then the Symbol Folder and double click on the symbol, here HPF.

# Copy and Paste the Circuit Design

**1** Click right on HPF  
And select Copy

**2** Click right on LPF  
And select Paste

**3** You can choice to use the same stackup as Parent circuit or to insert The sub-circuit as black box

**4** When Incorporate is selected  
The Merge layers window is displayed allowing to merge sub-circuit and parent circuit layers

**5** Symbol of the pasted sub-circuit appears in the schematic

**Synchronize Design**

- Incorporate**  
Design is copied, same stackup as parent, manufacturable with parent  
New design: HPF1
- Keep independent (black box)**  
Design is linked to original, unrelated stackups, not manufacturable with parent

**Merge Layers**

<input checked="" type="checkbox"/> Trace	Trace
<input checked="" type="checkbox"/> Dielectric	Dielectric
<input checked="" type="checkbox"/> Ground	Ground

Param Values: wine: 0.8 mm, lline: 1 mm, Status: Active

Click to place component instance

# Connect the Sub-Circuit Symbol

The screenshot displays the Ansoft Designer interface for a project named "LPFProject\_Complete\_Sub - LPF - Schematic". The Design tree on the left shows a hierarchy where the "LPF" folder is expanded, revealing a sub-circuit symbol labeled "U2: HPF1". A callout box points to this symbol with the text: "Sub-circuit folder appears in the circuit Design tree. If needed you can simulate it stand-alone from this location."

The main schematic area shows a circuit diagram. It features two ports, "Port1" and "Port2", connected to a central sub-circuit symbol. The sub-circuit symbol is a square with three pins labeled "1", "2", and "3". The circuit includes various components such as inductors (e.g., "LL1608\_F8N2K 8.2nH"), capacitors (e.g., "C0603338C98B200 3.3pF"), and transmission lines (e.g., "W=wline", "P=liline"). The sub-circuit symbol is also connected to a circular component at the bottom, which has parameters "W=wline", "D=0.3mm", and "DG=0.3mm".

A callout box at the bottom left points to the "Param Values" table in the software interface, with the text: "You can change the values of the sub-circuit parameters".

Name	Value	Unit
wline	0.8	mm
liline	1	mm
Status	Active	

**End of Exercise**

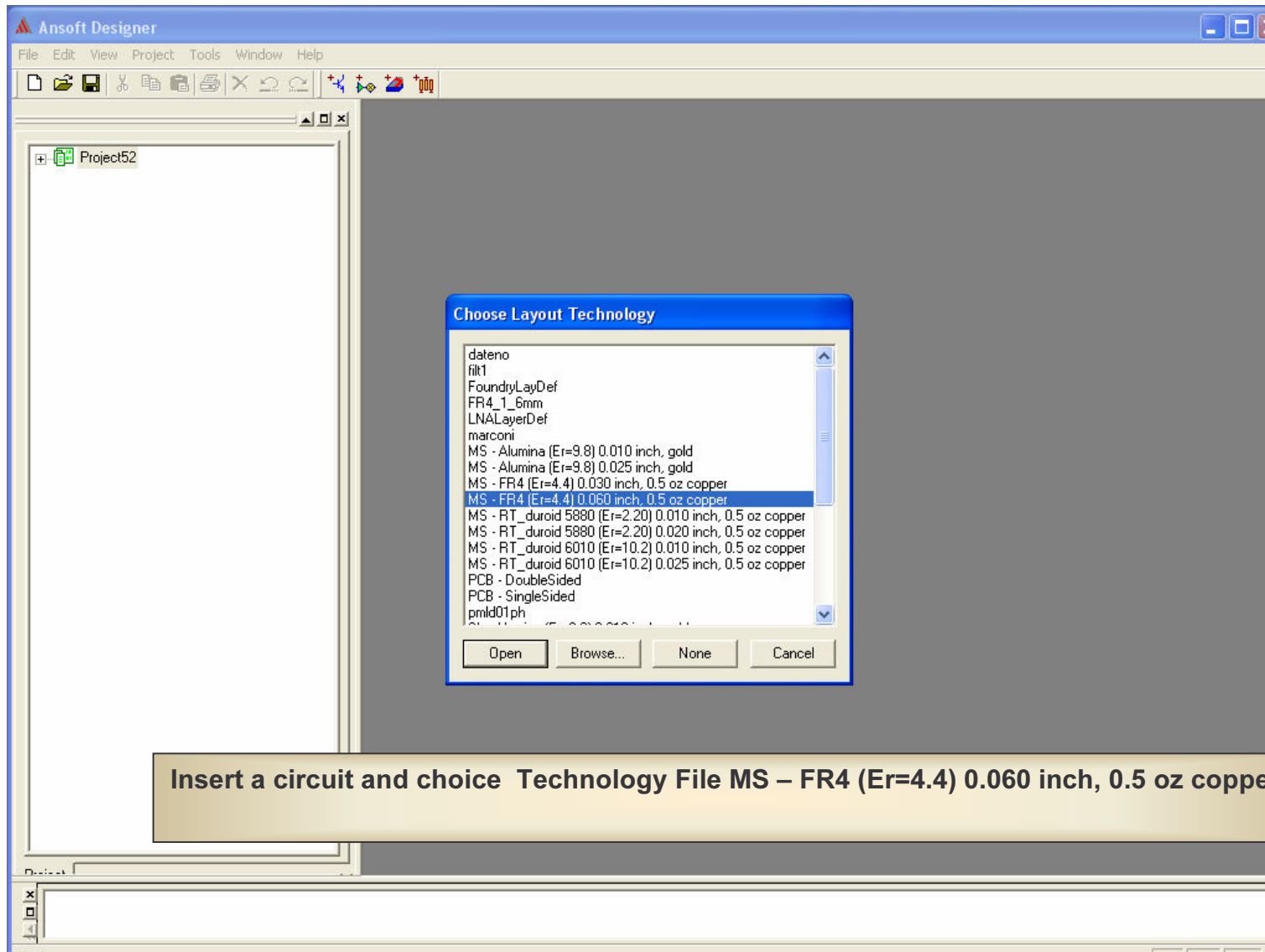




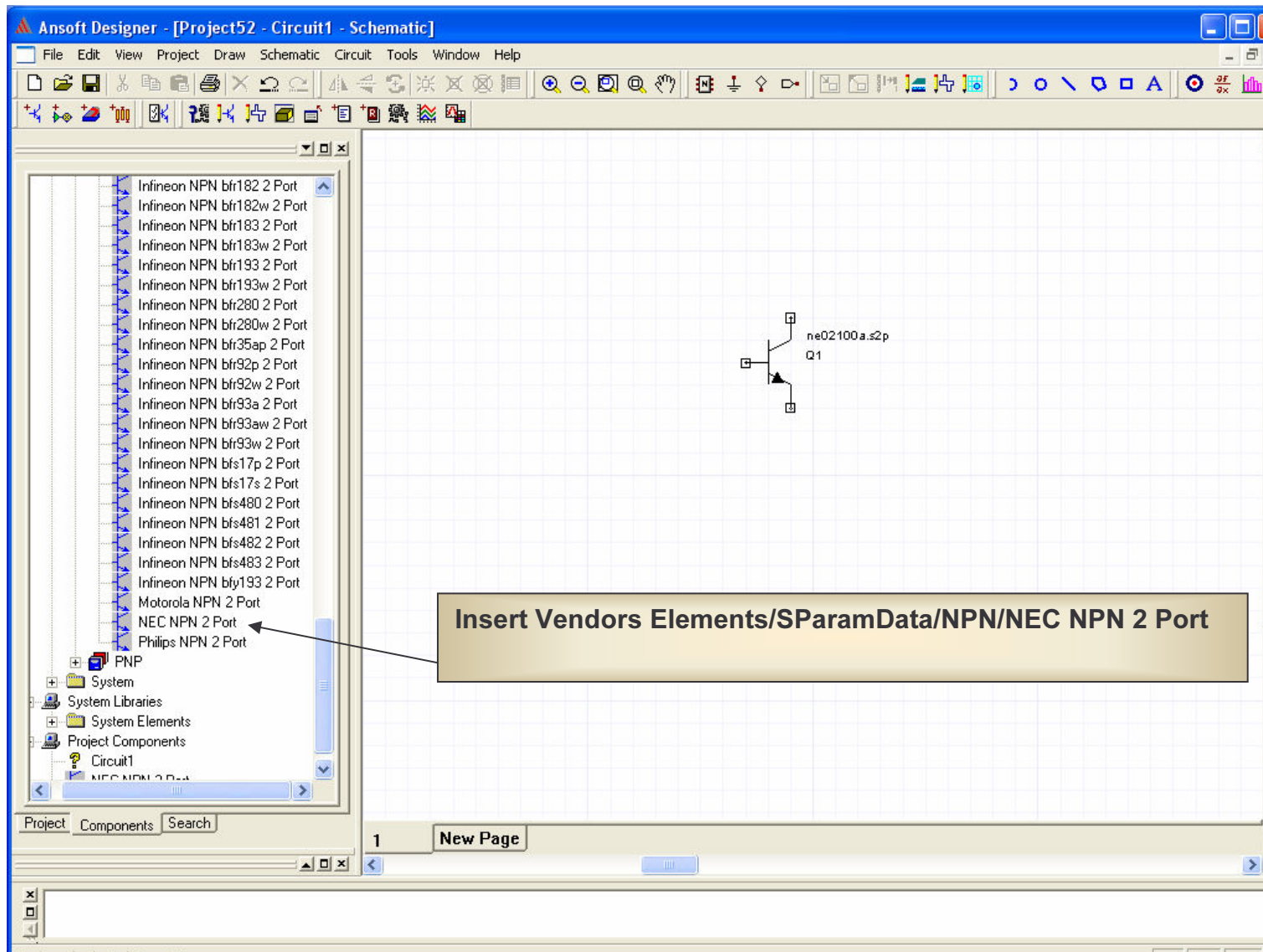
# Smith Tool .avi



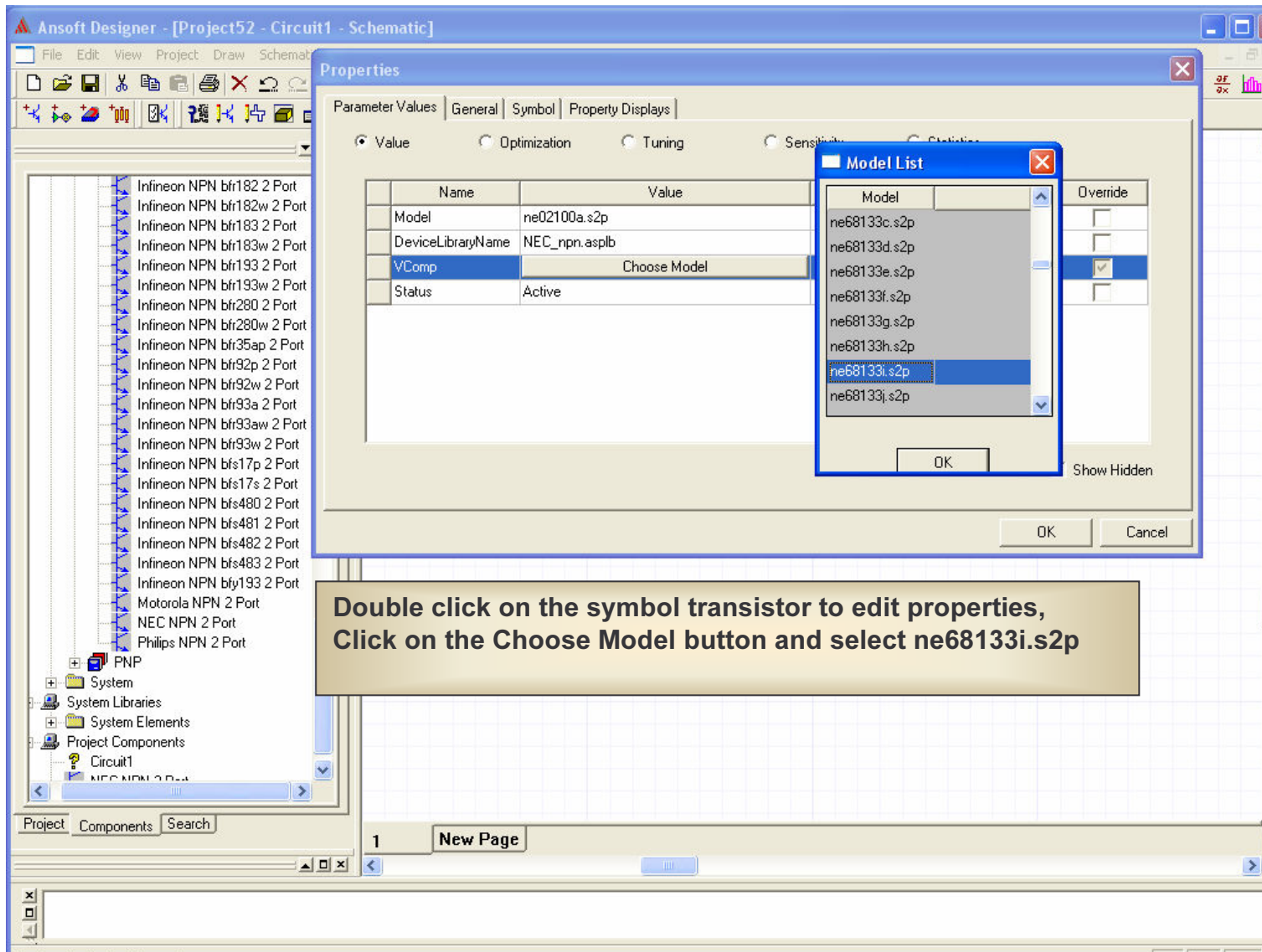
# Create a New Circuit



# Insert NPN NEC Transistor



# Choice the Transistor



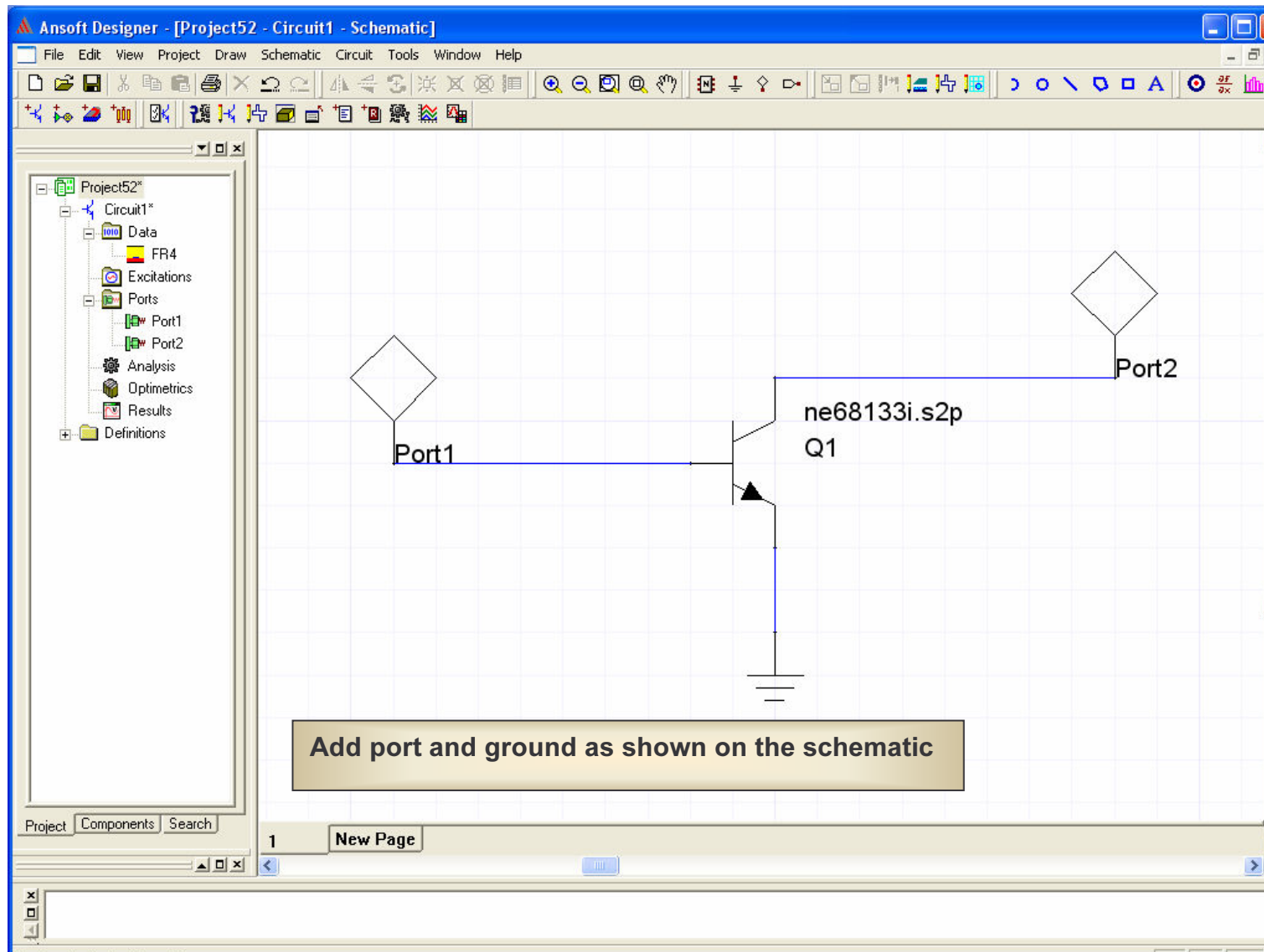
The screenshot shows the Ansoft Designer interface with a schematic project. A Properties dialog box is open for a transistor component. The 'VComp' property is set to 'Choose Model'. A 'Model List' dialog box is also open, showing a list of transistor models. The model 'ne68133i.s2p' is selected in the list.

Name	Value
Model	ne02100a.s2p
DeviceLibraryName	NEC_npn.asplb
VComp	Choose Model
Status	Active

Model	Override
ne68133c.s2p	<input type="checkbox"/>
ne68133d.s2p	<input type="checkbox"/>
ne68133e.s2p	<input checked="" type="checkbox"/>
ne68133f.s2p	<input type="checkbox"/>
ne68133g.s2p	<input type="checkbox"/>
ne68133h.s2p	<input type="checkbox"/>
ne68133i.s2p	<input checked="" type="checkbox"/>
ne68133j.s2p	<input type="checkbox"/>

**Double click on the symbol transistor to edit properties,  
Click on the Choose Model button and select ne68133i.s2p**

# Add Port and Ground



# Add Analysis setup

Linear Network Analysis, Frequency Domain

Analysis Setup Option: [Default Options]

Group Delay

Enable Group Delay Calculations

Perturbation (%) 0.1

Sweep Variables

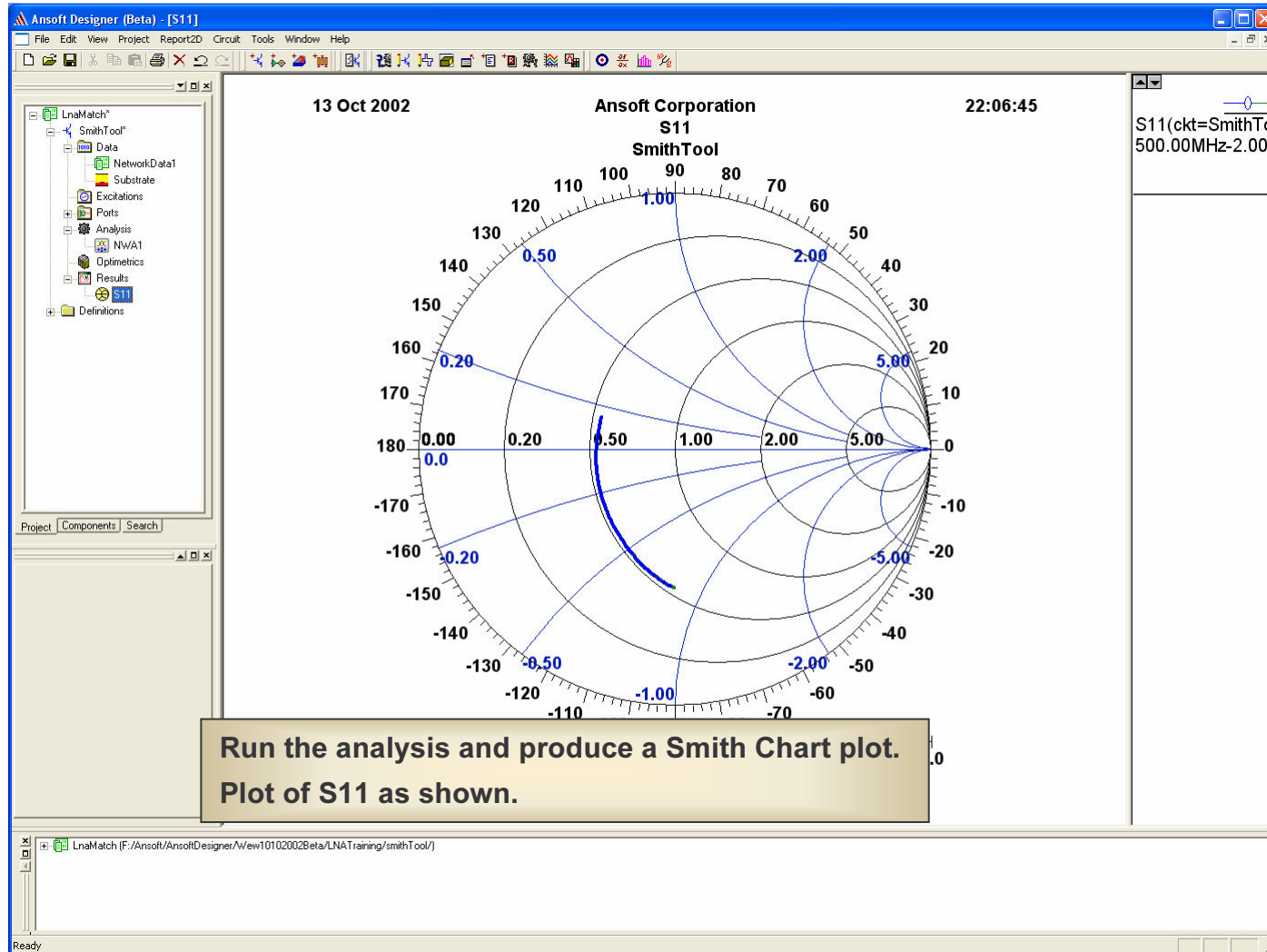
Name	Sweep/Value	Sync
F	LIN 0.5GHz 2GHz 0.01GHz	

Add... Remove Edit... Sync

< Back Finish Cancel Help

**Add Linear Network Analysis Setup.  
Sweep the frequency to 0.5GHz to 2GHz with a step of 0.01GHz**

# SmithTool - Polar Plot



# SmithTool - Opening the Utility

The screenshot shows the Ansoft Designer (Beta) interface with the Smith Tool utility open. The main window displays a Smith chart for a circuit named 'S11(ckt=Sm)' at a frequency of 500.00MHz. The chart shows a point at approximately 0.708 on the real axis and -92 on the imaginary axis. The VSWR is 5.84Z. The chart is overlaid with a grid and various scales.

Parameters displayed in the Smith Tool window:

- Display: Matching
- Grids:  Impedance,  Admittance,  Polar
- Step:  Start,  Stop,  Incr
- Circles:  Power Gain Gp (S-plane)
- Mapping:  Available Gain Ga (S->L)
- Gain (dB):  Mismatch:  Apply
- Marker Point: Conjugate, Z->Y, Y->Z
- ZRef: 50.00+j0.00, G: 17.95dB
- FMIN: 0.86dB, K: 0.56
- Circuit: SmithTool, Freq: 0.5GHz

MP: 0.708 -92

VSWR: 5.84Z

Click on Report2D menu and select Smith Tool



# SmithTool - Opening the Utility

13 Oct 2002 Ansoft Corporation 22:16:15 S11(ckt=Sm 500.00MHz-

The utility contains several areas:

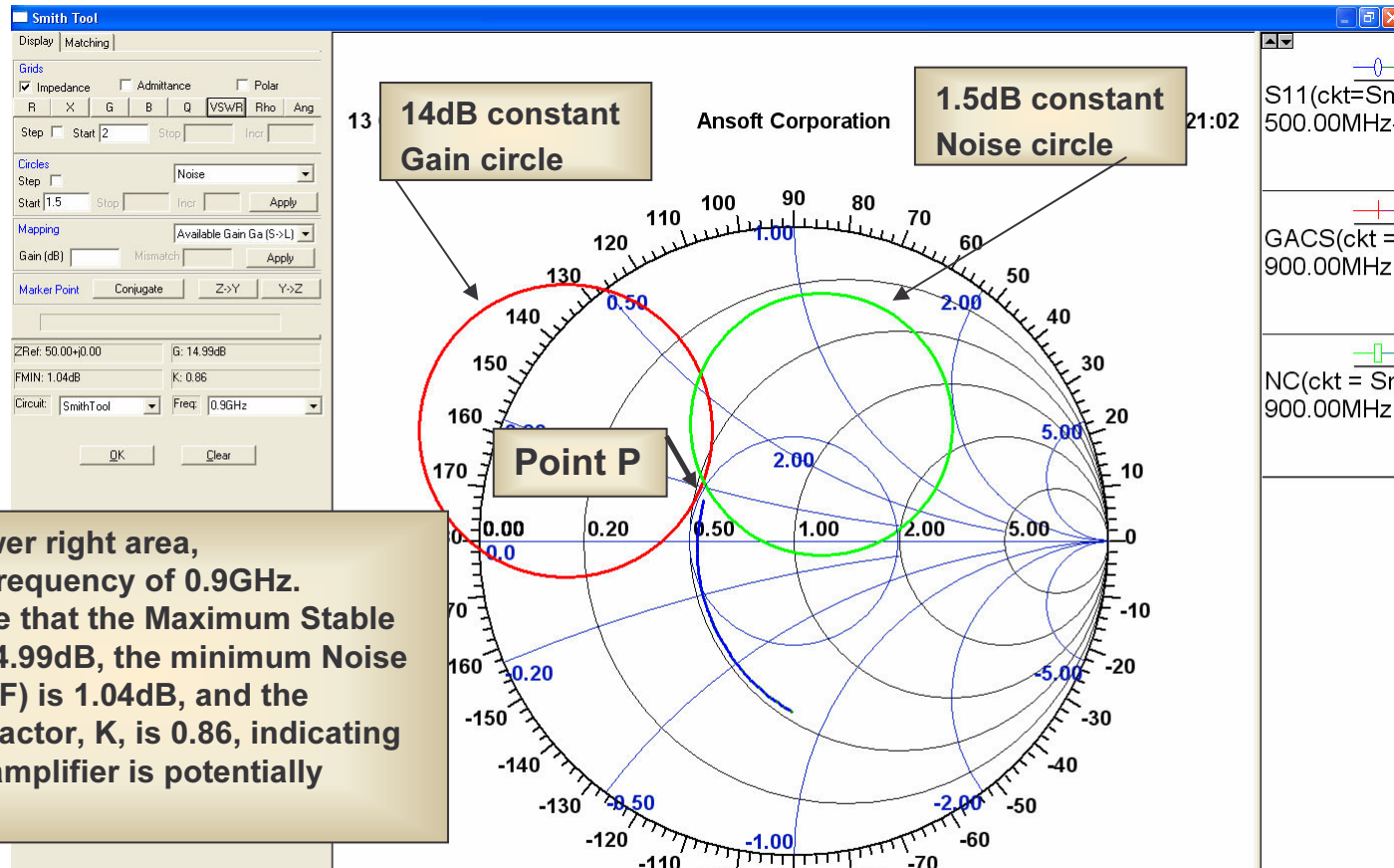
- The *Grids* area lets you draw constant R, X, G, B, Q, VSWR, and Rho circles on the plot.
- The *Circles* area lets you draw Gain, Noise, and Stability circles.
- The *Mapping* area lets you transform the responses from the source plane to the load plane and vice-versa.

At the top of the dialog, there are tabs to switch between this *Display* portion of the dialog and the *Matching* portion. We will exercise both areas as this example proceeds.

At the bottom of the dialog is information that is calculated from the device S Parameters, like Maximum Stable Gain, minimum NF, and stability factor, K.

MP: 0.708 -92  
RX: 0.318 - j0.4  
GB: 0.348 + j0  
Q: 2.832  
VSWR: 5.842

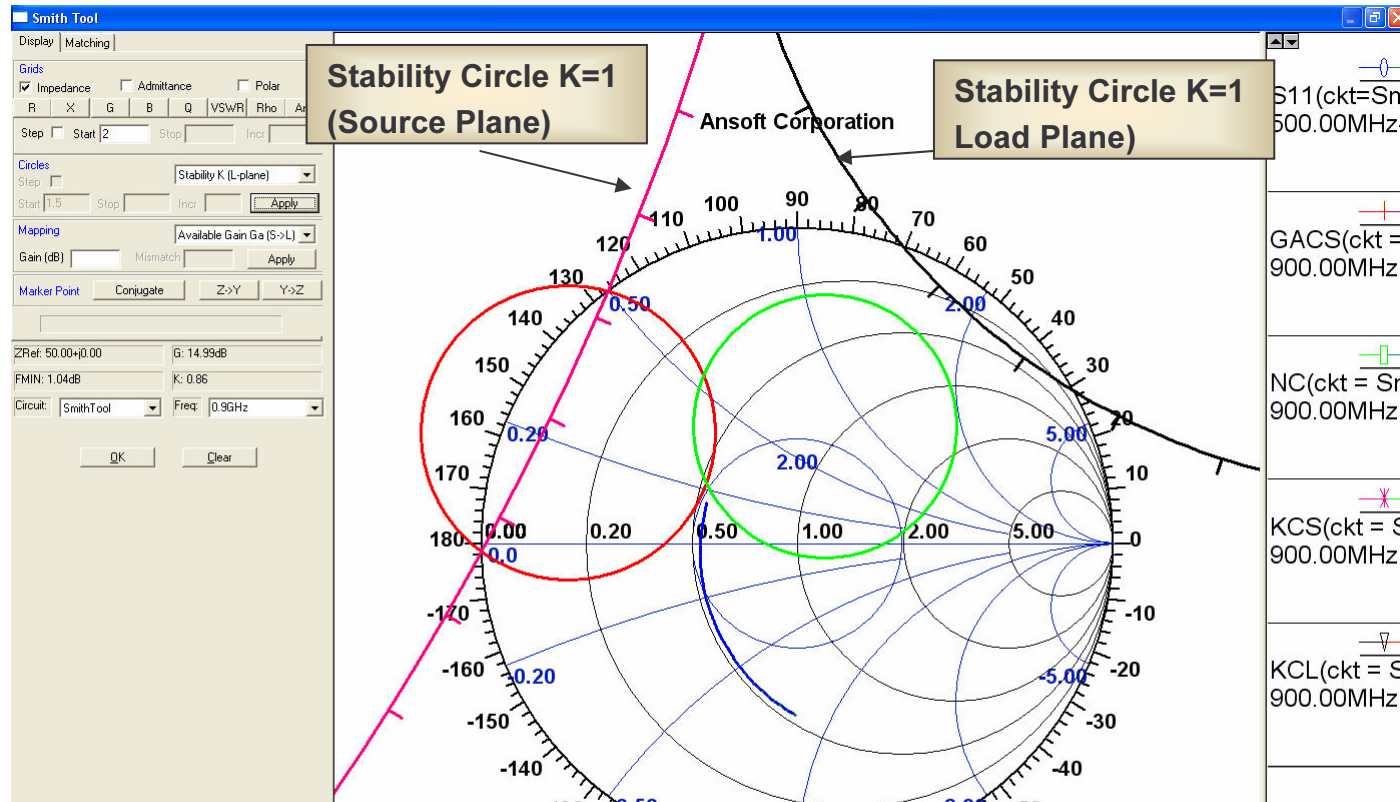
# SmithTool - Maximizing Performance



In the lower right area, select a frequency of 0.9GHz. Now, note that the Maximum Stable Gain is 14.99dB, the minimum Noise Figure (NF) is 1.04dB, and the stability factor, K, is 0.86, indicating that our amplifier is potentially unstable.

In the *Circles* area, select *Avail. Gain Ga (S-Plane)*. Enter 14 in the *Start* box and click on *Apply*. An 14dB gain circle appears. Now, select *Noise* and enter 1.5dB. Click *Apply* and a 1.5dB noise circle appears. In the *Start* box of section *Grids* enter 2 and click on *VSWR*.

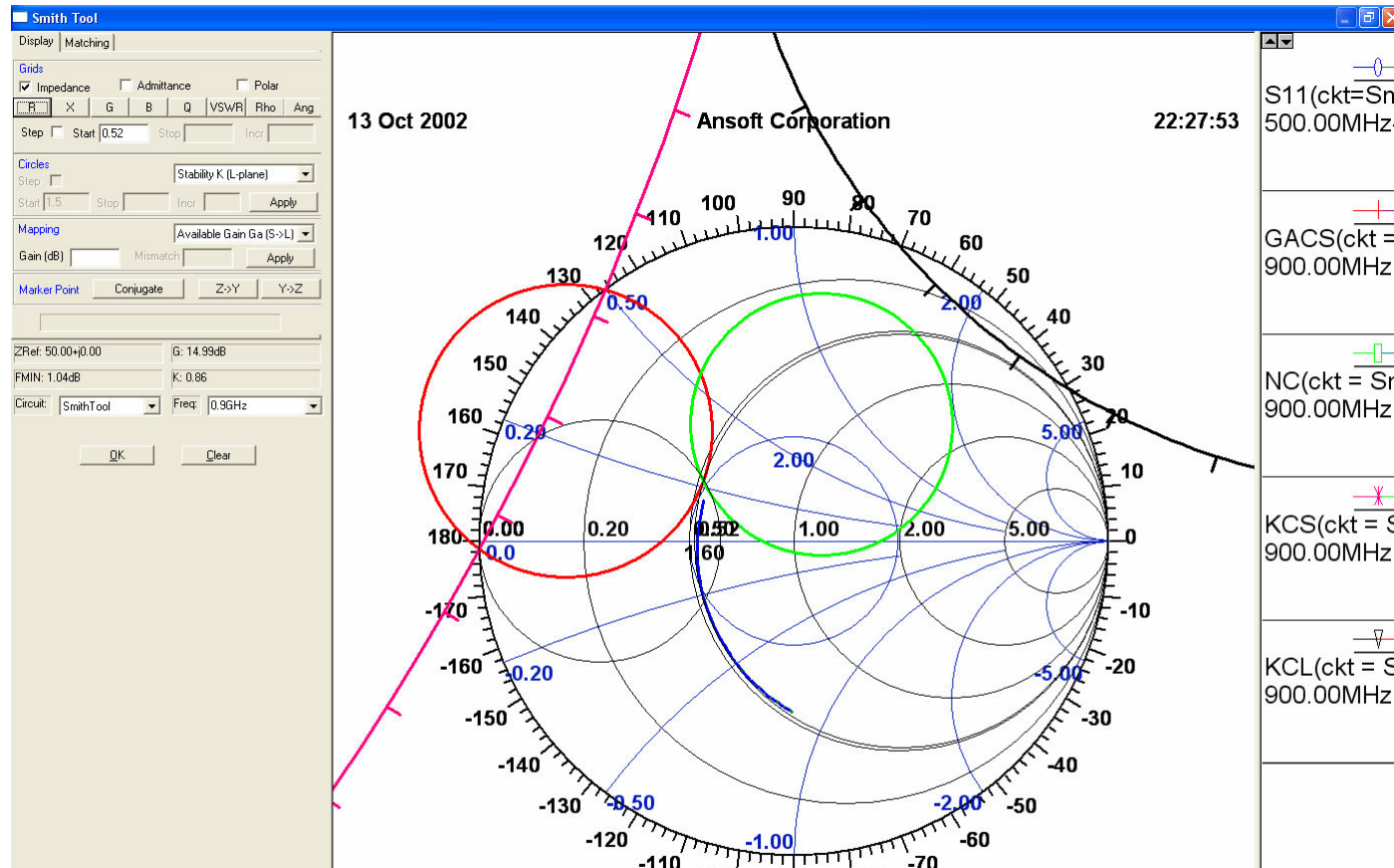
# SmithTool - Checking Stability



In the *Circles* area, select *Stability K (S Plane)* in the drop down list and click *Apply*. The source plane circle for  $K=1$  appears, with small lines, or spokes, indicating which side of the circle is stable.

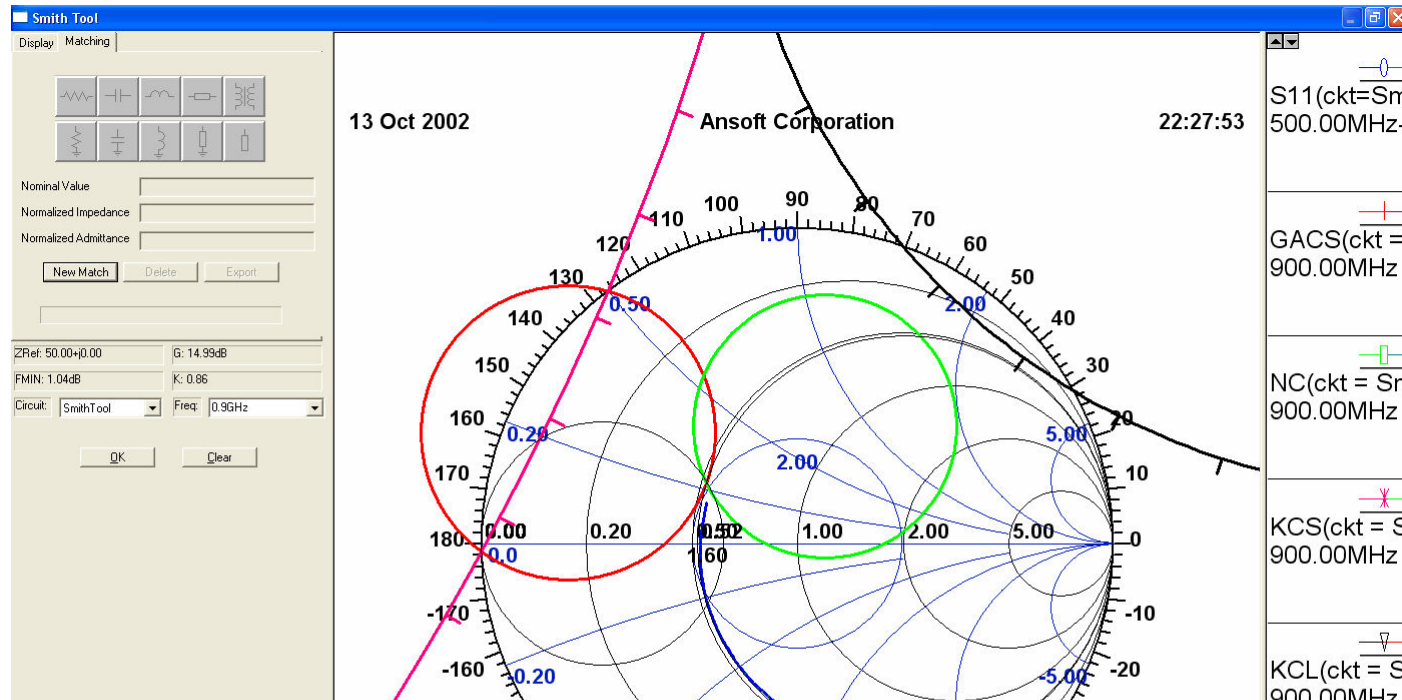
In the *Circles* area, select *Stability K (L Plane)* in the drop down list and click *Apply*. The load plane circle for  $K=1$  appears, with small lines, or spokes, indicating which side of the circle is stable.

# SmithTool - Drawing Aids



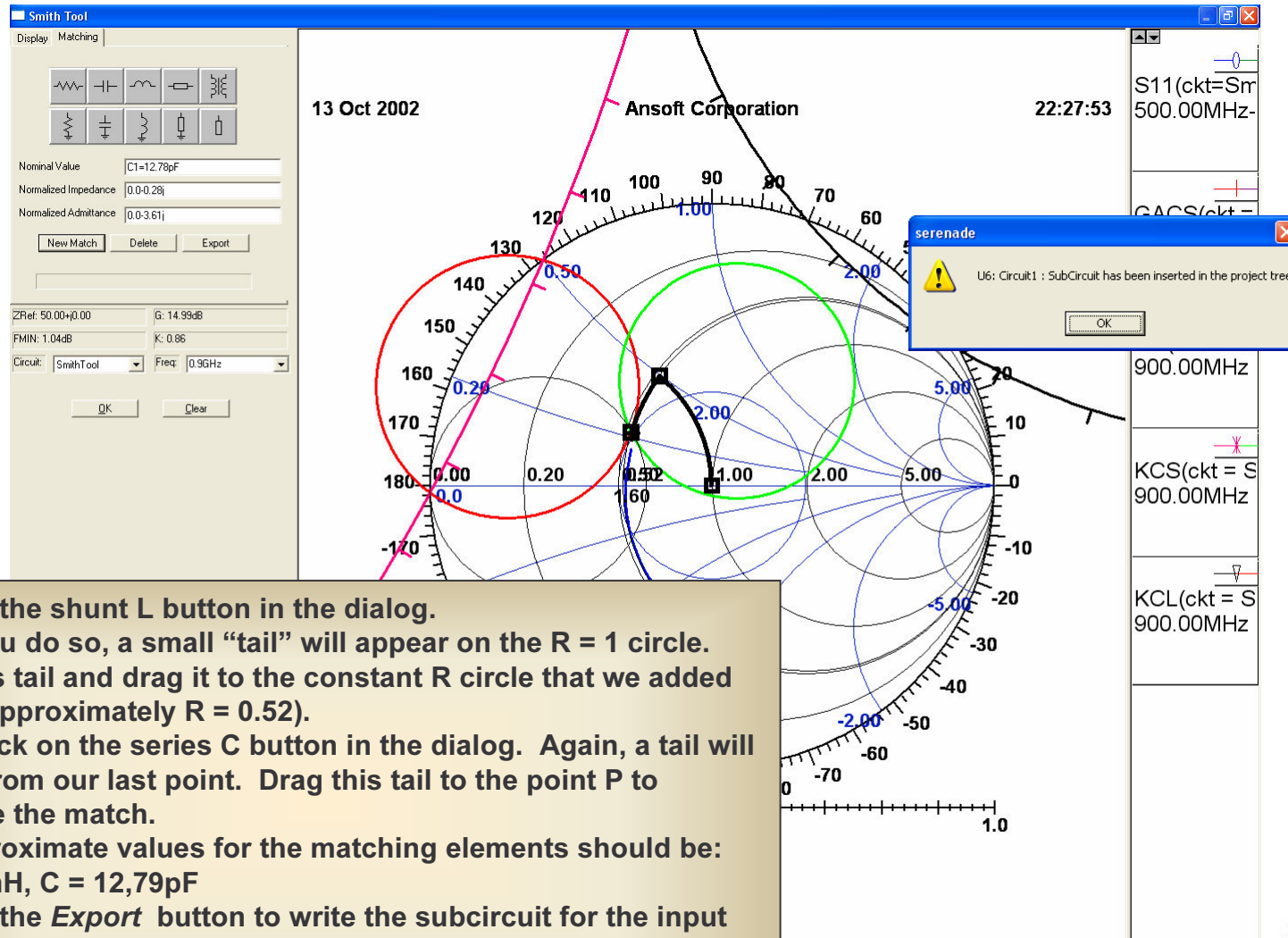
In the *Grids* area, click on *G*. The cursor jumps to the plot. Click on point P to draw a constant *G* circle through it. This circle should be approximately  $G = 1.60$ .  
In the *Grids* area, click on *R*. The cursor jumps to the plot. Click on point P to draw a constant *R* circle through it. This circle should be approximately  $G = 0.52$ .

# SmithTool - Matching Tab



Click on the *Matching* tab on the SmithTool dialog. The dialog changes as shown above. For the input matching circuit, we will move on the Smith chart from 50 Ohms at the center of the chart to our point P. Click on the *New Match* button. When you do this, the cursor will immediately jump to the center of the Smith chart. Without moving the mouse, click again to place the "crosshair" at 50 Ohms. After you do this, the ten element buttons in the dialog (shown above in gray) will activate. These are the available elements for use in the matching circuit, representing both lumped and distributed components.

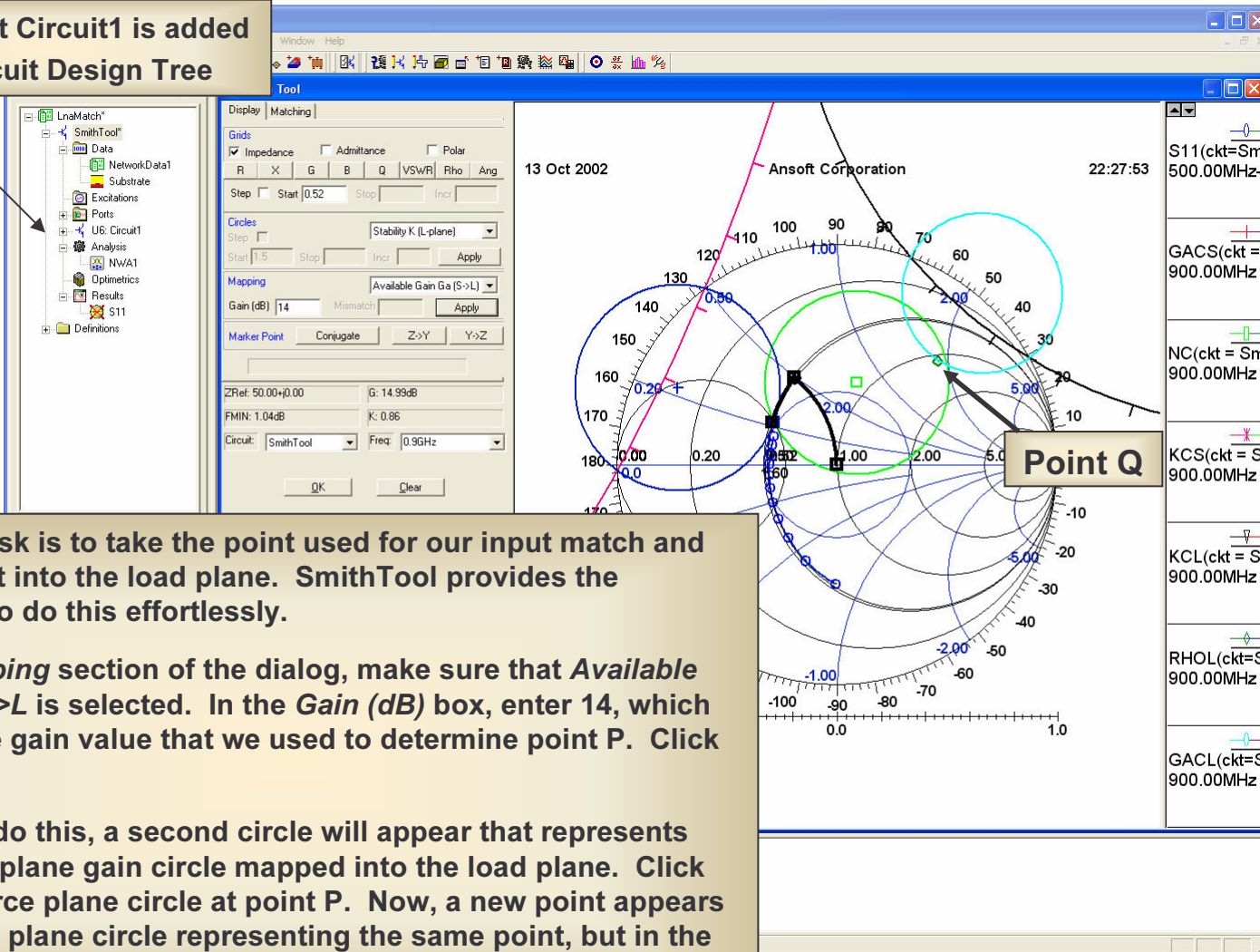
# SmithTool - Input Matching Circuit



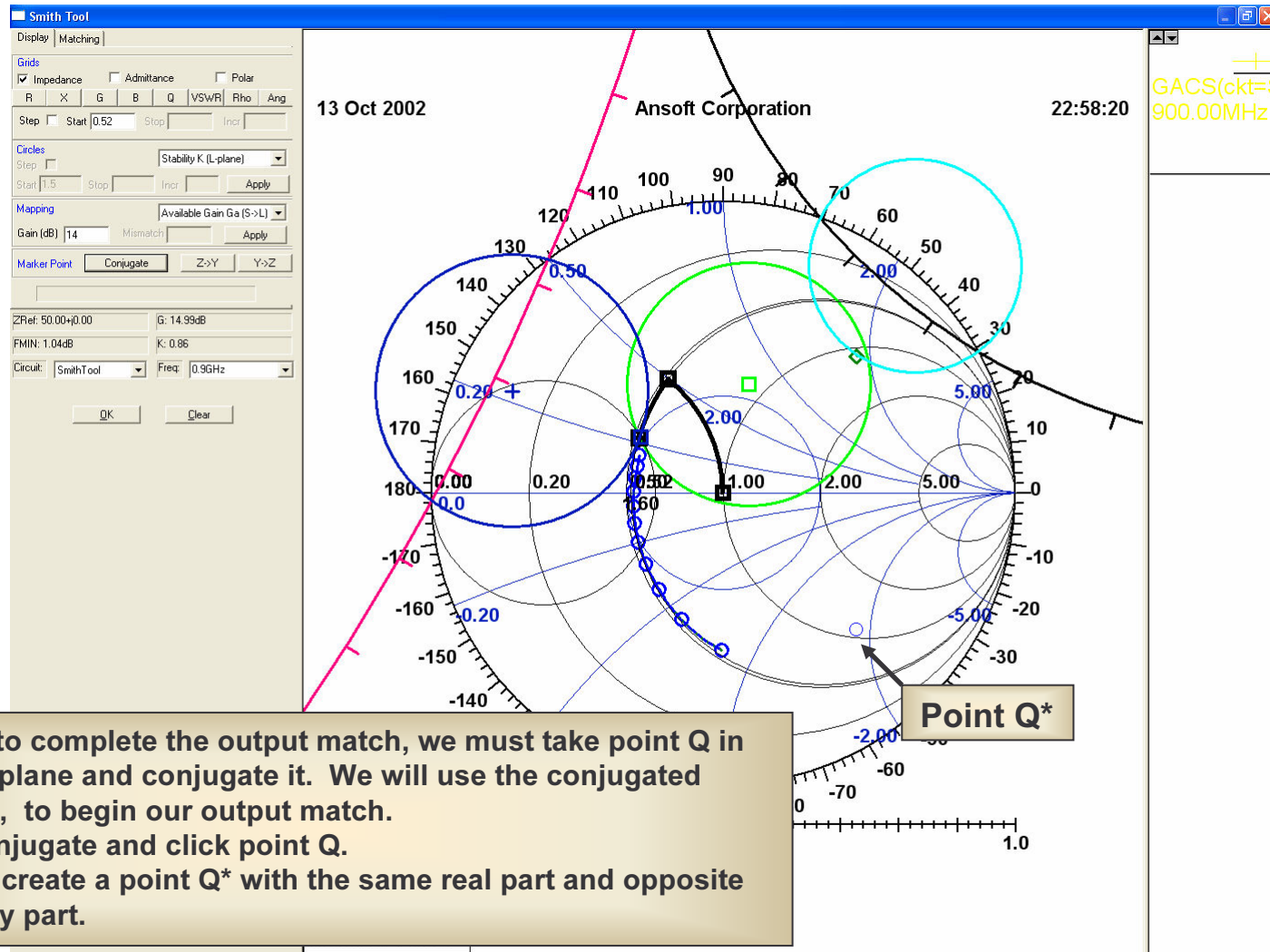
Click on the shunt L button in the dialog.  
When you do so, a small “tail” will appear on the  $R = 1$  circle.  
Grab this tail and drag it to the constant R circle that we added earlier (approximately  $R = 0.52$ ).  
Then, click on the series C button in the dialog. Again, a tail will appear from our last point. Drag this tail to the point P to complete the match.  
The approximate values for the matching elements should be:  
 $L = 9.19\text{nH}$ ,  $C = 12,79\text{pF}$   
Click on the *Export* button to write the subcircuit for the input match.  
Click OK

# SmithTool - Source/Load Mapping

Sub-Circuit Circuit1 is added  
To the Circuit Design Tree

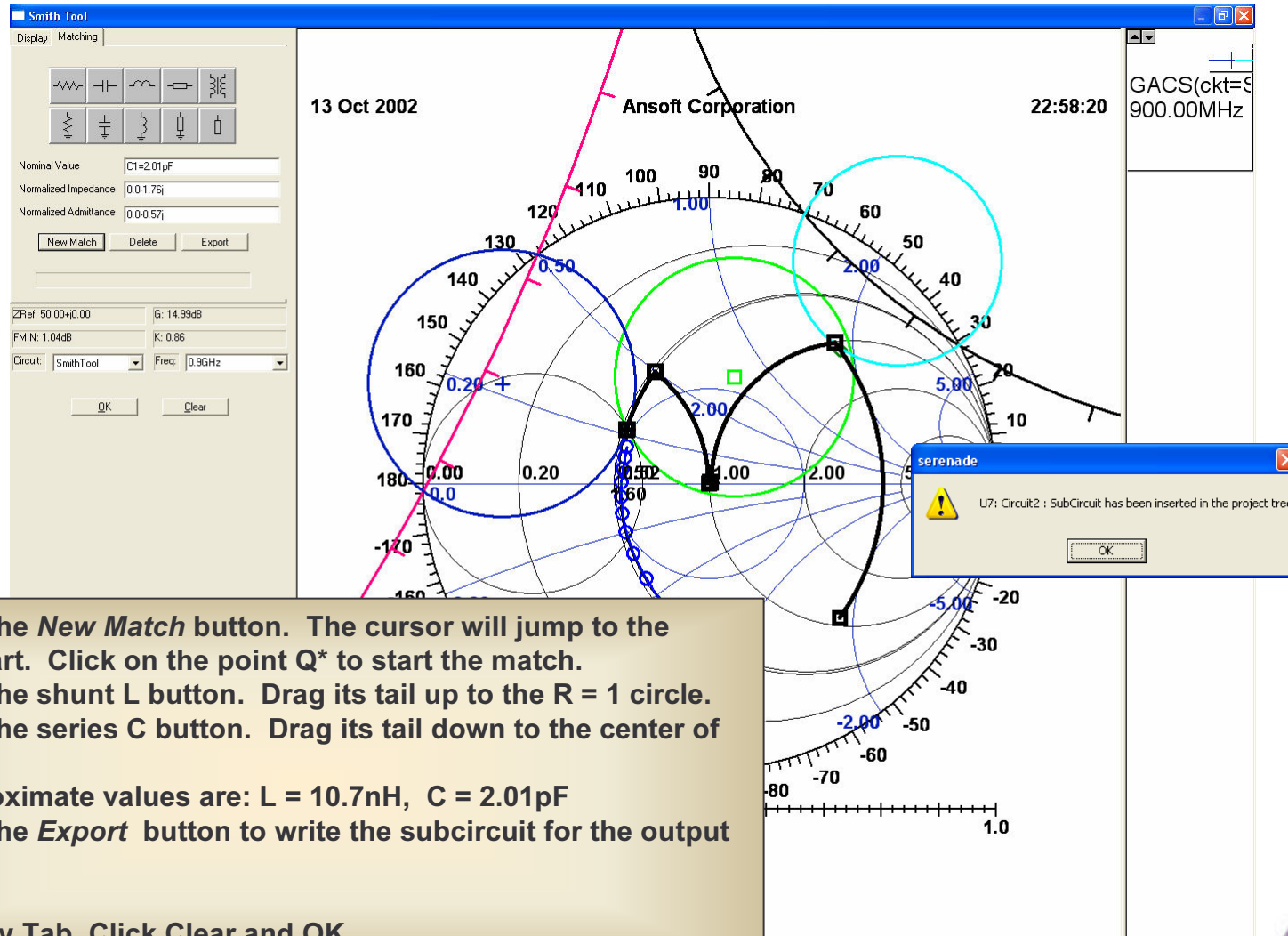


# SmithTool - Complex Conjugation



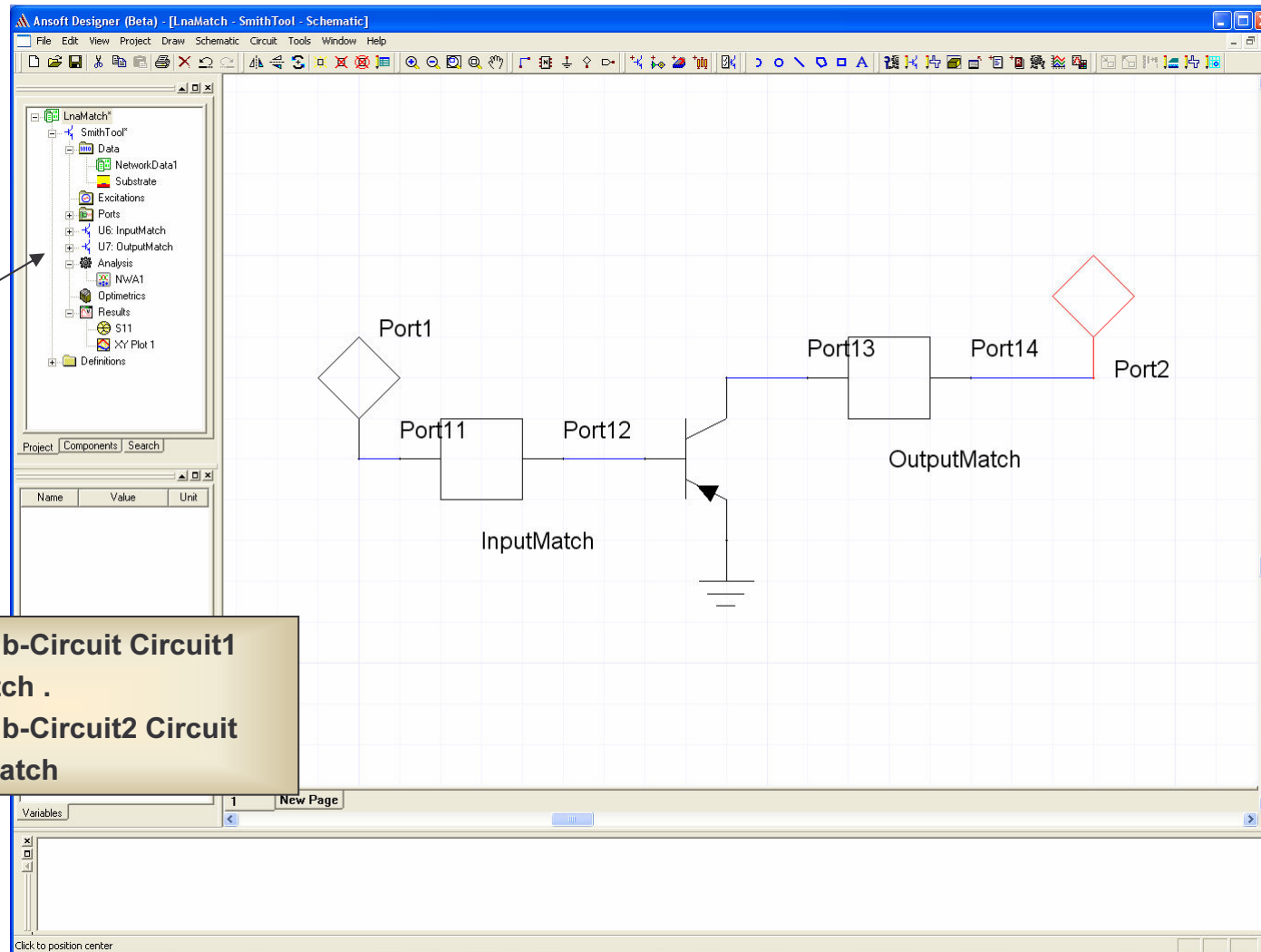


# SmithTool - Output Matching Circuit



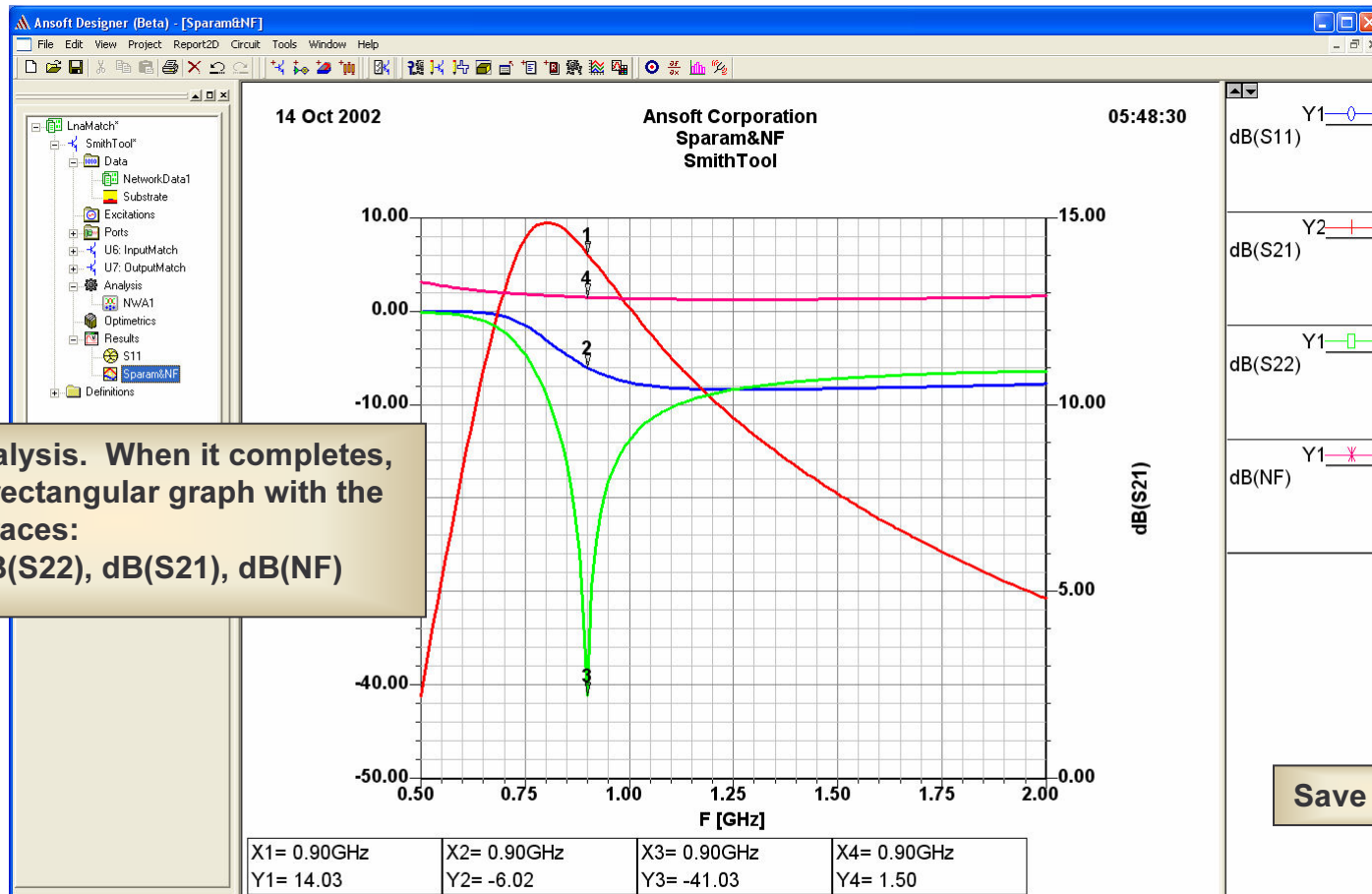
Click on the **New Match** button. The cursor will jump to the Smith chart. Click on the point Q\* to start the match.  
Click on the shunt L button. Drag its tail up to the R = 1 circle.  
Click on the series C button. Drag its tail down to the center of the chart.  
The approximate values are: L = 10.7nH, C = 2.01pF  
Click on the **Export** button to write the subcircuit for the output match  
Click OK  
Hit Display Tab, Click Clear and OK

# Building the Amplifier



**Rename Sub-Circuit Circuit1 to InputMatch .**  
**Rename Sub-Circuit2 Circuit to OutputMatch**

# Verifying Amplifier Performance



Run the analysis. When it completes, produce a rectangular graph with the following traces:  
dB(S11), dB(S22), dB(S21), dB(NF)

Save the Project

Right click on the graph and select *Data Marker*. Add a marker on each trace at 0.9GHz. As you can see, our design goals of 14dB gain and 1.5dB NF have been met. Note that the input is -6dB. Also, note that the output is resonated, providing an excellent match at the output. Save the current project.

# Non-Linear Analysis.avi



# Exercise: LNA Design Non-Linear Analysis

# Load LnaNLStart

Resistor were added for bias

Capacitor and Inductor use Normalized value

Non linear model replace S parameters file

Name	Value	Unit
Status	Active	

You can either load the circuit or created from the previous project.  
If you want to create the circuit: Open the project LNAMatch and save it as LnaNLR1TONE  
To use the existing circuit click File Open and select LnaNLStart.  
This circuit use a non linear model for the transistor (Vendor lib/nonlinear/npn/nec/ne68133)  
instead of the S parameters data file.  
Values of Capacitors and Inductors were replace by normalized values, resistors are added for bias.

# Insert DC Source

**Insert Voltage Source**

2.5V

0528

12nH

2.2pF

NE68133 Q1

100uF

12pF

10nH

0015

Name	Value	Unit
V	2.5	V
primitive	vdc	
EditSource	Edit	
Name	VDC1	
Status	Active	
Info	VDC	

Click on Components Tab.  
Expand Source Folder then Independent Sources folder and insert Voltage source as shown above.  
Set the source to 2.5V.

# View DC Bias

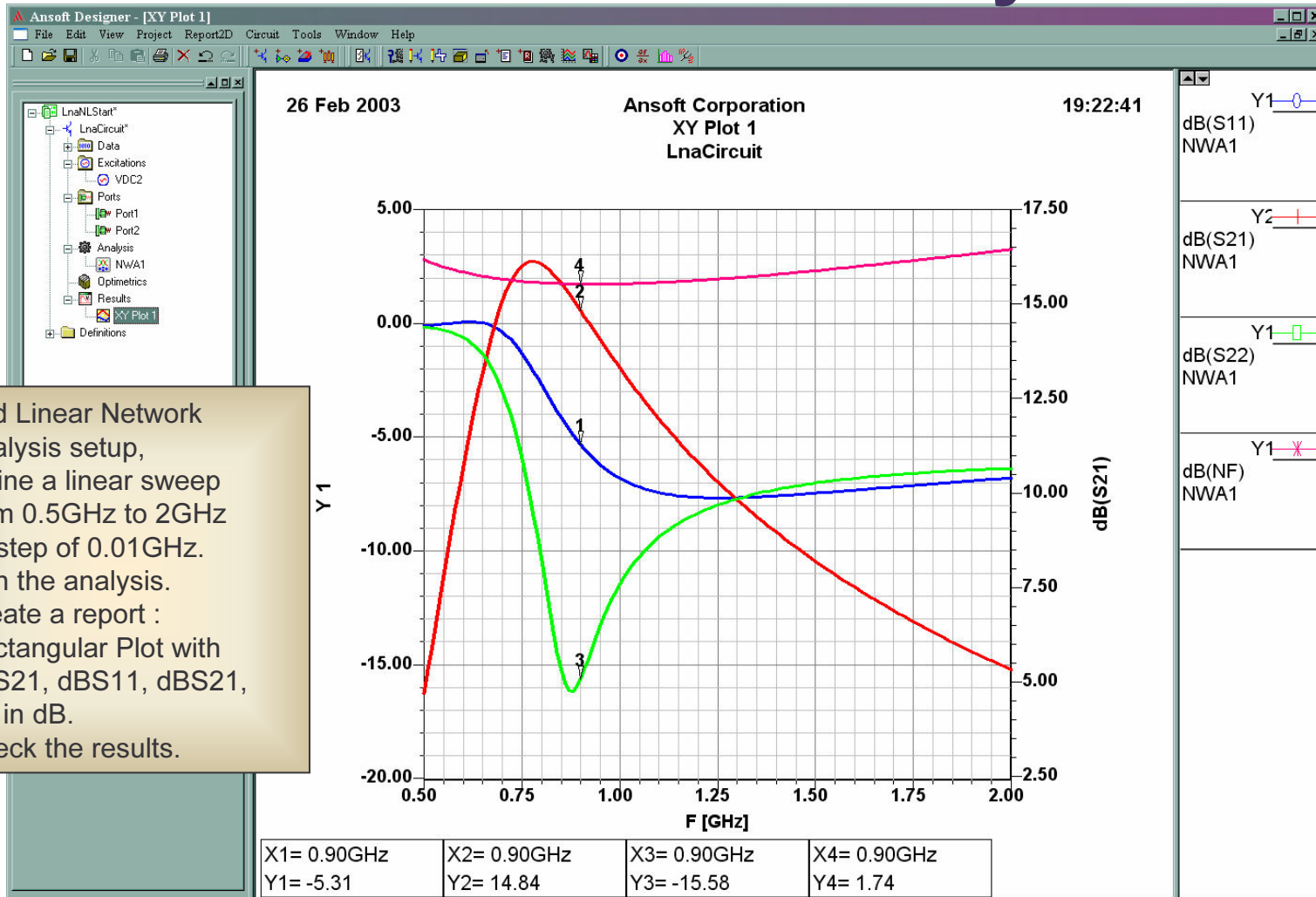
Click Right on LNA circuit and select View DC Bias.  
This will display on the schematic the DC current and Voltage.  
We can check the bias of the transistor 2.5V 3mA which was the one of the S parameter use for matching.

Save the Project

Toggle Display DC Bias



# Run Linear Analysis



Add Linear Network Analysis setup, define a linear sweep from 0.5GHz to 2GHz by step of 0.01GHz. Run the analysis. Create a report : Rectangular Plot with dB(S21, dB(S11), dB(S21), NF in dB. Check the results.

Even if we use a non linear model it is possible to run linear analysis, non linear model is linearised at the bias condition and the simulation will use the corresponding S parameters. This allow to check that the results with the non linear model are close to the one get with S parameter data file.

# Define RF 1 Tone Analysis

**1** Double click on the input port

**2** Click Add in the Sources section

**3** Select Power. Enter Pin for value of P

**4** Define Pin as local variable with value of -10dbm

Port Definition

Port name: Port1  
Port number: 1  
Symbol:  Interconnect  Microwave Port

Termination  
 Simple termination: Re: 50 Im: 0 Impedance  
 One-port data: Edit... Create New...

Source Definition  
Source type: Power  
Sources:  
Add... Edit... Delete

Load Pull Tuner and Reference Node  
Load Pull Tuner: <none> Edit... Create New...  
Reference Node: Ground

Source Selection

Voltage  Power  
Name: Sinusoidal1  
Type: Sinusoidal

Property	Value	Unit	Description
hnum	f1		Harmonic number of the source
vdc	0	mV	DC offset of the source
p	Pin		Available power of the source
phs	0	deg	Phase of the source
td	0	ns	Time delay

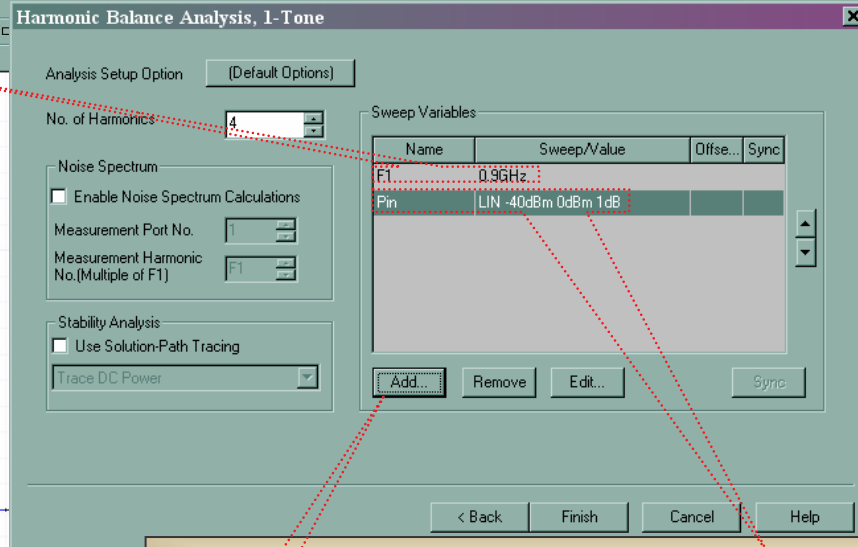
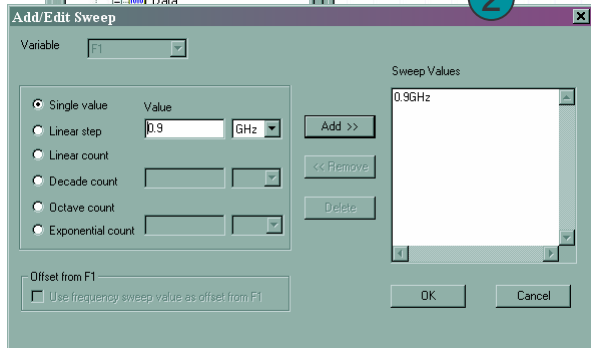
Add Variable to LnaCircuit

Name: Pin  
Value: -10dbm  
 Parameter Default  Local Variable  Project Variable

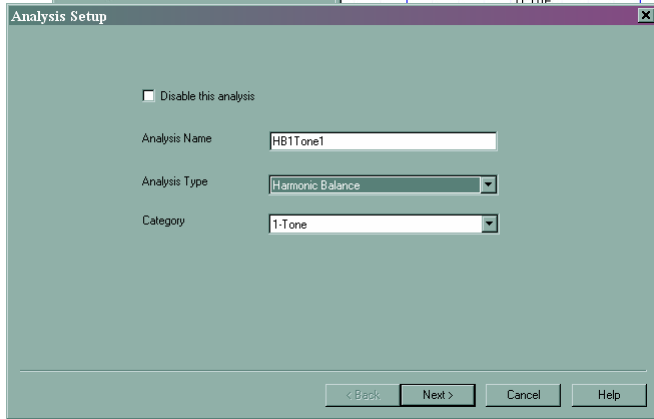
Double click on the input port symbol to open the Port Definition window. In the source section click Add to open the Source Selection window. Select power and enter Pin as the value of parameter P. Click ok in the Source Selection window. Click ok the Port Definition window. In the Project Manager window expand the Excitations folder to check that source named Sinussoidal1 is added.

# Define a Power Sweep

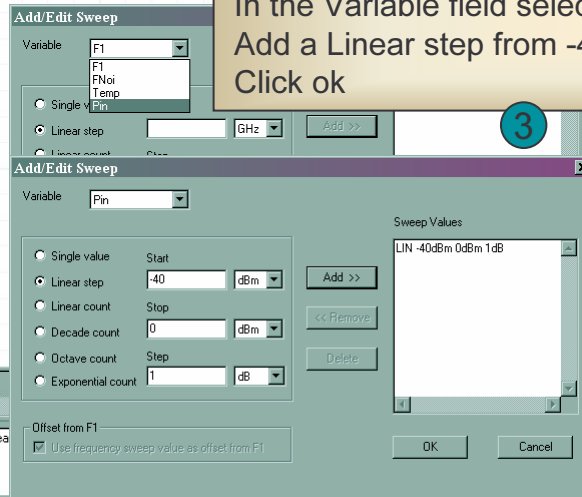
Highlight F1 and click Edit, add single value of 0.9GHz, Click OK.



Click **Add** in the Harmonic Balance Analysis, 1-Tone window, In the Variable field select Pin, Add a Linear step from -40dBm to 0dBm by step of 1dB, Click ok



Add Analysis Setup select :  
Analysis Type **Harmonic Balance**  
Analysis Name **HB1Tone1**  
Category **1-Tone**  
Click Next



Save the Project

# Run Harmonic Balance Analysis

**Circuit Analysis Details**

Circuit Name: LnaCircuit

Analysis Status:

Status: Analyzing

Sweeps: Tone1: 900.000000 MHz  
Pin: -26 dBm

Sweep Number: 1377 of 4001

Iteration: 0

Analysis Error: 2.307696e-007

Click right on Analysis Setup HBTone1 and select Analyze HBTone1.  
The Progress Bar appears click right in the progress bar and select details to Circuit Analysis Details.

# Create Results: Pou/TG21 vs Pin

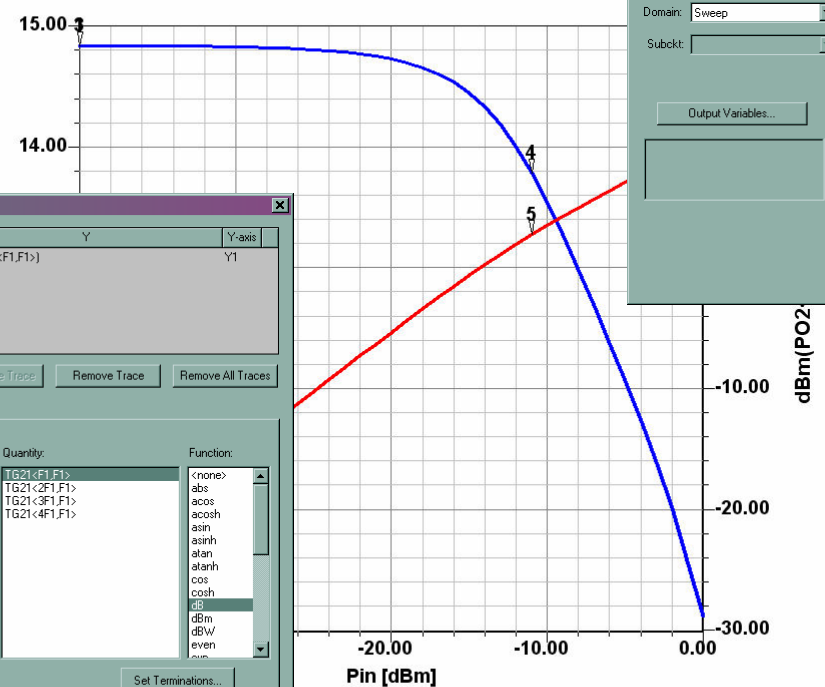
Select HB1Tone1 in field Solution  
 Select Sweep in field Domain  
 Select Power, PO2<F1>, dBm to plot the output power at port 2 for the fundamental.

2

Click right on Results and create report – Rectangular Plot

1

Ansoft Corporation  
 XY Plot 1  
 LnaCircuit



Traces

X	Y	Y-axis
Pin	dB(TG21<F1,F1>)	Y1
Pin	dBm(PO2<F1>)	Y2

Design: LnaCircuit | Sweeps: X | Y

Solution: HB1Tone1 | Category: Output Variables

Domain: Sweep | Subckt: [blank]

Quantity: TG21<F1,F1>, TG21<2F1,F1>, TG21<3F1,F1>, TG21<4F1,F1>

Function: <none>, abs, acos, acosh, asin, asinh, atan, atanh, cos, cosh, dB, dBm, dBW, even, ...

Buttons: Add Trace, Add Blank Trace, Replace Trace, Remove Trace, Remove All Traces, Output Variables..., Apply, Done, Cancel

Select Transducer Gain, TG21<F1,F1>, dB to plot the gain between fundamental at port1 and fundamental at port2. Click done

3

# Create Results: DCIV curves

The screenshot displays the Ansoft Designer interface for a DCIV analysis. The main window shows a graph of current  $I_c$  [mA] versus voltage Vce. The graph contains several curves: a blue curve representing the dynamic load line, and several other curves (green, black, pink, light green, red) representing the DC-IV characteristics of the device. A red circle highlights the intersection of the blue curve and one of the DC-IV curves, indicating the instantaneous operating point.

Three numbered callouts provide instructions:

- 1** Click right on Results and create report Device IV Characteristics.
- 2** Select HB1Tone1 Check DCIV Curves Check AC Load Line
- 3** Uncheck All Values Using CTRL key select -40, -30, -20, -10dBm Click Add Trace

The **Create Report** dialog box is open, showing the following settings:

- Target Design: LnaCircuit
- Report Type: Device IV Characteristics
- Display Type: Rectangular Plot

The **Traces** dialog box is also open, showing the following settings:

- Design: LnaCircuit
- Sweeps: Device Characteristics
- Solution: HB1Tone1
- Domain: Device
- Subckt: (empty)
- Number of Cycles: 2
- Characteristic: Output

The **Traces** table shows the following traces:

Trace	X	Y	Y-axis
1	Vce	Ic(Q1.Q1)	Y1
2	Vce	Ic(Q1.Q1)	Y1

The **Traces** dialog box also shows a list of dBm values for the sweep:

Name	Type	Description	Value
Pin	Primary Sweep	-40dBm..-3...	-40dBm
F1	Point(s)	All Values	-24dBm
			-23dBm
			-22dBm
			-21dBm
			-20dBm
			-19dBm
			-18dBm
			-17dBm
			-16dBm
			-15dBm
			-14dBm
			-13dBm
			-12dBm
			-11dBm
			-10dBm

The **Traces** dialog box also shows the following settings:

- Design: LnaCircuit
- Sweeps: Device Characteristics
- Solution: HB1Tone1
- Domain: Device
- Subckt: (empty)
- Number of Cycles: 2
- Characteristic: Output

The **Traces** dialog box also shows the following settings:

- Design: LnaCircuit
- Sweeps: Device Characteristics
- Solution: HB1Tone1
- Domain: Device
- Subckt: (empty)
- Number of Cycles: 2
- Characteristic: Output

The **Traces** dialog box also shows the following settings:

- Design: LnaCircuit
- Sweeps: Device Characteristics
- Solution: HB1Tone1
- Domain: Device
- Subckt: (empty)
- Number of Cycles: 2
- Characteristic: Output

Here, we see two types of traces:

1. Device DC-IV characteristics, taken from the nonlinear model
2. Instantaneous operating point, or dynamic load line over an RF cycle

# Create Results: Spectrum

28 Feb 2003

Click right on Results and create report Rectangular Plot. 1

Select Power, PO2, dBm  
Click Add Trace, Click done 3

Traces

Design: LnaCircuit Sweeps X Y  
Solution: HB1Tone1  
Domain: Spectral  
Subckt:

Name	Type	Description	
Spec...	Primary Sweep	All Values	-20dBm
Pin	Point(s)	-10dBm	-19dBm
F1	Point(s)	All Values	-17dBm
			-16dBm
			-15dBm
			-14dBm
			-13dBm
			-12dBm
			-11dBm
			-10dBm
			-9dBm
			-8dBm
			-7dBm
			-6dBm

Traces

1 Spectrum dBm(PO2) Y1

Add Trace Add Blank Trace Replace Trace Remove Trace Remove All Traces

Design: LnaCircuit Sweeps X Y  
Solution: HB1Tone1  
Domain: Spectral  
Subckt:

Category: Quantity: Function:

Variables: PO1, PO2, POdb[01.Q1], POcel[01.Q1]  
Output Variables: Power, Voltage, Current, Travelling Wave, State Variable, All

Function: (none), abs, acos, acosh, asin, asinh, atan, atanh, cos, cosh, dB, dBm, dBW, even, exp

Output Variables... Set Terminations...

Apply Done Cancel

2

Select HB1Tone1 in field Solution  
Select Spectral in field Domain  
Hit sweep Tab  
Select Pin = -10dbm for the Pin value.

20.00

1

2

Spectrum [GHz]

X1= 0.90GHz X2= 1.80GHz  
Y1= 3.54 Y2= -13.82

Ready

# Create Results: Wave Form

**1** Click right on Results and create report Rectangular Plot.

**2** Select HB1Tone1 in field Solution  
 Select Time in field Domain  
 Select Pin = -40, -30, -20, -10 dbm for the Pin values

**3** Select Voltage, V2, none  
 Click Add Trace, Click done

Save the Project



# Optional Exercises: (Intermod) Digital Modulation



# Add a Second RF Source to Port 1

**1** Double click on the input port

**2** Click Add in the Sources section

**3** Select Power. Set Fnum to f2 and Enter Pin for value of P. Click OK

**Source Selection**

Voltage  Power

Name: Sinusoidal2

Type: Sinusoidal

Property	Value	Unit	Description
hnum	f2		Harmonic number of the source
vdc	0	mV	DC offset of the source
p	pin		Available power of the source
phs	0	deg	Phase of the source
td	0	ns	Time delay

**Port Definition**

Port name: Port1

Port number: 1

Termination:  Simple termination: Re: 50 Im: 0 Impedance

Source Definition: Source type: Power

Enable	Name	Type	Modulation	Noise
<input checked="" type="checkbox"/>	Sinusoidal1	Sinusoidal	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	Sinusoidal2	Sinusoidal	<input type="checkbox"/>	<input type="checkbox"/>

Load Pull Tuner and Reference Node

Load Pull Tuner: <none>

Reference Node: Ground

Number of selected items: 1

# Add Intermodulation Analysis Setup

**1** Analysis Type Harmonic Balance  
 Analysis Name HB2ToneInter1  
 Category 2 Tones Intermodulation Spectrum  
 Click Next

**2** Highlight F2, Uncheck Offset option and click Edit, add single value of 0.901GHz, Click OK. Click Add

**3** Add a Linear step from -40dBm to 0dBm by step of 2dBm for variable Pin  
 Click ok

**Analysis Setup Dialog:**

- Analysis Name: HB2ToneInter1
- Analysis Type: Harmonic Balance
- Category: 2-Tone, Intermodulation Spectrum

**Harmonic Balance Analysis, 2-Tone, Intermodulation Spectrum Dialog:**

- Intermodulation Order: 5
- Stability Analysis:  Use Solution-Path Tracing
- Trace AC Power:

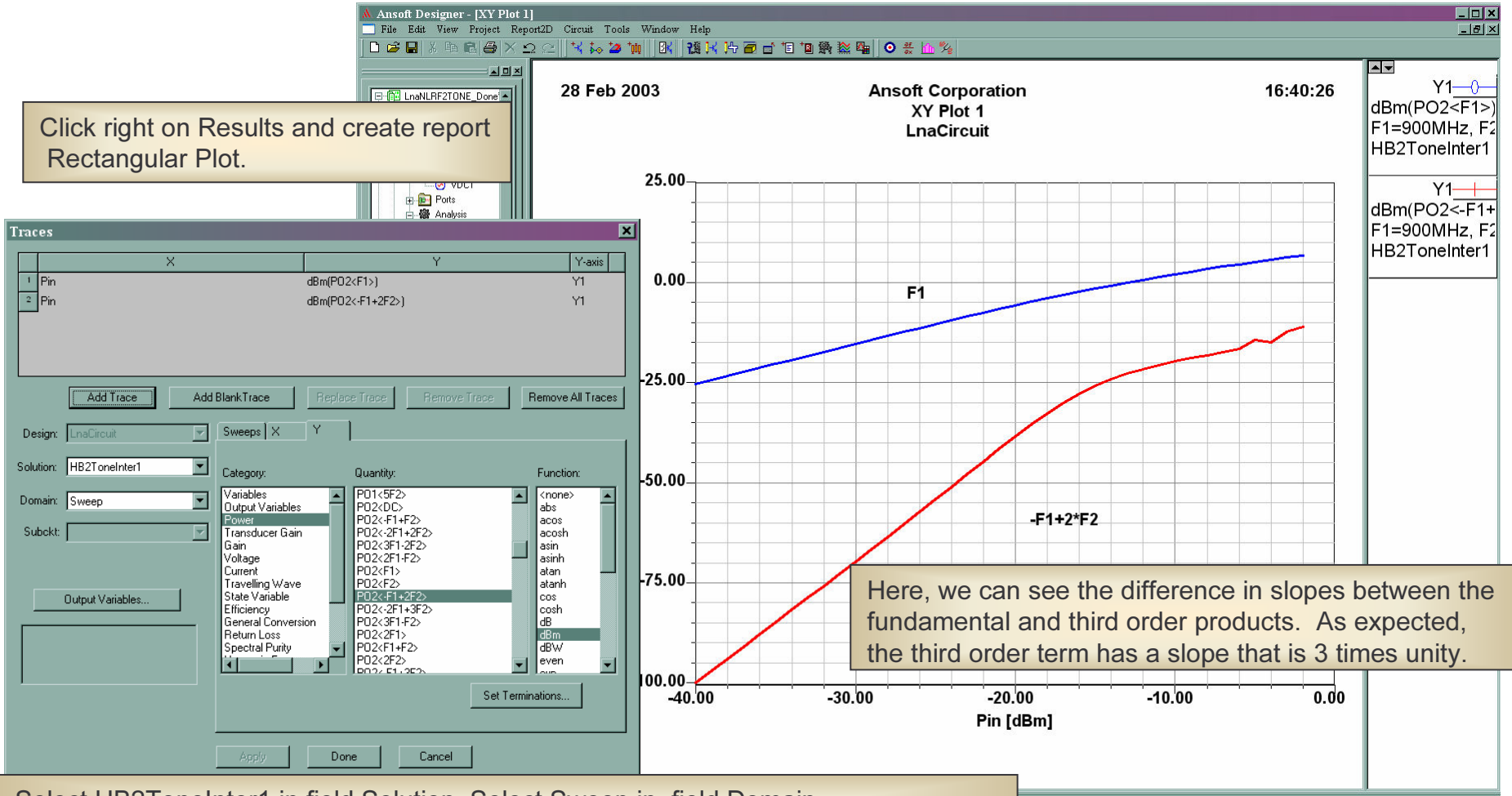
Name	Sweep/Value	Offset ...	Sync
F1	900mhz		
F2	901mhz	<input type="checkbox"/>	
Pin	LIN -40dBm -2dBm 1dB		

**Add/Edit Sweep Dialog:**

- Variable: Pin
- Single value:
- Linear step:  Start: -40 dBm, Stop: 0 dBm, Step: 2 dBm
- Linear count:
- Decade count:  Stop: -2 dBm
- Octave count:  Step: 1 dB
- Exponential count:
- Offset from F1:  Use frequency sweep value as offset from F1

# Create Results: Pout vs Pin

Click right on Results and create report Rectangular Plot.



Here, we can see the difference in slopes between the fundamental and third order products. As expected, the third order term has a slope that is 3 times unity.

Select HB2ToneInter1 in field Solution, Select Sweep in field Domain  
 Select Power, PO2<F1>, dBm to plot the output power at port 2 for the fundamental.  
 Select Power, PO2<-f1+2\*f2> to plot the output power at port2 for the IM3.  
 Click done

# Create Results: Calculate IP3

**1** Click right on Results and create report – Data Table.

Select HB2ToneInter1 in field Solution  
 Select Sweep in field Domain  
 Click on Output Variables button to open the output Variables window.

**2**

Select Output Variable in Category, IP3 in Quantity  
 None as function, Click Add Trace and Done **4**

**3** Enter in field expression :  
 $\text{dBm}(\text{PO2}<\text{F1}>)+(\text{dBm}(\text{PO2}<\text{F1}>)-\text{dBm}(\text{PO2}<2\text{F1}-\text{F2}>))/2$ .  
 Enter in field Name IP3, Click Add, Click done

**Output Variables**

Name	Expression
1 IP3	$\text{dBm}(\text{PO2}<\text{F1}>)+(\text{dBm}(\text{PO2}<\text{F1}>)-\text{dBm}(\text{PO2}<2\text{F1}-\text{F2}>))/2$

Name: IP3  
 Expression:  $\text{dBm}(\text{PO2}<\text{F1}>)+(\text{dBm}(\text{PO2}<\text{F1}>)-\text{dBm}(\text{PO2}<2\text{F1}-\text{F2}>))/2$

**Traces**

Design: LnaCircuit  
 Solution: HB2ToneInter1  
 Domain: Sweep  
 Subckt:

Category: Variables  
 Quantity: IP3  
 Function: <none>

dBm(PO2<F1>)+((dBm(PO2<F1>)-dBm(PO2<2F1-F2>))/2)

# Create Results: Intermodulation Spectrum

Click right on Results and create report – Rectangular Plot  
 Select HB2ToneInter1 in field Solution  
 Select Spectral in field Domain

1

Select Pin, uncheck All Values,  
 Select -5dbm

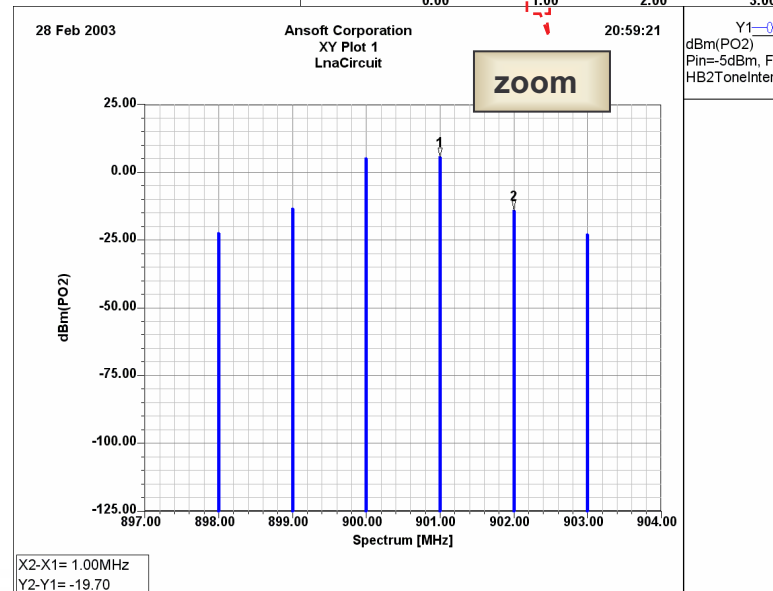
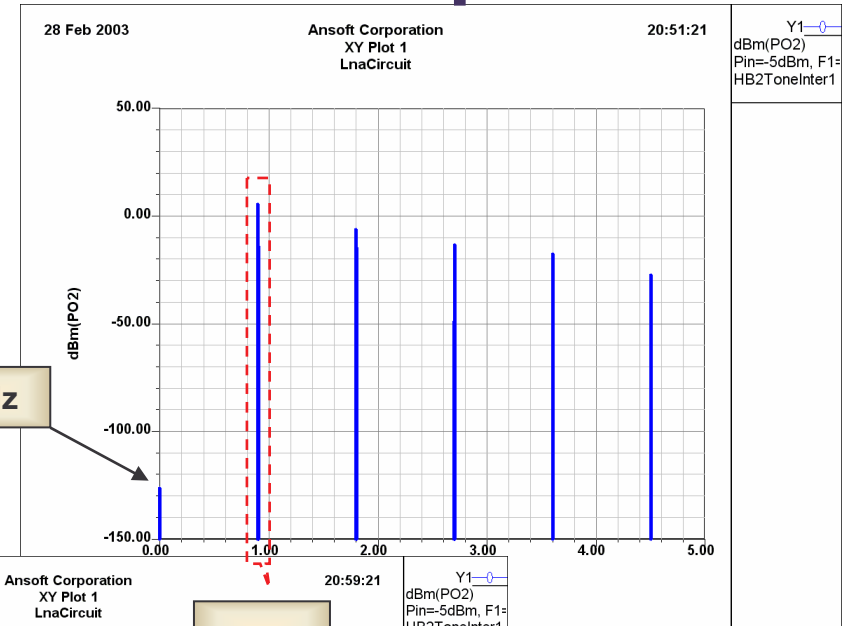
2

Select Power, PO2, dBm  
 Click Add Trace, Click Done

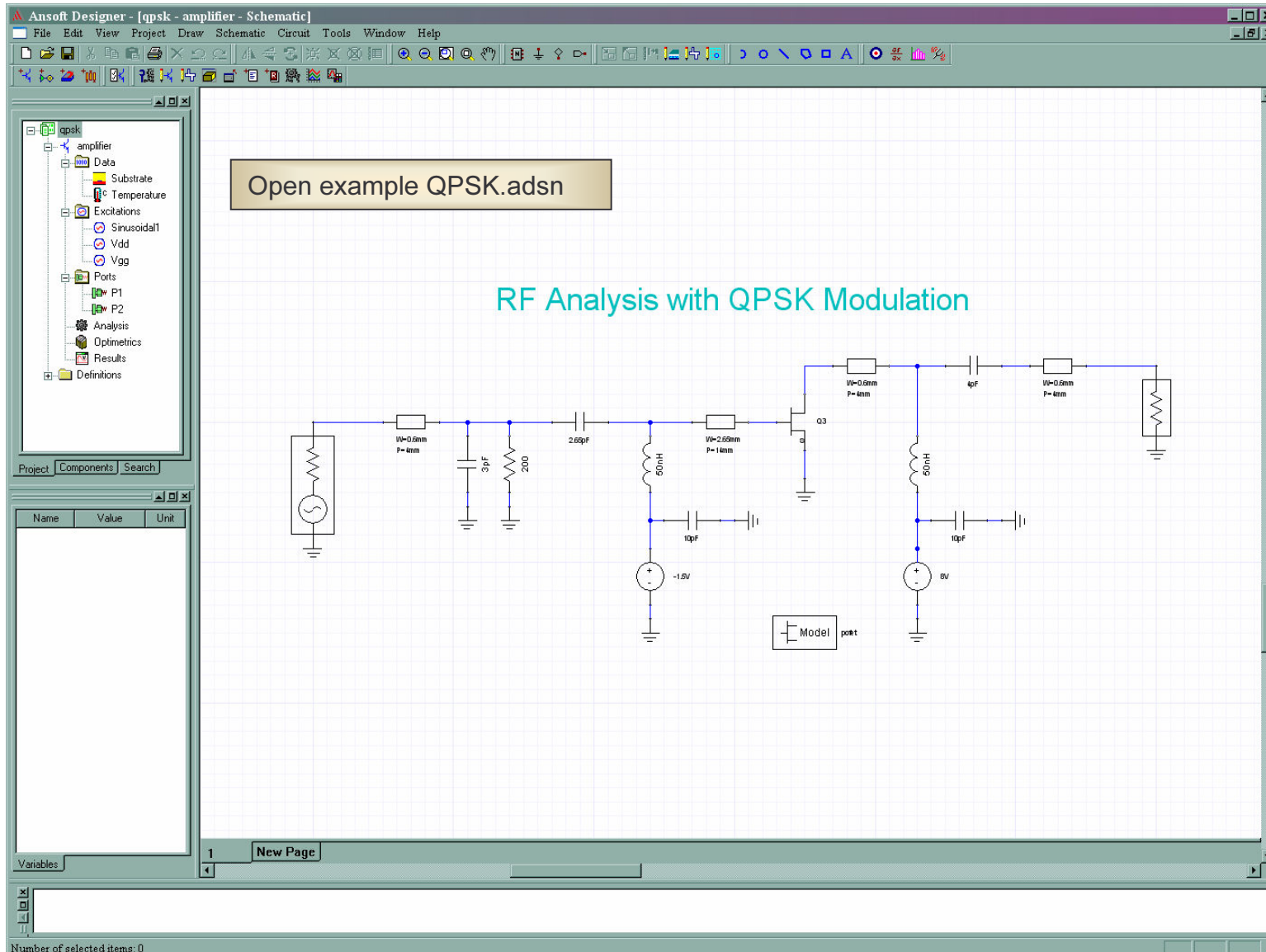
3

Name	Type	Description
-16dBm		
-15dBm		
-14dBm		
-13dBm		
-12dBm		
-11dBm		
-10dBm		
-9dBm		
-8dBm		
-7dBm		
-6dBm		
-5dBm		
-4dBm		
-3dBm		

Variables	Quantity	Function
PO1		krone>
PO2		abs
PObe(Q1.Q1)		acos
POce(Q1.Q1)		acosh
		asin
		asinh
		atan
		atanh
		cos
		cosh
		dB
		dBm
		dBW
		even
		exp



# Digital Modulation



# Digital Modulation: Modulation Source (1)

**Edit source Sinussoidal1**

**Channel Meas**  
specify channel bandwidth

**Filter Tab**  
select the filter Butterworth, Gaussian, Root-Raised, Cosine, Raised Cosine

**Parameters Tab**  
Bit Rate, Delay, I/Q Imbalance

You can select from the type of the Modulation Source CDMA2000, GSM, GSM Edge, GMSK, ..., User Defined.

**Source Selection**

Voltage  
 Power  
 Name: Sinussoidal1  
 Type: Sinussoidal

Property	Value	Unit	Description
td	0	ns	Time delay
f	0	GHz	Frequency for transient analysis
df	0.		Damping factor
noise	<none>		Noise model name
mod	<none>		Modulation name

**Modulation Source**

Name: SourceModulation1  
Type: IS-95

Parameters | Filter | Channel Meas | Diagram

Property	Value	Unit	Description
Br	1.2288	MHz	Bit rate of the binary system
Dly	0.5		Fractional bit delay
Iasc	1		I channel amplitude scale
Qasc	1		Q channel amplitude scale

**Click to Edit the Modulation Source window**



# Digital Modulation: Modulation Source (1)

**Parameter Tab**  
set Type to PSK, Br=1.2288MHz, m=4, Dly=0, lasc=Qasc=1

**Filter Tab**  
set Type Butterworth, LPFC=665KHz, LPFN=3

**Channel Meas Tab**  
set BW2=BW3=590KHz and FS2=1.99KHz,FS3=3,24KHz

**Modulation**

**Click OK**

**Modulation Source**

Name: SourceModulation1  
Type: PSK

Parameters Filter Channel Meas Diagram

Property	Value	Unit	Description
LPFC	665	kHz	-3dB cutoff freq. of lpf
LPFN	3		Number of resonators if lpf=1

Property Value Unit Description

Property	Value	Unit	Description
Br	1.2288		Binary system
m	4		Order of the signal space (power of 2)
Dly	0		Fractional bit delay
lasc	1		I channel amplitude scale
Qasc	1		Q channel amplitude scale

Property Value Unit Description

Property	Value	Unit	Description
BW2	590	kHz	one sided bandwidth of main channel in P2IB
BW3	590	kHz	one sided bandwidth of main channel in P3IB
FS2	1.99	MHz	start baseband frequency of second adjacent c
FS3	3.24	MHz	start baseband frequency of third adjacent char

# Digital Modulation: Analysis Setup

**1** Click right on analysis and select add Analysis setup. Select type Analysis Type Modulation Envelope, Category 1-Tone Click Next

The Modulation Source is added to the Data Folder.

You can select Modulation Envelope from the Analysis Type field. Category allows you to select  
 1-Tone,  
 2-Tone/3-Tone, Intermododulation Spectrum  
 2-Tone/3-Tone, Mixer Intermododulation Spectrum

**3** Set Length of Analysis to 104.2us  
 (=1/(br\*8)\*1024, with 1024=nb of sample)  
 Set Time step to 0.1us (= 1/(br\*8)  
 this means we oversample the bit rate by 8)

**2** Set F1 to 2GHz  
 define a power sweep on Pavs from 0dbm to 23dbm by step of 1db  
 Set the number of Harmonic to 8,

**4** Click Finish

Name	Sweep/Value	Sync
F1	2ghz	
Pavs	LIN 0dBm 23dBm 1dB	

# Digital Modulation: Eye Diagram

**1** Click right on Results and select Create  
Select Report Type=Eye Diagram,  
Display Type=Rectangular Plot,  
Click OK.

Tab Sweeps uncheck All  
for Pavs and select 0dBm.

**2**

Name	Type	Description	0dBm
Time	Primary Sweep	All Values	0dBm
Pavs	Point(s)	0dBm	3dBm
F1	Point(s)	All Values	4dBm
			5dBm
			6dBm

Traces

Eye Diagram

1 IchEye2<F1>

Add Trace Add BlankTrace Replace Trace Remove Trace Remove All Traces

Design: amplifier Sweeps Eye Diagram

Category: Modulation Responses

Quantity: IchEye1<F1> IchEye1<2F1> IchEye1<3F1> IchEye1<4F1> IchEye1<5F1> IchEye1<6F1> IchEye1<7F1> IchEye1<8F1> IchEye2<F1> IchEye2<2F1> IchEye2<3F1> IchEye2<4F1> IchEye2<5F1> IchEye2<6F1>

Function: <none>

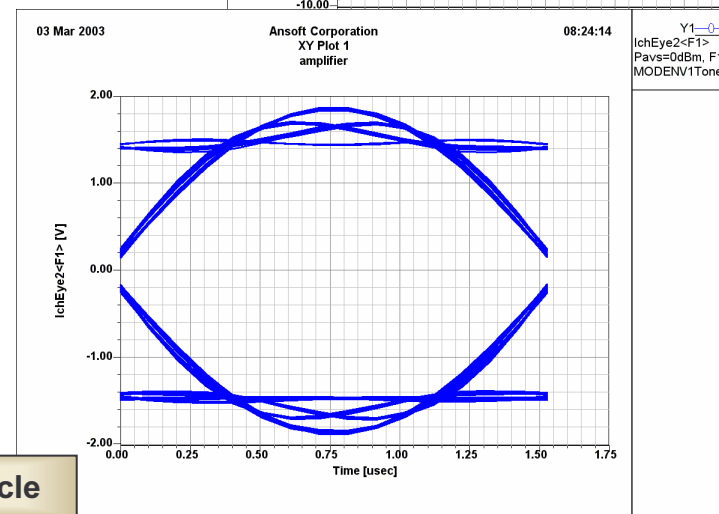
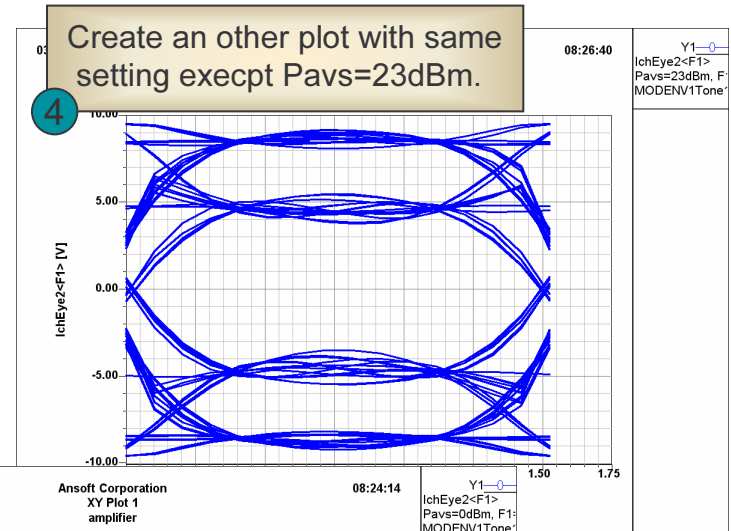
of Cycles 1

Done Cancel

**3**

Tab Eye Diagram select  
Modulation  
Response,  
Quantity=  
IchEye2<F1>

Available plot for Modulation Envelope are:  
Eye Diagram, Constellation, IQ Spectrum, ACPR



Click Done

Number of Cycle

# Digital Modulation: IQ Spectrum

The image shows two windows from the ANSYS HFSS software. The top window is the 'Create Report' dialog, and the bottom window is the 'Spectrum' plot configuration window.

**Create Report Dialog (Step 1):**

- Target Design: amplifier
- Report Type: Standard
- Display Type: Rectangular Plot

**Spectrum Plot Configuration (Step 3):**

- Design: amplifier
- Solution: MODENV1Tone1
- Domain: Spectral
- Category: Modulation Responses
- Quantity: IQ2<F1>
- Function: dB

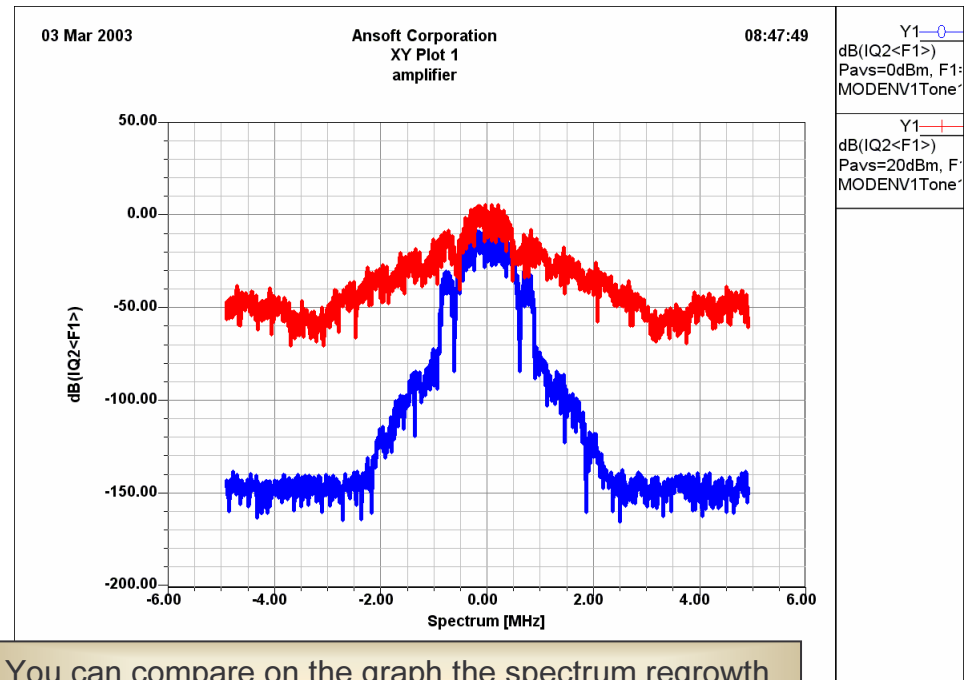
Click right on Results and select Create. 1

Select Report Type=Standard, Display Type=Rectangular Plot, Click OK. 2

Tab Sweeps uncheck All for Pavs, select 0dBm and 20dBm using the CTRL key. 3

Tab Y select Category=Modulation Responses, Quantity=IQ2<F1>, Function dB.

Click Done.



You can compare on the graph the spectrum regrowth produced for Pin=0dBm and Pin=20dBm.

# Digital Modulation: Constellation Plot

Click right on Results and select Create  
 Select Report Type=Constellation, Display Type=Rectangular Plot, Click OK.  
 Tab Sweeps uncheck All for Pavs and select 0dBm.  
 Tab Constellation select Modulation Response, Quantity= Constlltn2<F1>.  
 Click Done

1  
2  
3

**Create Report**

Target Design: amplifier

Report Type: Standard

Display Type: Constellation

OK Cancel

Design: amplifier

Sweeps: Constellation

Solution: MODENV1Tone1

Domain: Time

Name	Type	Description	0dBm
Time	Primary Sweep	All Values	
Pavs	Point(s)	0dBm	
F1	Point(s)	All Values	

**Traces**

Constellation

1 Constlltn2<F1>

Add Trace Add Blank Trace Replace Trace Remove Trace Remove All Traces

Design: amplifier

Sweeps: Constellation

Solution: MODENV1Tone1

Domain: Time

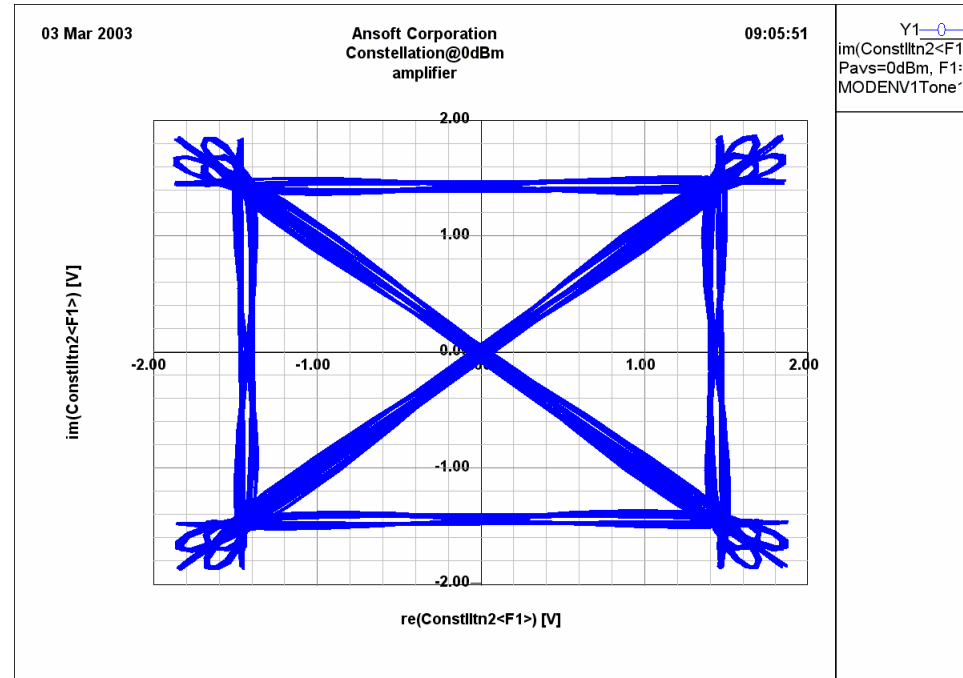
Output Variables...

Category	Quantity	Function
Variables	Constlltn1<F1>	<none>
Output Variables	Constlltn1<2F1>	
Modulation Responses	Constlltn1<3F1>	
	Constlltn1<4F1>	
	Constlltn1<5F1>	
	Constlltn1<6F1>	
	Constlltn1<7F1>	
	Constlltn1<8F1>	
	Constlltn2<F1>	
	Constlltn2<2F1>	
	Constlltn2<3F1>	
	Constlltn2<4F1>	
	Constlltn2<5F1>	
	Constlltn2<6F1>	

Vector Constellation

Sample waveform every 32 samples Offset 0

Apply Done Cancel



# Load Pull Analysis

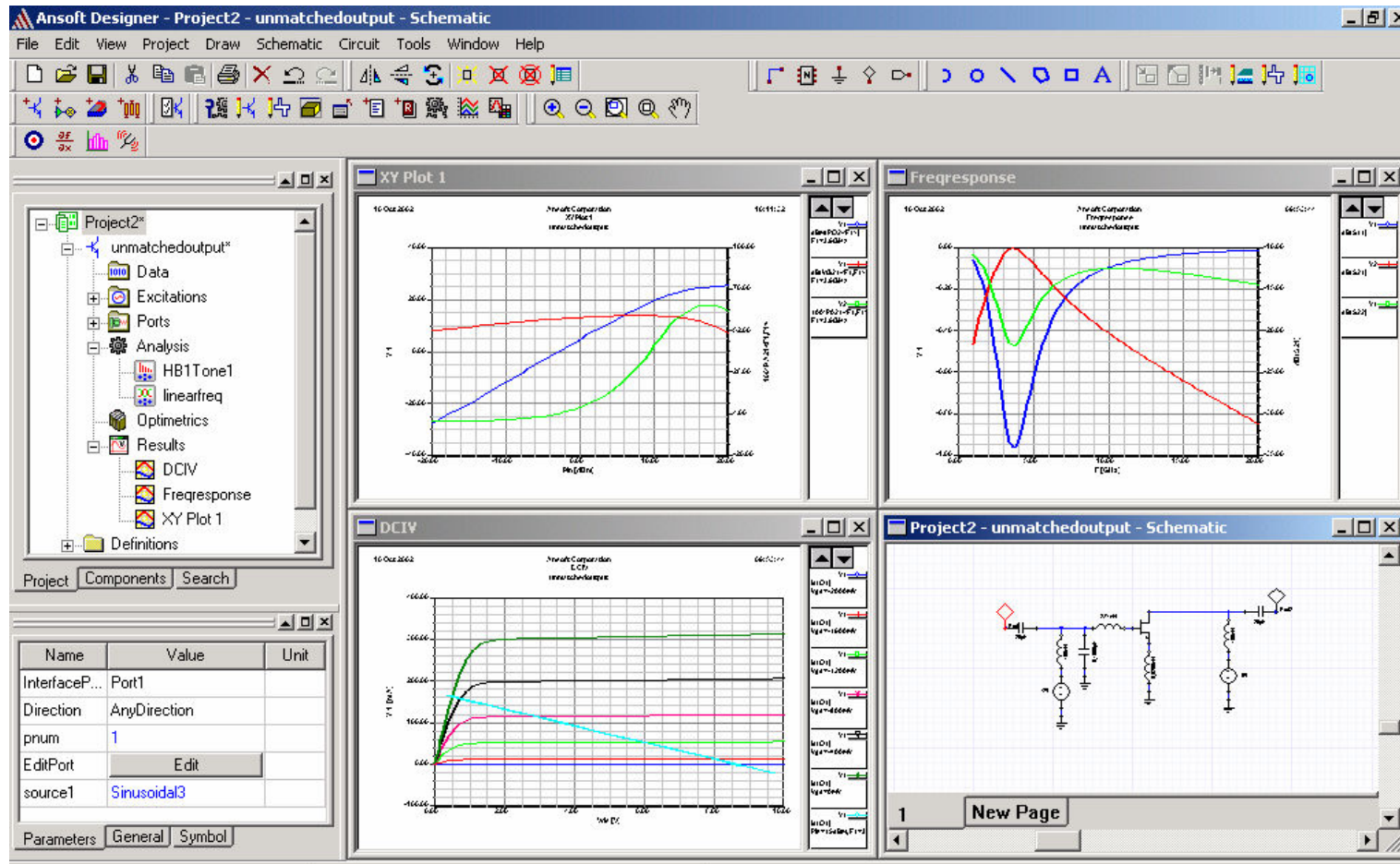


# Overview

There are 7 essential Steps to perform load-pull analysis and review response

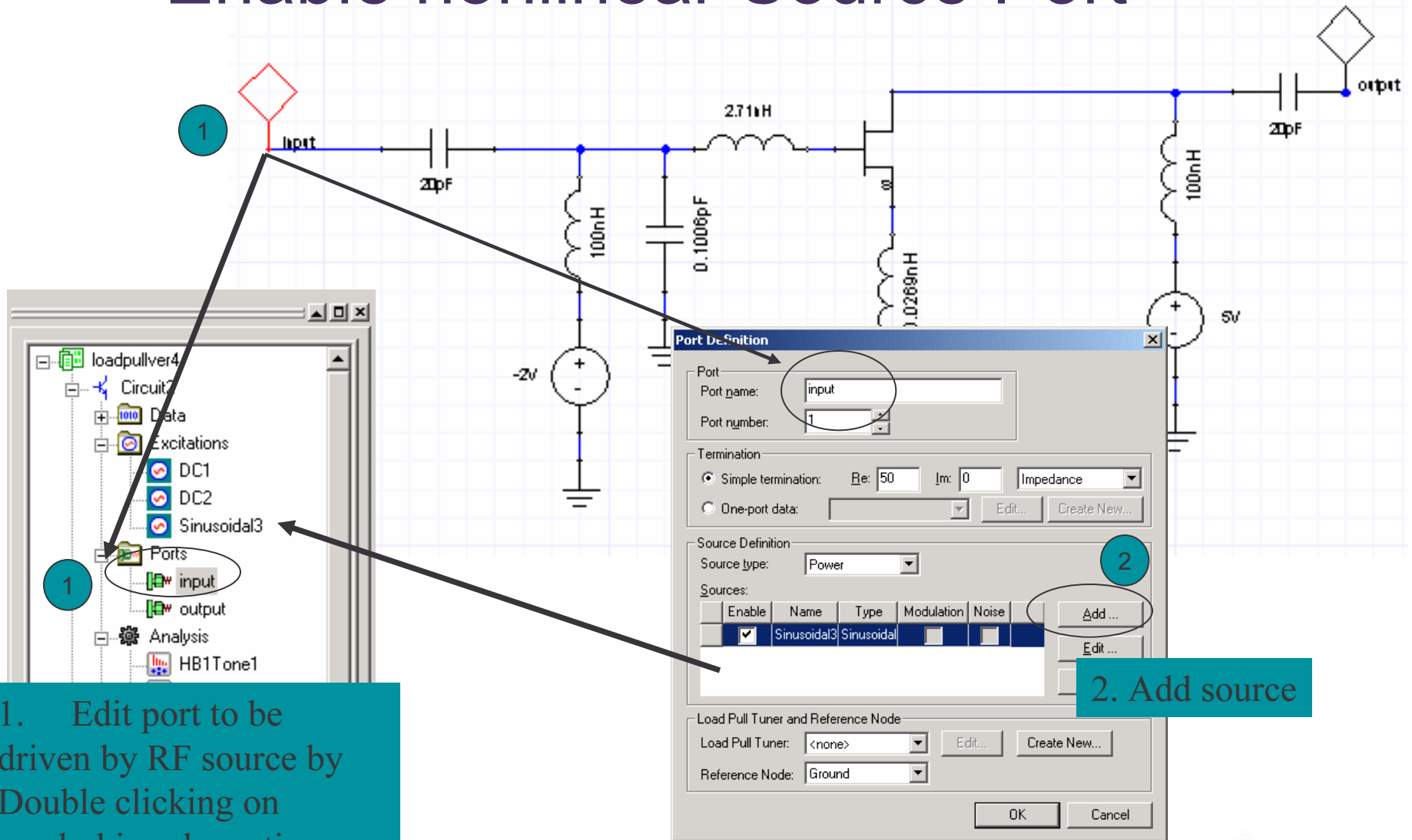
1. Define nonlinear circuit
  2. Specify nonlinear source
  3. Specify load/source pull tuner port
  4. Specify nonlinear analysis
  5. Specify loadpull analysis details based on selected nonlinear analysis
  6. Analyze
  7. Plot contours
- } Viewed in previous slides

# Loadpull Analysis in Ansoft Designer





# Enable nonlinear Source Port

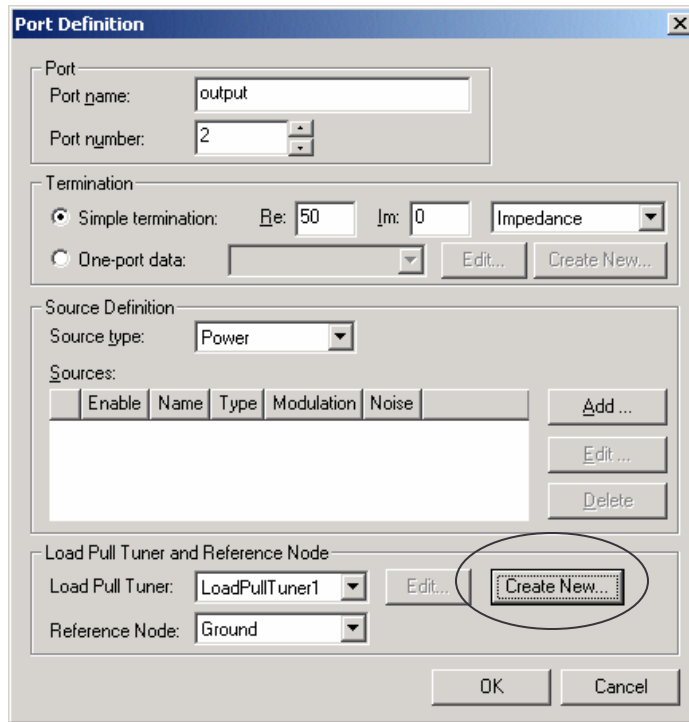


# Enable Loadpull termination

The image shows a screenshot of the ANSYS HFSS software interface. On the left is a project tree for a project named 'loadpullver4\*'. Under the 'Ports' folder, 'LoadPullTuner1' is highlighted with a red circle and an arrow pointing to a teal instruction box. The main workspace shows a circuit diagram with a 2.71nH inductor, a 100nH inductor, a 0.1006pF capacitor, and a 30pF capacitor. A red diamond-shaped port is labeled 'output' and has a blue circle with the number '1' next to it. A 'Port Definition' dialog box is open, showing the 'output' port name and port number '2'. The 'Termination' section has 'Simple termination' selected with a real impedance of 50 and an imaginary impedance of 0. The 'Source Definition' section has 'Power' selected. In the 'Load Pull Tuner and Reference Node' section, 'Load Pull Tuner' is set to 'LoadPullTuner1' and 'Reference Node' is set to 'Ground'. The 'Create New...' button in this section is highlighted with a red circle and a blue circle with the number '2'. A teal box on the left contains the following instructions:

1. Edit desired tuner port
2. Create new tuner Definition – settings shown in next slide
2. Specify Reference node (ground default)
3. Tuner automatically added to data definitions folder in project tree

# Define Loadpull tuner characteristics



Port Definition dialog box showing port configuration and termination options.

Port: Port name: output, Port number: 2

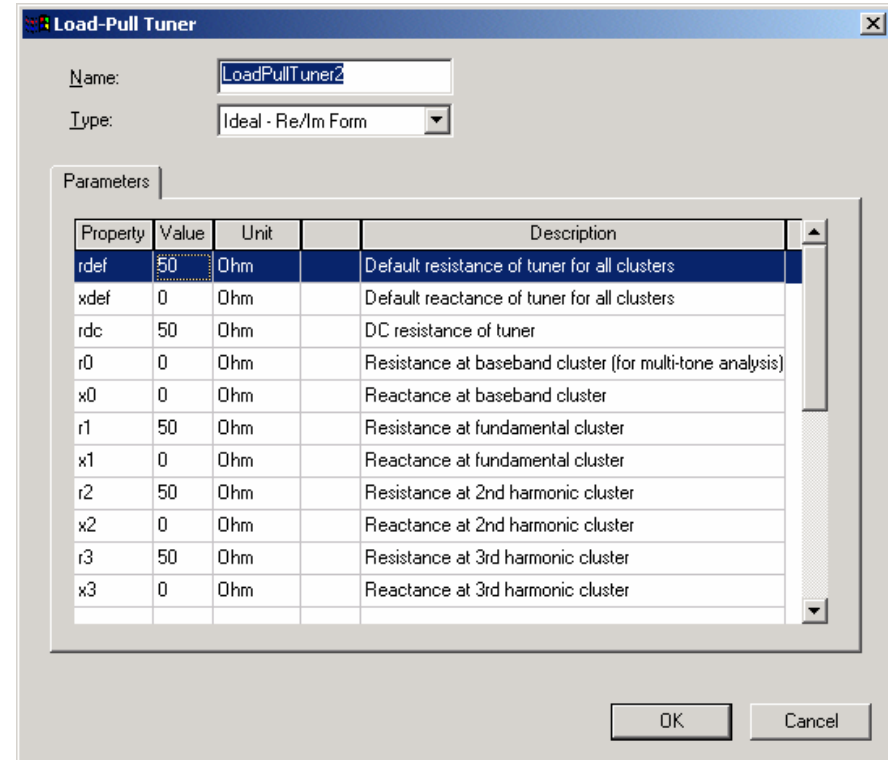
Termination:  Simple termination: Re: 50, Im: 0, Impedance;  One-port data: [ ] Edit... Create New...

Source Definition: Source type: Power

Sources: [ ] Add... Edit... Delete

Load Pull Tuner and Reference Node: Load Pull Tuner: LoadPullTuner1 Edit... Create New...; Reference Node: Ground

OK Cancel



Load-Pull Tuner dialog box showing tuner configuration and parameters.

Name: LoadPullTuner2, Type: Ideal - Re/Im Form

Parameters table:

Property	Value	Unit	Description
rdef	50	Ohm	Default resistance of tuner for all clusters
xdef	0	Ohm	Default reactance of tuner for all clusters
rdc	50	Ohm	DC resistance of tuner
r0	0	Ohm	Resistance at baseband cluster (for multi-tone analysis)
x0	0	Ohm	Reactance at baseband cluster
r1	50	Ohm	Resistance at fundamental cluster
x1	0	Ohm	Reactance at fundamental cluster
r2	50	Ohm	Resistance at 2nd harmonic cluster
x2	0	Ohm	Reactance at 2nd harmonic cluster
r3	50	Ohm	Resistance at 3rd harmonic cluster
x3	0	Ohm	Reactance at 3rd harmonic cluster

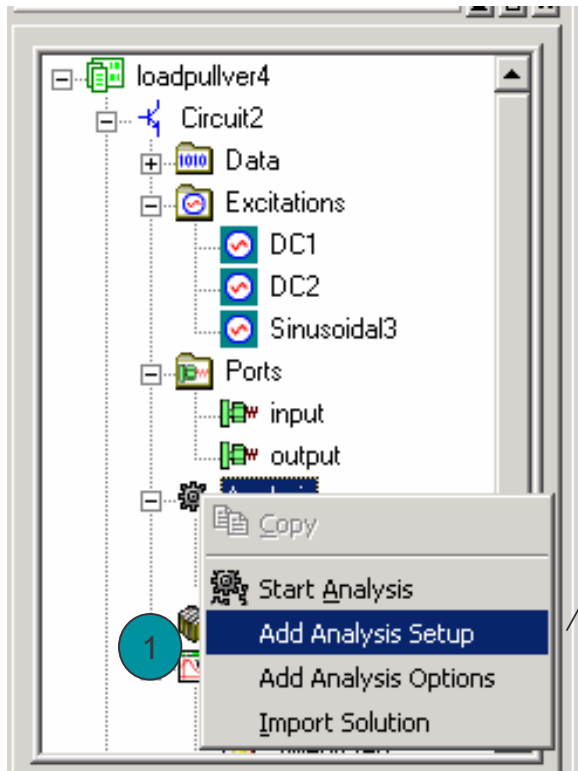
OK Cancel

Loadpull tuner definition specifies tuner type (ideal, double stub) and default complex impedance at all specified harmonic frequencies (50ohm default). To modify edit from port dialog or double click on tuner icon in project tree.

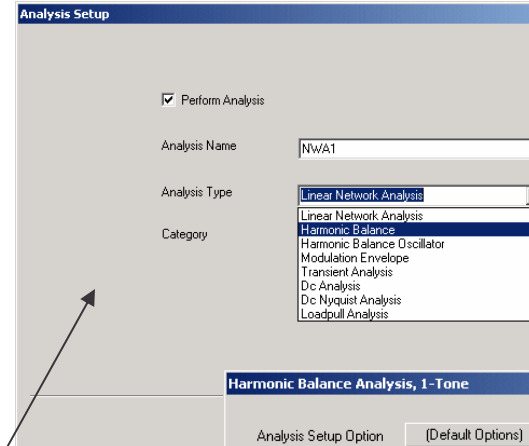
**Note: multiple tuners may be defined**



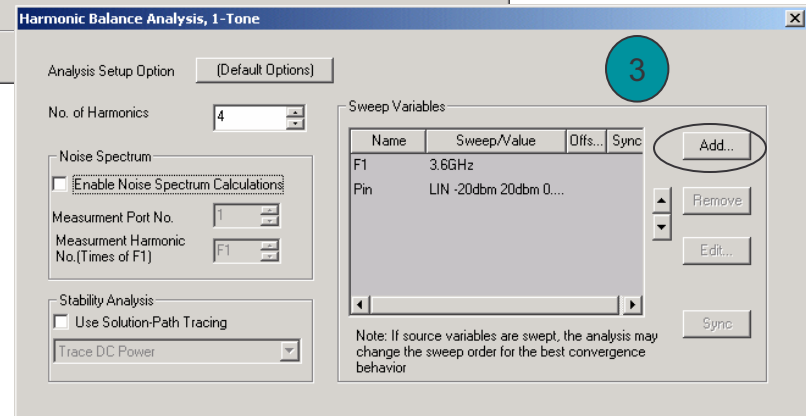
# Create new nonlinear analysis



1. Add analysis setup

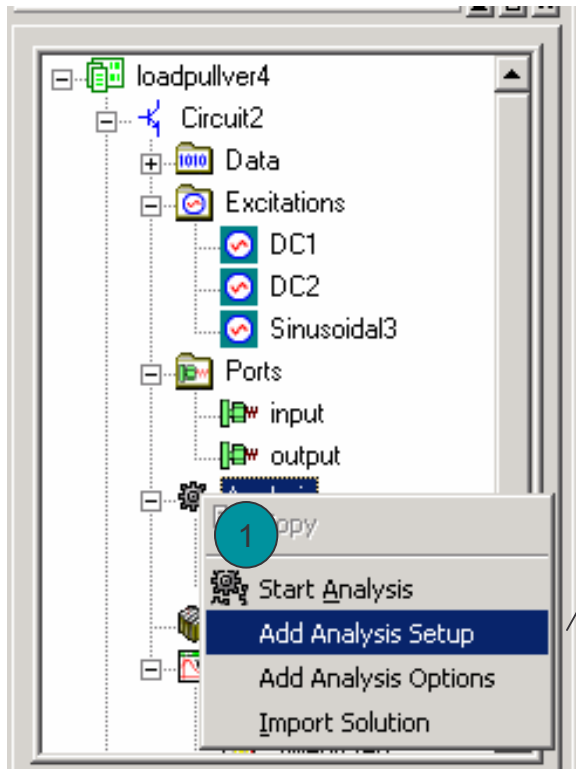


2. Select nonlinear analysis such as Harmonic Balance or HB Oscillation

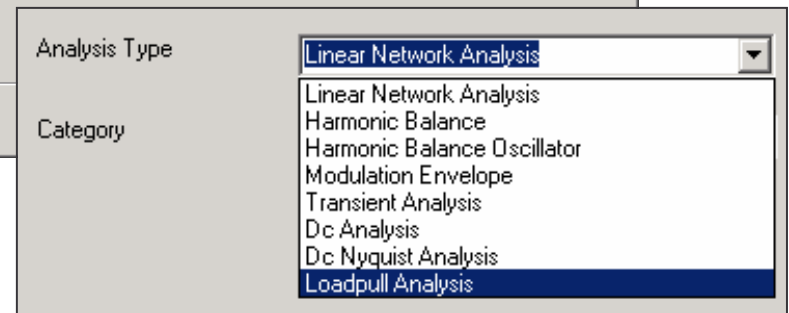
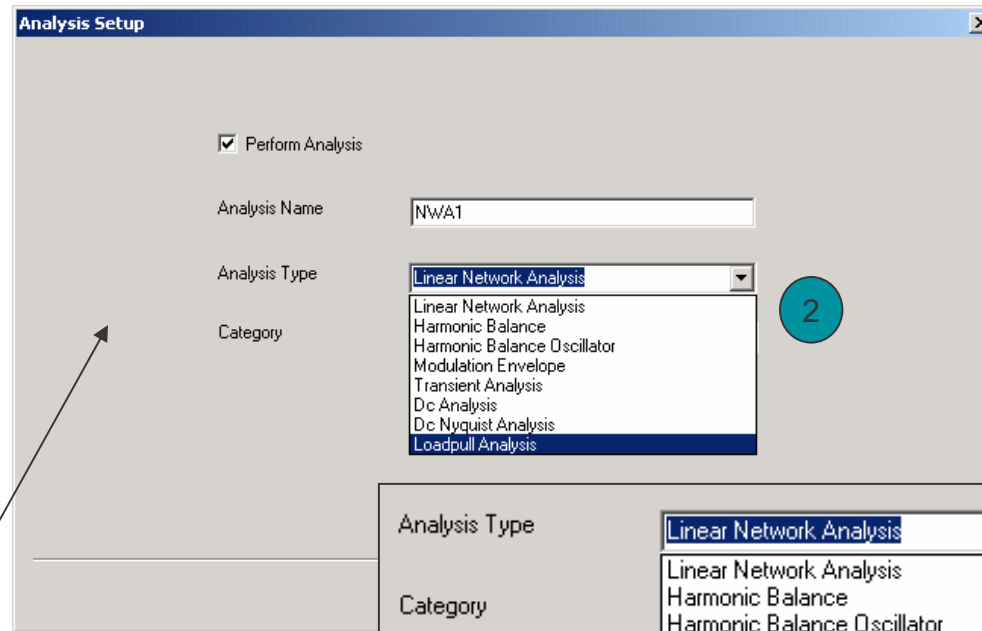


3. Specify swept parameters such as operating Frequencies and swept parameters such as Bias, tuning or power level (using the Add... Button)

# Create Loadpull analysis



1. Add analysis setup



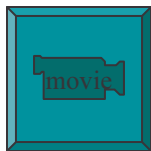
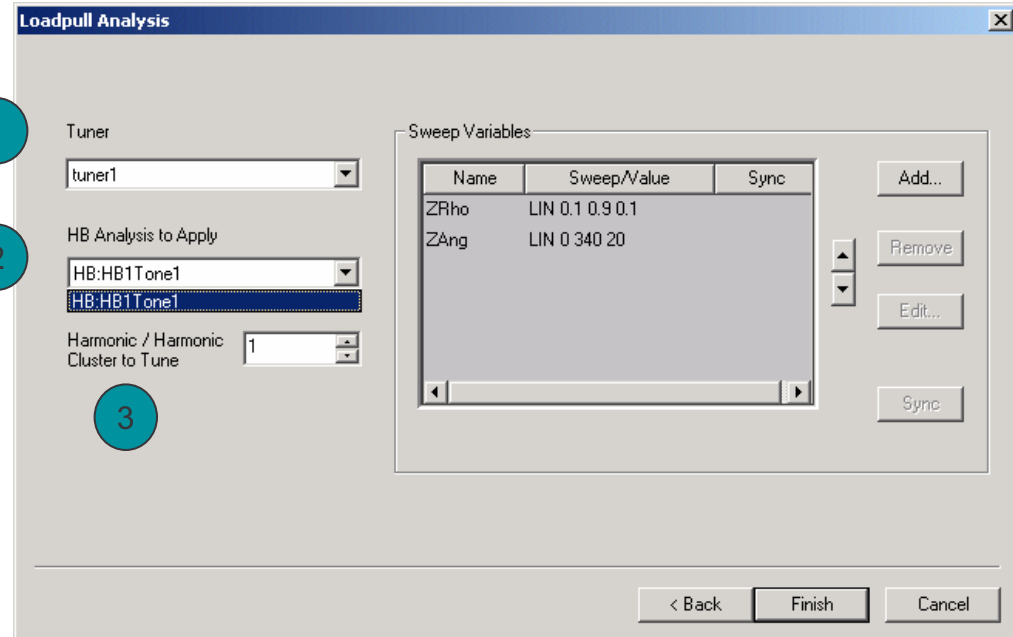
2. Specify swept parameters such as operating Frequencies and swept parameters such as Bias, tuning or power level (using the Add... Button)

# Create Loadpull analysis

1. Select loadpull tuners from pull down list (if multiple tuners are defined)

2. Select HB analysis to apply (multiple analyses may be defined for a given design)

3. Specify Harmonic frequency index, For example 1=fundamental  
2=second harmonic ( $2*f$ )



# Create Loadpull analysis

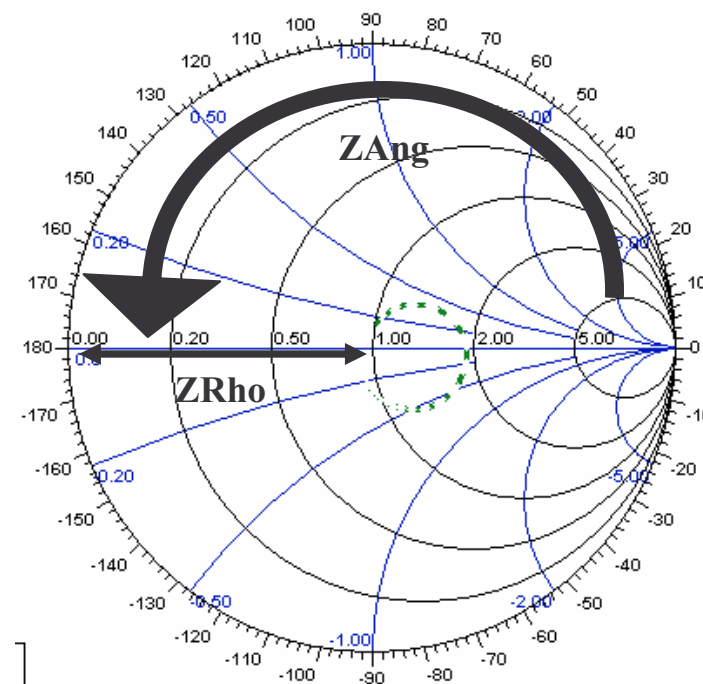
Set tuner range

a. ZRho controls the magnitude of the reflection coefficient

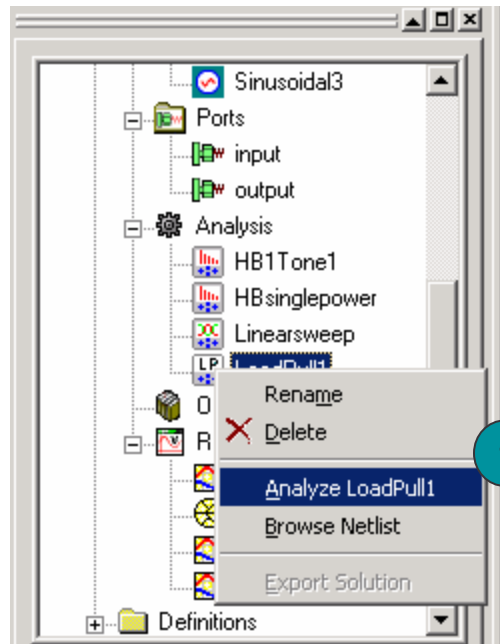
b. ZAng controls the angle of the reflection coefficient

**Hint: low impedance matching networks that are common to high power amplifiers are simulated more quickly by using a directed loadpull sweep of low impedances**

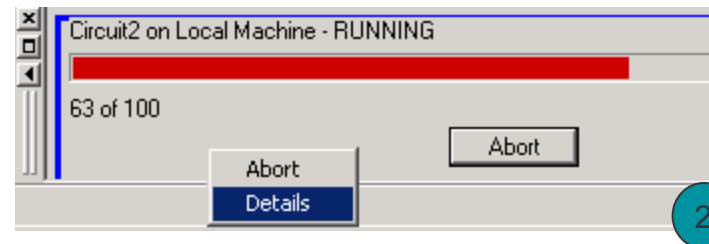
Sweep Variables		
Name	Sweep/Value	Sync
ZRho	LIN 0.1 0.9 0.1	
ZAng	LIN 0 340 20	



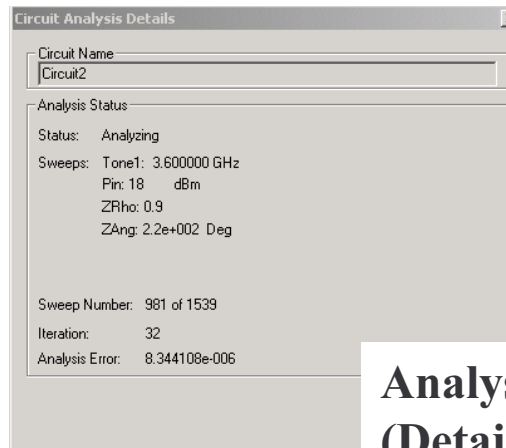
# Perform Loadpull analysis



1. Right mouse click on the defined loadpull analysis setup in the Analysis folder of the project tree



2. Right mouse click on the progress bar to bring up abort/details menu, select details to view HB convergence



Analysis status window (Details)

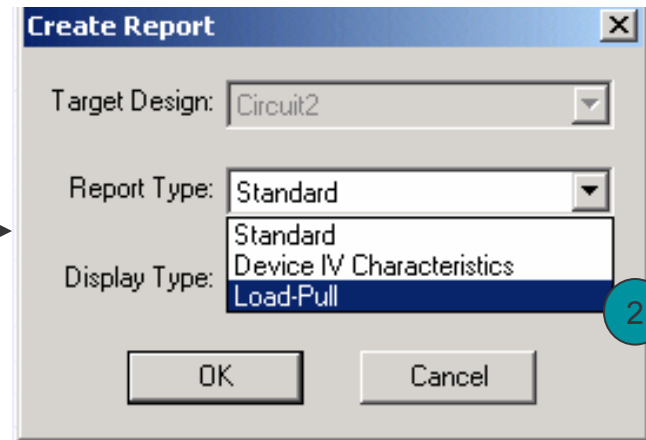




# Plotting Results (Contours)

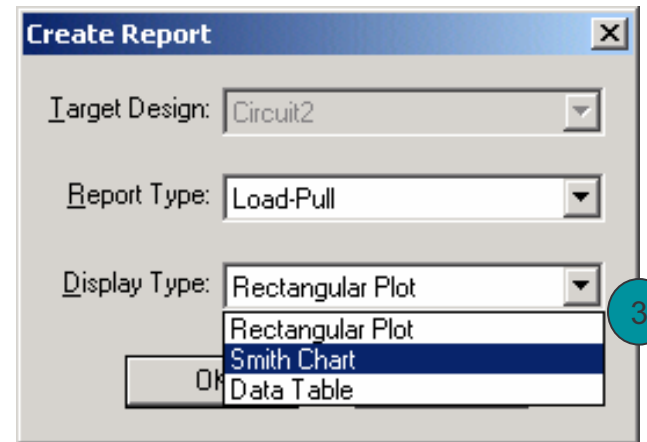


1. Right mouse click on the Results folder to create new report

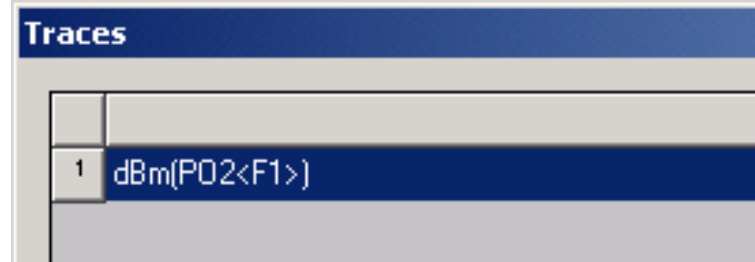
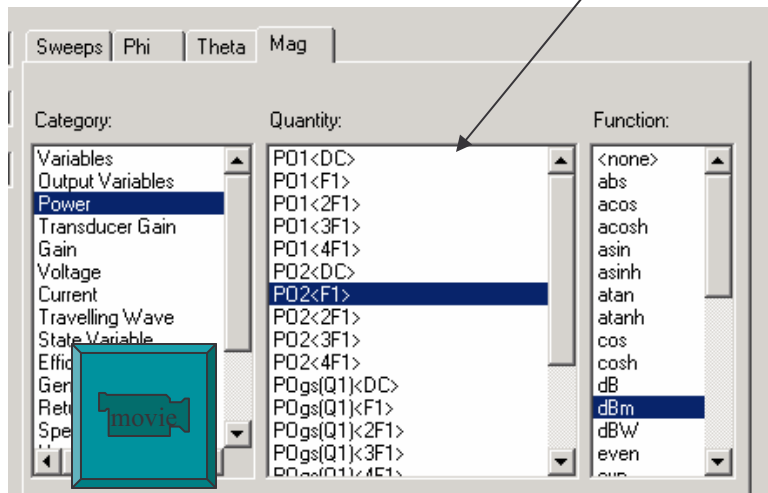
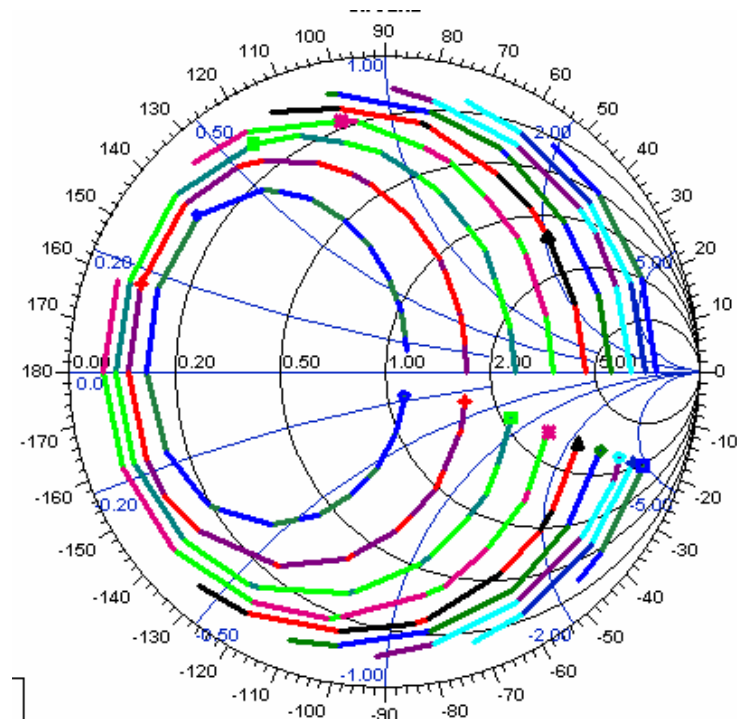
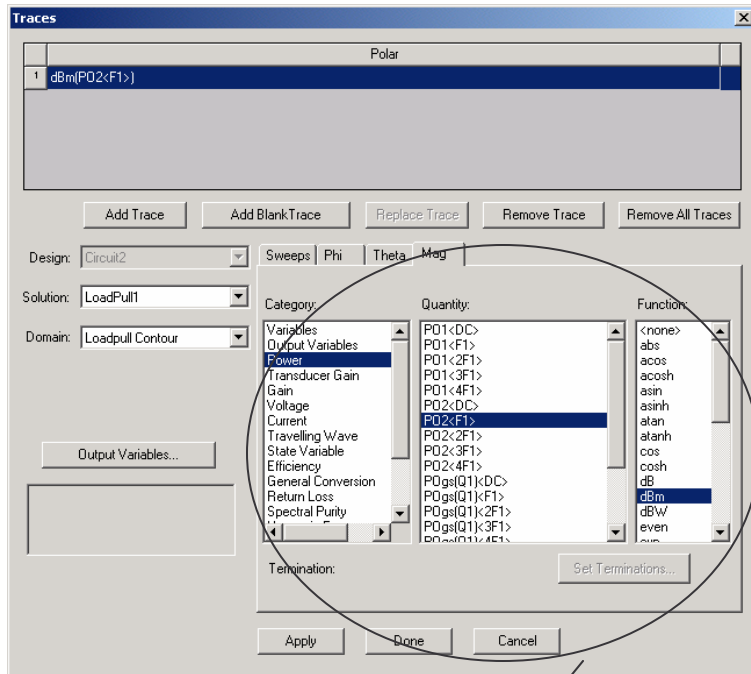


2. Select report type as Load-pull (only available after successful load-pull simulation)

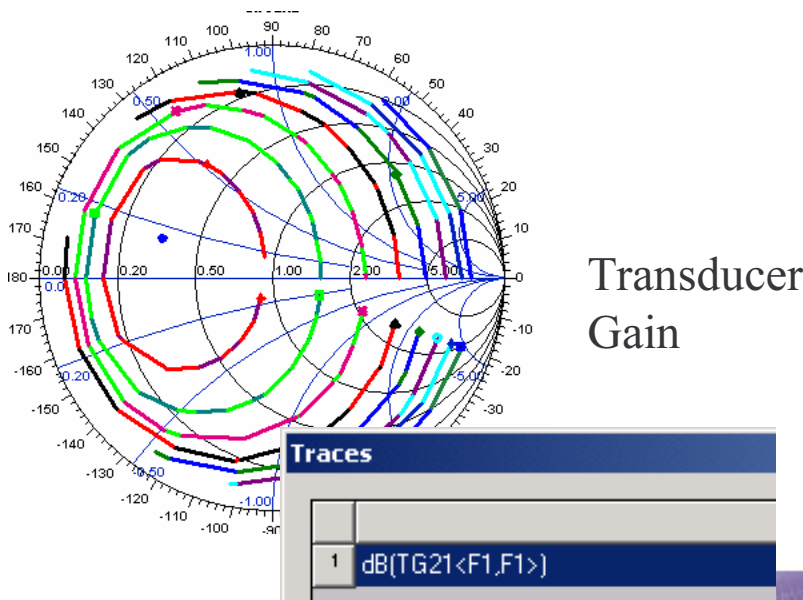
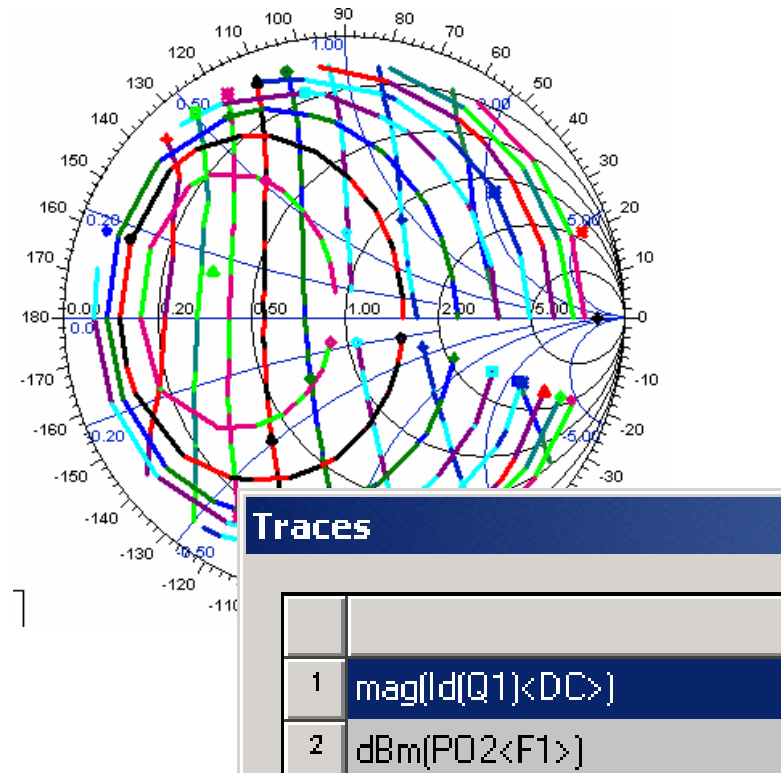
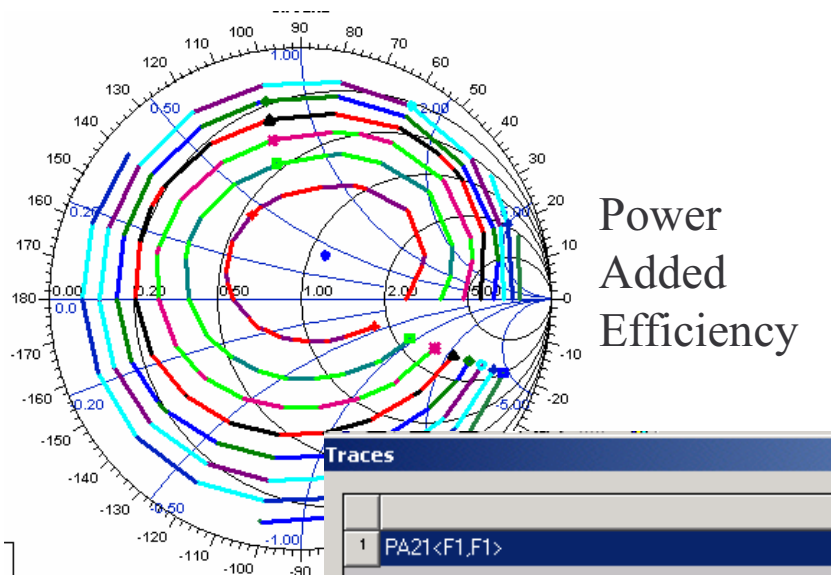
3. Select display type as smith chart (optional rectangular or polar plots also allowed)



# Plotting Output Power Contours

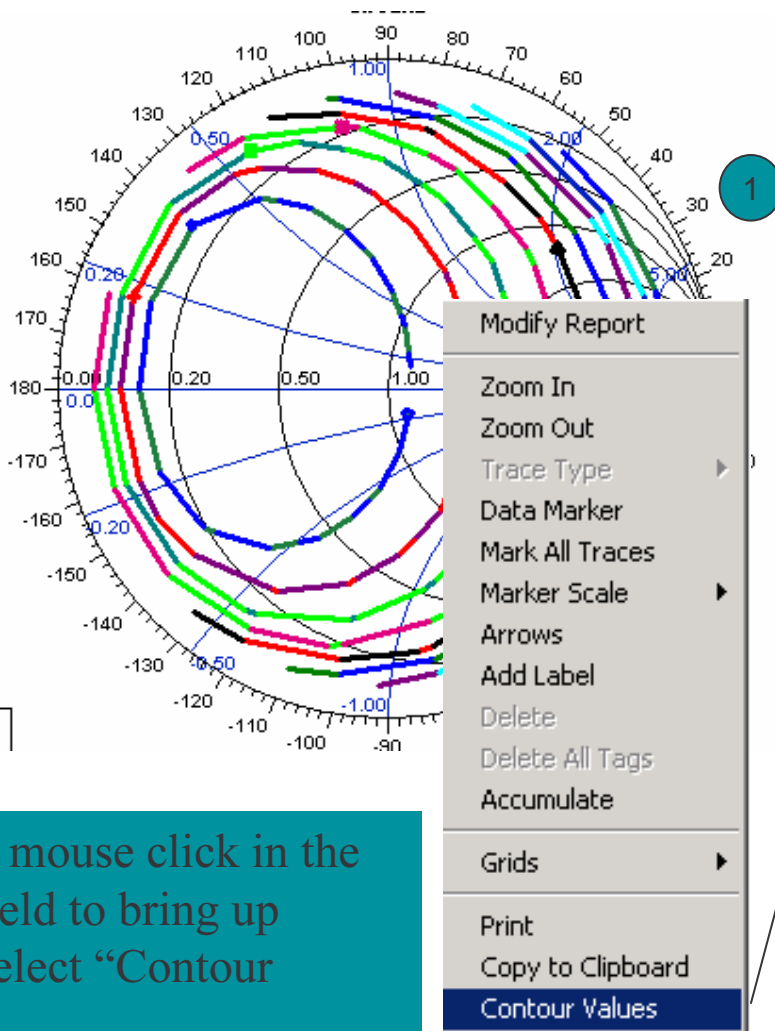


# Other Contour Examples



Multiple plots  
Drain current  
Output power

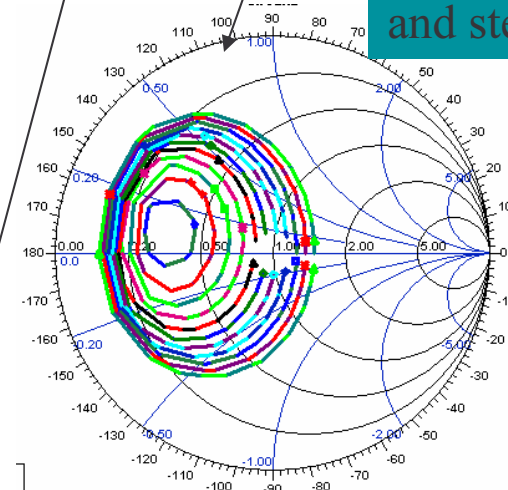
# Changing Contour Values



1. Right mouse click in the report field to bring up menu, select "Contour Values"

A dialog box titled 'Contour Values' with a close button (X) in the top right corner. It contains a 'List of traces:' section with a dropdown menu showing 'dBm(P02<F1>'. To the right, under 'Contour Values:', there are three input fields: 'Min value' with '24', 'Max value' with '26.5', and 'Step value' with '0.25'. Below these fields is a 'Max Z value:' label with '26.6683'. At the bottom are 'Apply' and 'Cancel' buttons. A blue circle with the number '2' is positioned to the right of the 'Max value' field.

2. Set new min, max and step values

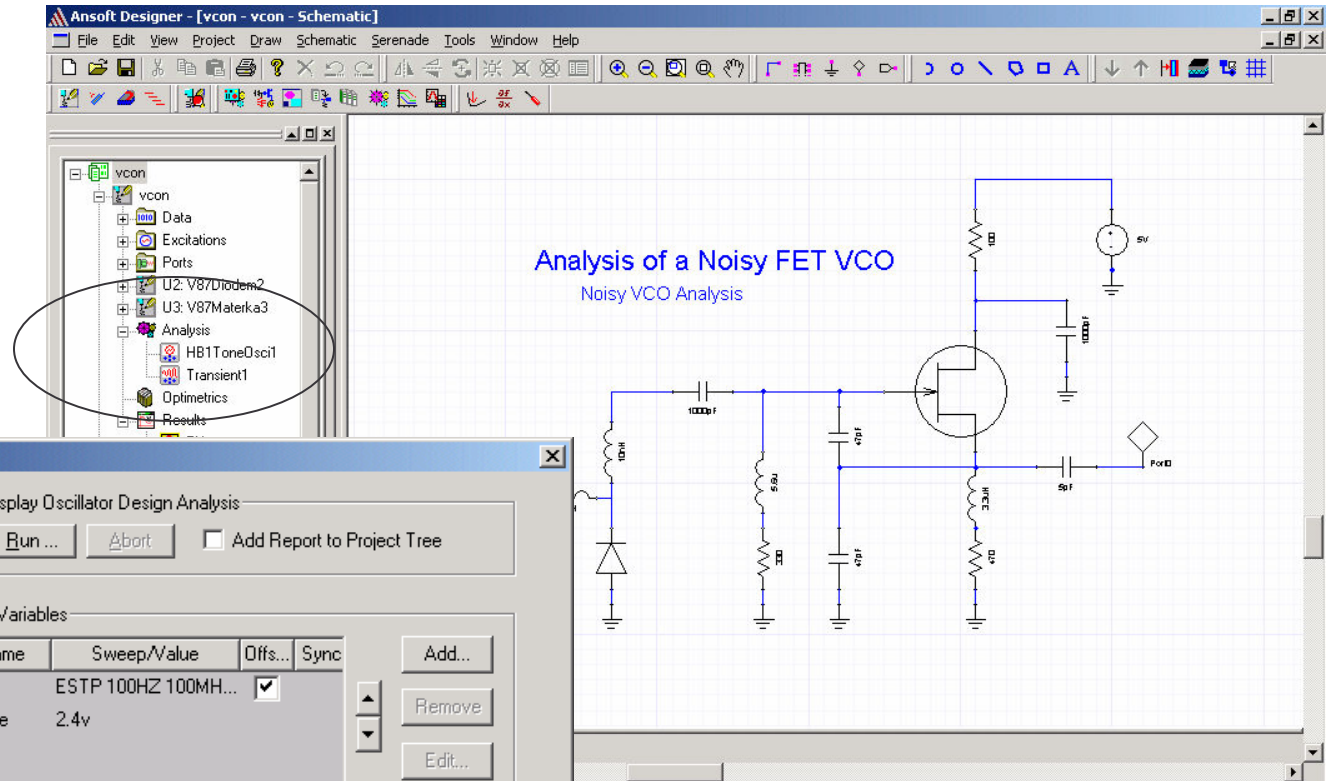


# Oscillator Analysis



# Oscillator Analysis

Multiple analyses  
(ex. HB, Transient,  
Phase noise)  
from single  
Circuit design



Harmonic Balance Oscillator Analysis, 1-Tone

Analysis Setup Option: (Default Options)

Enable Oscillator Design Analysis

No. of Harmonics: 3

Oscillator Search Range:  
Start: 120 MHz  
Stop: 250 MHz

Noise Spectrum:  
 Enable Noise Spectrum Calculations  
Measurement Port No.: 1  
Measurement Harmonic No. (Times of F1): F1

Stability Analysis:  
 Use Solution-Path Tracing

Display Oscillator Design Analysis:  
   Add Report to Project Tree

Sweep Variables:

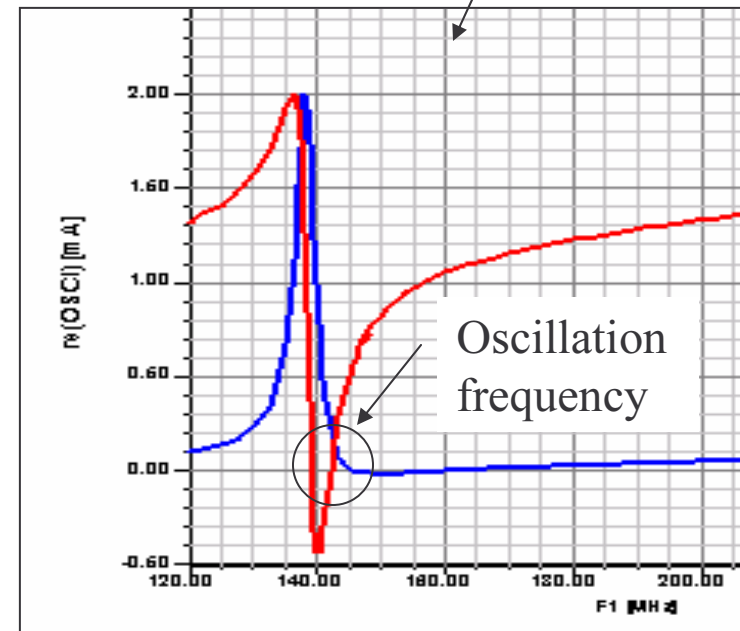
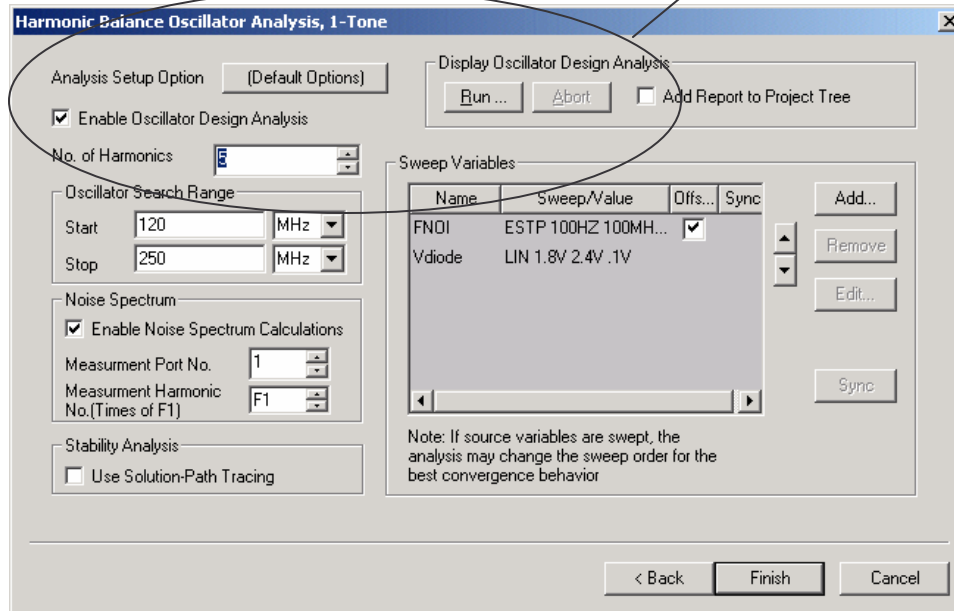
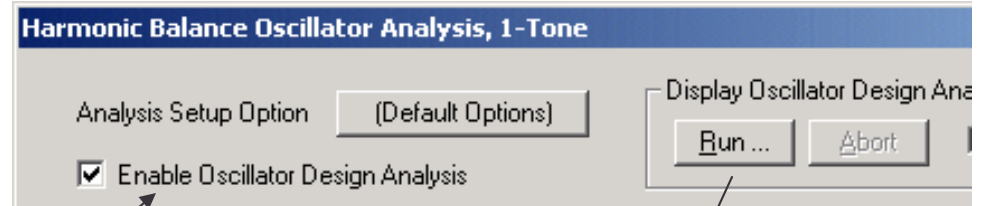
Name	Sweep/Value	Offs...	Sync
FNOI	ESTP 100HZ 100MH...		<input checked="" type="checkbox"/>
Vdiode	2.4v		<input type="checkbox"/>

Note: If source variables are swept, the analysis may change the sweep order for the best convergence behavior

Analysis set-up allows user to define multiple sweep parameters and oscillation search range

# Oscillator Analysis

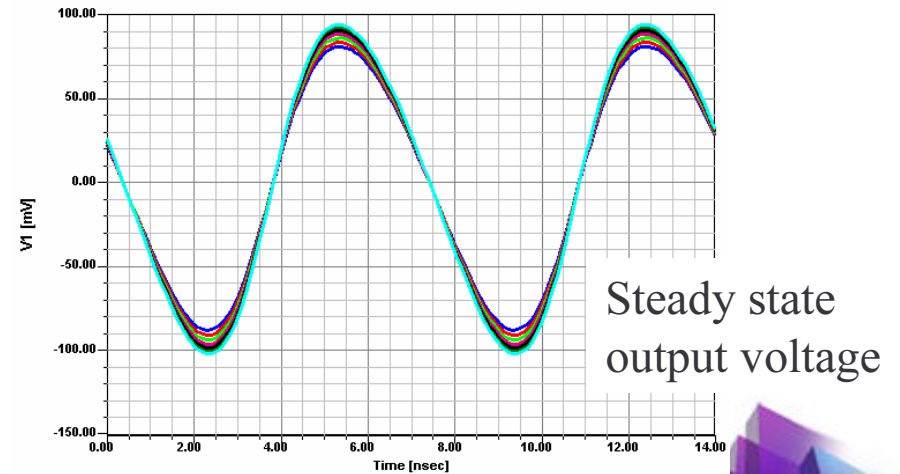
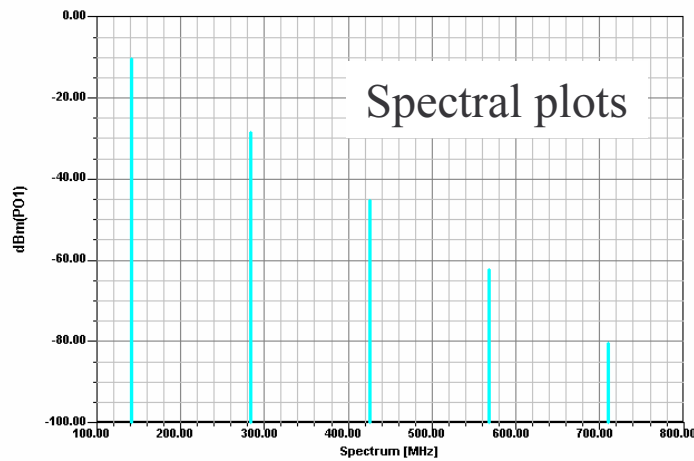
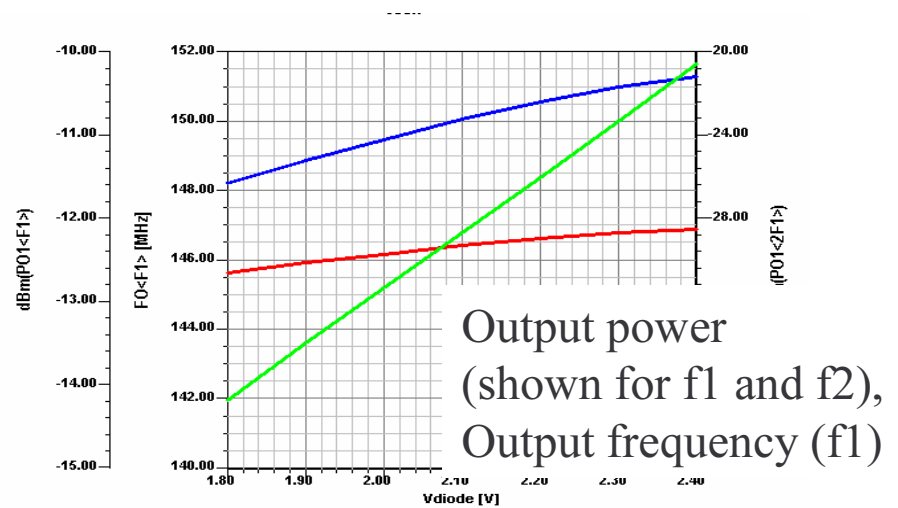
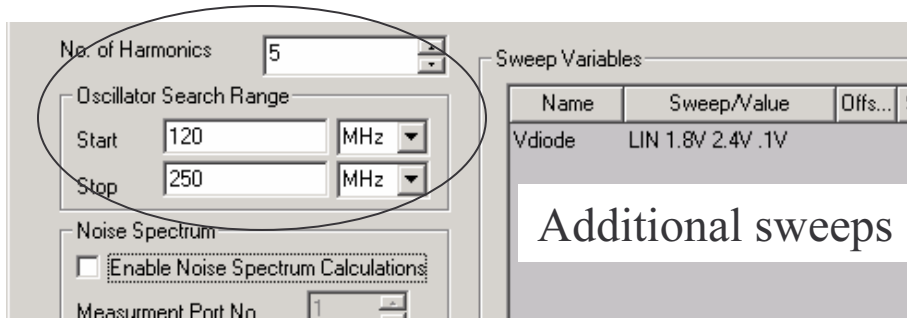
The oscillator design aid sweeps a user specified frequency range and plots complex currents through device. Negative resistance corresponds to potential oscillation frequency.



# Oscillator Analysis

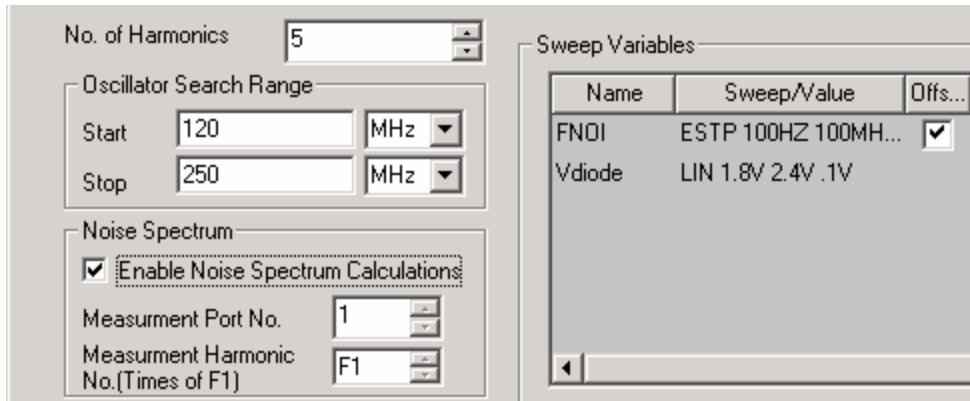
Harmonic Balance Oscillator analysis provides nonlinear circuit performance

Search range and harmonics

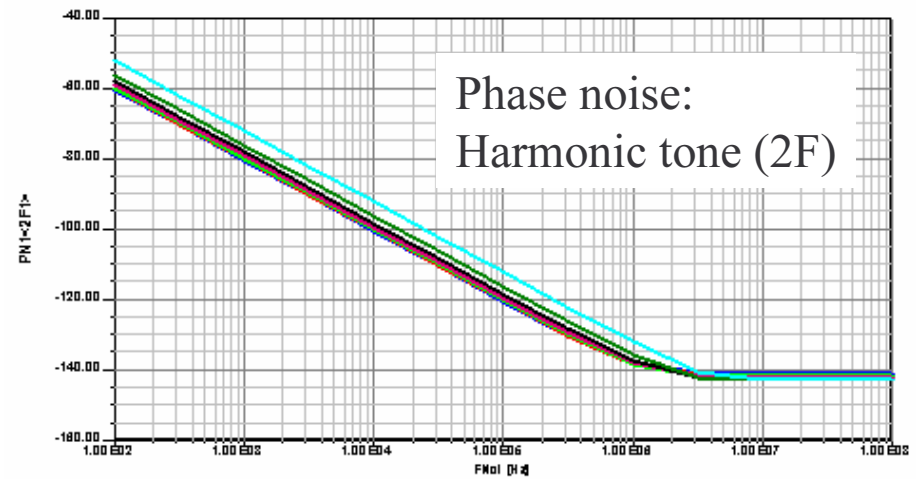
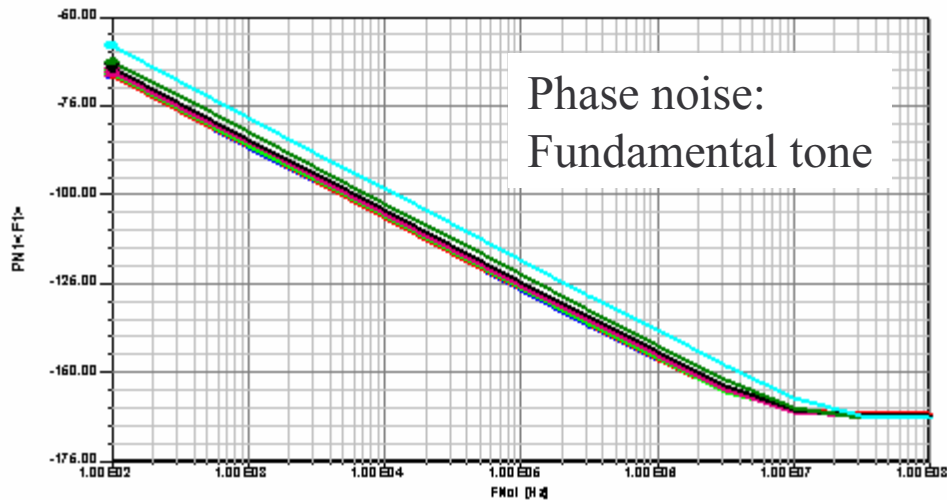




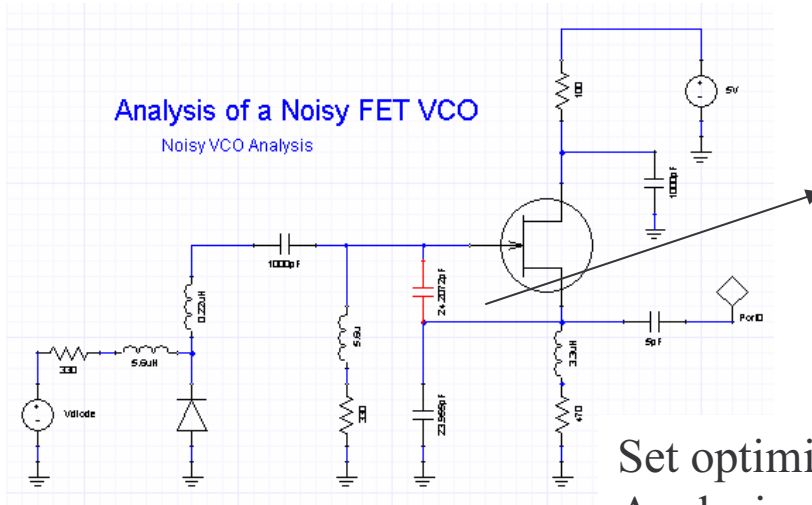
# Nonlinear Noise Analysis



Noise Spectrum analysis provides nonlinear circuit noise spectrum data such as phase noise and amplitude noise



# Optimization of Noise Performance



Properties

Passed Parameters | General | Symbol | Property Displays

Value  Optimization  Tuning  Sensitivity  Statistics

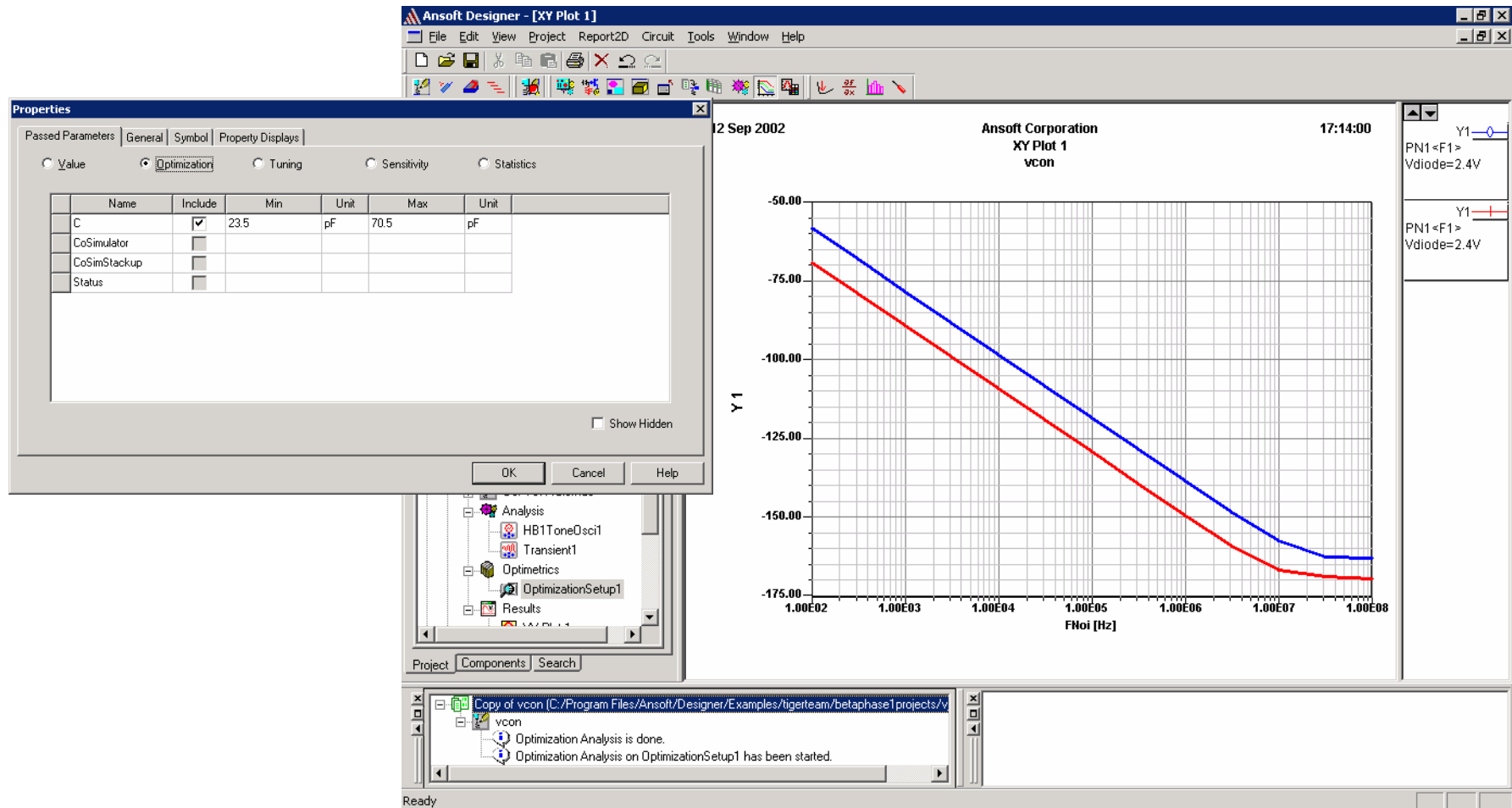
Name	Include	Min	Unit	Max	Unit
C	<input checked="" type="checkbox"/>	23.5	pF	70.5	pF
CoSimulator	<input type="checkbox"/>				
CoSimStackup	<input type="checkbox"/>				
Status	<input type="checkbox"/>				

Show Hidden

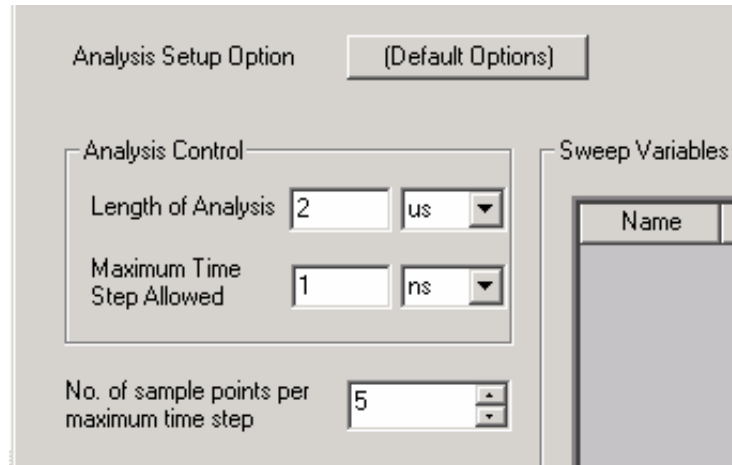
OK Cancel Help

Set optimization, tuning and statistical Analysis parameters/ranges through component properties

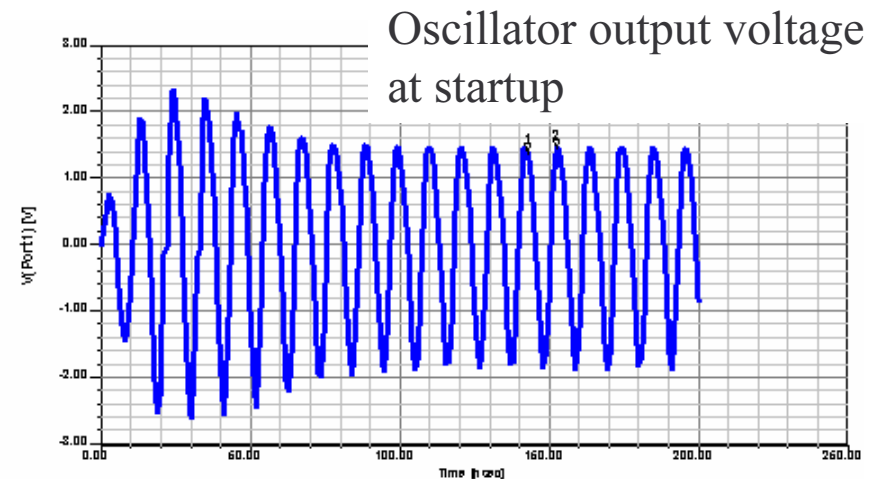
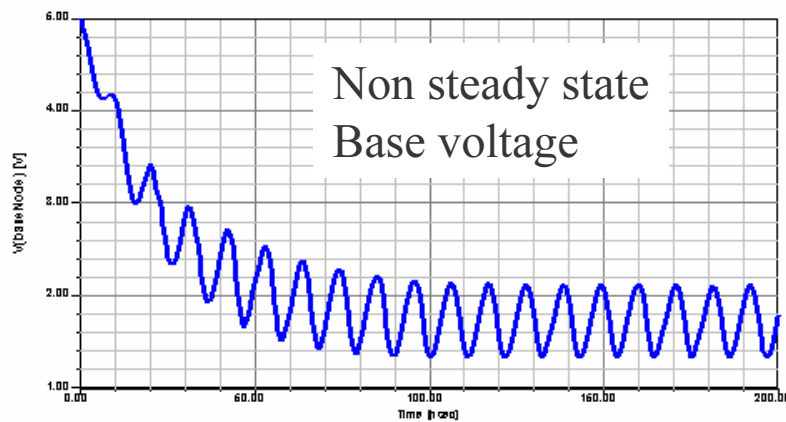
# Optimization of Noise Performance



# Transient Analysis



Transient analysis allows the Circuit designer to investigate non steady state behavior



# Field Solver Design Basics (.avi File)



# **Exercise: Create a Planar EM Design**



# Insert Planar EM Design

Click Right on Project and select Insert Planar EM Design

Choose Layout Technology

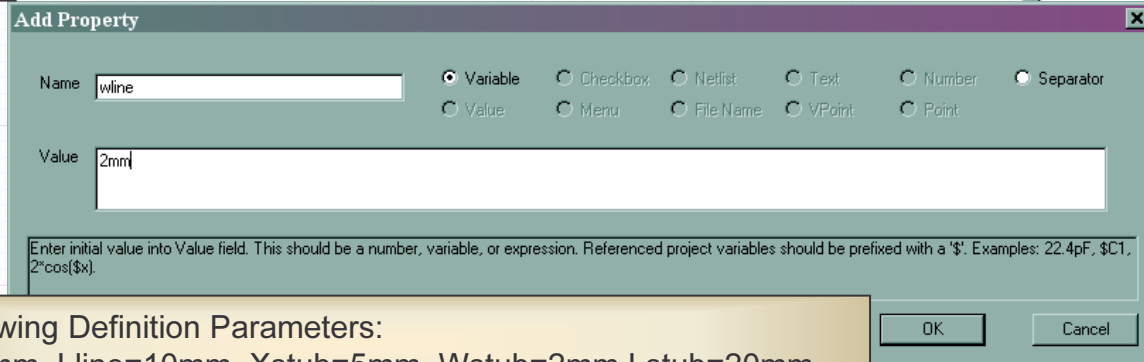
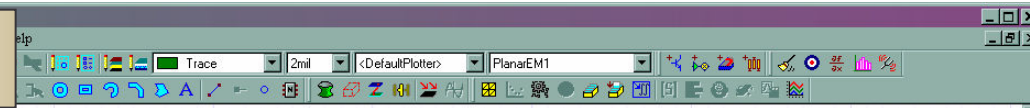
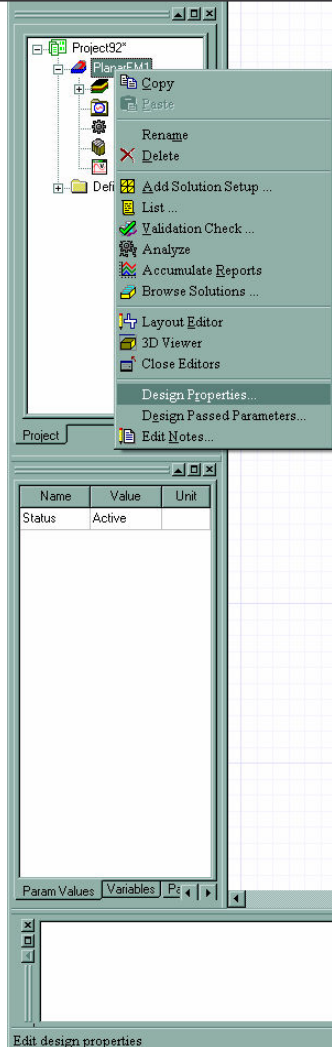
- dateno
- filt1
- FILTER\_Stackup
- FiltreSL
- FoundryLayDef
- FR4\_1\_6mm
- LNALayerDef
- marconi
- MS - Alumina (Er=9.8) 0.010 inch, gold
- MS - Alumina (Er=9.8) 0.025 inch, gold
- MS - FR4 (Er=4.4) 0.030 inch, 0.5 oz copper
- MS - FR4 (Er=4.4) 0.060 inch, 0.5 oz copper**
- MS - RT\_duroid 5880 (Er=2.20) 0.010 inch, 0.5 oz copper
- MS - RT\_duroid 5880 (Er=2.20) 0.020 inch, 0.5 oz copper
- MS - RT\_duroid 6010 (Er=10.2) 0.010 inch, 0.5 oz copper
- MS - RT\_duroid 6010 (Er=10.2) 0.025 inch, 0.5 oz copper
- ms9\_7

Select MS-FR4 (Er=4.4) 0.060inch, 0.5 oz copper

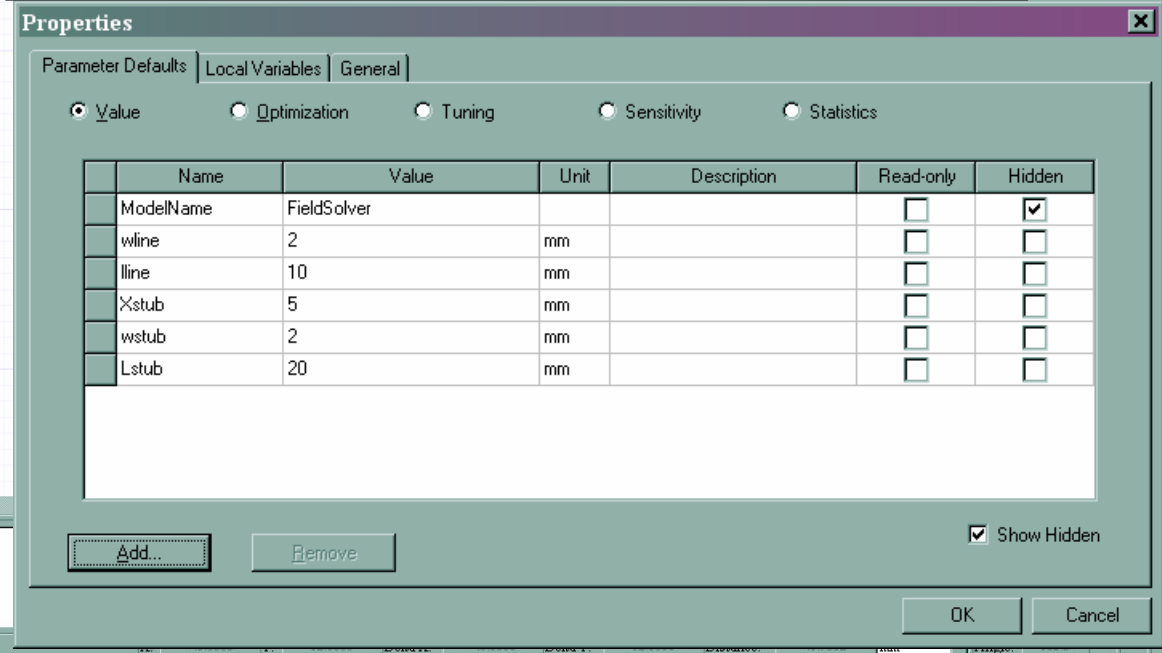
Insert Planar EM Design

# Add Definitions Parameters

Click Right on Planar EM Design and select Design Properties.



Add following Definition Parameters:  
Wline=2mm, Lline=10mm, Xstub=5mm, Wstub=2mm, Lstub=20mm





# Draw A Simple Open Stub

**Select Add Line and Draw two lines as shown**

**Use Definitions Parameters Xstub, Wstub, Lstub to set stub properties**

Name	Value	Unit	Description
Name	line104		
PlacementLayer	Trace		
Net	net_1		
LineWidth	wstub		
BendType	Corner		
CapType	Flat		
TotalLength	20		
Pt0	Xstub , 0		
Pt1	Xstub , lstub		

**Use Definitions Parameters Wline, Lline to set line properties**

Name	Value	Unit	Description
Name	line102		
PlacementLayer	Trace		
Net	net_0		
LineWidth	wline		
BendType	Corner		
CapType	Flat		
TotalLength	10		
Pt0	0 , 0		
Pt1	lline , 0		

Ready X: 24.0000 Y: 6.0000 Delta X: -11.0000 Delta Y: 9.0000 Distance: 14.2127 mm Angle: 140.7

# Add Ports

**Use select Edge to select edge line**

**Draw Port**

**The inserted Port are added to Excitations folder**

**Edge Port Definition allows to select Port options.**

**Click on Select Edge icon and select the left edge of the horizontal line, then click on draw Port icon.**

**Click on Select Edge icon and select the right edge of the horizontal line, then click on draw Port icon.**

Name	Value	Unit
wline	2	mm
lline	10	mm
xstub	5	mm
wstub	2	mm
lstub	20	mm
Status	Active	

Name	Value	Unit
InterfaceP...	Port2	
Direction	AnyDirection	

Name	Value	Unit
Port Name	Port2	
Gap Source	<input checked="" type="checkbox"/>	
Use Port Solver	<input checked="" type="checkbox"/>	
Port Type	Single Strip Gap Source	
Port Excitation		
Magnitude	1mA	
Phase	0deg	
Characteristic Impedance	500hm	
Post Processing Settings		
Post Process Port	<input type="checkbox"/>	
Renormalize	500hm + 0i 0hm	
Deembed	0mm	

# Add setup

Click right on Analysis folder and select Add Solution Setup.

Select Fixed Mesh with frequency = 5GHz.

**Meshing Parameters Tab**  
to set meshing options ;Fixed Mesh, Edge Mesh, Adaptative Mesh, Initial Mesh Frequency.  
**Mesh Refinement tab**  
More Meshing option

Set up mesh for circuit analysis X: -2.0000 Y: 12.0000 Delta X: -19.0000 Delta Y: 11.0000 Distance: 21.9545 mm Angle: 149.9

# Edge Meshing

## Using Edge Meshing

When you define a non-adaptive solution setup, you can instruct Ansoft Designer to add narrow rectangles along the edges of the model. These rectangles efficiently capture electromagnetic effects close to the model edges, resulting in faster solution times and/or higher accuracy.

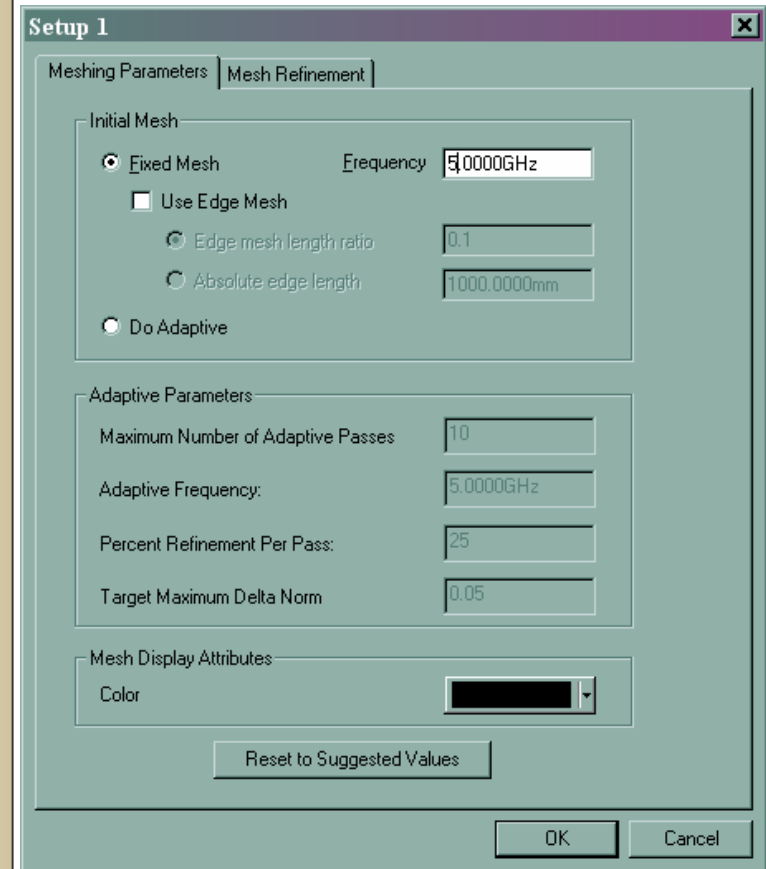
When Ansoft Designer refines the fixed mesh, it determines the length of the rectangles (the longer edges, which lie parallel to the model edge) by making them smaller than a fraction of the guided wavelength at the frequency you specified. You determine the width of the rectangles by either specifying the ratio of the rectangles' length to the width, or the *edge mesh length ratio*, or by specifying the absolute width of the rectangles, or the *absolute edge length*.

To use edge meshing:

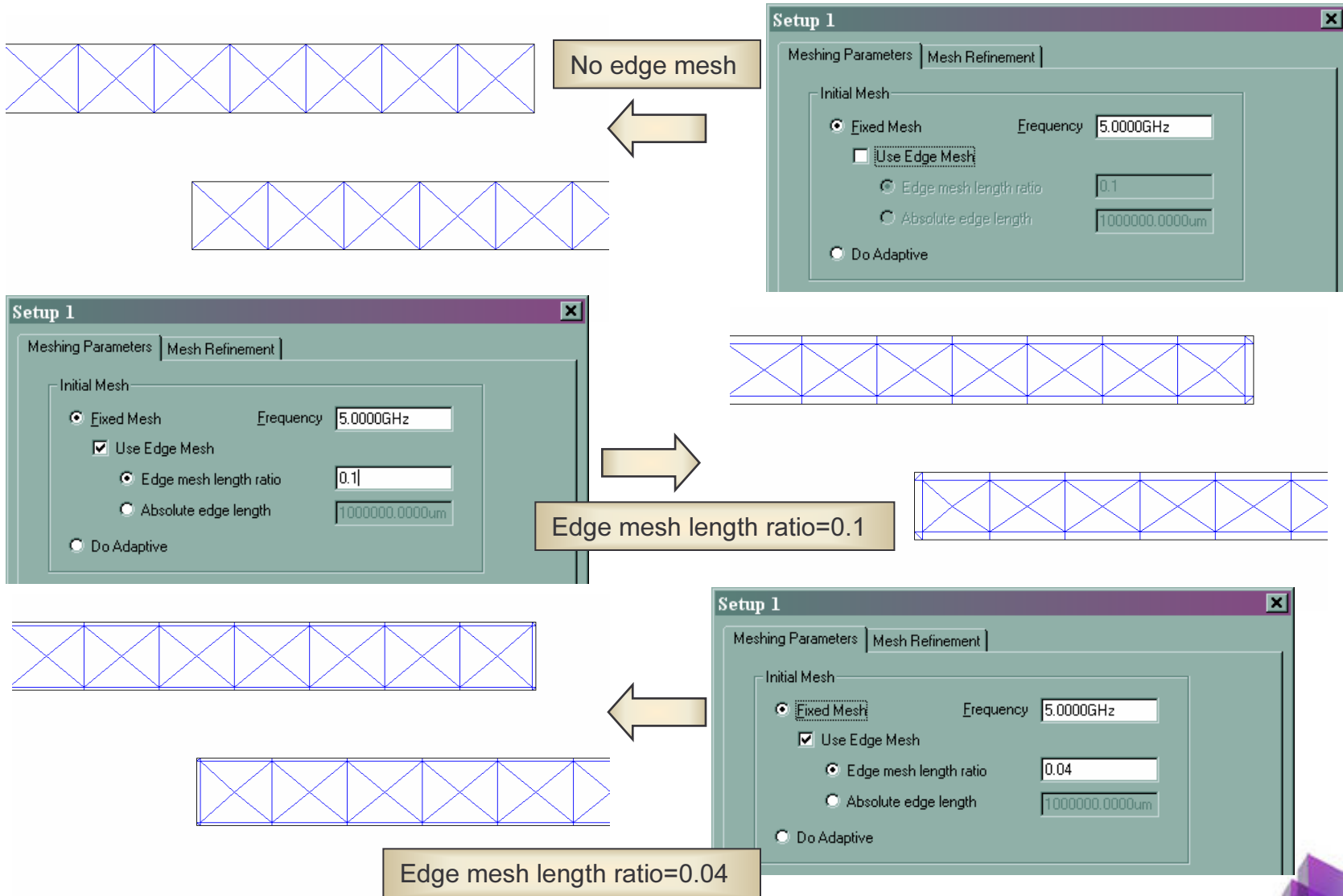
1. In the setup dialog box, click the **Meshing Parameters** tab.
2. Select **Fixed Mesh**, and then specify a frequency at which to generate the fixed mesh in the **Frequency** text box.
3. Select **Use Edge Mesh**.
4. Specify how the edge mesh is created by selecting one of the following:
  - **Edge mesh length ratio**
  - **Absolute edge length**
5. If you selected **Edge mesh length ratio**, type the ratio of the length to the width of the rectangles.

After refinement, the width of the rectangles will be nominally equal to ratio  $\times$  length. The ratio value should be between 0.02 and 0.2 to prevent extremely narrow rectangles and extremely wide triangles.

6. If you selected **Absolute edge length**, type the absolute width of the rectangles, including the model units.



# Example of Edge Meshing



# Mesh Refinement

## Setting Lambda Refinement

Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength. It is recommended and selected by default. To specify the size of wavelength by which Ansoft Designer will refine the mesh:

1. Under the **Mesh Refinement** tab in the solution setup dialog box, select **Lambda Refinement**.
2. Type a value in the **Edge Length Factor** text box. The ratio of the guided wavelength and the length of the longest triangle edge will be greater than or equal to the **Edge Length Factor**.

## Refining for Quality

When **Refine for Quality** is selected in the setup dialog box, Ansoft Designer will produce a mesh with triangles that have approximately the same size angles. The mesh is refined until all triangle angles in the mesh are at least the **Minimum Angle** value.

### To refine for quality:

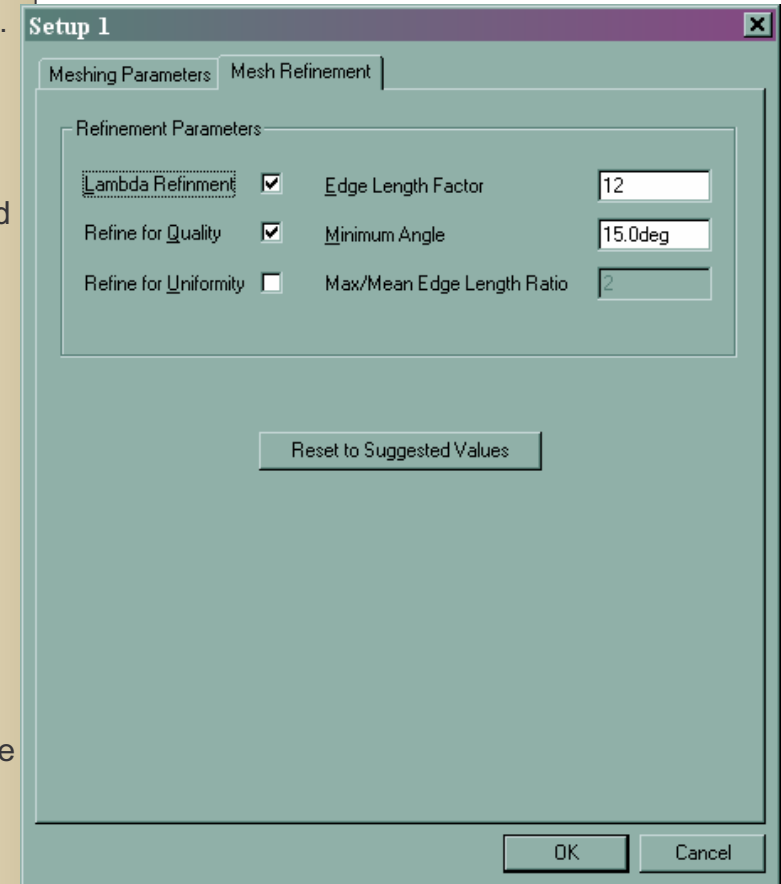
1. Under the **Mesh Refinement** tab in the solution setup dialog box, select **Refine for Quality**.
2. Type a value in the **Minimum Angle** text box in degrees.

## Refining for Uniformity

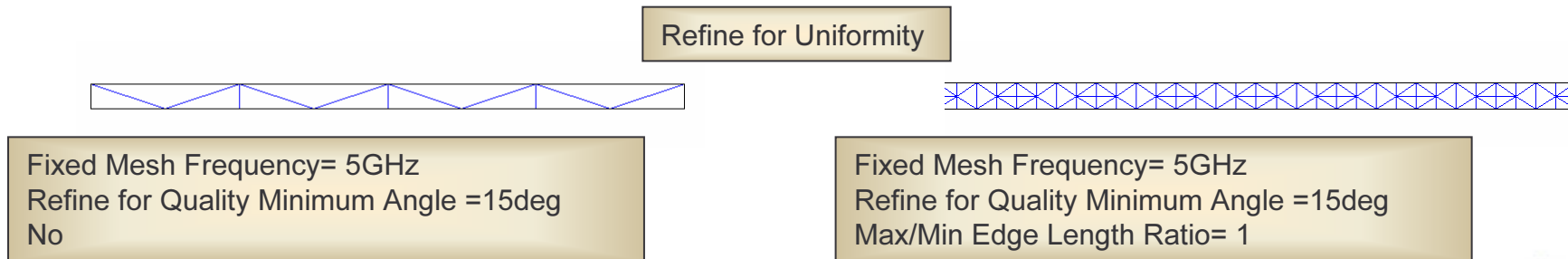
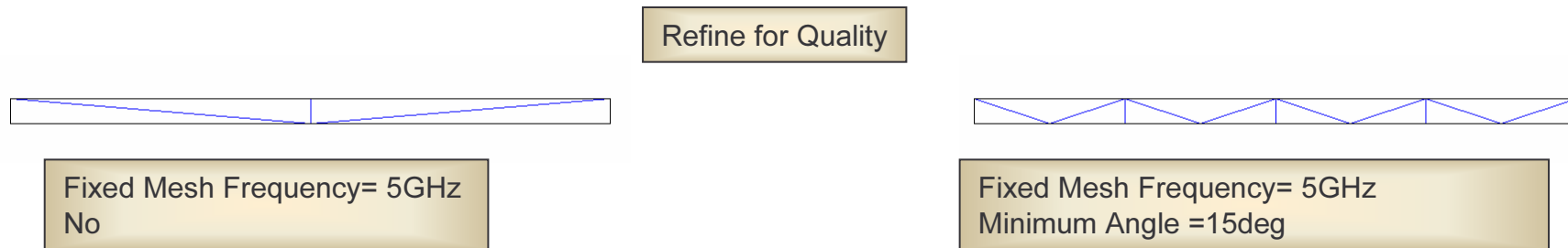
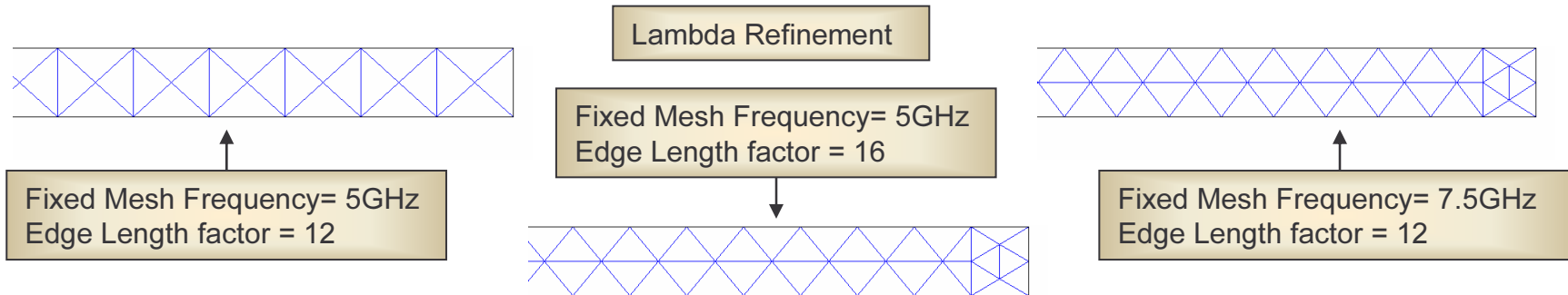
When **Refine for Uniformity** is selected in the setup dialog box, Ansoft Designer will produce a mesh with triangles that are approximately the same size. The ratio of the maximum length of any triangle edge to the average length of the triangle edges in the mesh will not be greater than **Max/Mean Edge Length Ratio** value, resulting in greater uniformity of the mesh.

To refine for uniformity:

1. Under the **Mesh Refinement** tab in the solution setup dialog box, select **Refine for Uniformity**.
2. Type a value in the **Max/Mean Edge Length Ratio** text box.



# Example of Mesh refinement



# Add Frequency Sweep

The screenshot shows the Ansoft Designer interface with a project named 'Project92 - PlanarEM1 - Layout'. A green meshed T-shaped structure is visible in the center, with two ports labeled 'Port1' and 'Port2'. A context menu is open over the 'Setup' object in the project tree, with 'Add Frequency Sweep...' selected. The 'Sweep 1' dialog box is open, showing the following settings:

- Type:  Interpolating Fast
- Specify Frequency Sweep:  Linear step
- Start: 0.1 GHz
- Stop: 5 GHz
- Step: 0.01 GHz

The 'Sweep Description' field contains: 'Linear Step from 0.1GHz to 5GHz, step=0.01GHz'. The 'View Sweep Points List...' button is highlighted.

Annotations in the image provide the following instructions:

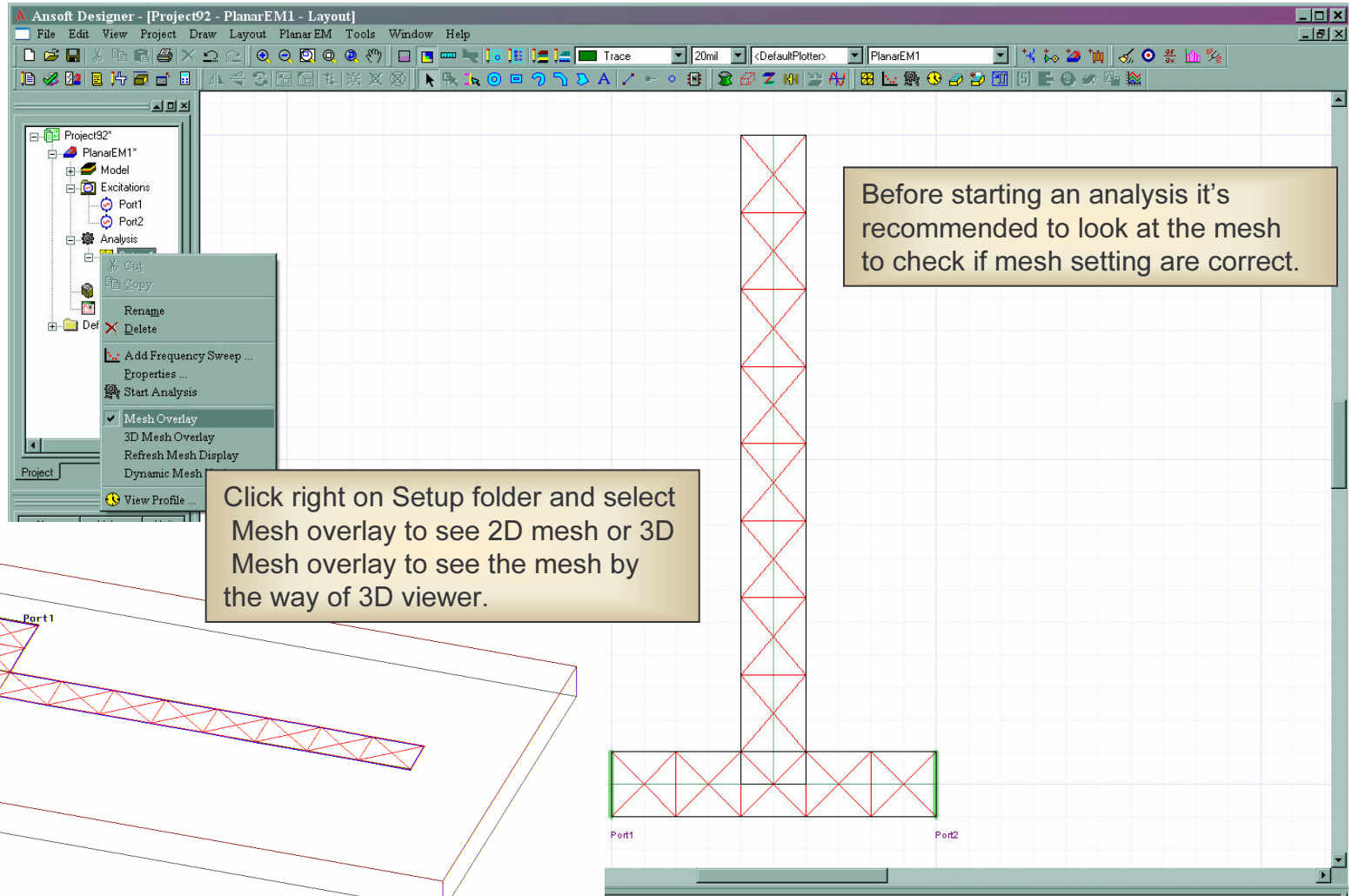
- Click Right on the Setup just created and select Add Frequency Sweep..
- Select Interpolating Fast  
Remove the existing sweep.  
Select Linear Step and enter  
Start 0.1GHz Stop 5GHz Step 0.01GHz  
click Add.  
Click on View Sweep Point List to see  
all frequency points.  
Click OK
- You can select multiple type sweep or single value,  
Discrete sweep or Interpolating Fast Sweep.

Name	Value	Unit
Setup	Setup 1	

Set up frequency sweeps for Planar EM analysis X: Y: Delta X: Delta Y: Distance: mm Angle:



# View Mesh



# Run Analysis

The screenshot displays the Ansoft Designer software interface. The main window shows a 2D layout of a planar EM structure with a grid. A 'Solution' dialog box is open, showing simulation parameters and a table of analysis results. Two callout boxes provide instructions: 'Click on icon start analysis' points to a green play button in the toolbar, and 'Click on icon view profile to open this window' points to a profile icon in the toolbar. The 'Solution' dialog box has tabs for 'Matrix', 'Convergence', and 'Profile'. The 'Profile' tab is active, showing a table with columns for Task, Real Time, CPU Time, Memory, and Information. The table lists tasks for 'PlanarEM1' and 'Fast Frequency Sweep' with their respective times and resource usage. A 'Close' button is at the bottom of the dialog. The status bar at the bottom shows coordinates and dimensions.

Click on icon start analysis

Click on icon view profile to open this window

Task	Real Time	CPU Time	Memory	Information
PlanarEM1				03/03/2003 11:11:49; Host: PCALAIN; Process...
Fast Frequency Sweep				
0.100000	00:00:02	00:00:01	71664 K	66 Unknowns; 52 Triangles
5.000000	00:00:01	00:00:01	71664 K	66 Unknowns; 52 Triangles
2.501000	00:00:02	00:00:01	71664 K	66 Unknowns; 52 Triangles
2.403000	00:00:02	00:00:01	71664 K	66 Unknowns; 52 Triangles
1.276000	00:00:01	00:00:01	71664 K	66 Unknowns; 52 Triangles
Total	00:00:08	00:00:05		

Port1 Port2

# Create results: DB S11&S21

The screenshot displays the Ansoft Designer interface for a project named 'Project92 - PlanarEM1 - Layout'. The main window shows a plot titled 'Ansoft Corporation XY Plot 1 PlanarEM1' with the date '03 Mar 2003' and time '11:16:43'. The plot shows two curves: a blue curve for  $\text{dB}(S(\text{Port1}, \text{Port1}))$  and a red curve for  $\text{dB}(S(\text{Port2}, \text{Port1}))$ . The x-axis is labeled 'F [GHz]' and ranges from 1.00 to 5.00. The y-axis is labeled 'Y1' and ranges from 0.00 to -10.00. A legend on the right indicates that the blue line represents  $\text{dB}(S(\text{Port1}, \text{Port1}))$  and the red line represents  $\text{dB}(S(\text{Port2}, \text{Port1}))$ .

The 'Traces' dialog box is open, showing the following configuration:

X	Y	Y-axis
1 F	$\text{dB}(S(\text{Port1}, \text{Port1}))$	Y1
2 F	$\text{dB}(S(\text{Port2}, \text{Port1}))$	Y1

The 'Create Report' dialog box is also open, with the following settings:

- Target Design: PlanarEM1
- Report Type: Standard
- Display Type: Rectangular Plot

The 'Create Report' dialog box has 'OK' and 'Cancel' buttons. The 'Traces' dialog box has 'Add Trace', 'Add Blank Trace', 'Replace Trace', 'Remove Trace', and 'Remove All Traces' buttons. The 'Create Report' dialog box has 'OK' and 'Cancel' buttons. The 'Traces' dialog box also has a 'Get Terminations...' button and 'Apply', 'Done', and 'Cancel' buttons at the bottom.

# Optional Exercise: Planar EM Co-Simulation



ANSOFT CORPORATION

# Edit The Symbol and Modify It

The screenshot shows the Ansoft Designer (Beta) Symbol Editor interface. The main workspace contains a circuit diagram with two ports labeled "Port1" and "Port2" connected to a central component labeled "Planar EM OpenStub". The component is represented by a rectangular symbol with a vertical line extending upwards. The interface includes a menu bar (File, Edit, View, Project, Draw, Symbol, Planar EM, Tools, Window, Help), a toolbar with various drawing and editing tools, and a project tree on the left side. A "Desktop" dialog box is open in the foreground, asking "Save changes?" with "Yes" and "No" buttons. Three callout boxes provide instructions: one pointing to the drawing tools in the toolbar, one pointing to the close button in the window title bar, and one pointing to the "Yes" button in the "Save changes?" dialog.

You can draw arc, circle, line, polygon, Rectangle and add text.

Click the cross to close the Symbol editor widow

Save the symbol

# Copy and Paste the Circuit Design

**Click right on LPF and select Paste** ②

**Symbol of the pasted sub-circuit appears in the schematic** ⑤

**Click right on OpenStub and select Copy** ①

**You can choice to use the same stackup as Parent circuit or to insert The sub-circuit as black box** ③

**Incorporate selected the Merge layers window allows to merge sub-circuit and parent circuit layers** ④

**Merge Layers**

M Trace	Trace
M Dielectric	Dielectric
M Ground	Ground

Flip Source Layers

Merge Layers

Cancel

**Synchronize Design**

Incorporate  
Design is copied, same stackup as parent, manufacturable with parent  
New design: OpenStub

Keep independent (black box)  
Design is linked to original, unrelated stackups, not manufacturable with parent

OK  
Cancel

Click to position center

# Connect the Sub-Circuit Symbol

The screenshot displays the Ansoft Designer interface for a project named 'LPFProject'. The main workspace shows a circuit schematic with two ports, Port1 and Port2, connected through a series of components including waveguides, a central inductor (0.45634nH), and two parallel branches. Each branch contains a waveguide, a parallel inductor, and a capacitor. The design tree on the left shows a 'Sub-circuit' folder, which is highlighted by a blue callout box with the text 'Sub-circuit folder appears in the circuit Design tree'. Below the design tree is a table of parameters for the sub-circuit, with a blue callout box pointing to it that says 'You can change the values of the sub-circuit parameters'. The table lists parameters such as 'wline', 'iline', 'xstub', 'wstub', 'lstub', and 'Status'. The status is 'Active'. At the bottom of the window, there is a log showing simulation results for 'OpenStub'.

Sub-circuit folder appears in the circuit Design tree

Name	Value	Unit
wline	2	mm
iline	10	mm
xstub	5	mm
wstub	2	mm
lstub	20	mm
Status	Active	

You can change the values of the sub-circuit parameters

OpenStub Training (F:\Ansoft\Ansoft Designer\Monday14102002Beta\project\)  
OpenStub  
Planar EM simulation complete.  
A fast frequency sweep with 50 points has been started. (63 unknowns)  
A discrete frequency sweep with 50 point(s) has been started. (63 unknowns)

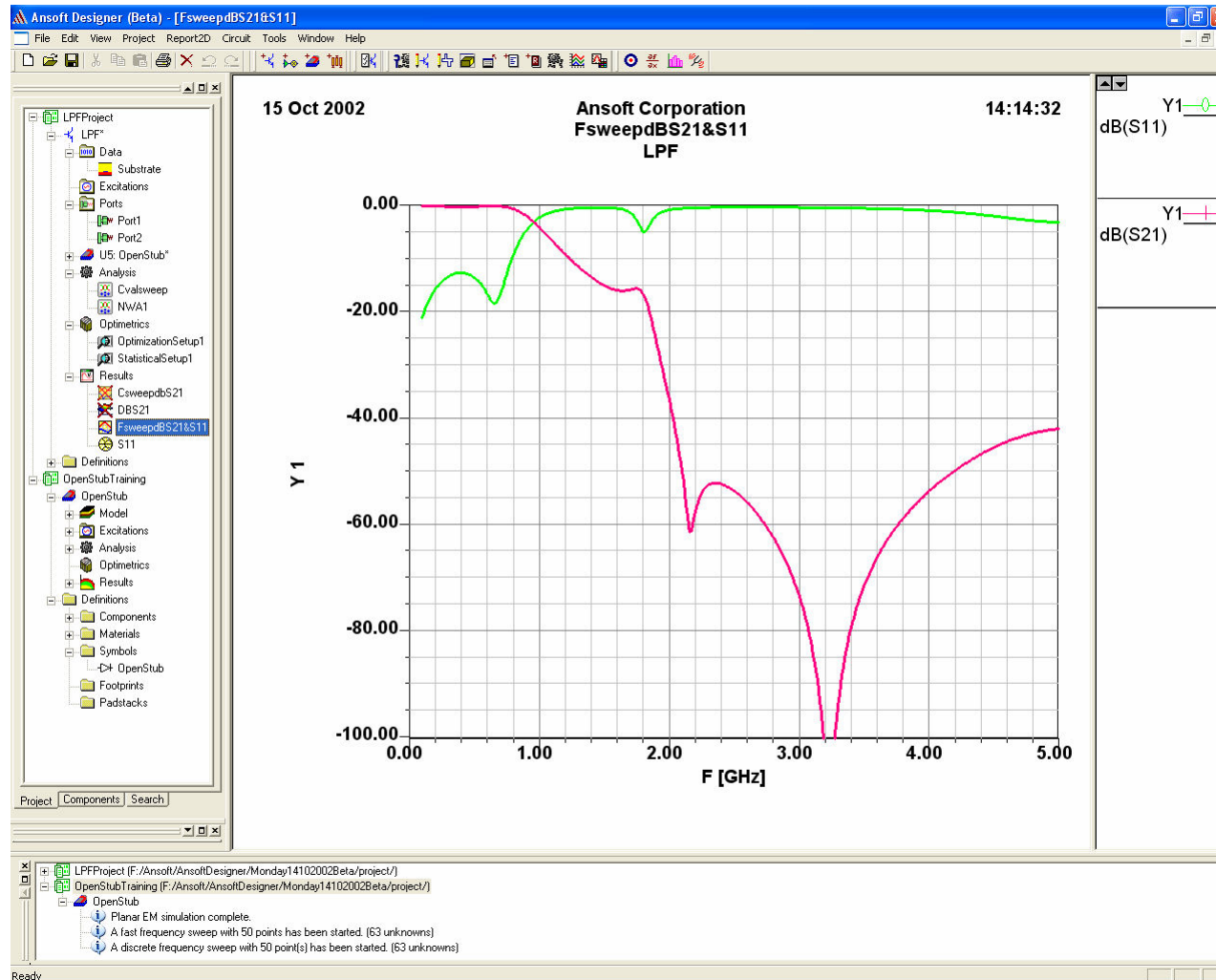
# Run Analysis

The screenshot displays the Ansoft Designer (Beta) interface for a project named 'LPFProject - LPF - Schematic'. The main workspace shows a circuit schematic with components like 'Port1', 'Port2', 'W1=wire', 'W2=wire', 'W3=wire', 'W=wire', 'P=wire', '0.45634nH', 'Voltage Probe', and 'Current Probe'. A context menu is open over the 'Analysis' folder in the left-hand tree, with 'Analyze NWA1' selected. A 'Progress' dialog box is overlaid on the schematic, showing two simulation tasks: 'LPF on Local Machine - RUNNING' and 'OpenStub : Fast Frequency Sweep on Local Machine - RUNNING'. The second task includes a progress bar and the text '5.000000 GHz MoM Matrix Fill'. At the bottom of the window, a status bar indicates: 'Analyze control block', '1', 'New P', and a list of simulation results: 'Planar EM simulation complete.', 'A fast frequency sweep with 50 points has been started. (63 unknowns)', and 'A discrete frequency sweep with 50 point(s) has been started. (63 unknowns)'.

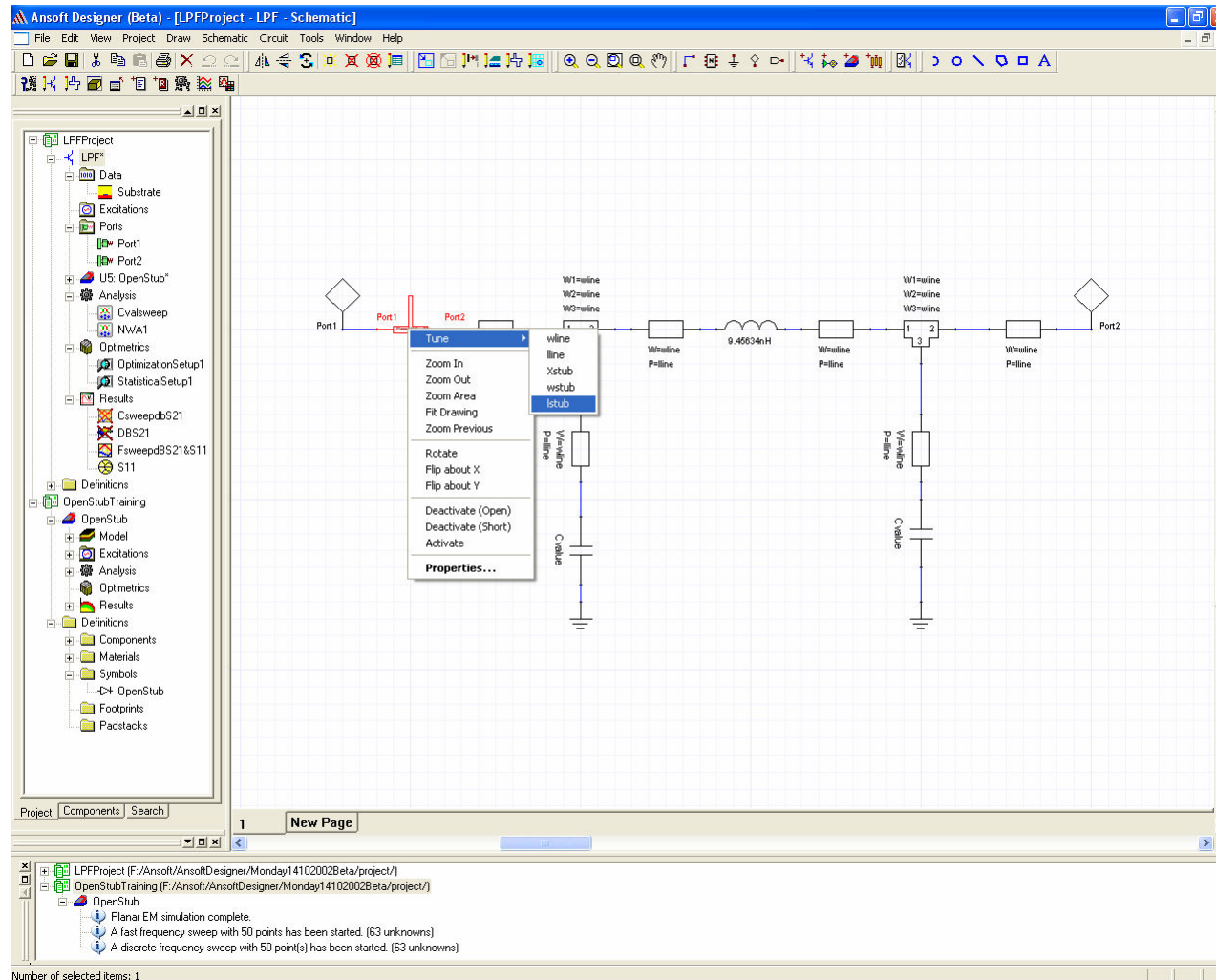
Ansoft Designer parse the Netlist and identify a Field Solver EM  
An EM simulation is automatically running



# Create Results: DBS21&S11



# Define Lstub as Tunable



# Run Tuning

The screenshot displays the Ansoft Designer (Beta) interface for a project named 'LPFProject - LPF - Schematic'. The main workspace shows a schematic diagram of a Low Pass Filter (LPF) circuit. The circuit includes two ports, Port1 and Port2, connected to a series of components: a W-line, a P-line, a series combination of W1=uline, W2=uline, and W3=uline, a central inductor labeled '0.45634nH', another series combination of W1=uline, W2=uline, and W3=uline, and finally another W-line and P-line leading to Port2. Two vertical branches are connected to the central W-line sections, each containing a 'Variable P-line' and a 'Capacitor'.

A context menu is open over the 'Optimetrics' component in the left-hand project tree. The menu options are: Copy, Add, Tuning..., and View Analysis Result... The 'Tuning...' option is highlighted.

The bottom status bar shows the following messages:

- Planar EM simulation complete.
- A fast frequency sweep with 50 points has been started. (63 unknowns)
- A discrete frequency sweep with 50 point(s) has been started. (63 unknowns)

At the bottom left of the status bar, the text reads: "Start tuning analysis on the design."

# Run Accumulate Sweep Tuning

**15 Oct 2002** **Ansoft Corporation** **14:22:42**  
**FsweepdB\_S21&S11**  
**LPF**

Y1  
dB(S11)  
Y1  
dB(S21)  
Y1  
dB(S11)  
Istub:45=10mm  
Y1  
dB(S21)  
Istub:45=10mm

0.00  
-20.00  
-40.00  
-60.00

F [GHz]

**Progress**  
LPF Tuning on Local Machine - RUNNING  
Solving: //LPF:45/Istub=12mm/Cvalue=4.64273pF  
LPF on Local Machine - RUNNING  
Perform EM Simulation  
OpenStub : Fast Frequency Sweep on Local Machine - RUNNING  
5.000000 GHz MoM Matrix Fill

**Tune - LPF**  
 Real Time  
 Accumulate  
Sim. Setups: Tune  
NWA1  
Cvalsweep  
Istub (L) [mm]: 30  
Step: 2  
10

Project | Components | Search

LPFProject (F:/Ansoft/AnsoftDesigner/Monday14102002beta/project/)  
OpenStubTraining (F:/Ansoft/AnsoftDesigner/Monday14102002beta/project/)  
OpenStub  
Planar EM simulation complete.  
A fast frequency sweep with 50 points has been started. (63 unknowns)  
A discrete frequency sweep with 50 point(s) has been started. (63 unknowns)

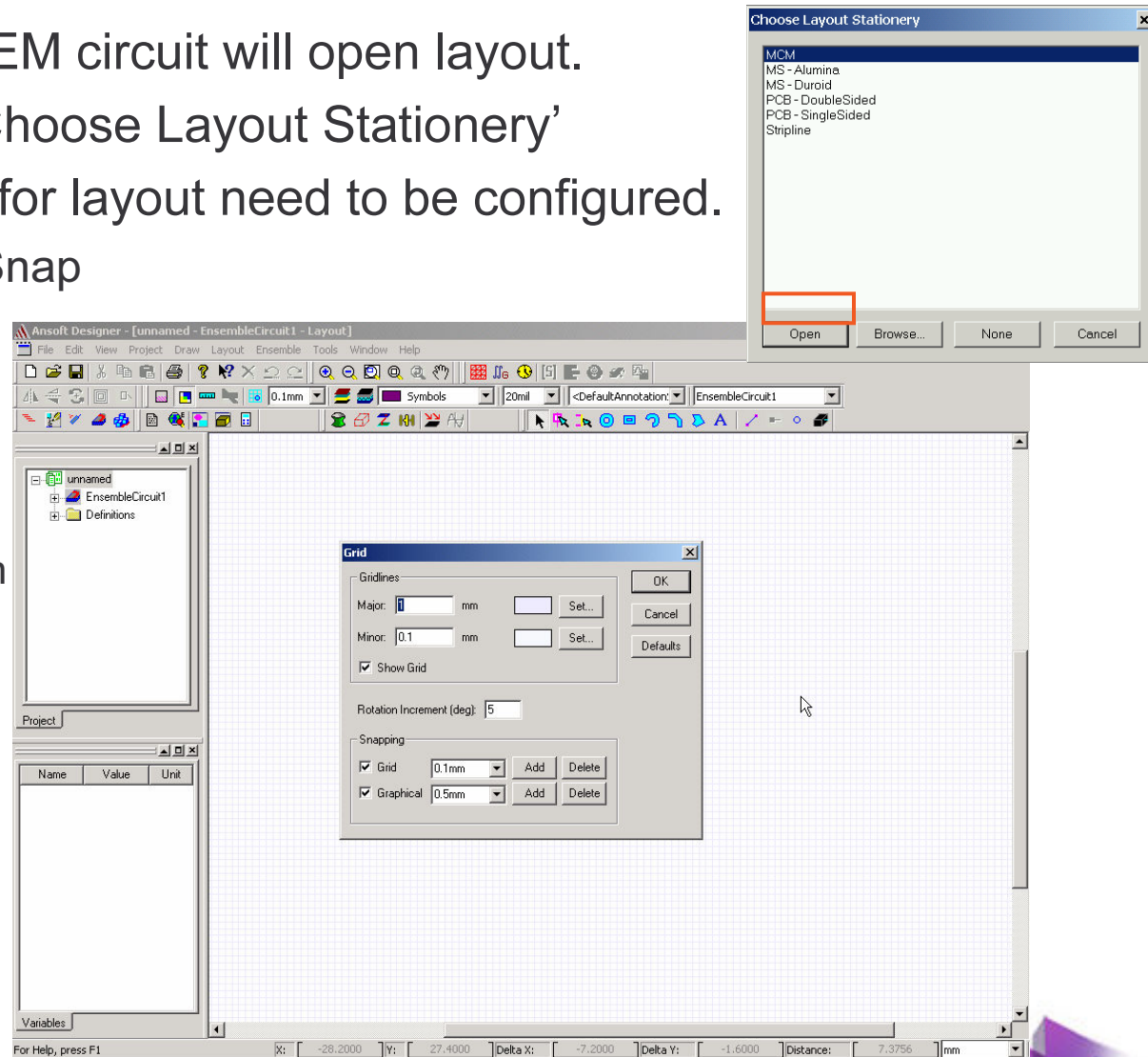
Ready

# Optional Exercise: Planar EM Antenna Design





# Configure Layout

- ◆ Creating an Planar EM circuit will open layout.
- ◆ Click on 'None' in 'Choose Layout Stationery'
- ◆ The grid and snaps for layout need to be configured.
  - ◆ Layout -> Grid & Snap
  - ◆ Set:
    - Major = 1
    - Minor = 0.1
    - Grid = 0.1mm
    - Graphical = 0.5mm
  - ◆ OK




# Insert Layers

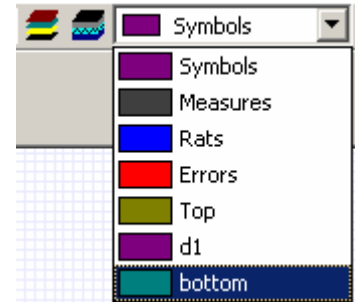
- ◆ Open the Stackup Editor in one of three ways:
  - ◆ Layout -> Layers. Select Stackup tab.
  - ◆ Click  icon. Select Stackup tab.
  - ◆ Click  icon.
- ◆ Insert an infinite ground layer
  - ◆ Add Layer -> Name = “top” -> Type = “signal”
- ◆ Insert a dielectric layer
  - ◆ Add Layer -> Name = “d1” -> Type = “dielectric”
- ◆ Insert a trace layer
  - ◆ Add Layer -> Name = “bottom” -> Type = “signal”

	Name	Type	Material	Drag Mode	Thickness	Lower Elevation	Upper Elevation	Roughness
—	Top	signal	copper	middle align	0mm	4mm	4mm	0mm
▨	d1	dielectric	air		4mm	0mm	4mm	
—	bottom	signal	copper	middle align	0mm	0mm	0mm	0mm

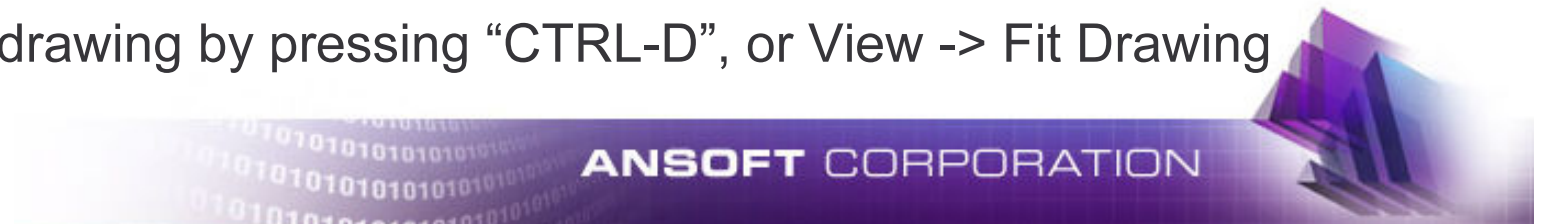
- ◆ Change the “d1” thickness to 4 mm
- ◆ Layers can be moved by dragging and dropping the entire row..
- ◆ Material attributes can be defined by selecting the material button

# Drawing the model (1)

- ◆ Change the active layer to “bottom”
  - ◆ Select the layer pull-down menu and highlight “bottom”
- ◆ Begin drawing a rectangle object in one of two ways:
  - ◆ Draw -> Primitive -> Rectangle
  - ◆ Select the  icon.
- ◆ Enter the lower left-hand coordinates for the rectangle.
  - ◆ In the Status Bar, enter X=0.0, Y=0.0. Use the TAB key to move between entries and press ENTER when finished.




- ◆ Finish drawing the rectangle
  - ◆ In the Status Bar enter Delta X = 40, Delta Y = 46.8.
  - ◆ OR, drag the upper righthand corner until Delta X:40, Delta Y:46.8
- ◆ Fit the drawing by pressing “CTRL-D”, or View -> Fit Drawing



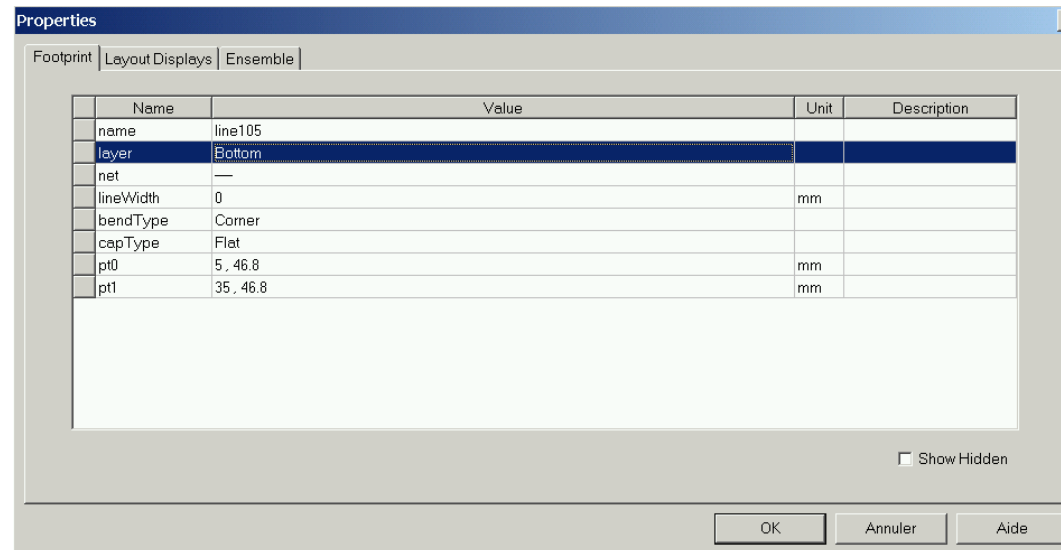


# Drawing the model (2)

- ♦ We will now define a shortcut to the PIFA in order to reduce the physical dimension to a quarter of the resonant wavelength.
- ♦ Select the  icon
- ♦ Enter the lower left-hand coordinates for the line.
  - ♦ In the Status Bar, enter X=5, Y=46.8
  - ♦  $\Delta X = 30$  ,  $\Delta Y = 0$

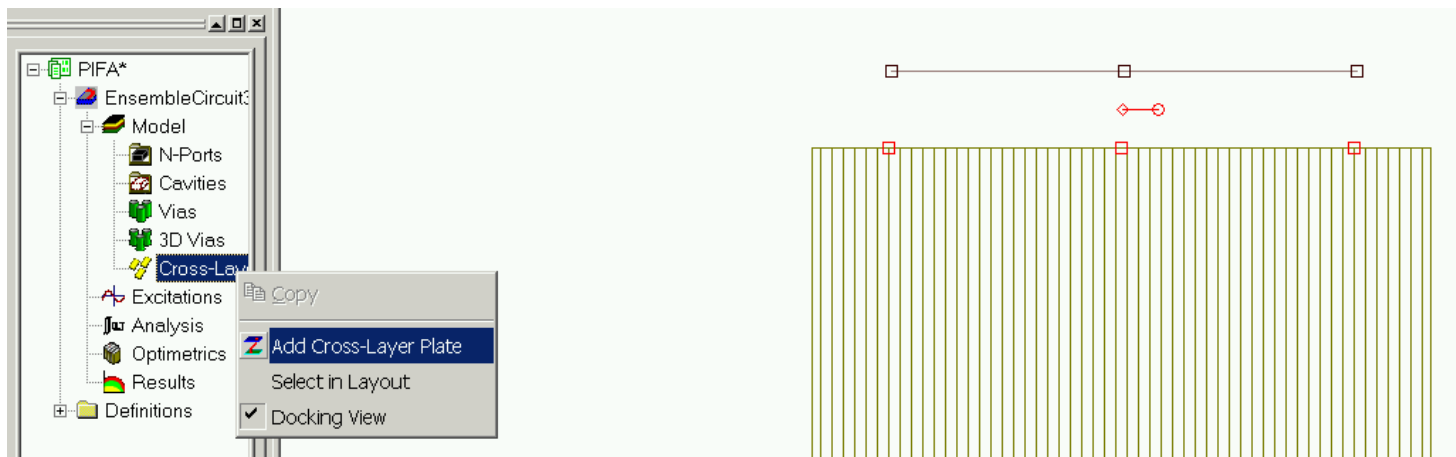
X:	35.0000	Y:	46.8000	Delta X:	30.0000	Delta Y:	0.0000
----	---------	----	---------	----------	---------	----------	--------

- ♦ Use the TAB key to move between entries and press ENTER when finished.
- ♦ Verify the following properties



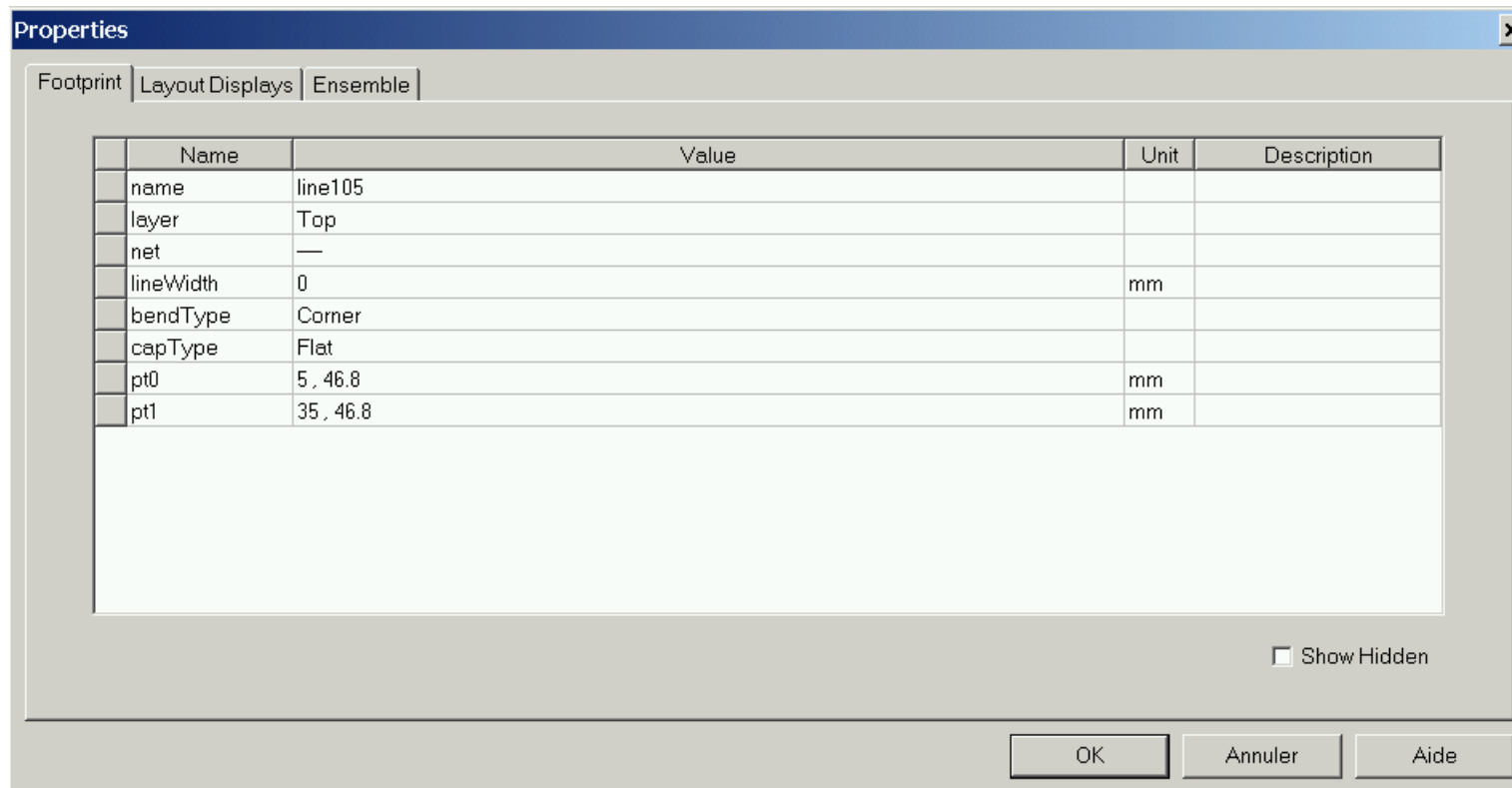
# Drawing the model (3)

- Select the line and Copy it (CTRL+C command)
- In the layer, choose Top
- Paste the line (CTRL-V)
- Select the 2 lines by maintaining the CTRL button when you select them.
- In the menu on the left, click on Cross-Layer and Add Cross-Layer Plate




# Drawing the model (4)

- ◆ Change the active layer to “Top”
  - ◆ Select the layer pull-down menu and highlight “Top”
  - ◆ Double Click on the line and adjust the properties




# Drawing the model (5)

- ◆ Begin drawing a rectangle object in one of two ways:
  - ◆ Draw -> Primitive -> Rectangle
  - ◆ Select the  icon.
- ◆ Enter the lower left-hand coordinates for the rectangle.
  - ◆ In the Status Bar, enter X = 5, Y = 46.8. Use the TAB key to move between entries and press ENTER when finished.

X:	30.0000	Y:	46.8000	Delta X:	0.0000	Delta Y:	0.0000	Distance:	0.0000
----	---------	----	---------	----------	--------	----------	--------	-----------	--------

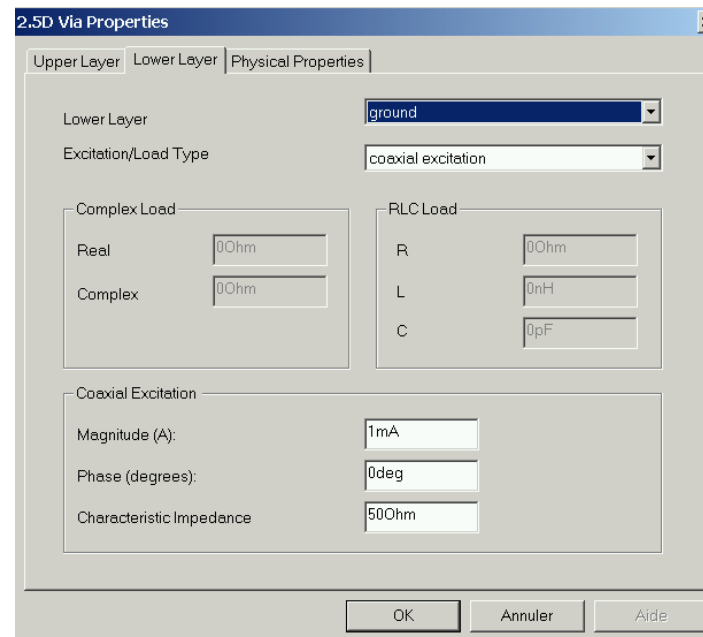
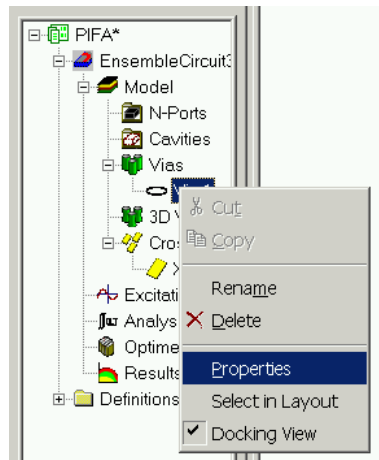
- ◆ Finish drawing the rectangle
  - ◆ In the Status Bar enter Delta X = 30, Delta Y = -36.8.
  - ◆ OR, drag the upper righthand corner until Delta X : 30, Delta Y : -36.8
- ◆ Fit the drawing by pressing “CTRL-D”, or View -> Fit Drawing

# Define Excitation

- ◆ Insert a hole 
- ◆ Double Click on the hole and change the properties

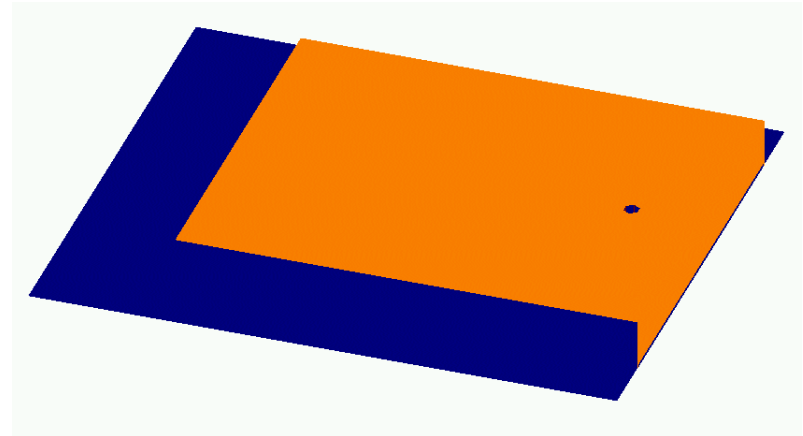
X: [ 20.0000 ] Y: [ 41.3000 ] Delta X: [ 0.0000 ] Delta Y: [ 0.0000 ] Distance: [ 0.0000 ]

- ◆ Enter the lower left-hand coordinates for the rectangle.
  - ◆ In the Status Bar, enter X = 20, Y = 41.8.
  - ◆ Use the TAB key to move between entries and press ENTER when finished.
- ◆ Click on via1 and choose Properties



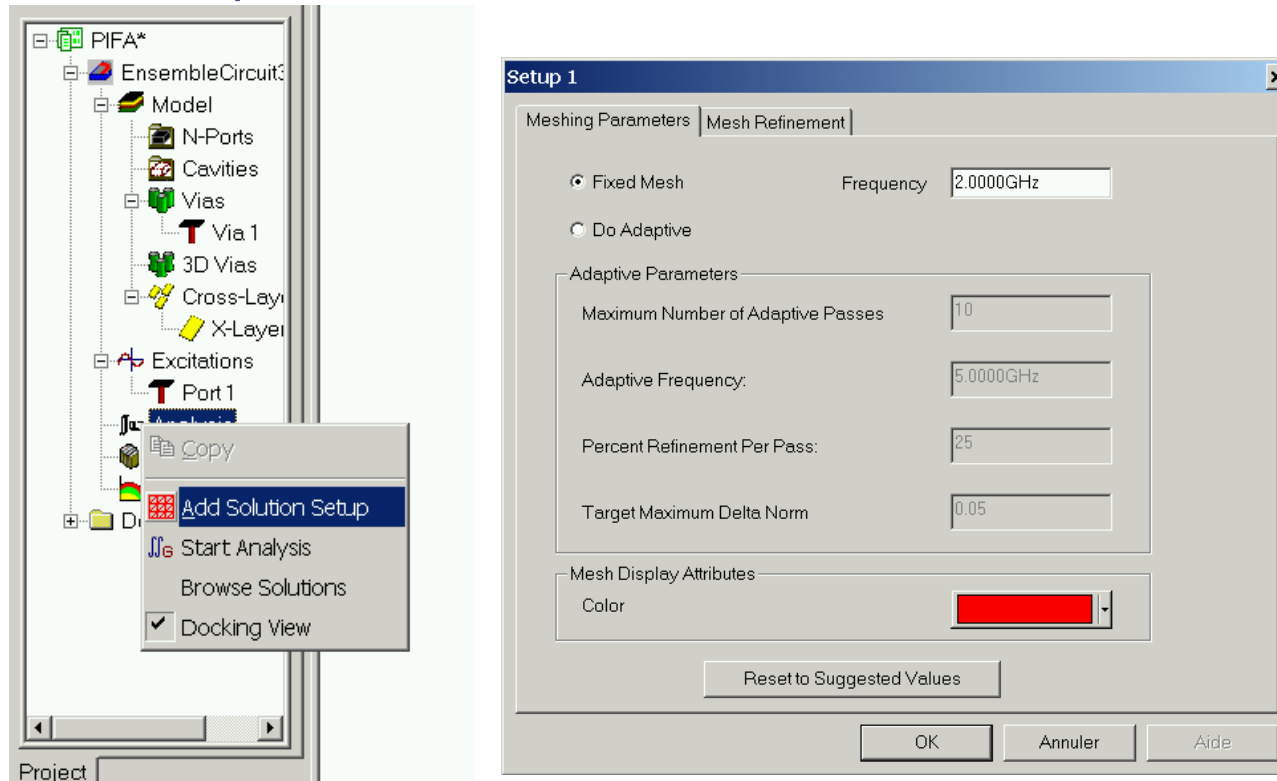
# 3D View

- ◆ Open a 3D view
  - ◆ Select Planar EM -> 3D Viewer OR
  - ◆ Right-click on PlanarEM1 in the Project Manager and select “3D Viewer”
- ◆ Explore 3D view functionality
  - ◆ Right-click in the 3D view window
  - ◆ Many options are available here....
- ◆ Change the 3D lighting
  - ◆ View -> modify attributes -> lighting
- ◆ Change the background color
  - ◆ View -> modify attributes -> background color



# Setup Analysis (1)

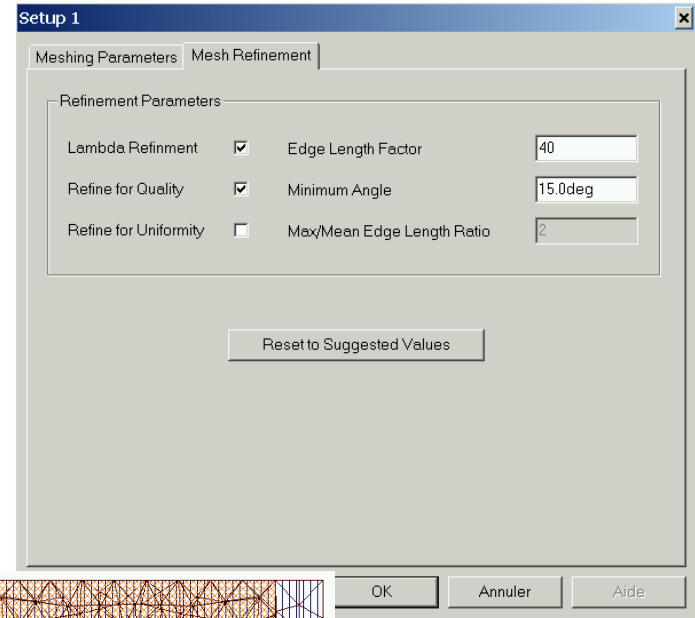
- ◆ Right click on the **Analysis** entry in the project tree and select **add solution setup**



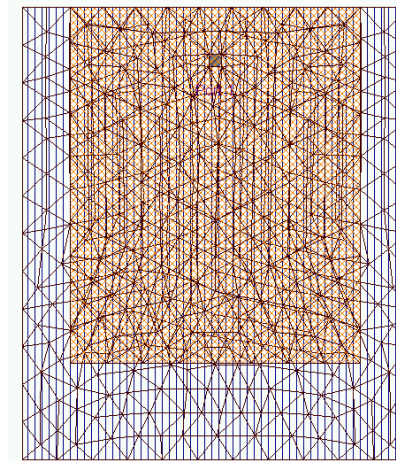
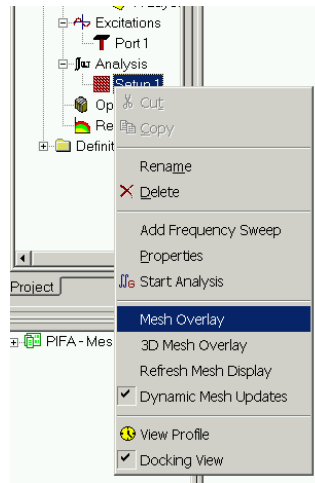
- ◆ Use a fixed mesh frequency of 2GHz. You can also change the mesh display color.

# Setup Analysis (2)

- ◆ Modify Mesh refinement



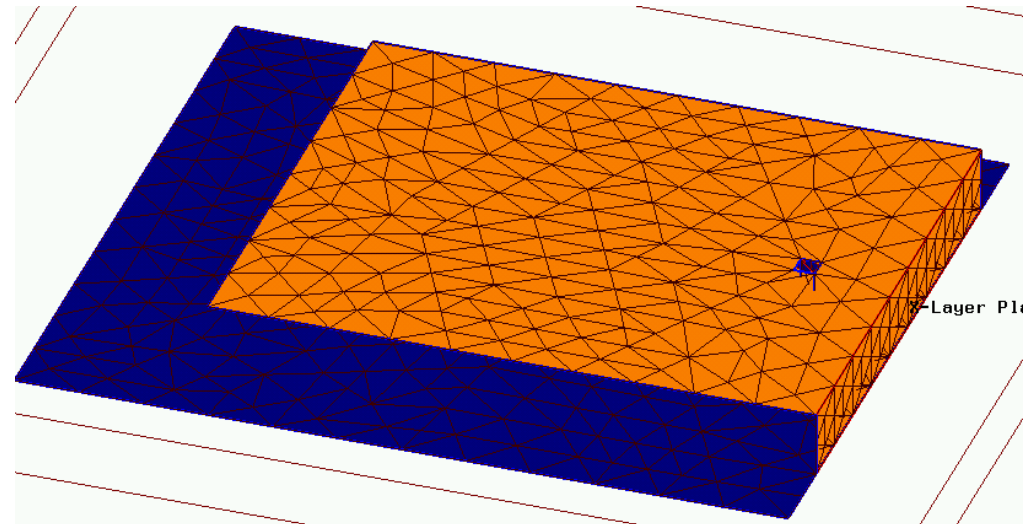
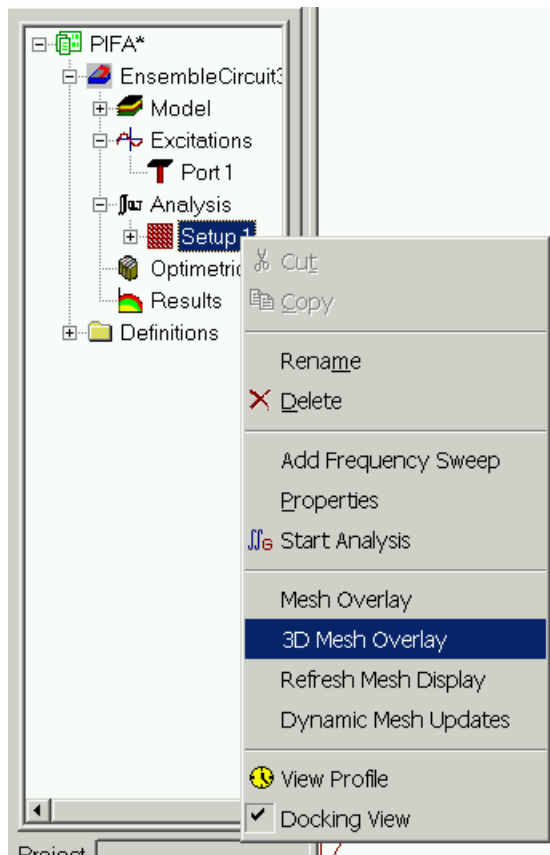
- ◆ Visualize the mesh





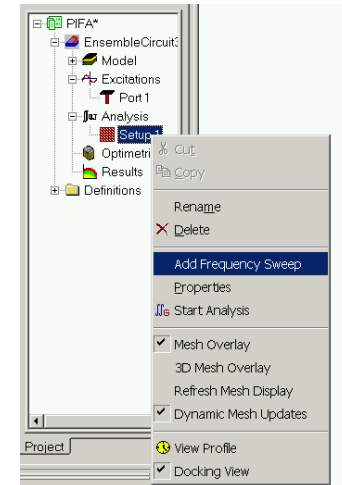
# 3D Mesh

- ◆ Visualize 3D mesh

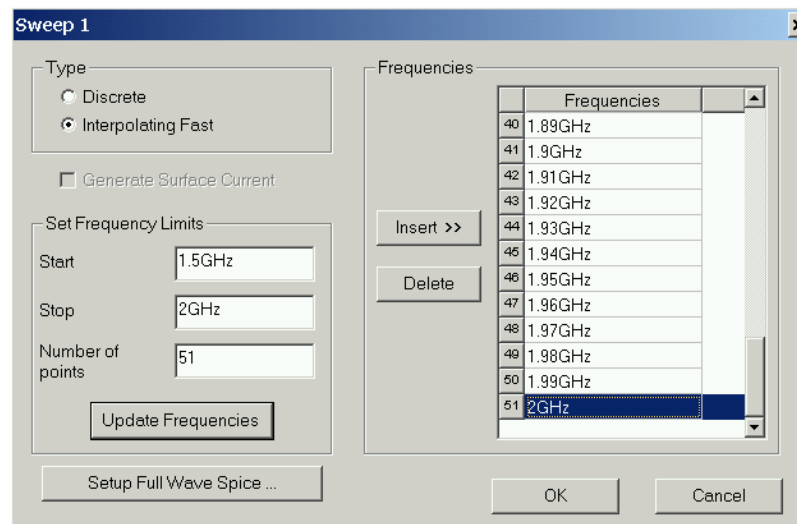


# Setup Analysis (3)

- ◆ Add a sweep to the analysis in one of two ways
  - ◆ Planar EM -> Add Frequency Sweep OR
  - ◆ Right-click on the “Sweep 1” branch in the Project Manager and select “Add Frequency Sweep”

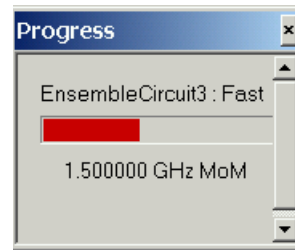
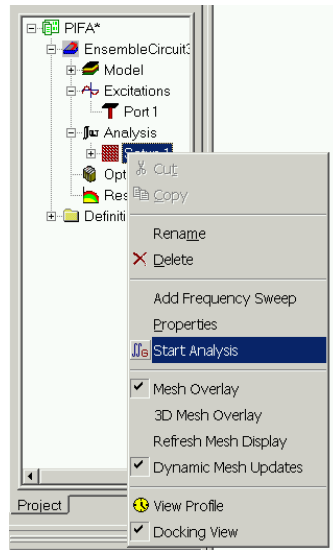


- ◆ Fill in the Sweep dialogue as seen below. Enter “Update Frequencies” to add the points to the sweep table.
  - ◆ Interpolating Fast Sweep
    - ◆ Fstart = 1.5GHz
    - ◆ Fstop = 2GHz
    - ◆ Number of points = 51

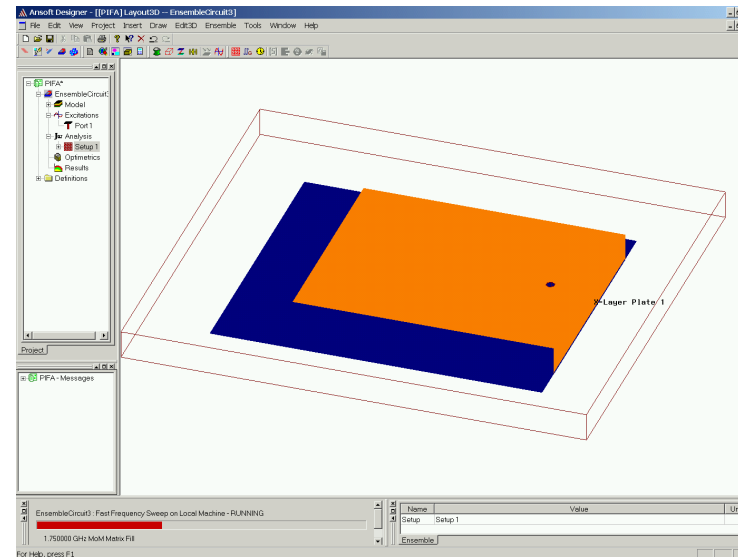


# Setup Solution

- ◆ Right click on Sweep1 and select Start Analysis.

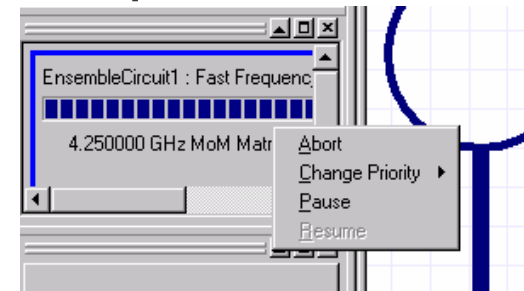


- ◆ Click on 3D Editor.

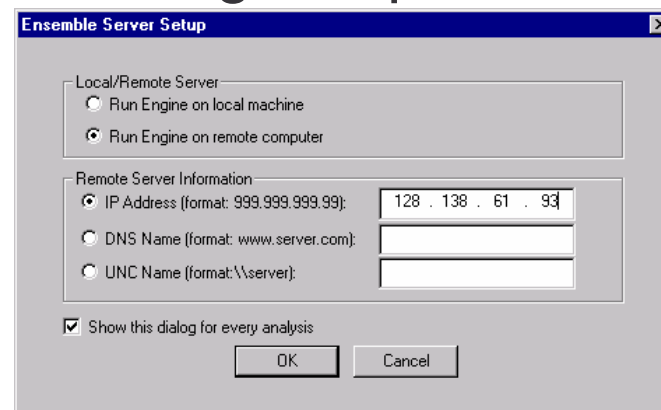


# Simulation Odds and Ends

- ◆ By Right Clicking on the progress bar during simulation, the process can be aborted, paused, or have the priority level changed.

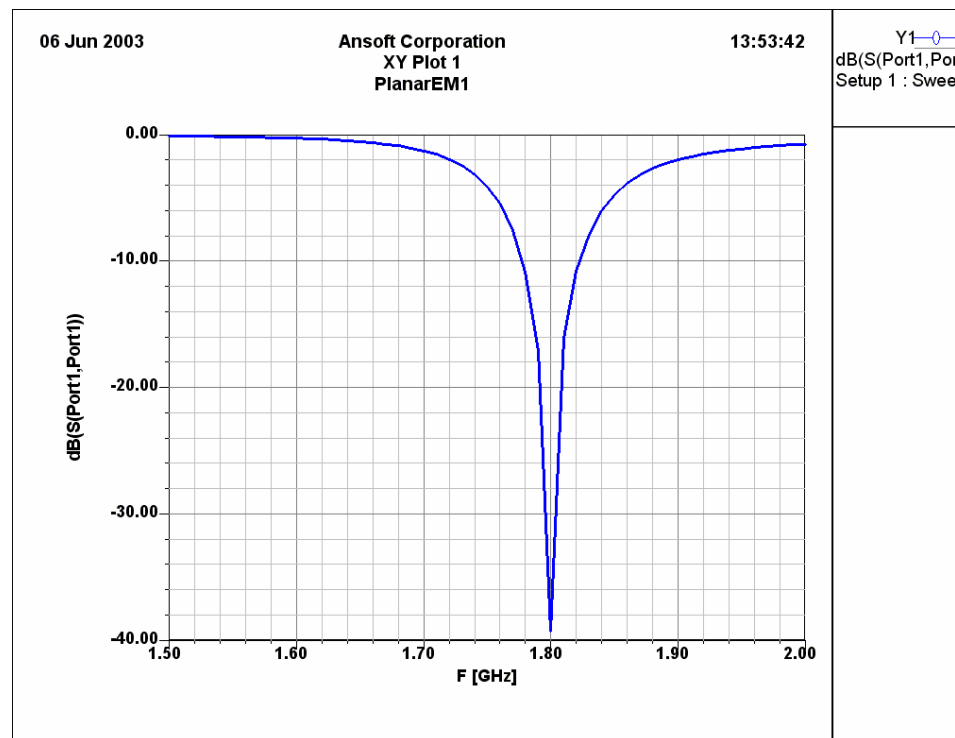


- ◆ The simulation engines have been turned into COM/DCOM objects. This allows a simulation to be run on a remote machine, given permissions and an IP address.



# Generate Reports (1)

- ◆ View tabular S-parameters
  - ◆ Right-click on “Sweep1”
  - ◆ Select Results -> Matrix Data
- ◆ Plot Return Loss
  - ◆ Right-click on “Sweep1”
  - ◆ Select Results -> Plot Templates -> Return Loss



# Generate Reports (2)

