Ansoft Designer Training

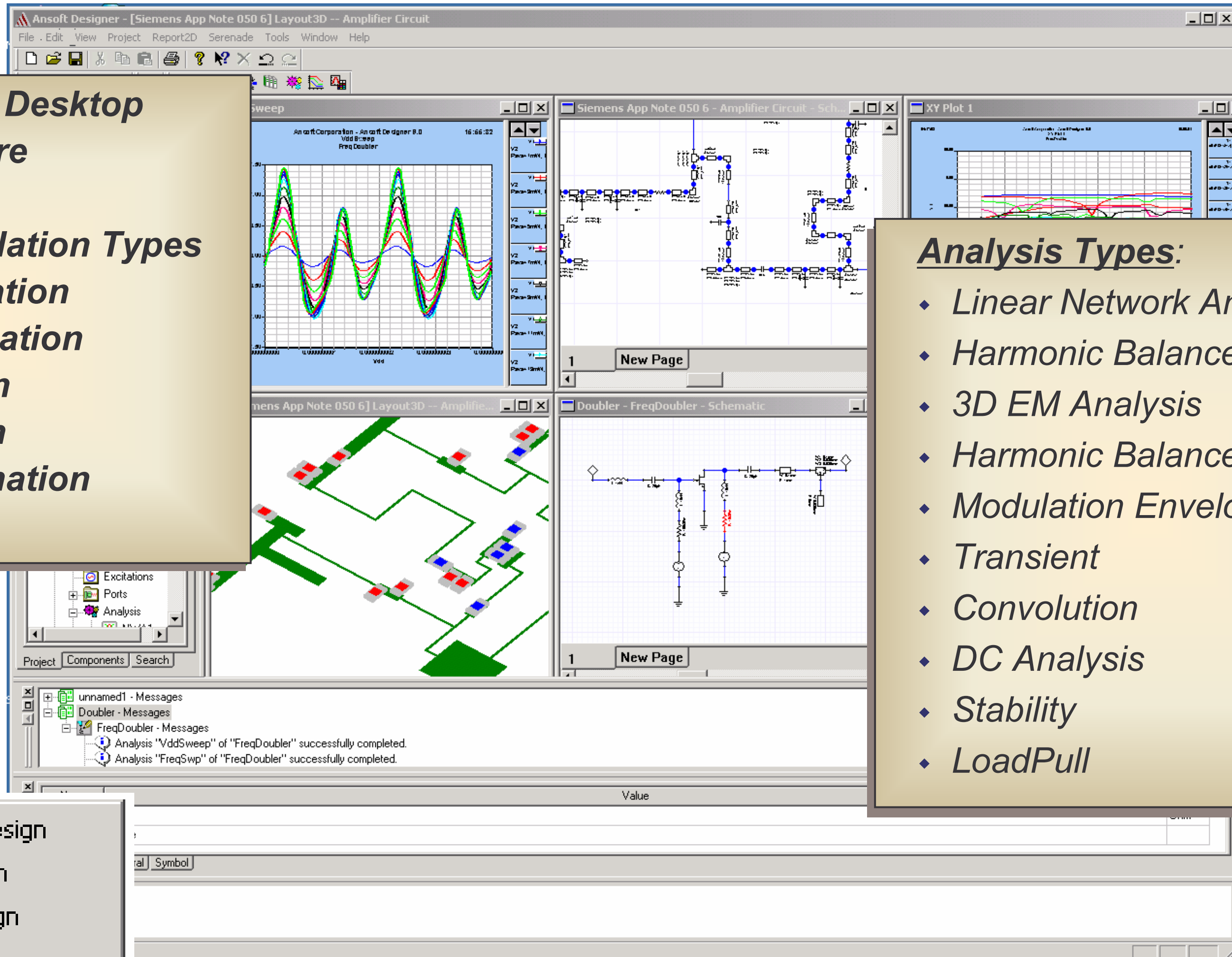
Agenda

- 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*

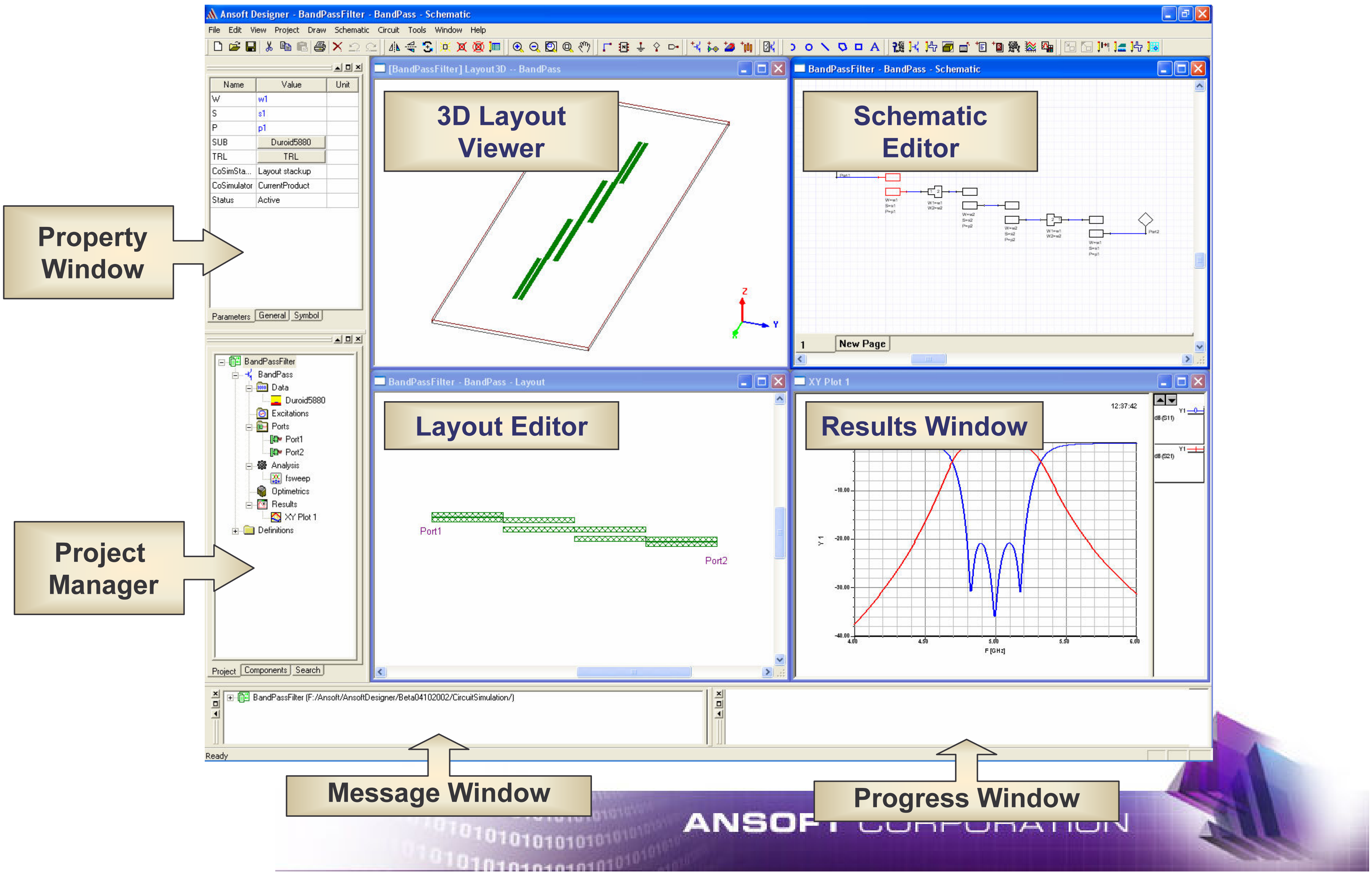


Analysis Types:

- *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 Designer Windows



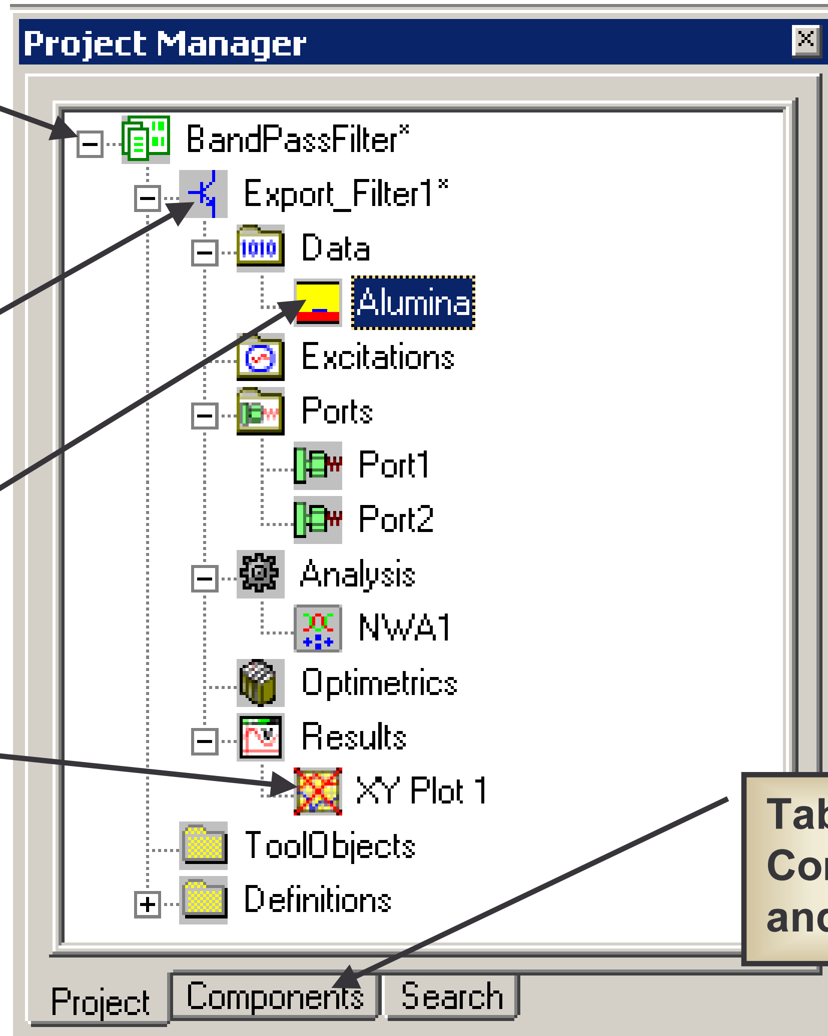
Project Manager Window

Click on the +/- to expand or collapse the tree

Double click on the design icon to open the schematic editor

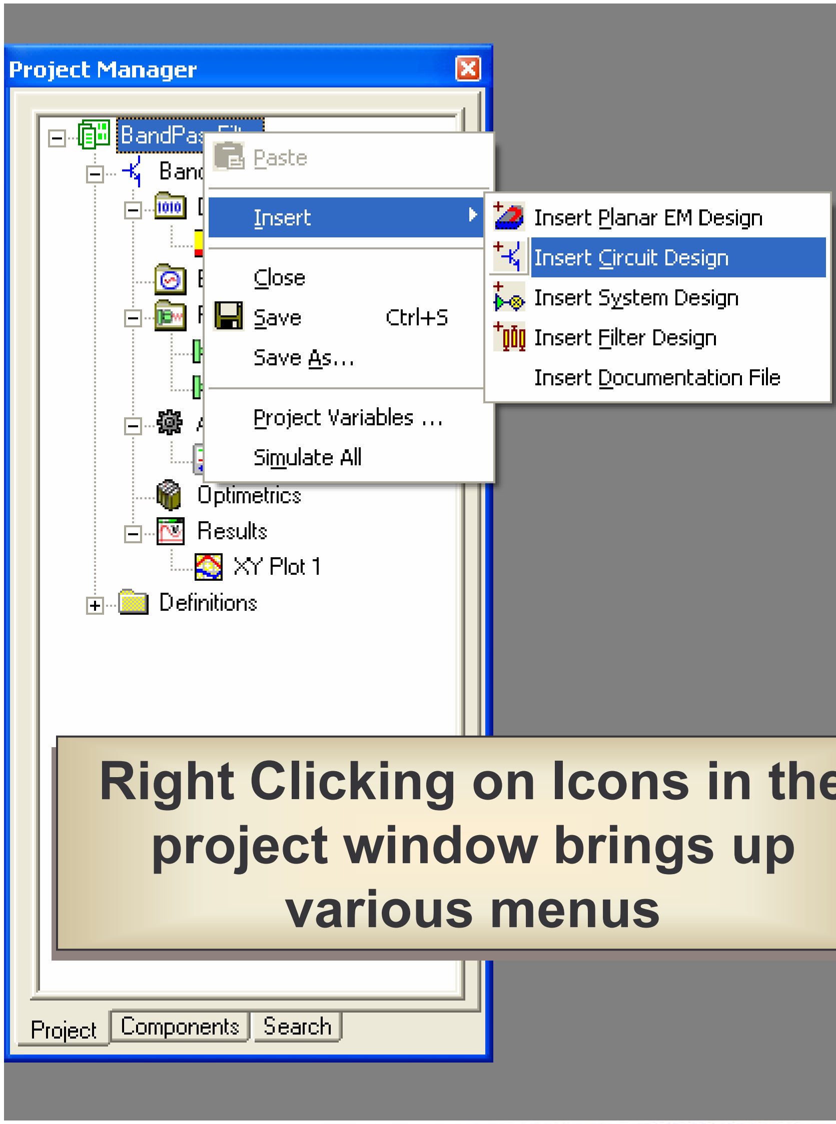
Double click on the substrate or analysis icon to open the Definition dialog for those objects

Double click on the graph icon to display results

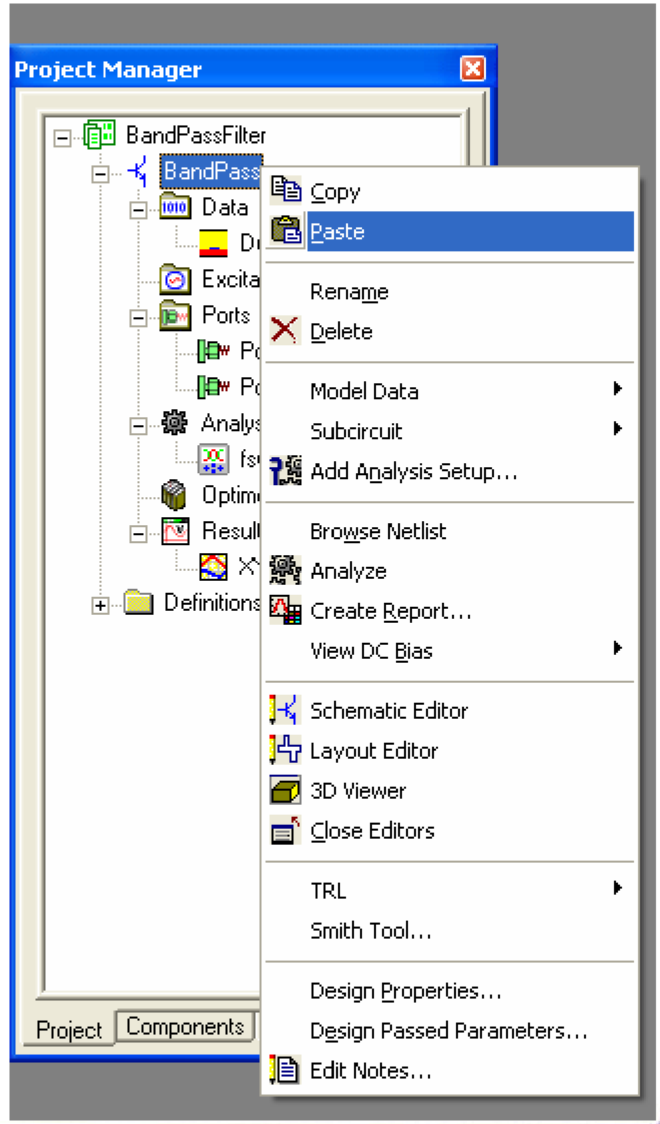


Tabs for project, Components, and Search

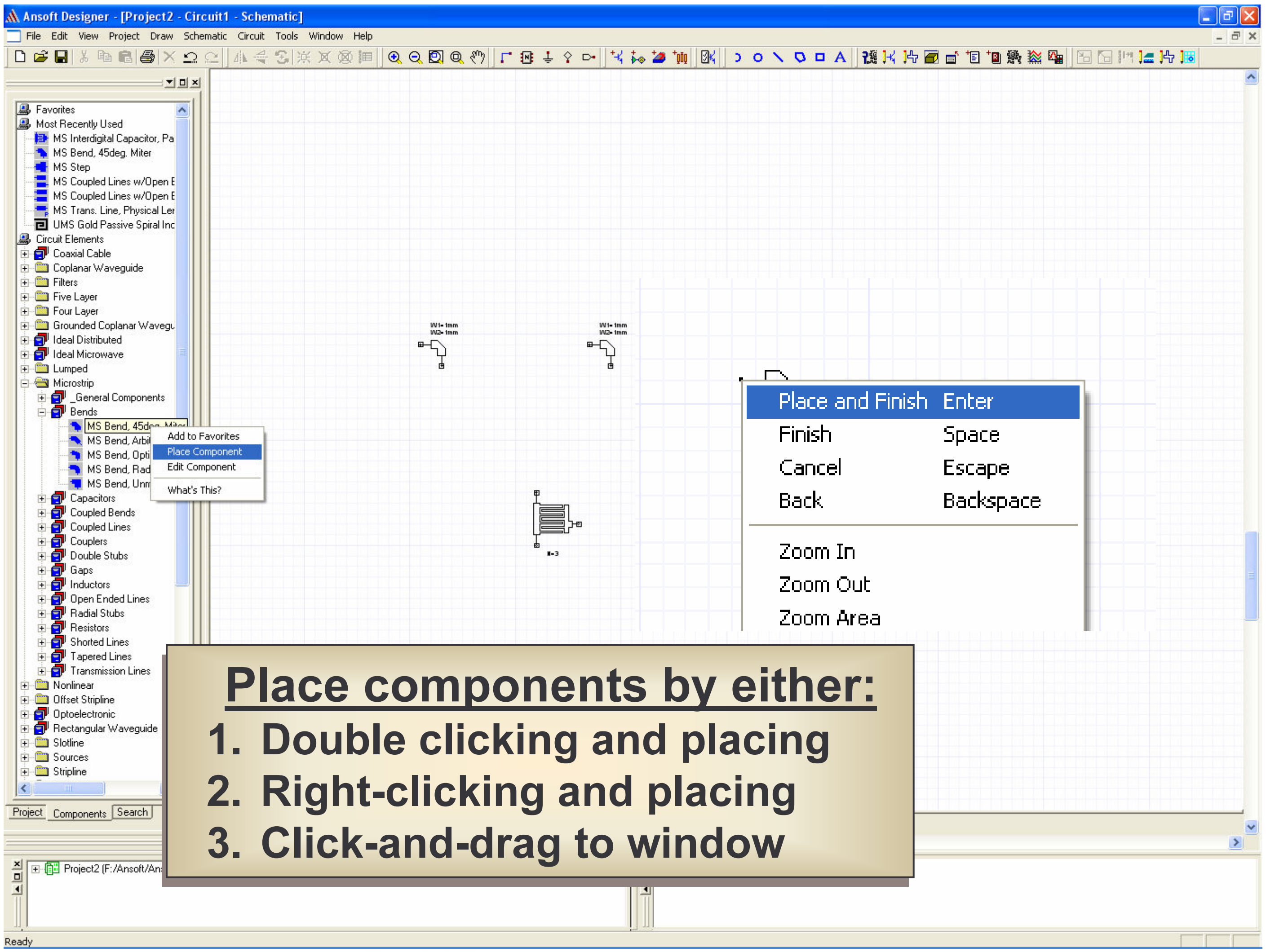
Right Click Pop Up Menu



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



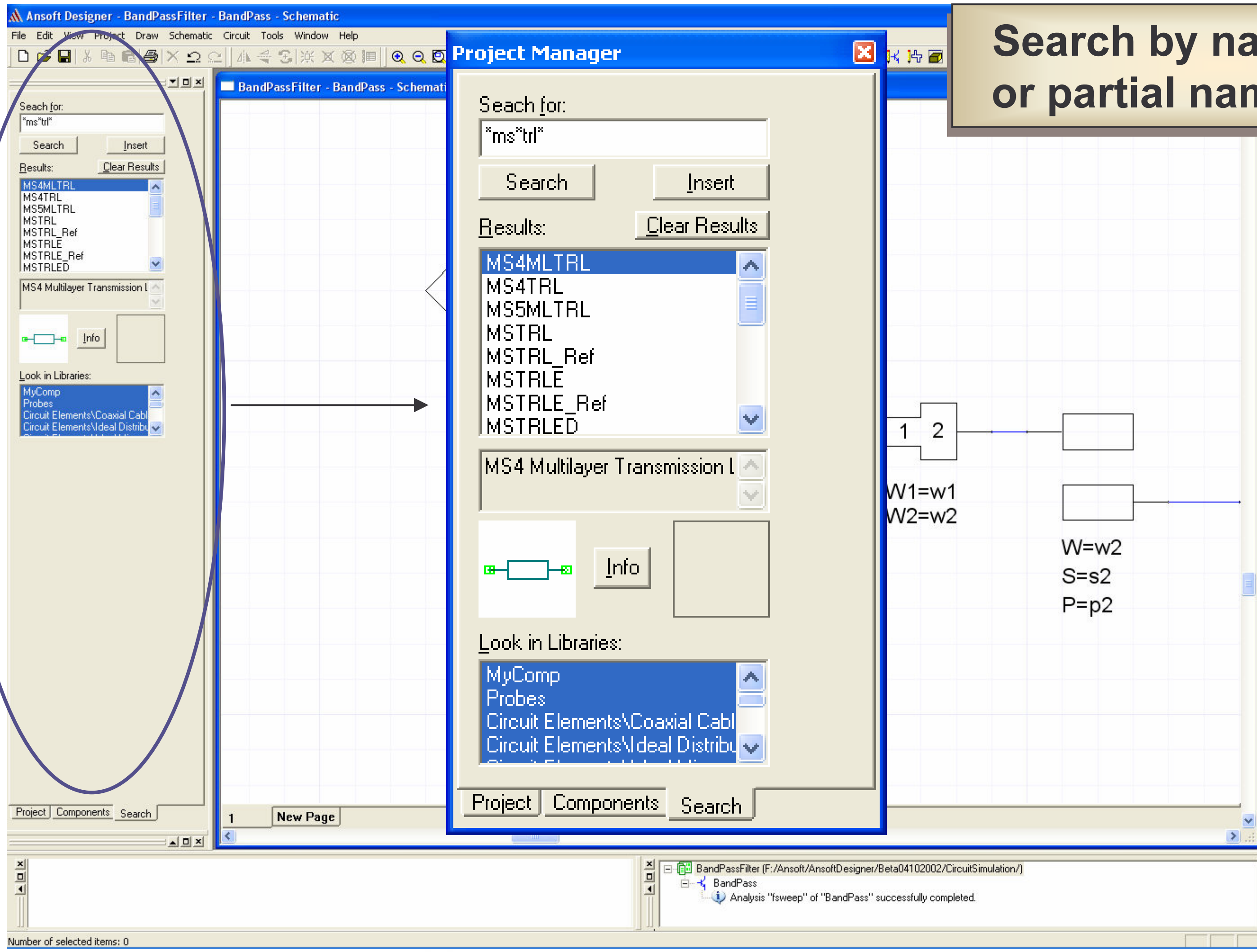
Component Tab



Place components by either:

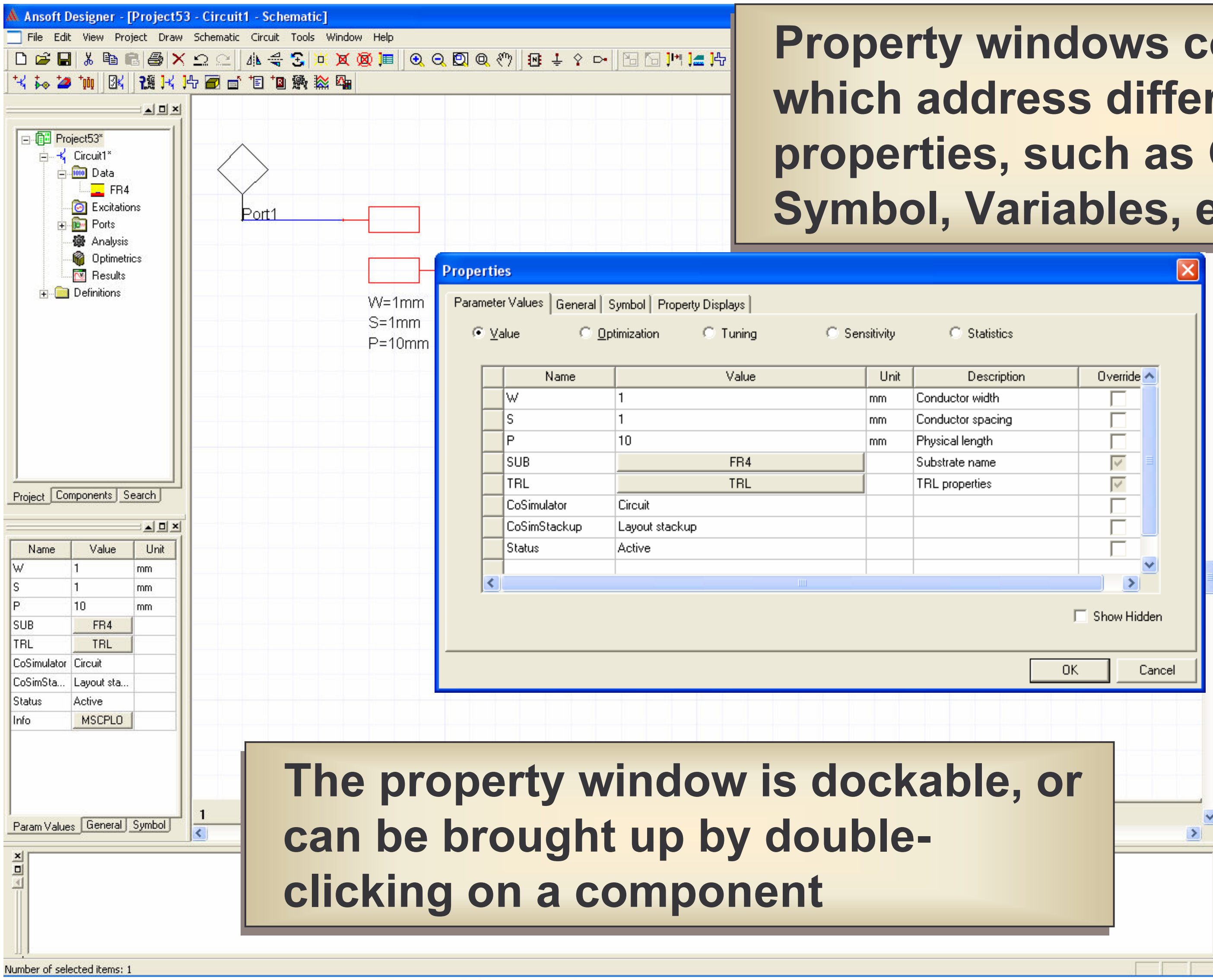
1. Double clicking and placing
2. Right-clicking and placing
3. Click-and-drag to window

Search Tab



Search by name, type, or partial names

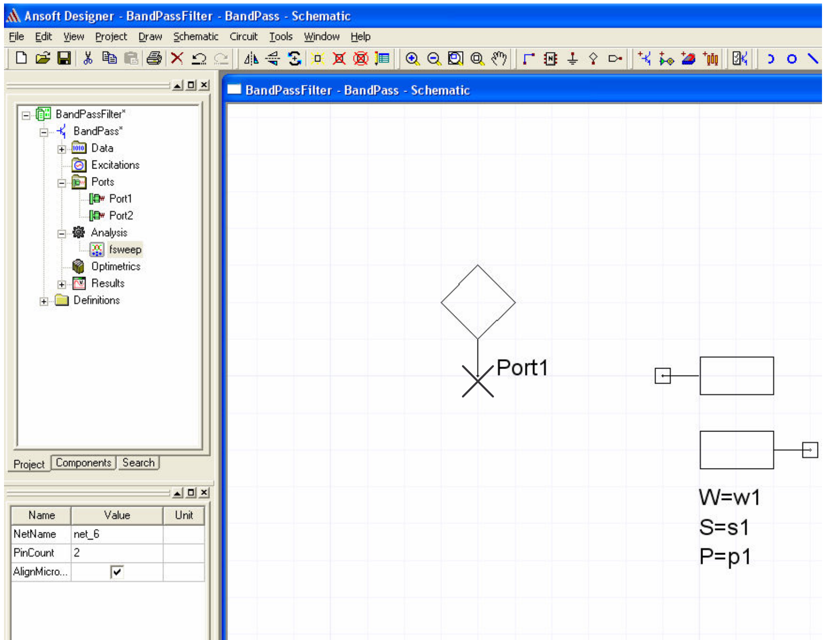
Property Window

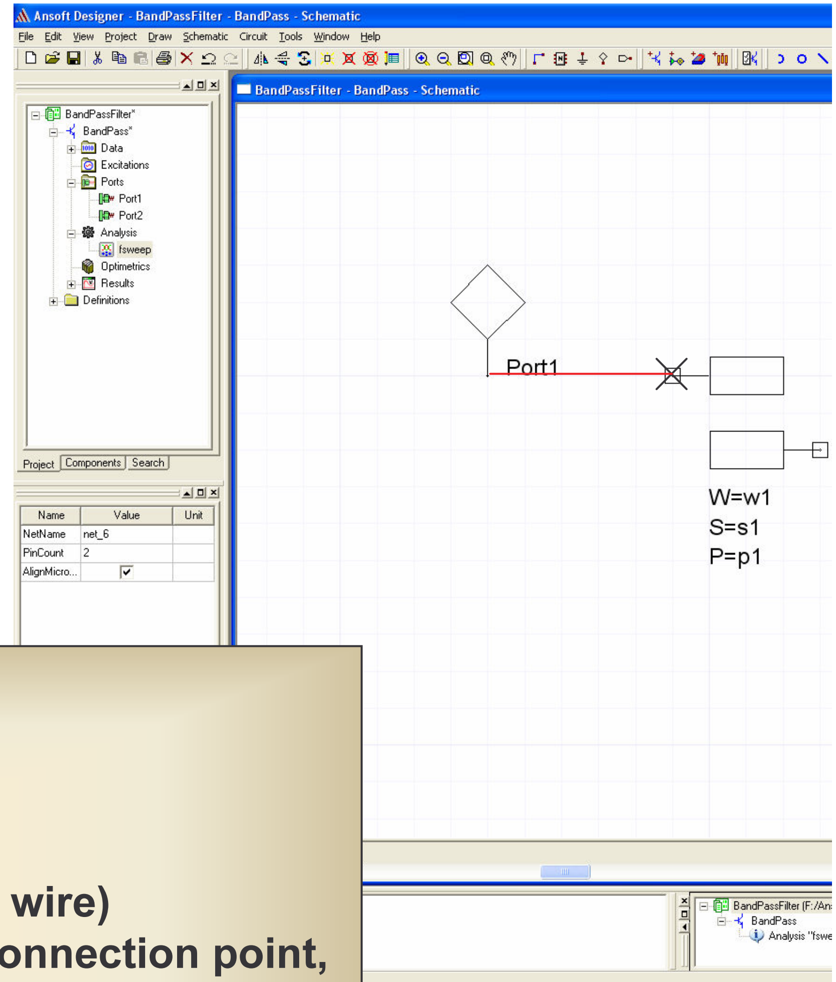


Property windows contains tabs which address different types of properties, such as General, Symbol, Variables, etc.

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

Schematic Window

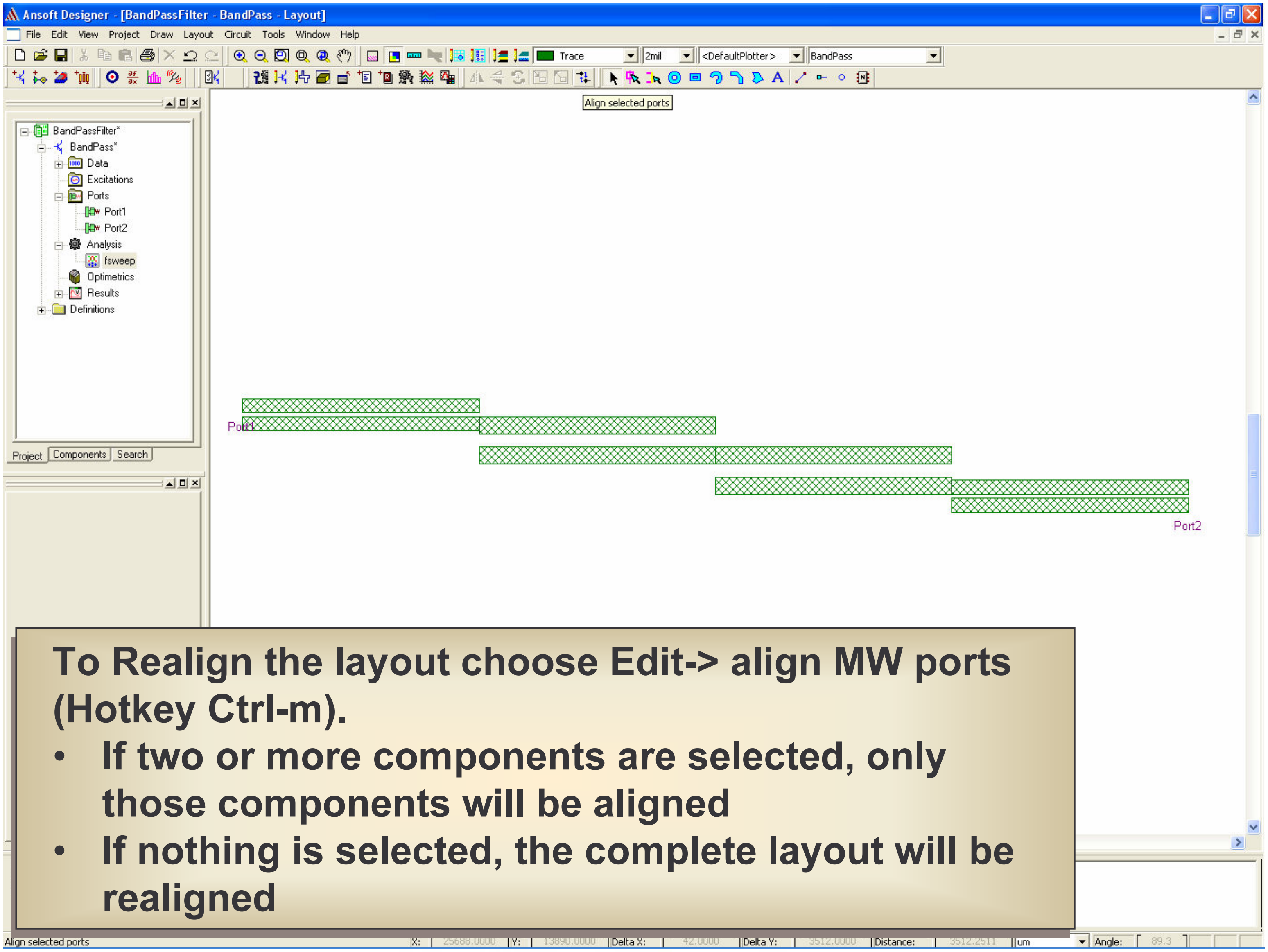




<u>To wire a component:</u>

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

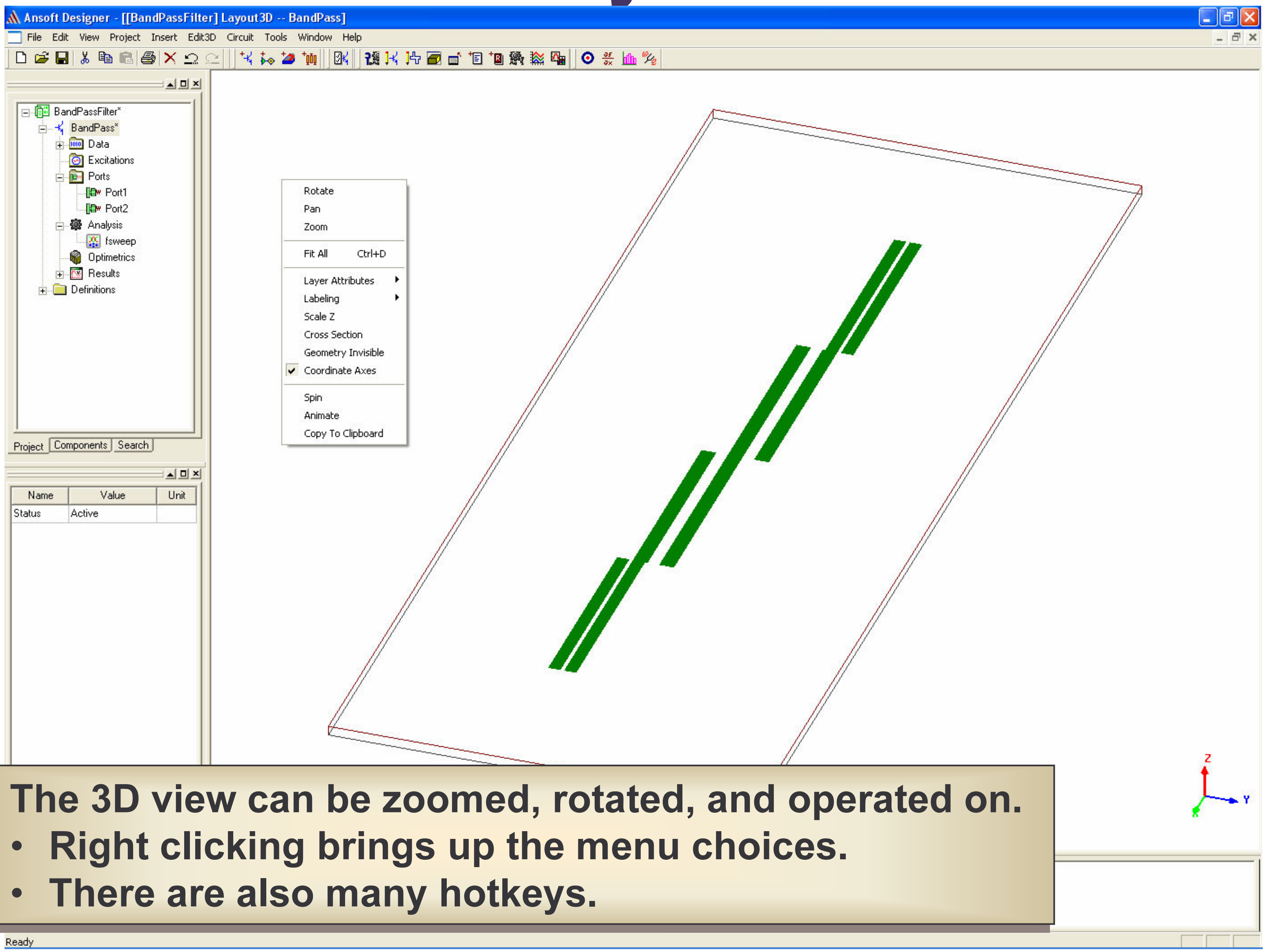
Layout Editor



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

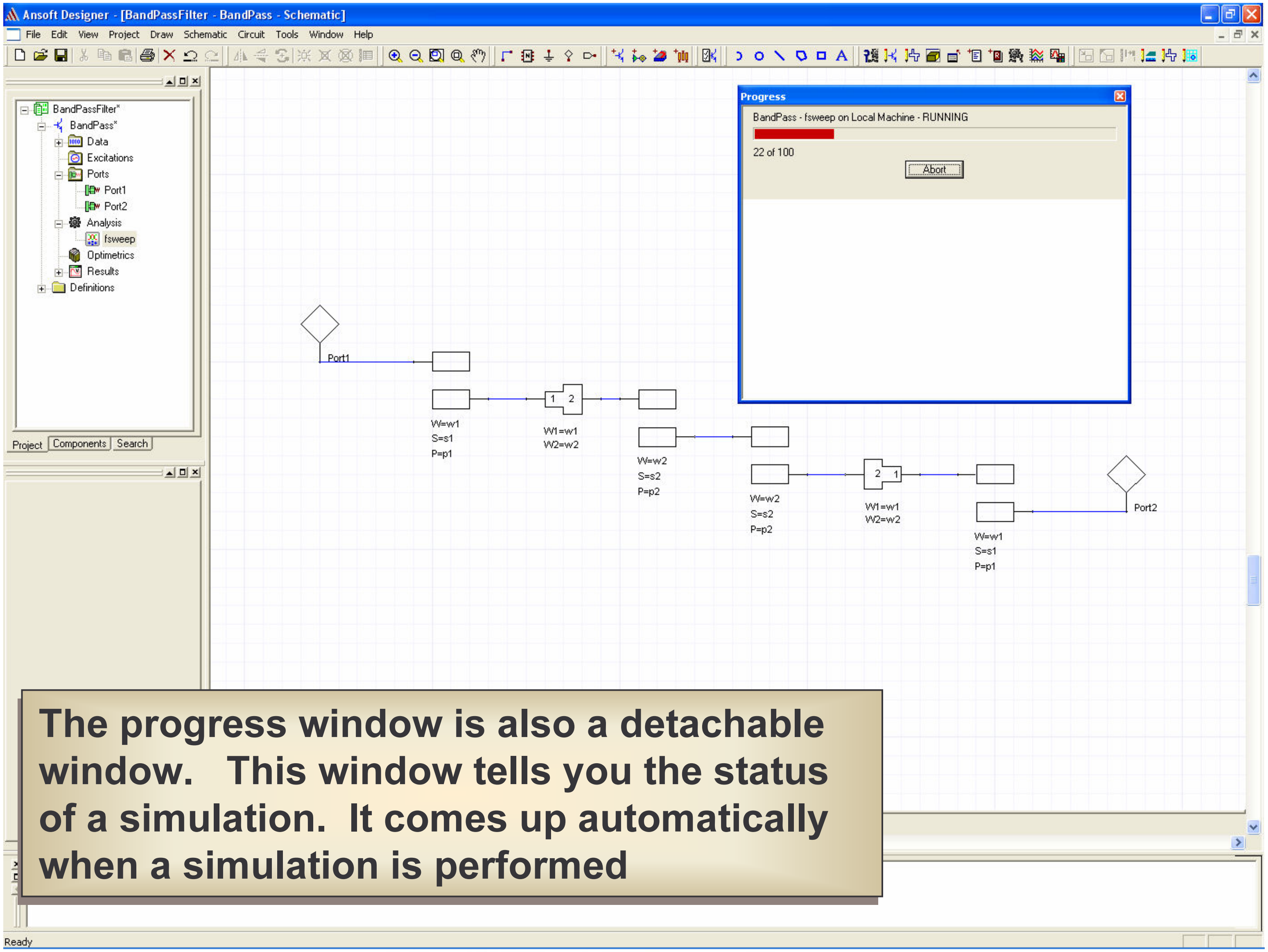
3D Layout Viewer



The 3D view can be zoomed, rotated, and operated on.

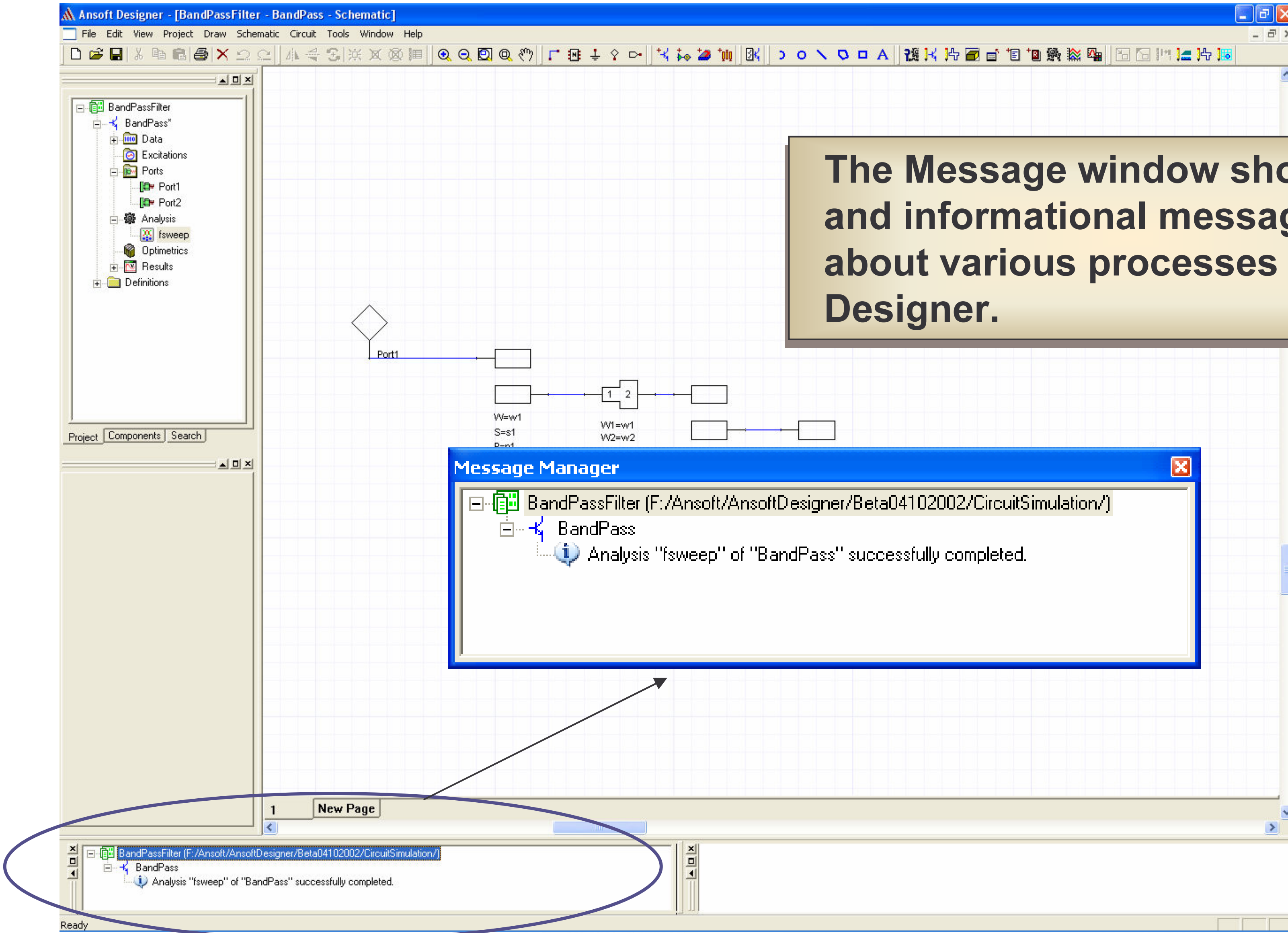
- **Right clicking brings up the menu choices.**
- **There are also many hotkeys.**

Progress Window



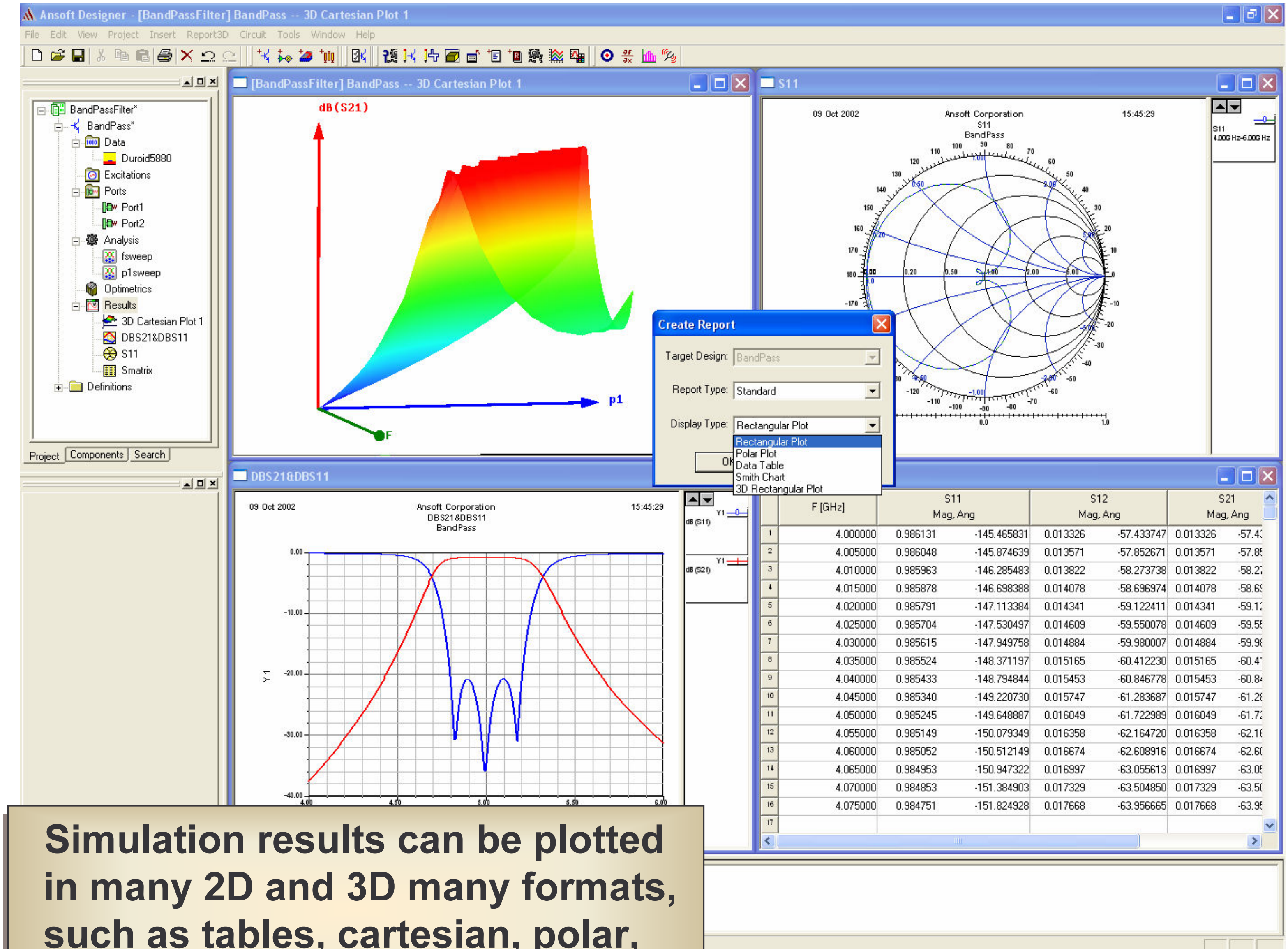
The progress window is also a detachable window. This window tells you the status of a simulation. It comes up automatically when a simulation is performed

Message Window



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

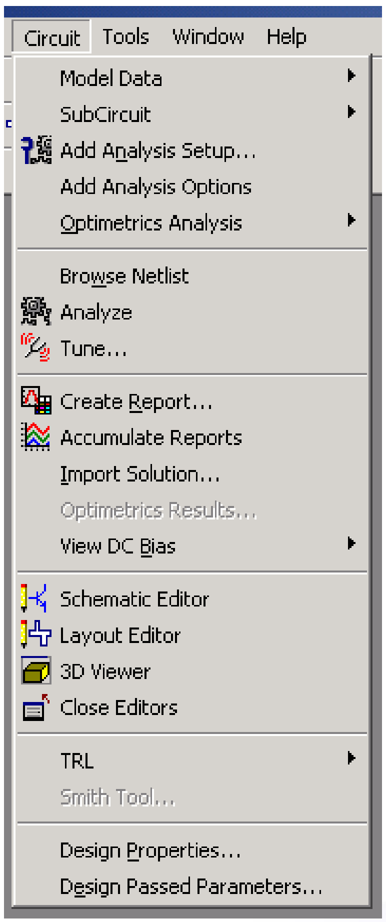
Results Window

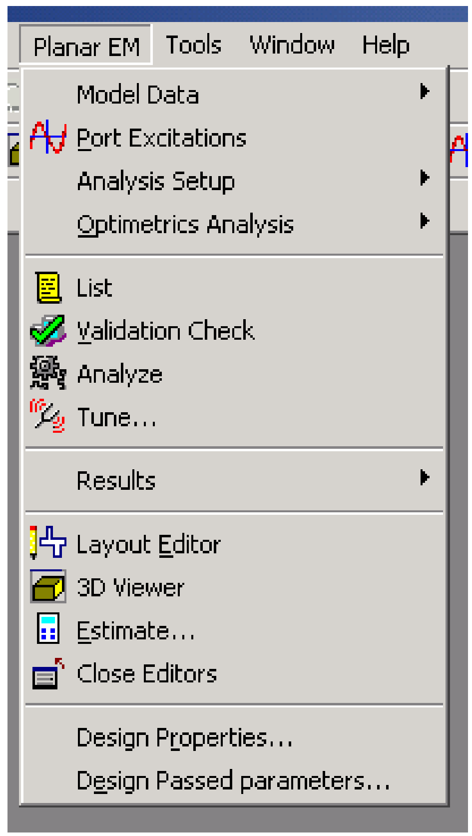


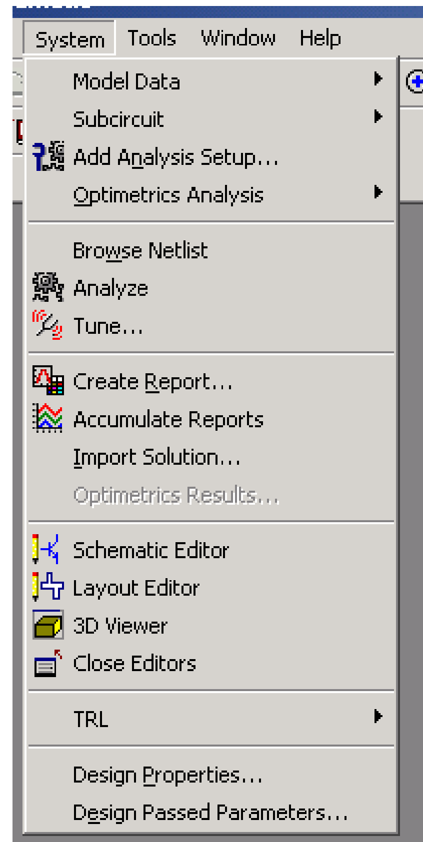
Simulation results can be plotted in many 2D and 3D many formats, such as tables, cartesian, polar, smith, and others.

Dynamic Menus

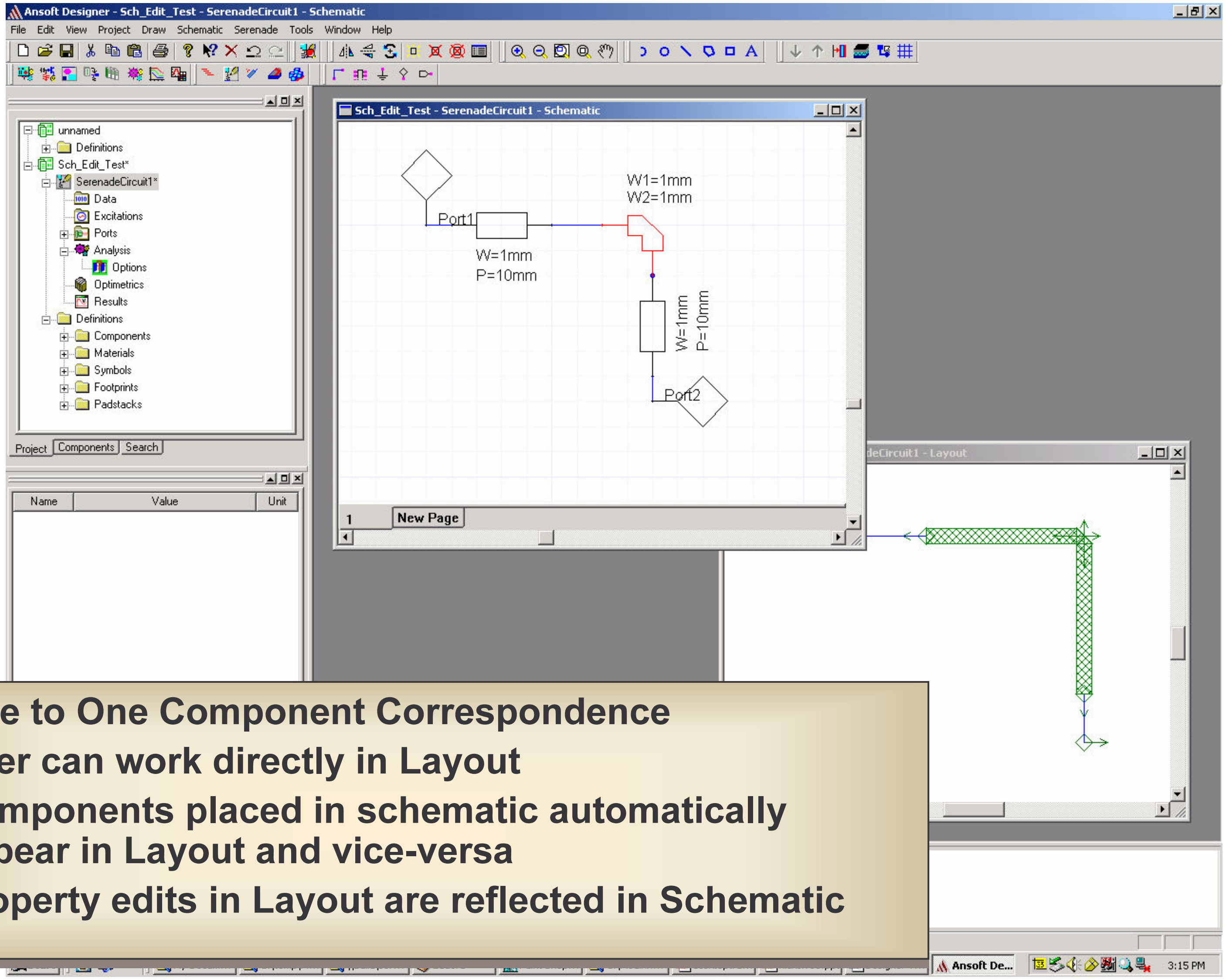
Menus change dynamically depending on which Design window is highlighted (Circuit, Planar EM or System)





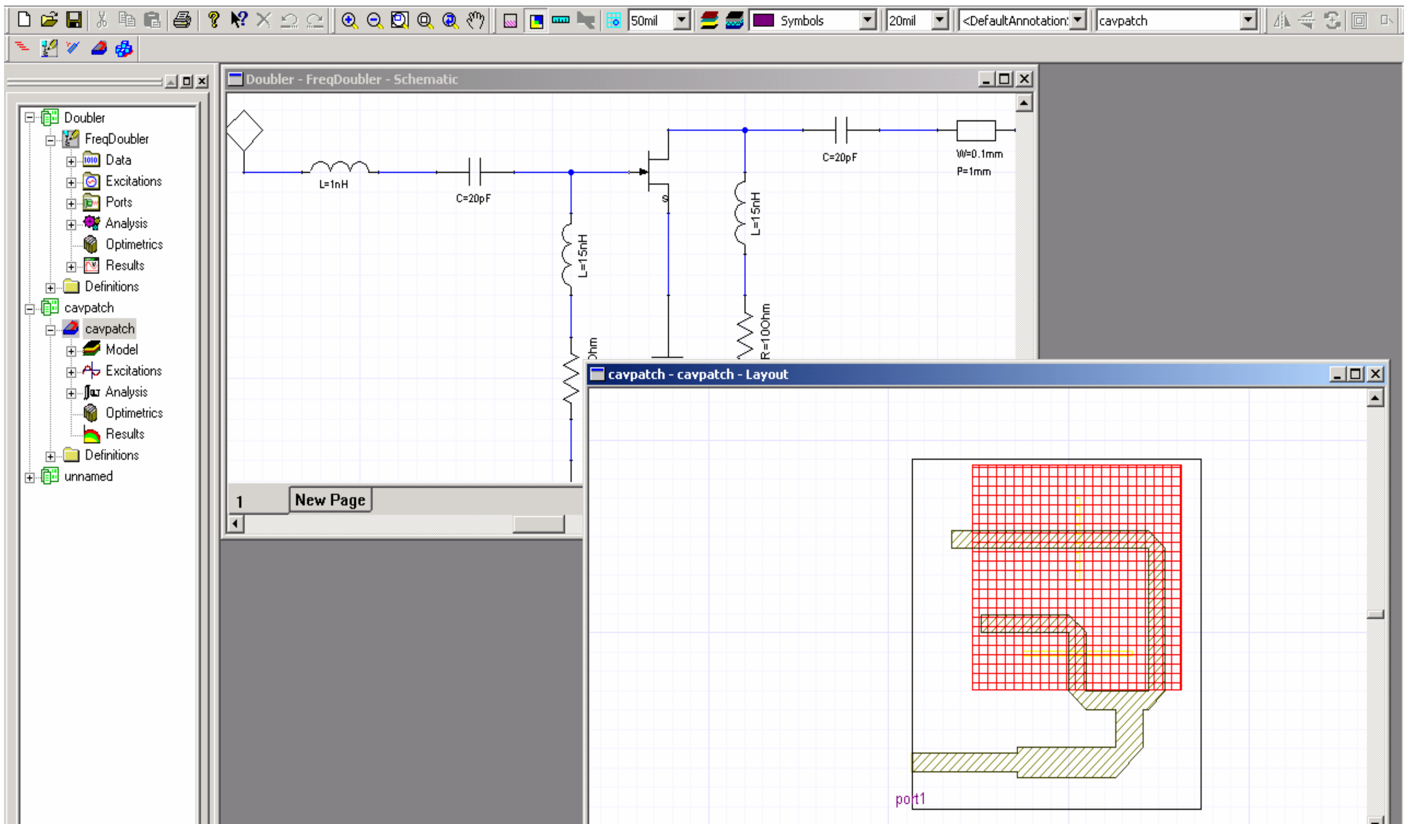


Schematic/Layout Integration



- 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

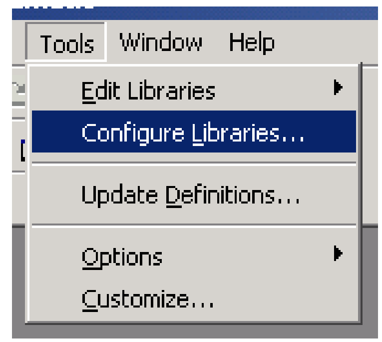
Simulation Tool Integration

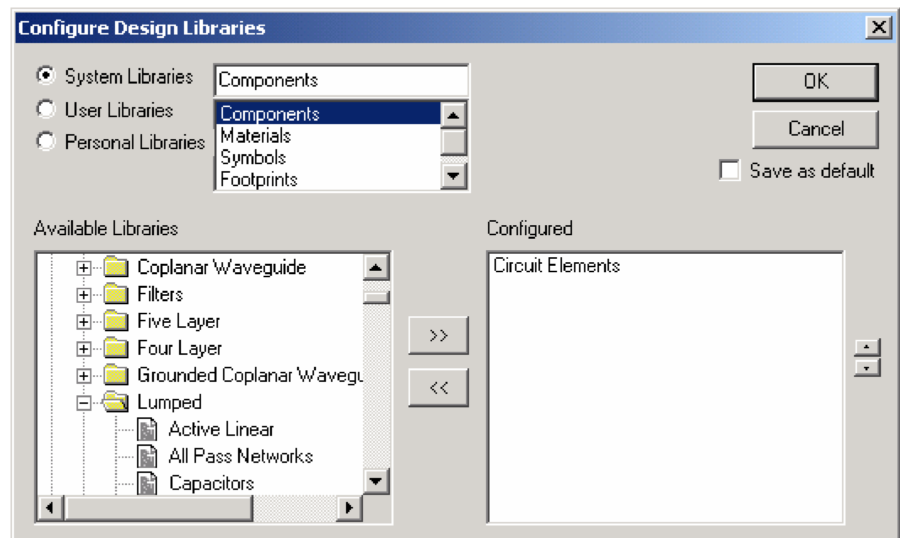


- Common Desktop/GUI: Circuit, Planar EM, System all work from a common GUI.
- Co-Simulation: Circuit/System/Planar EM
- Solver on demand: EM Kernel components
- Common project, component, library management.

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.

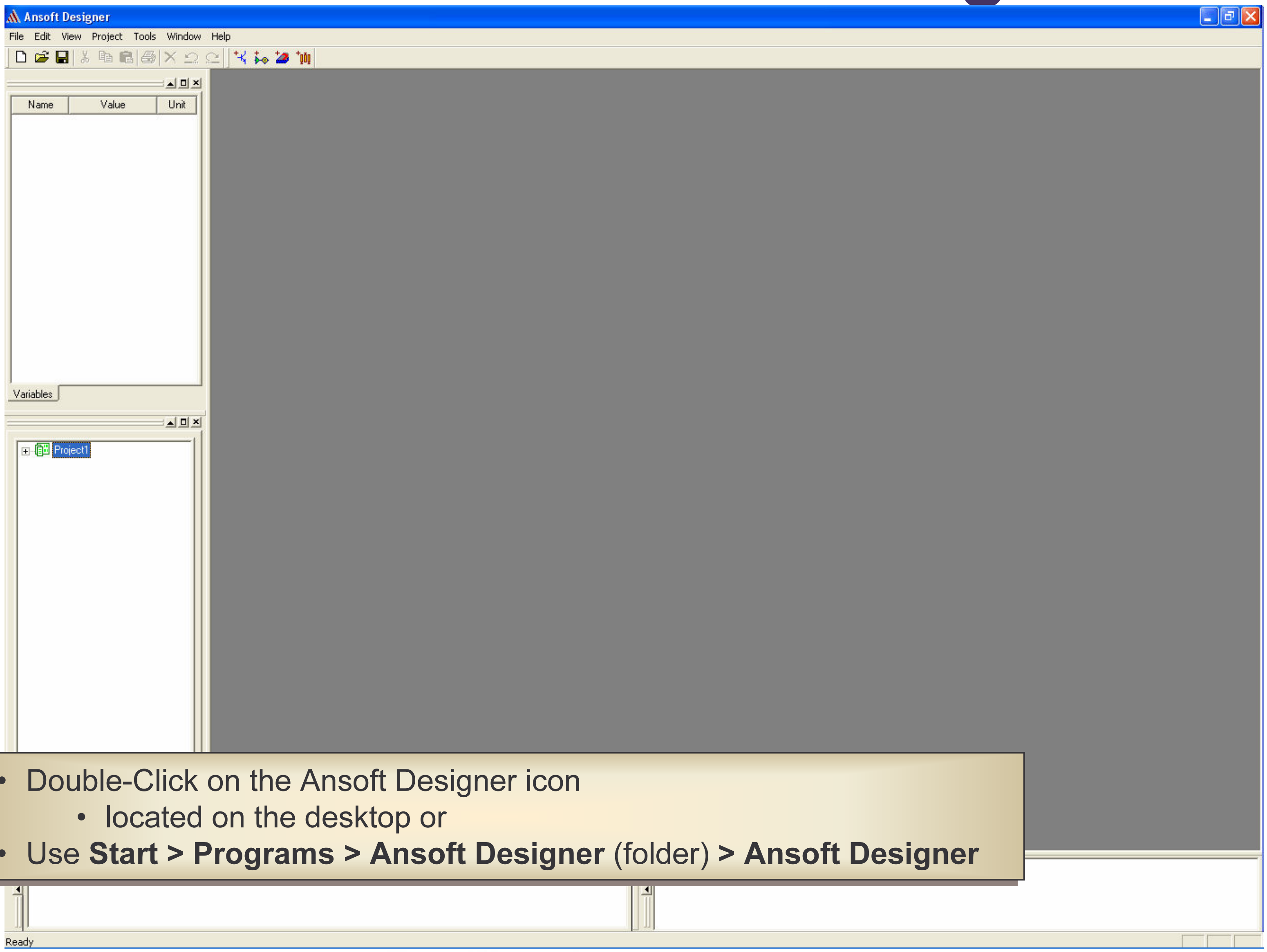
Exercise 1: Using The Interface

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

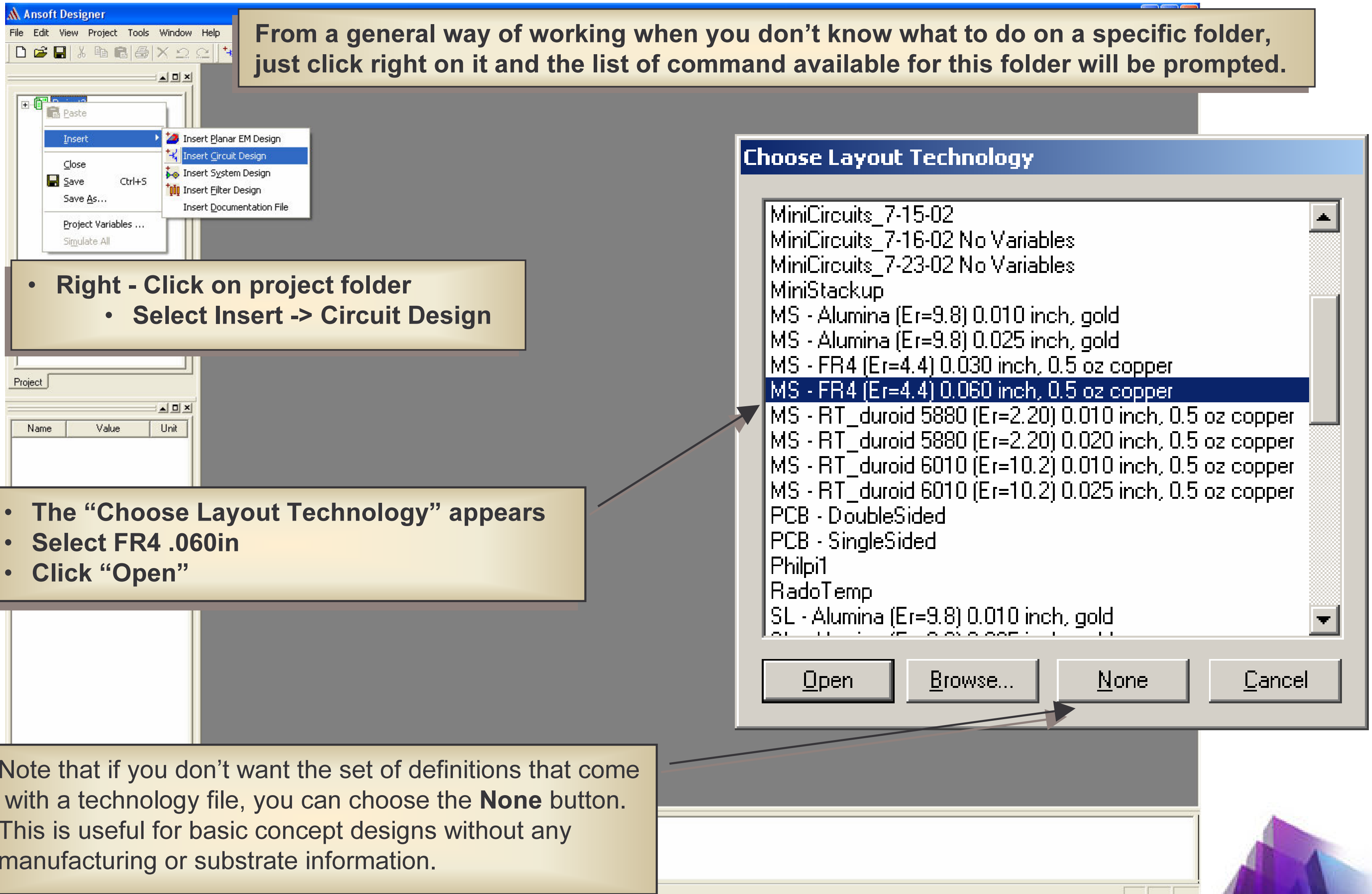


- Double-Click on the Ansoft Designer icon
  - located on the desktop or
- Use **Start > Programs > Ansoft Designer** (folder) **> Ansoft Designer**

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**

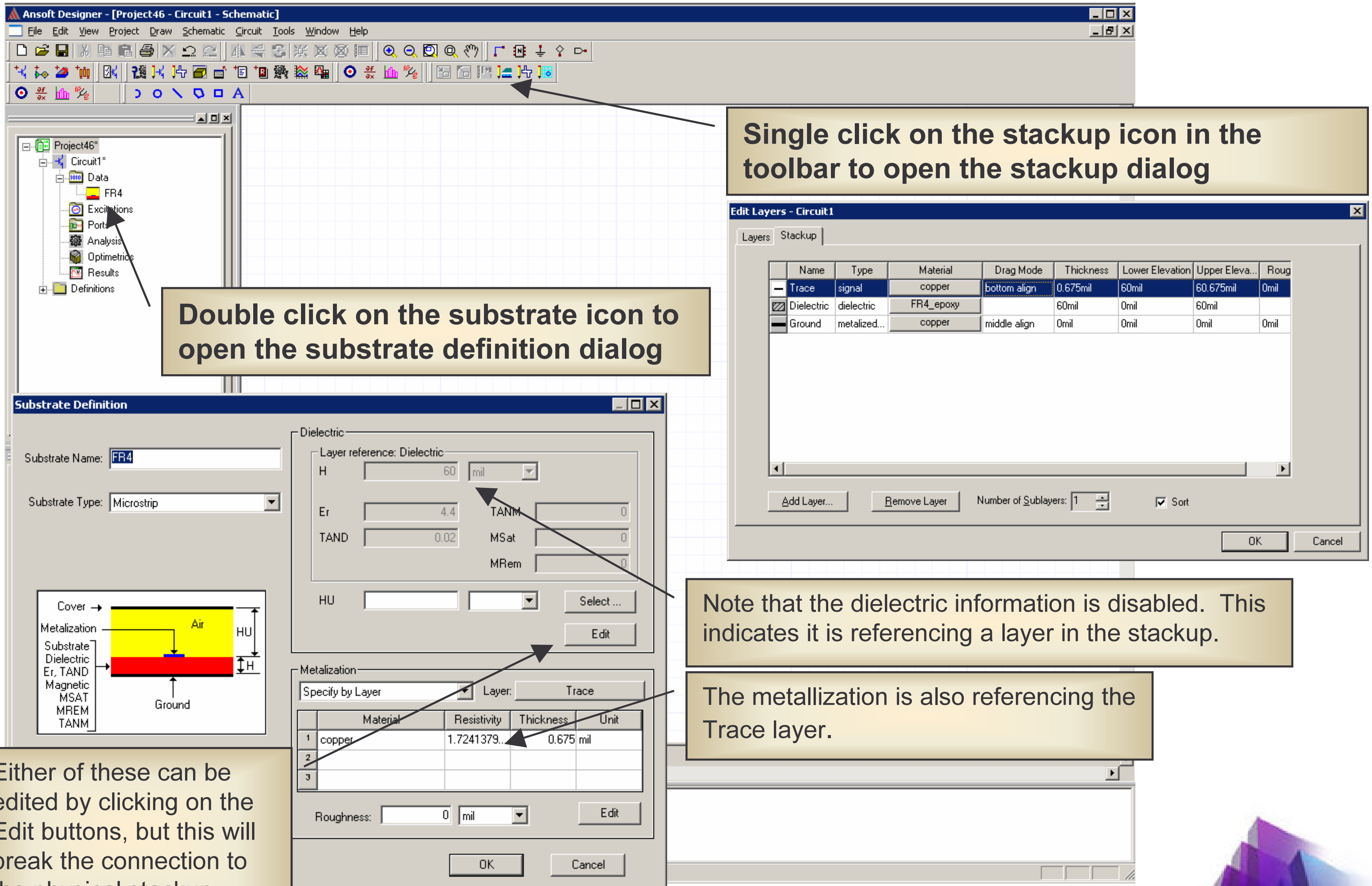
- **The "Choose Layout Technology" appears**
- **Select FR4 .060in**
- **Click "Open"**

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

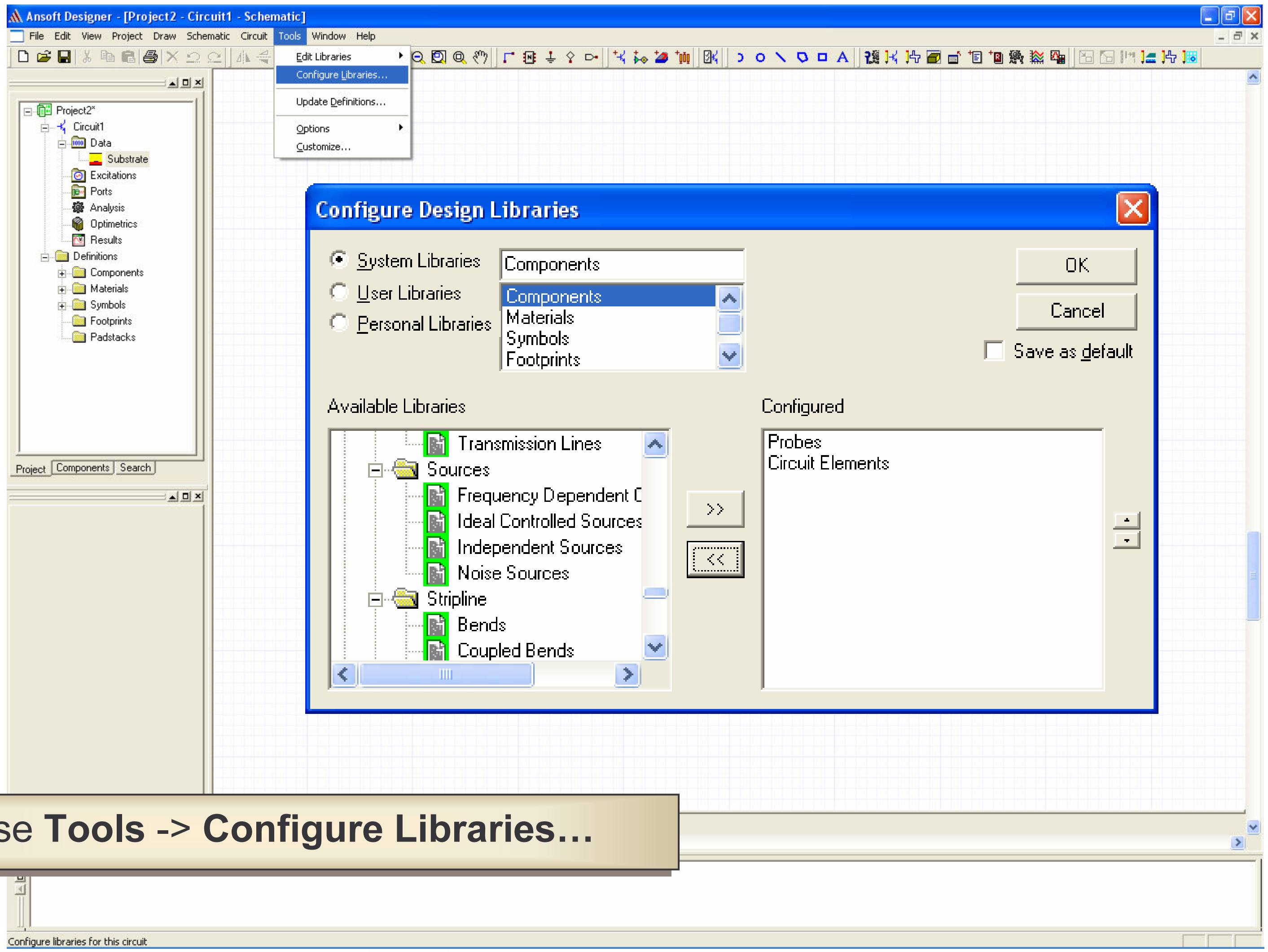
Double click on the substrate icon to open the substrate definition dialog

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.

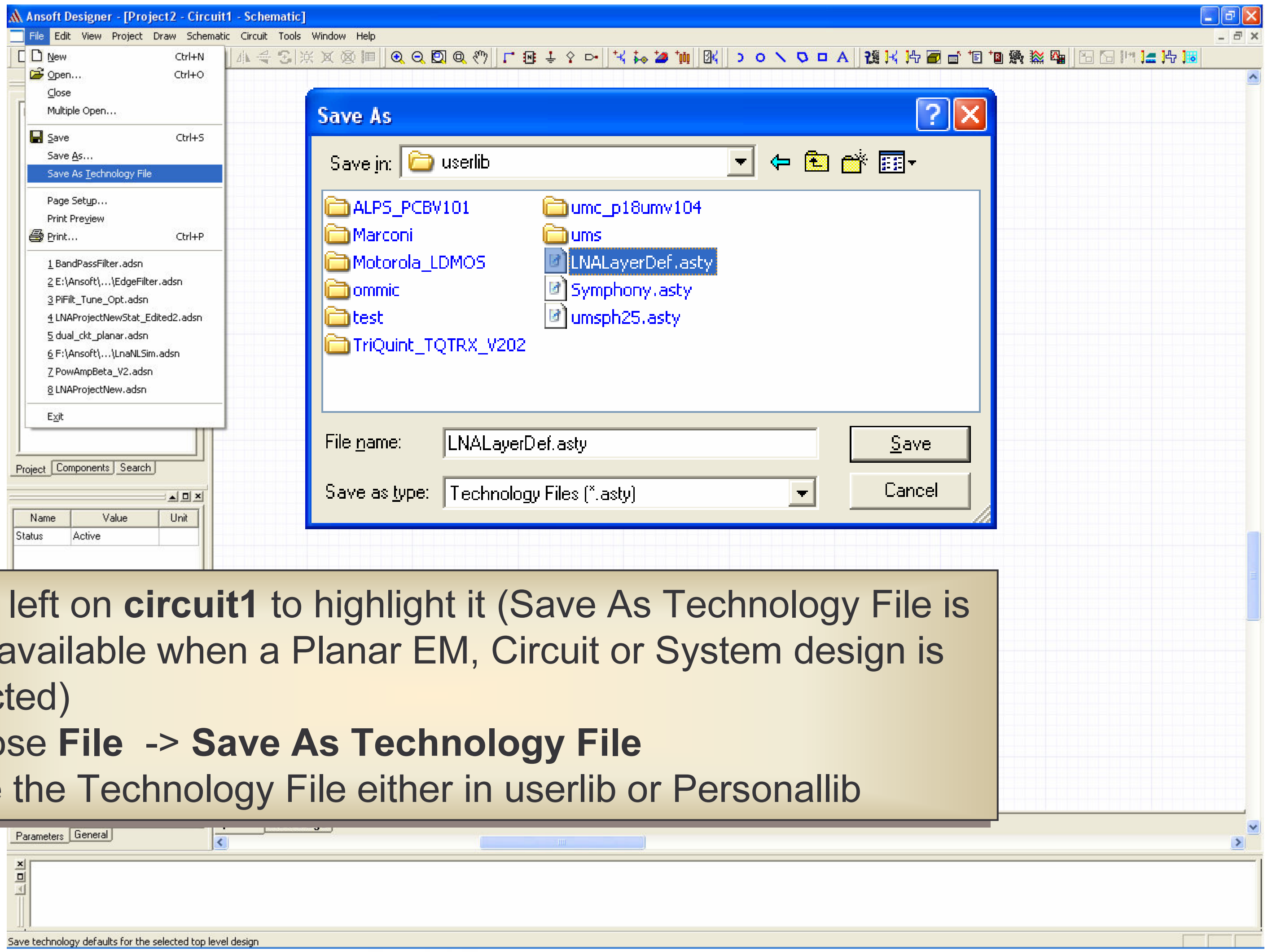
Either of these can be edited by clicking on the Edit buttons, but this will break the connection to the physical stackup.

View Loaded Libraries



- Choose **Tools -> Configure Libraries…**

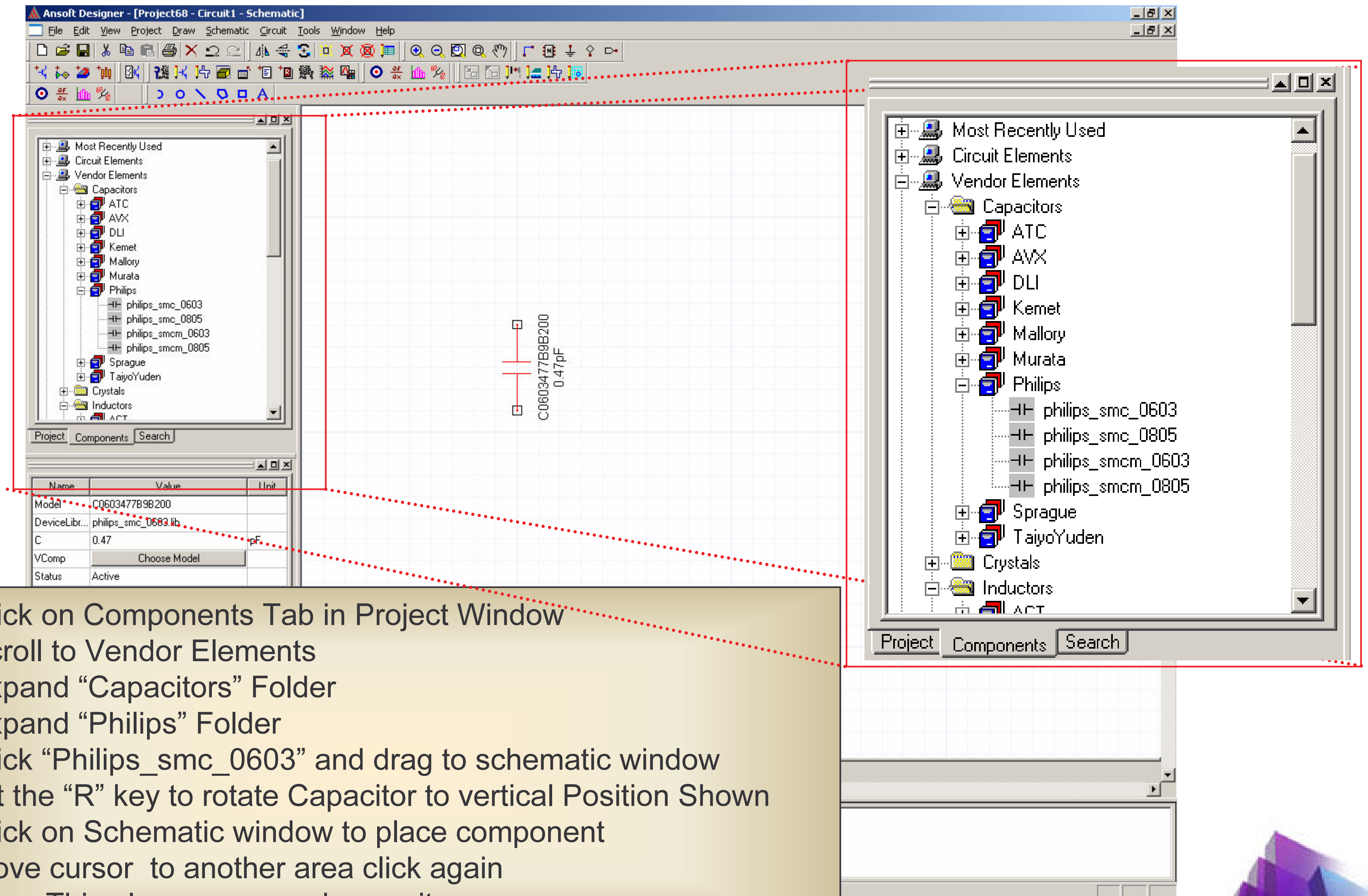
Save Technology File



- 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

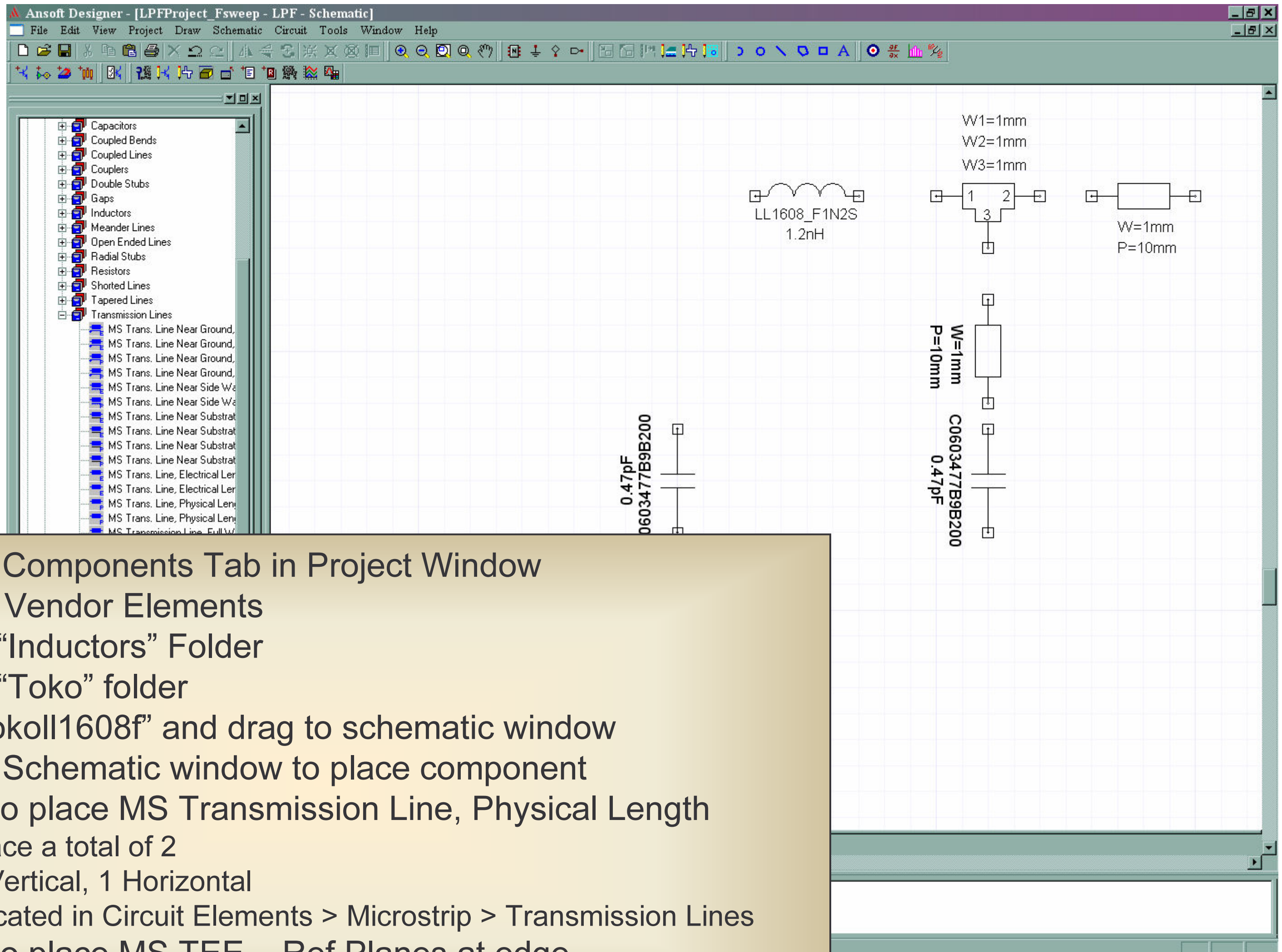
(Vendor Library Capacitors)



- 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

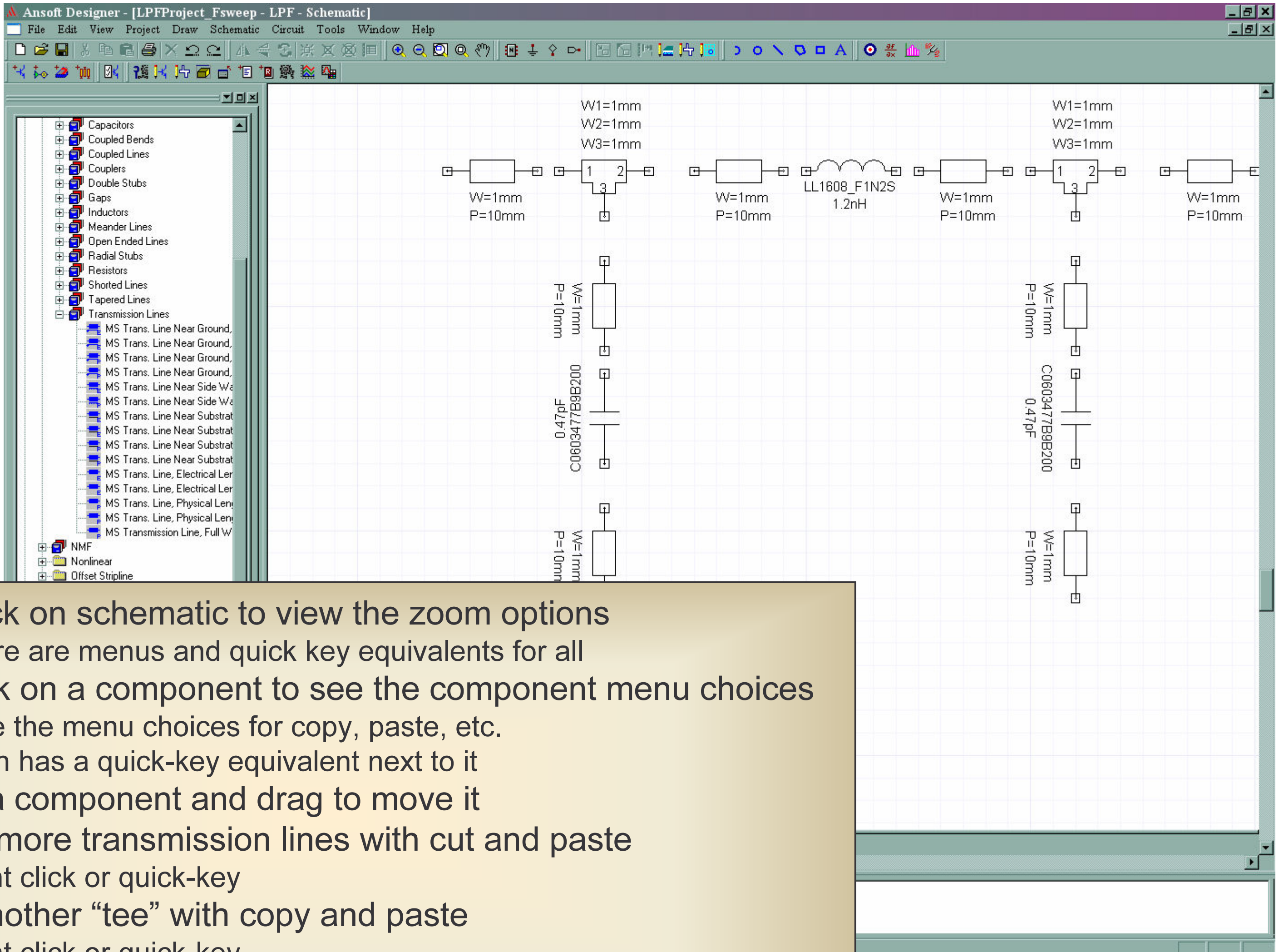
(Vendor Library Inductor, Transmission Lines)



- 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

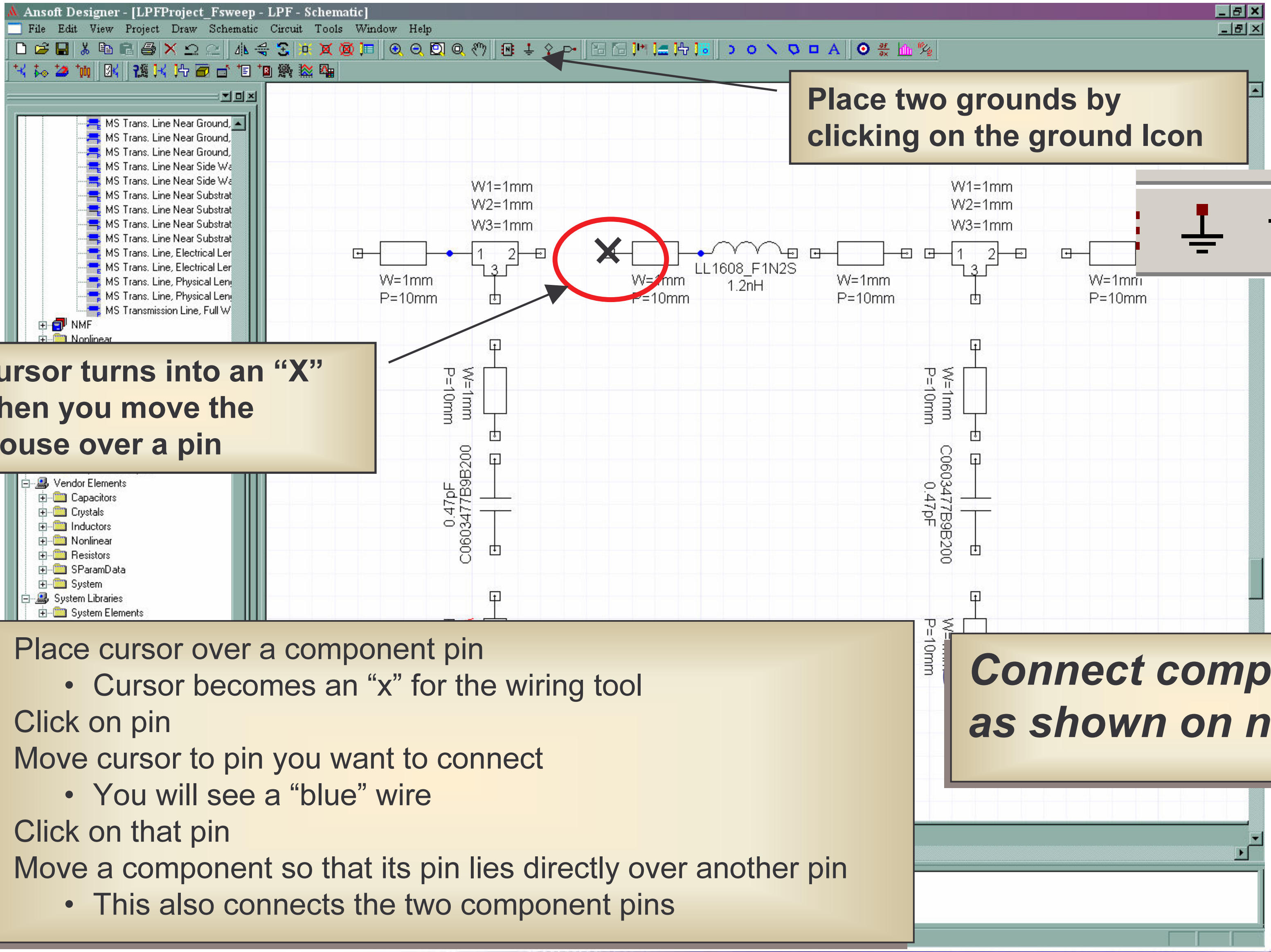
Viewing Window

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



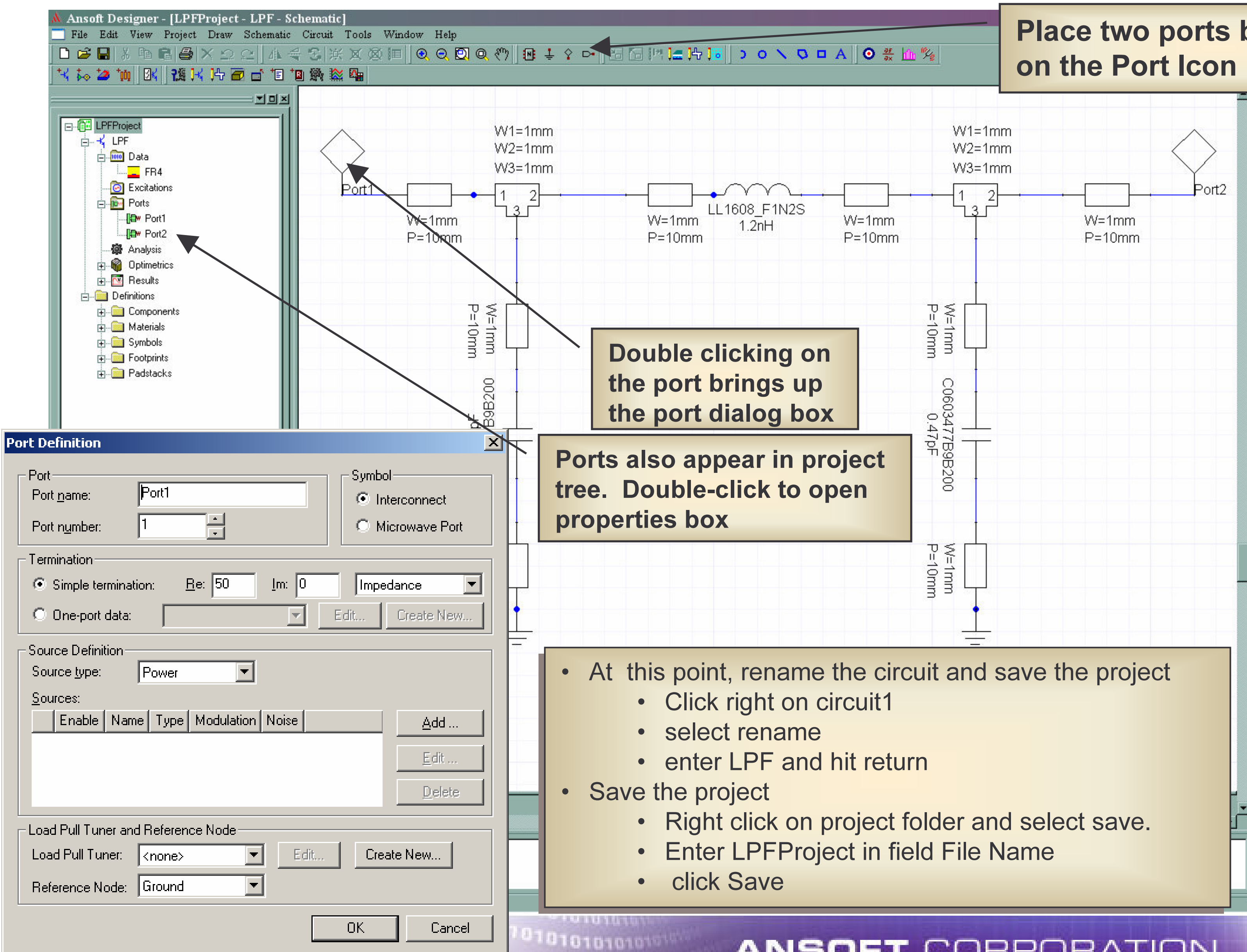
Place two grounds by clicking on the ground Icon

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

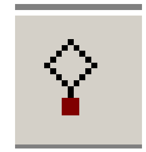
- Place cursor over a component pin
  - Cursor becomes an "x" for the wiring tool
- Click on pin
- Move cursor to pin you want to connect
  - You will see a "blue" wire
- Click on that pin
- Move a component so that its pin lies directly over another pin
  - This also connects the two component pins

Connect components as shown on next slide

Add Ports



Place two ports by clicking on the Port Icon

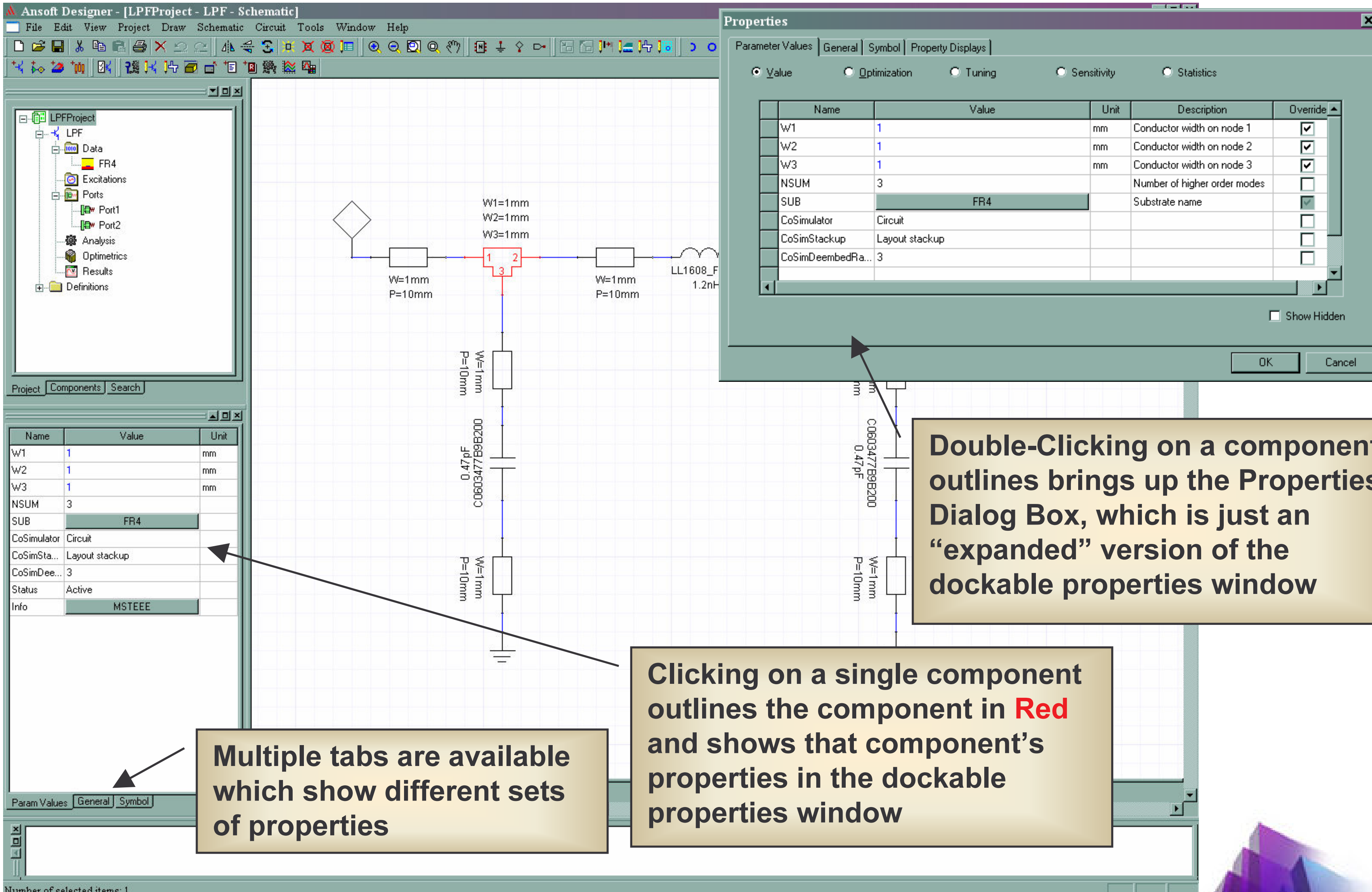


Double clicking on the port brings up the port dialog box

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

Component Properties

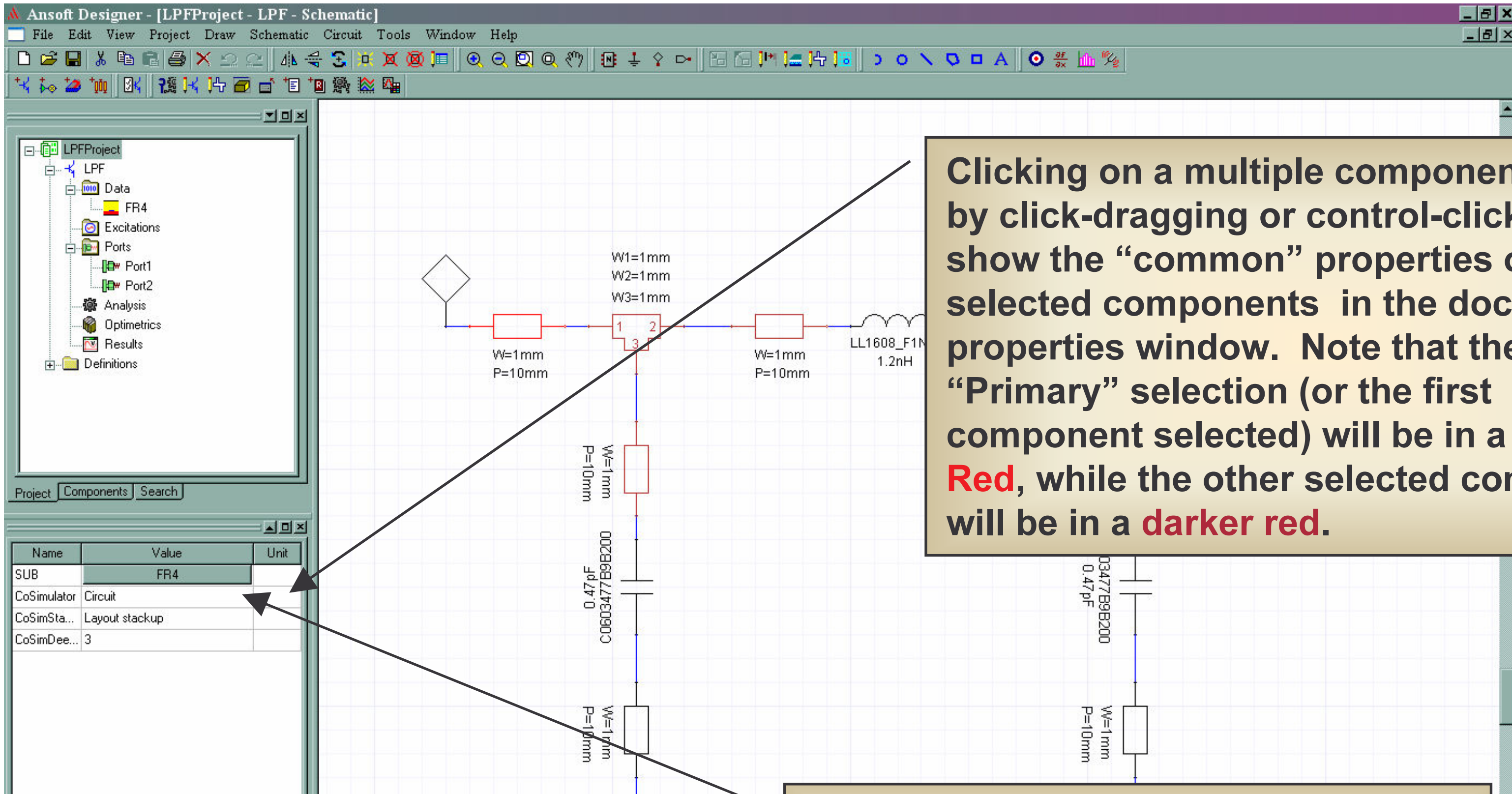


Double-Clicking on a component outlines brings up the Properties Dialog Box, which is just an "expanded" version of the dockable properties window

Clicking on a single component outlines the component in Red and shows that component's properties in the dockable properties window

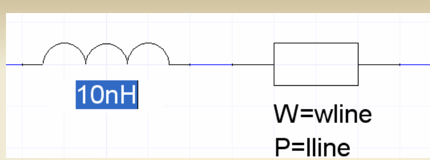
Multiple tabs are available which show different sets of properties

“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.

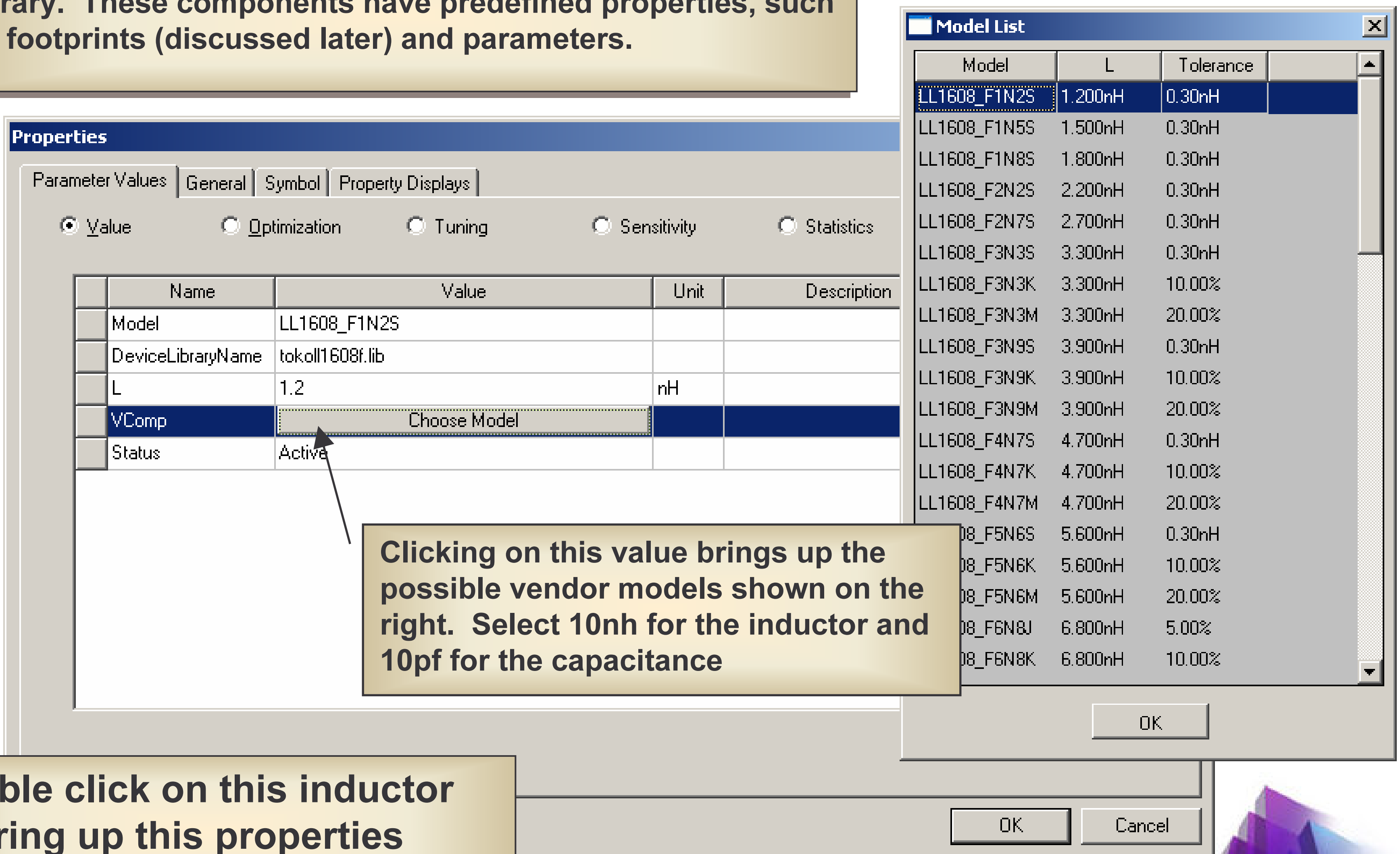
In this schematic, the selected components are 3 MS transmission lines and 1 MS TEE. Substrate is a common parameter. This allows the user to quickly change the “common” values of many components at once.



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

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.

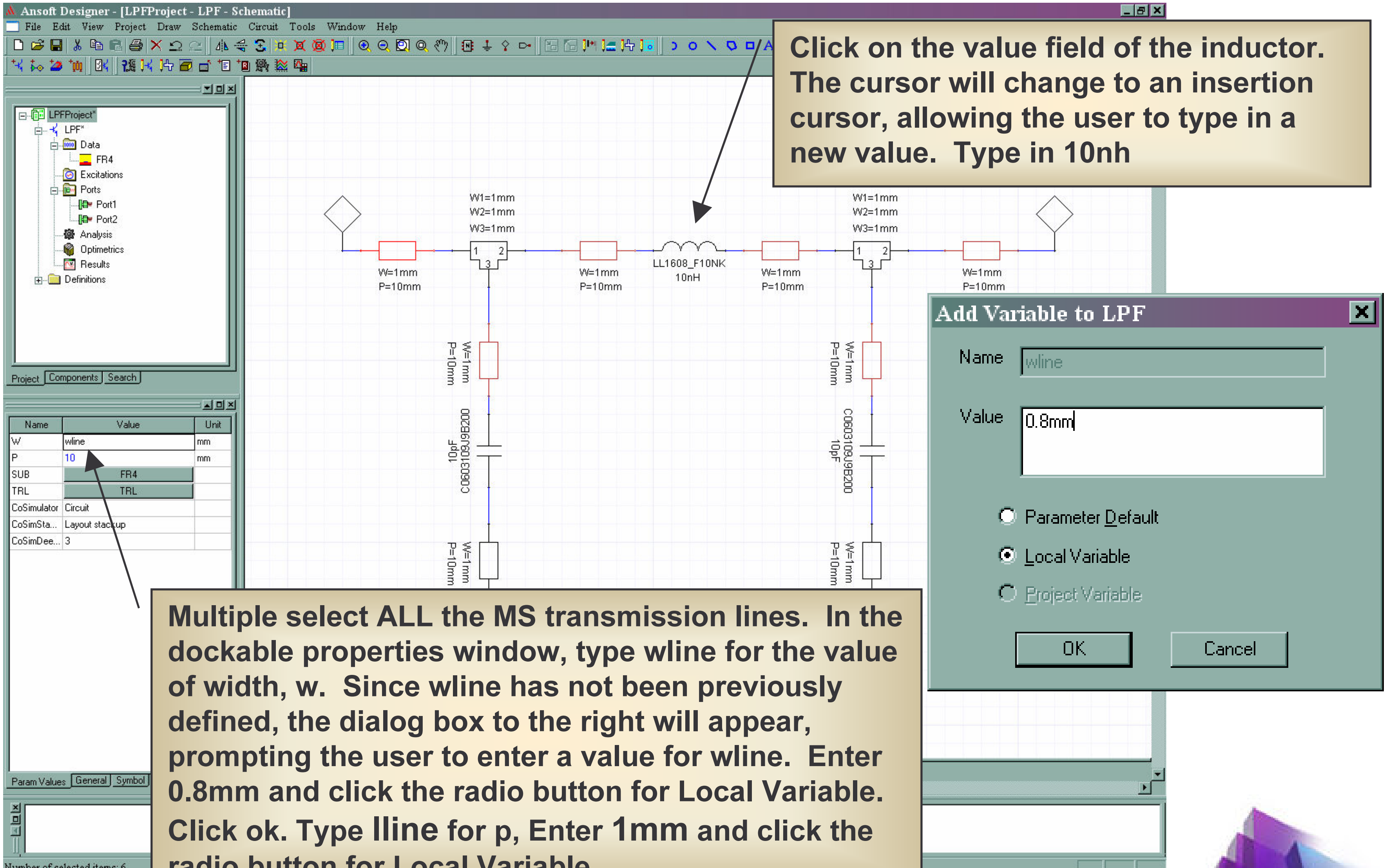


Clicking on this value brings up the possible vendor models shown on the right. Select 10nh for the inductor and 10pf for the capacitance

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

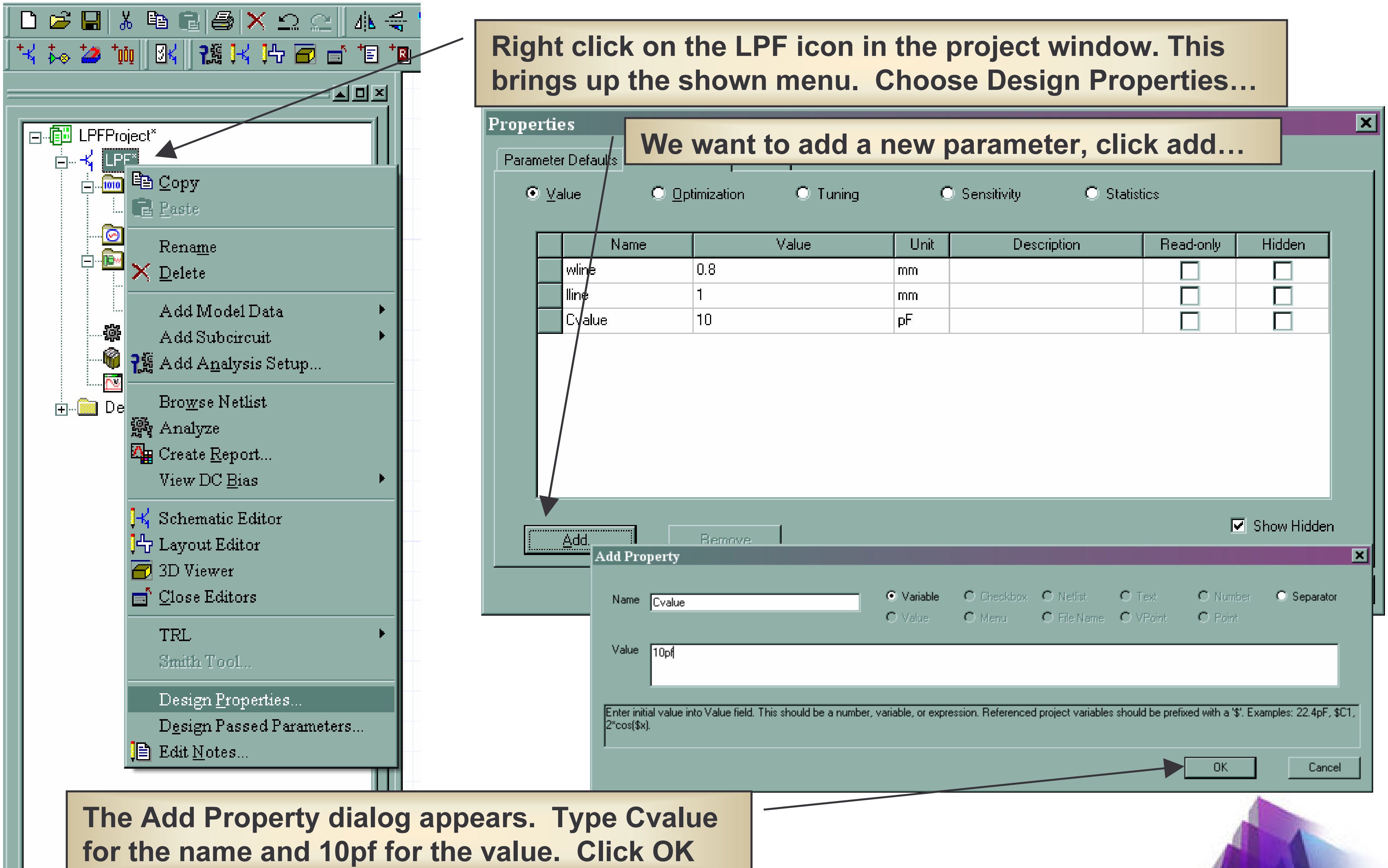
Multiple select ALL the MS transmission lines. In the dockable properties window, type wline for the value of width, w. Since wline has not been previously defined, the dialog box to the right will appear, prompting the user to enter a value for wline. Enter 0.8mm and click the radio button for Local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

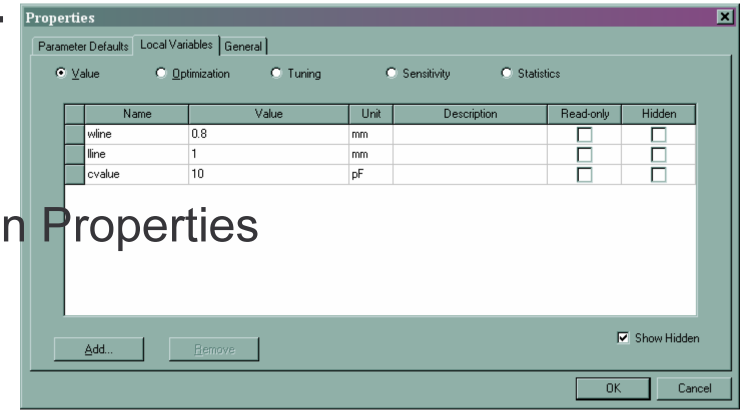
We want to add a new parameter, click add…



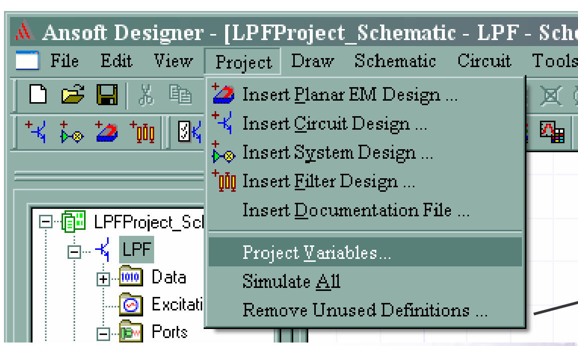
The Add Property dialog appears. Type Cvalue for the name and 10pf for the value. Click OK

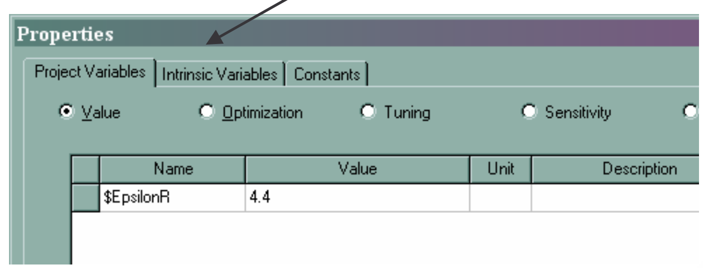
Variable Assignment

- 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



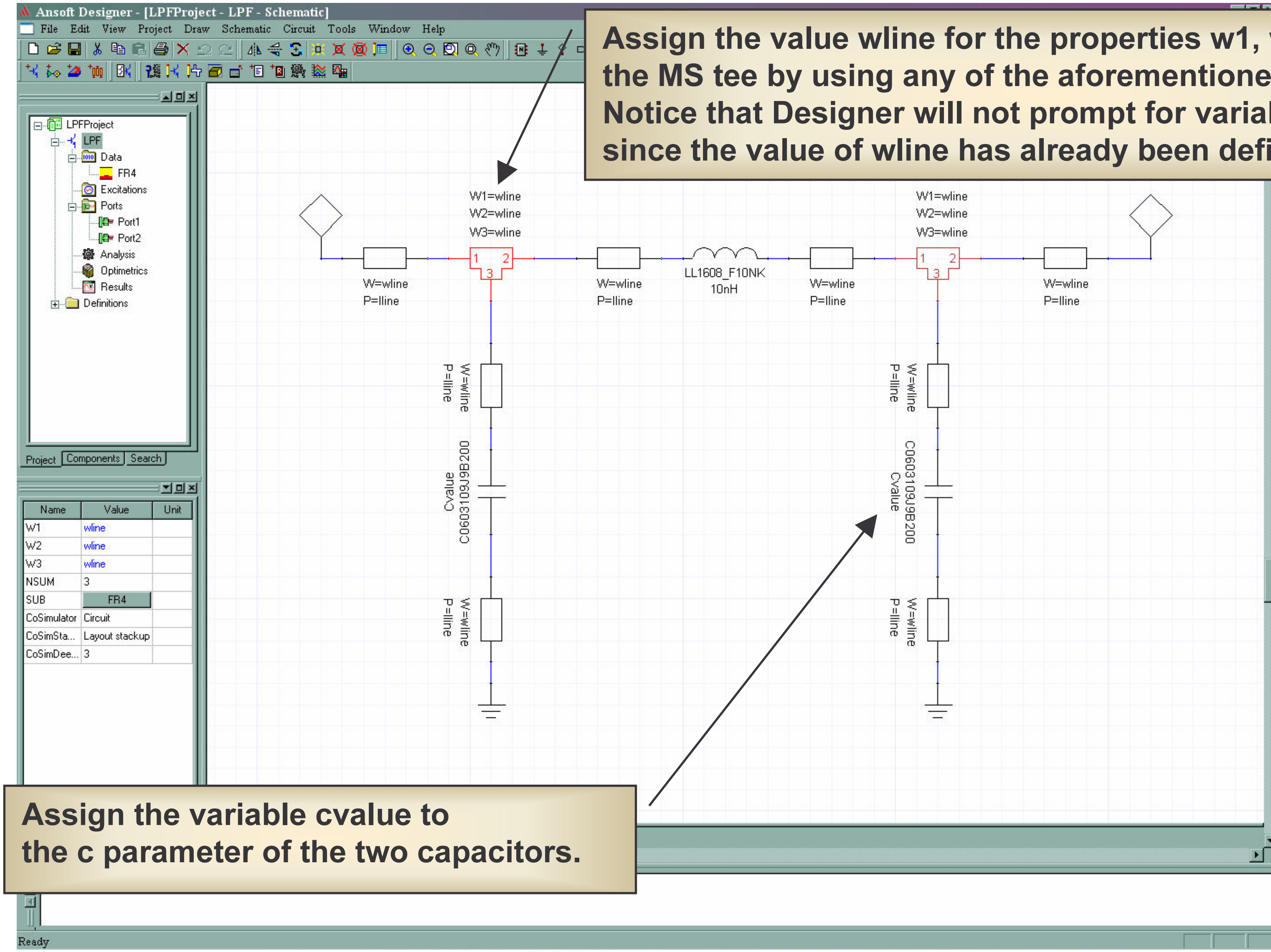
List of predifined variables





Defining Variables

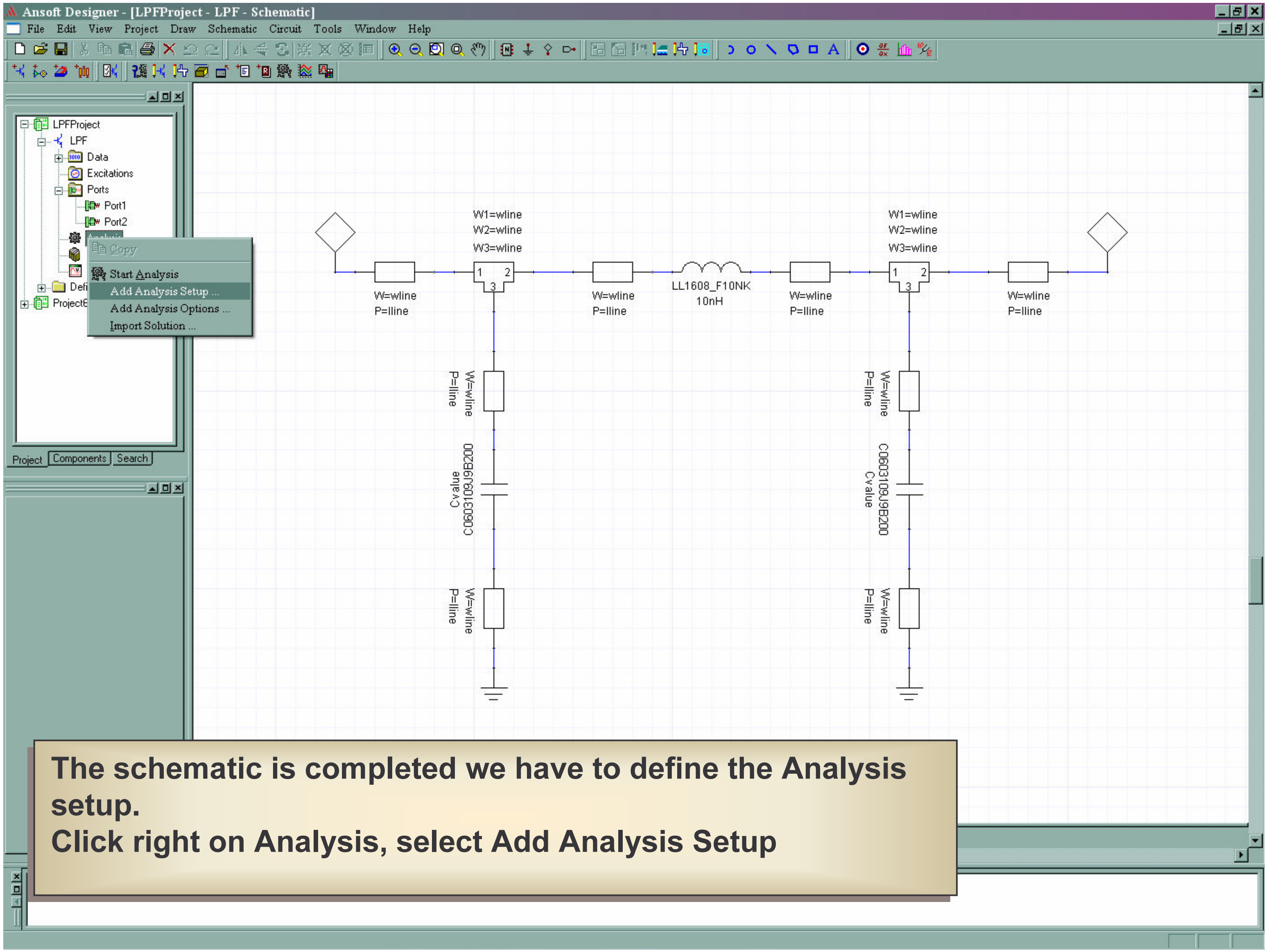
Final Variable Assignment



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

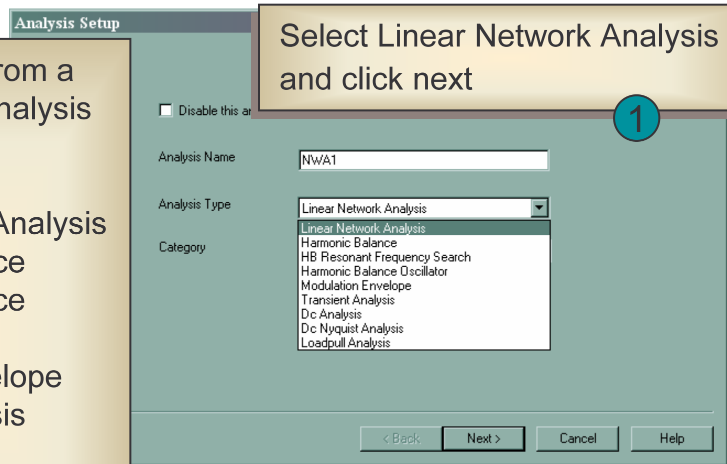


The schematic is completed we have to define the Analysis setup.
Click right on Analysis, select Add Analysis Setup

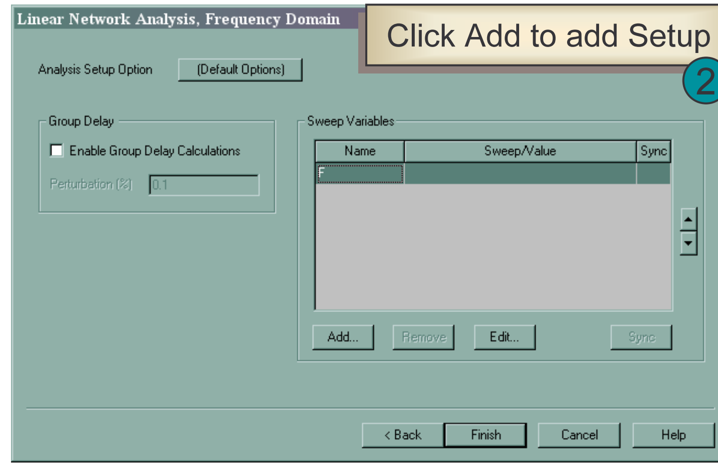
Set Analysis Setup

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

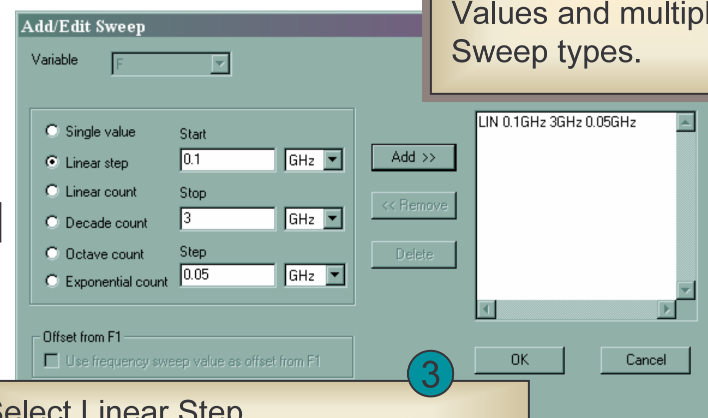


1 Select Linear Network Analysis and click next

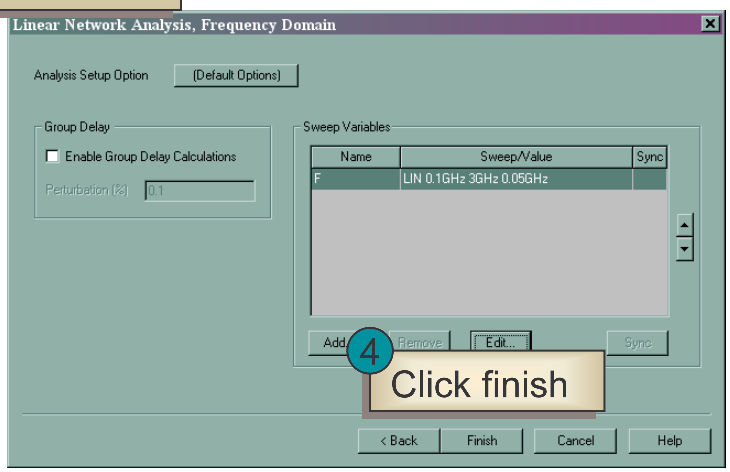


2 Click Add to add Setup

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



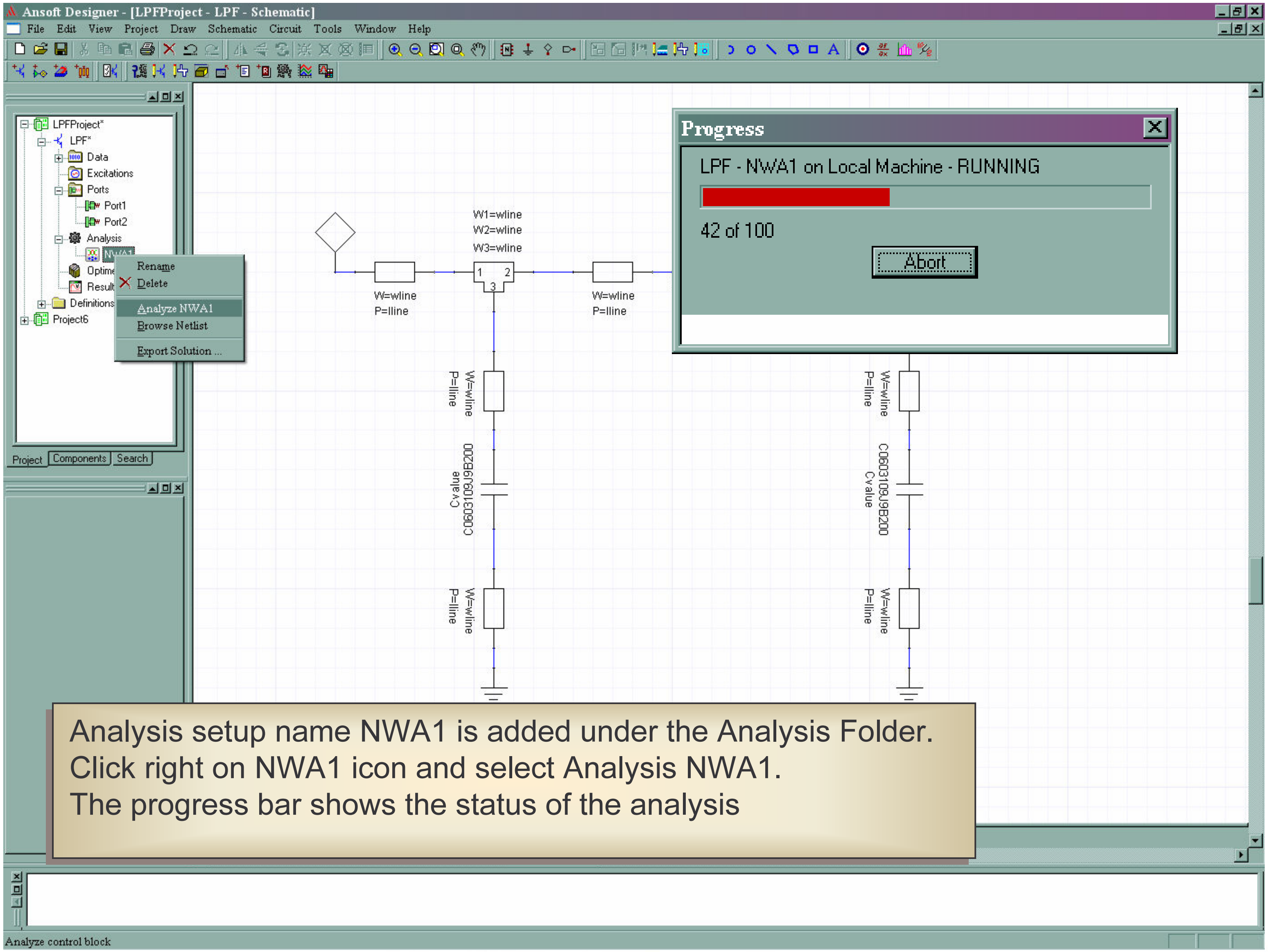
3 Select Linear Step
Enter Start 0.1Ghz Stop 3Ghz Step 0.05Ghz and click Add and OK.



4 Click finish

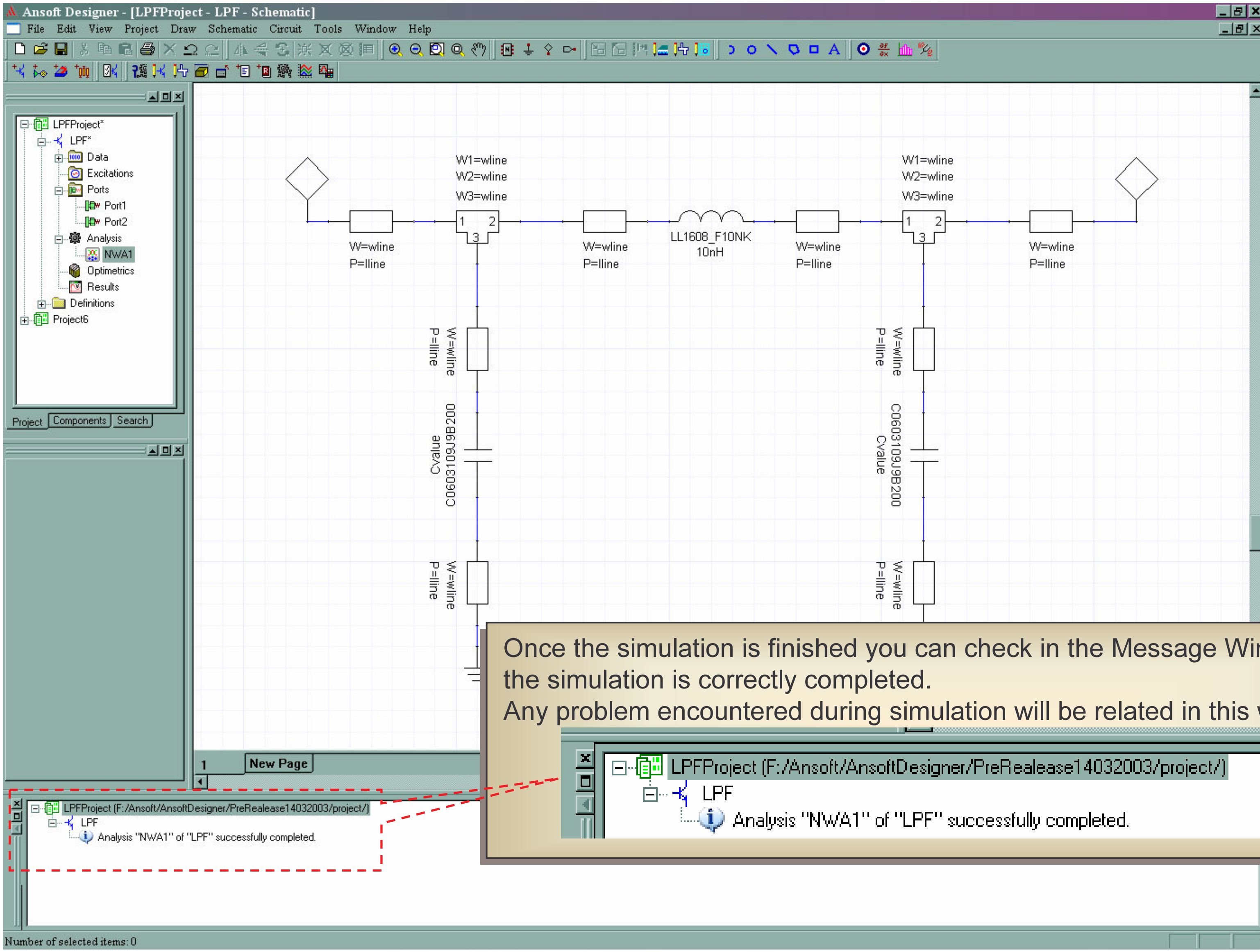
Hit CTRL+S to save the project

Run Analysis



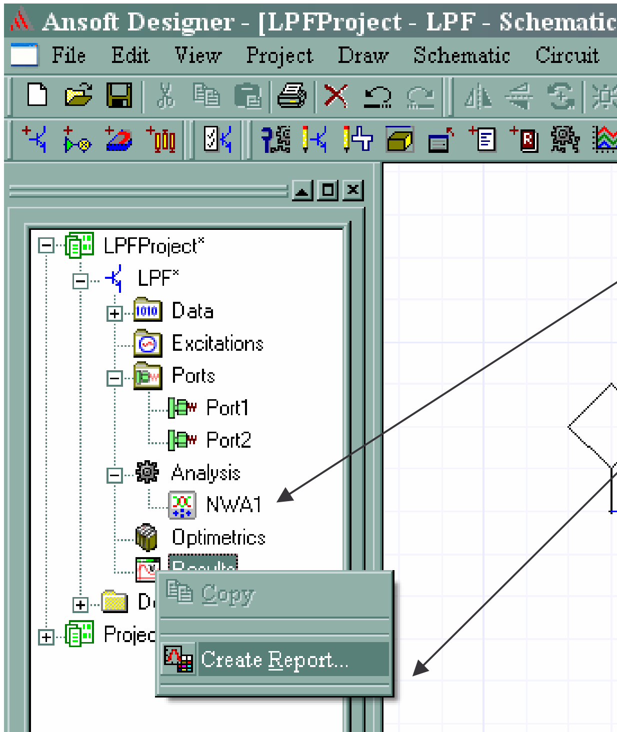
Analysis setup name NWA1 is added under the Analysis Folder.
Click right on NWA1 icon and select Analysis NWA1.
The progress bar shows the status of the analysis

Simulation Successfully Completed



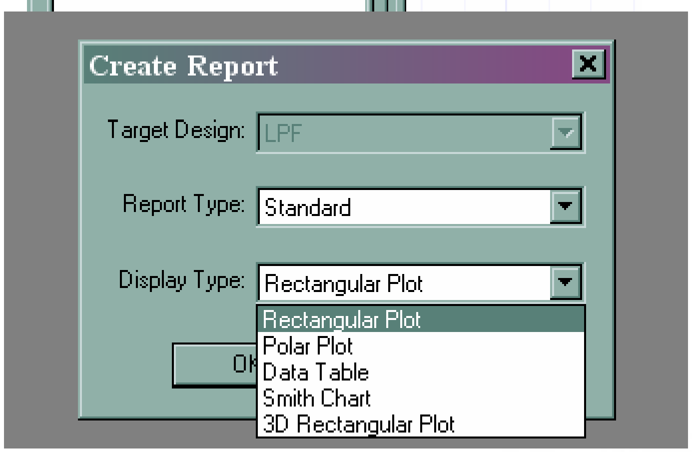
Once the simulation is finished you can check in the Message Window that the simulation is correctly completed.
Any problem encountered during simulation will be related in this window.

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

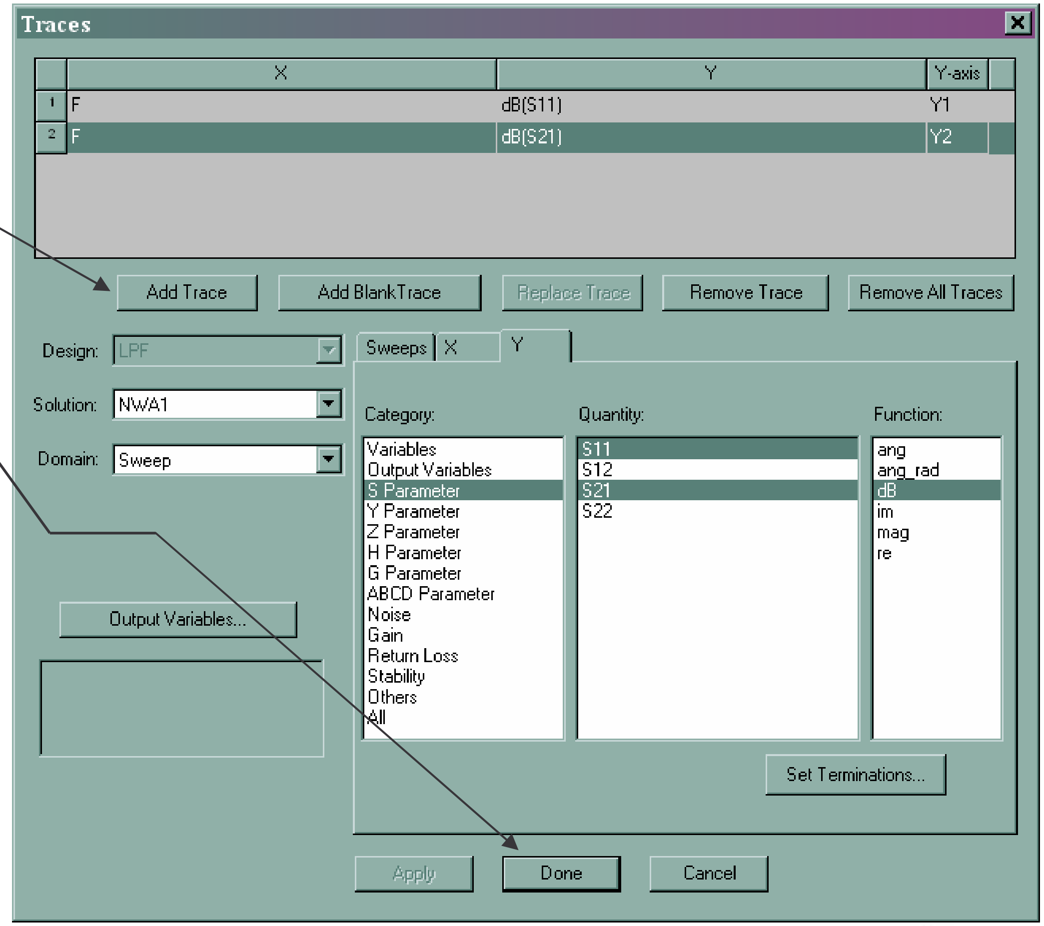
Click on S11 and S21

Click on dB

Select Add Trace

Select Done

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



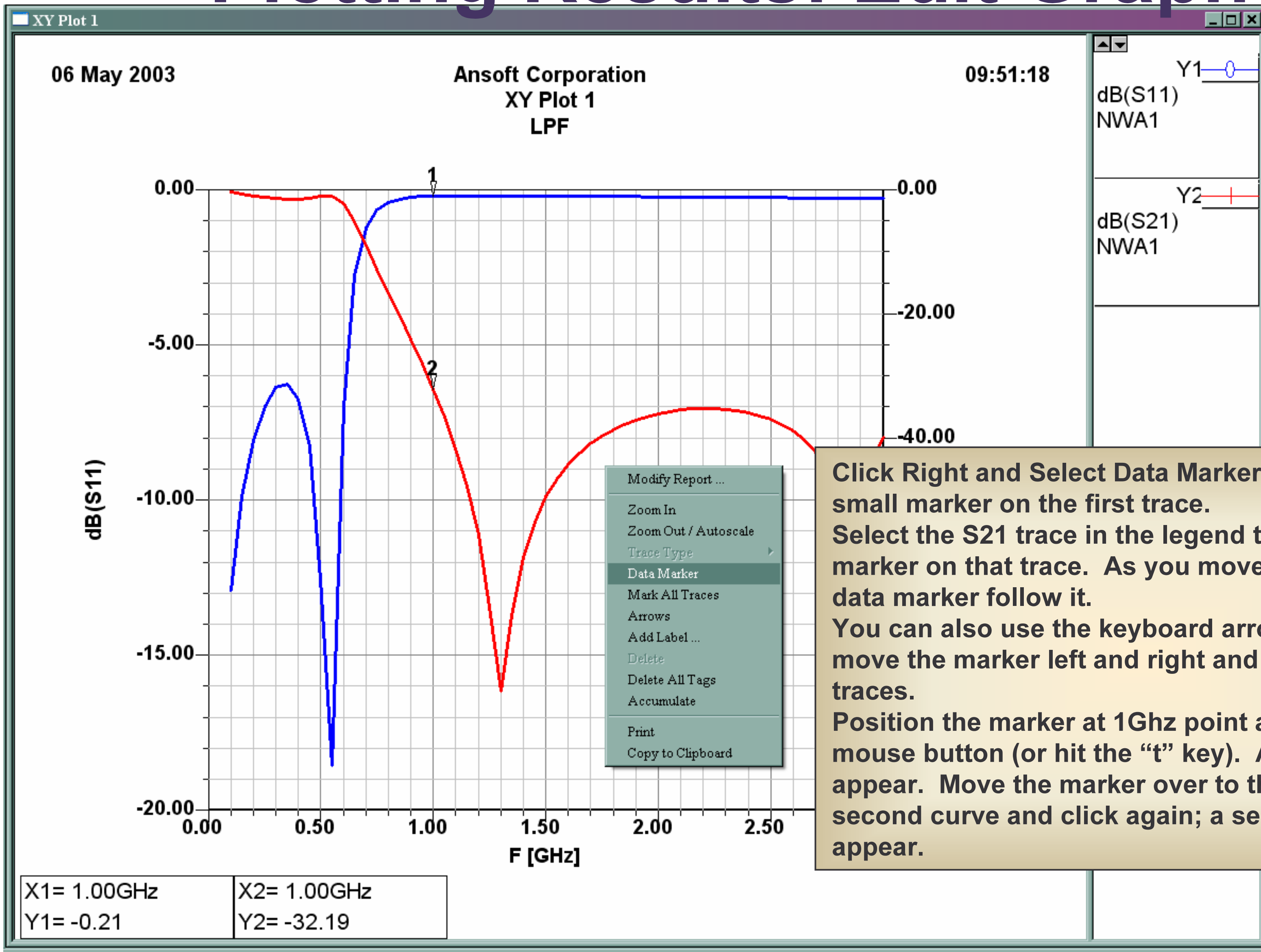
Design – if the project has multiple top-level designs, select the desired design

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

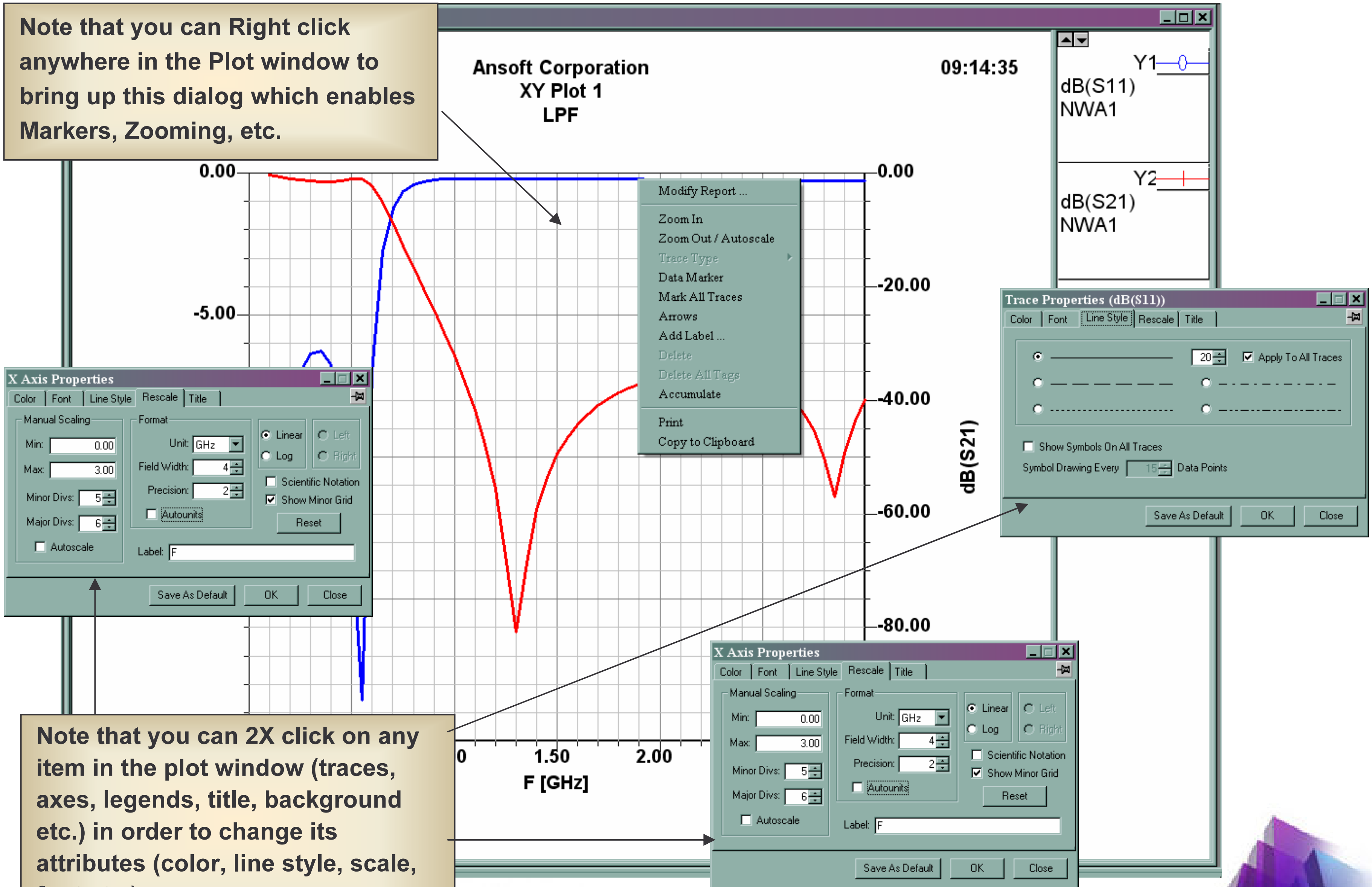
Plotting Results: Edit Graph



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

There are also a number of operations available from the right mouse popup menu from within the graph:

Modify Report – show the Reporter dialog for this graph

Zoom In/Out/Autoscale – view a magnified or original graph

Trace Type – Enabled when a trace is selected. Modify the method of viewing the trace.

Data Marker – Put a tag on the trace to mark a specific point

Mark All Traces – Show data points for all traces at once

Arrows – Display arrows on the trace to show increasing calculation index

Add Label – Put text on the trace

Delete – Enabled when a trace is selected to delete the trace

Delete All Tags – Delete the marker tags

Accumulate – Keep prior traces on the graph up to the Accumulate Depth (settable in Tools>Options>Report 2D Options)

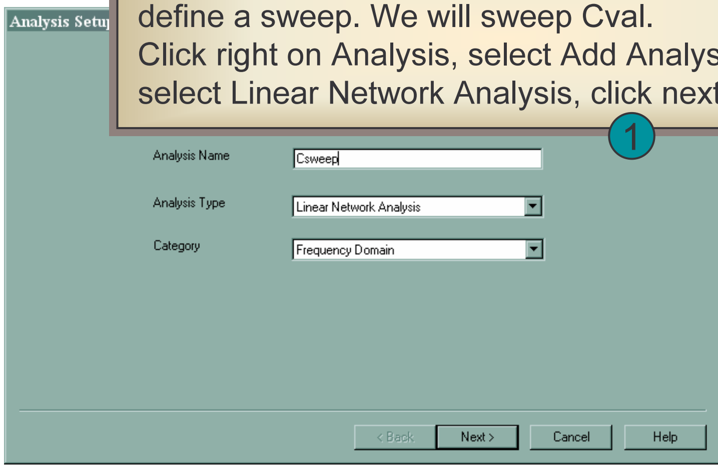
Print – Send to printer

Copy to Clipboard – Put a copy of the graph on the clipboard to paste into other applications

Parametric Sweep

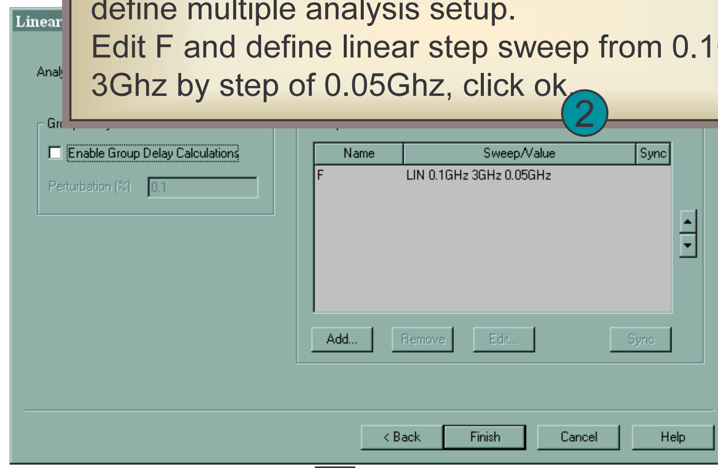
In Ansoft Designer you can use any variable to define a sweep. We will sweep Cval.
Click right on Analysis, select Add Analysis Setup, select Linear Network Analysis, click next.

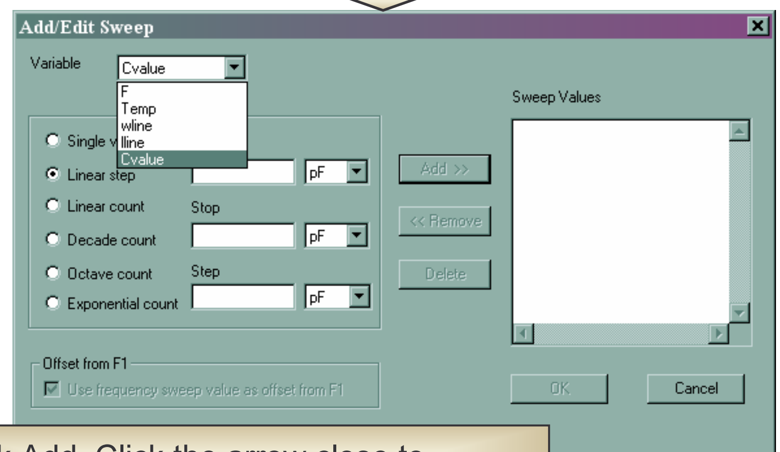
1



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.

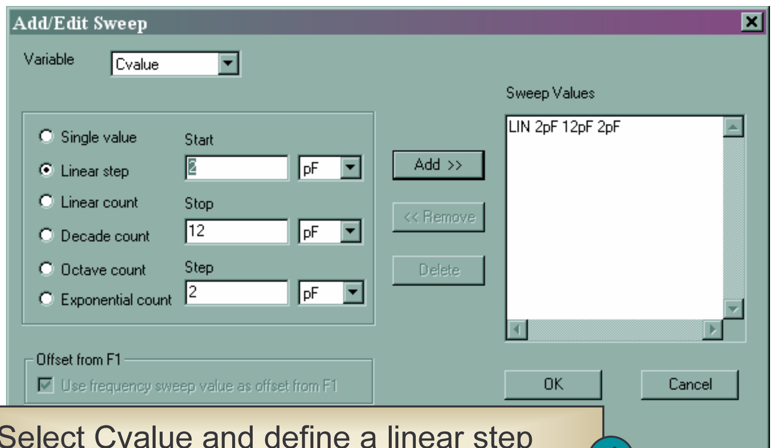
2





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

3

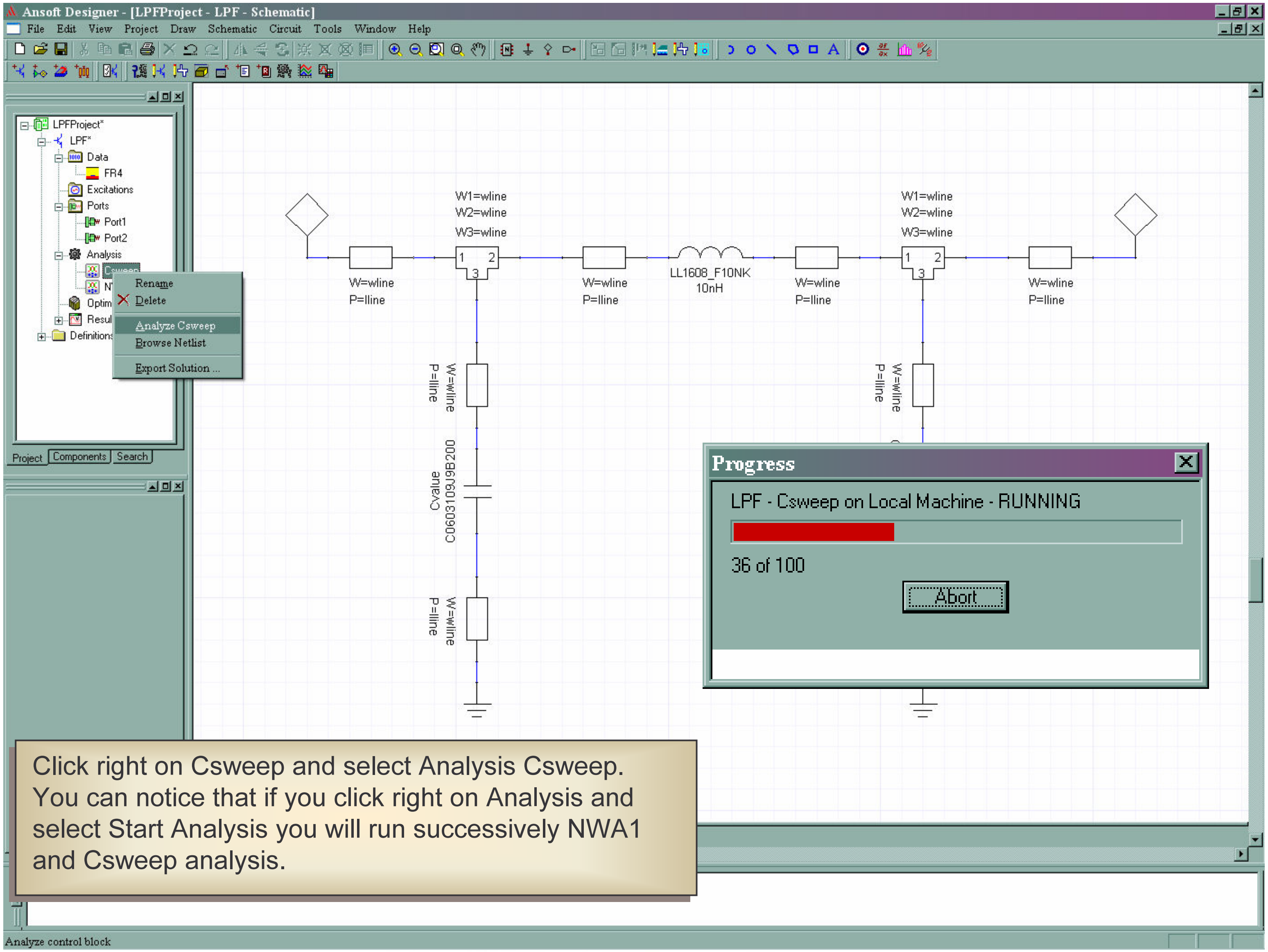


Select Cvalue and define a linear step from 2pf to 12pf by step of 2pf.
Click ok and finish.

4

Hit CTRL+S to save the project

Run Analysis Csweep

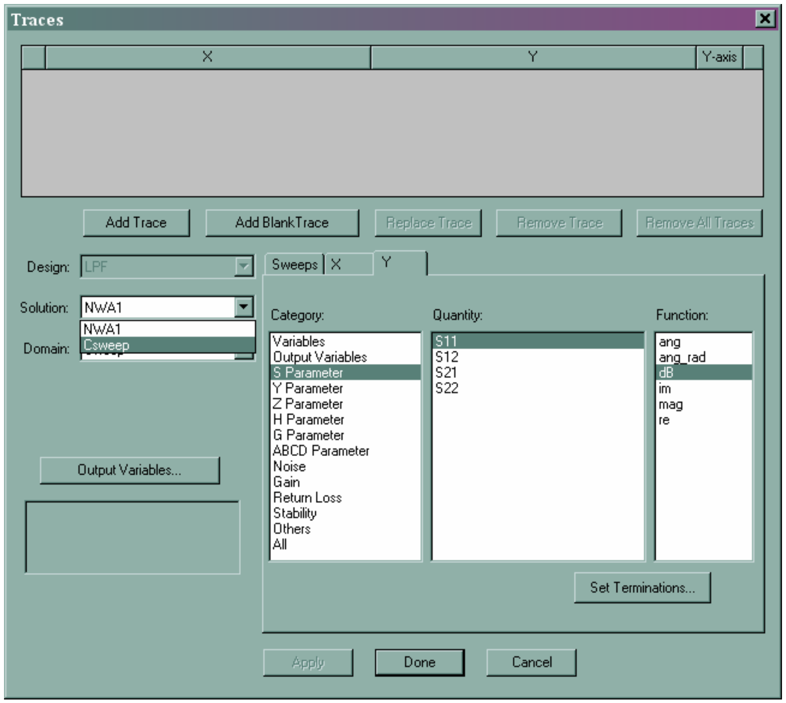


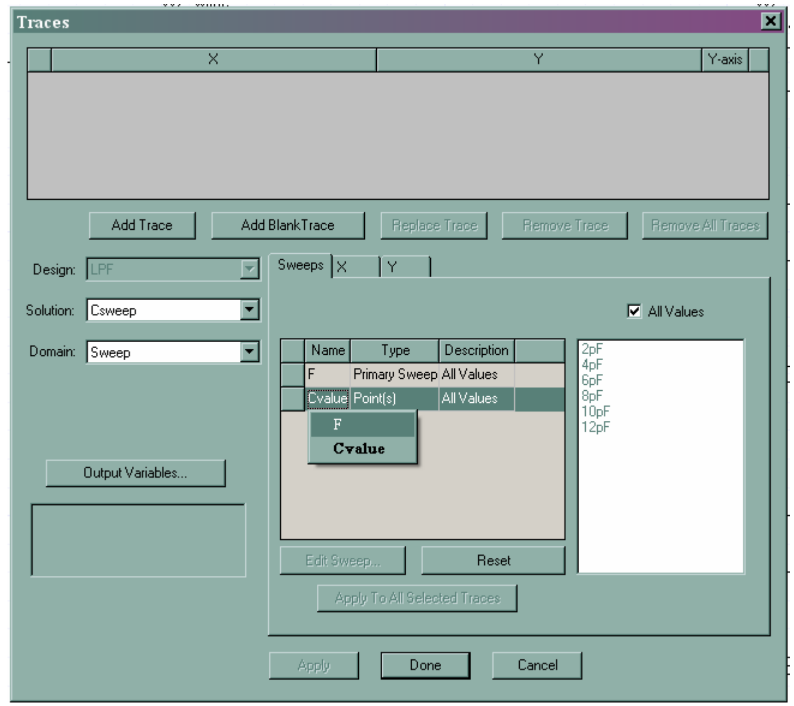
Click right on Csweep and select Analysis Csweep. You can notice that if you click right on Analysis and select Start Analysis you will run successively NWA1 and Csweep analysis.

Plotting Results

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

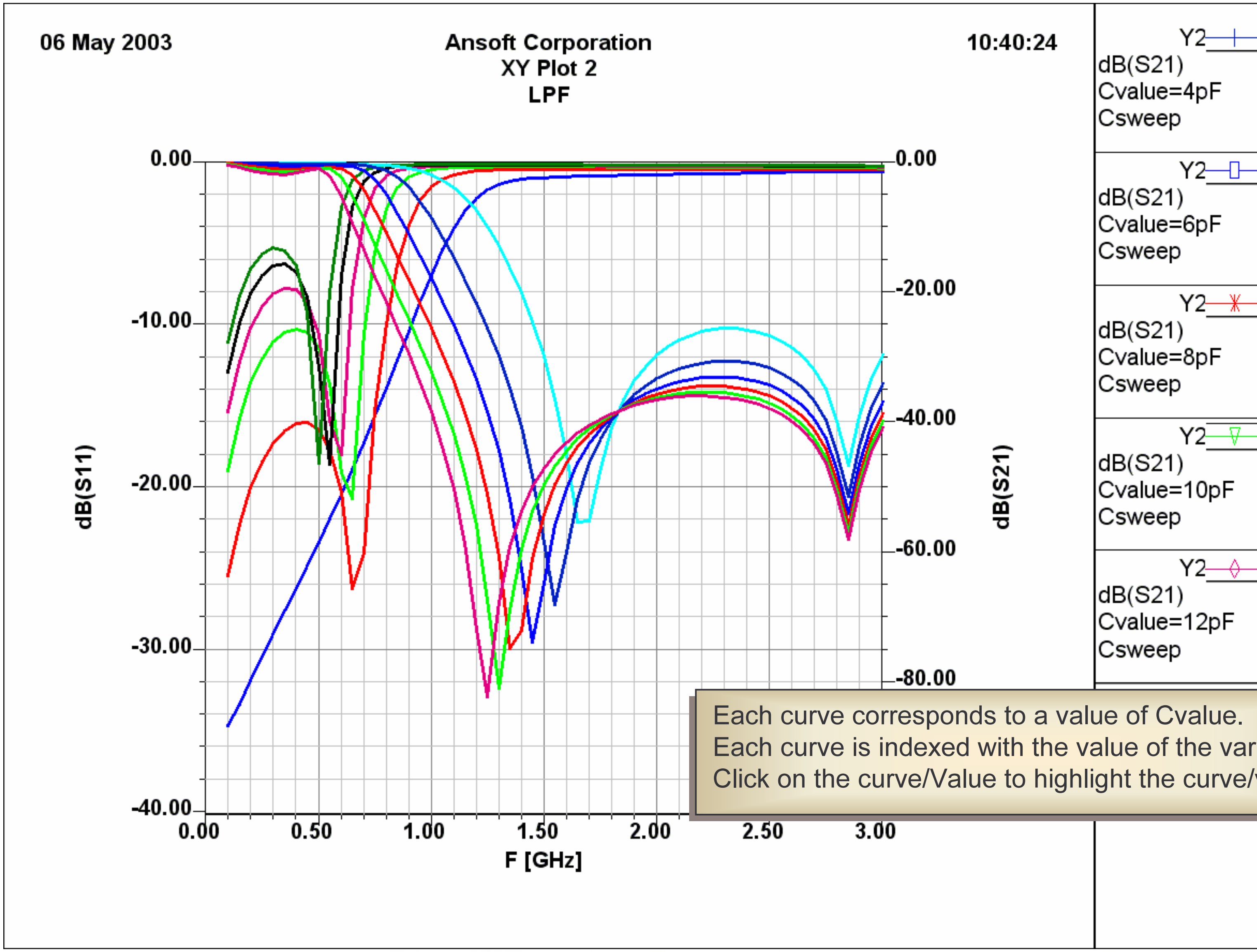
Clicking on one of the sweep variable shows the swept values, you can select all the value, one value or several value.
Use the shift and CTRL key to select multiple value.





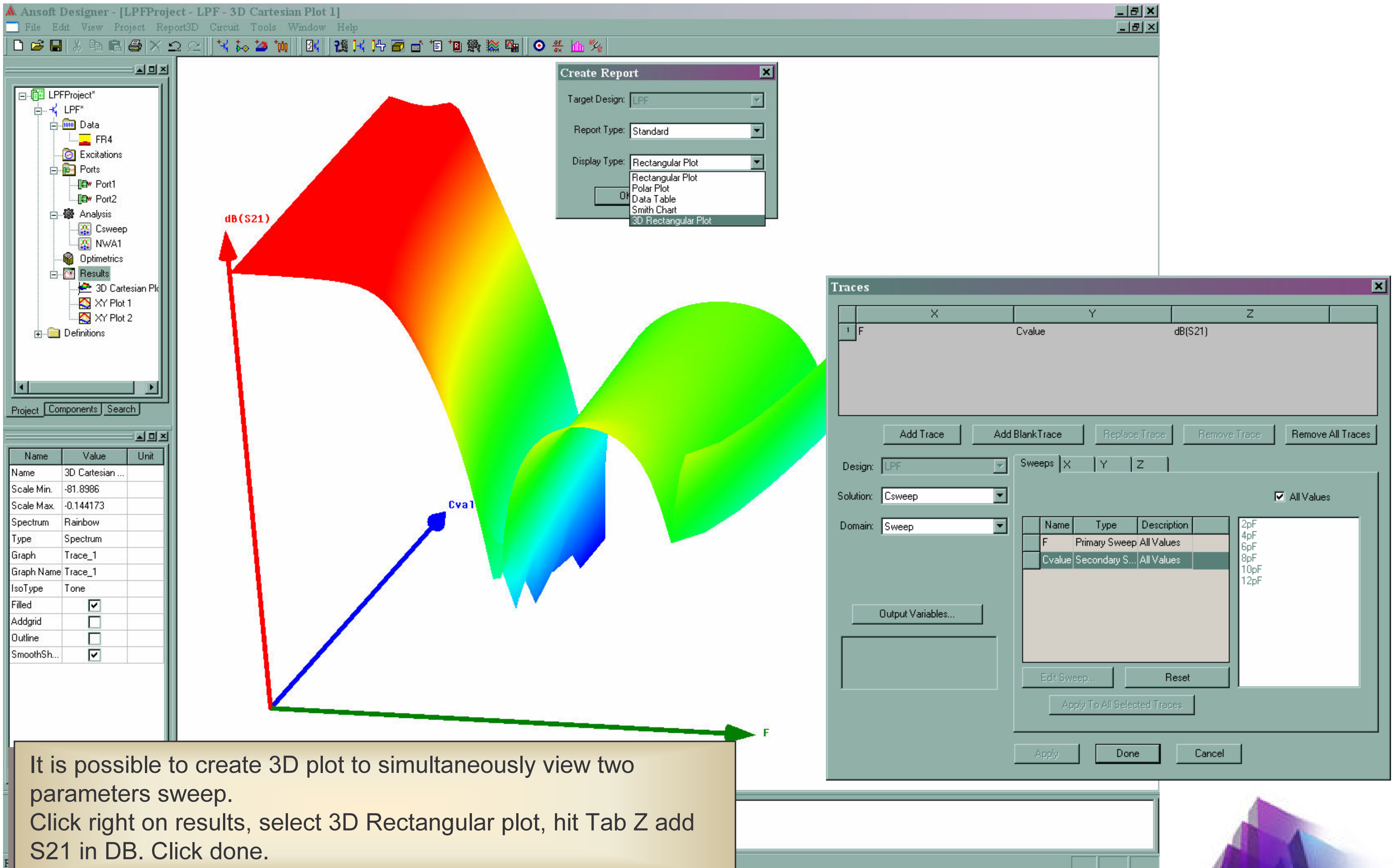
Clicking on the name of the variable you can change the sweep order.
Hit Tab Y and add S21 in DB, Click Done

Plotting Results: Edit Graph



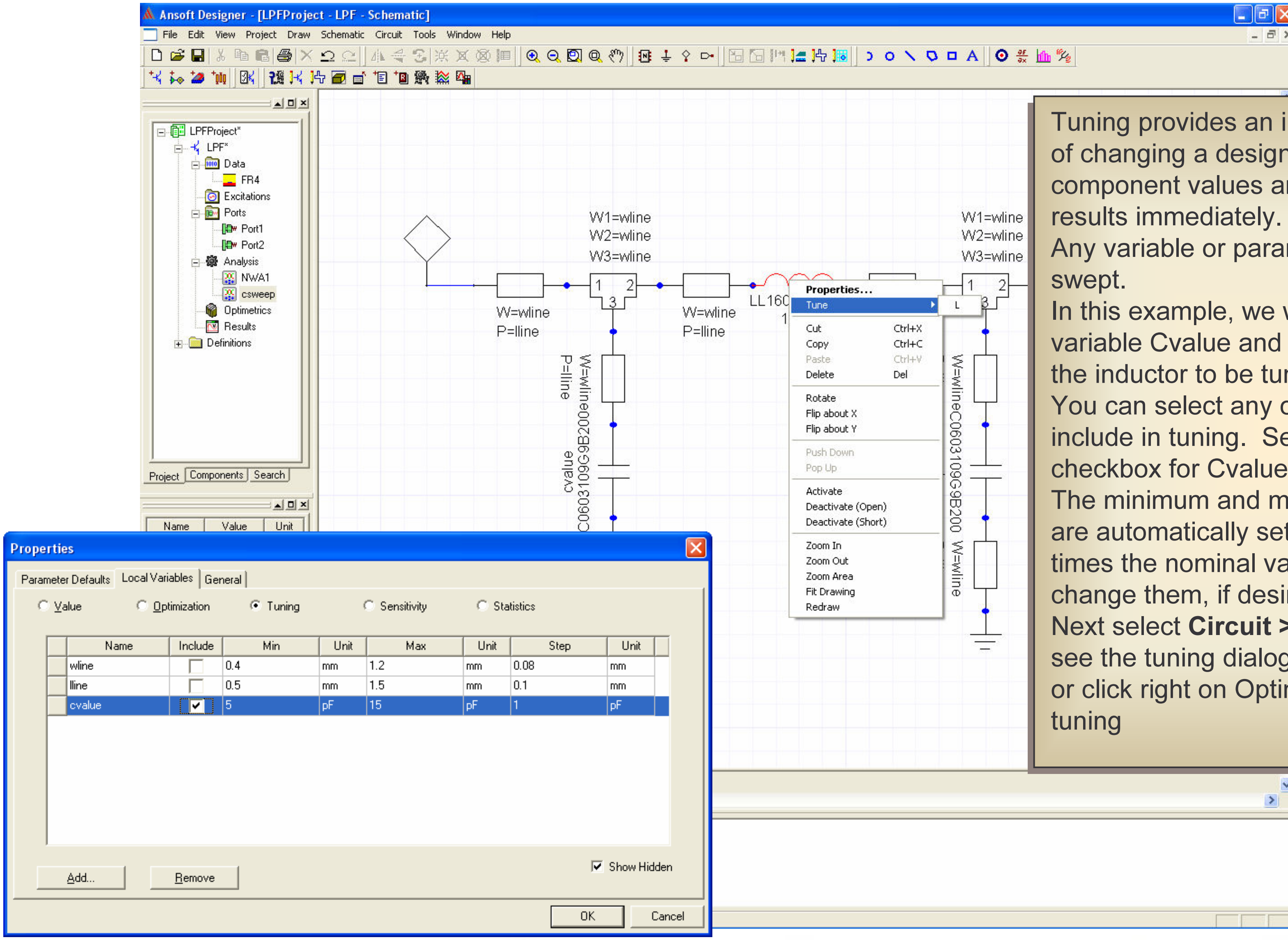
Each curve corresponds to a value of Cvalue.
Each curve is indexed with the value of the variable
Click on the curve/Value to highlight the curve/value

Plotting Results: 3D Plot



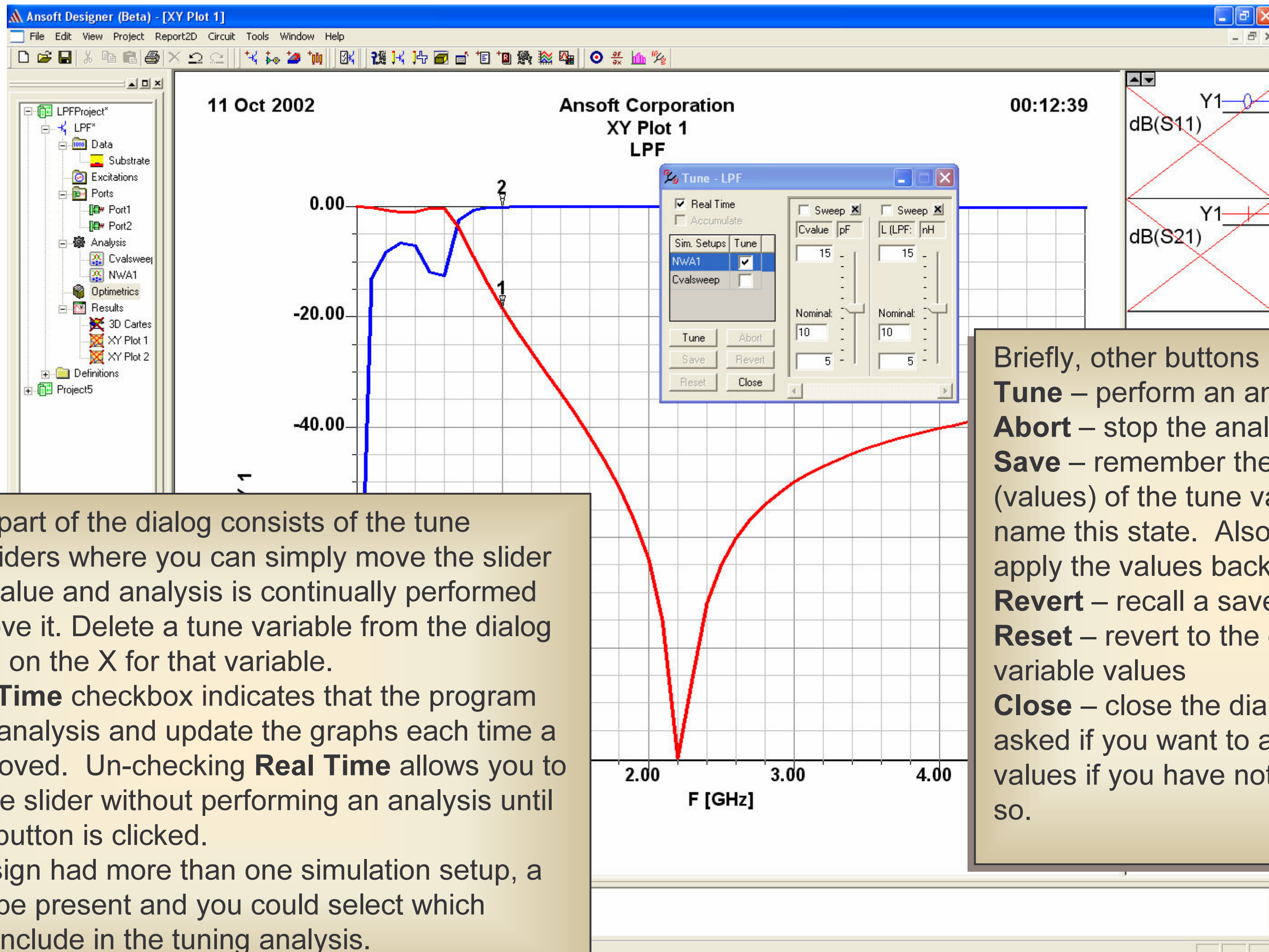
It is possible to create 3D plot to simultaneously view two parameters sweep.
Click right on results, select 3D Rectangular plot, hit Tab Z add S21 in DB. Click done.

Tuning



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 ½ and 1 ½ 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

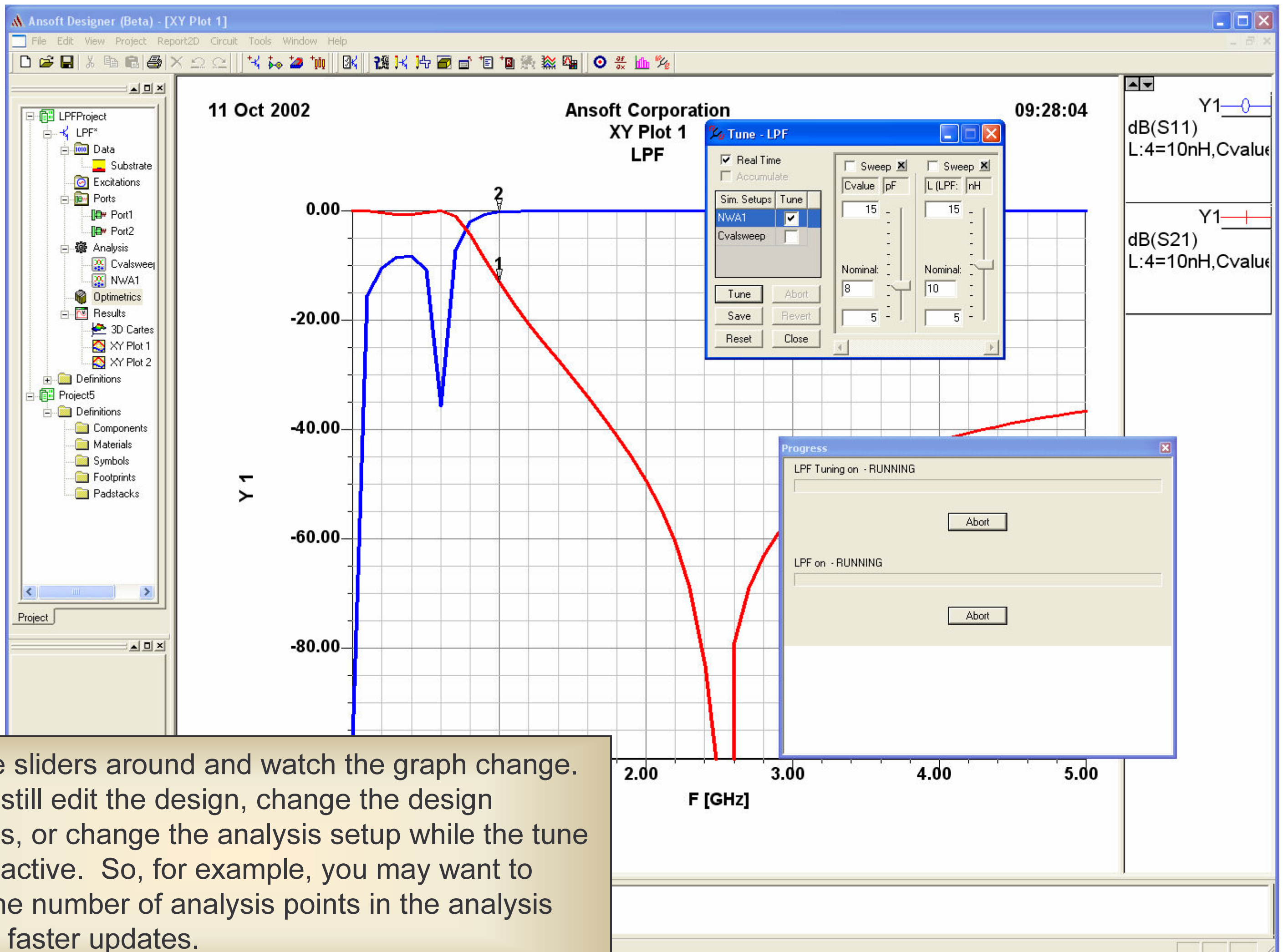
Tuning



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.

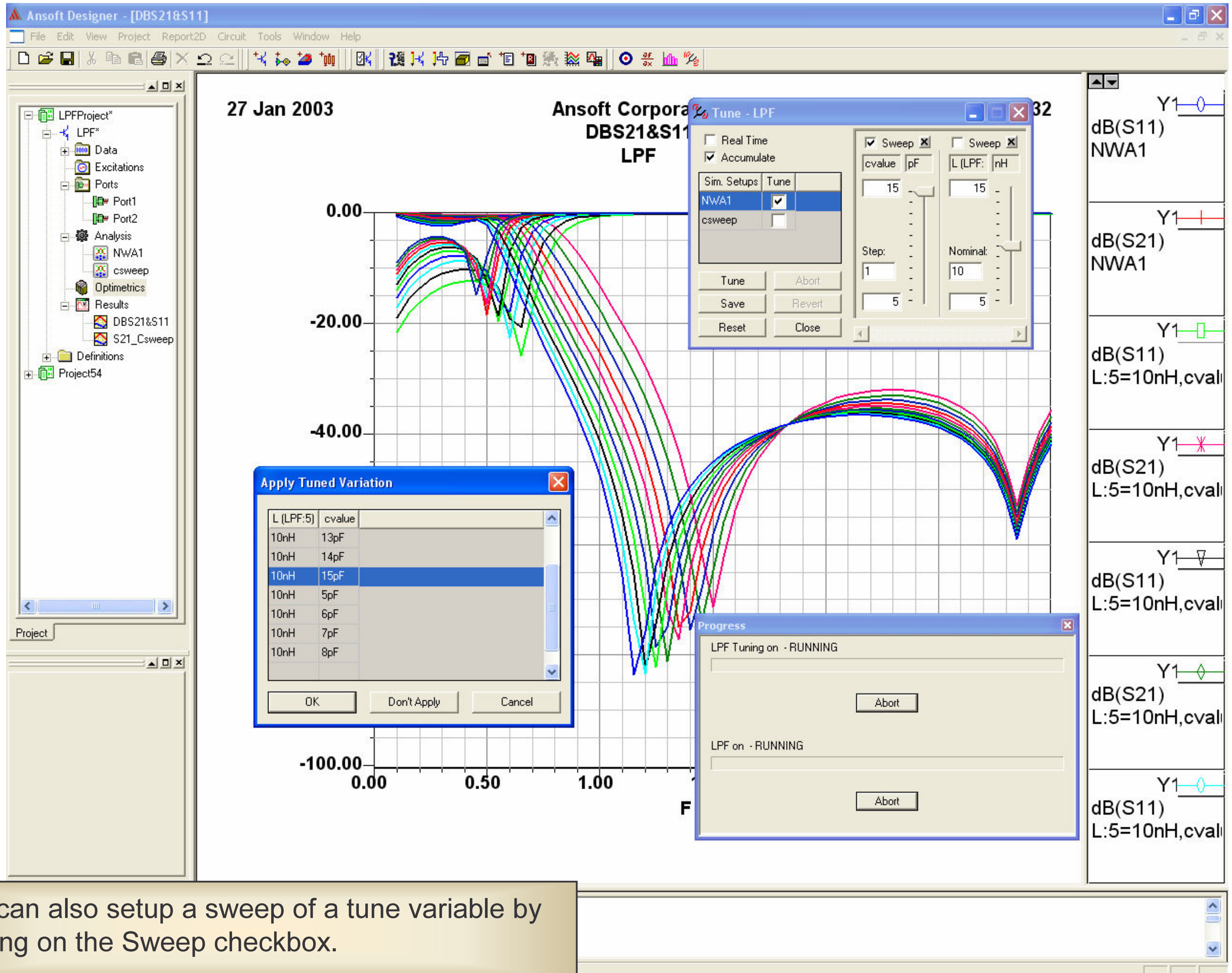
Briefly, other buttons include:
Tune – perform an analysis
Abort – stop the analysis
Save – remember the current state (values) of the tune variables and name this state. Also, optionally apply the values back to the design.
Revert – recall a saved state
Reset – revert to the original tune variable values
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



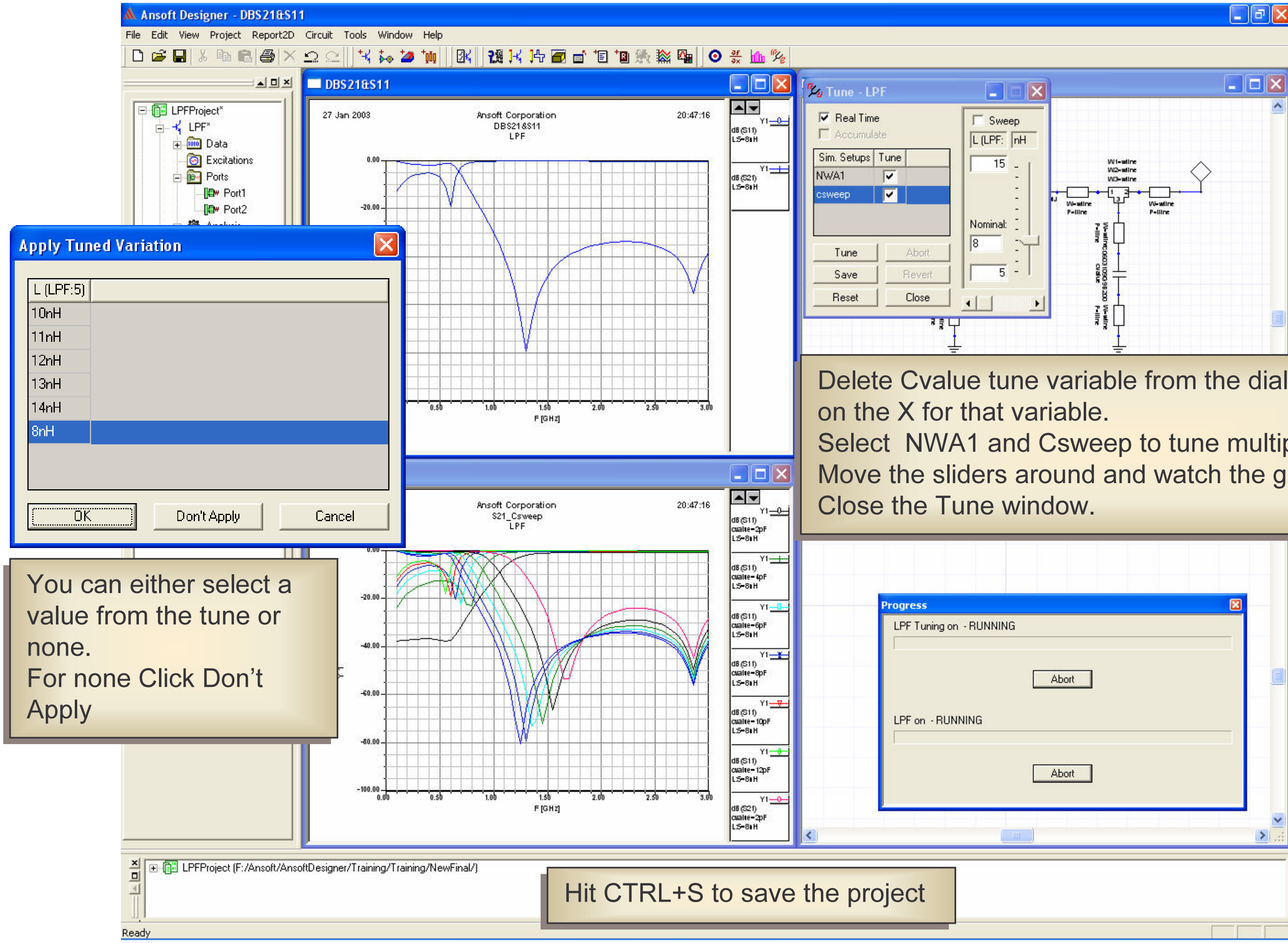
Move the sliders around and watch the graph change. You can still edit the design, change the design properties, or change the analysis setup while the tune dialog is active. So, for example, you may want to reduce the number of analysis points in the analysis setup for faster updates.

Tuning: Accumulate



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

Tuning: Multiple Analysis Setup



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.

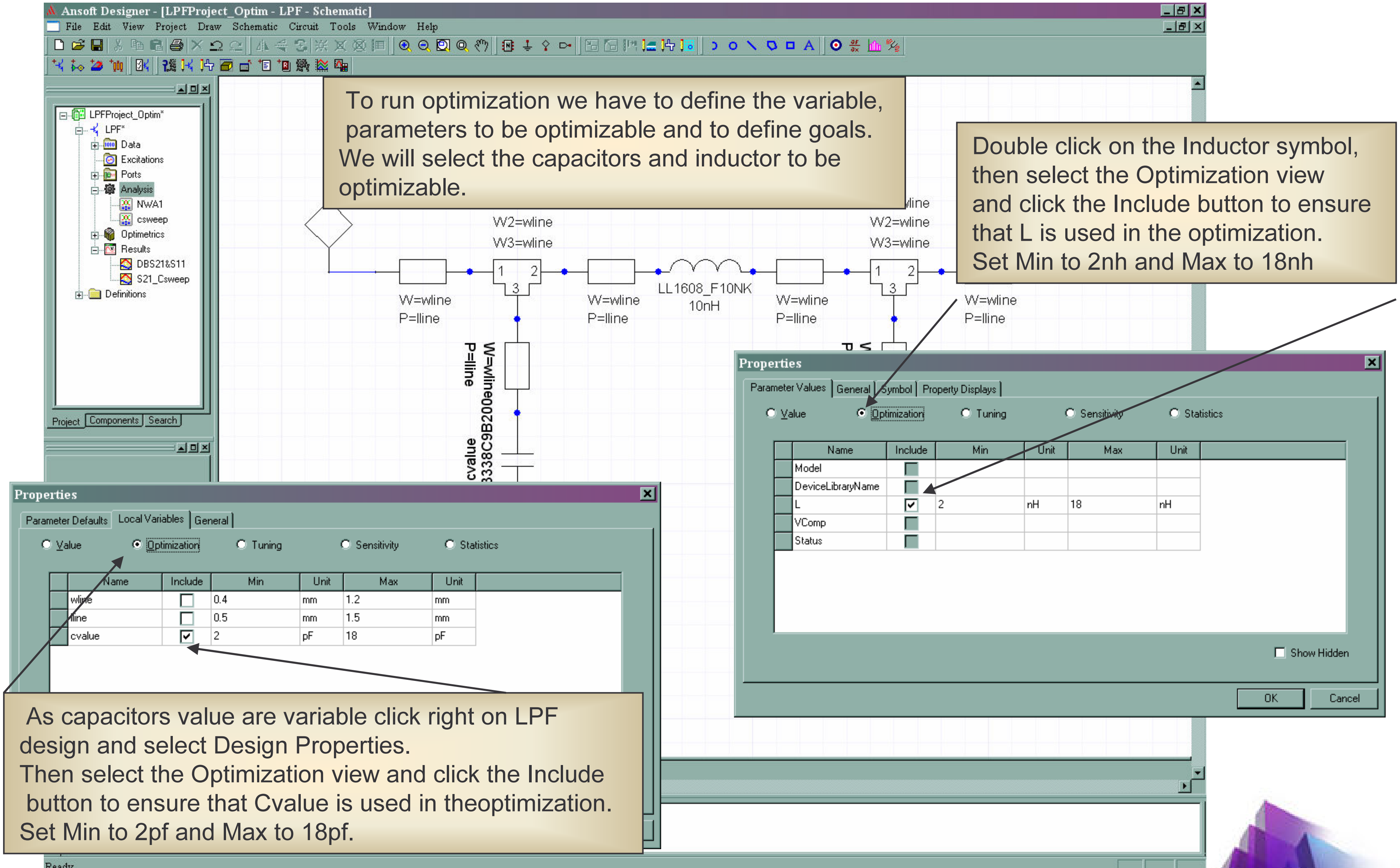
You can either select a value from the tune or none.
For none Click Don't Apply

Hit CTRL+S to save the project

Optimization: Define optimizable parameters

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.

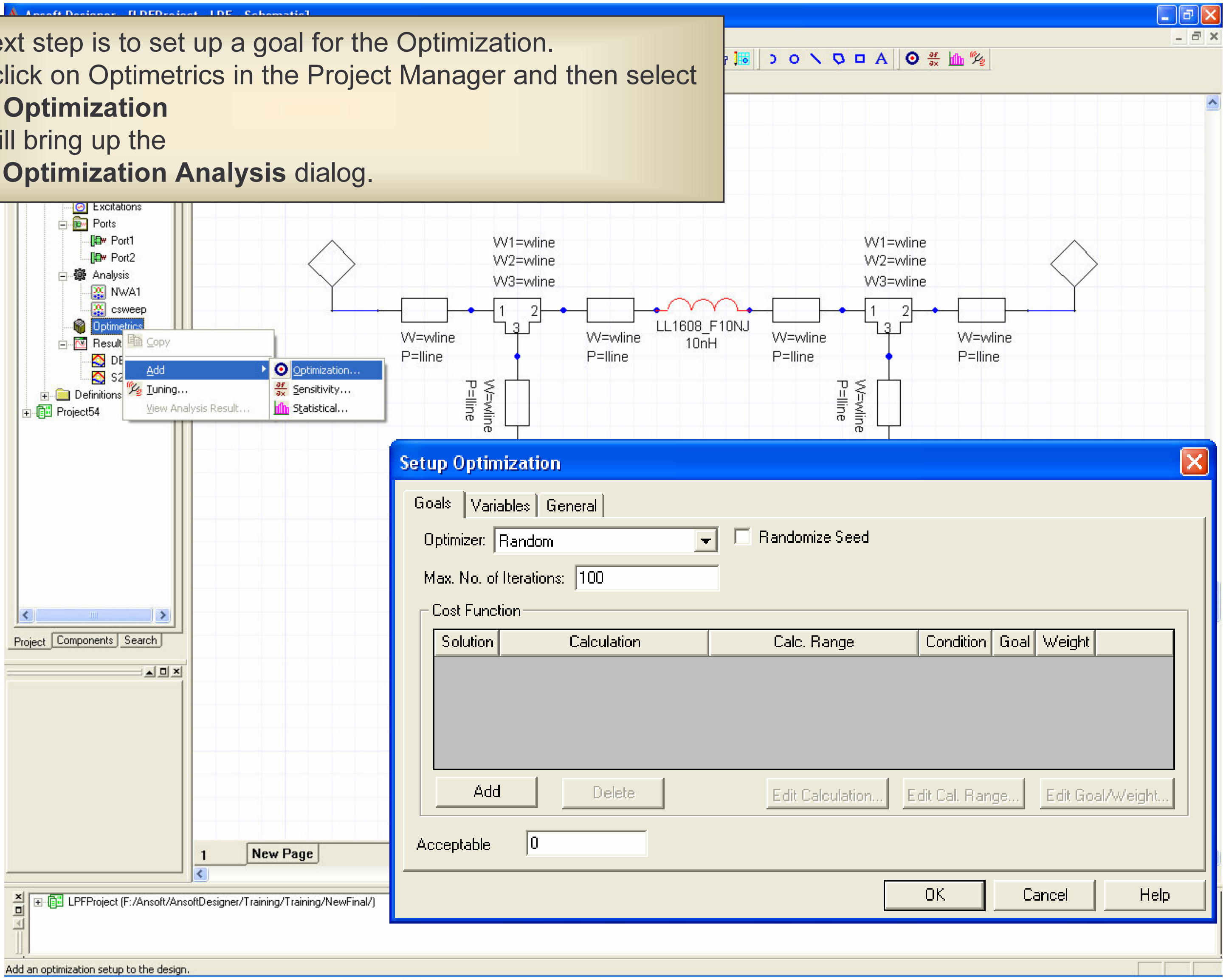
Double click on the Inductor symbol, then select the Optimization view and click the Include button to ensure that L is used in the optimization. Set Min to 2nh and Max to 18nh



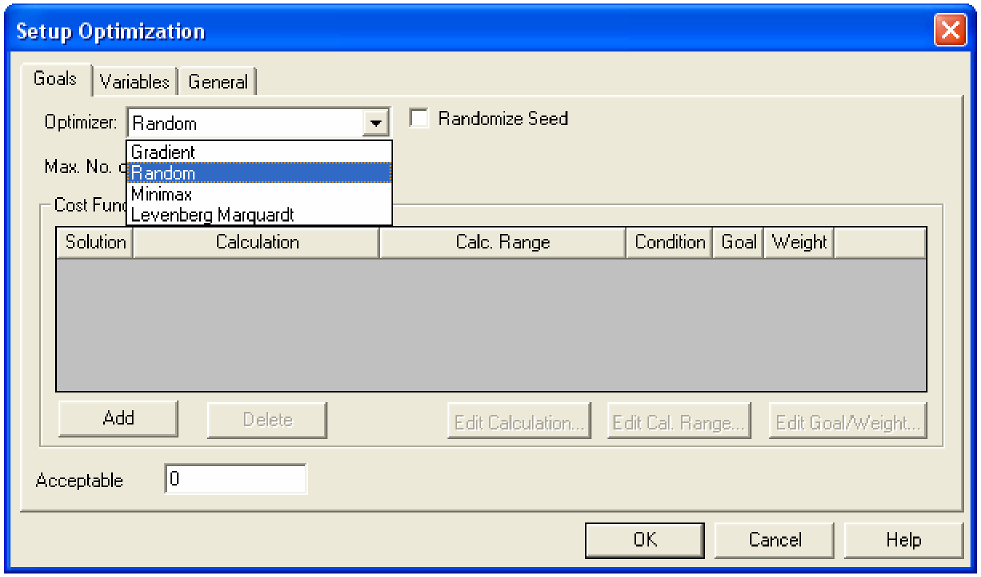
As capacitors value are variable click right on LPF design and select Design Properties. Then select the Optimization view and click the Include button to ensure that Cvalue is used in theoptimization. Set Min to 2pf and Max to 18pf.

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.

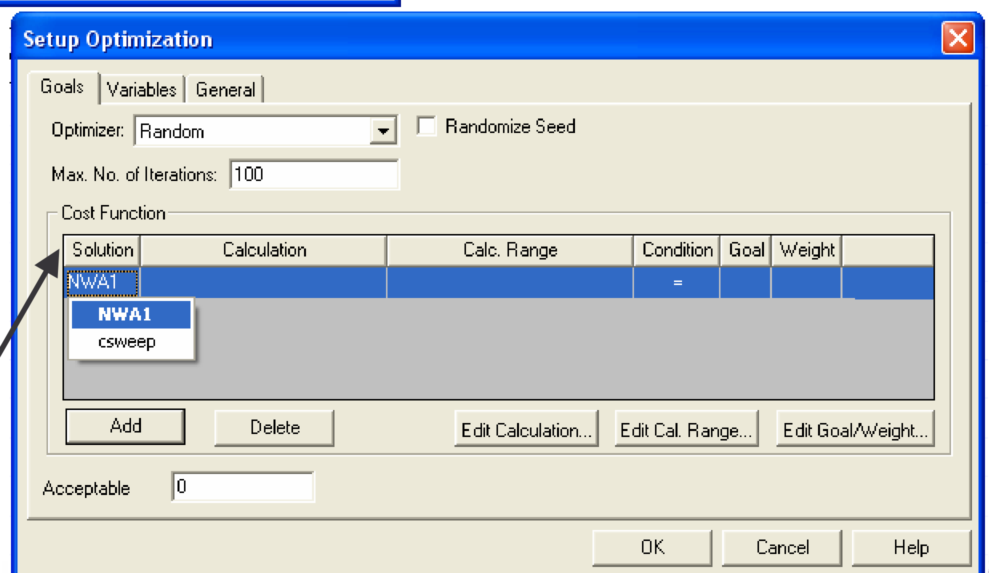


Optimization: Select Solution



Optimizer allows to select from the 4th different algorithms.

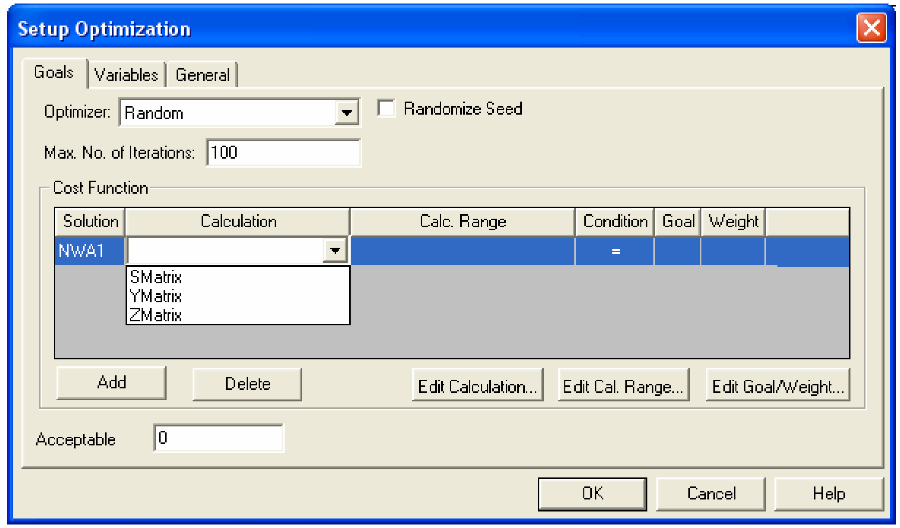
Select Random
Select Max. No of iterations 100



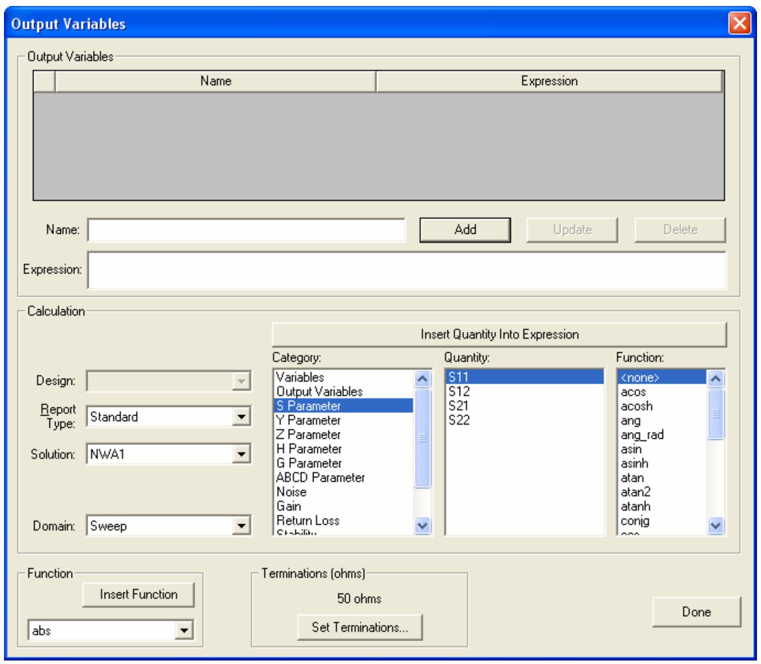
Clicking on **Solution** gives the user a dropdown dialog of all applicable Analyses.
Click Add and Select **NWA1**

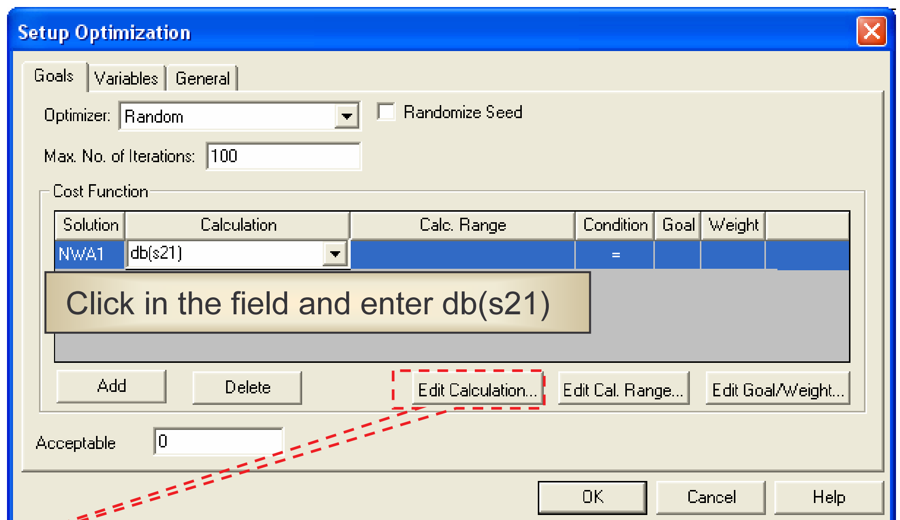
Optimization: Define Calculation

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

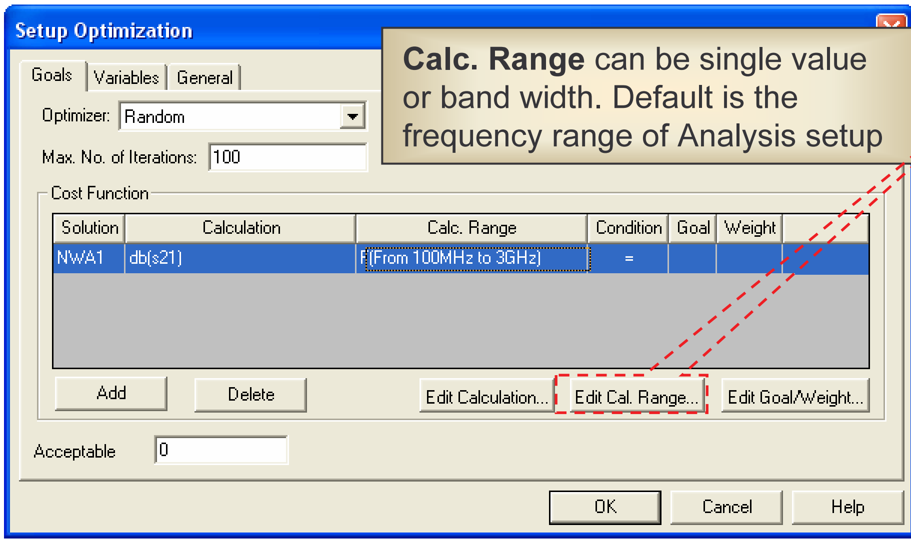


Ansoft Designer lets the user create custom expressions for plotting and optimizing.

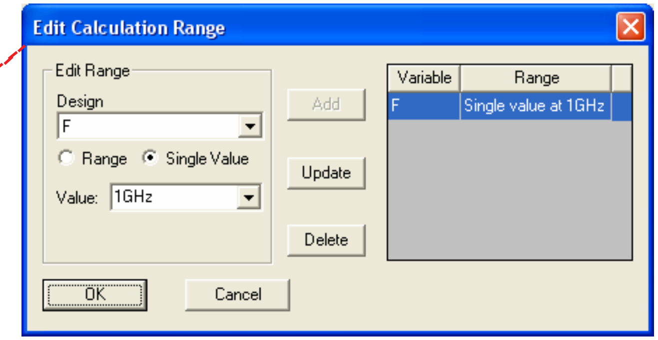




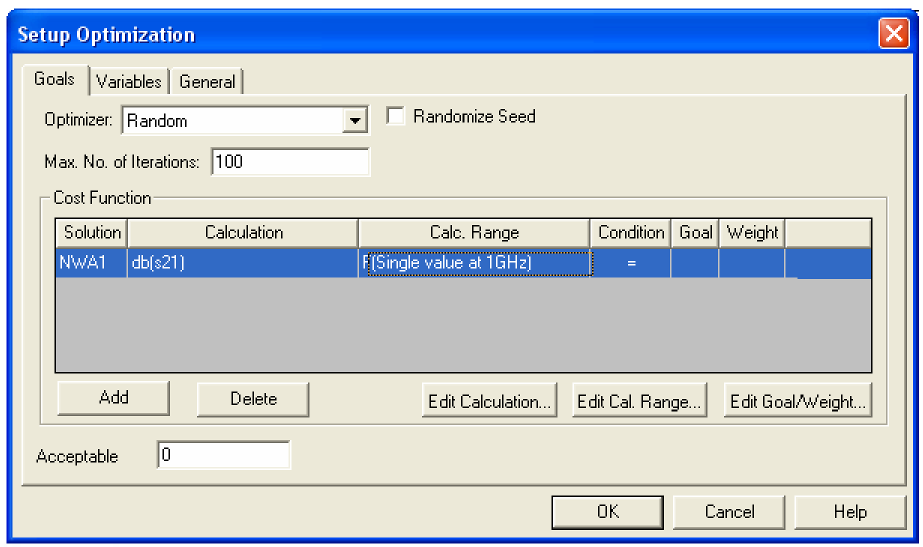
Optimization: Define Calculation Range



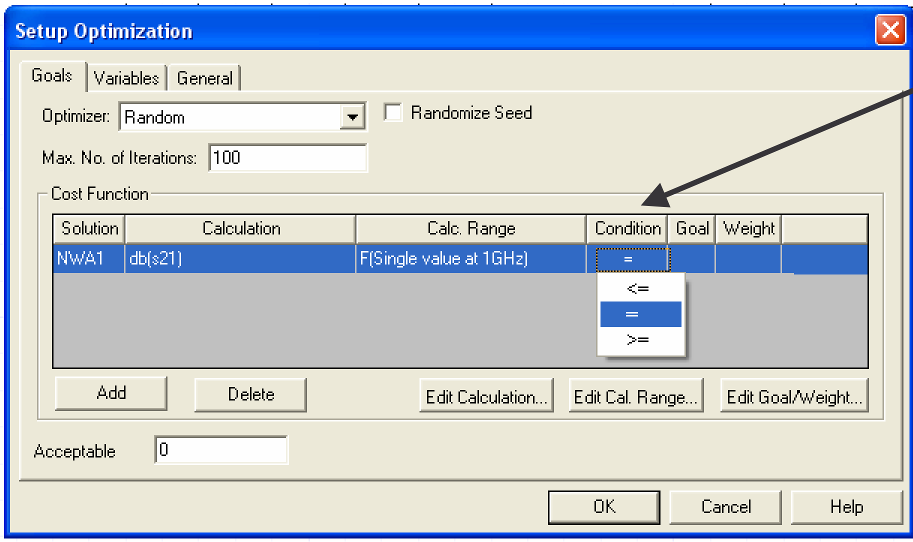
Calc. Range can be single value or band width. Default is the frequency range of Analysis setup



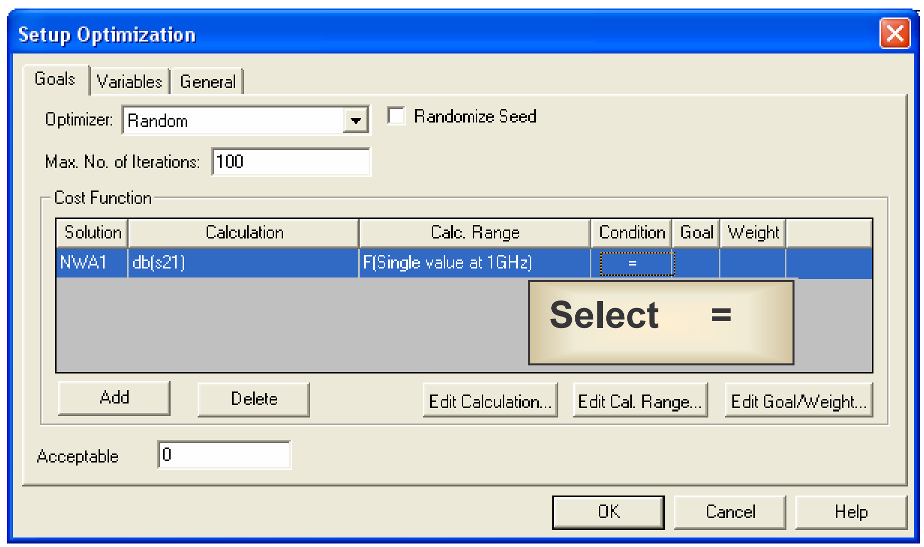
Select single Value,
Value = 1GHz
Click Update and Ok



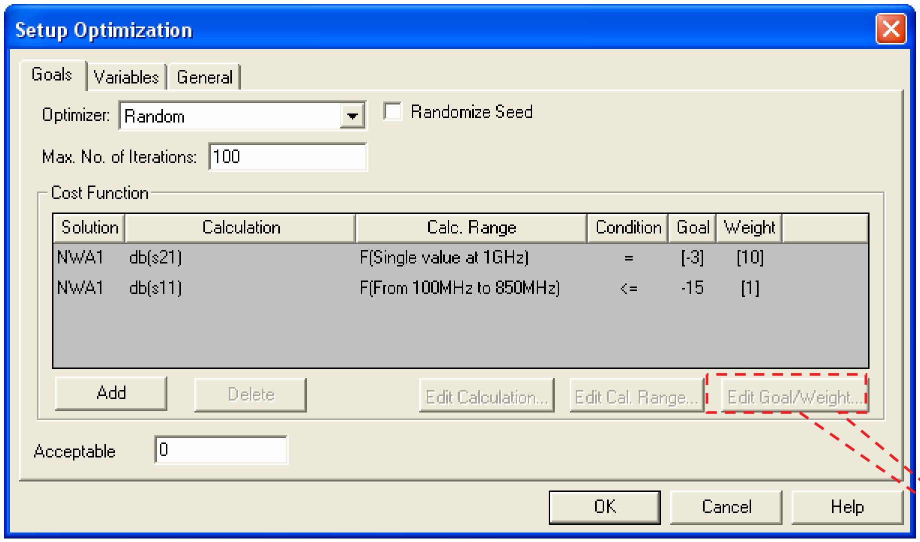
Optimization: Select Condition



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

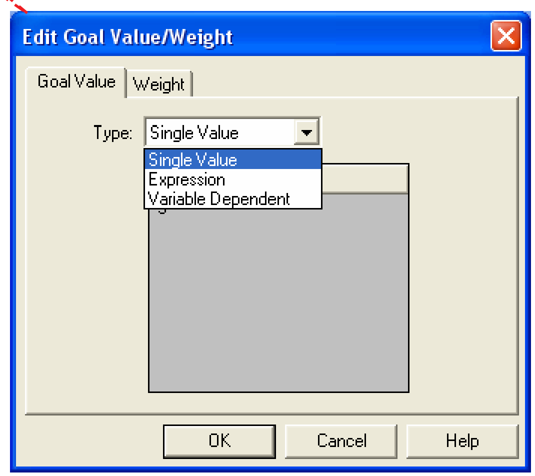
NWA1 db(s11) F (from 100MHz to 850MHz) <= -15 (1)

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

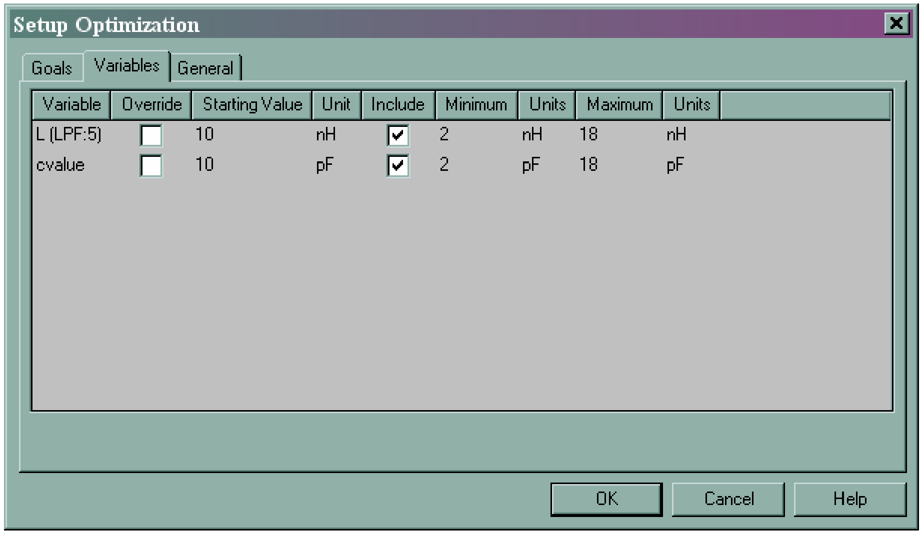
The Goal can be defined as :

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

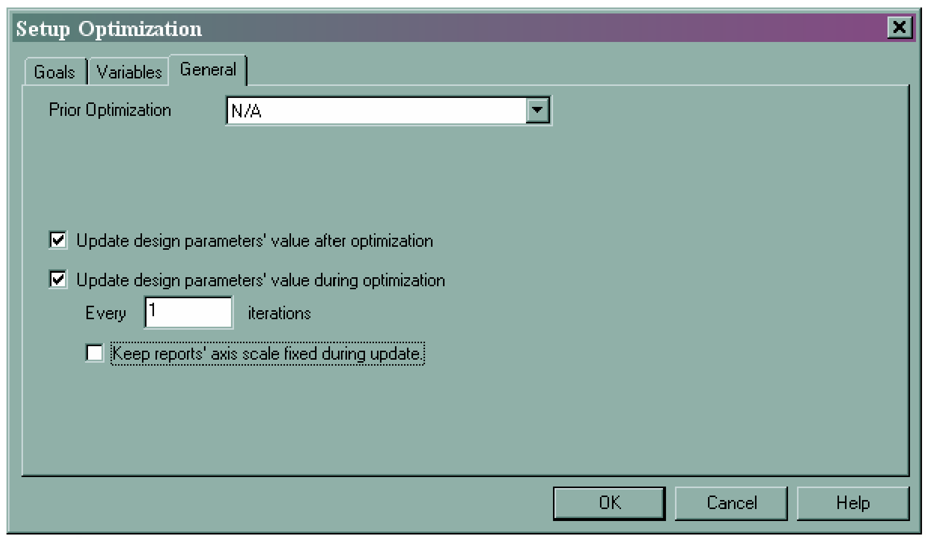
Weight allows to increase the contribution of a specific goal to the whole error function when multiple goals are defined



Optimization Setup: Tab Variable and General

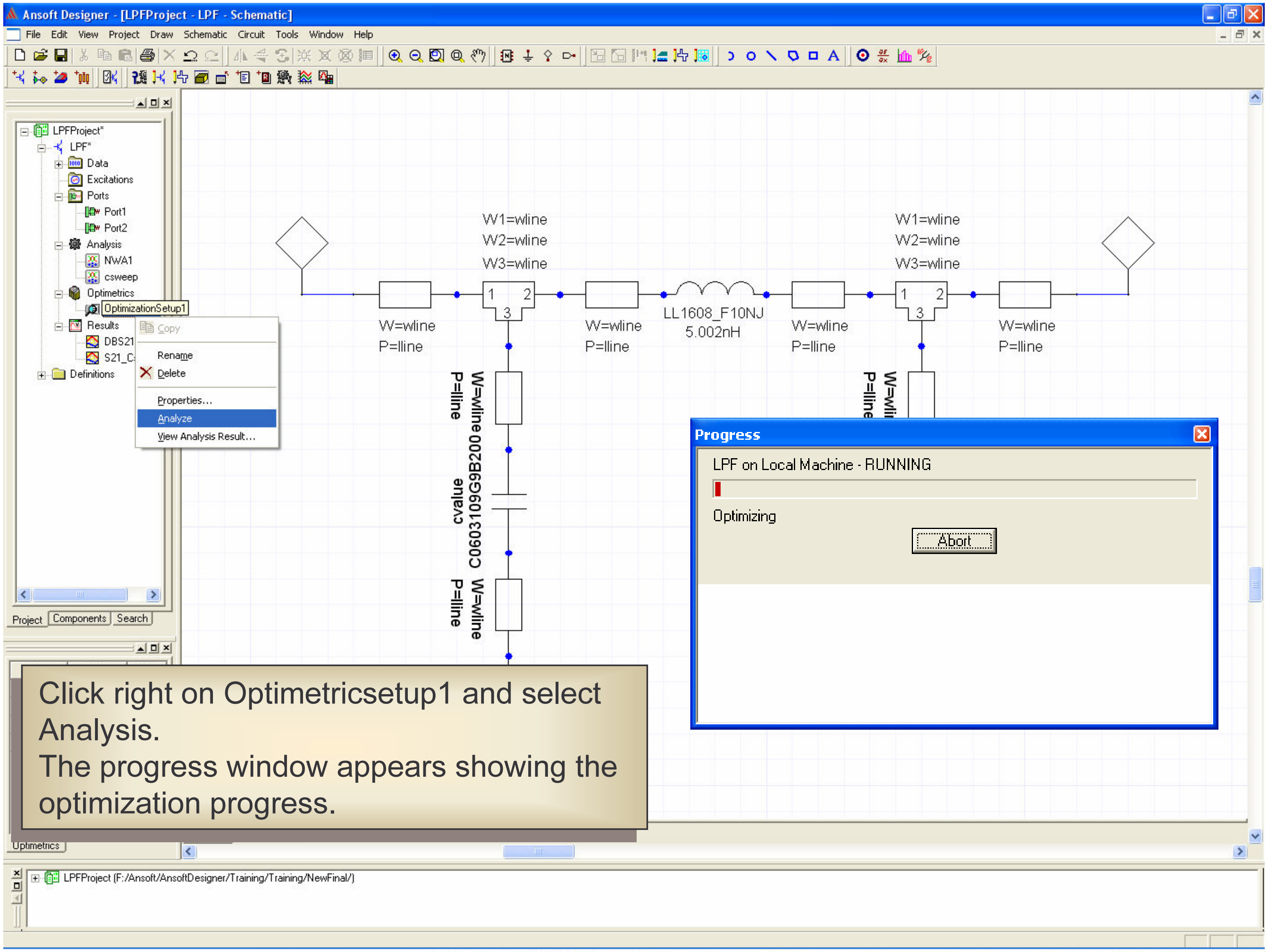


Hit the Variable tab.
This allows to change the minimum and maximum values or to exclude one or more parameters from the optimization



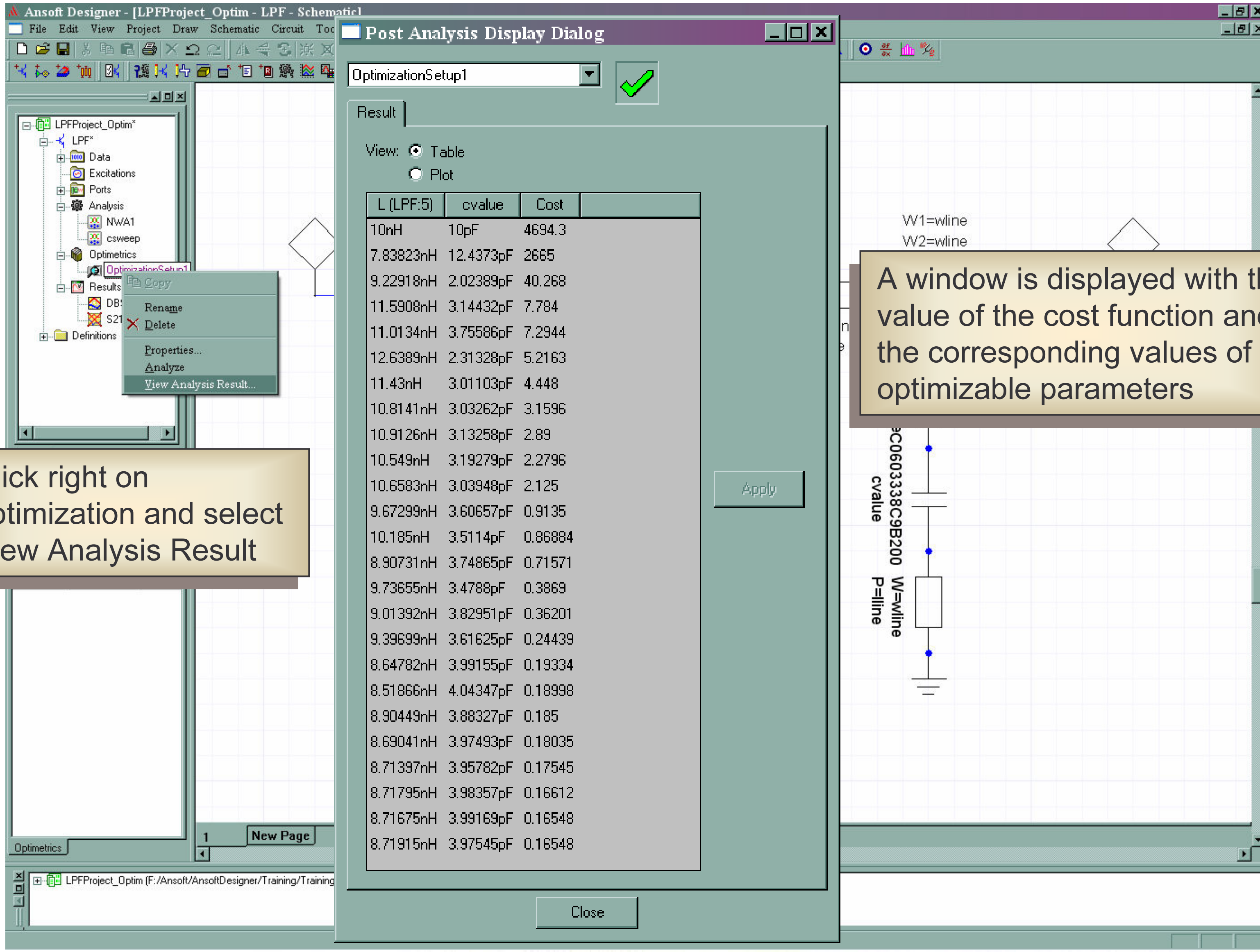
Hit the General tab
This is used to define the strategy of updating the design parameters during and after optimization

Running Optimization



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

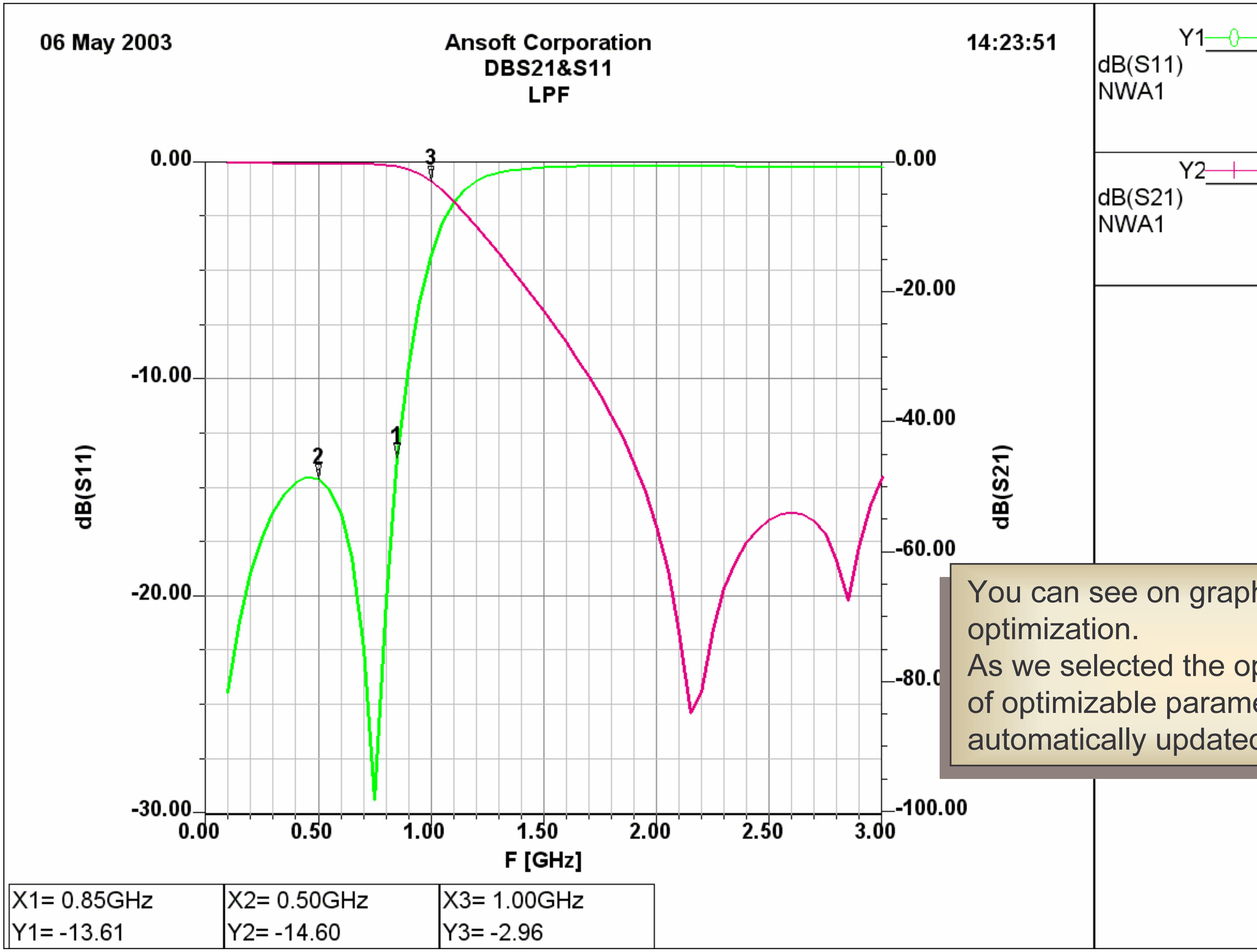
View Optimization Results



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

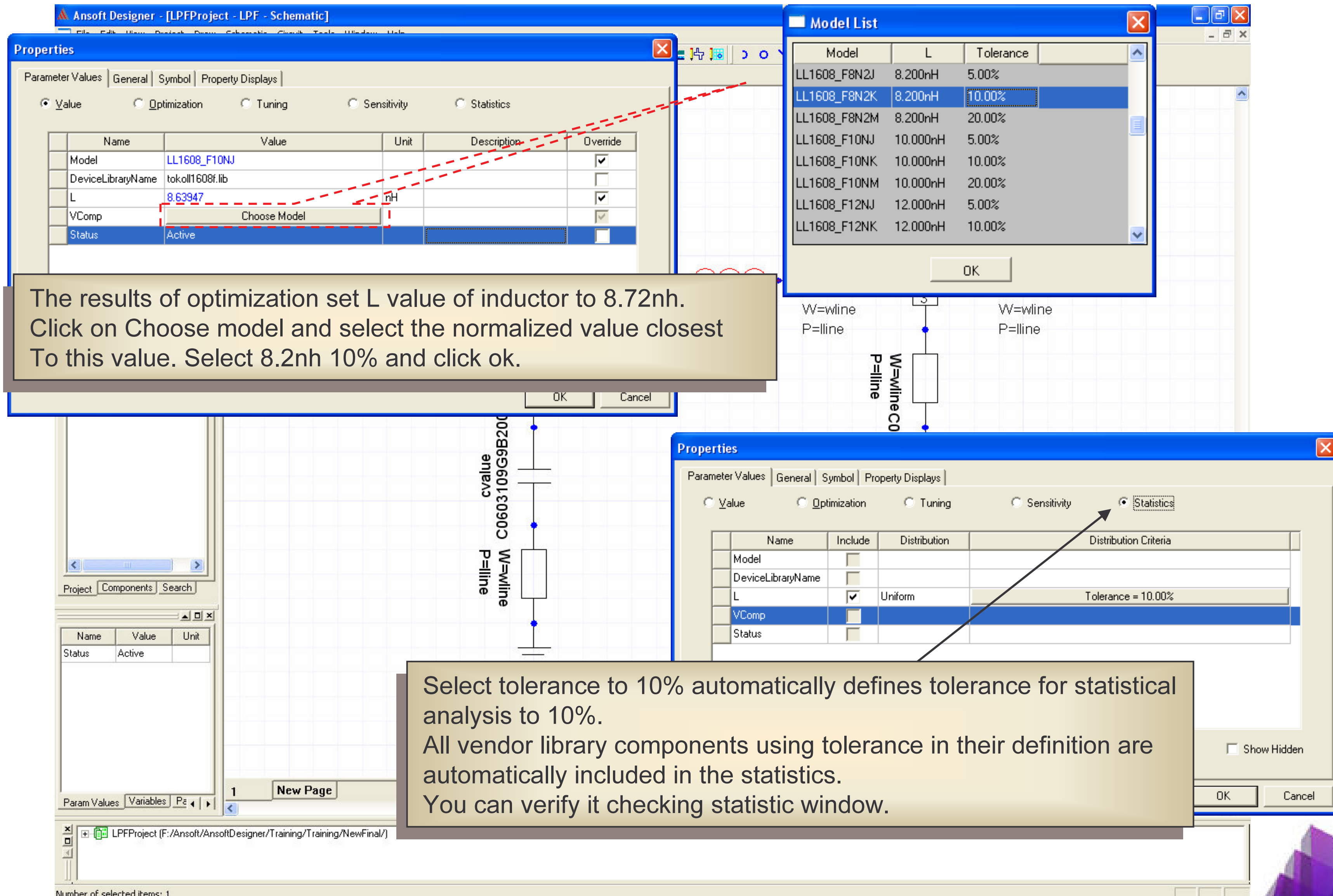
Click right on optimization and select View Analysis Result

View Optimization Result on Graph



You can see on graph the result of optimization.
As we selected the option, the value of optimizable parameters were automatically updated.

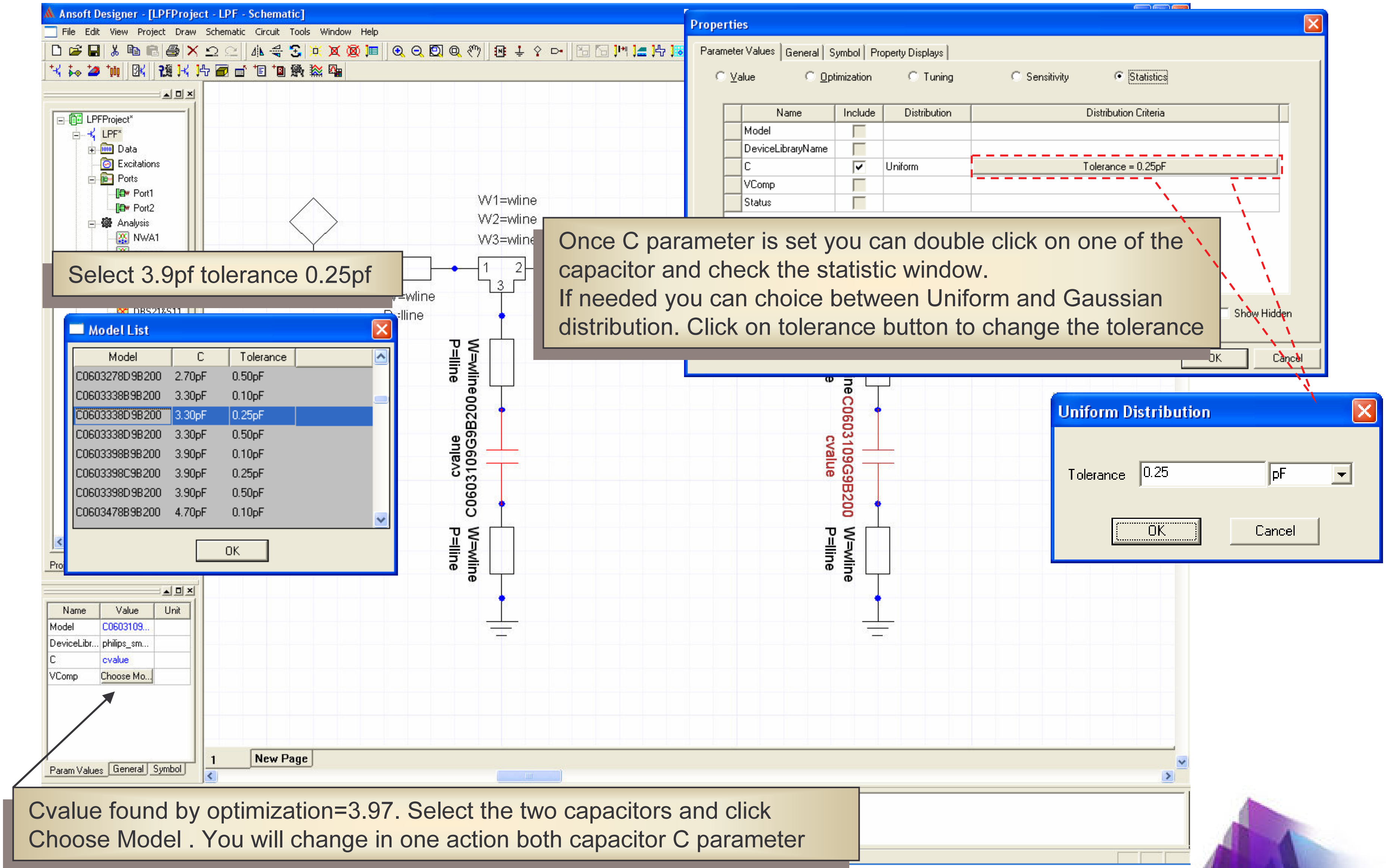
Set the parameters for Inductor



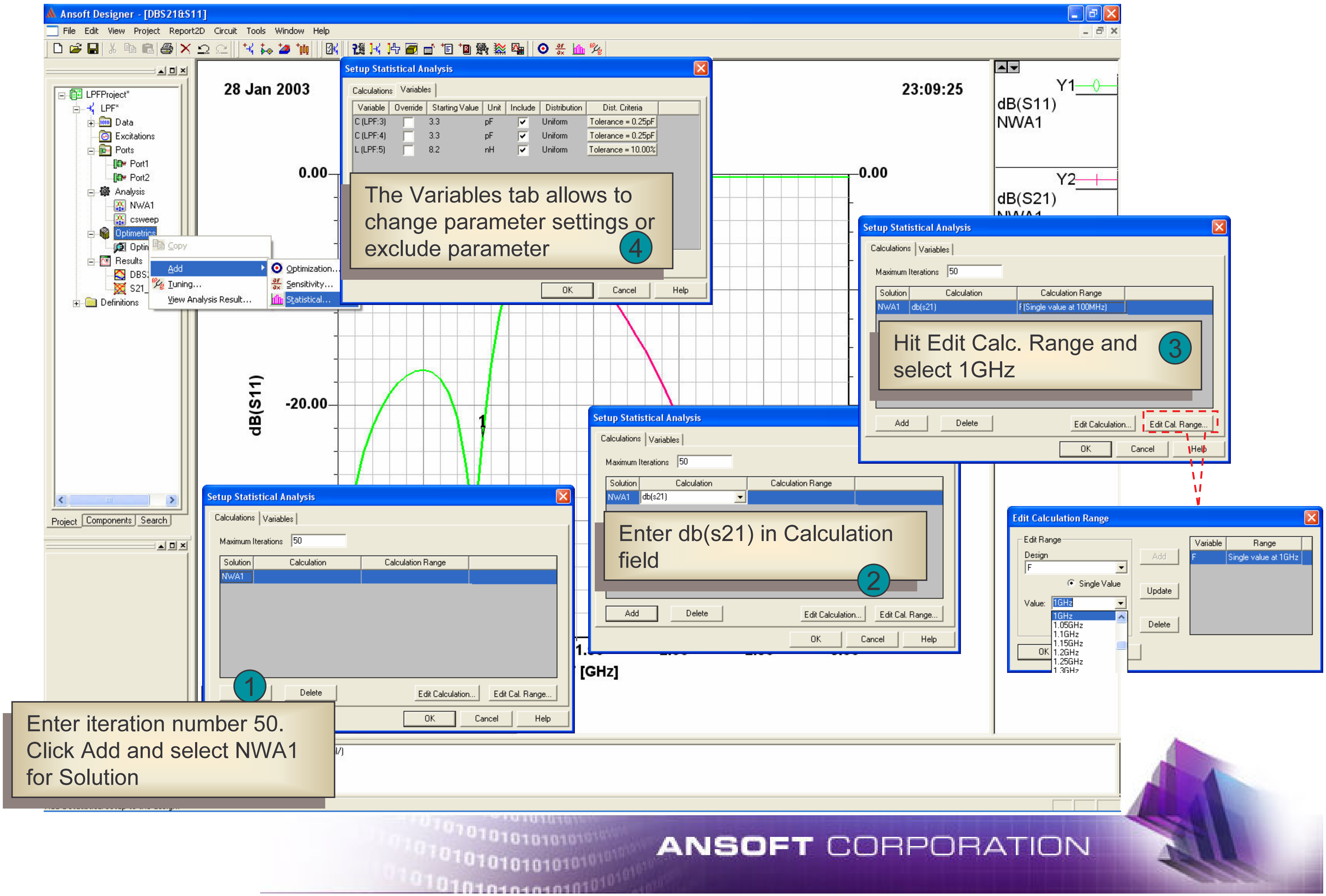
The results of optimization set L value of inductor to 8.72nh.
Click on Choose model and select the normalized value closest
To this value. Select 8.2nh 10% and click ok.

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.

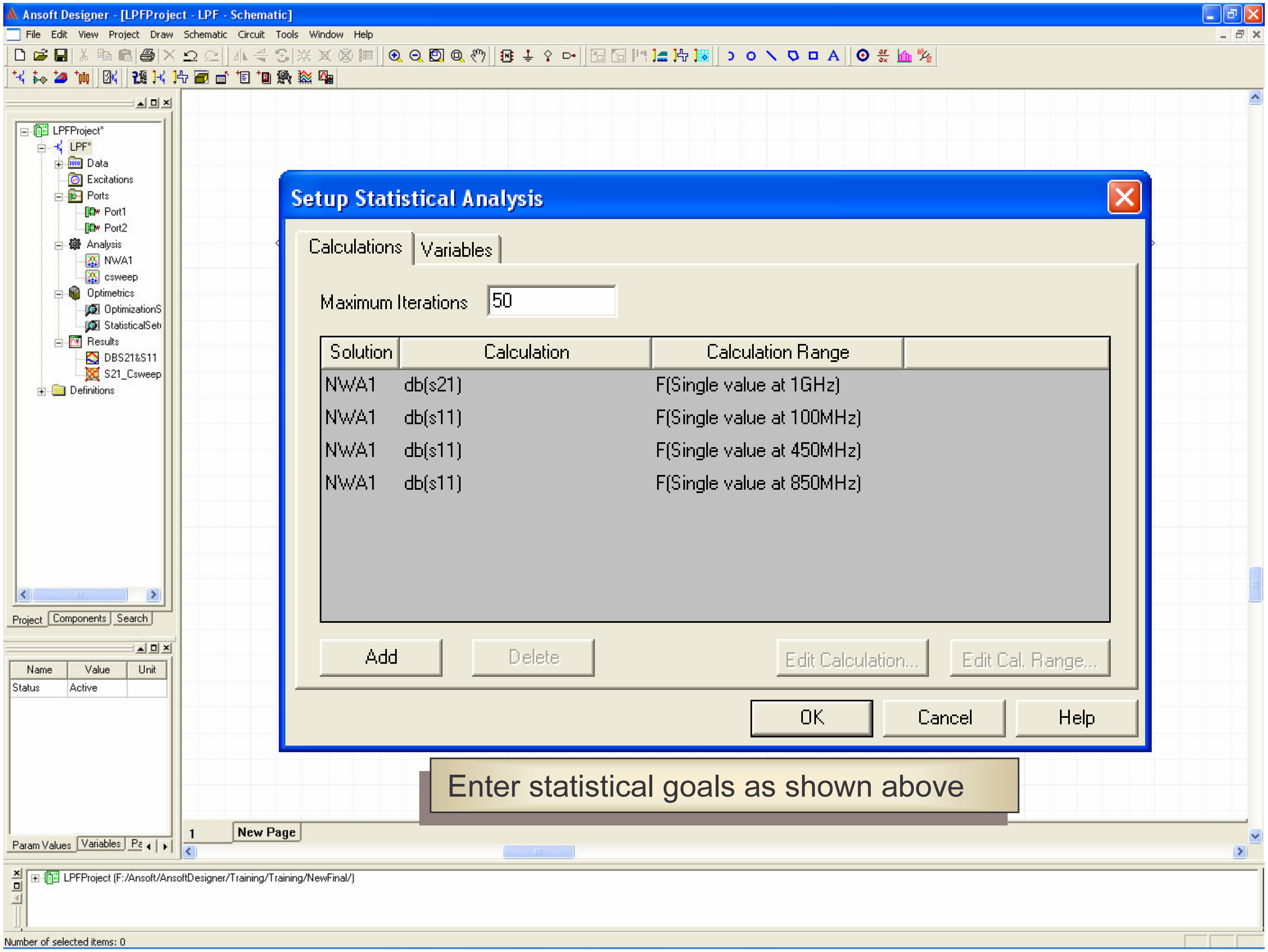
Set the parameters for Capacitor



Statistical Analysis: Define goals

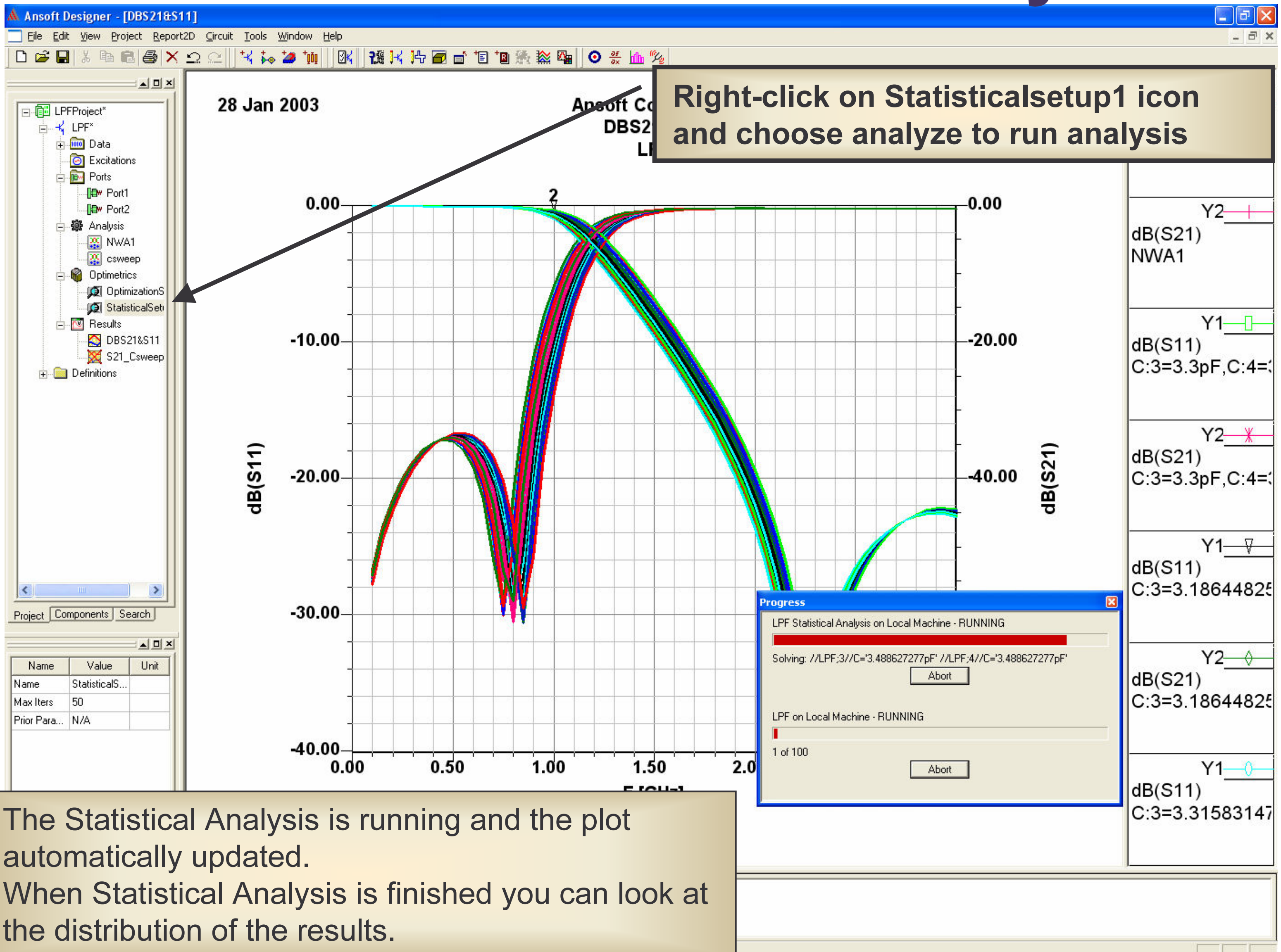


Statistical Analysis: Define goals



Enter statistical goals as shown above

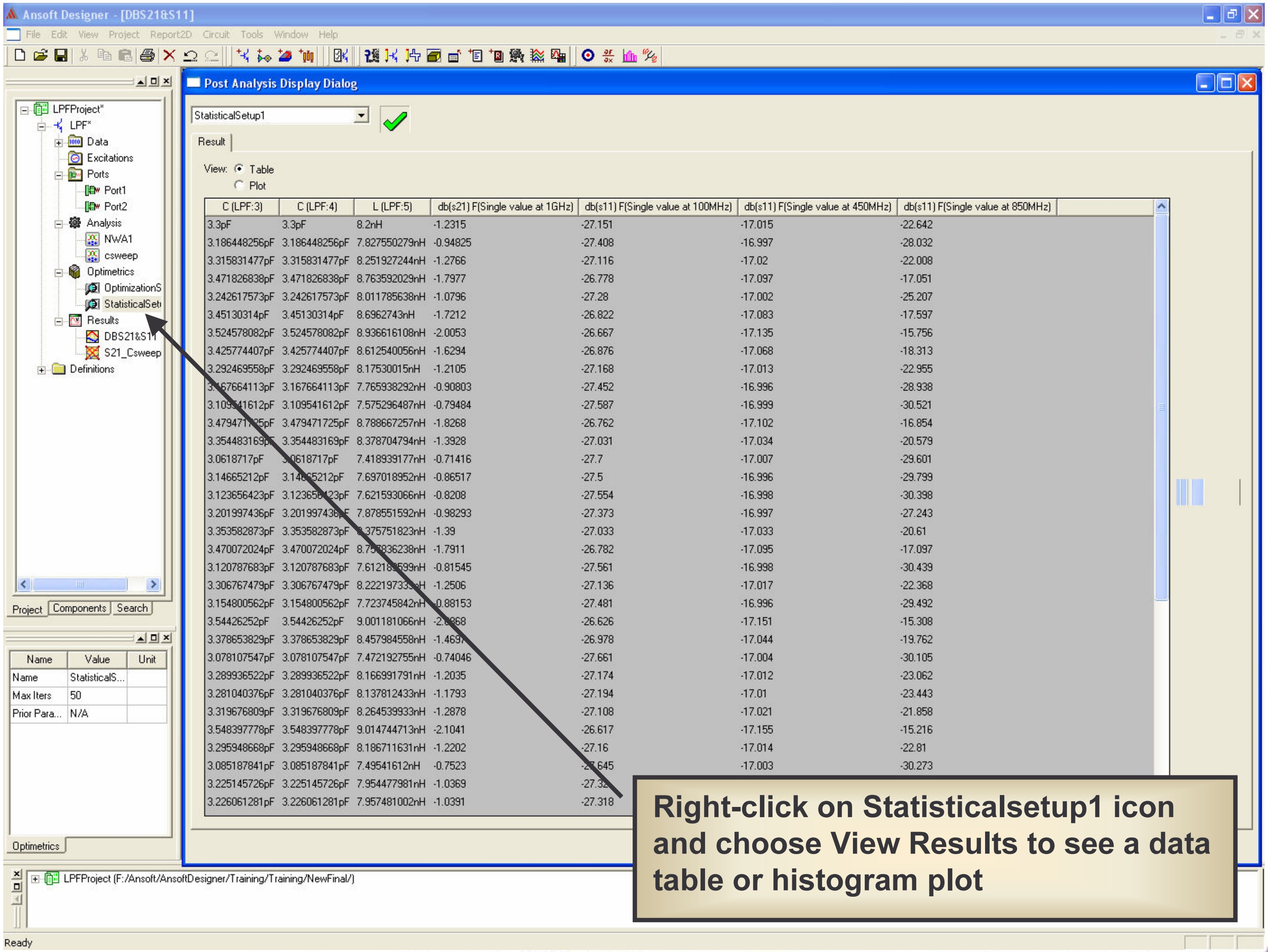
Run Statistical Analysis



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

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

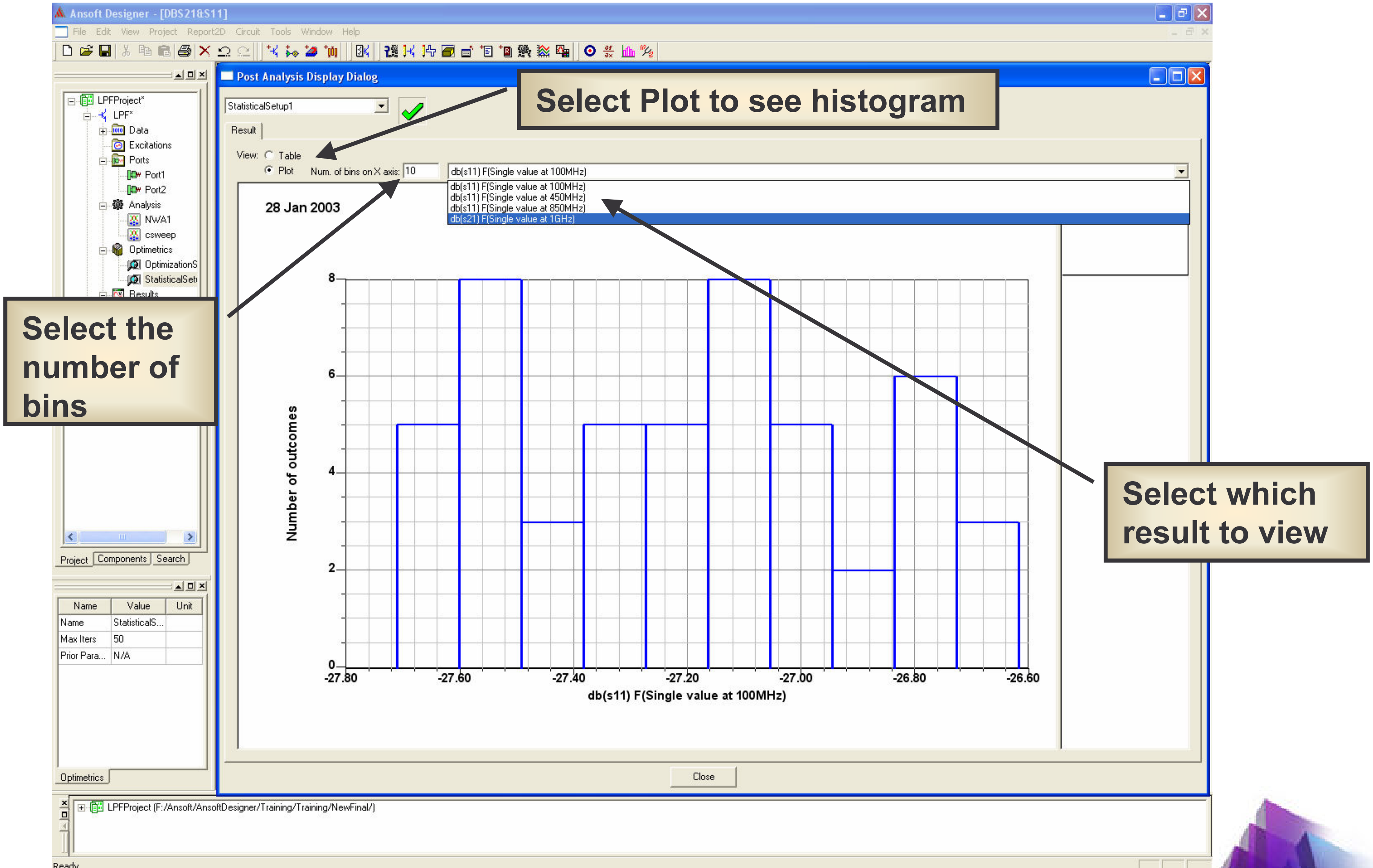
View Analysis Result: Data Table



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.[illegible]67664113pF	3.167664113pF	7.765938292nH	-0.90803	-27.452	-16.996	-28.938
3.109[illegible]41612pF	3.109541612pF	7.575296487nH	-0.79484	-27.587	-16.999	-30.521
3.479471[illegible]25pF	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[illegible]0618717pF	7.418939177nH	-0.71416	-27.7	-17.007	-29.601
3.14665212pF	3.14[illegible]65212pF	7.697018952nH	-0.86517	-27.5	-16.996	-29.799
3.123656423pF	3.123656[illegible]23pF	7.621593066nH	-0.8208	-27.554	-16.998	-30.398
3.201997436pF	3.201997436pF	7.878551592nH	-0.98293	-27.373	-16.997	-27.243
3.353582873pF	3.353582873pF	[illegible].375751823nH	-1.39	-27.033	-17.033	-20.61
3.470072024pF	3.470072024pF	8.7[illegible]836238nH	-1.7911	-26.782	-17.095	-17.097
3.120787683pF	3.120787683pF	7.61218[illegible]599nH	-0.81545	-27.561	-16.998	-30.439
3.306767479pF	3.306767479pF	8.2221973[illegible]H	-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.[illegible]868	-26.626	-17.151	-15.308
3.378653829pF	3.378653829pF	8.457984558nH	-1.469[illegible]	-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[illegible]	[illegible]	[illegible]
3.226061281pF	3.226061281pF	7.957481002nH	-1.0391	-27.318	[illegible]	[illegible]

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

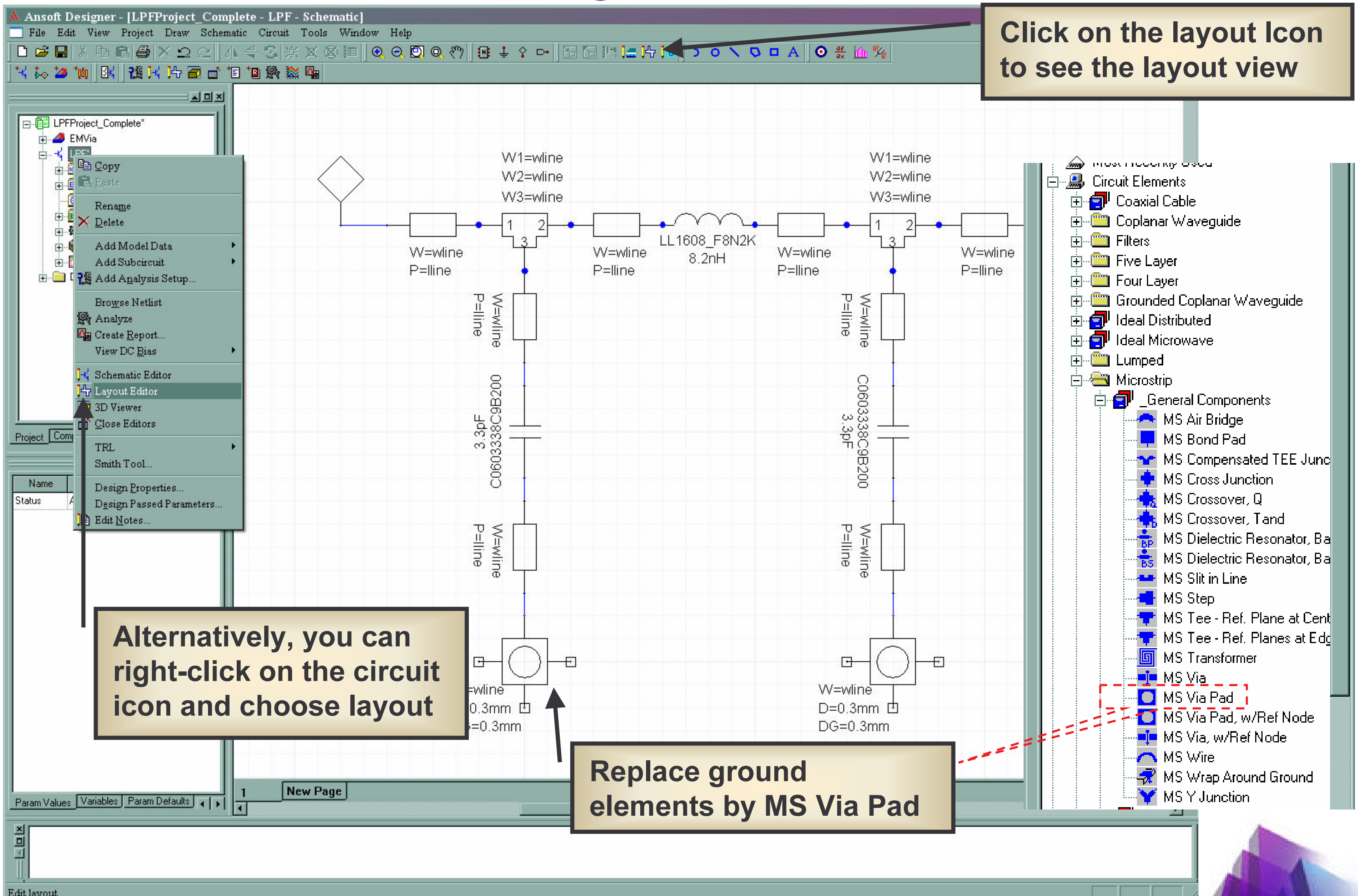
View Analysis Result: Histogram



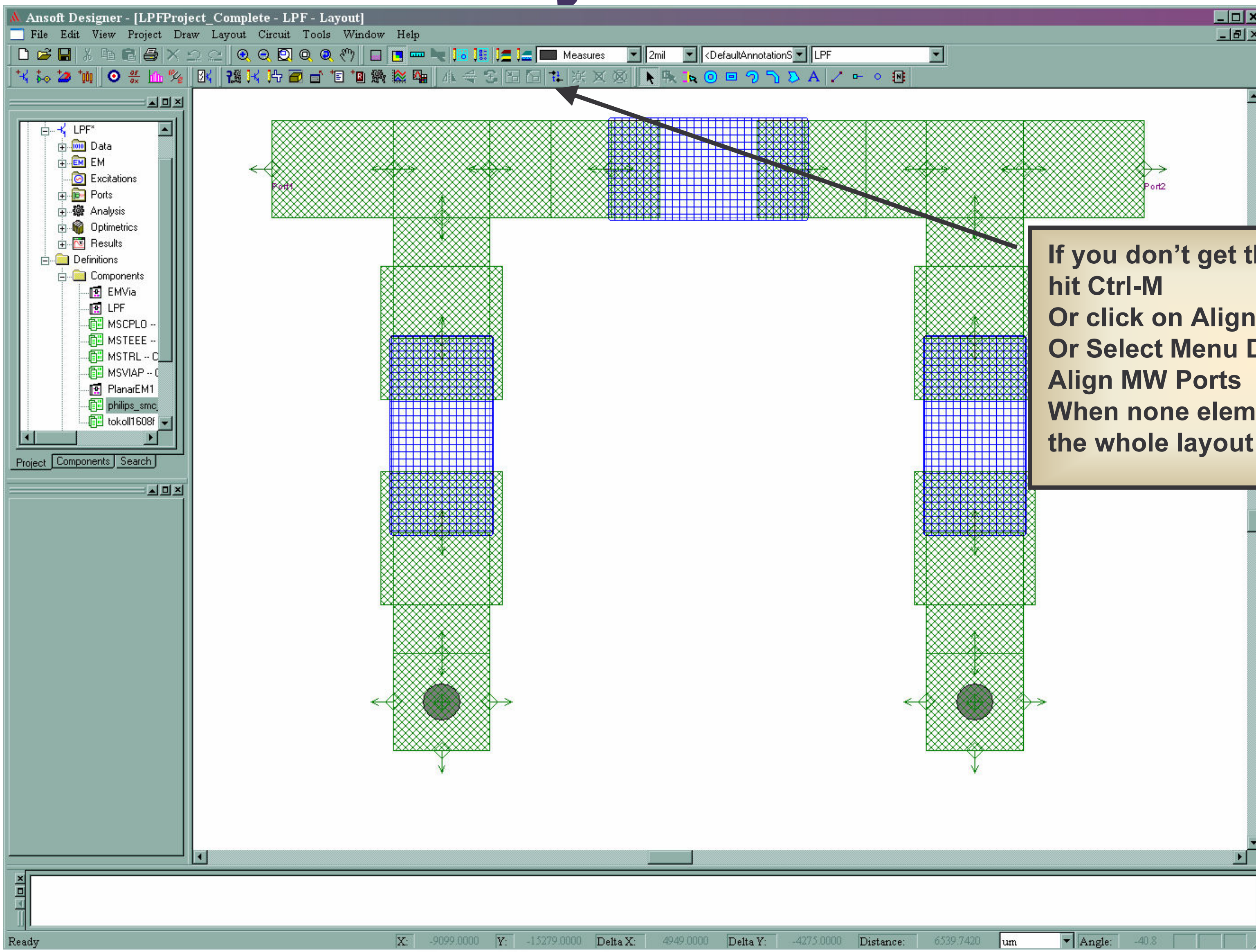
End of Exercise 1

Layout Basics

View Layout Window

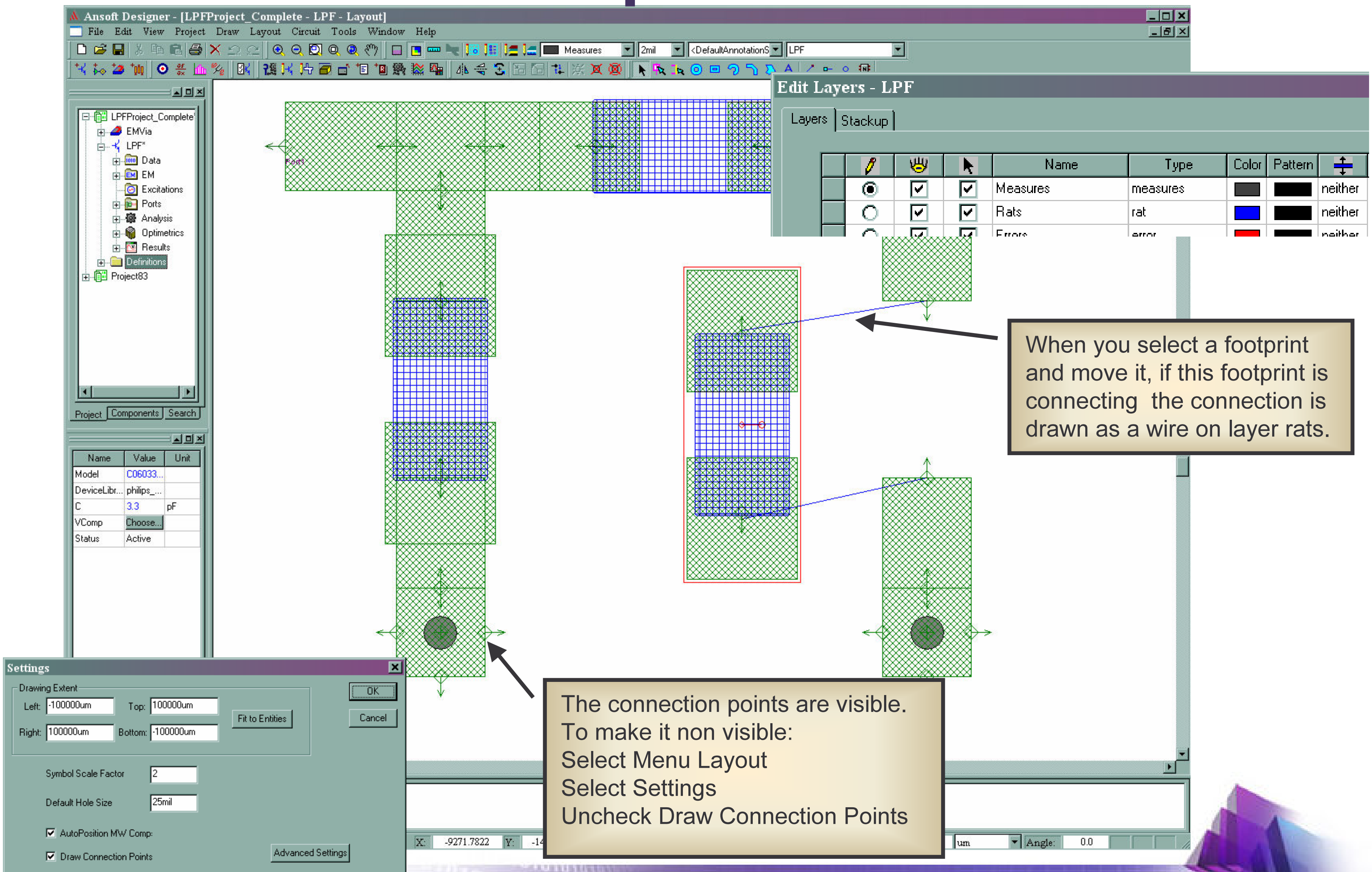


Layout Window



If you don't get the layout aligned, hit Ctrl-M
Or click on Align MW Ports icon
Or Select Menu Draw and click on Align MW Ports
When none element is selected the whole layout is align.

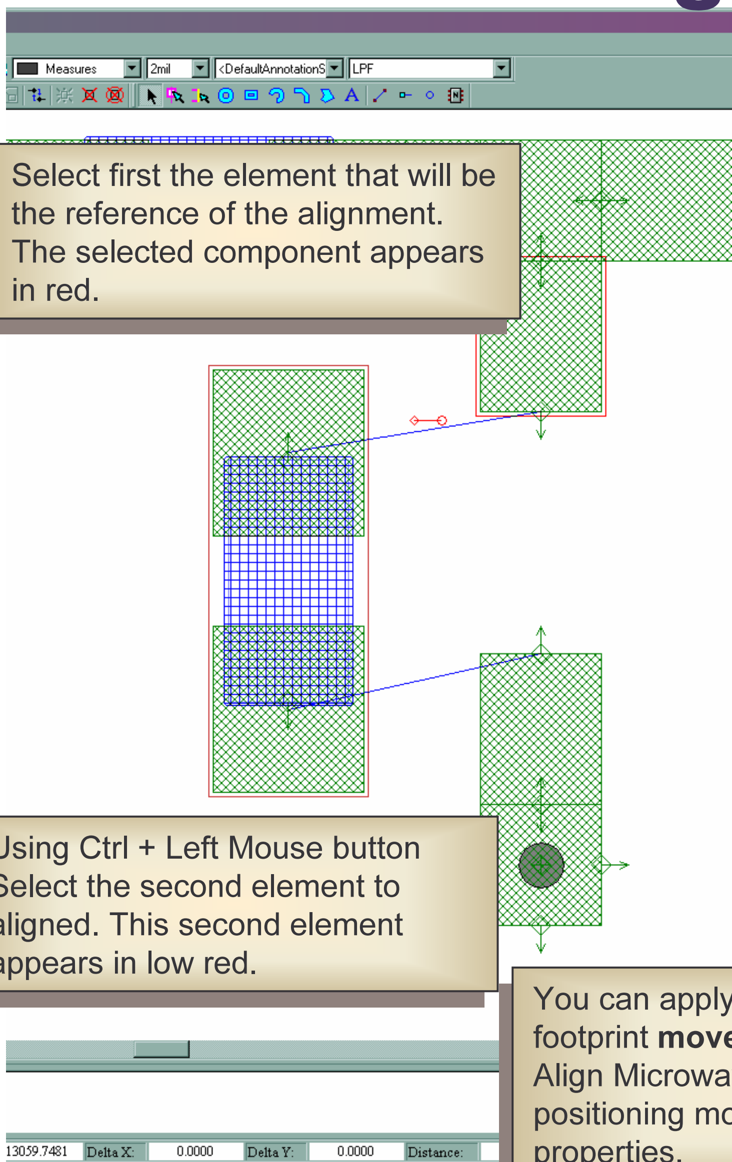
Connection points and wires



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

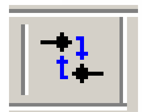
The connection points are visible.
To make it non visible:
Select Menu Layout
Select Settings
Uncheck Draw Connection Points

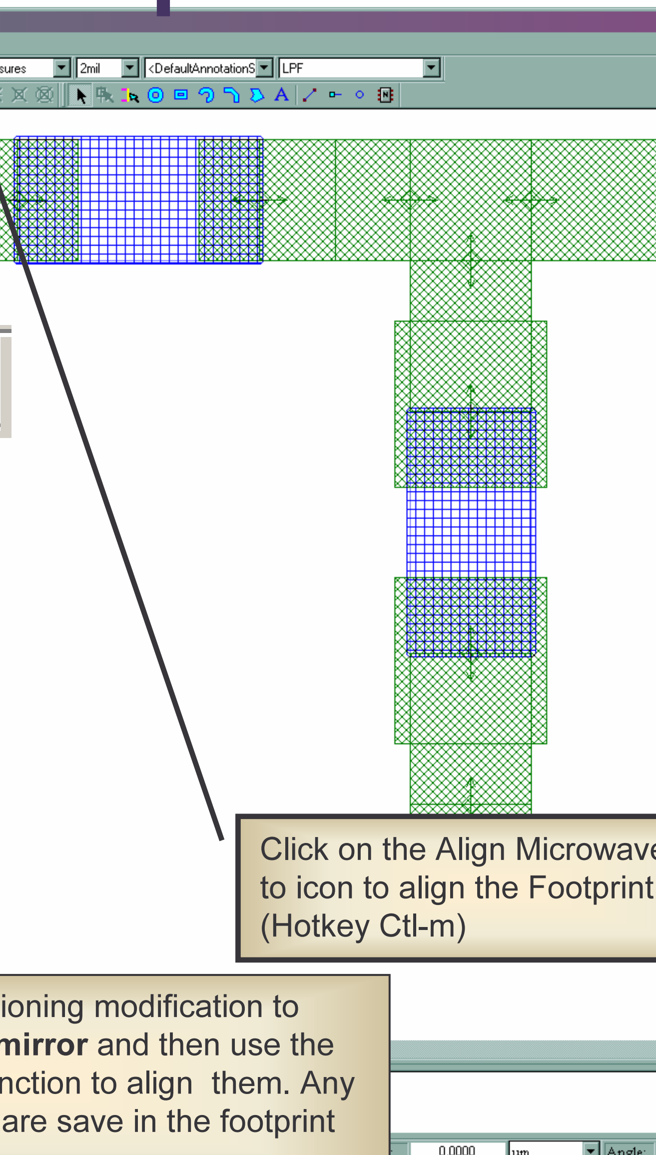
Align Footprint



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.

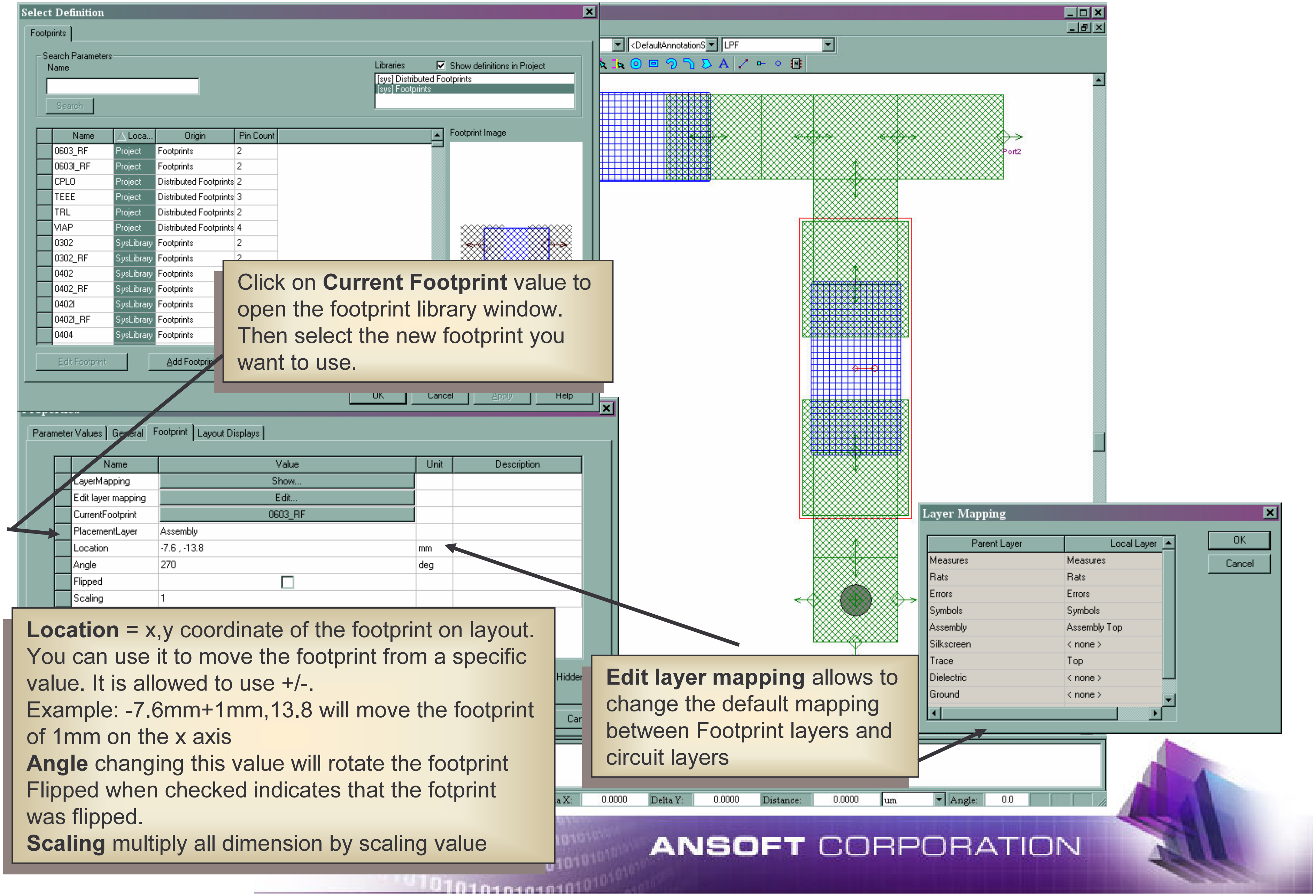




Click on the Align Microwave Port to icon to align the Footprint (Hotkey Ctl-m)

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.

Layout Footprint Properties

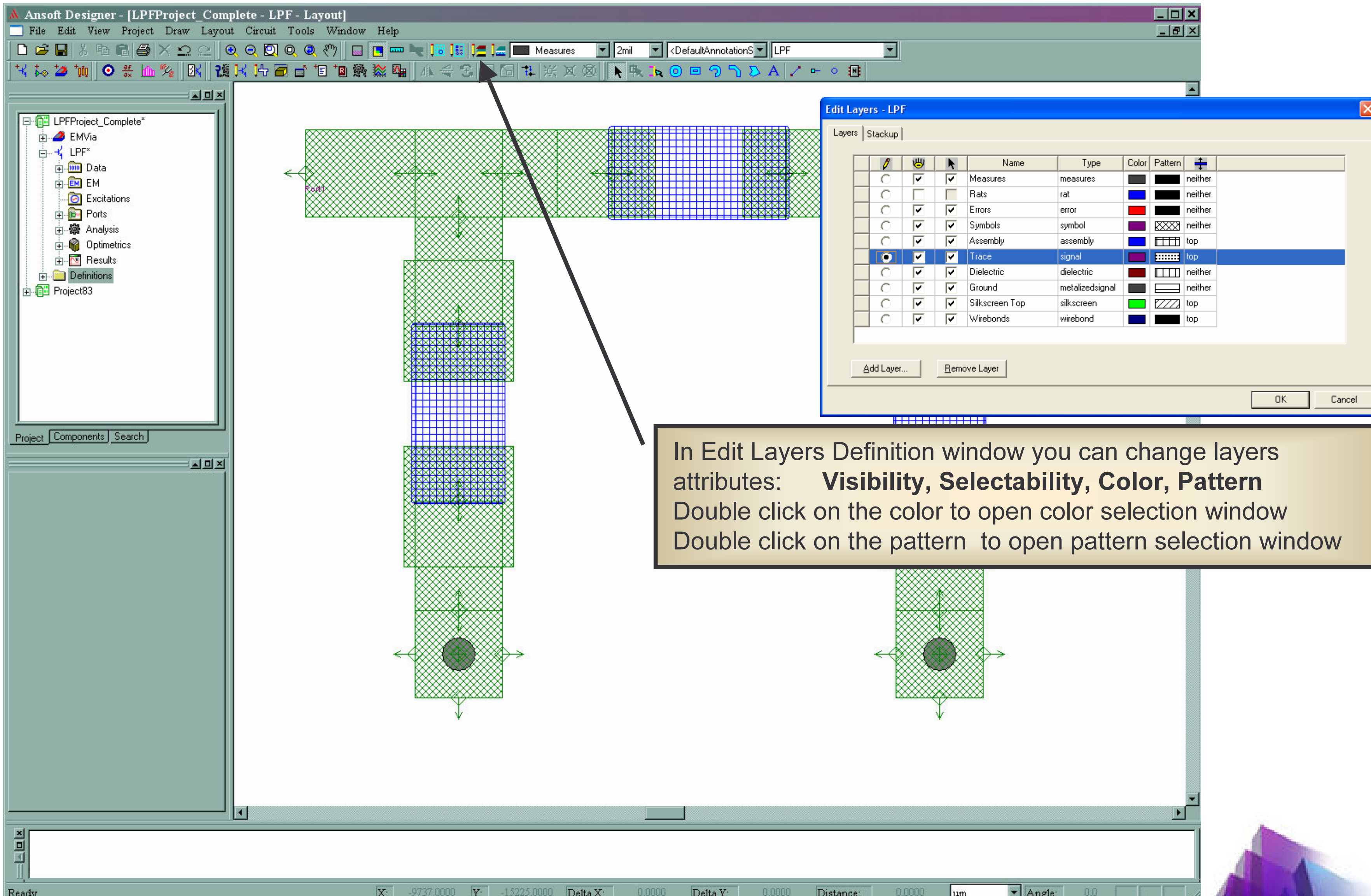


Click on **Current Footprint** value to open the footprint library window. Then select the new footprint you want to use.

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 fotprint was flipped.
Scaling multiply all dimension by scaling value

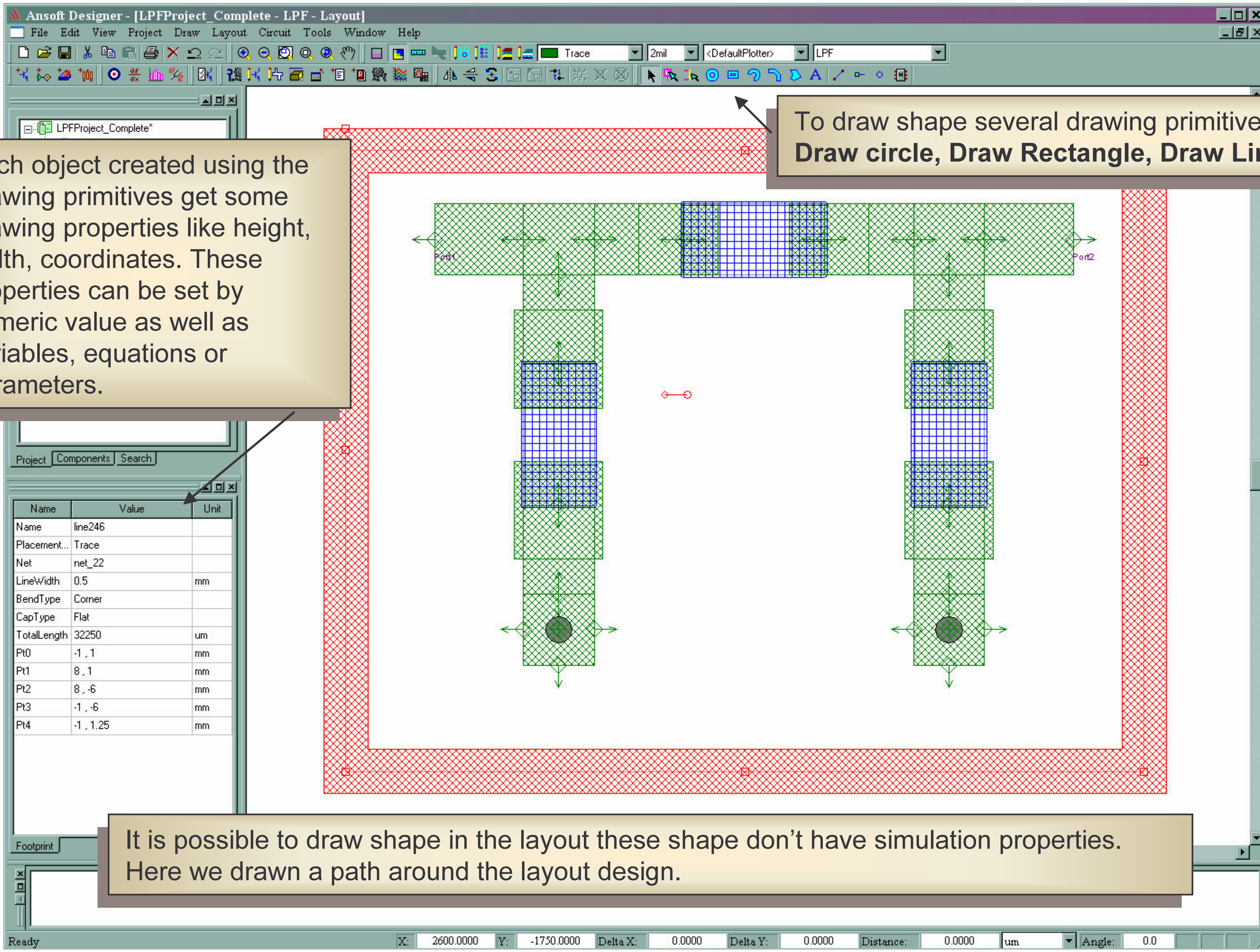
Edit layer mapping allows to change the default mapping between Footprint layers and circuit layers

Layout Window Option



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

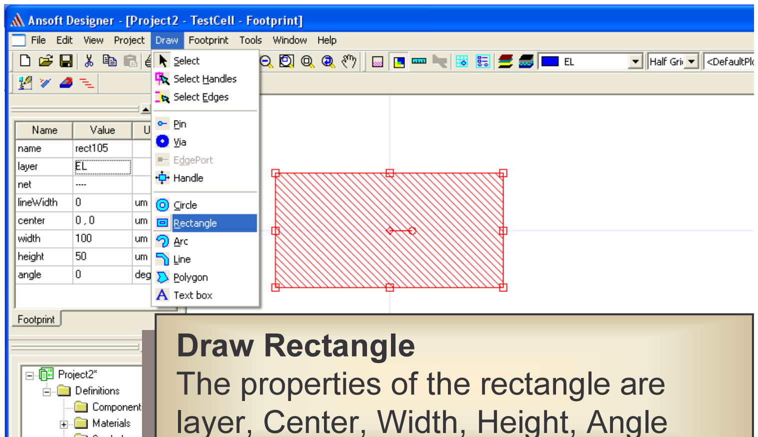


To draw shape several drawing primitives are available :
Draw circle, Draw Rectangle, Draw Line, Draw Polygon

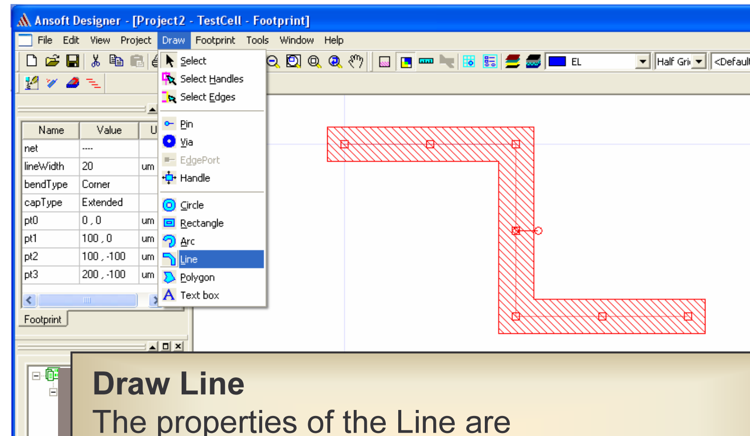
Each object created using the drawing primitives get some drawing properties like height, width, coordinates. These properties can be set by numeric value as well as variables, equations or parameters.

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.

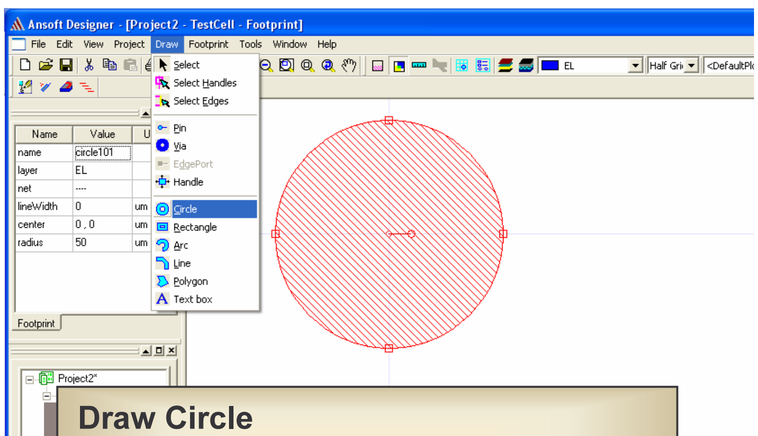
Drawing Primitives



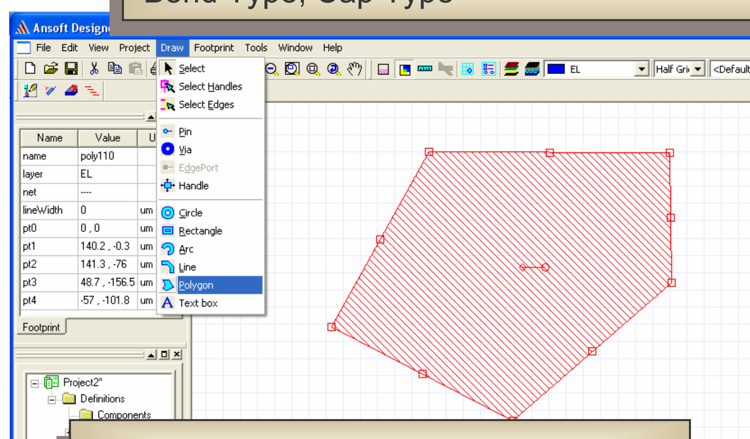
Draw Rectangle
The properties of the rectangle are
layer, Center, Width, Height, Angle



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

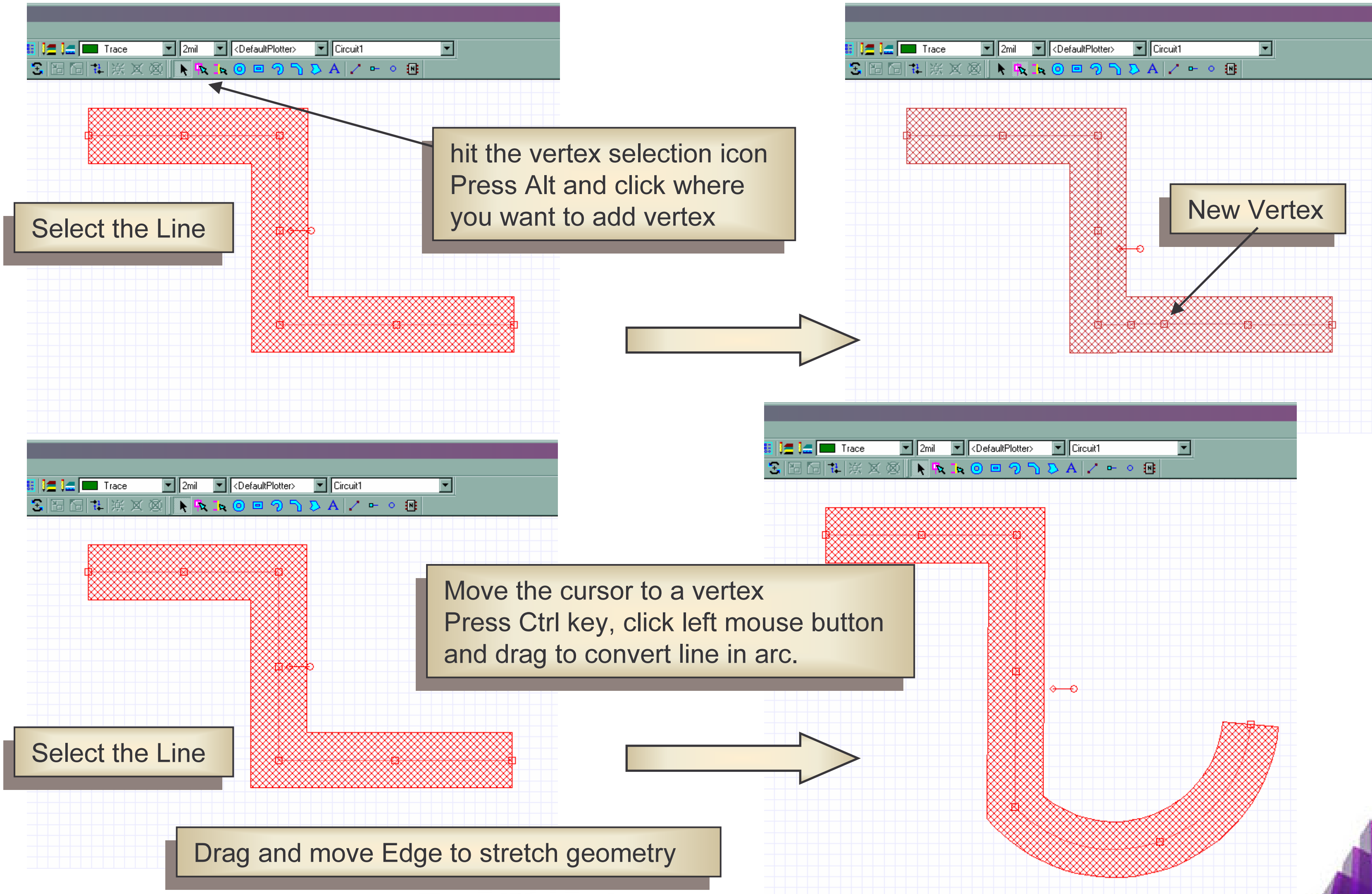


Draw Circle
The properties of the Circle are
layer, Center, Radius



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

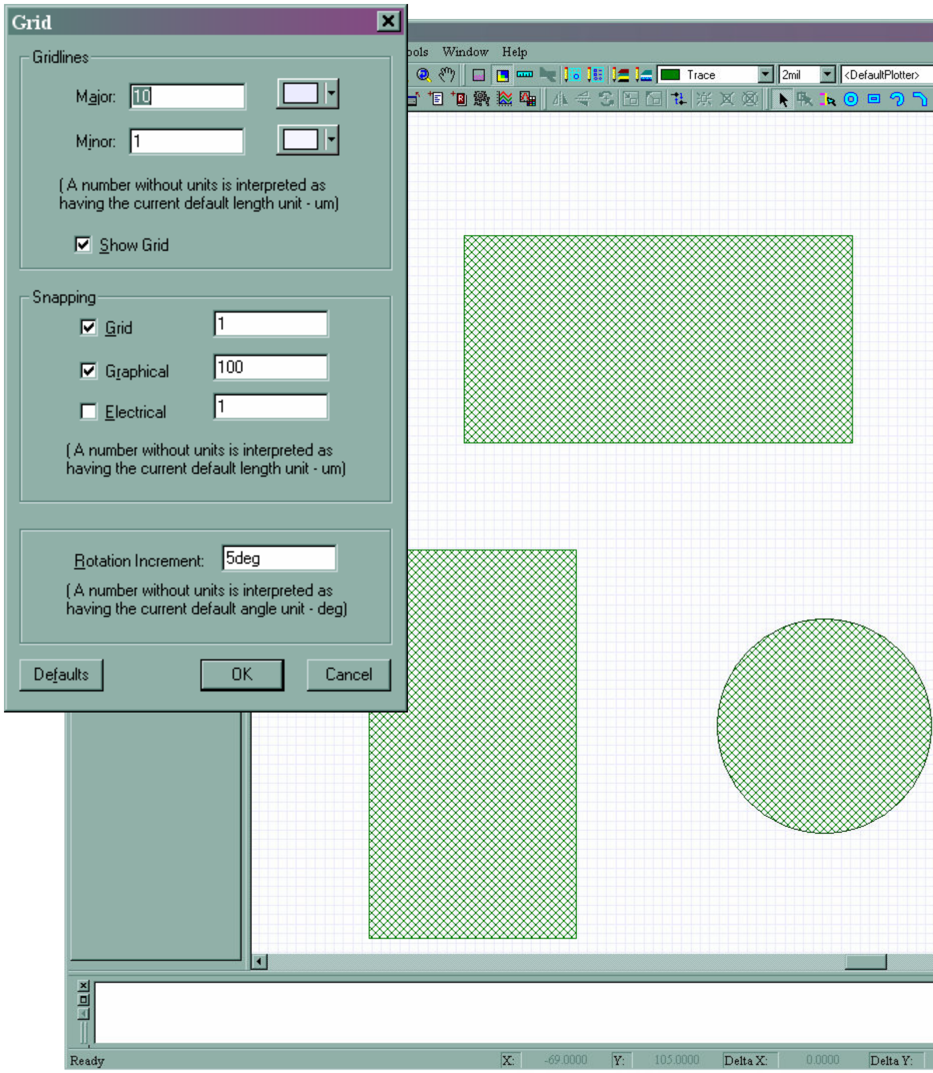
Specific Operation on Primitive

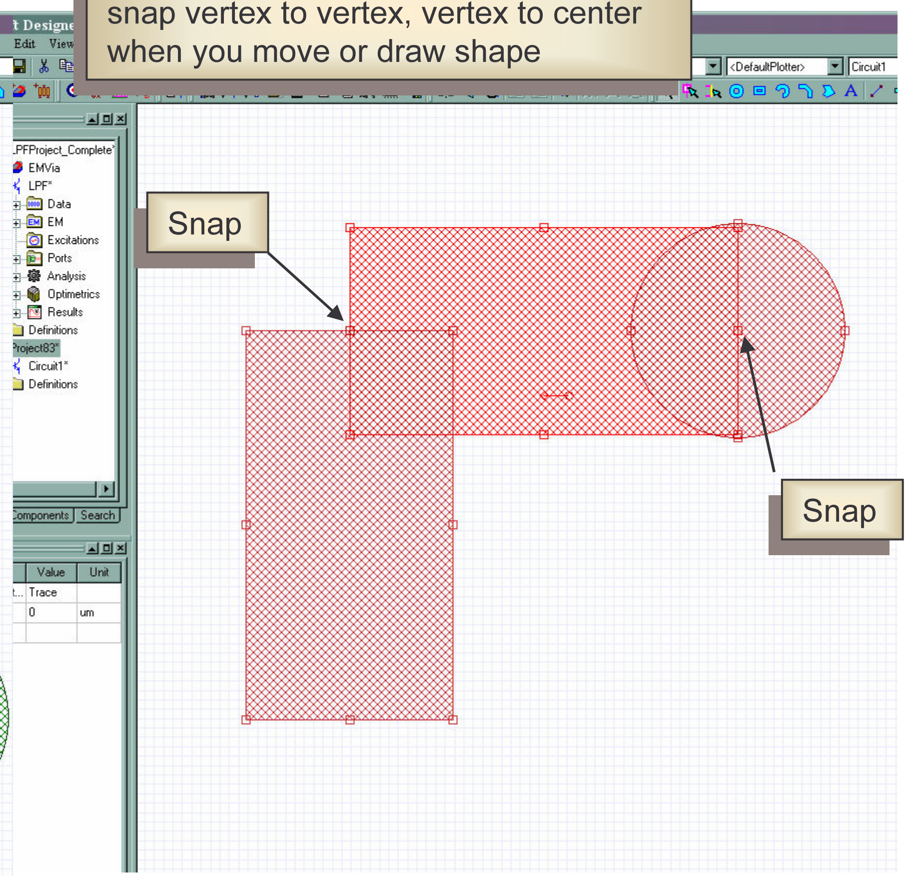


Snapping

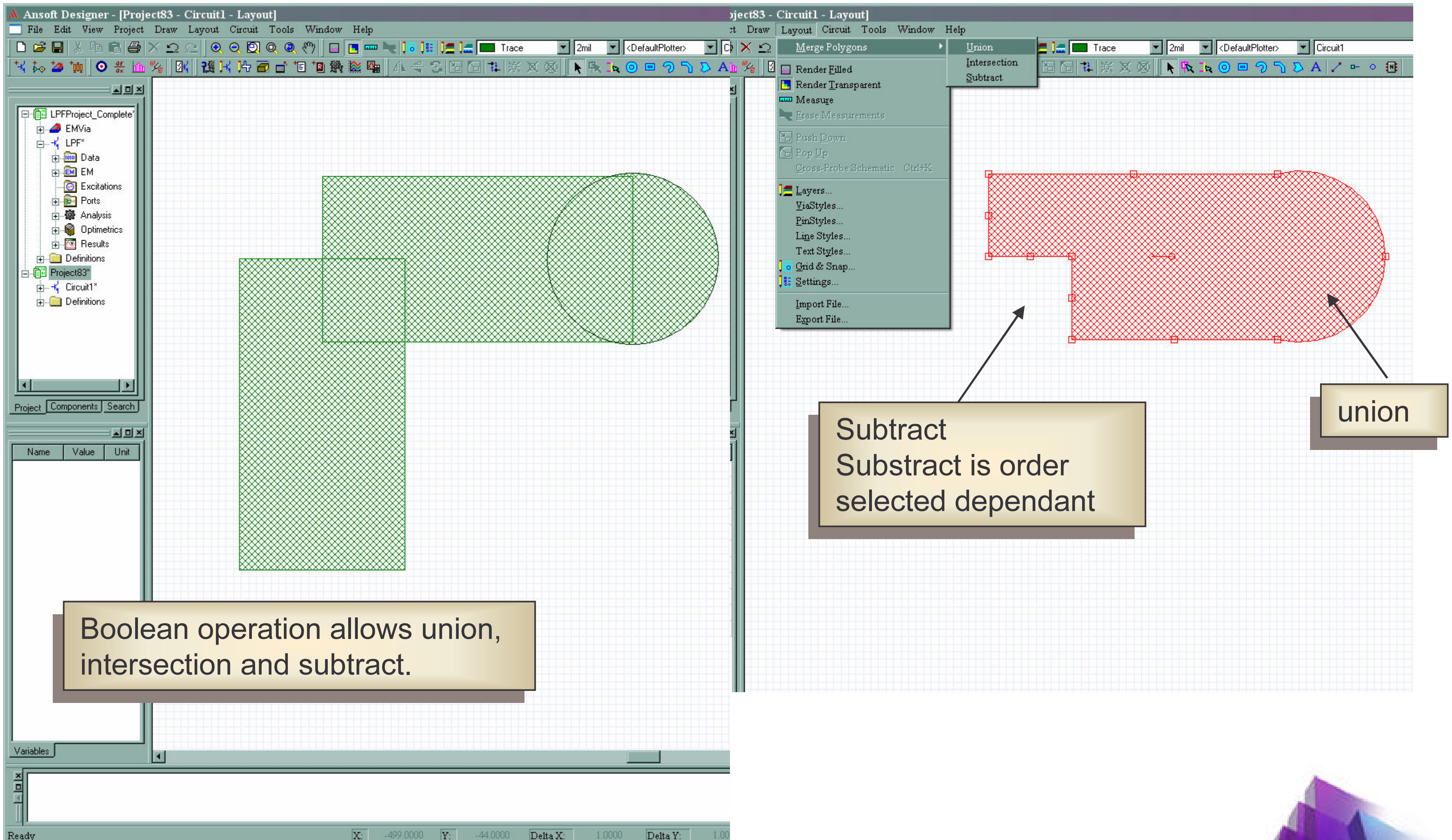
Snap to grid: snap the shape on the grid

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

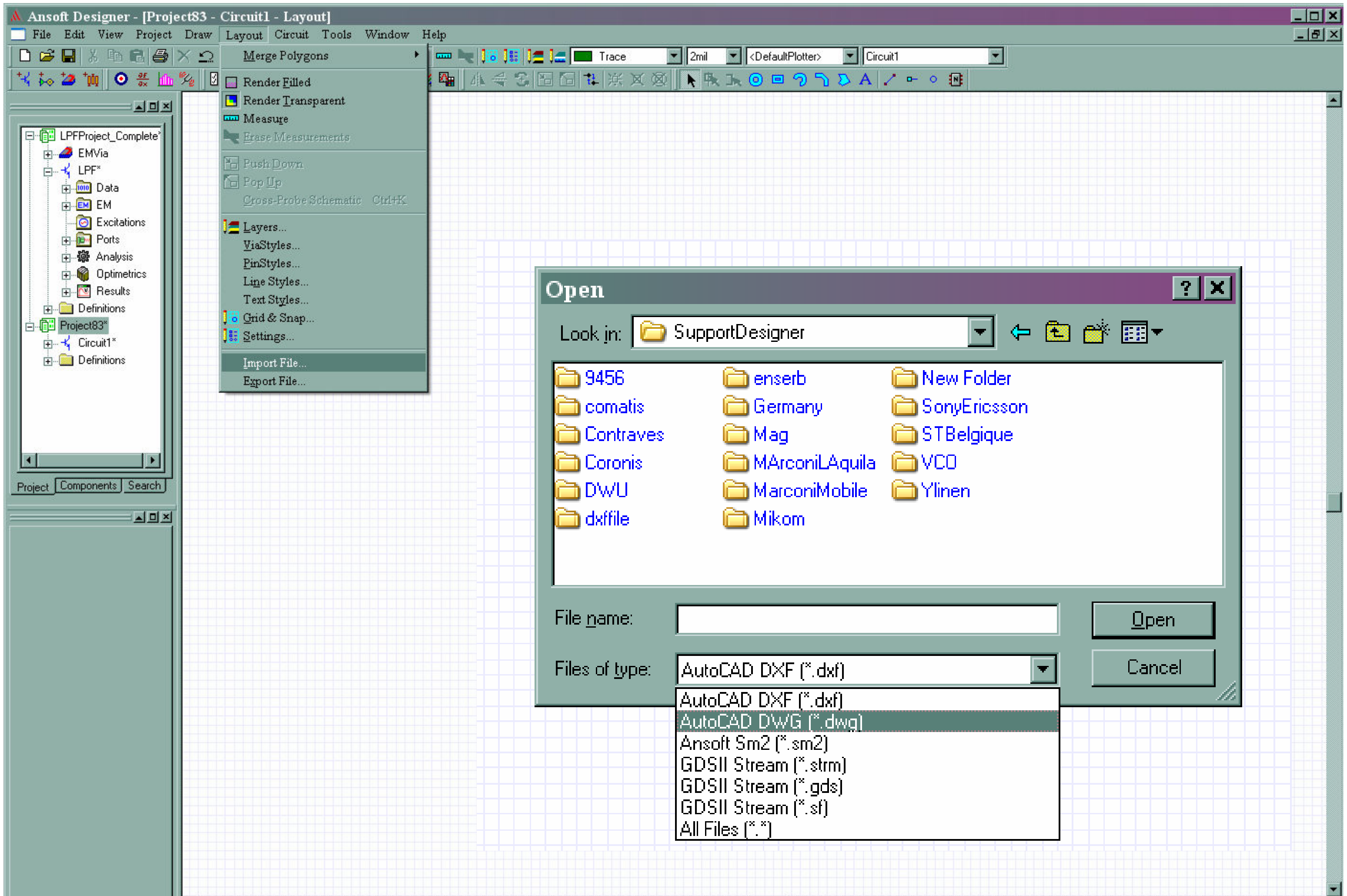




Boolean Operations

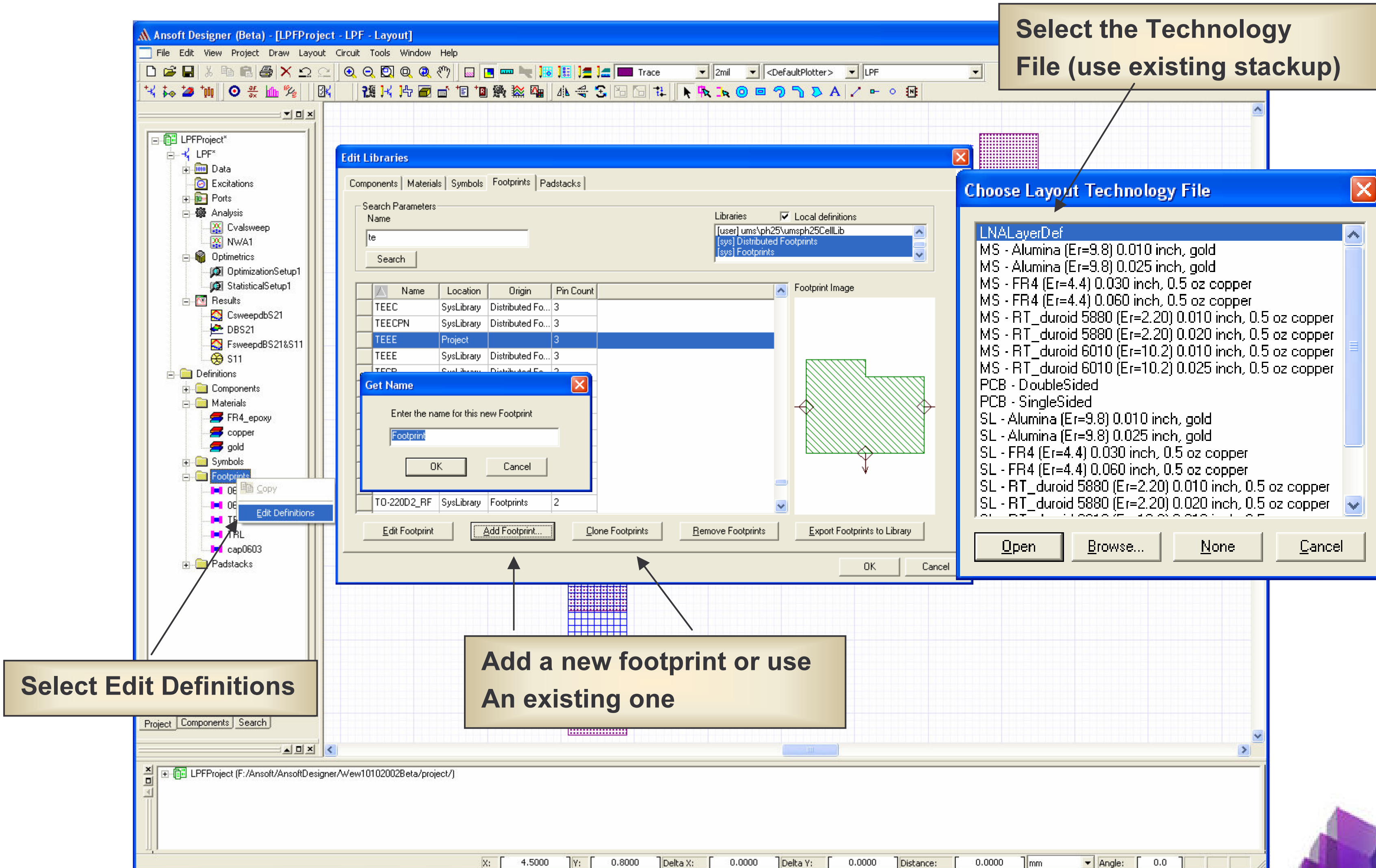


Import DXF and GDSII Files

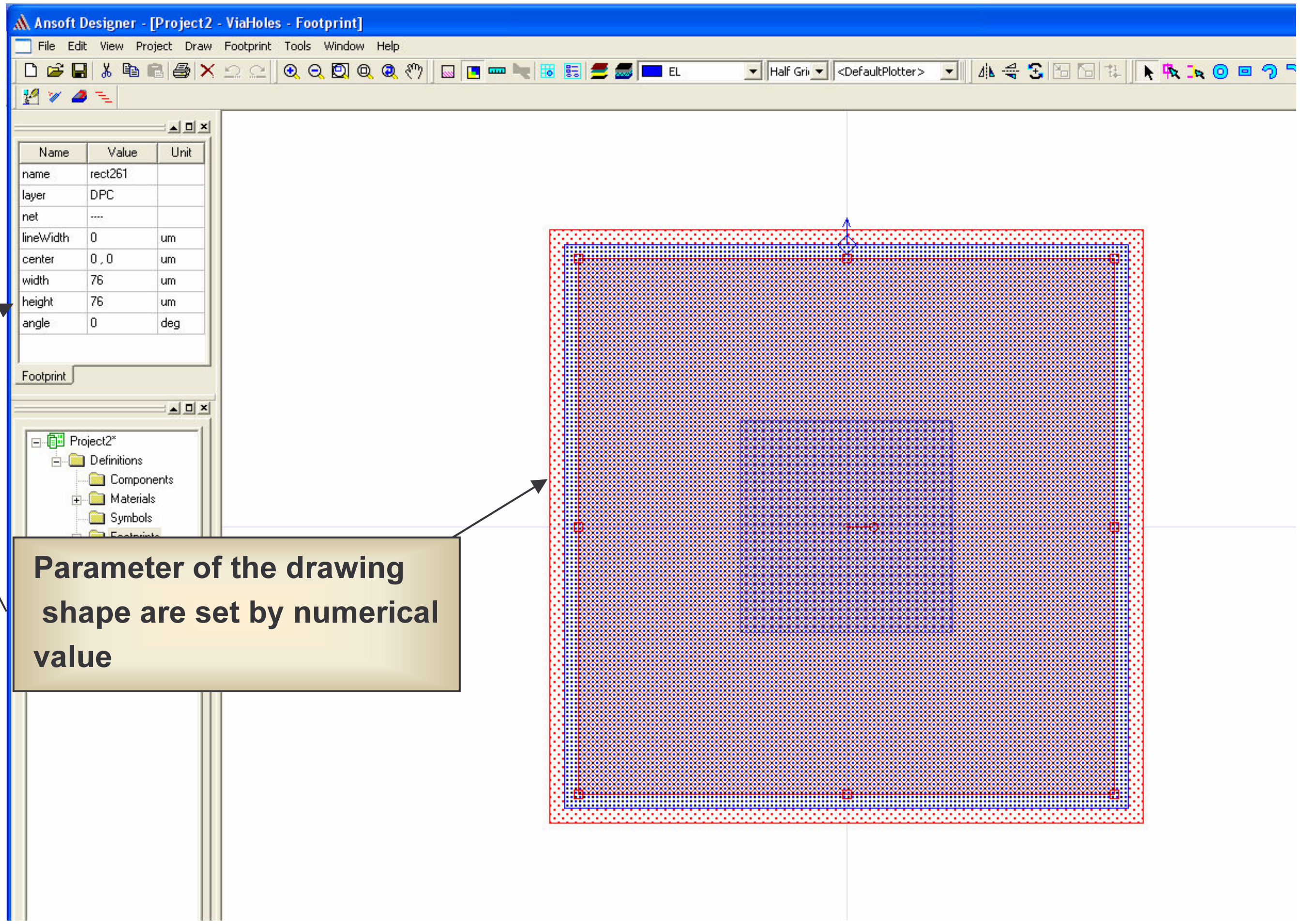


You can also import existing GDSII or DXF file.
Using map file allow you to map the layer names in Ansoft Designer with the GDSII layer number

Edit and Create Footprint

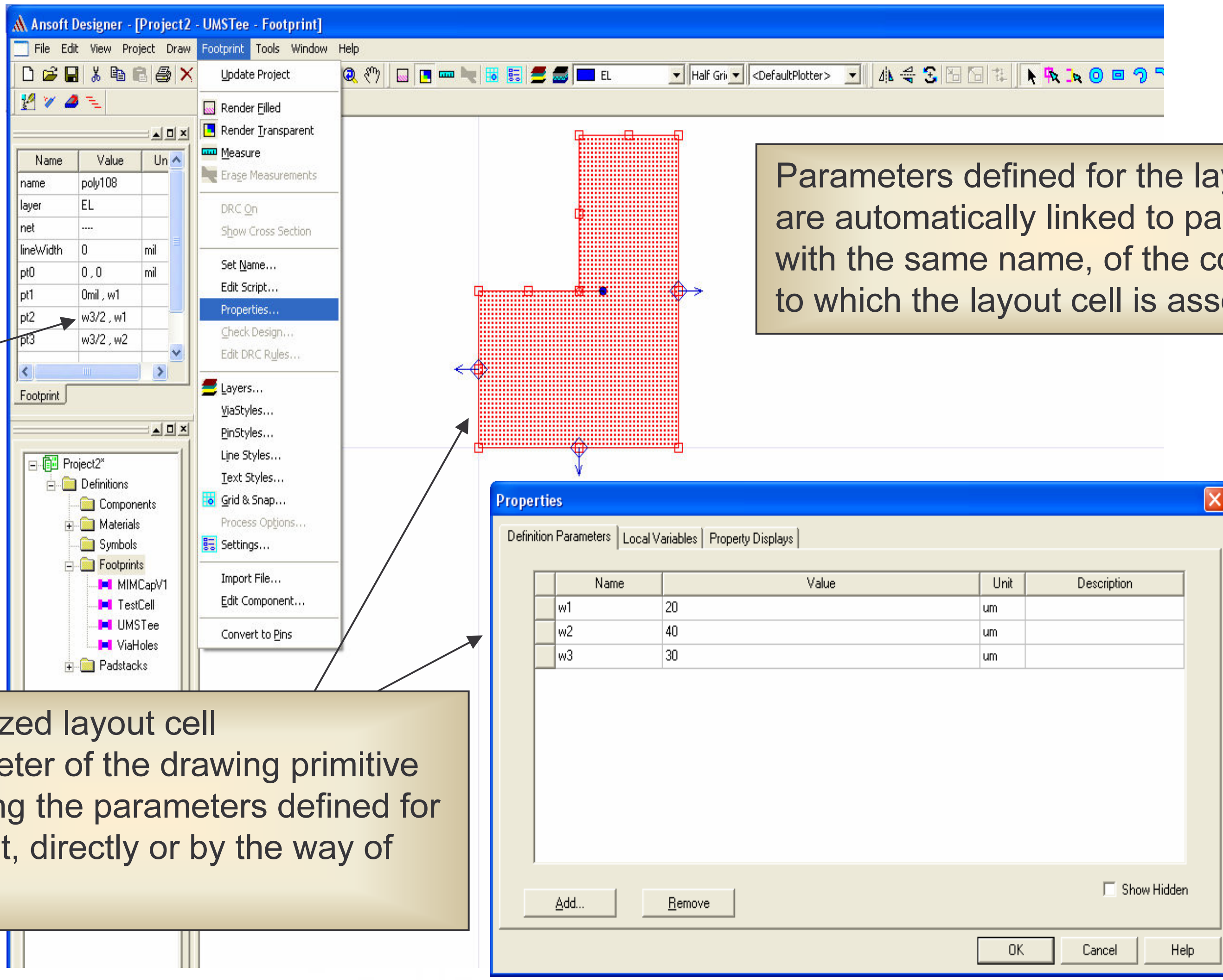


Fixed Footprint



Parameter of the drawing shape are set by numerical value

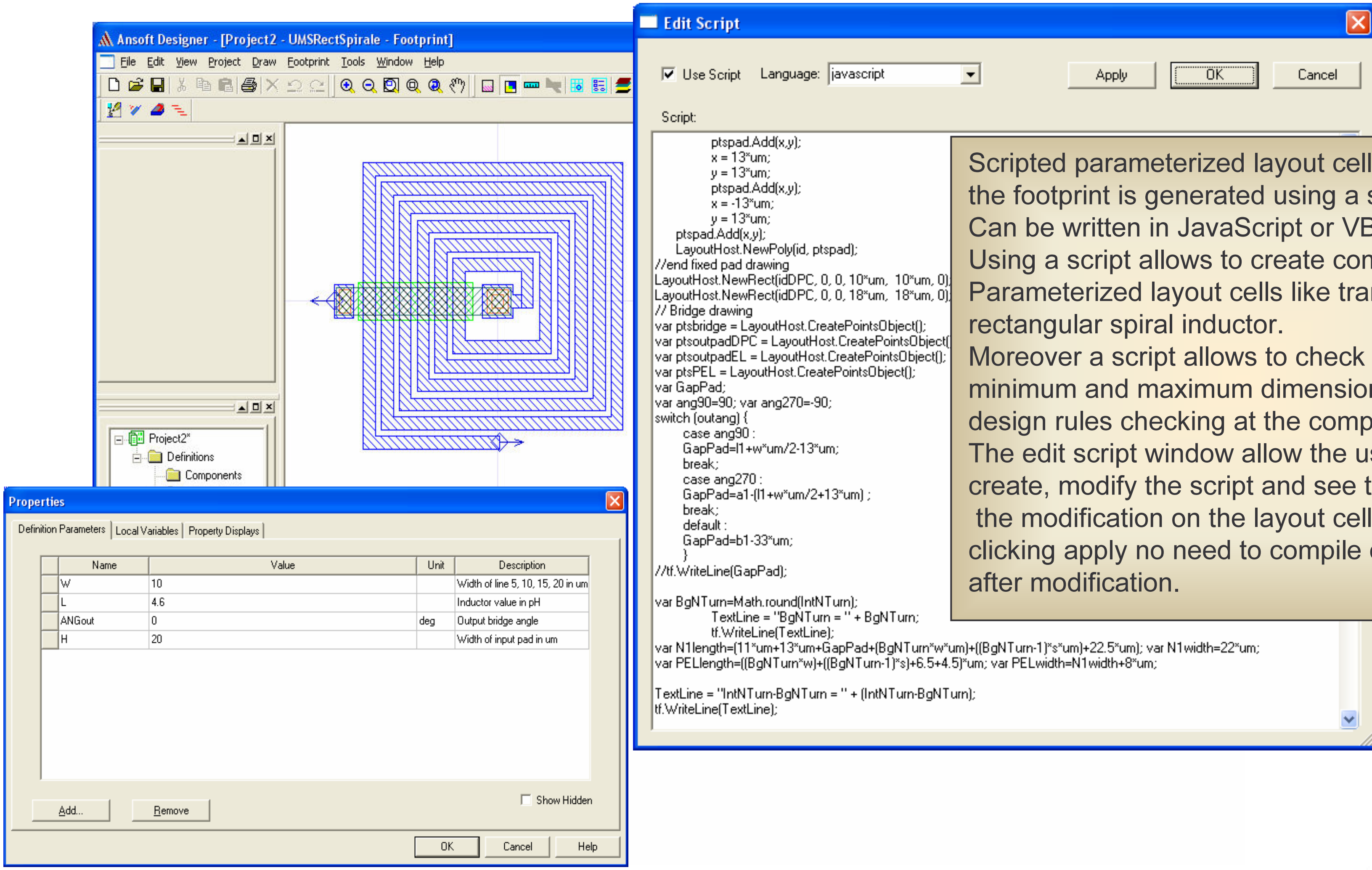
Parameterized Layout Cell



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

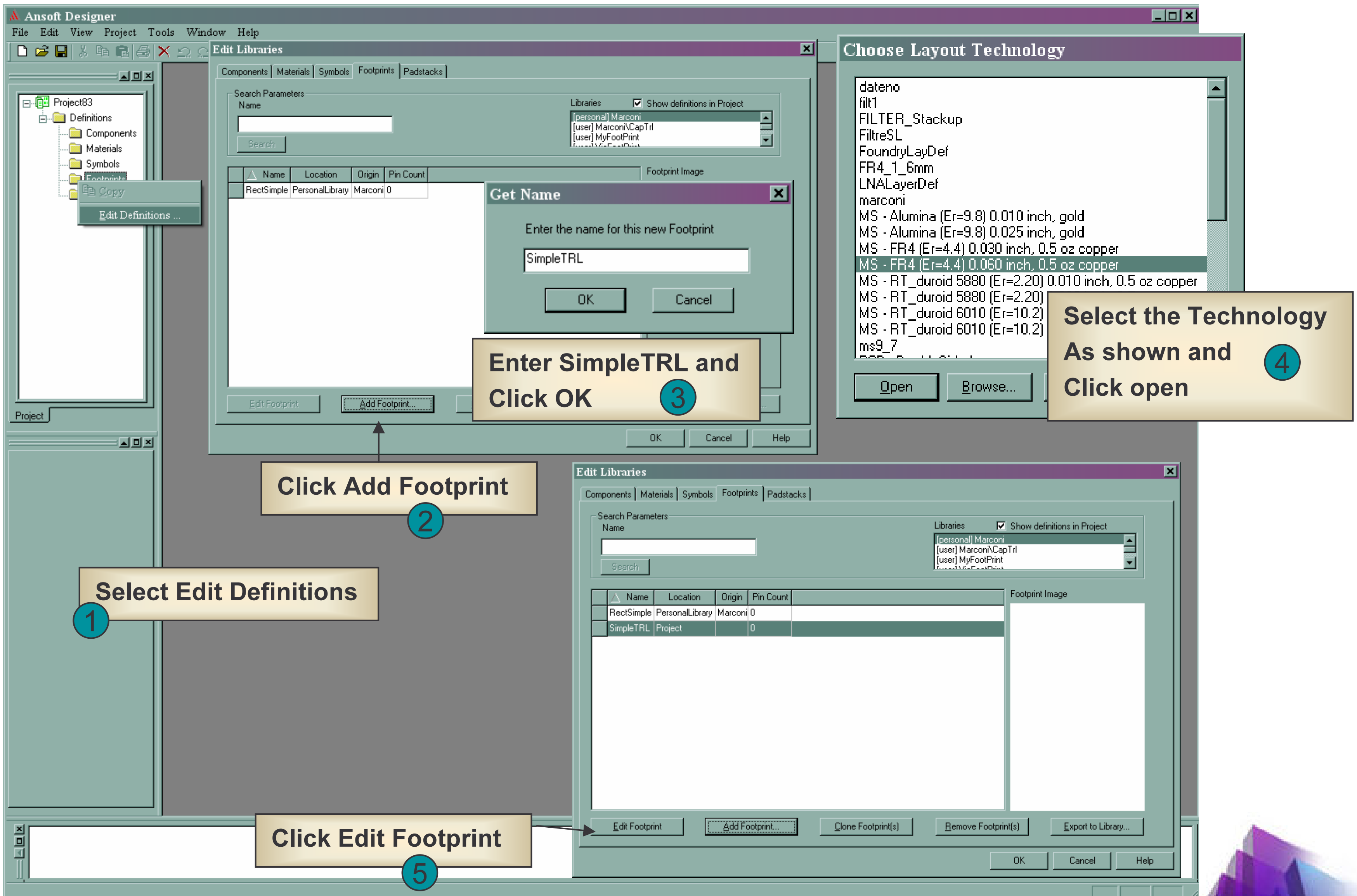


Scripted parameterized layout cell.
the footprint is generated using a script which
Can be written in JavaScript or VBScript.
Using a script allows to create complex
Parameterized layout cells like transistor,
rectangular spiral inductor.
Moreover a script allows to check
minimum and maximum dimensions 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.

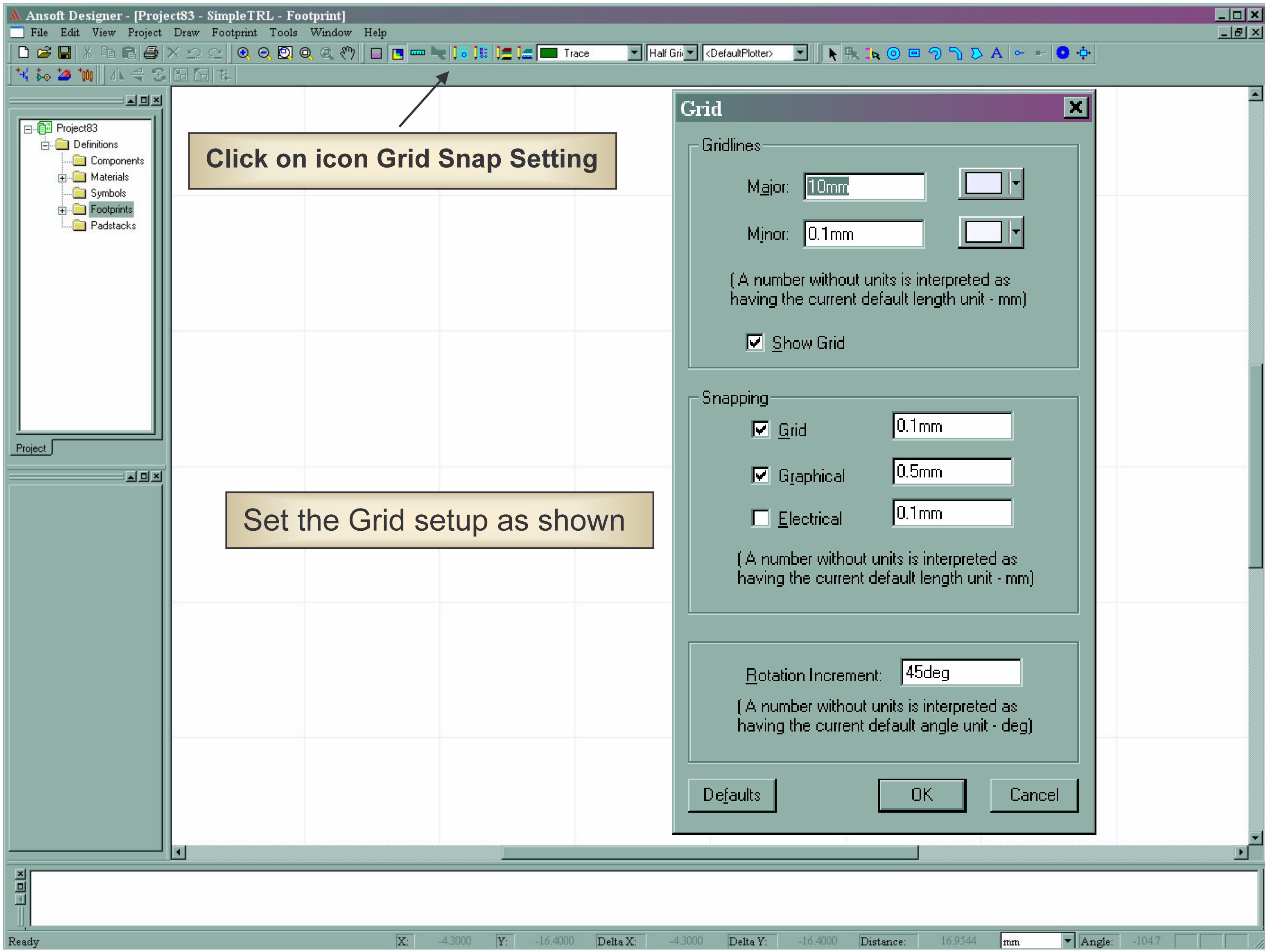
Exercise 2: Creating A Footprint

- 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

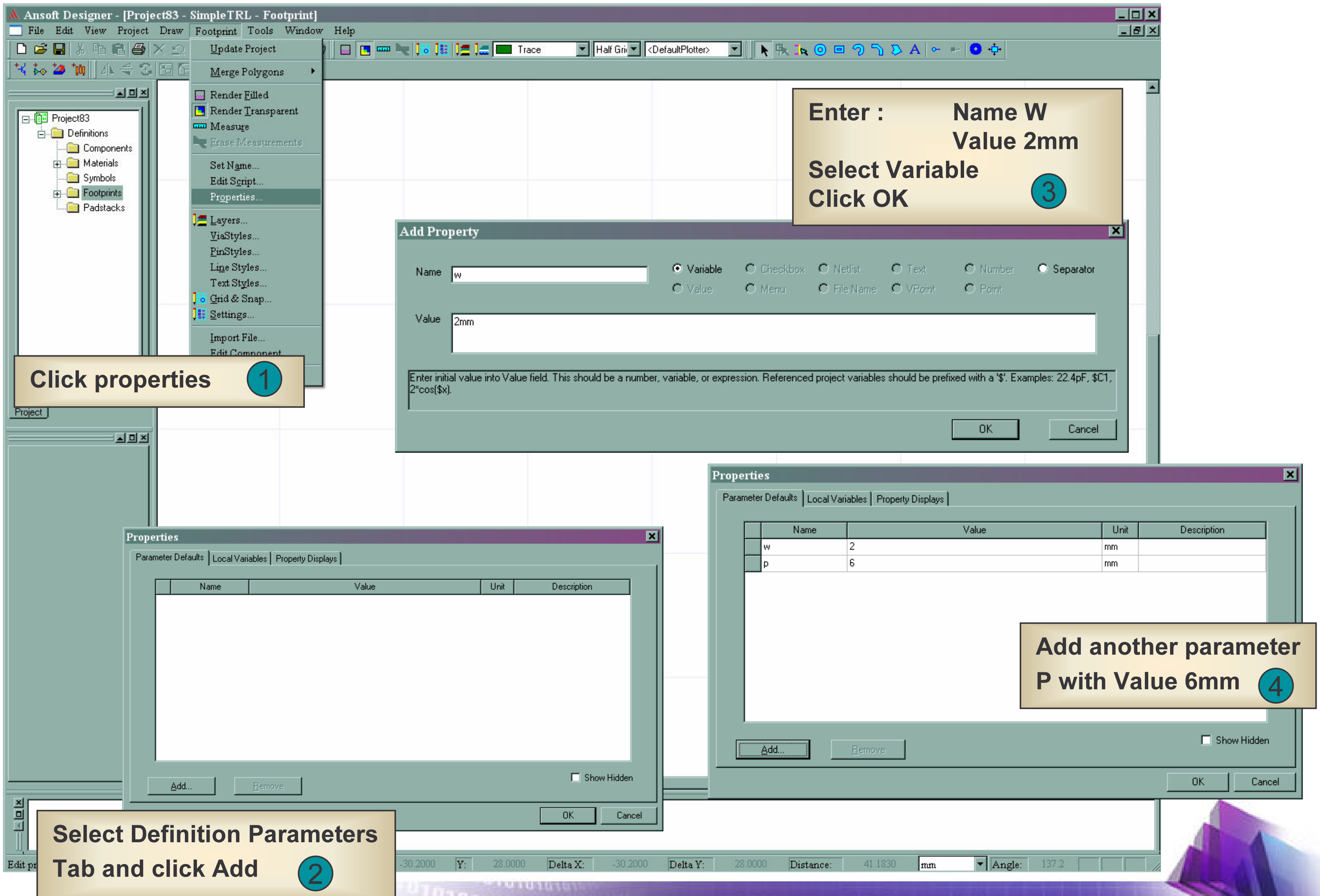


Set the Grid

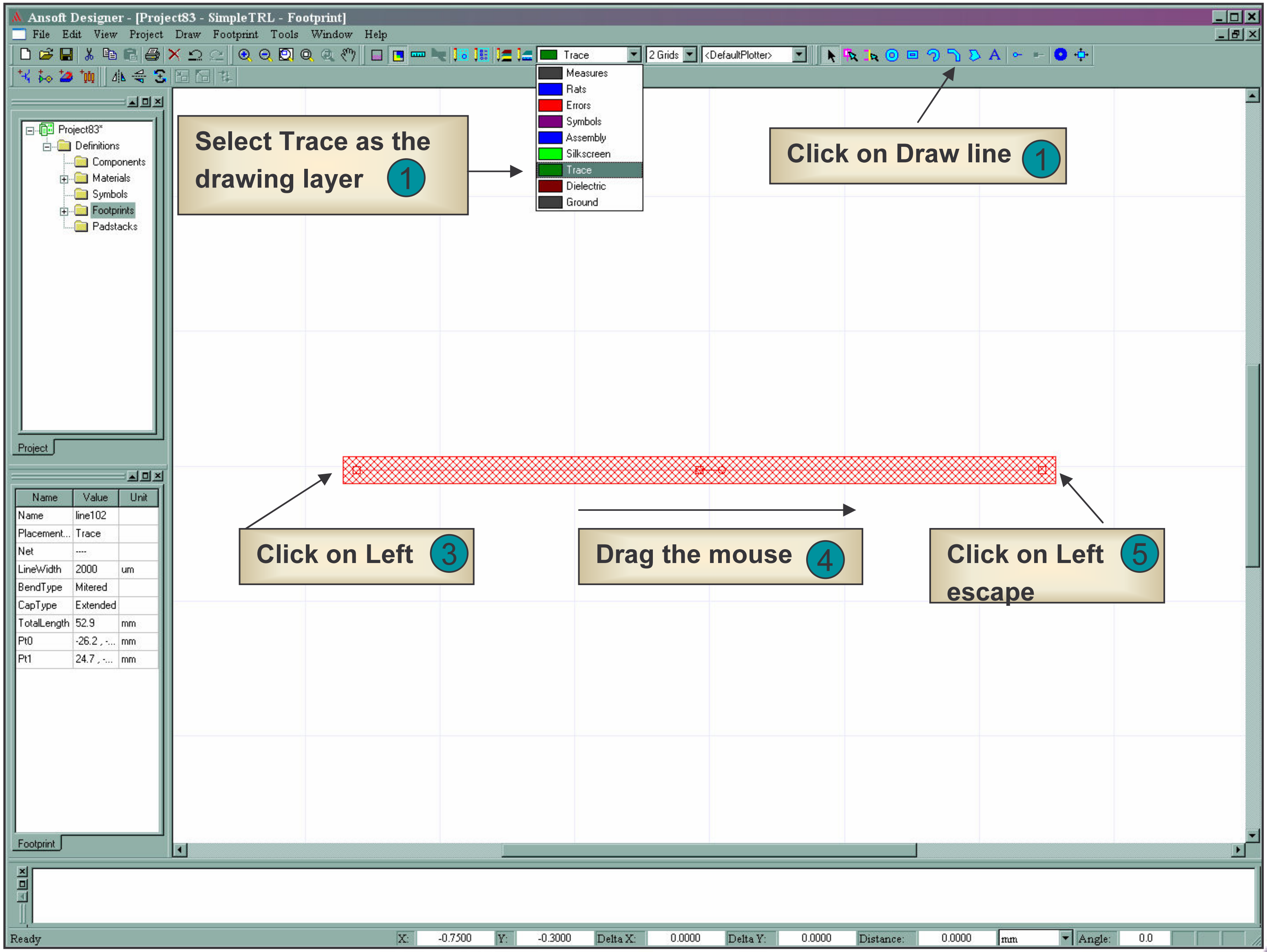


Click on icon Grid Snap Setting
Set the Grid setup as shown
Grid
Gridlines
Major: 10mm
Minor: 0.1mm
(A number without units is interpreted as having the current default length unit - mm)
Show Grid
Snapping
Grid 0.1mm
Graphical 0.5mm
Electrical 0.1mm
(A number without units is interpreted as having the current default length unit - mm)
Rotation Increment: 45deg
(A number without units is interpreted as having the current default angle unit - deg)
Defaults
OK
Cancel

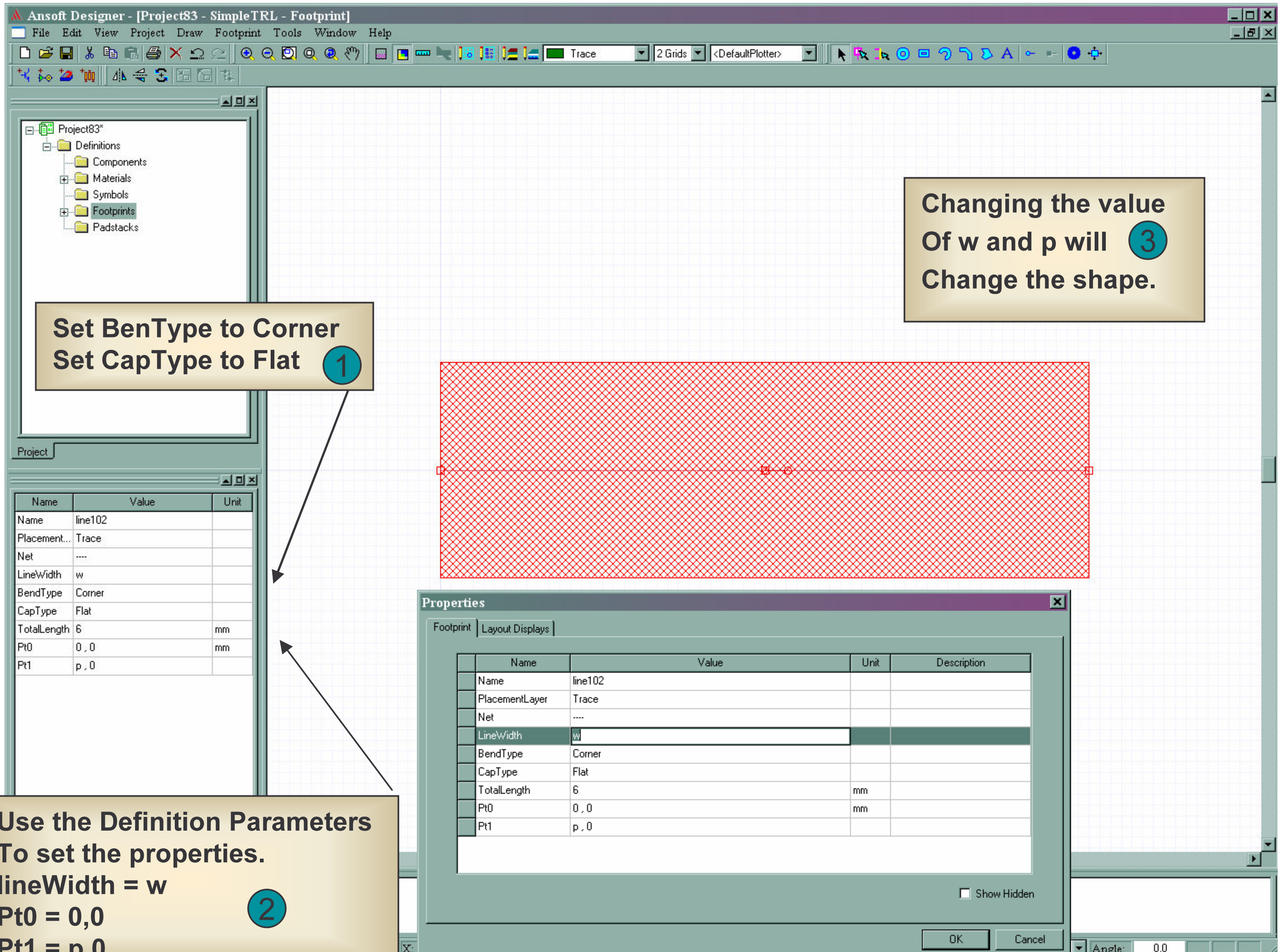
Define the parameters



Draw Shape


Ansoft Designer - [Project83 - SimpleTRL - Footprint]
File Edit View Project Draw Footprint Tools Window Help
Trace
2 Grids
<DefaultPlotter>
Measures
Rats
Errors
Symbols
Assembly
Silkscreen
Trace
Dielectric
Ground
Project83*
Definitions
Components
Materials
Symbols
Footprints
Padstacks
Project
Select Trace as the drawing layer 1
Click on Draw line 1
Click on Left 3
Drag the mouse 4
Click on Left escape 5
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
Footprint
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

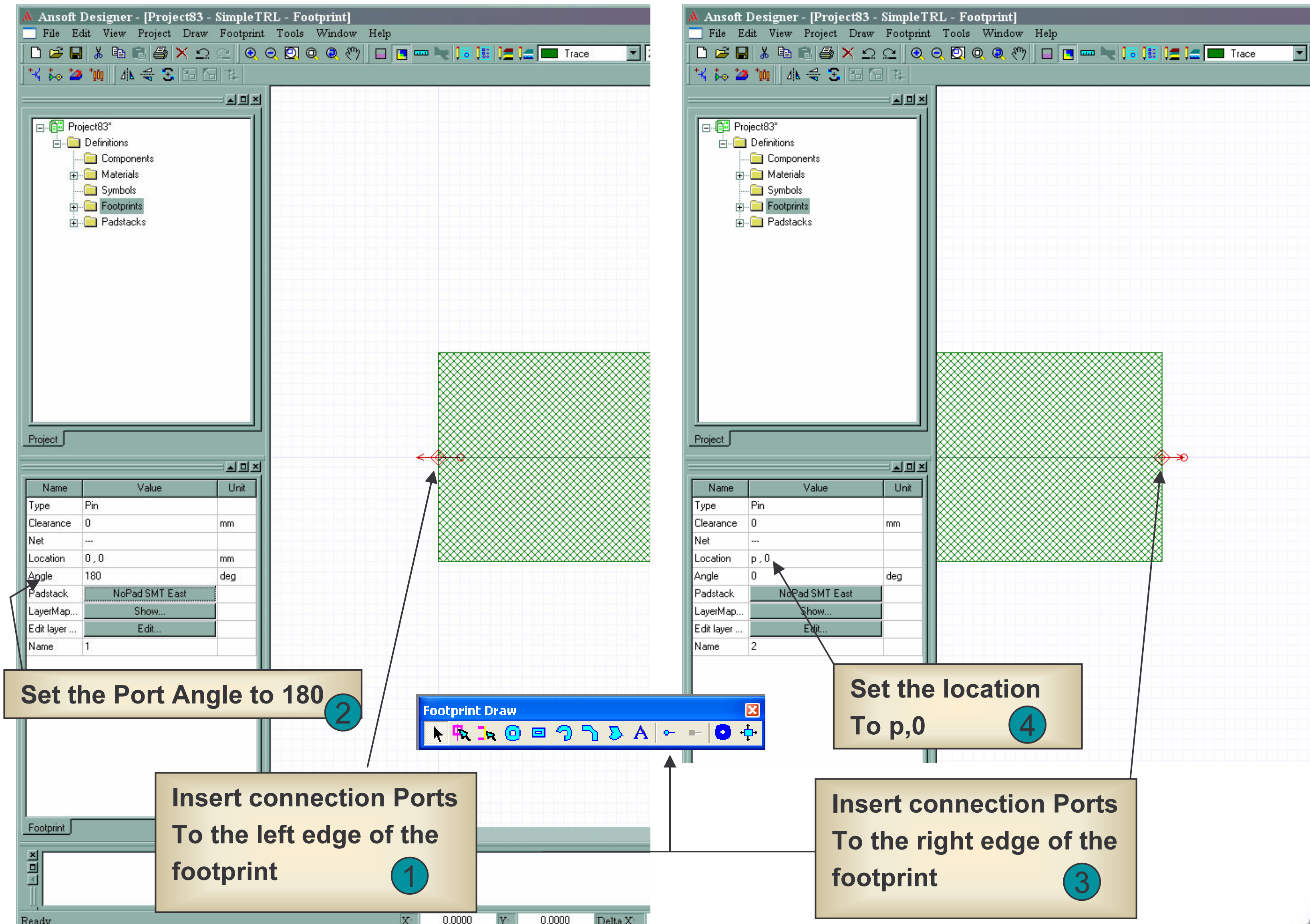


Set BenType to Corner
Set CapType to Flat
1

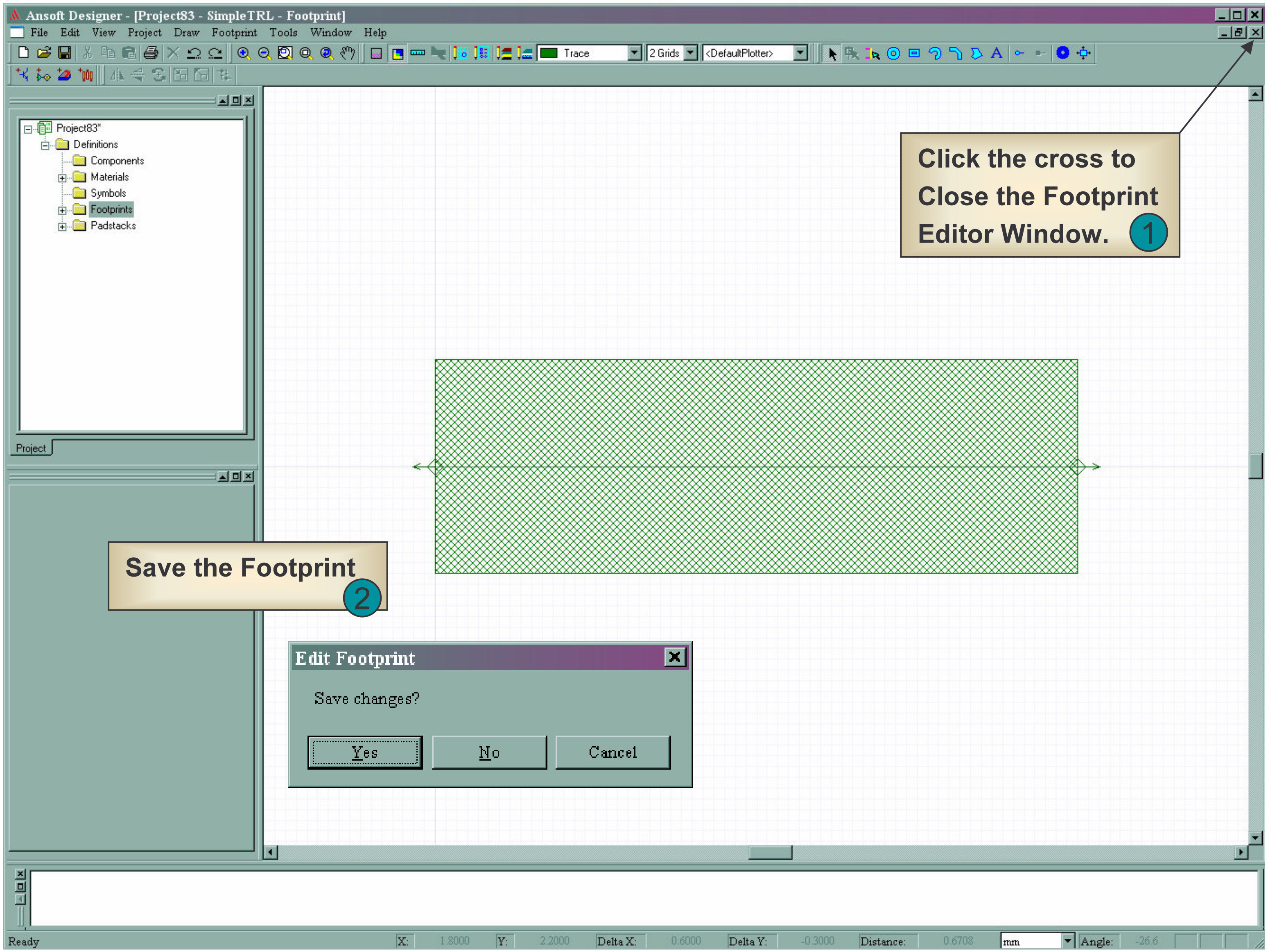
Changing the value
Of w and p will
3
Change the shape.

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

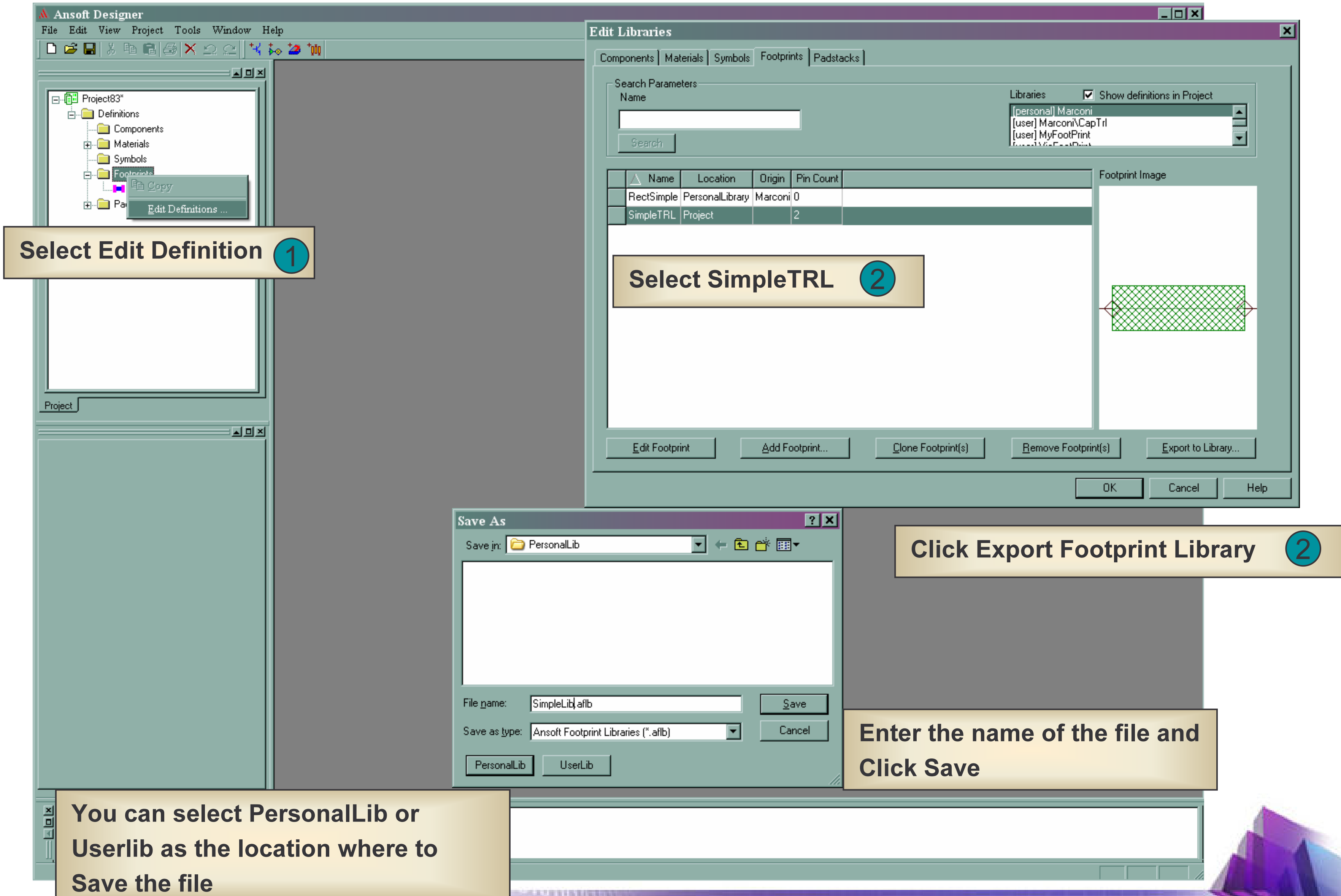
Add Connection Ports



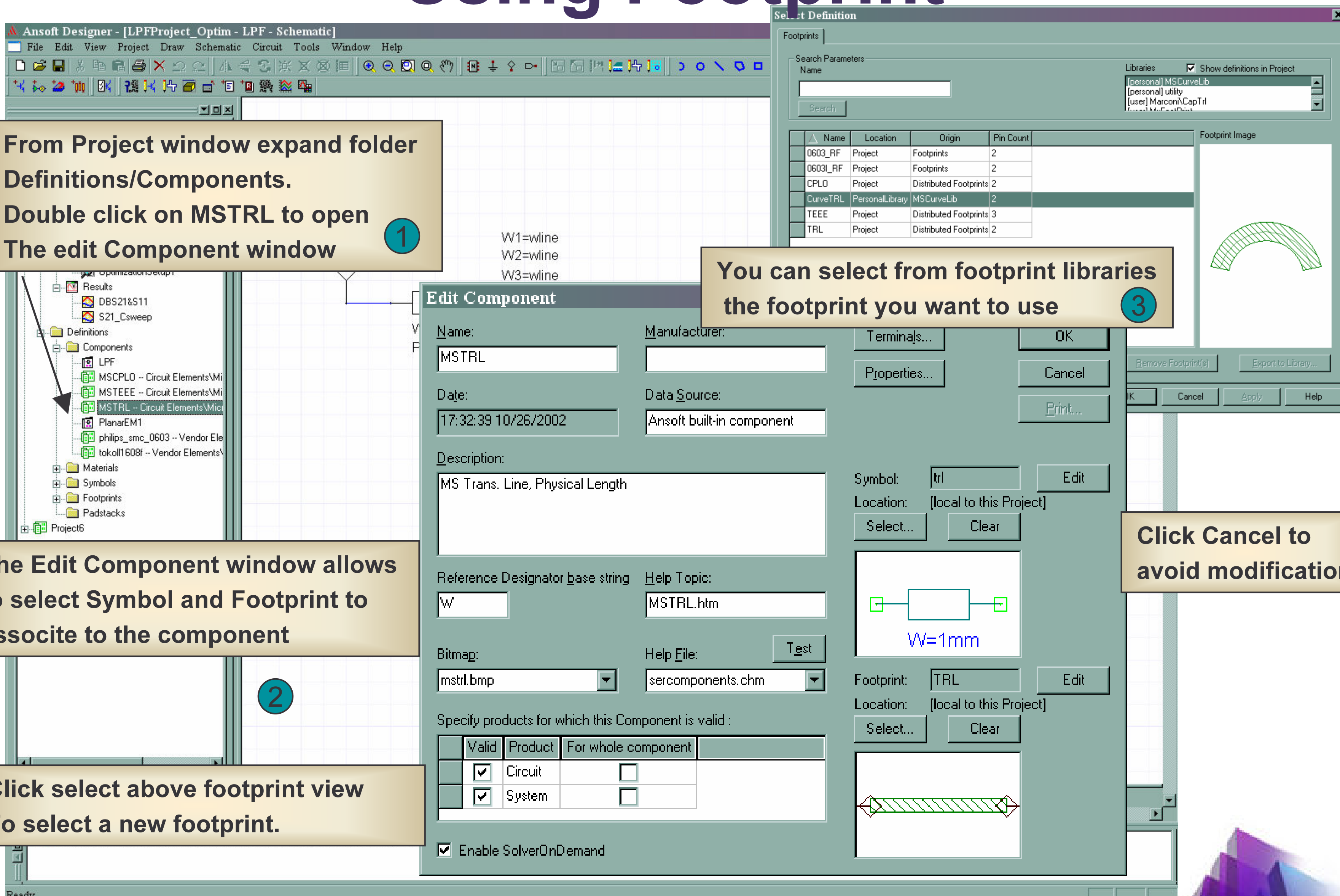
Save The Layout Cell to Library



Export The Layout Cell to Library



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

Click Cancel to avoid modification

The Edit Component window allows to select Symbol and Footprint to associte to the component

2

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

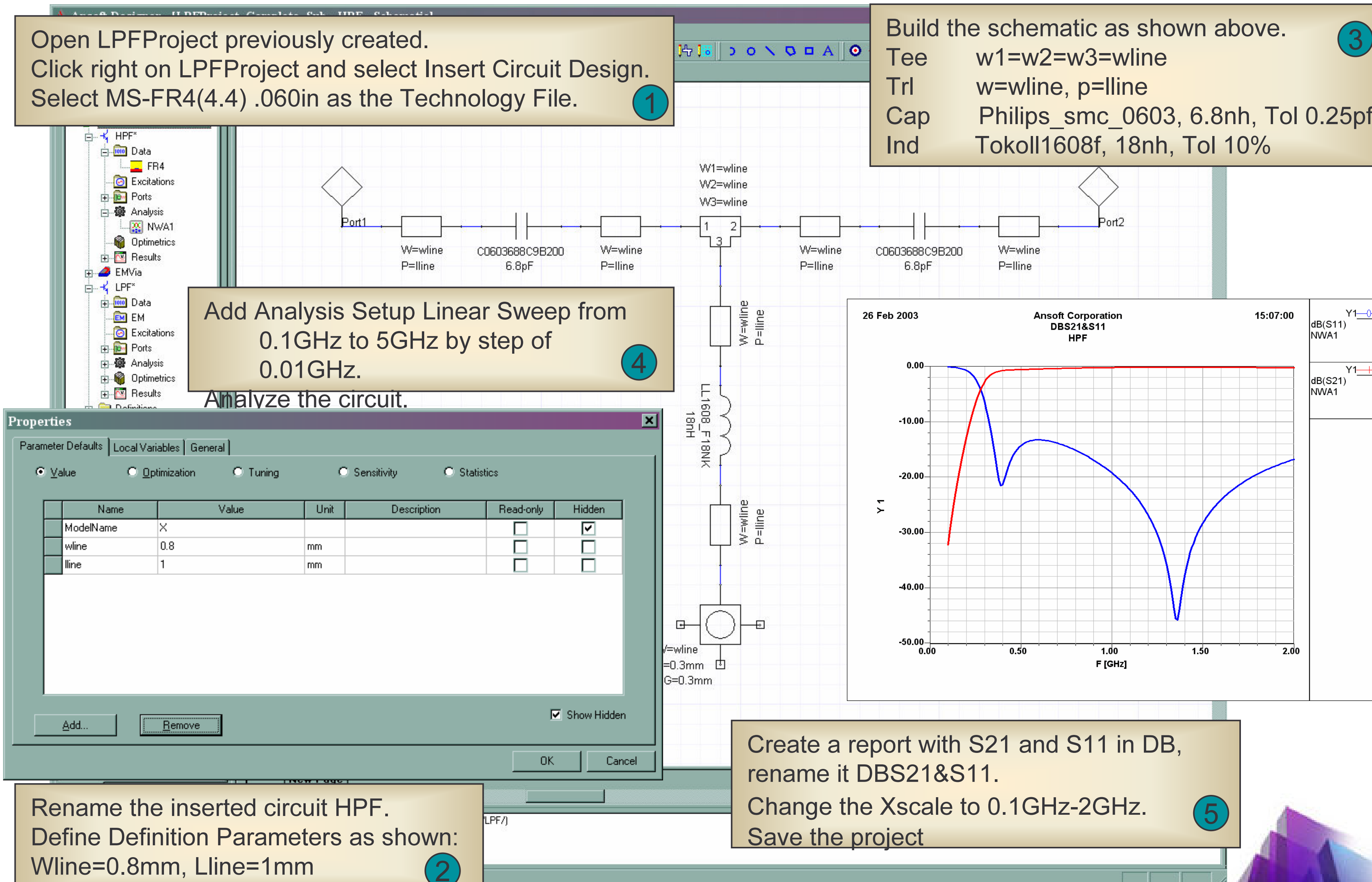
Rename the inserted circuit HPF.
Define Definition Parameters as shown:
Wline=0.8mm, Lline=1mm 2

Build the schematic as shown above. 3

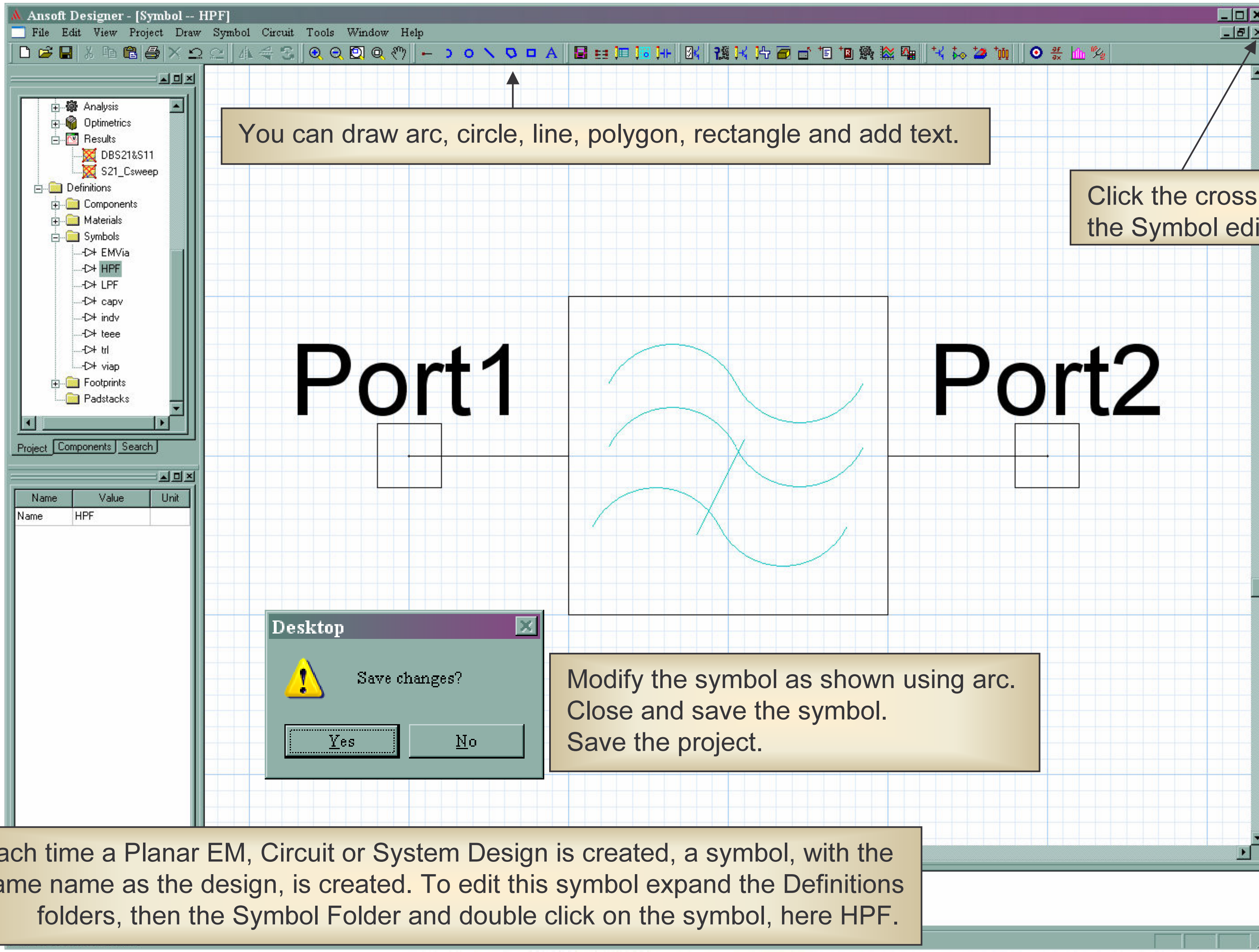
Tee	w1=w2=w3=wline
Trl	w=wline, p=lline
Cap	Philips_smc_0603, 6.8nh, Tol 0.25pf
Ind	Tokoll1608f, 18nh, Tol 10%

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



Edit The Symbol and Modify It



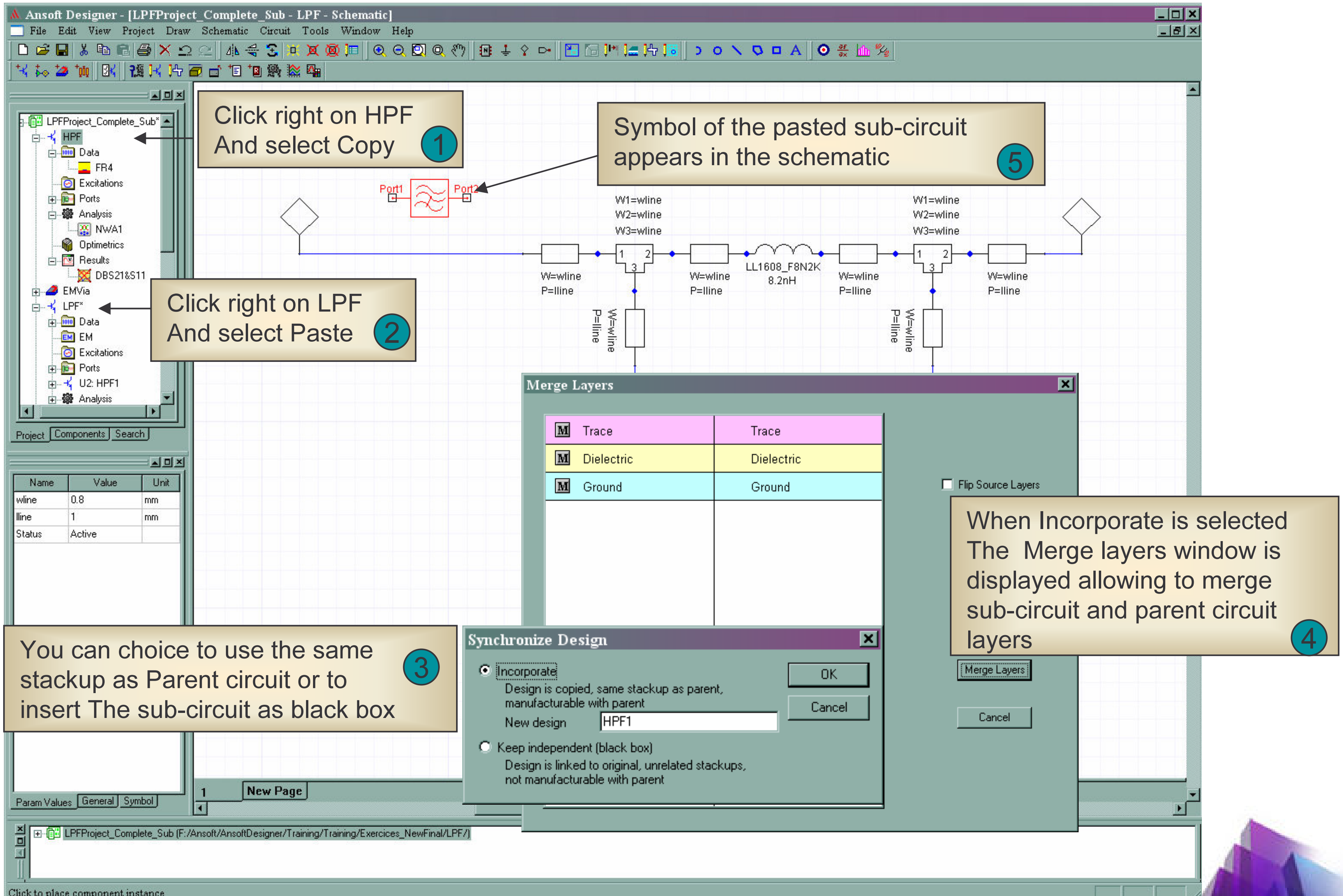
You can draw arc, circle, line, polygon, rectangle and add text.

Click the cross to close the Symbol editor widow

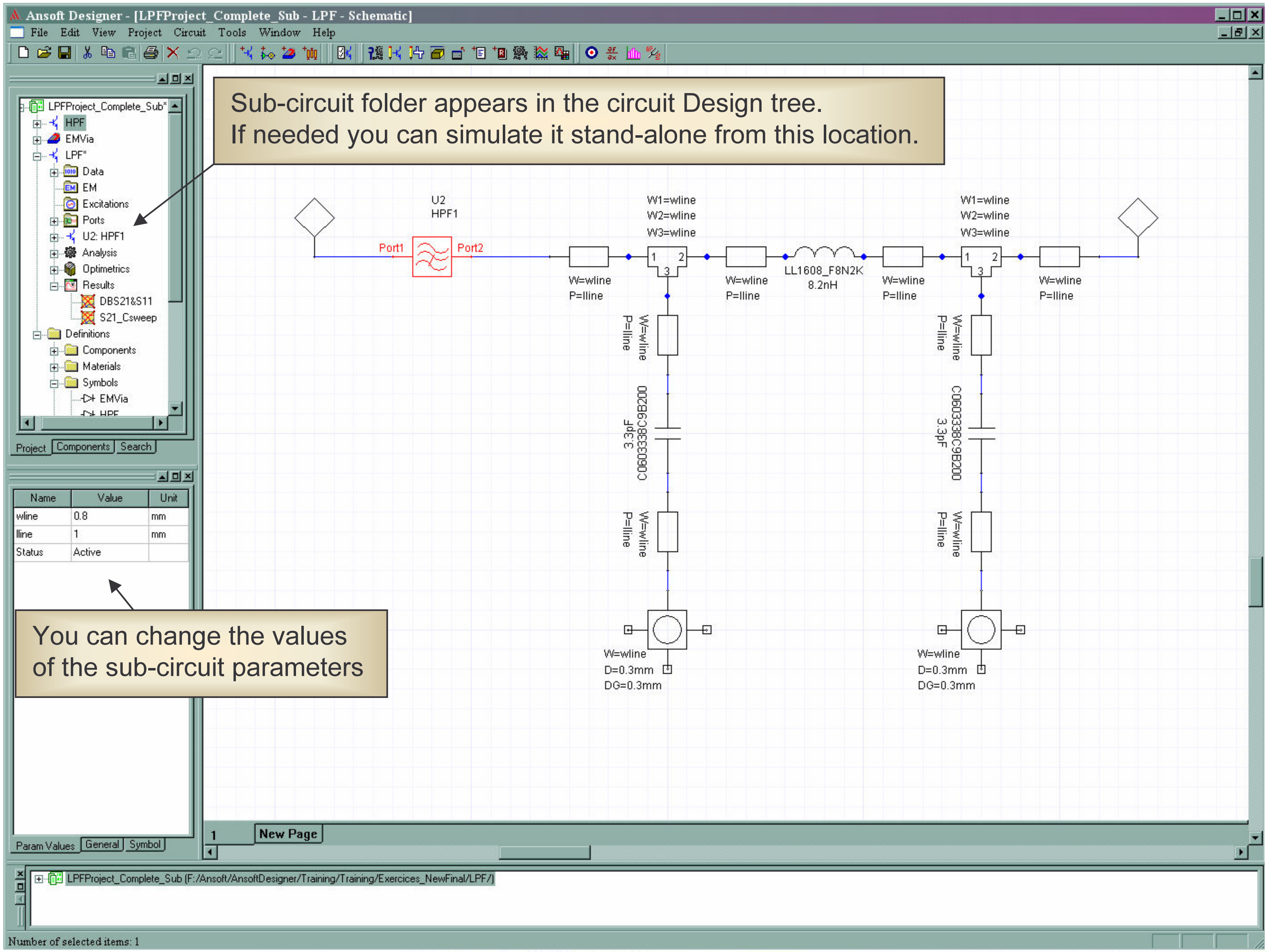
Modify the symbol as shown using arc.
Close and save the symbol.
Save the project.

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



Connect the Sub-Circuit Symbol

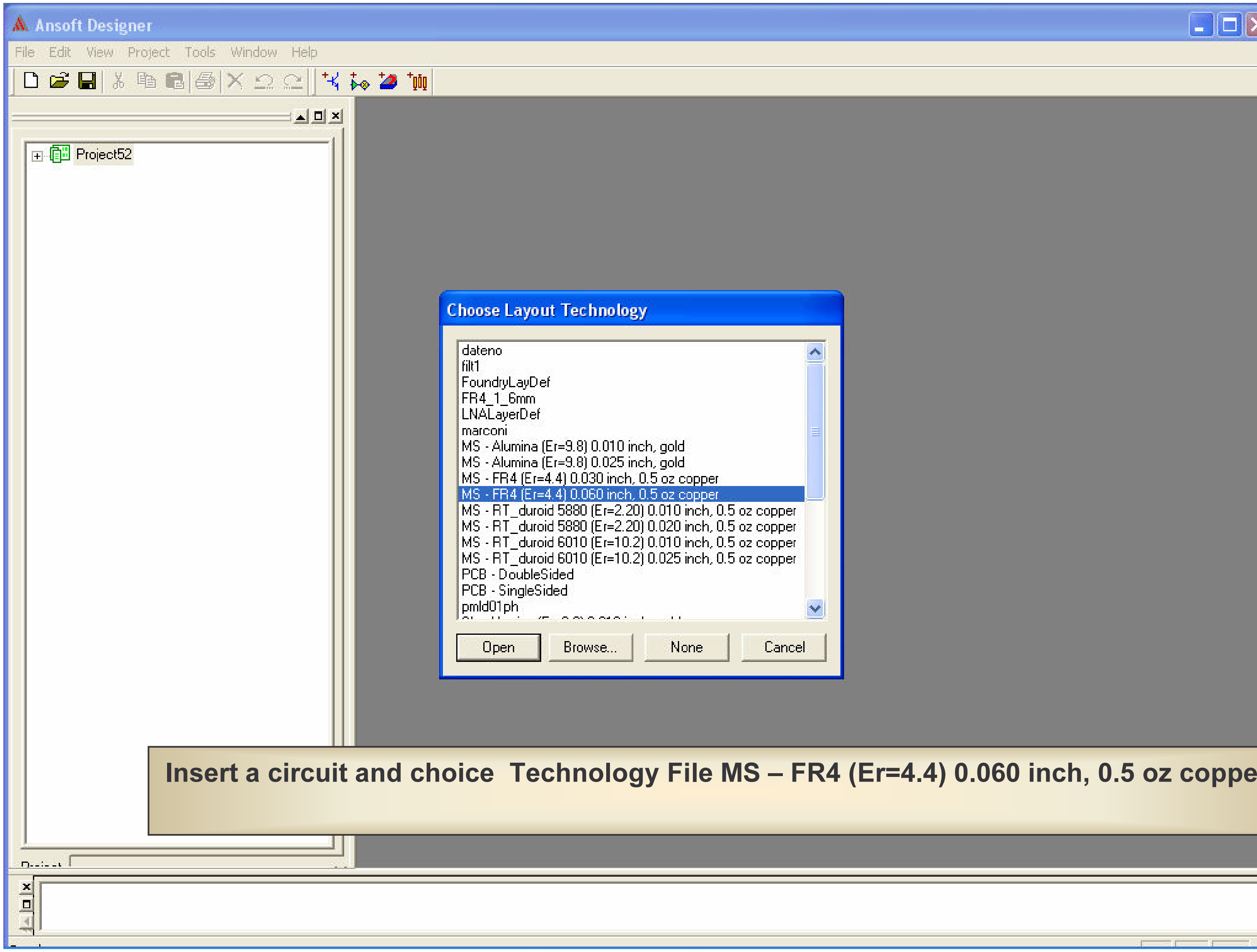


Name	Value	Unit
wline	0.8	mm
lline	1	mm
Status	Active	

End of Exercise

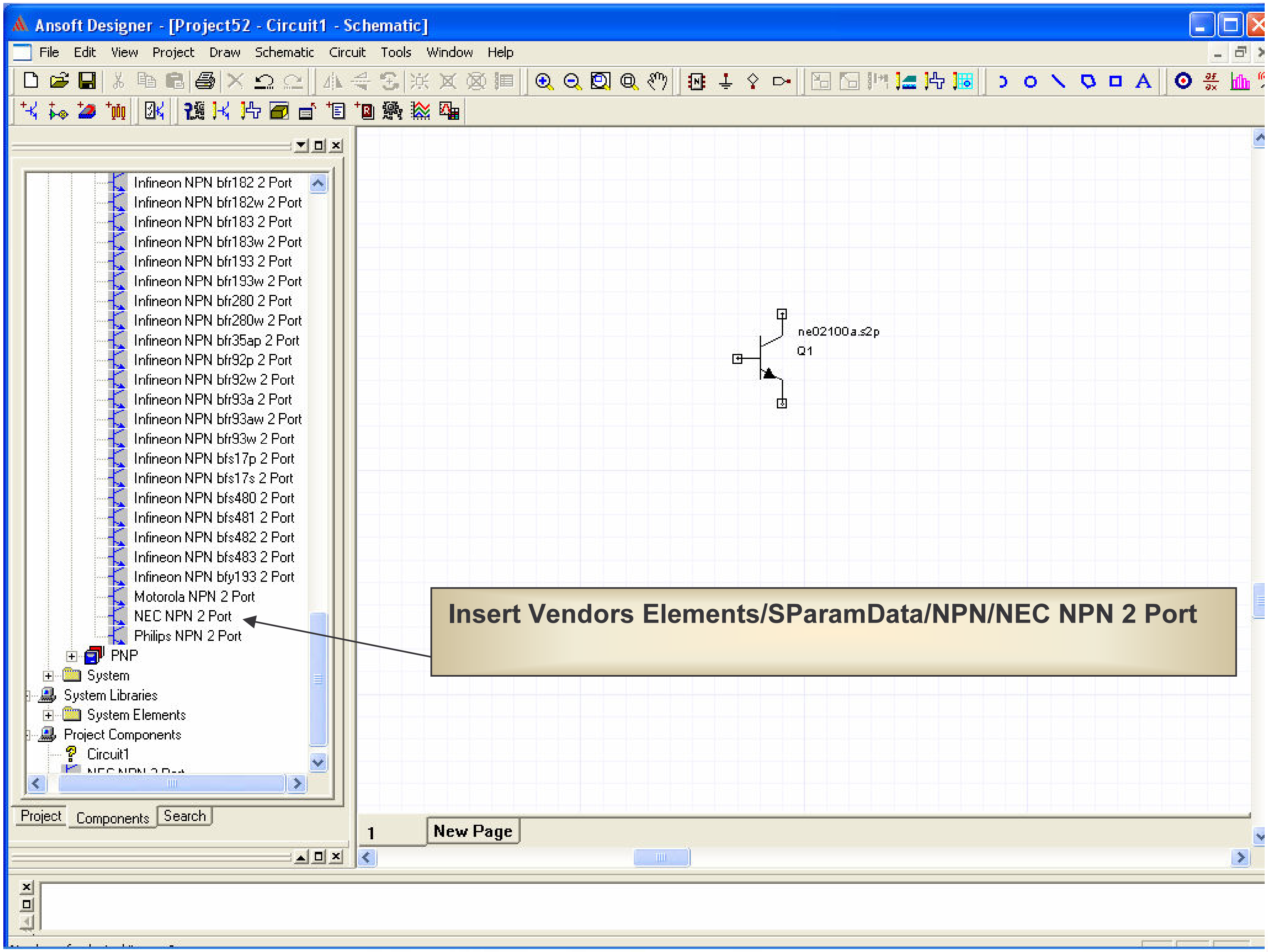
Smith Tool .avi

Create a New Circuit

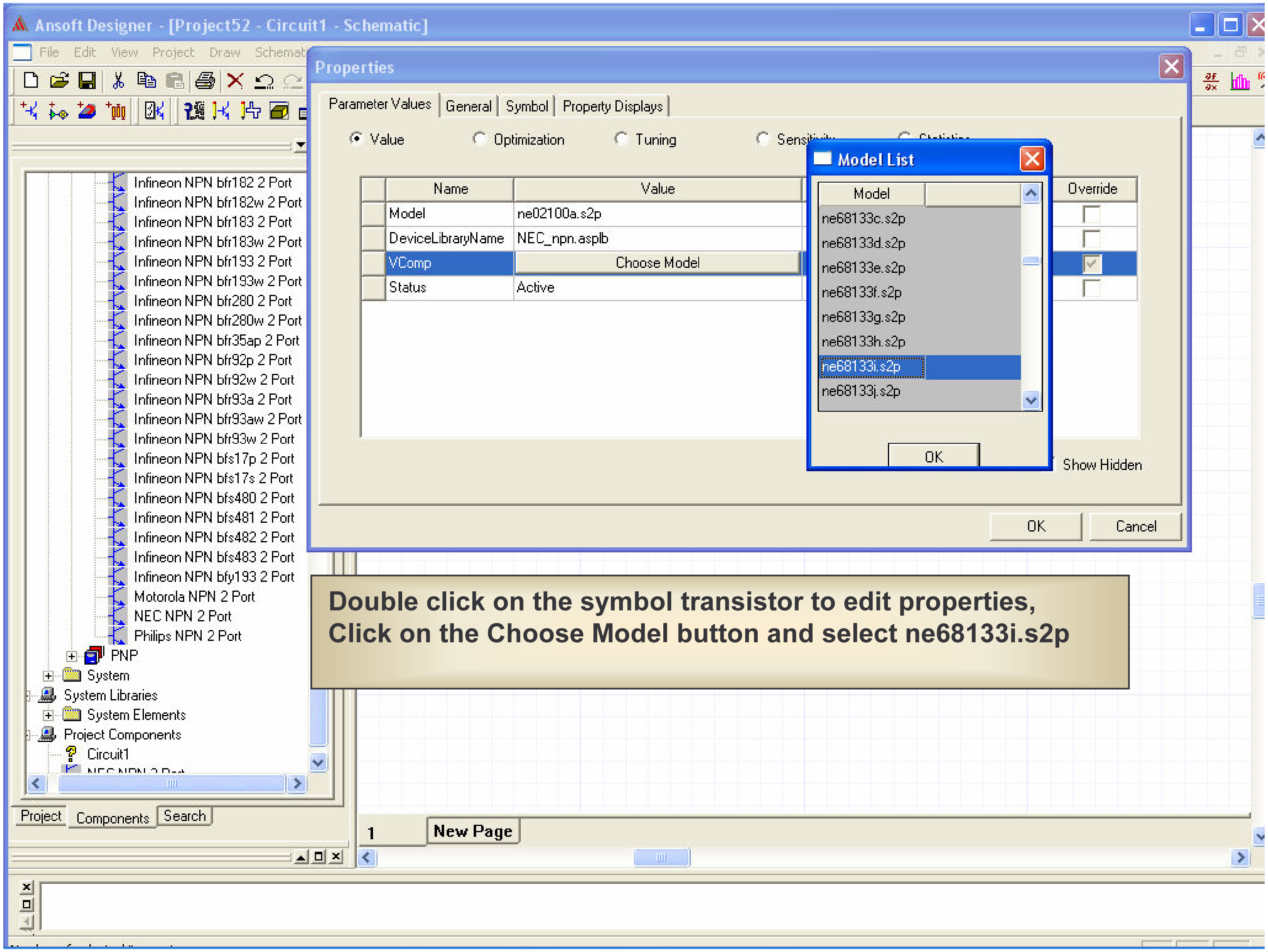


Insert a circuit and choice Technology File MS – FR4 (Er=4.4) 0.060 inch, 0.5 oz copper.

Insert NPN NEC Transistor

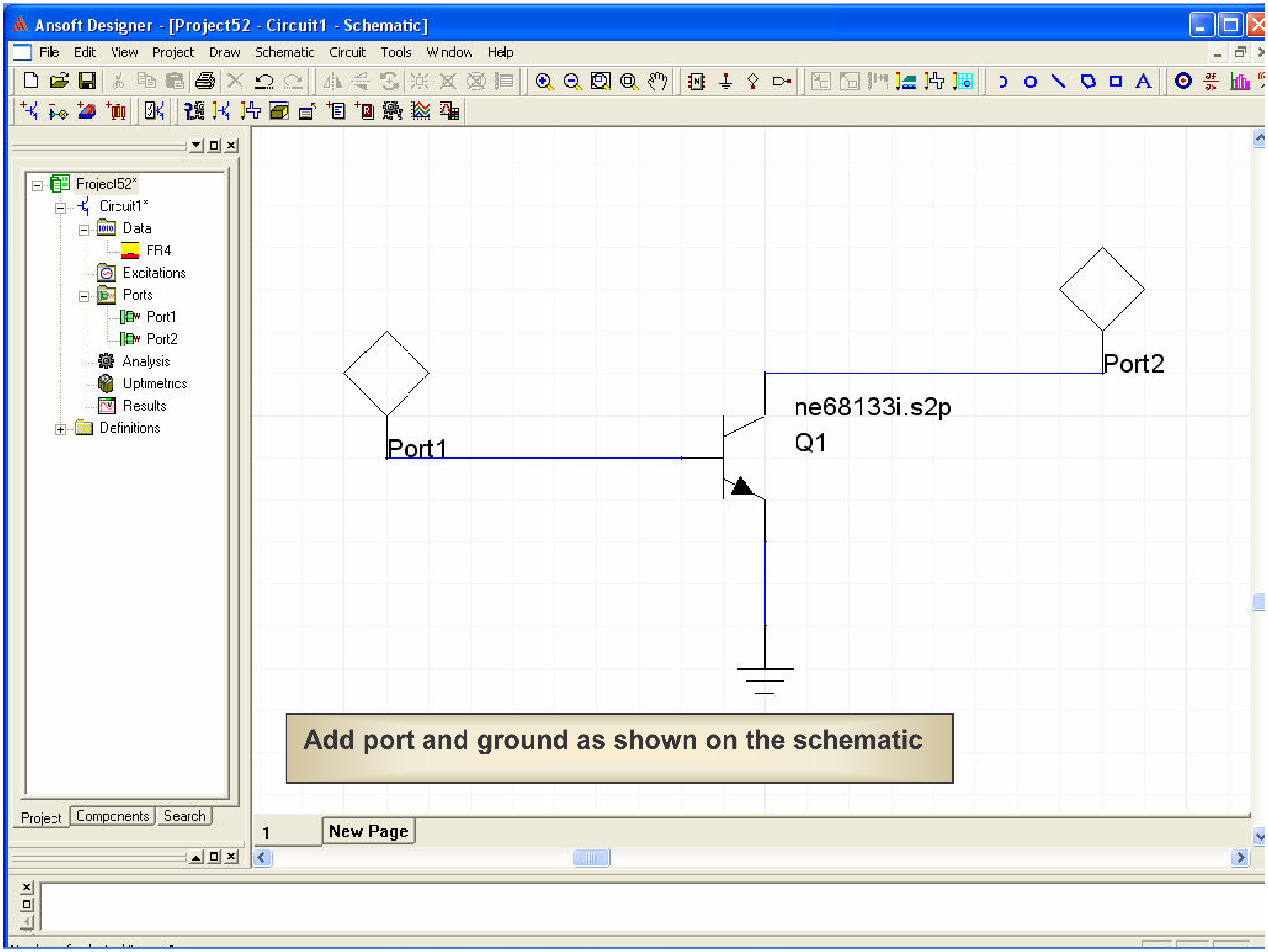


Choice the Transistor

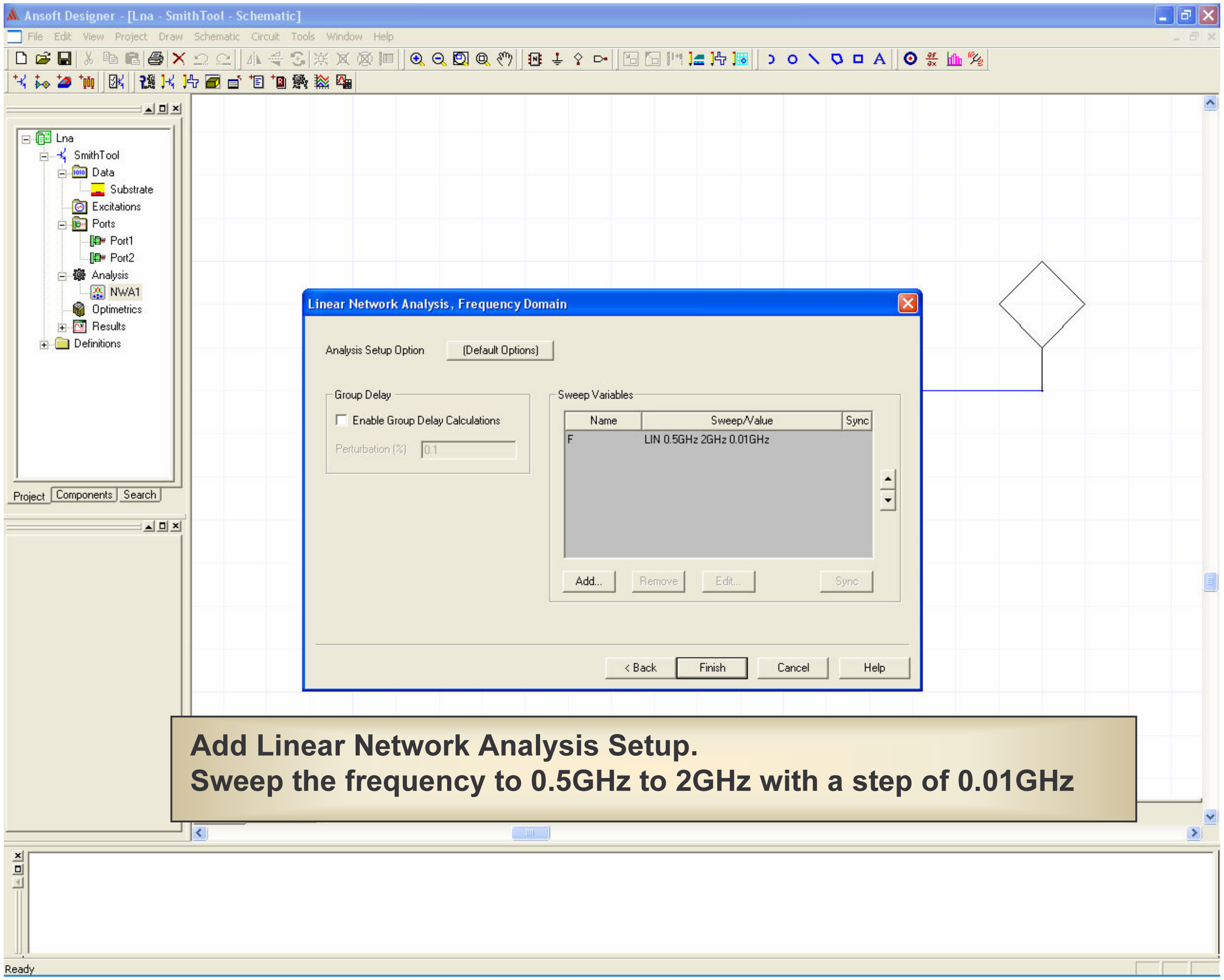


Double click on the symbol transistor to edit properties,
Click on the Choose Model button and select ne68133i.s2p

Add Port and Ground

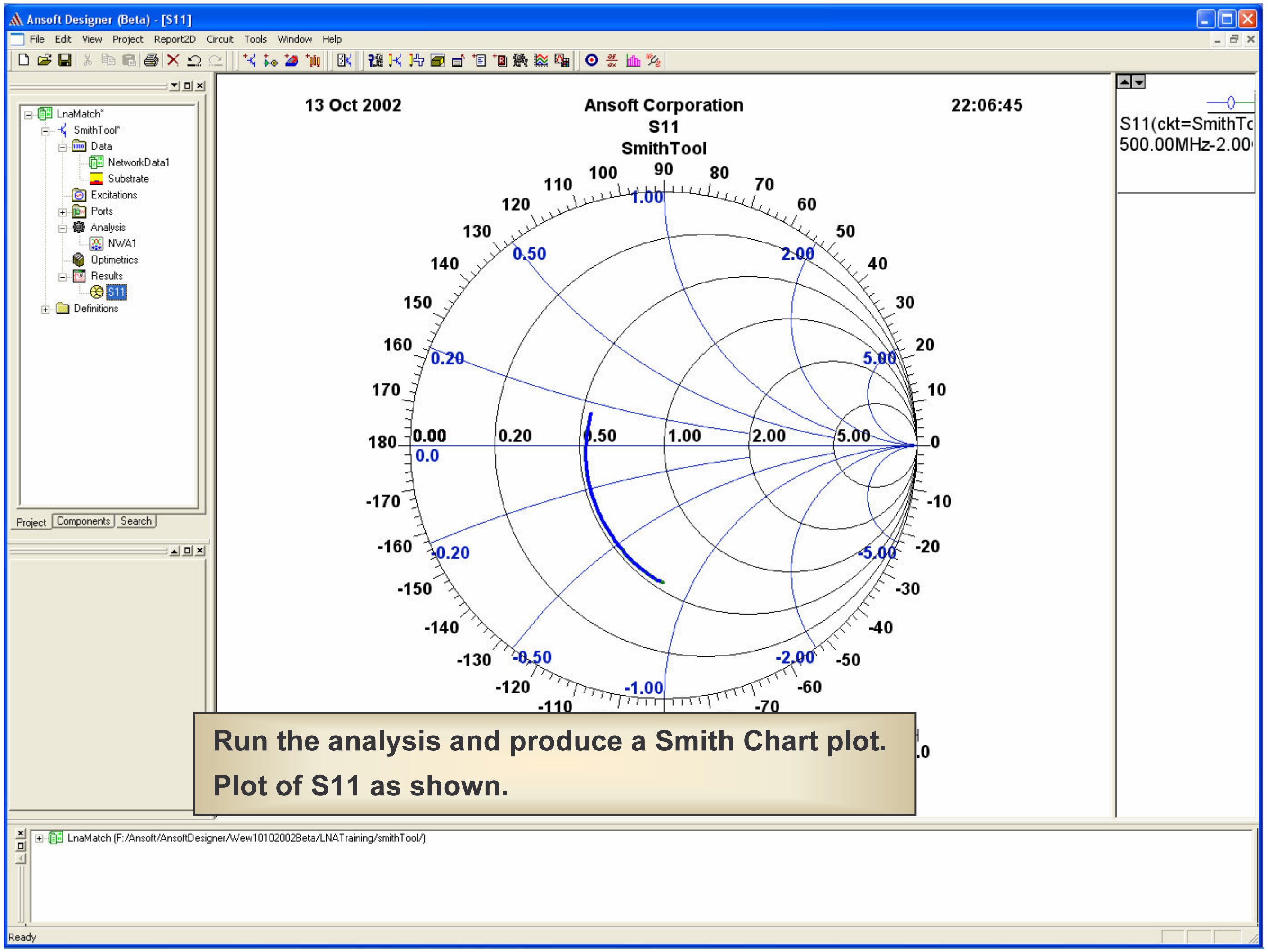


Add Analysis setup



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

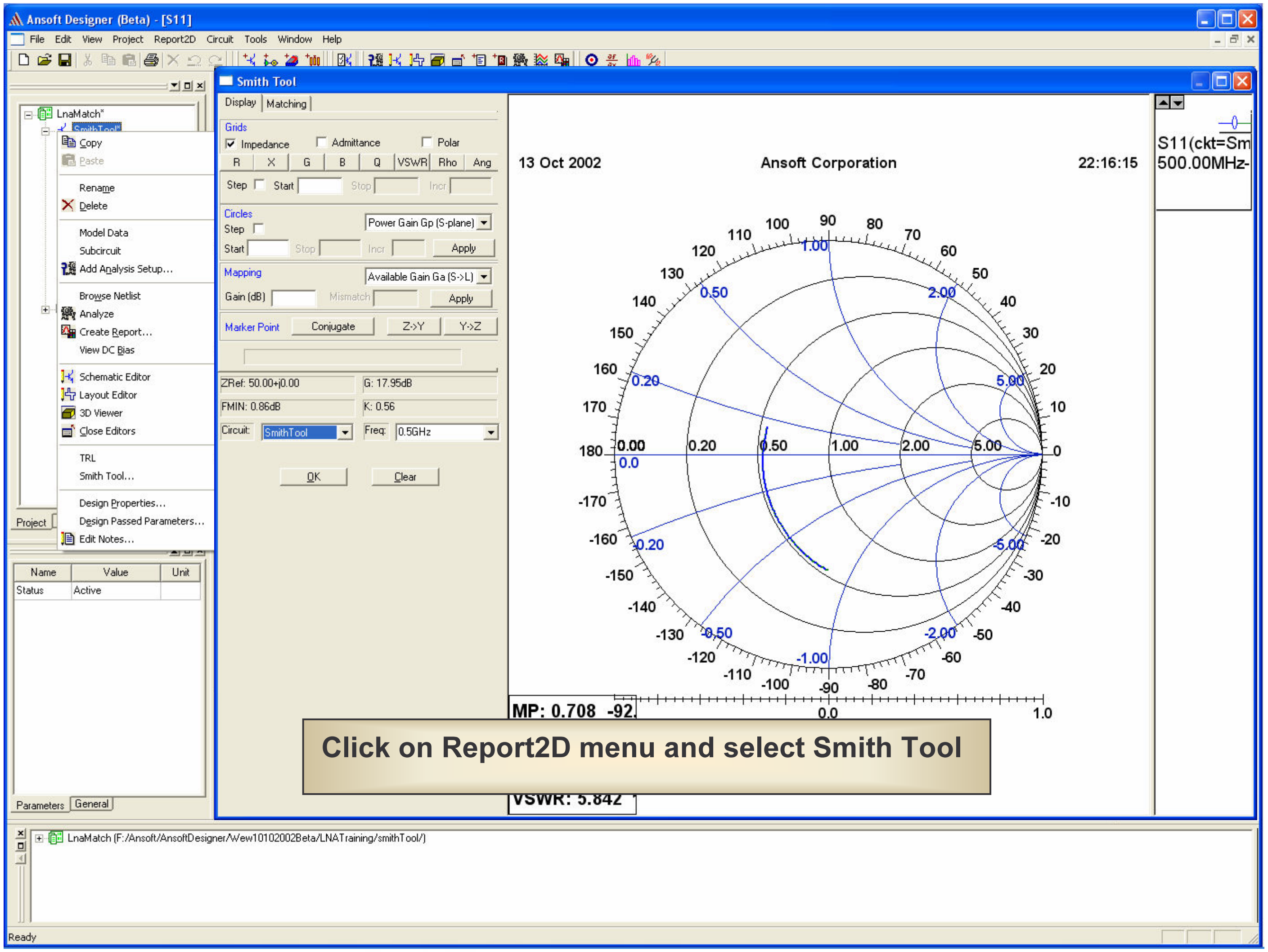
SmithTool - Polar Plot



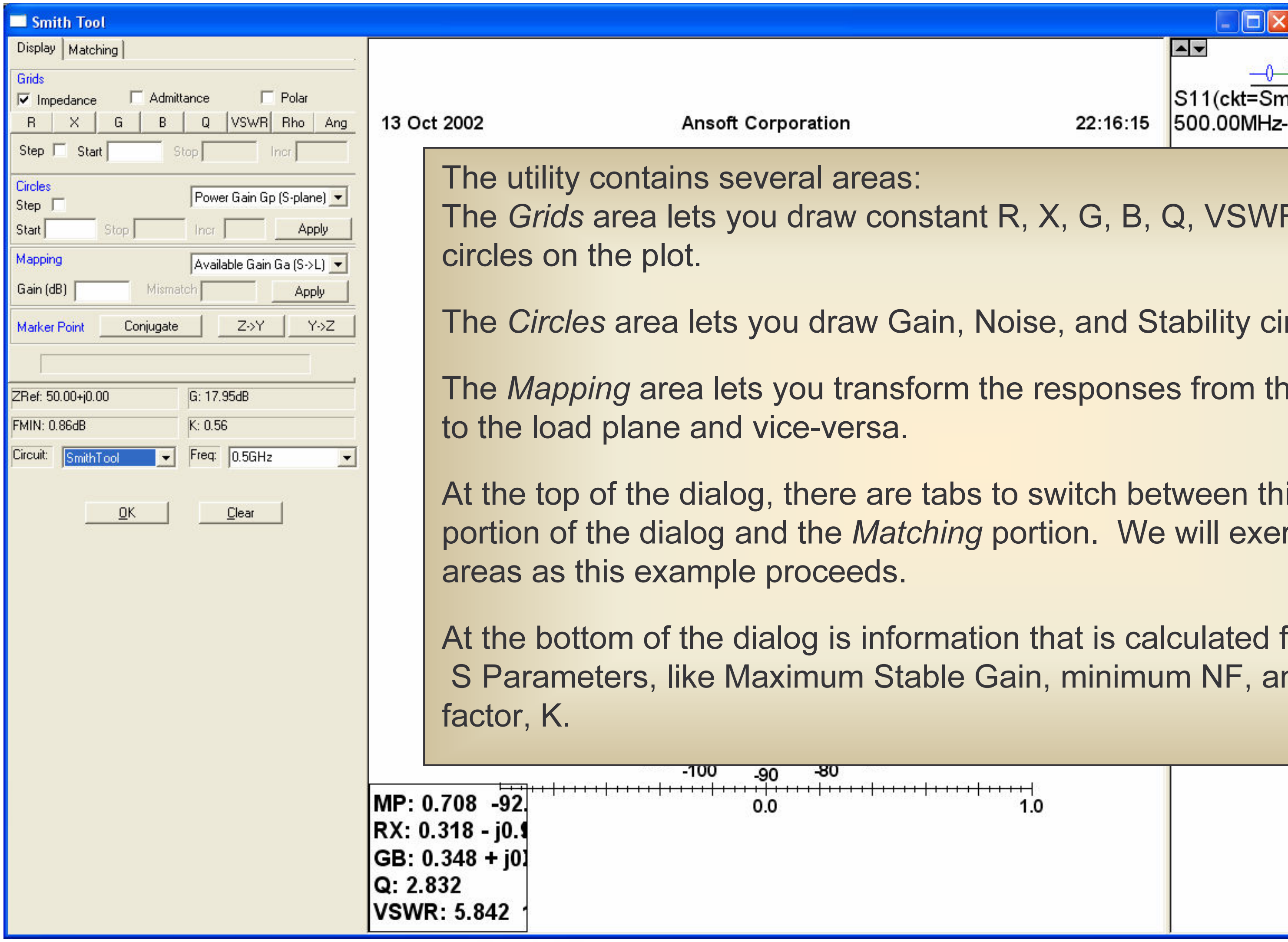
Run the analysis and produce a Smith Chart plot.

Plot of S11 as shown.

SmithTool - Opening the Utility



SmithTool - Opening the Utility



The utility contains several areas:
The *Grids* area lets you draw constant R, X, G, B, Q, VSWR, and Rho circles on the plot.

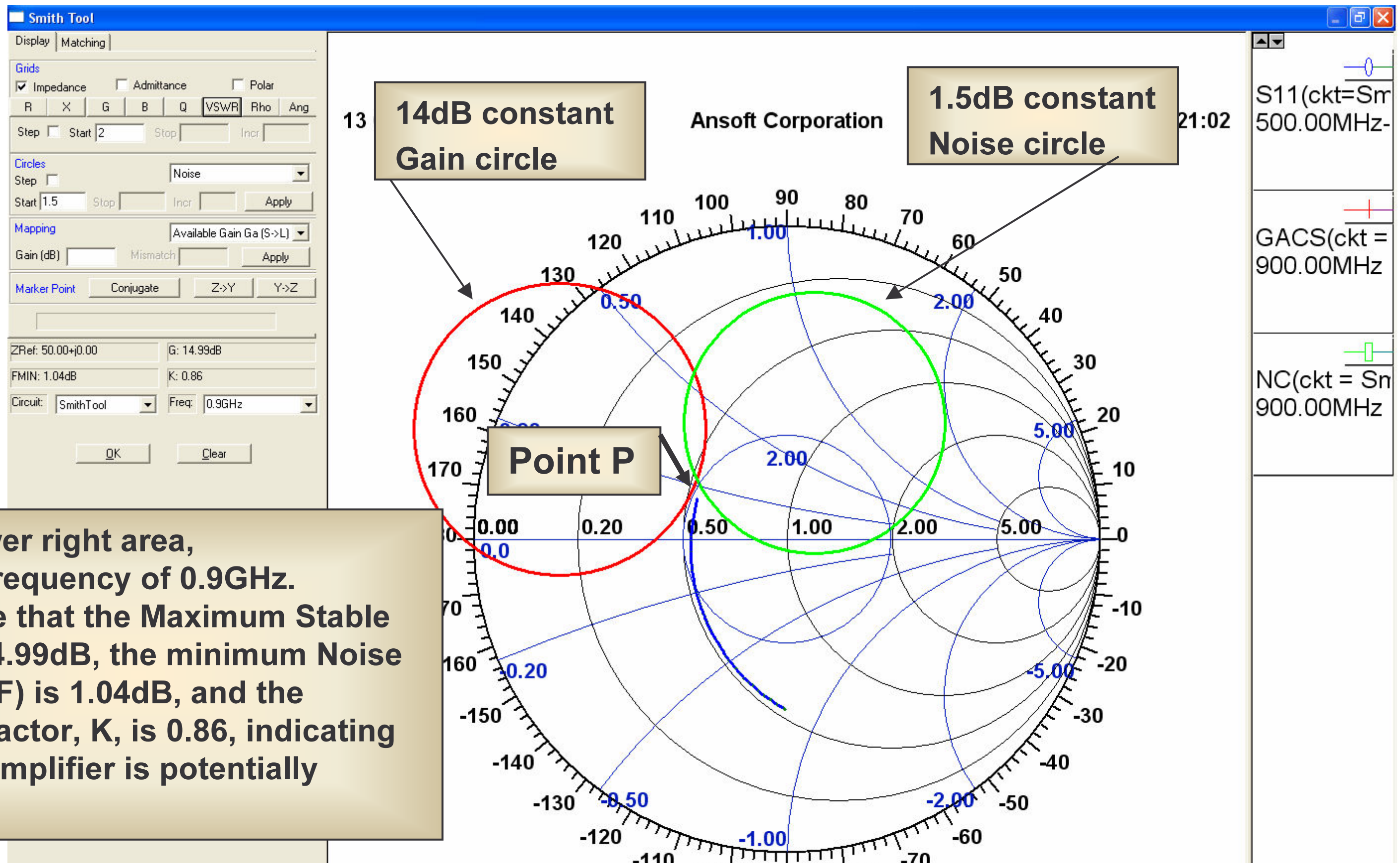
The *Circles* area lets you draw Gain, Noise, and Stability circles.

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.

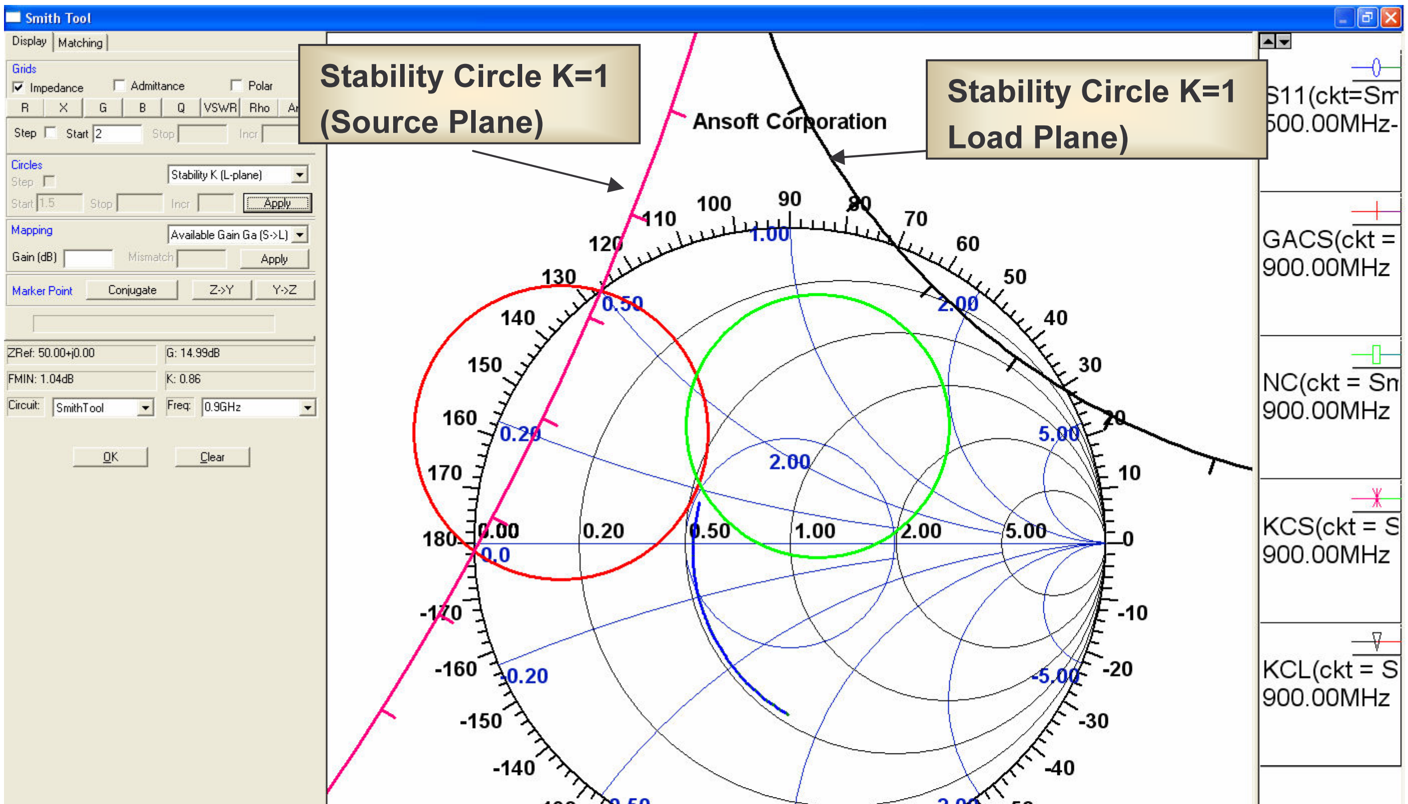
SmithTool - Maximizing Performance



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*.

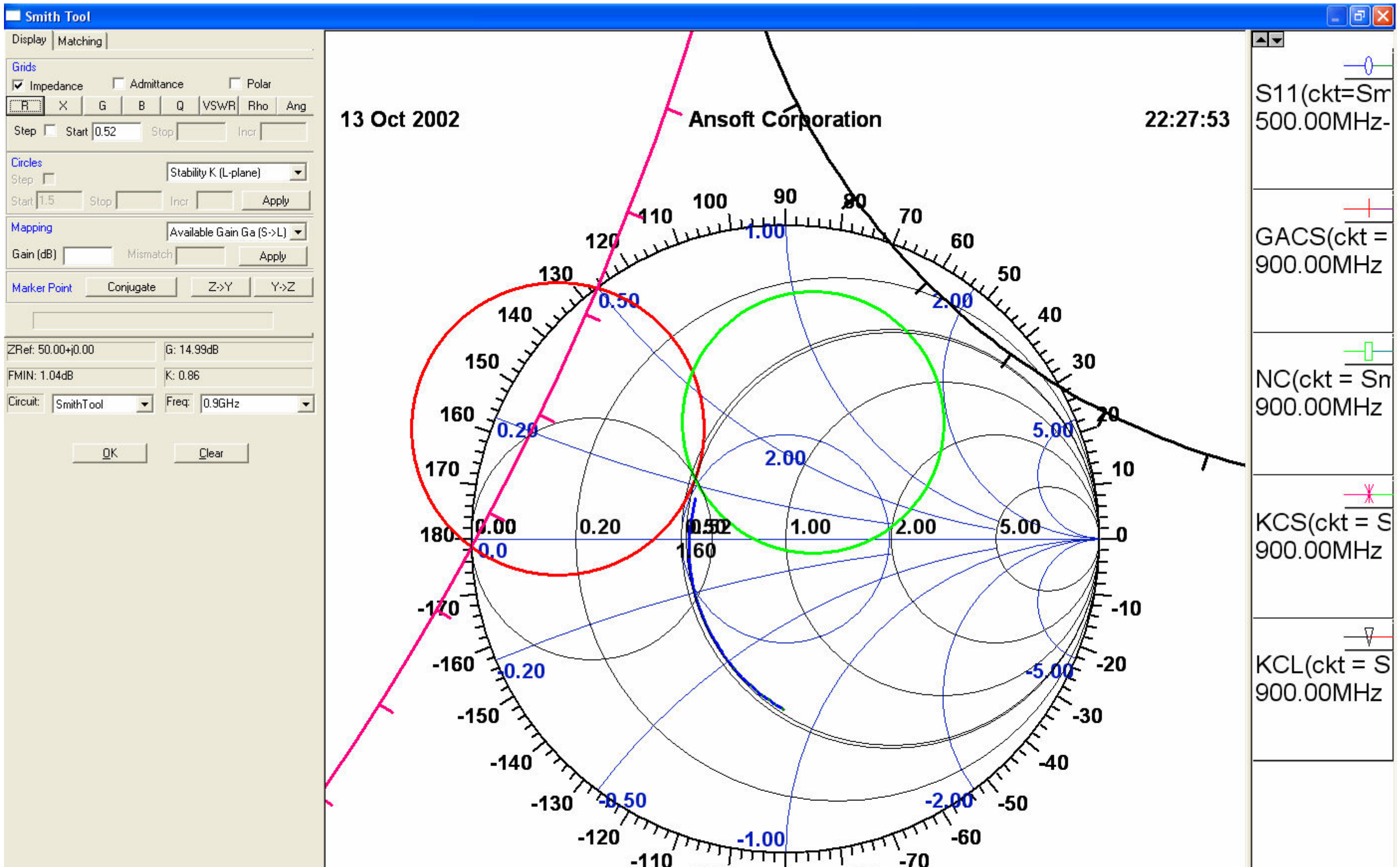
SmithTool - Checking Stability



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.

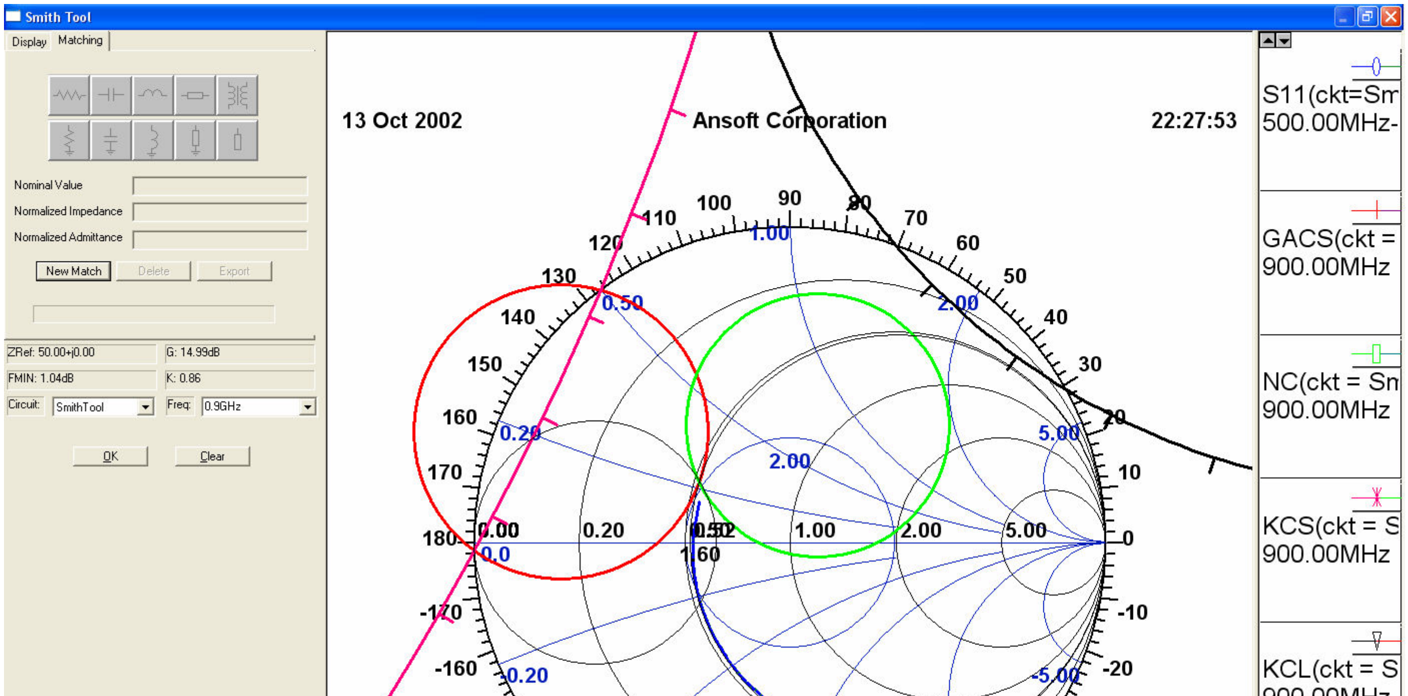
In the *Circles* area, select *Stability K (L Plane)* in the drop down list and click *Apply*. The load plane circle for K=1 appears, with small lines, or spokes, indicating which side of the circle is stable.

SmithTool - Drawing Aids



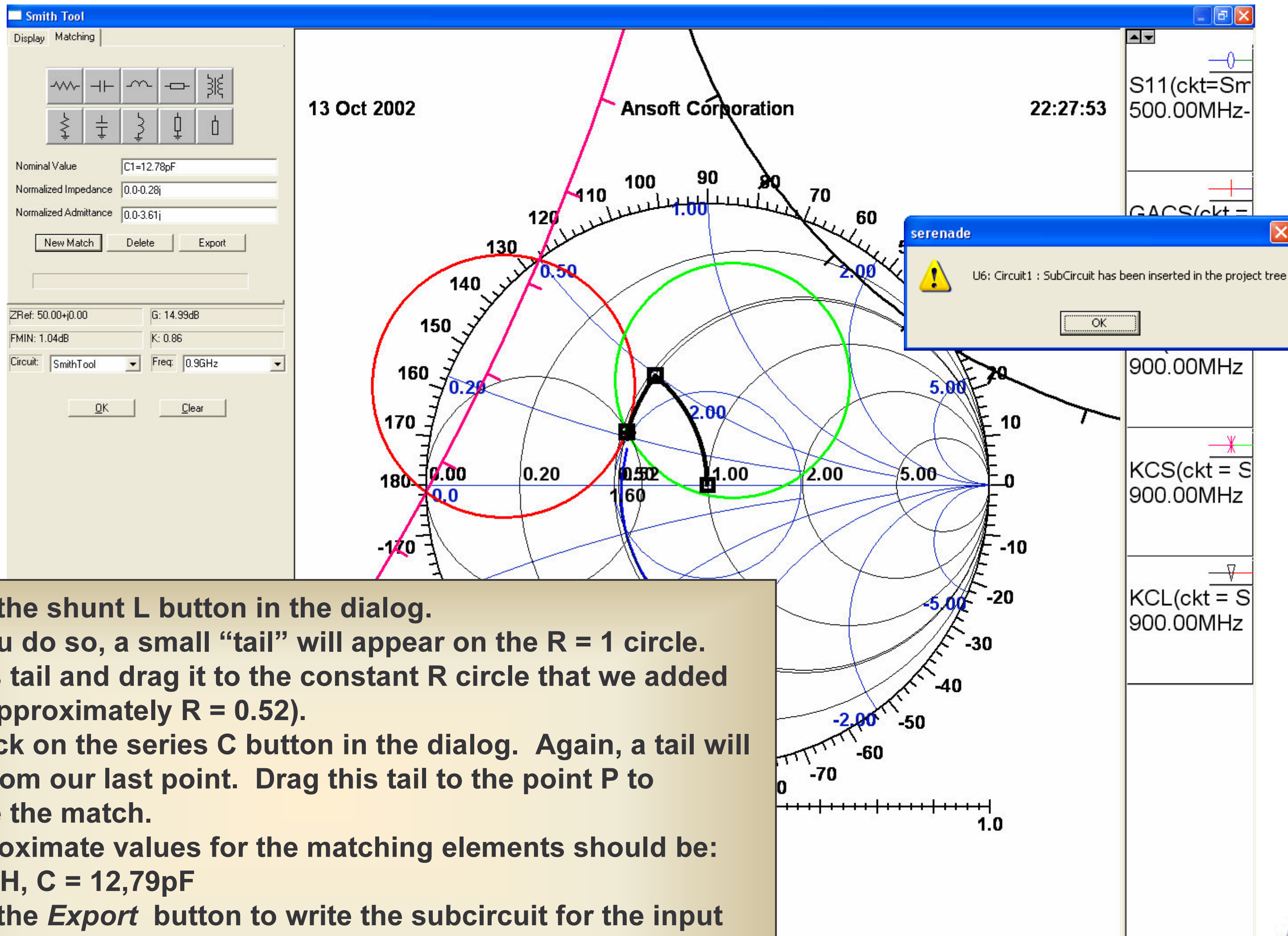
In the *Grids* area, click on *G*. The cursor jumps to the plot. Click on point P to draw a constant G circle through it. This circle should be approximately G = 1.60.
In the *Grids* area, click on *R*. The cursor jumps to the plot. Click on point P to draw a constant R circle through it. This circle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

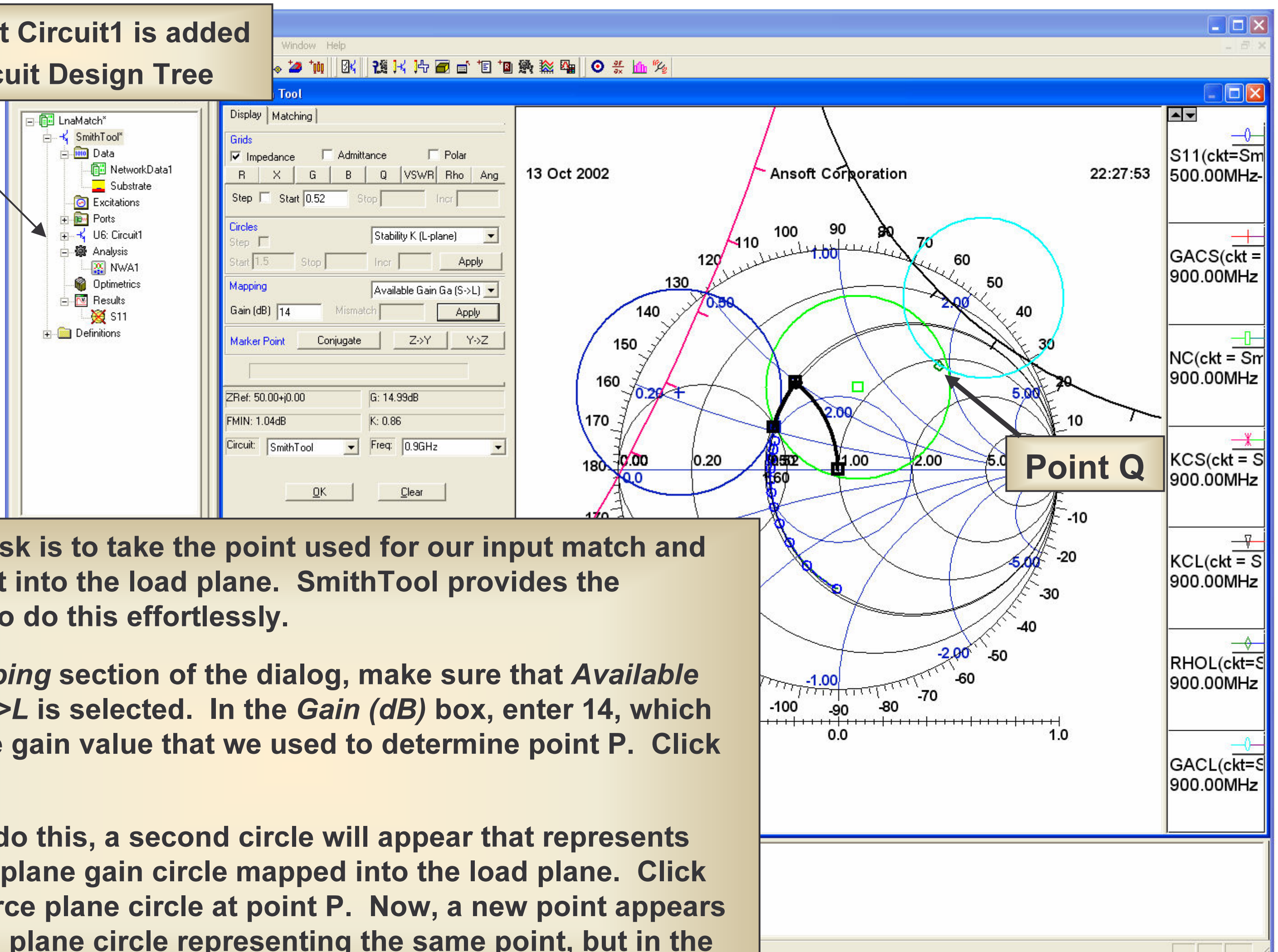
SmithTool - Input Matching Circuit



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 = 9.19nH, C = 12,79pF
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

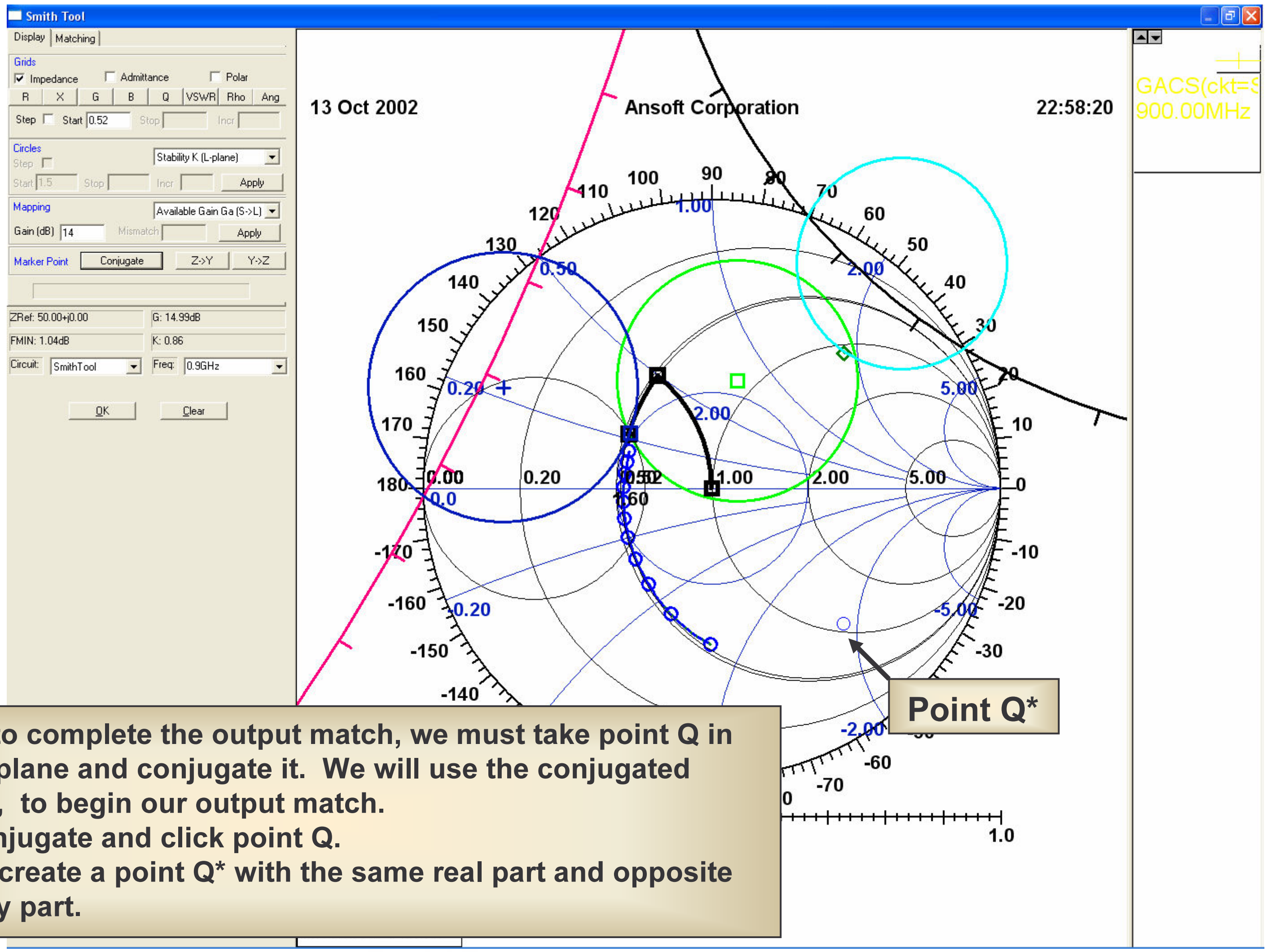


Our next task is to take the point used for our input match and transform it into the load plane. SmithTool provides the capability to do this effortlessly.

In the *Mapping* section of the dialog, make sure that *Available Gain Ga S->L* is selected. In the *Gain (dB)* box, enter 14, which is the same gain value that we used to determine point P. Click on *Apply*.

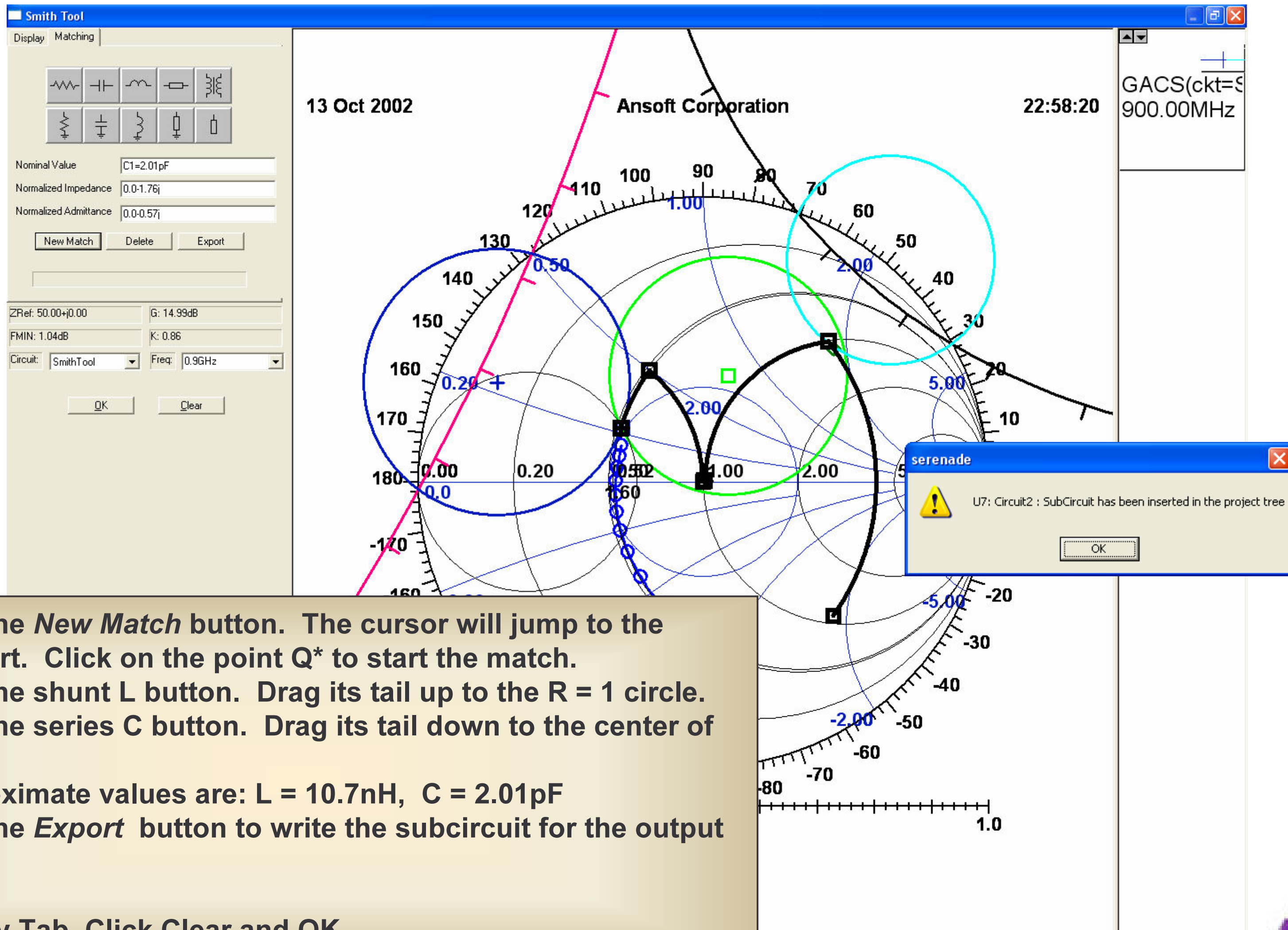
When you do this, a second circle will appear that represents the source plane gain circle mapped into the load plane. Click on the source plane circle at point P. Now, a new point appears on the load plane circle representing the same point, but in the load plane, as shown above. We will call this point Q.

SmithTool - Complex Conjugation



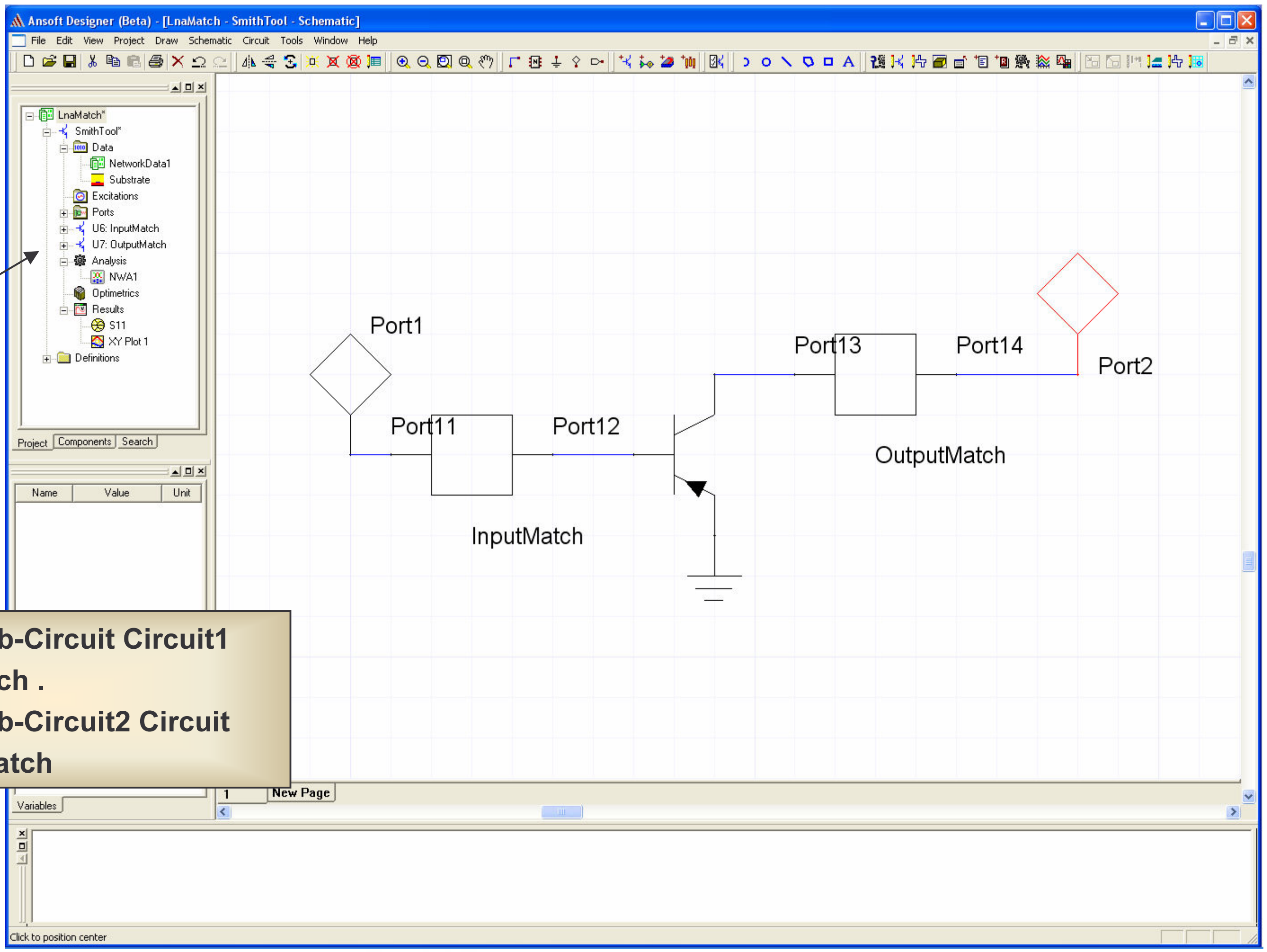
In order to complete the output match, we must take point Q in the load plane and conjugate it. We will use the conjugated point, Q*, to begin our output match.
Click Conjugate and click point Q.
This will create a point Q* with the same real part and opposite imaginary part.

SmithTool - Output Matching Circuit



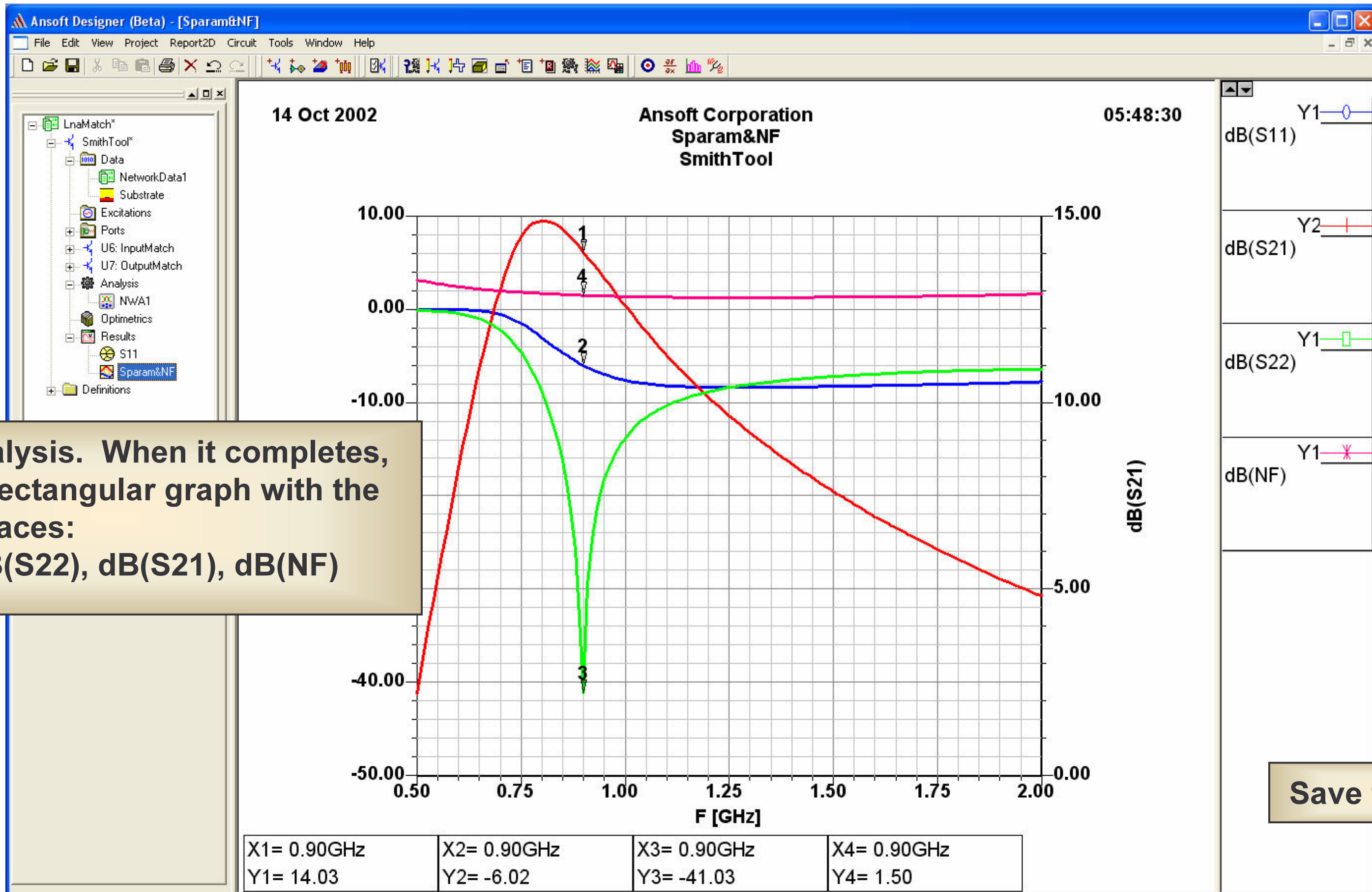
Click on the *New Match* button. The cursor will jump to the Smith chart. Click on the point Q* to start the match.
Click on the shunt L button. Drag its tail up to the R = 1 circle.
Click on the series C button. Drag its tail down to the center of the chart.
The approximate values are: L = 10.7nH, C = 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)

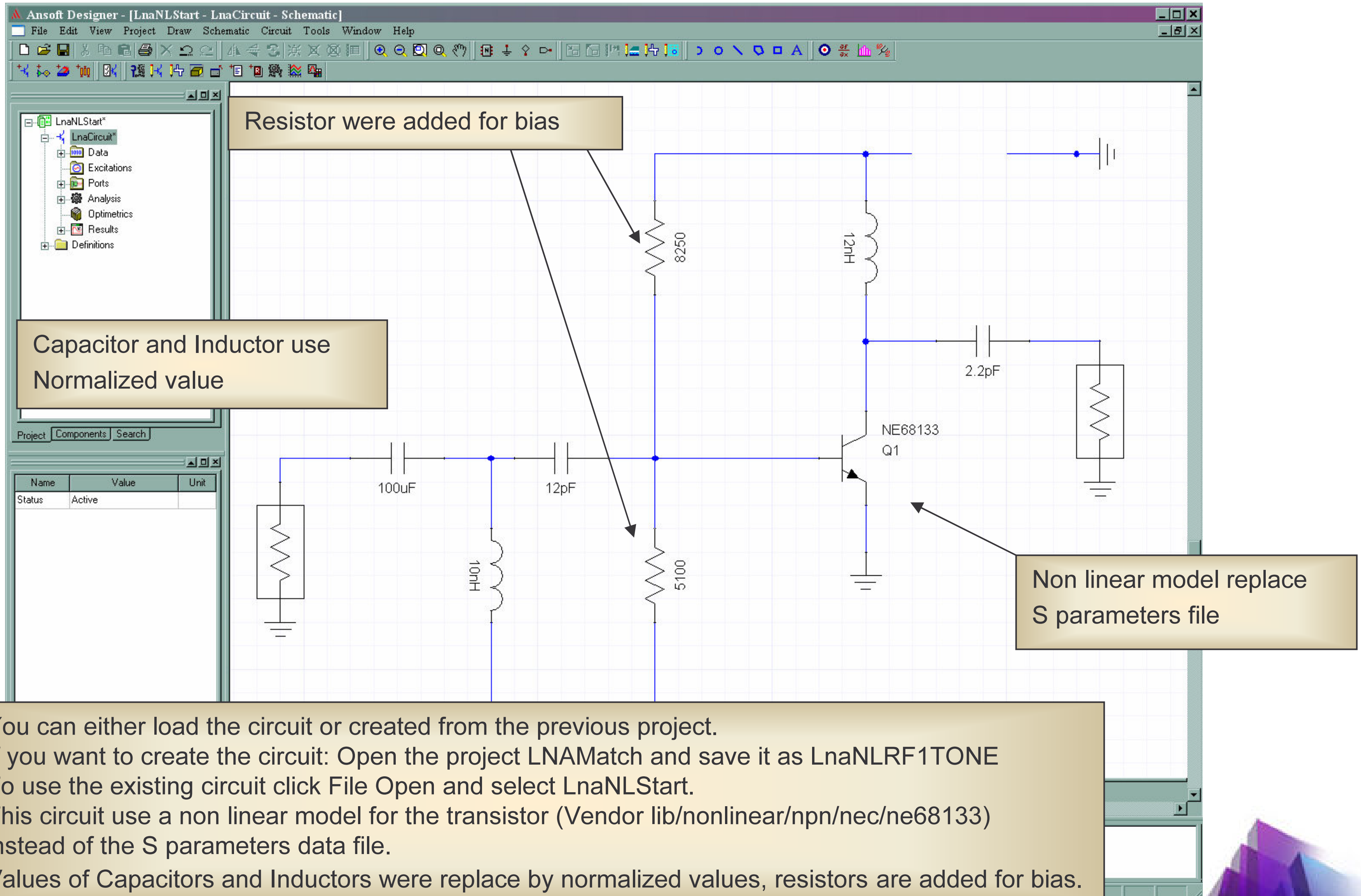
Save the Project

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

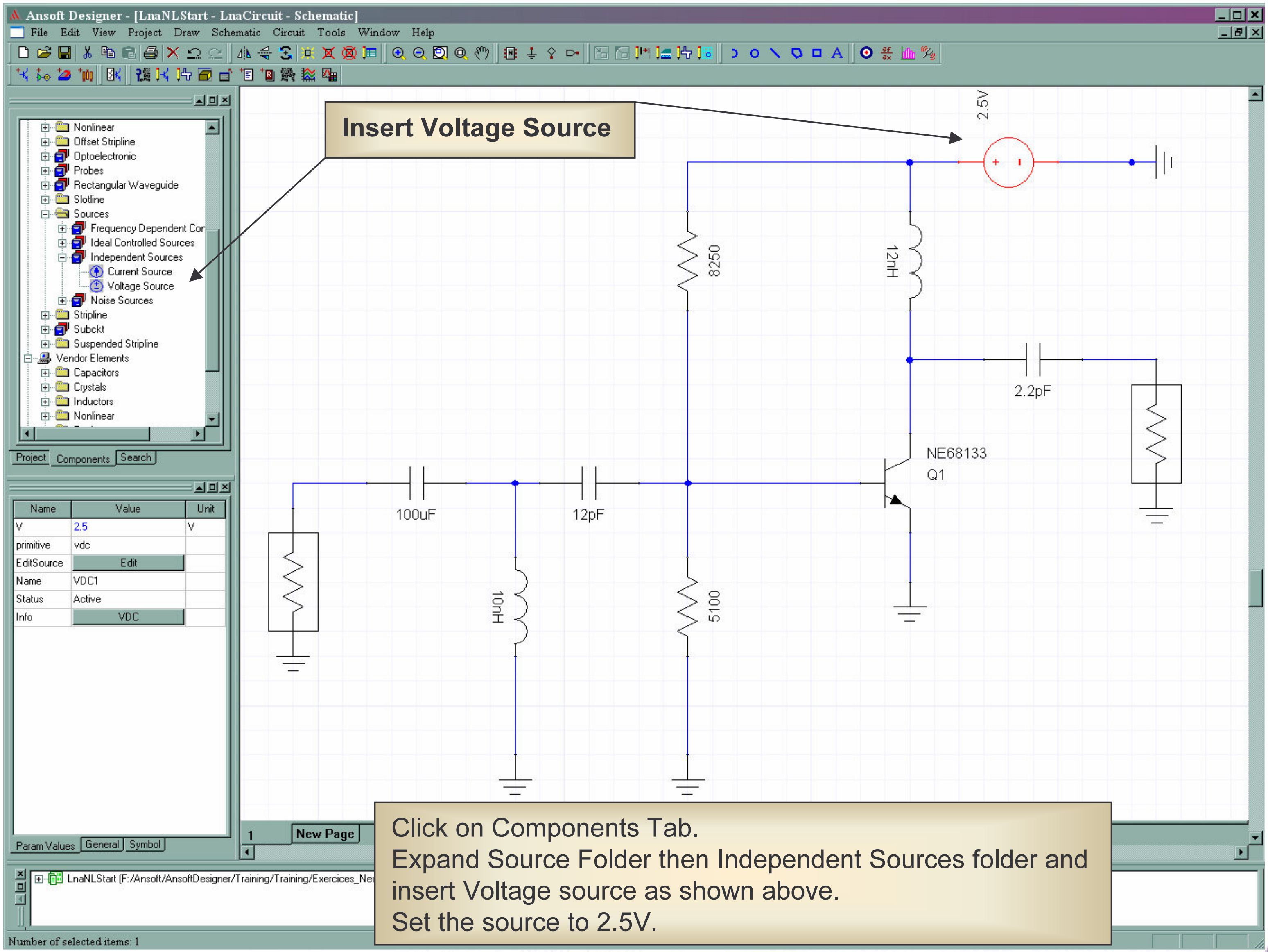
Exercise:
LNA Design
Non-Linear Analysis

Load LnaNLStart



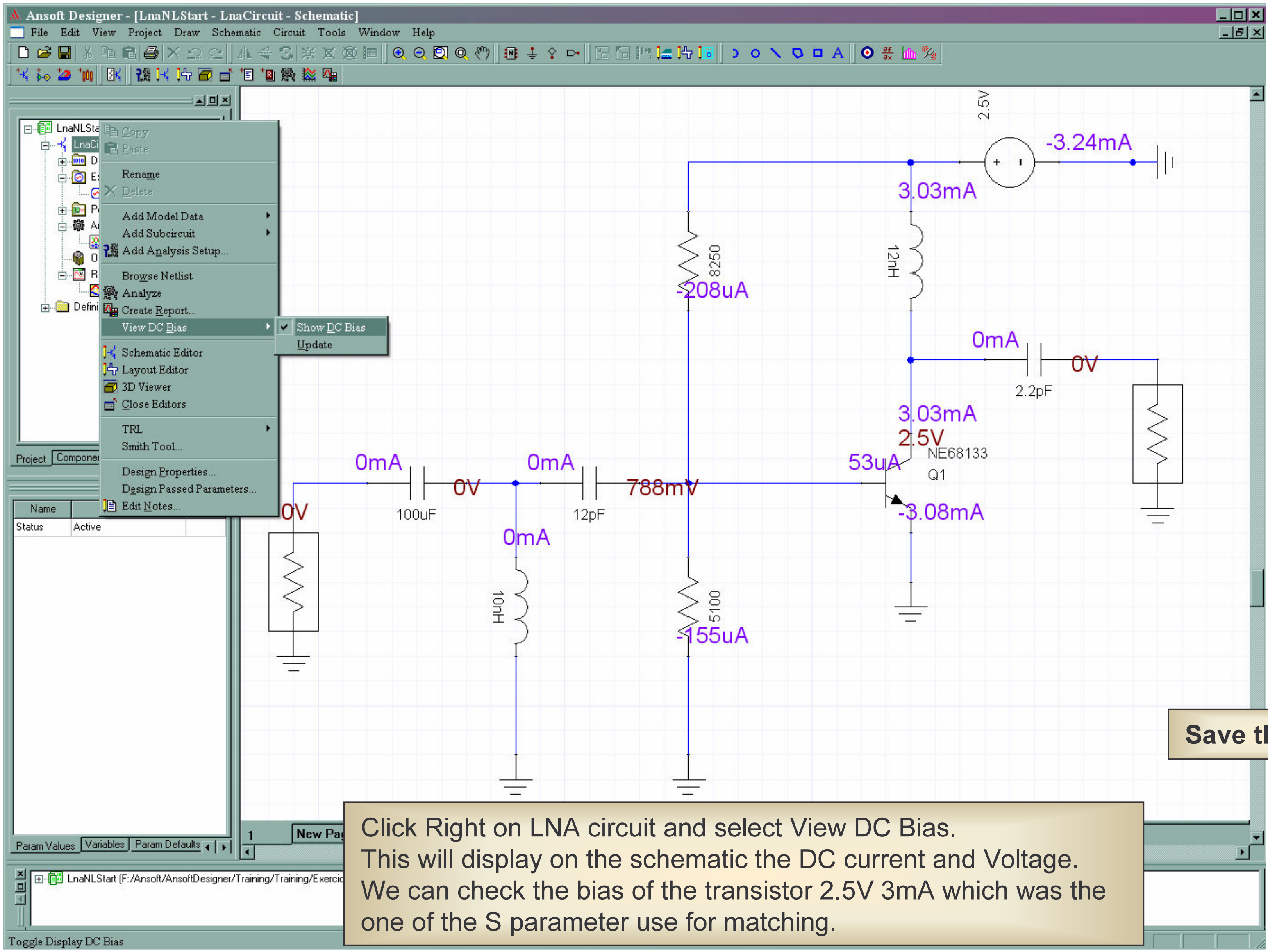
You can either load the circuit or created from the previous project.
If you want to create the circuit: Open the project LNAMatch and save it as LnaNLRF1TONE
To use the existing circuit click File Open and select LnaNLStart.
This circuit use a non linear model for the transistor (Vendor lib/nonlinear/npn/nec/ne68133) 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



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

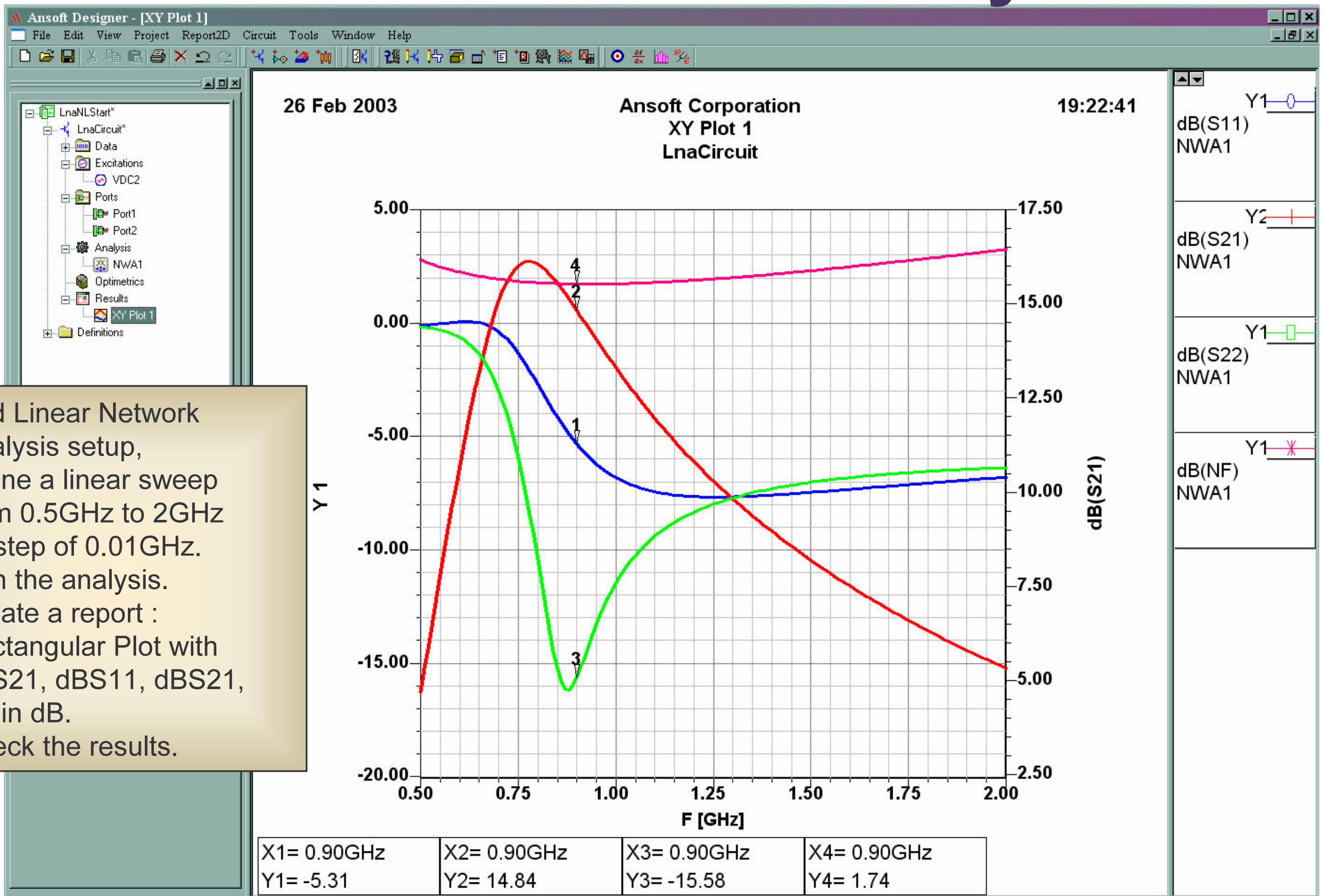
View DC Bias



Save the Project

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.

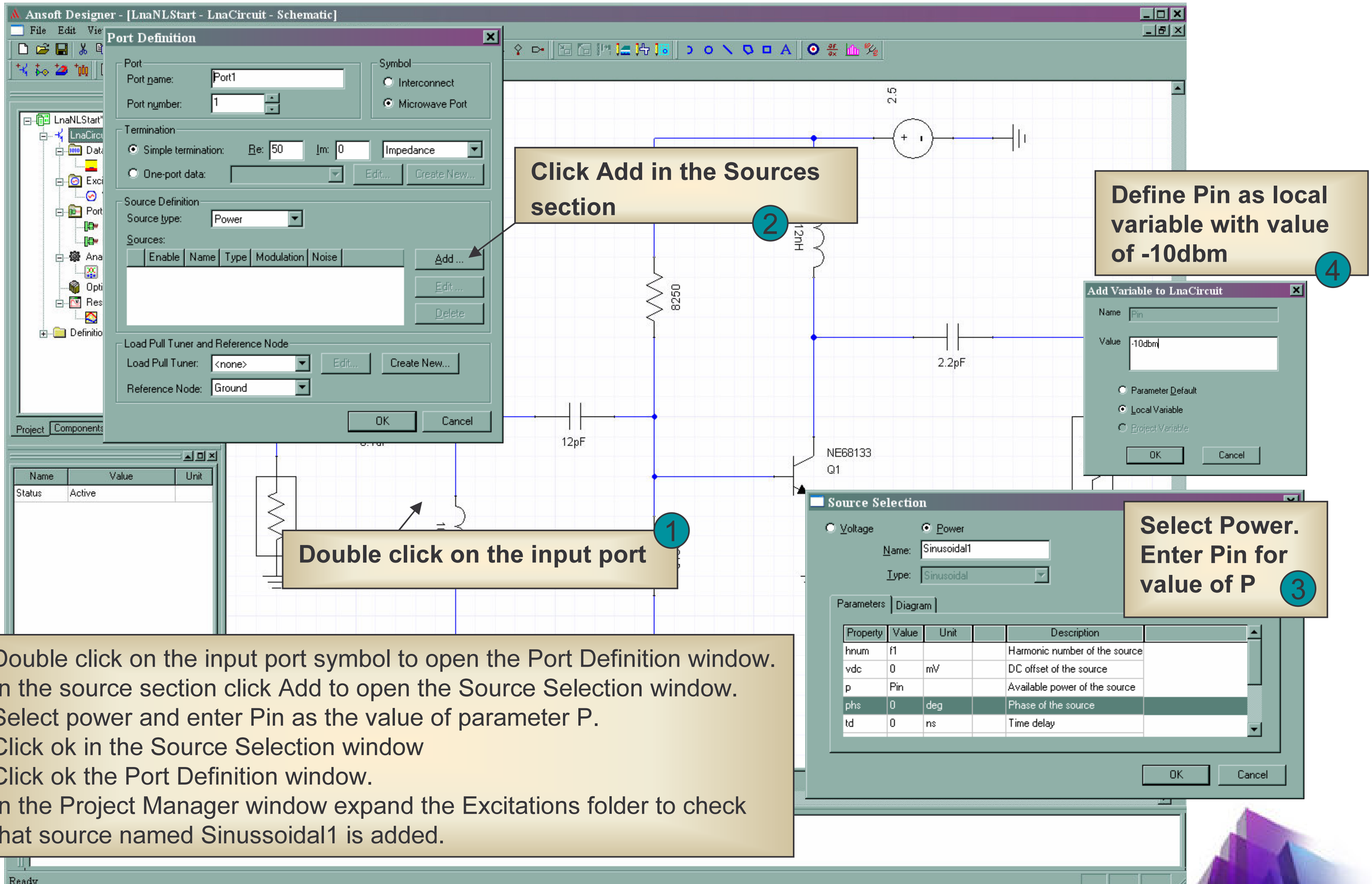
Run Linear Analysis



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 dBS21, dBS11, dBS21, NF in dB.
Check the results.

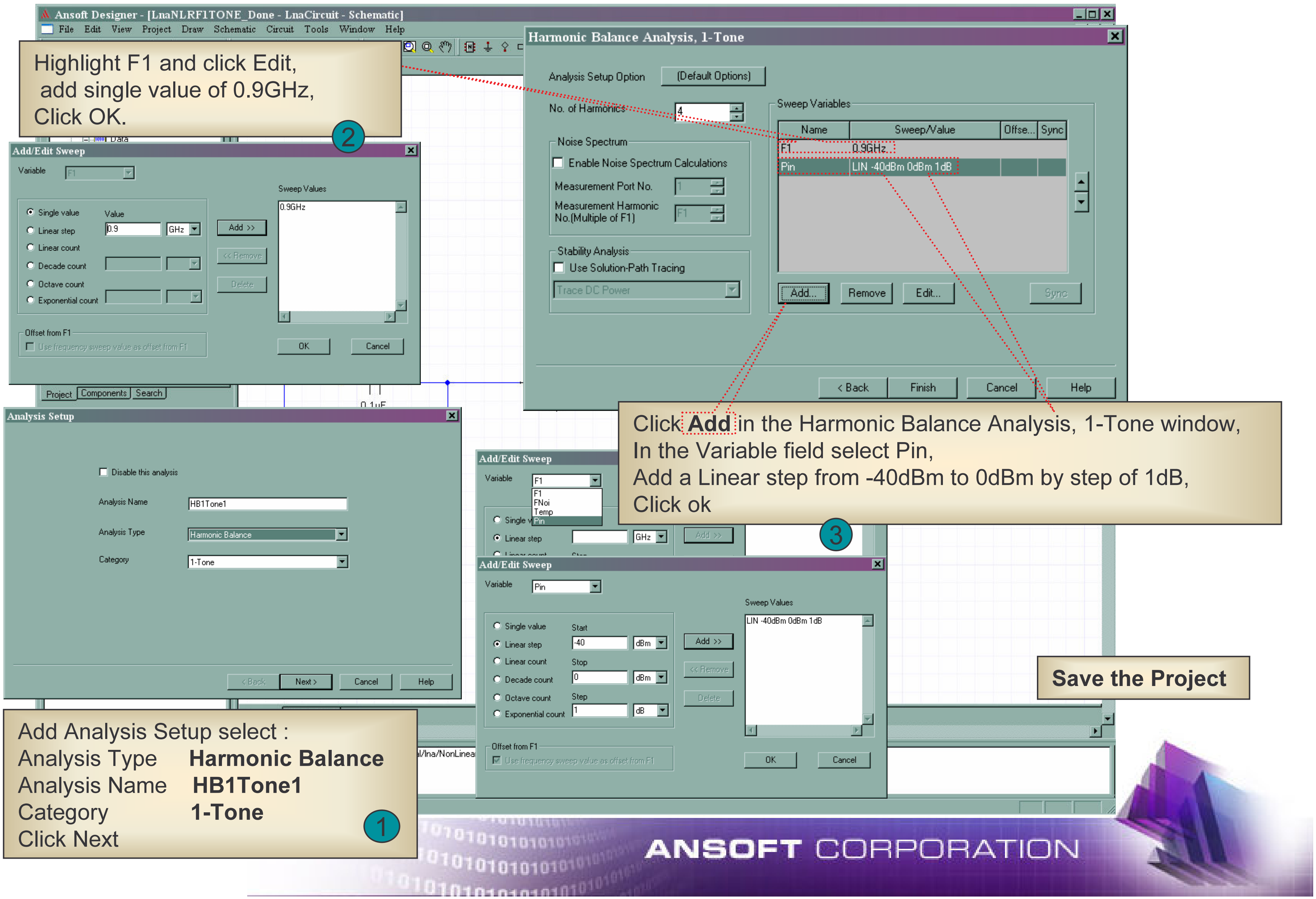
Even if we use a non linear model it is possible to run linear analysis, non linear model is linearised at the bias condition and the simulation will use the corresponding S parameters. This allow to check that the results with the non linear model are close to the one get with S parameter data file.

Define RF 1 Tone Analysis



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.

2

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

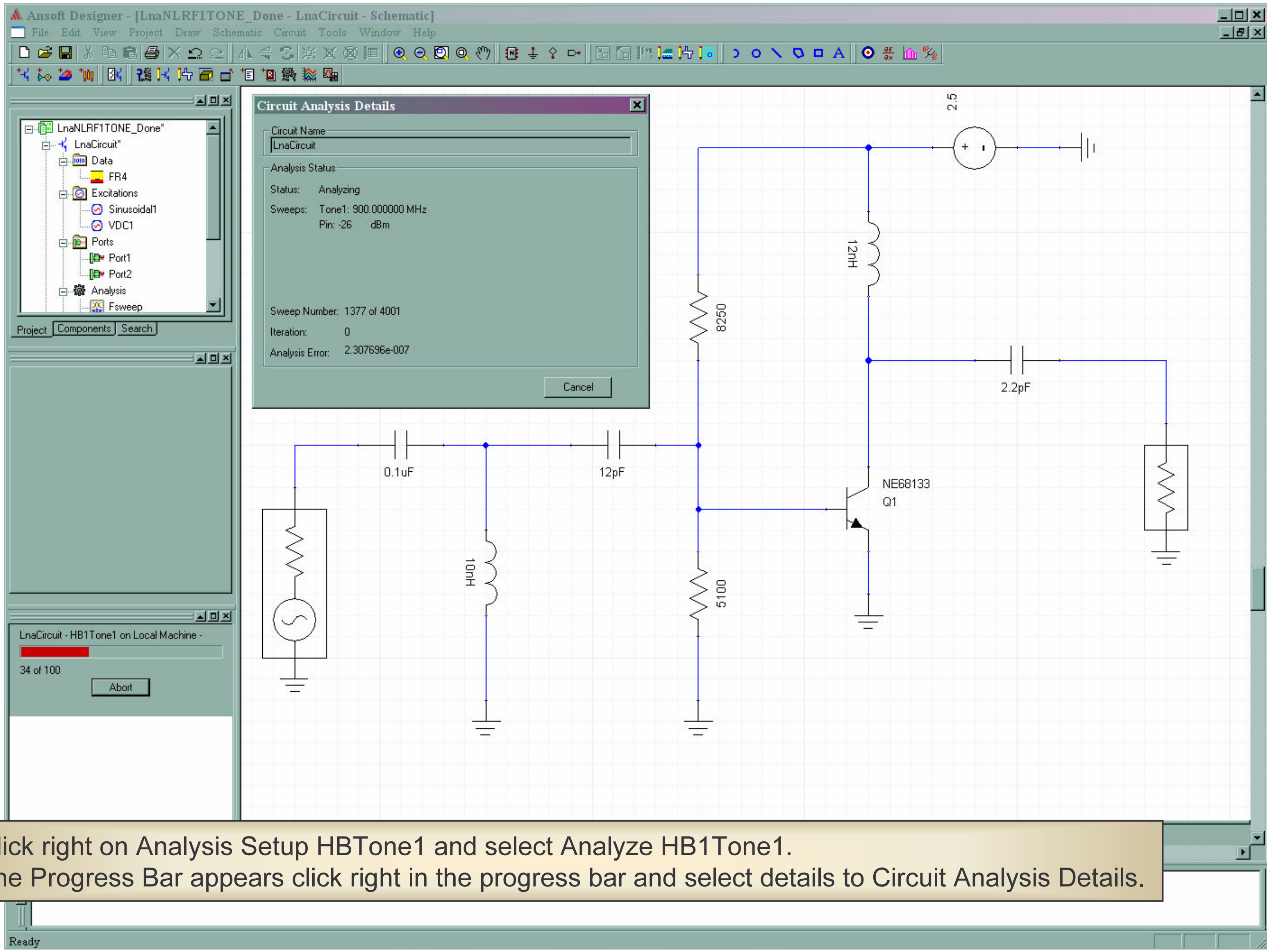
3

Save the Project

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

1

Run Harmonic Balance Analysis

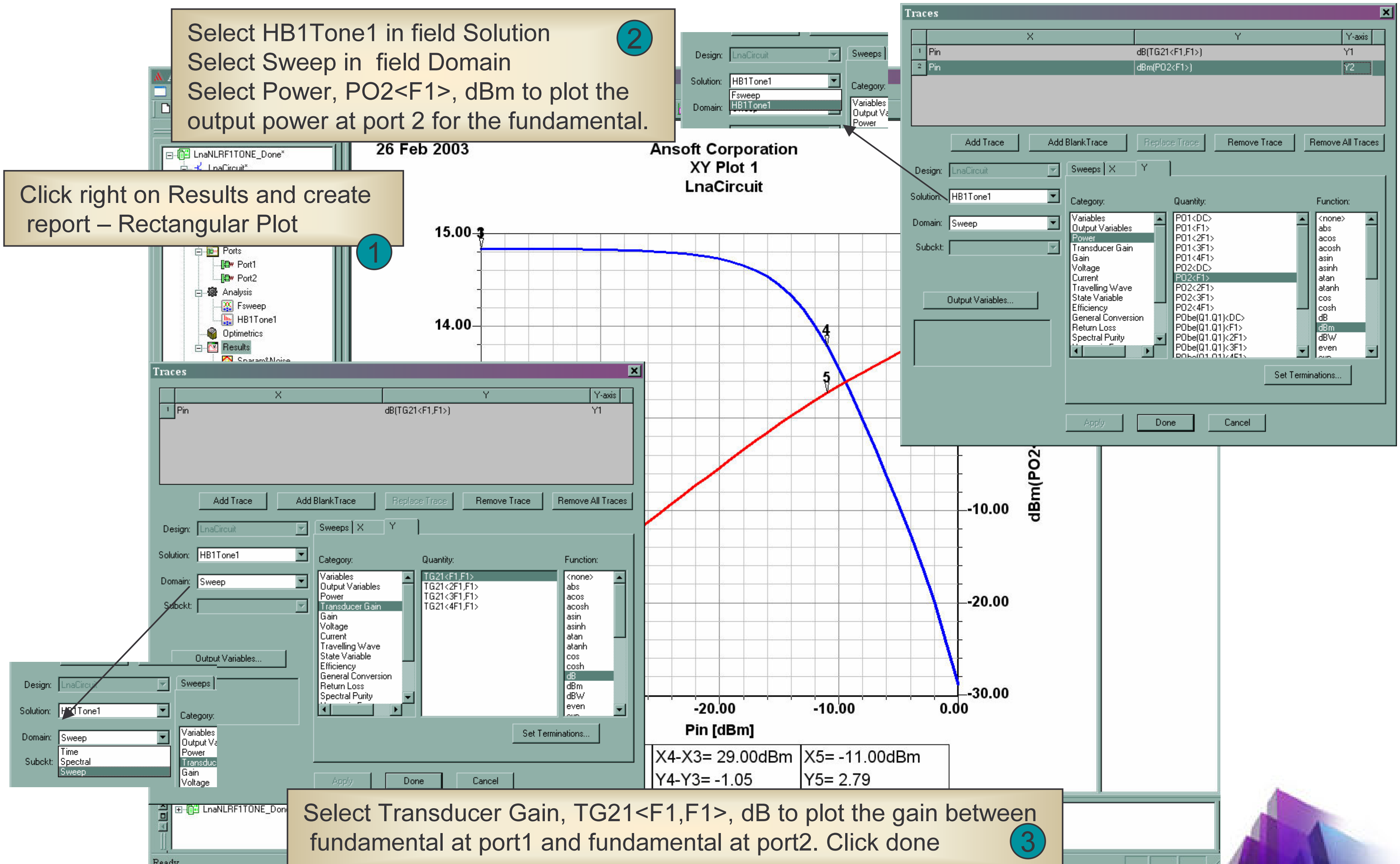


Click right on Analysis Setup HBTone1 and select Analyze HB1Tone1.
The Progress Bar appears click right in the progress bar and select details to Circuit Analysis Details.

Create Results: Pou/TG21 vs Pin

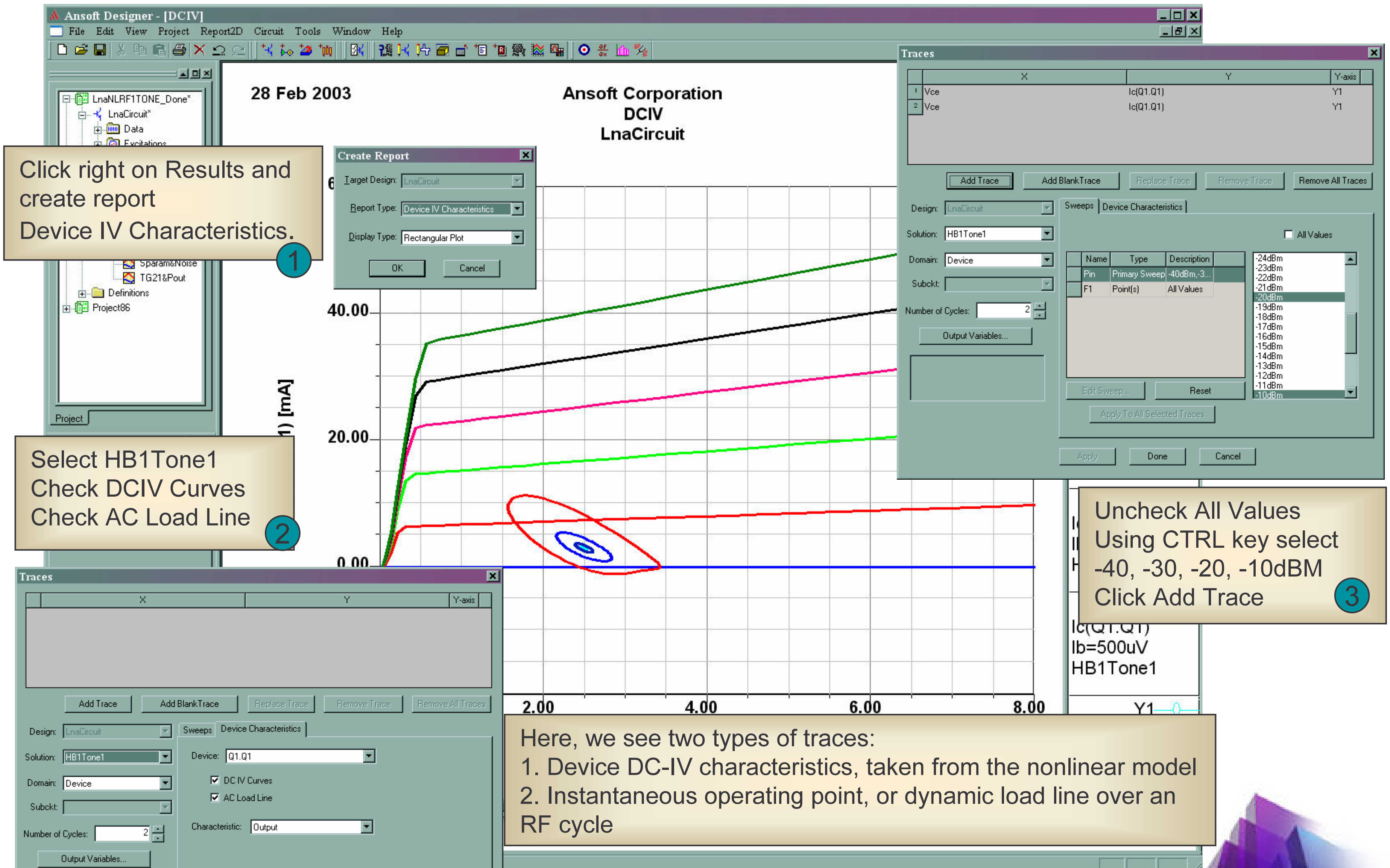
Select HB1Tone1 in field Solution
Select Sweep in field Domain
Select Power, PO2<F1>, dBm to plot the output power at port 2 for the fundamental. 2

Click right on Results and create report – Rectangular Plot 1



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



Click right on Results and create report Device IV Characteristics. 1

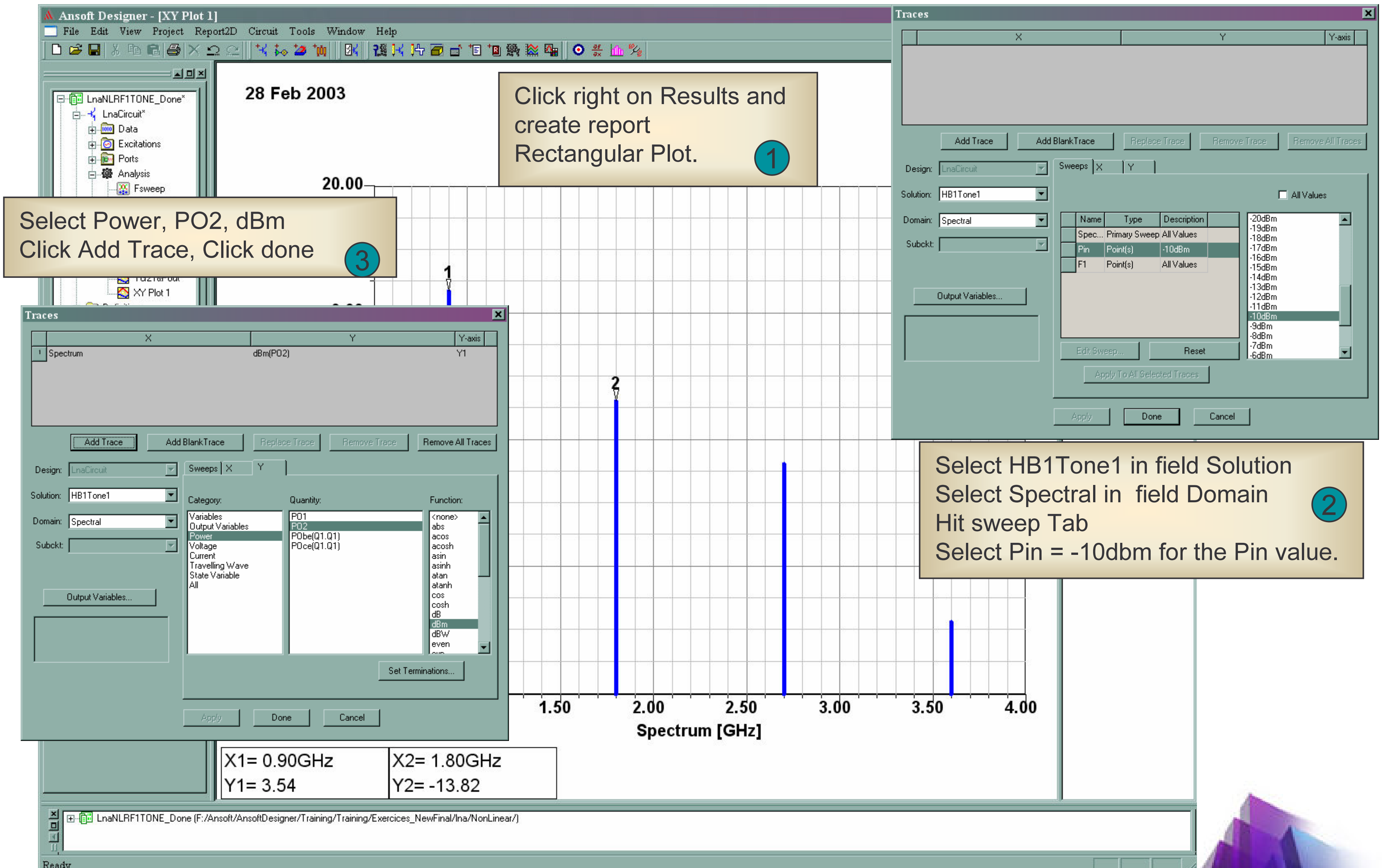
Select HB1Tone1
Check DCIV Curves
Check AC Load Line 2

Uncheck All Values
Using CTRL key select -40, -30, -20, -10dBM
Click Add Trace 3

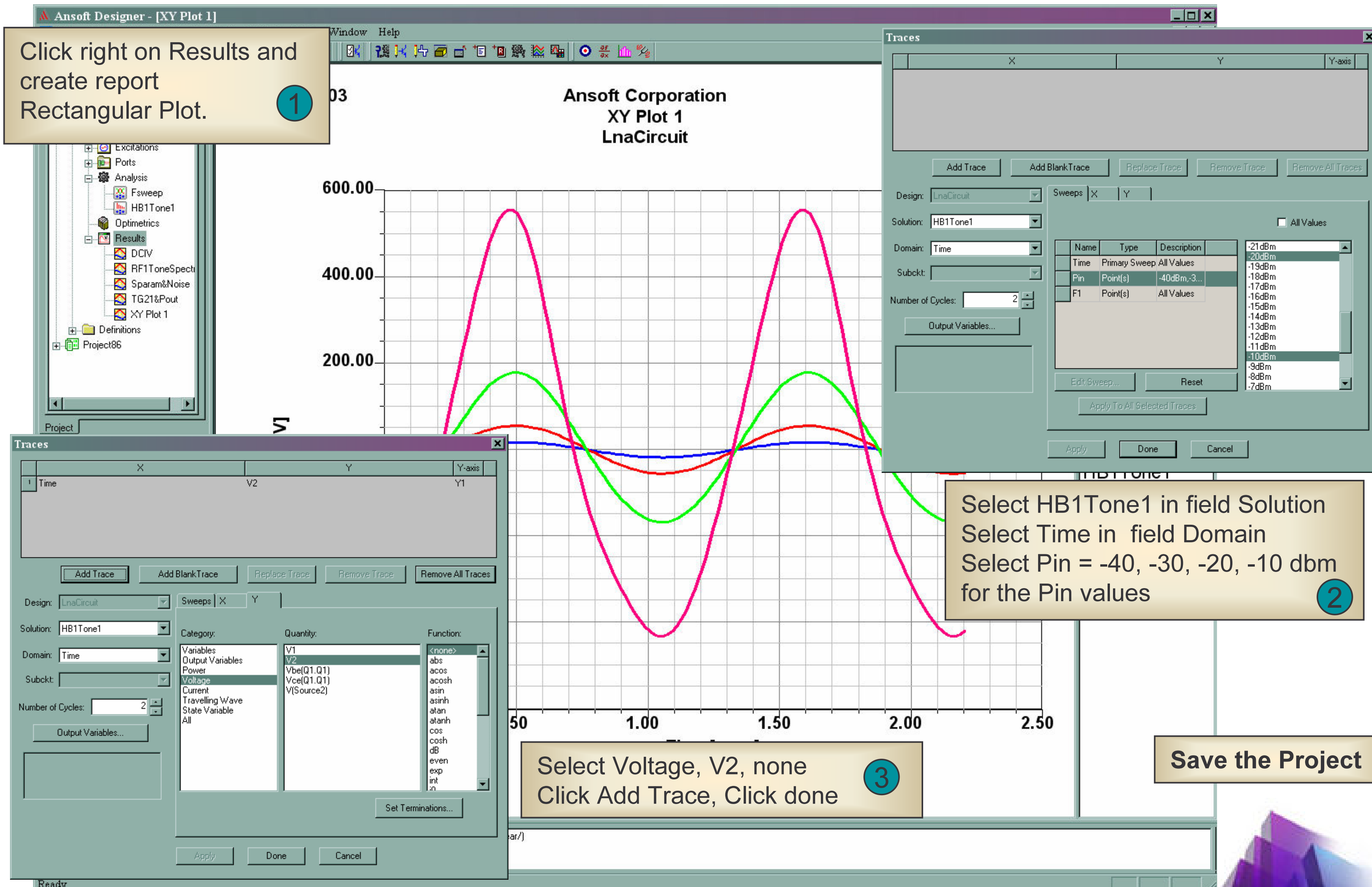
Here, we see two types of traces:
1. Device DC-IV characteristics, taken from the nonlinear model
2. Instantaneous operating point, or dynamic load line over an RF cycle

ANSOFT CORPORATION

Create Results: Spectrum



Create Results: Wave Form



Click right on Results and create report Rectangular Plot. 1

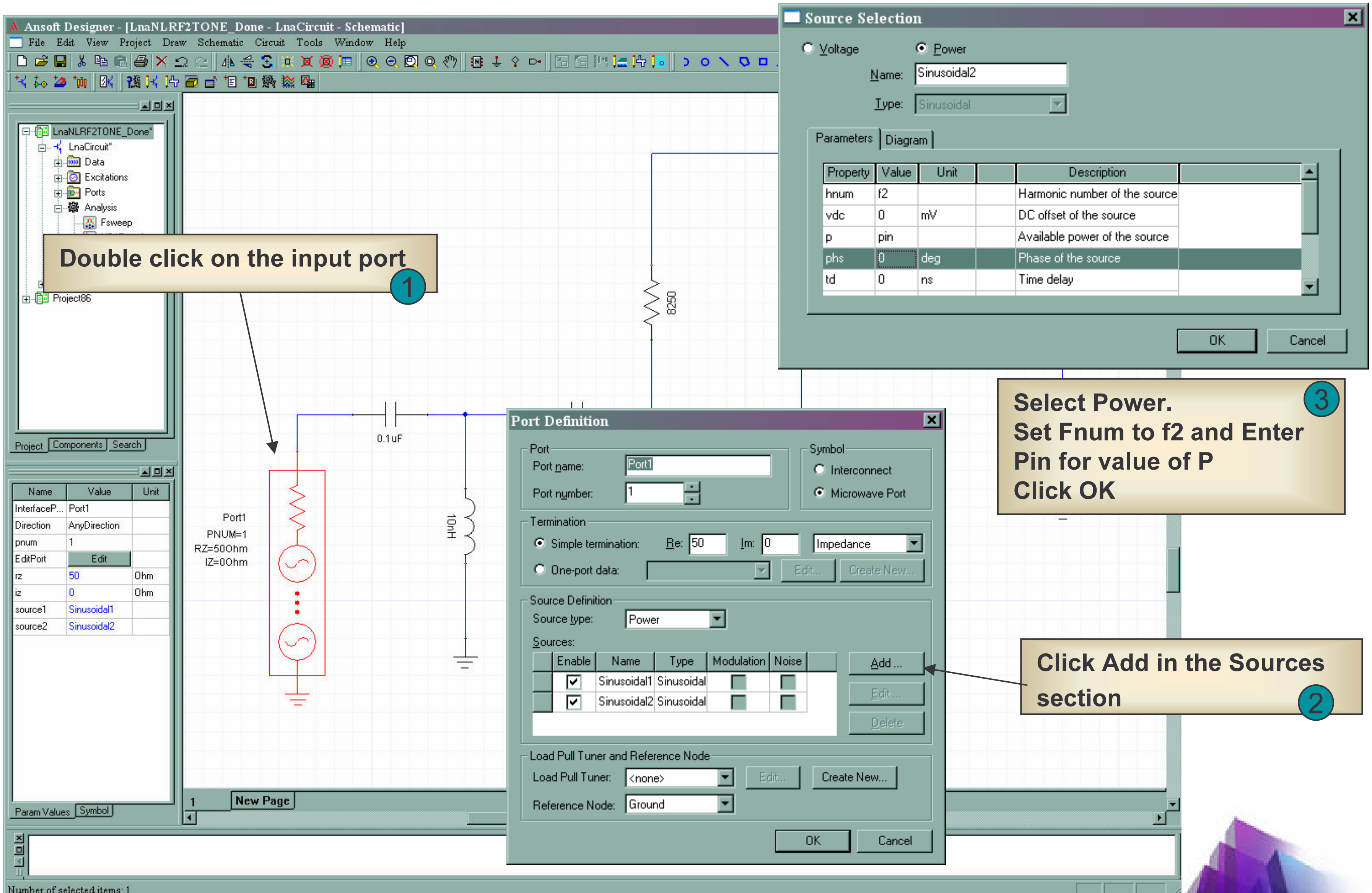
Select HB1Tone1 in field Solution
Select Time in field Domain
Select Pin = -40, -30, -20, -10 dbm for the Pin values 2

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

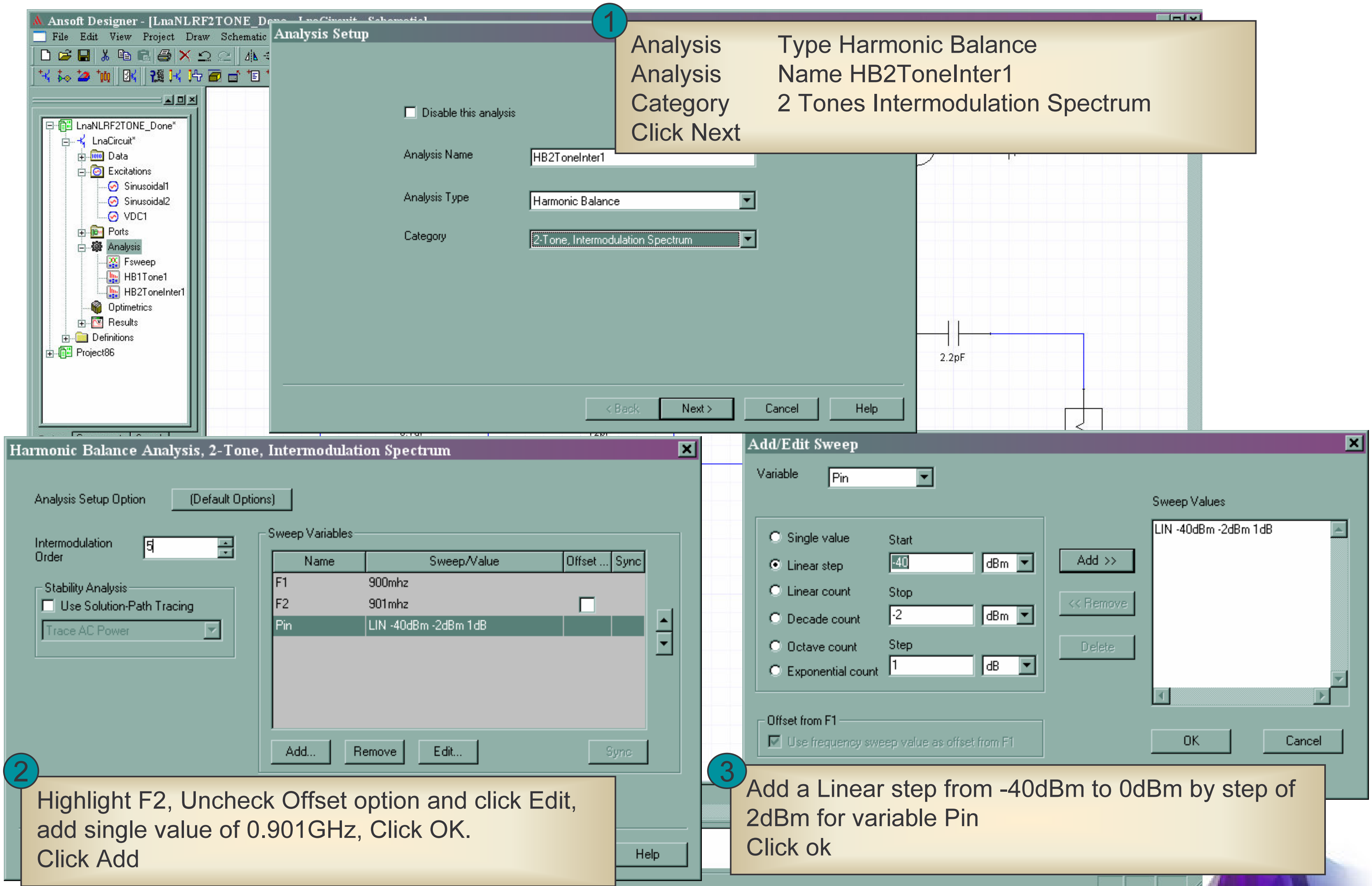
Save the Project

Optional Exercises:
(Intermod)
Digital Modulation

Add a Second RF Source to Port 1



Add Intermodulation Analysis Setup



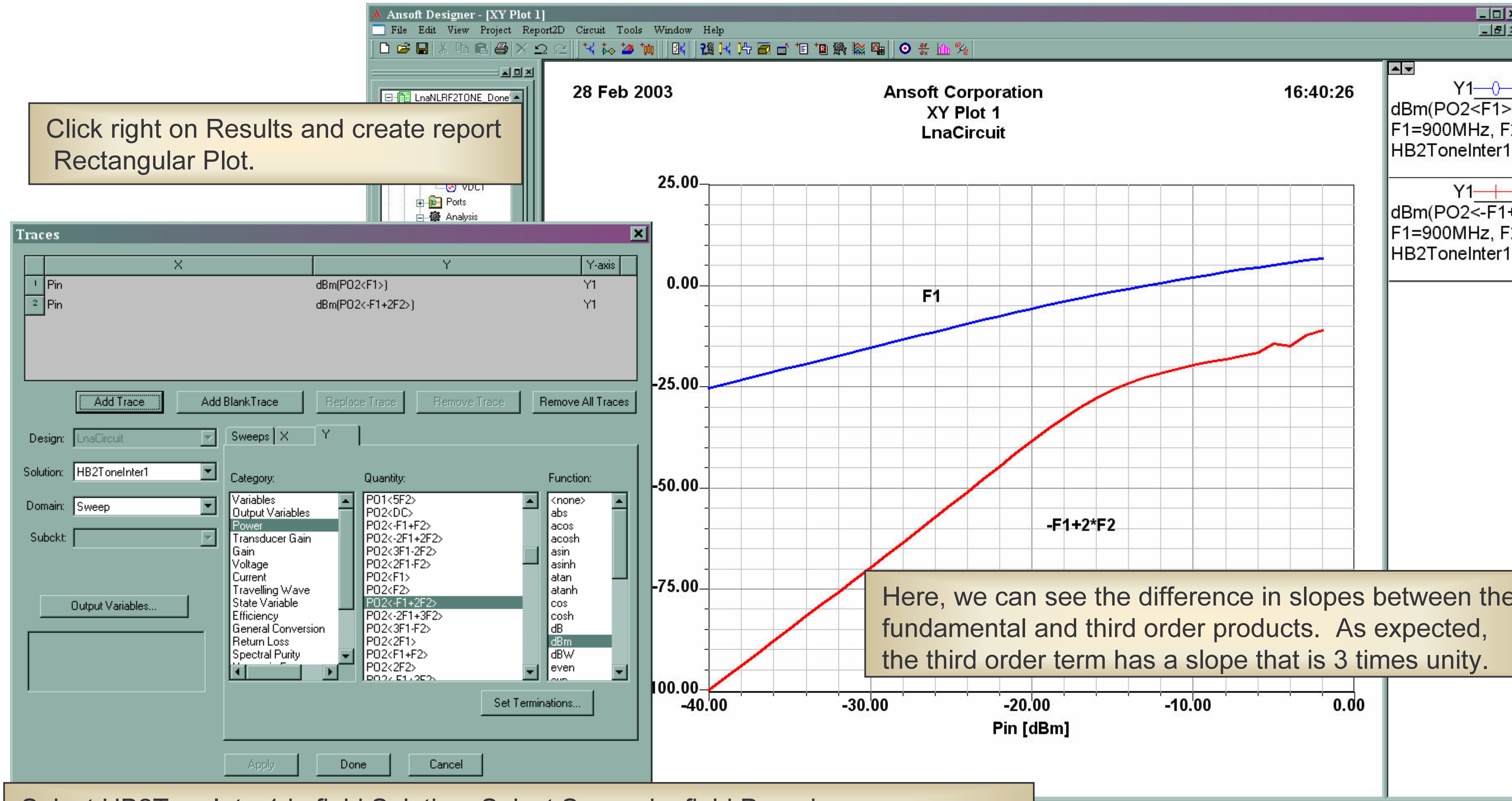
1. Analysis Type Harmonic Balance
Analysis Name HB2ToneInter1
Category 2 Tones Intermodulation Spectrum
Click Next

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

Create Results: Pout vs Pin

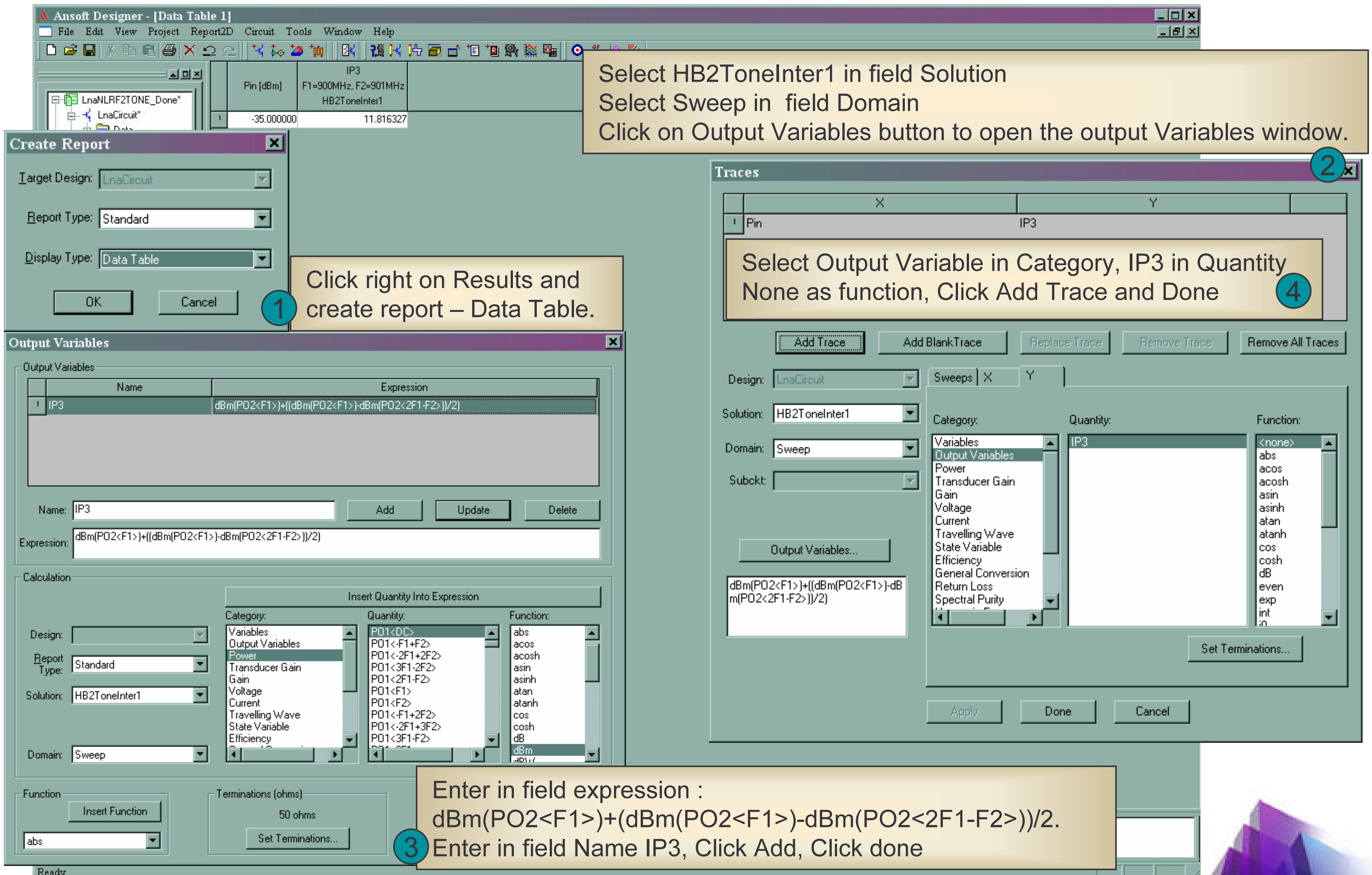
Click right on Results and create report Rectangular Plot.



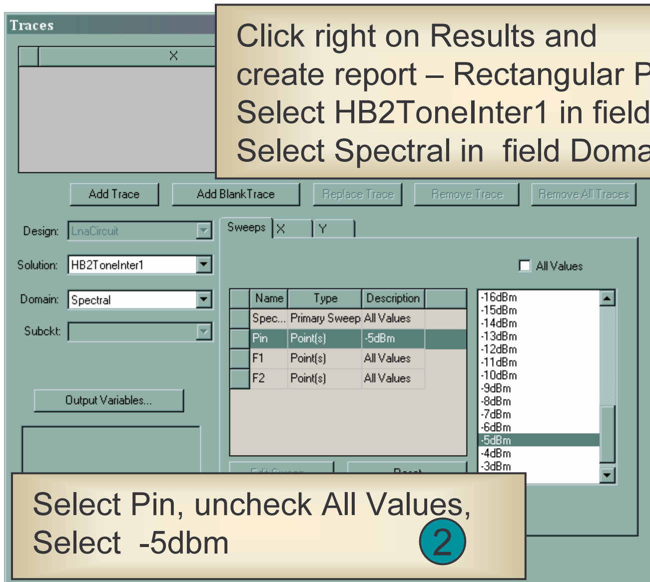
Here, we can see the difference in slopes between the fundamental and third order products. As expected, the third order term has a slope that is 3 times unity.

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

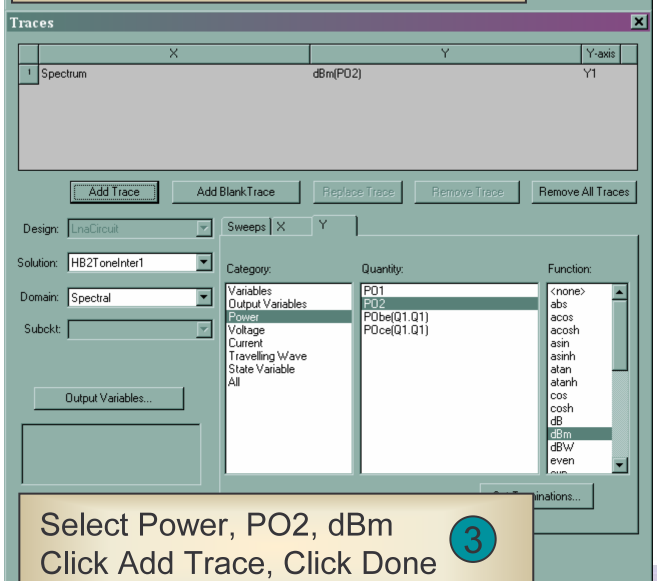


Create Results: Intermodulation Spectrum

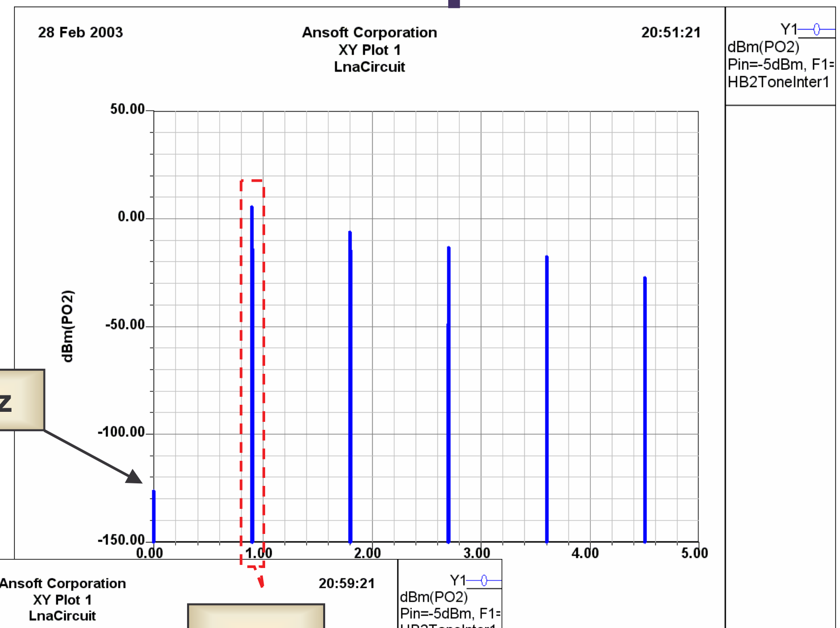


Click right on Results and create report – Rectangular Plot
Select HB2ToneInter1 in field Solution
Select Spectral in field Domain 1

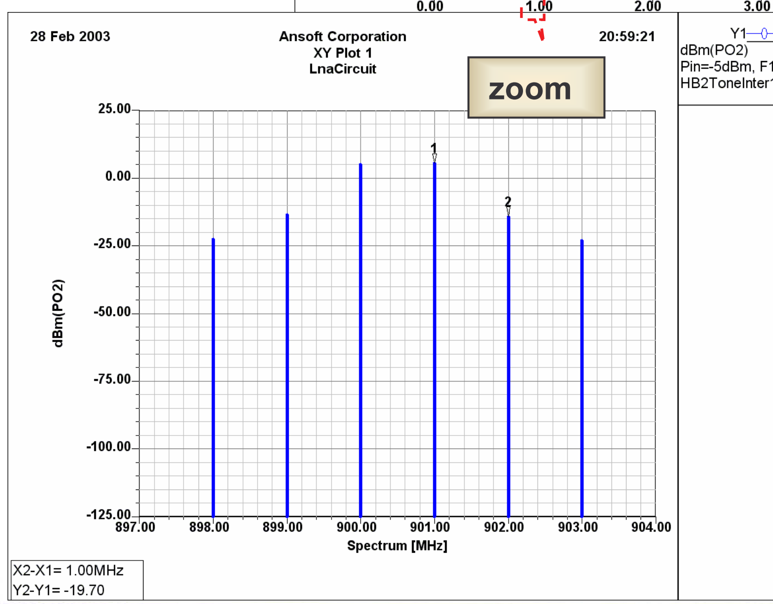
Select Pin, uncheck All Values,
Select -5dbm 2



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

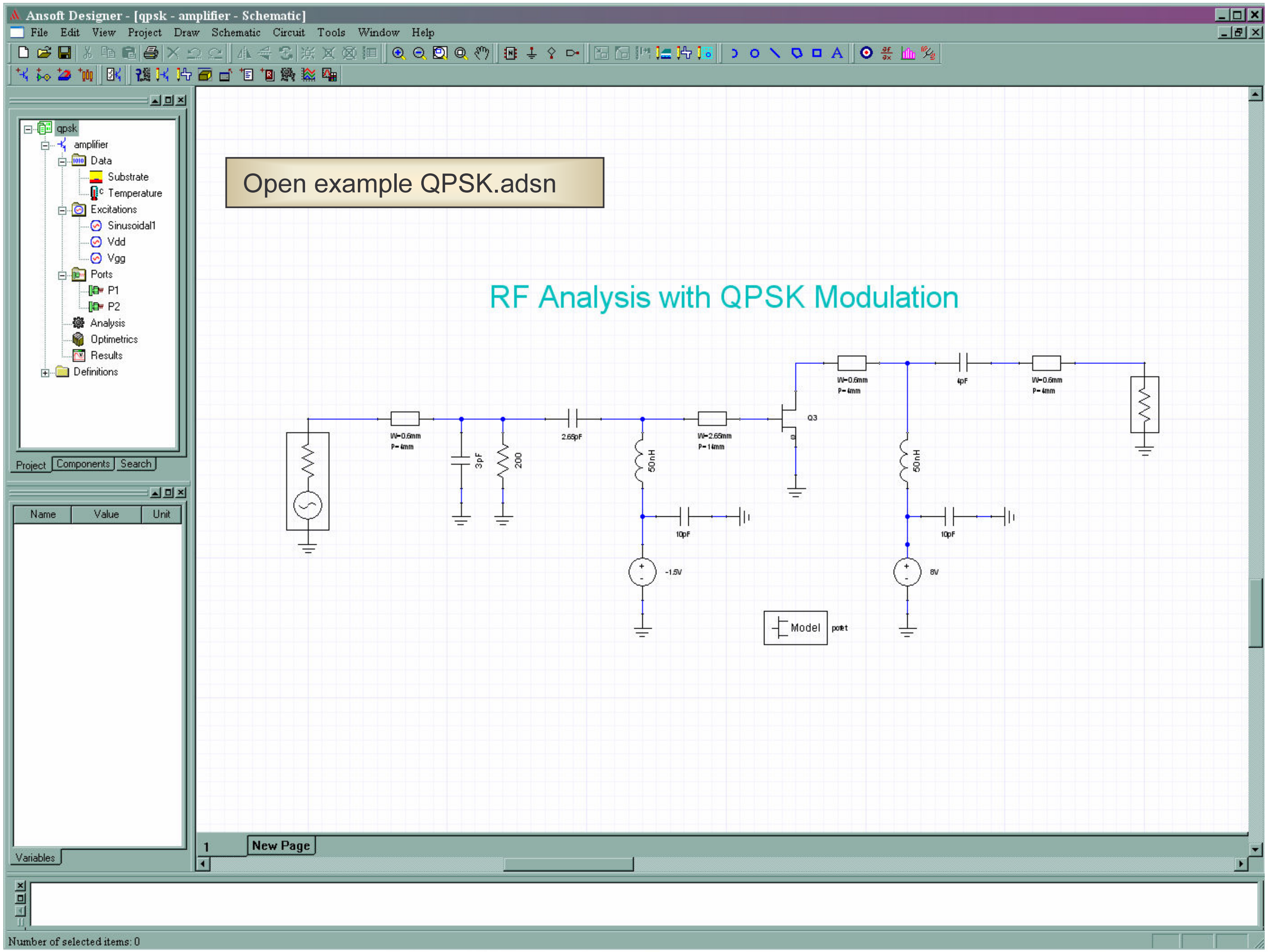


1MHz

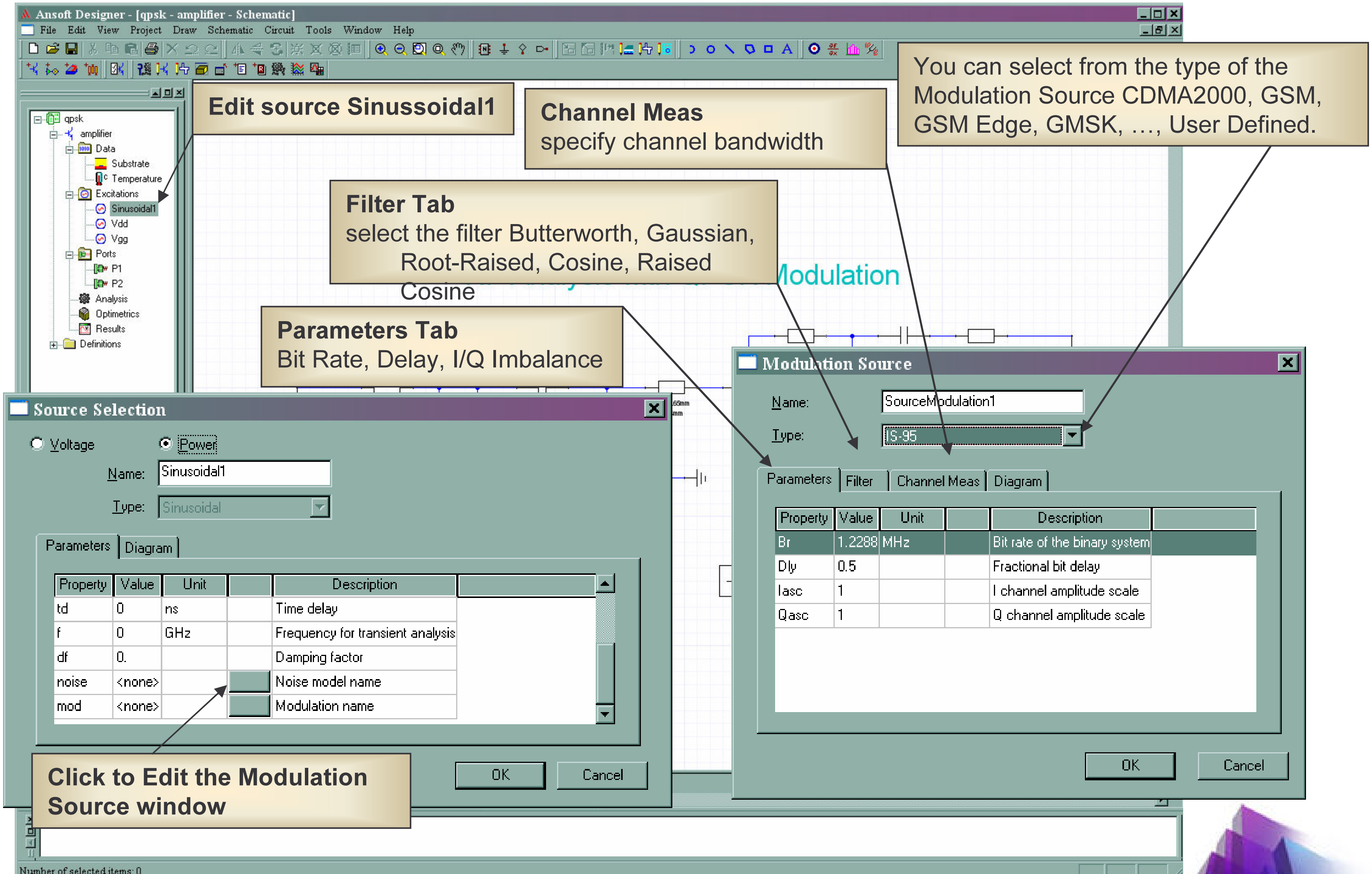


zoom

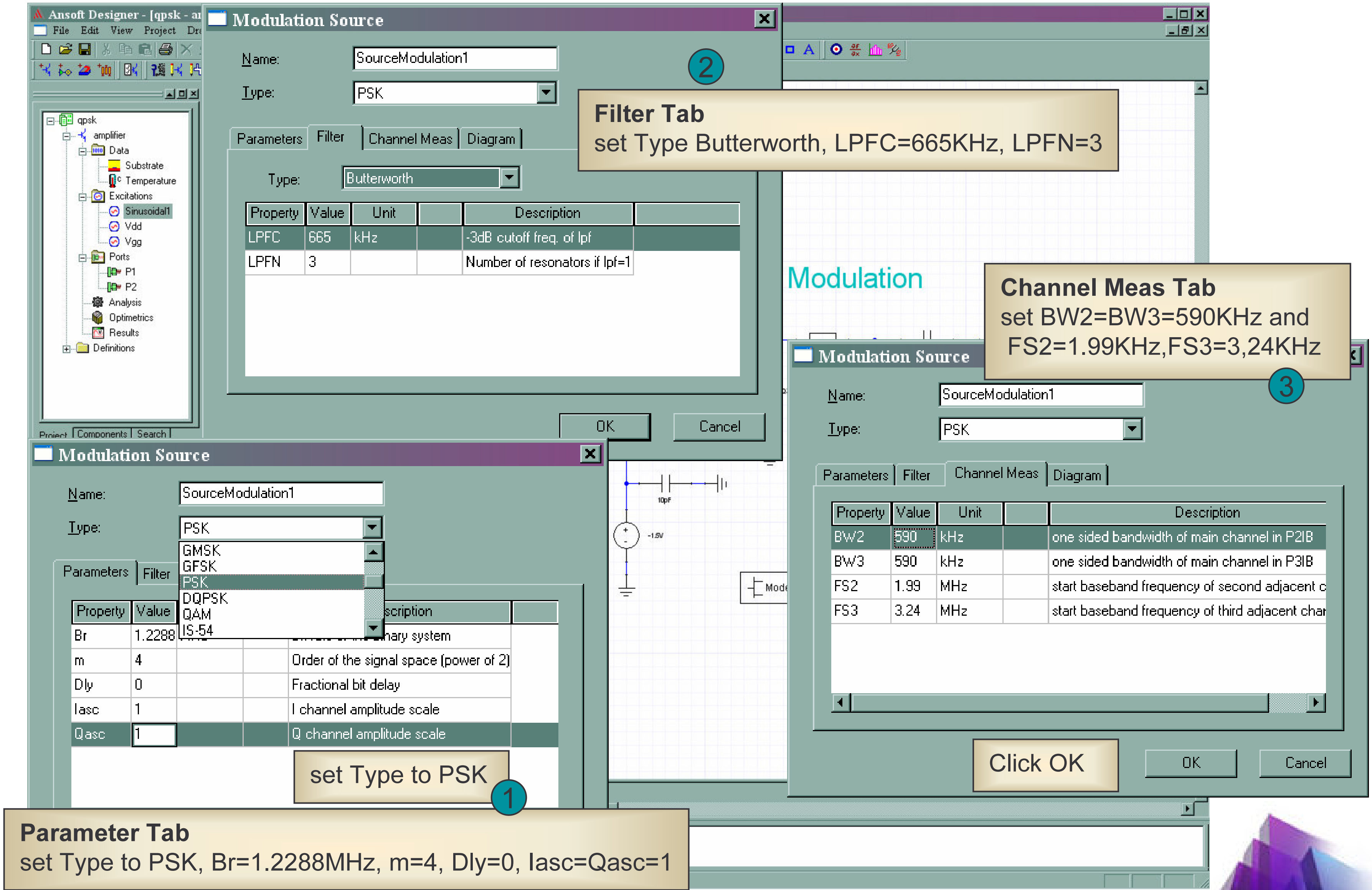
Digital Modulation



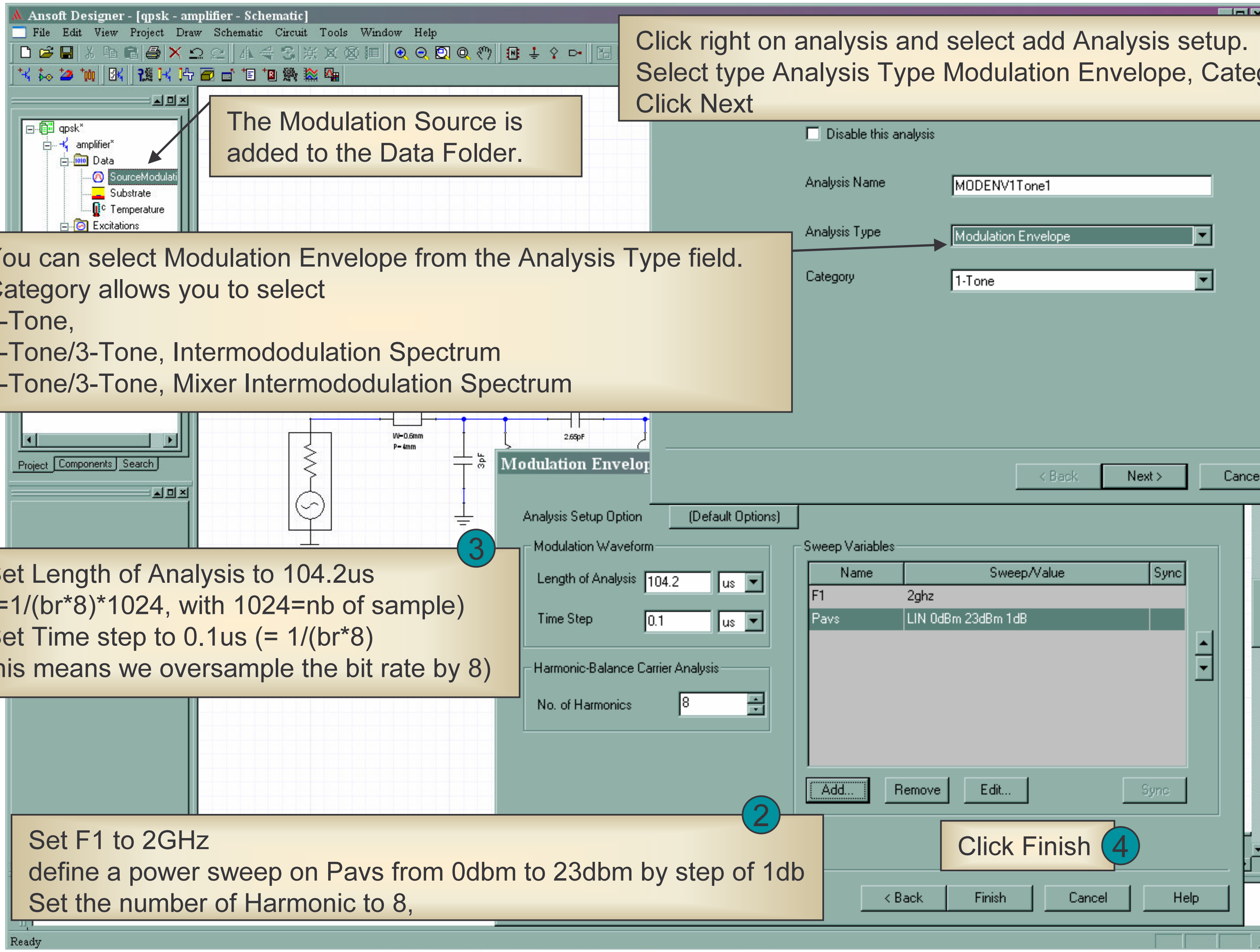
Digital Modulation: Modulation Source (1)



Digital Modulation: Modulation Source (1)



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,

Click Finish 4

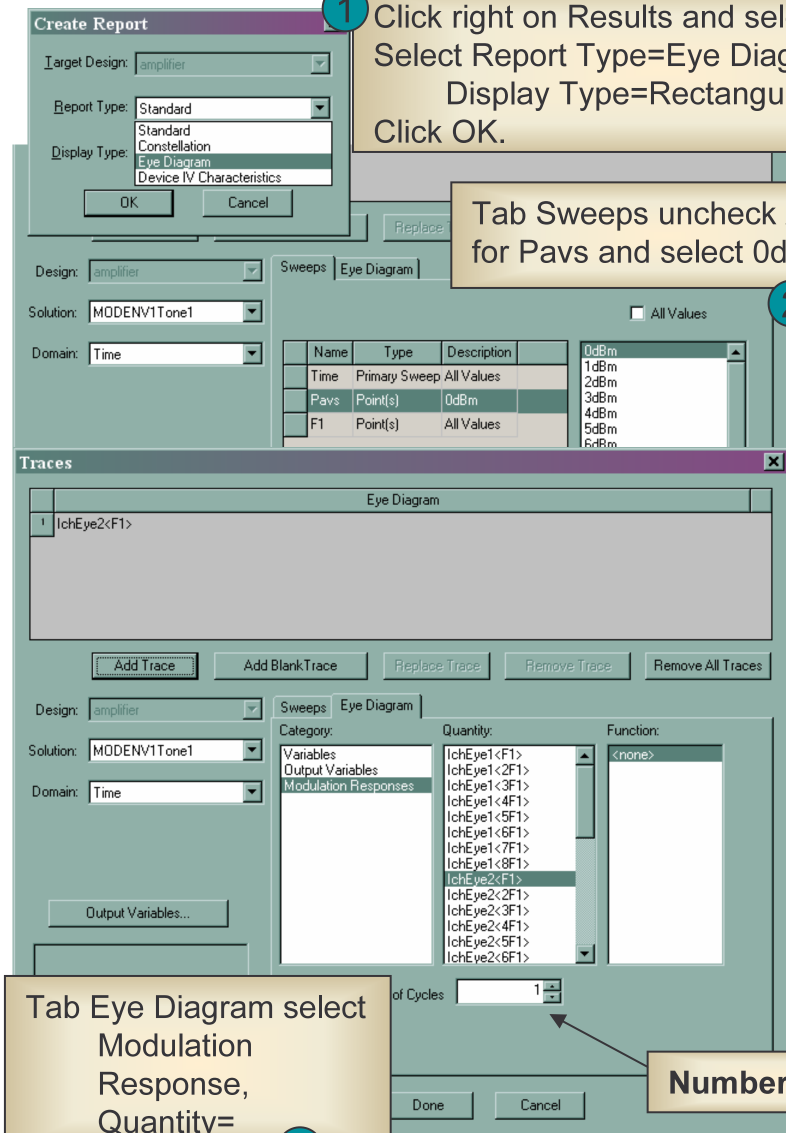
Digital Modulation: Eye Diagram

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

2 Tab Sweeps uncheck All for Pavs and select 0dBm.

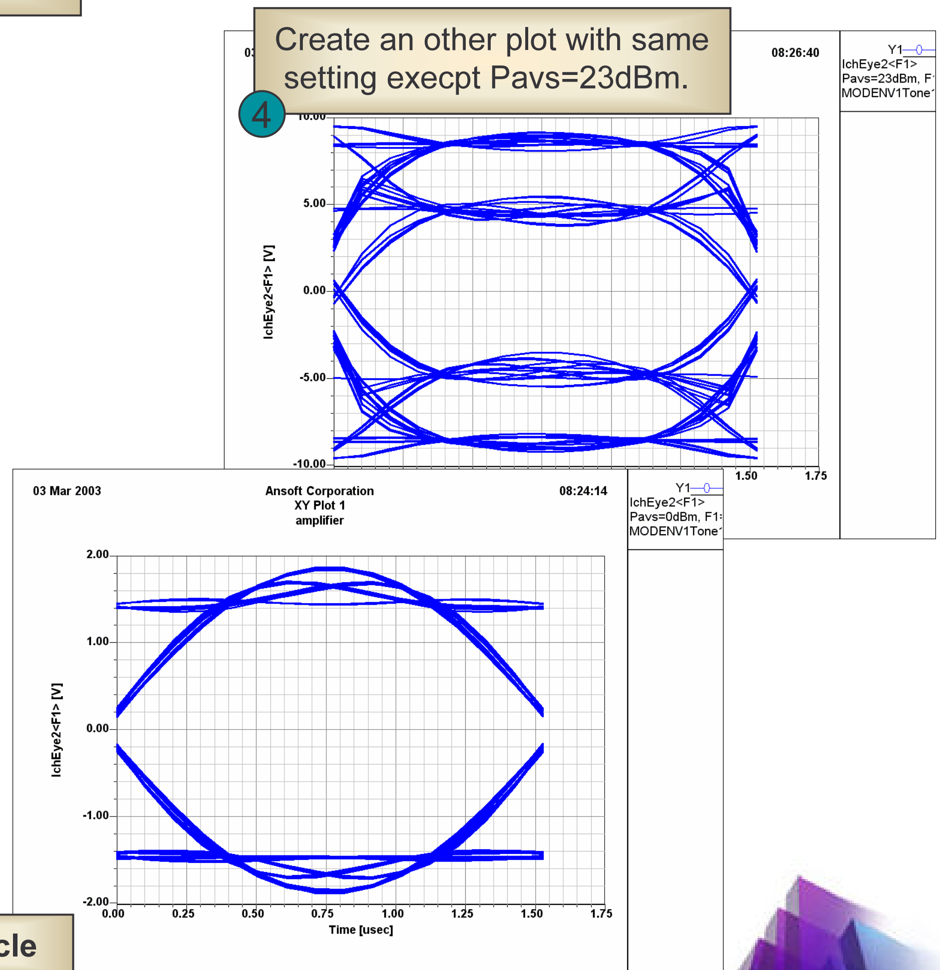
3 Tab Eye Diagram select
Modulation Response,
Quantity=IchEye2<F1>
Click Done

Number of Cycle



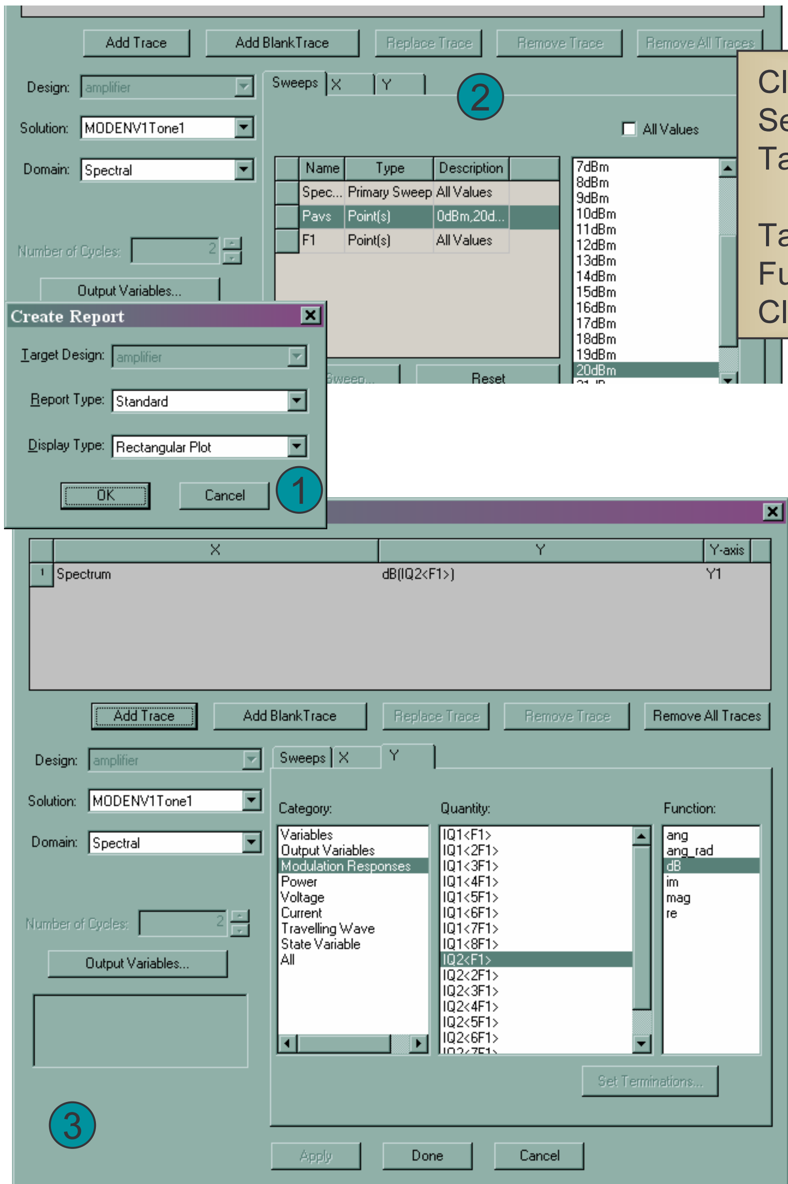
Available plot for Modulation Envelope are:
Eye Diagram, Constellation, IQ Spectrum, ACPR

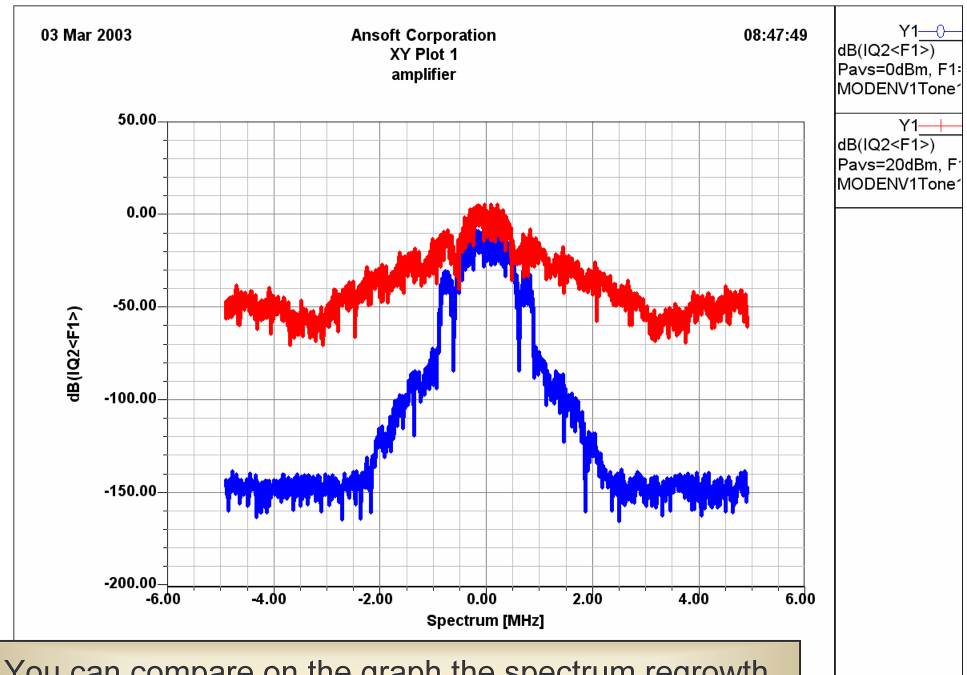
4 Create an other plot with same setting execpt Pavs=23dBm.



Digital Modulation: IQ Spectrum

Click right on Results and select Create.
Select Report Type=Standard, Display Type=Rectangular Plot, Click OK. 1
Tab Sweeps uncheck All for Pavs, select 0dBm and 20dBm using the CTRL key. 2
Tab Y select Category=Modulation Responses, Quantity=IQ2<F1>, Function dB. 3
Click Done.

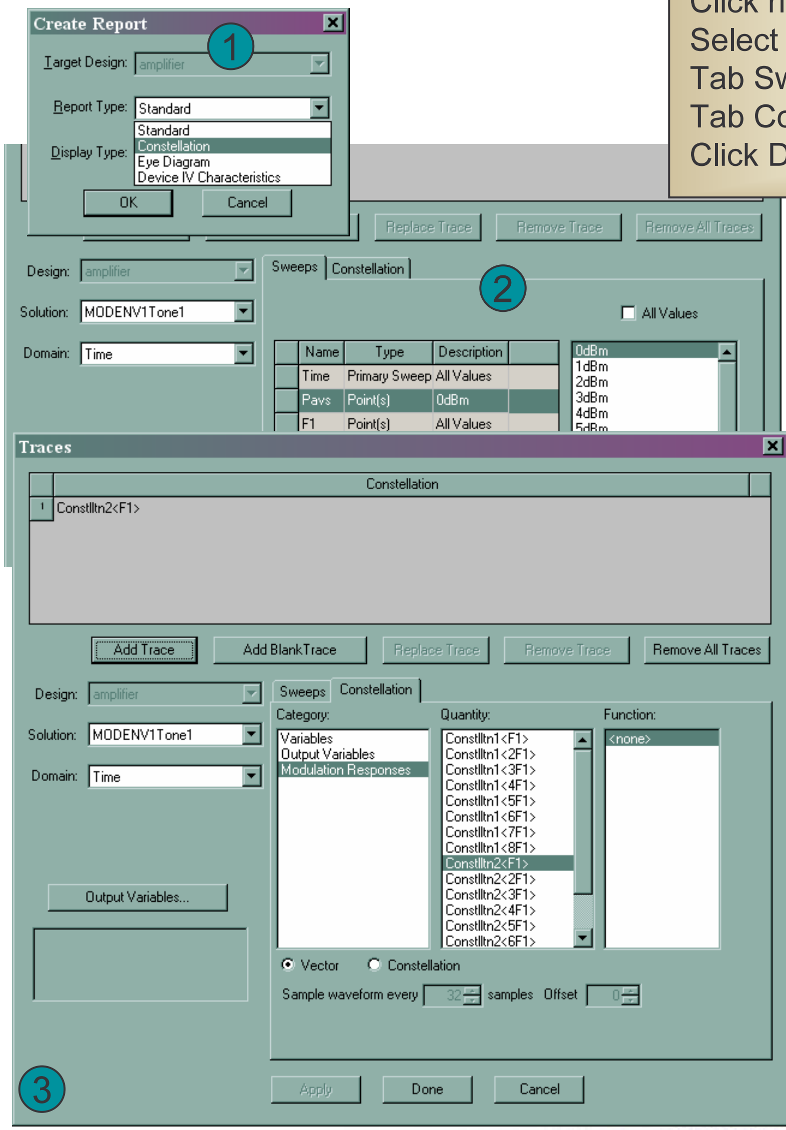


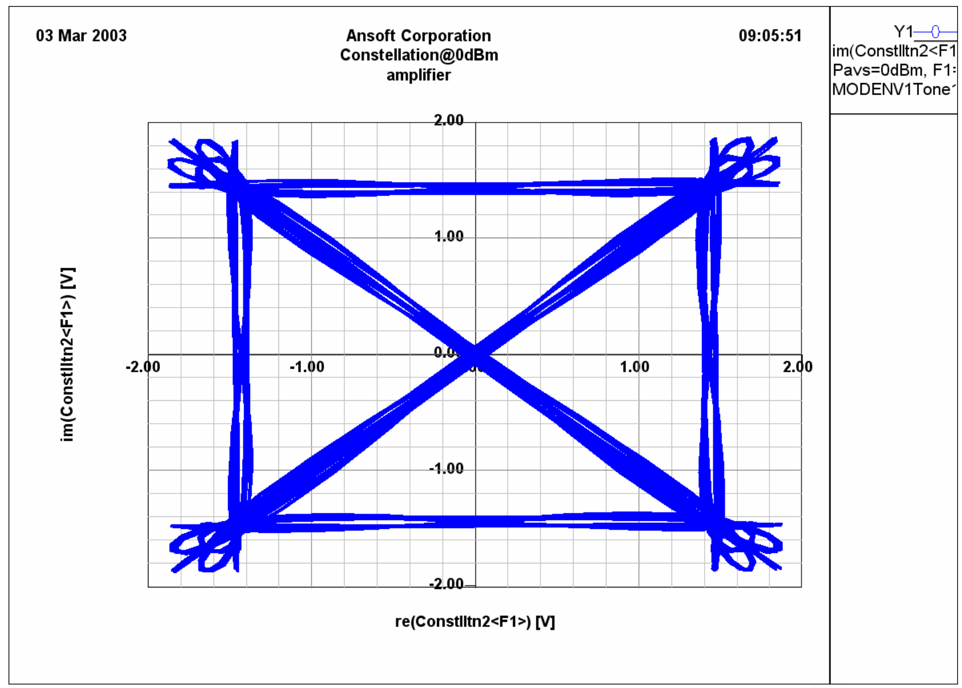


You can compare on the graph the spectrum regrowth produced for Pin=0dBm and Pin=20dBm.

Digital Modulation: Constellation Plot

Click right on Results and select Create
Select Report Type=Constellation, Display Type=Rectangular Plot, Click OK. 1
Tab Sweeps uncheck All for Pavs and select 0dBm. 2
Tab Constellation select Modulation Response, Quantity= Constllltn2<F1>. 3
Click Done





Load Pull Analysis

Overview

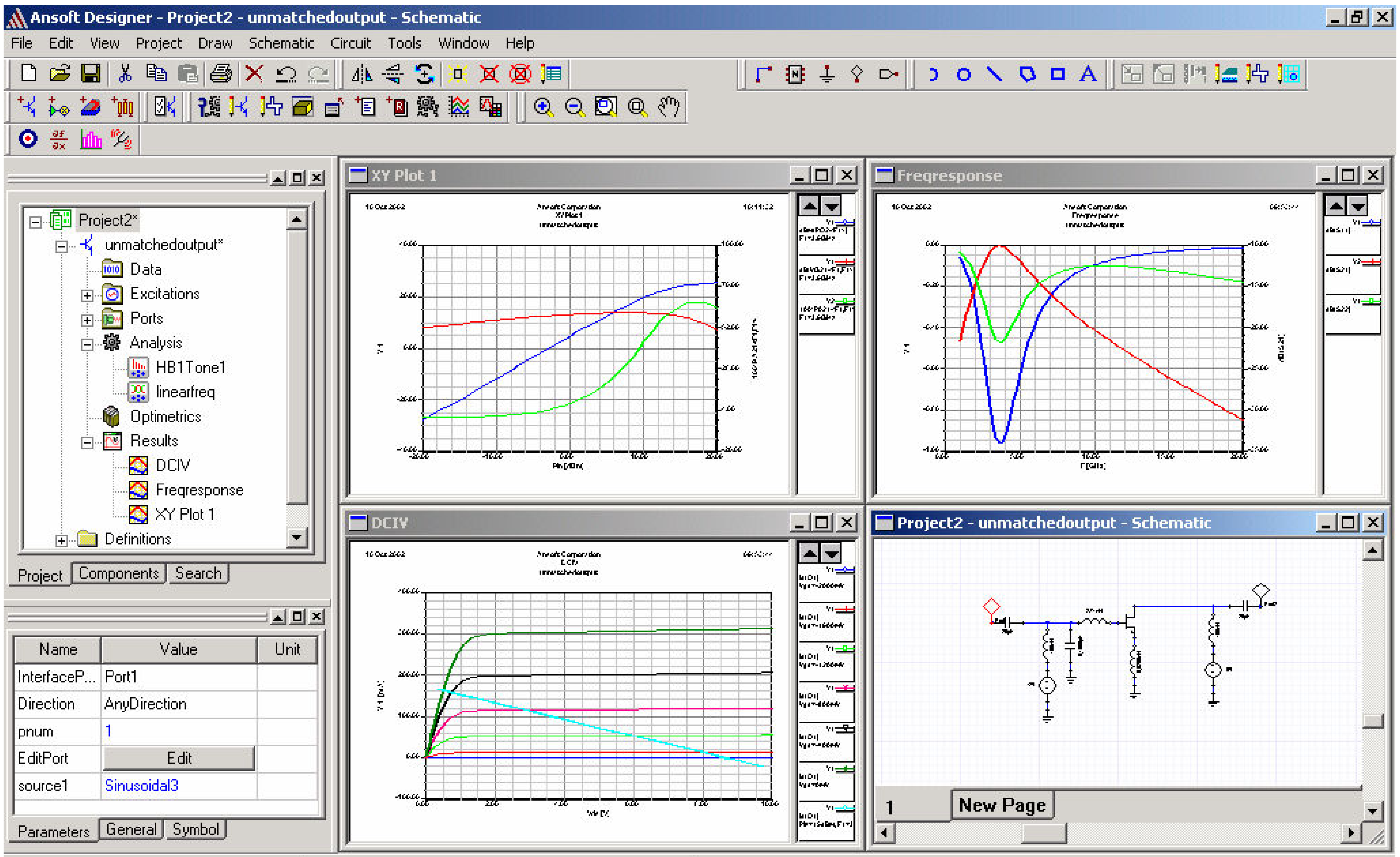
There are 7 essential Steps to perform load-pull analysis and review response

1. Define nonlinear circuit
2. Specify nonlinear source

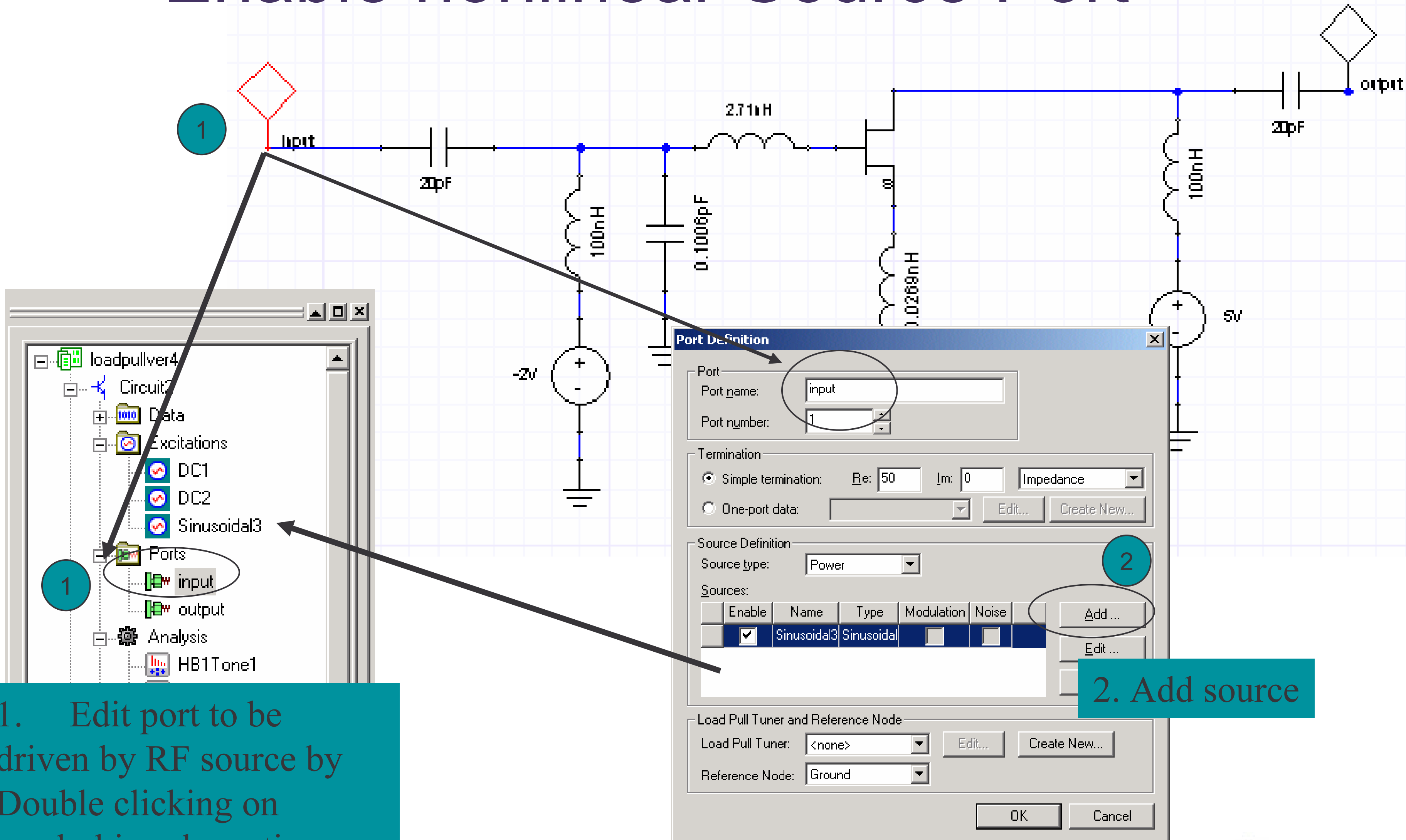
Viewed in previous slides

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

Loadpull Analysis in Ansoft Designer



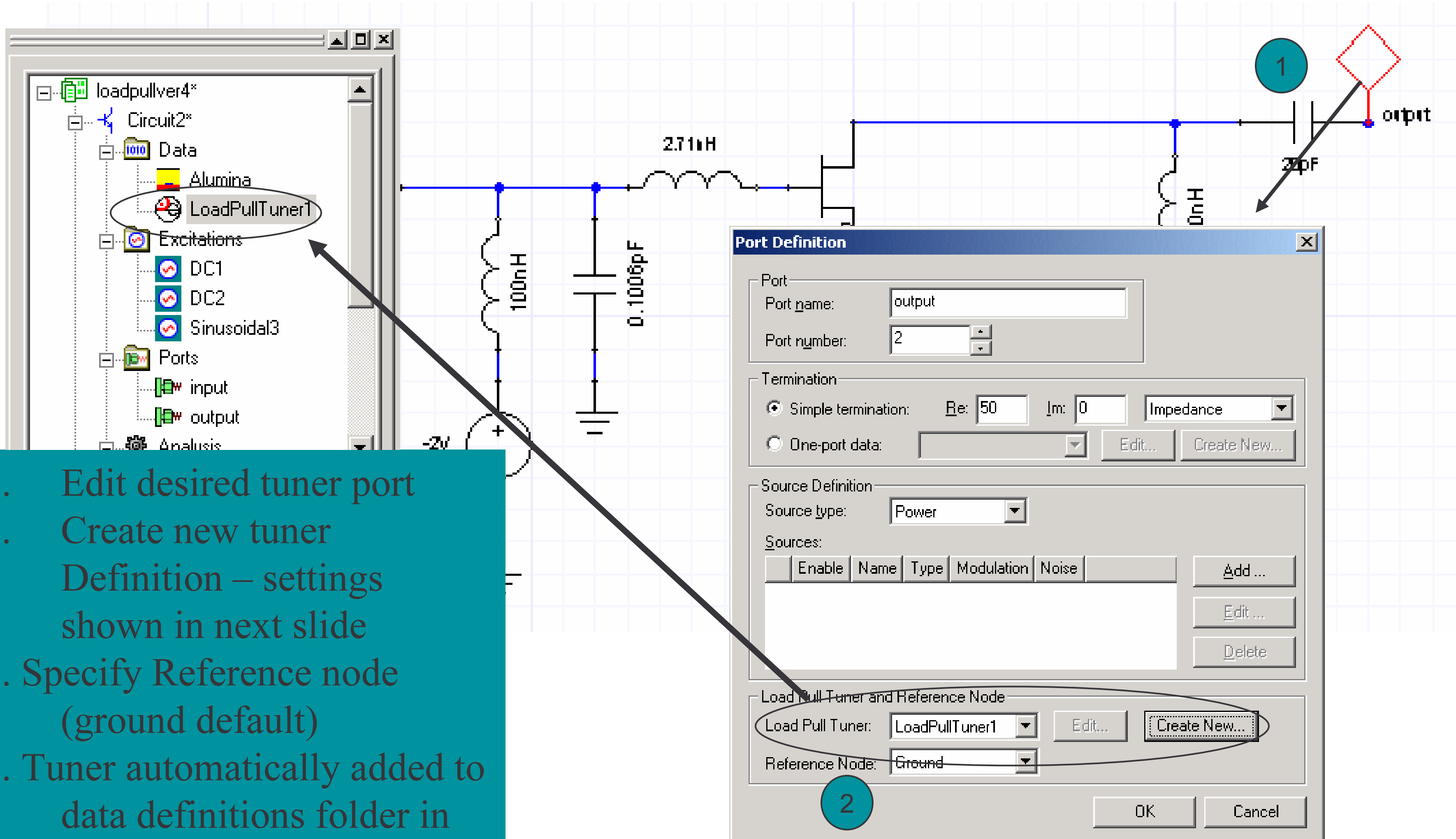
Enable nonlinear Source Port



2. Add source

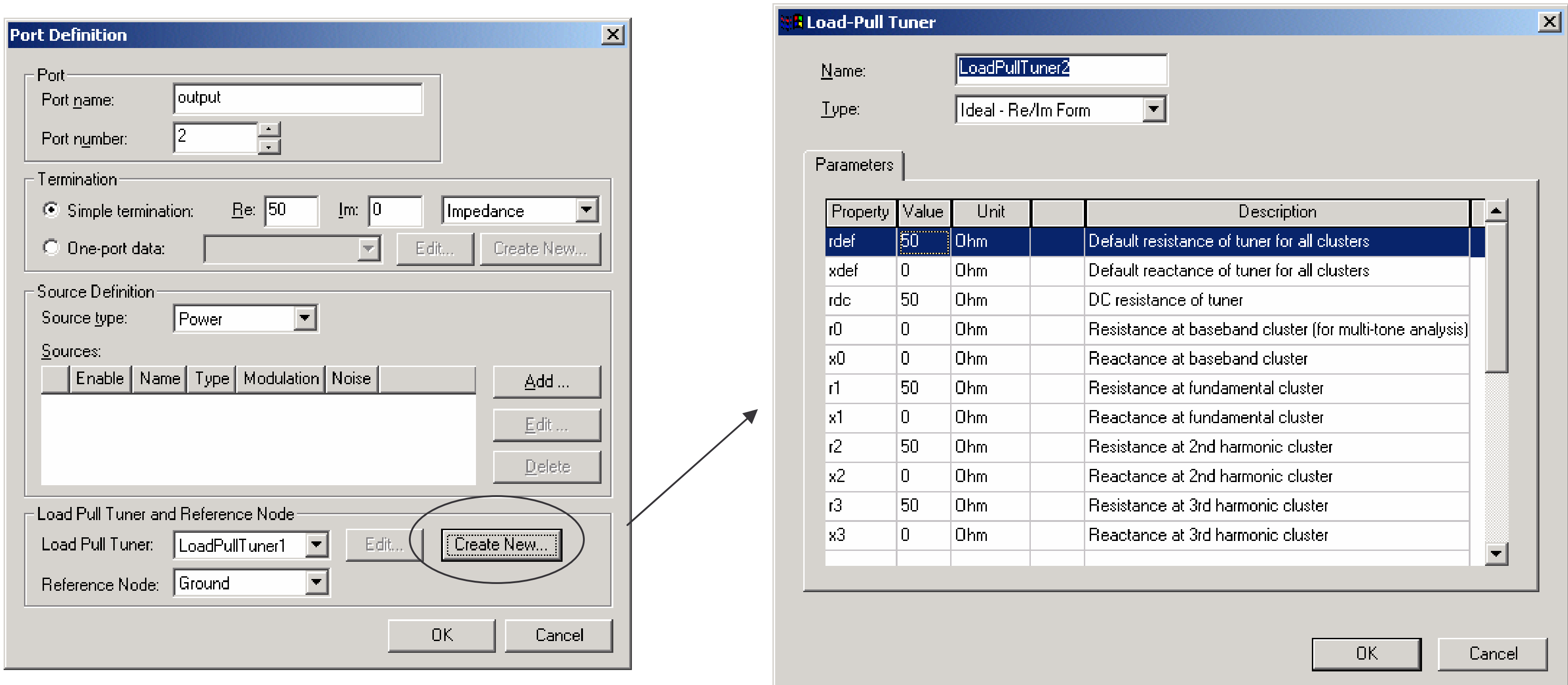
1. Edit port to be driven by RF source by Double clicking on symbol in schematic or icon in project tree

Enable Loadpull termination



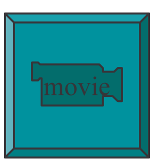
1. Edit desired tuner port
2. Create new tuner Definition – settings shown in next slide
2. Specify Reference node (ground default)
3. Tuner automatically added to data definitions folder in project tree

Define Loadpull tuner characteristics

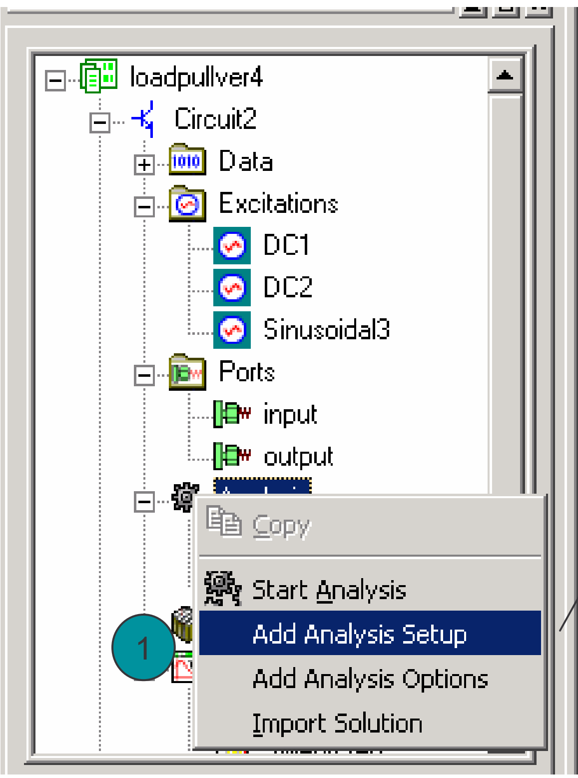


Loadpull tuner definition specifies tuner type (ideal, double stub) and default complex impedance at all specified harmonic frequencies (50ohm default). To modify edit from port dialog or double click on tuner icon in project tree.

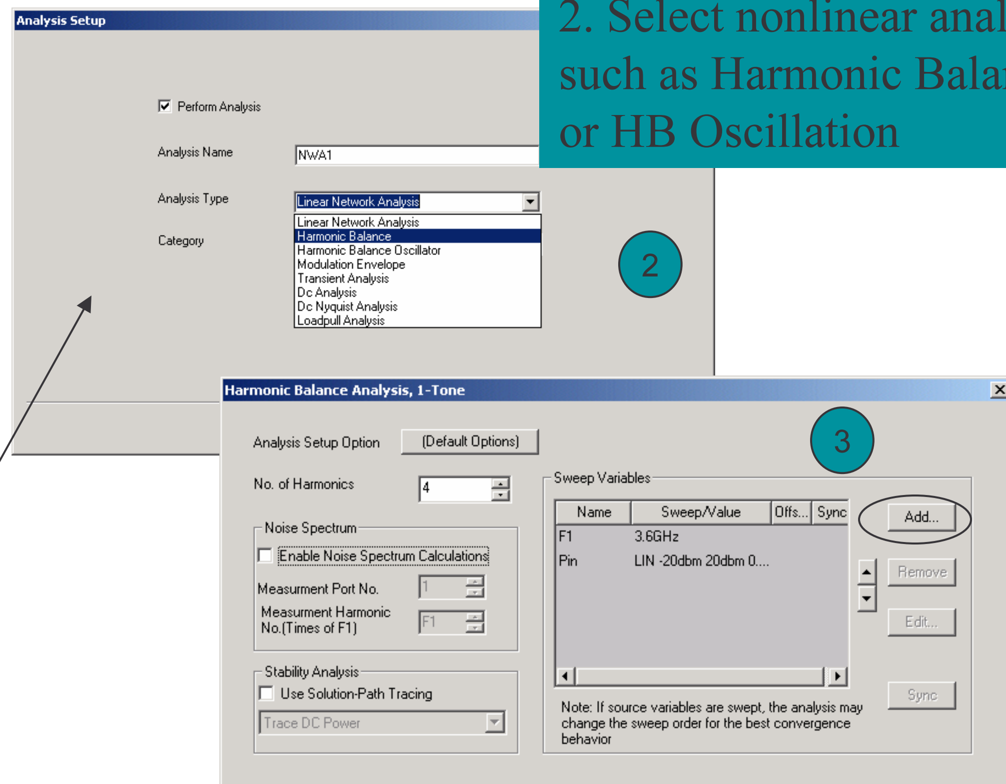
Note: multiple tuners may be defined



Create new nonlinear analysis



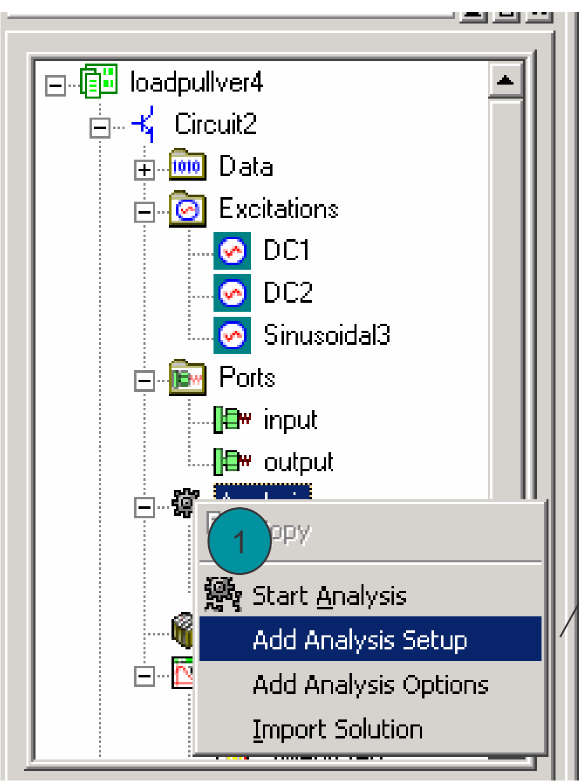
1. Add analysis setup



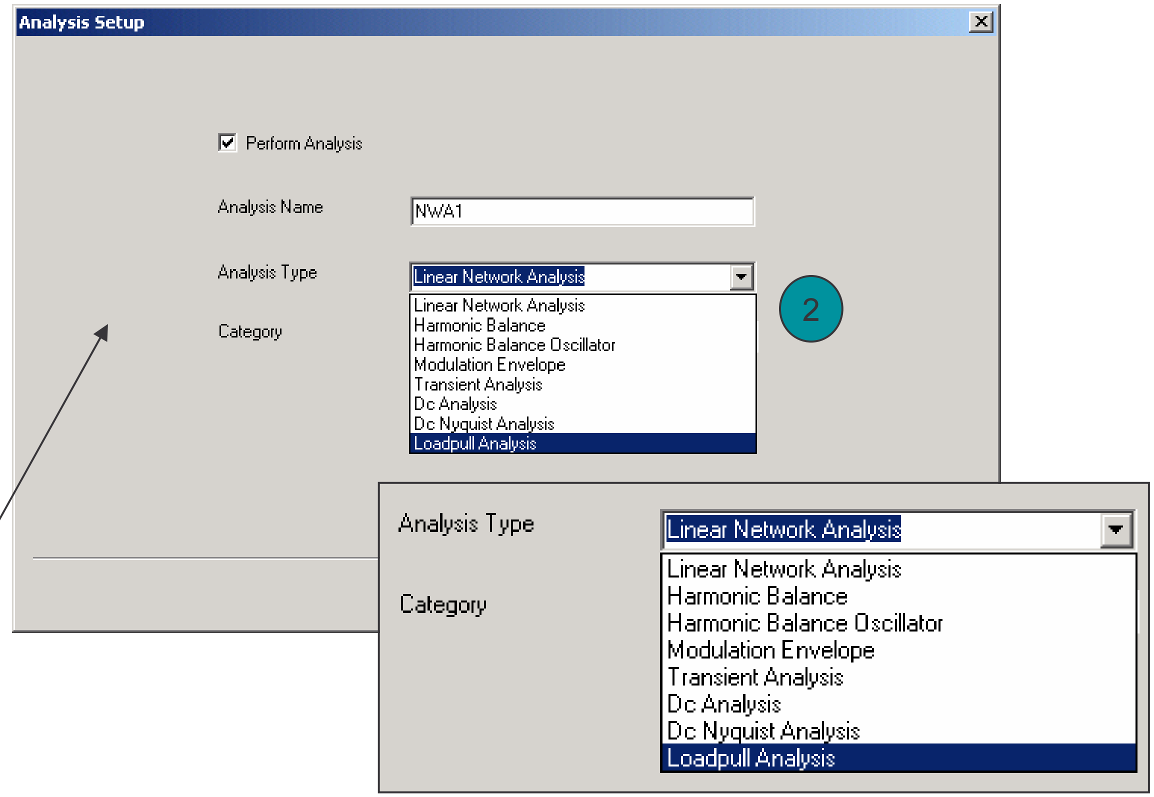
2. Select nonlinear analysis such as Harmonic Balance or HB Oscillation

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



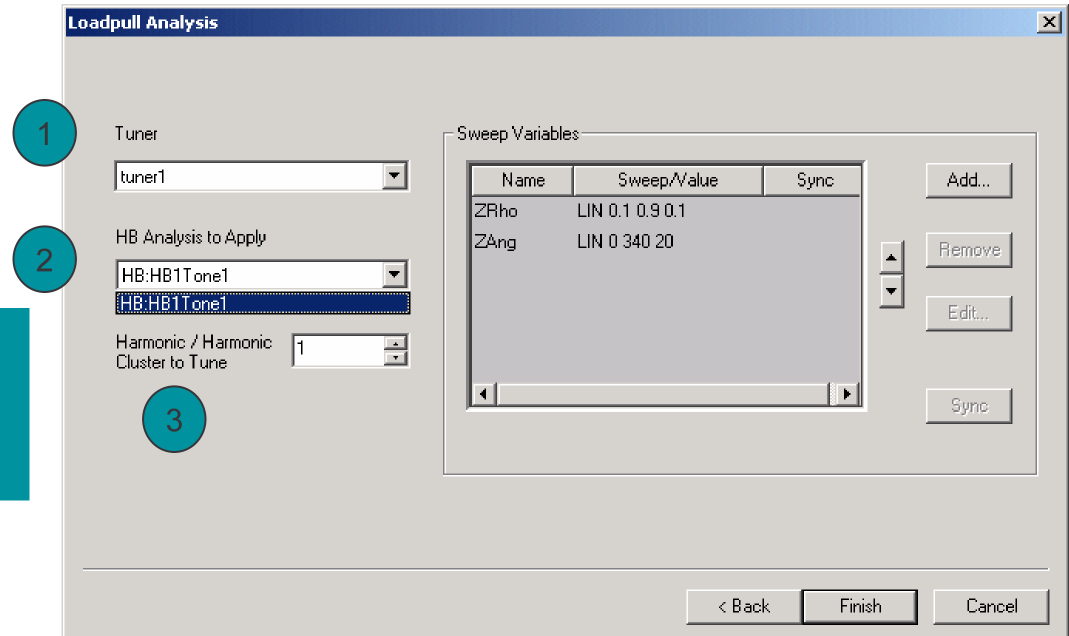
2. Specify swept parameters such as operating Frequencies and swept parameters such as Bias, tuning or power level (using the Add… Button)

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)



movie

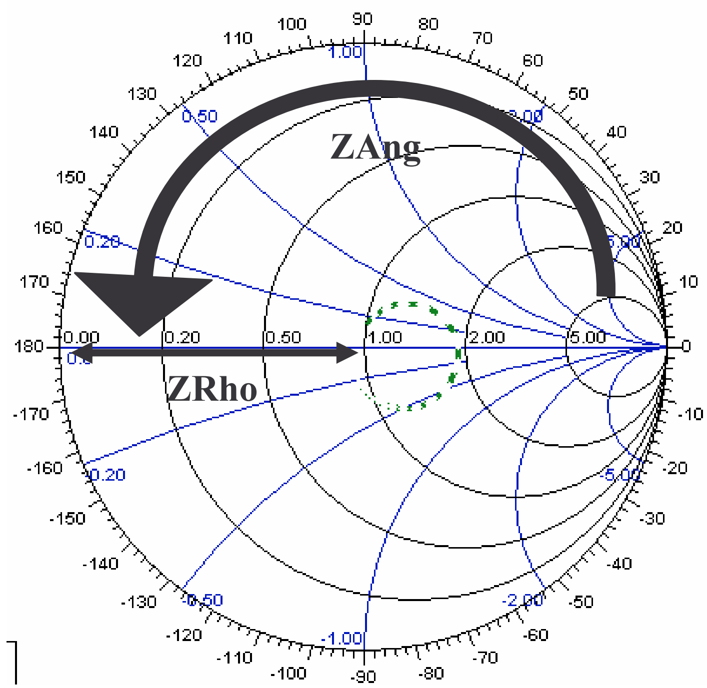
Create Loadpull analysis

Set tuner range
a. ZRho controls the magnitude of the reflection coefficient
b. ZAng controls the angle of the reflection coefficient

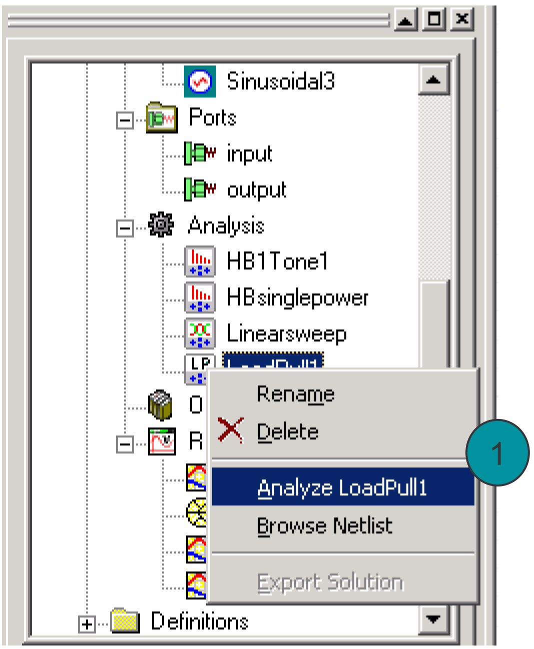
Hint: low impedance matching networks that are common to high power amplifiers are simulated more quickly by using a directed loadpull sweep of low impedances

Sweep Variables

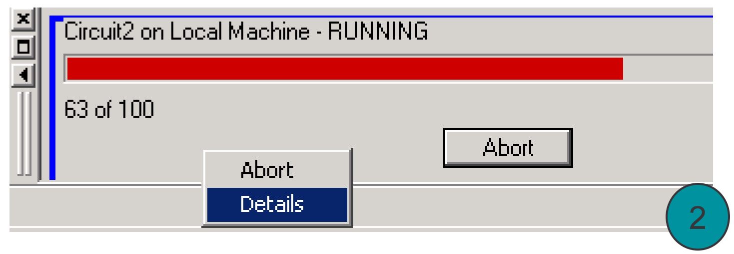
Name	Sweep/Value	Sync
ZRho	LIN 0.1 0.9 0.1	
ZAng	LIN 0 340 20	



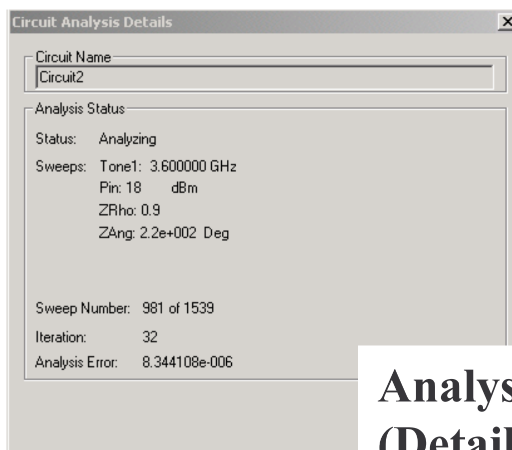
Perform Loadpull analysis



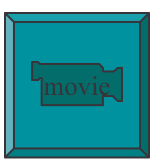
1. Right mouse click on the defined loadpull analysis setup in the Analysis folder of the project tree



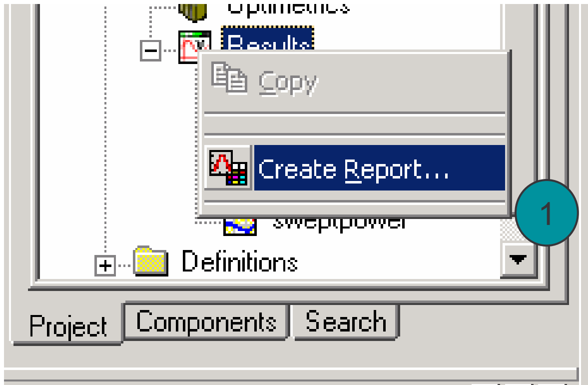
2. Right mouse click on the progress bar to bring up abort/details menu, select details to view HB convergence

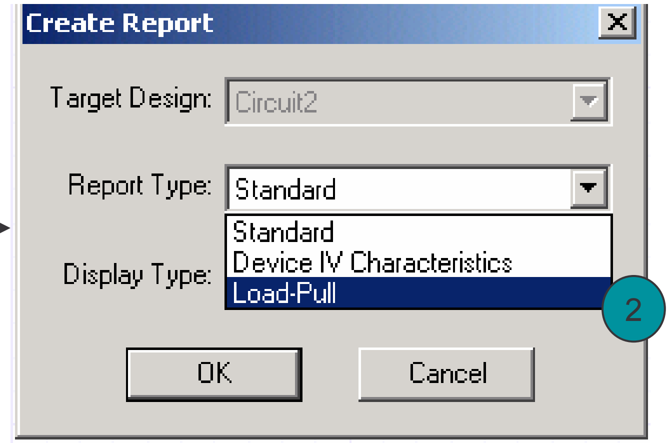


Analysis status window (Details)



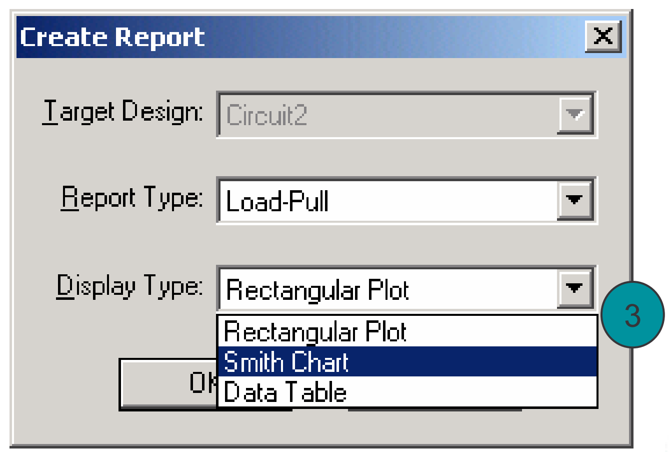
Plotting Results (Contours)





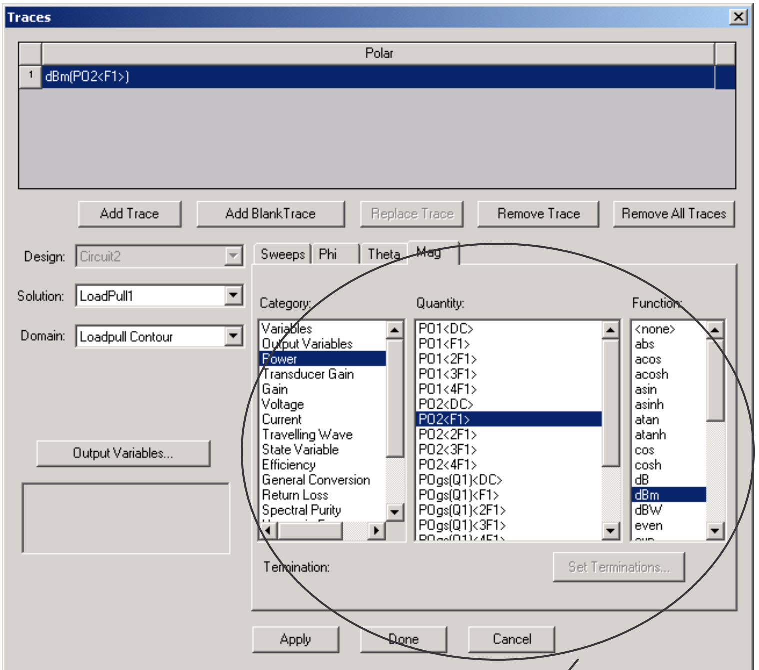
1. Right mouse click on the Results folder to create new report

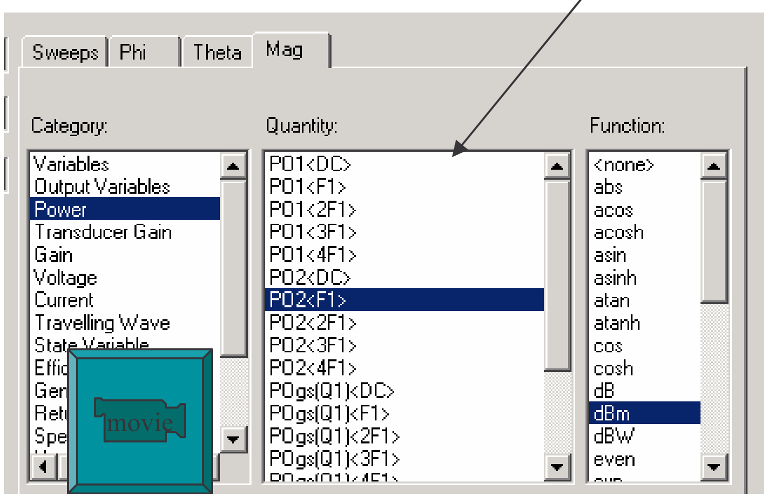
2. Select report type as Load-pull (only available after successful load-pull simulation)

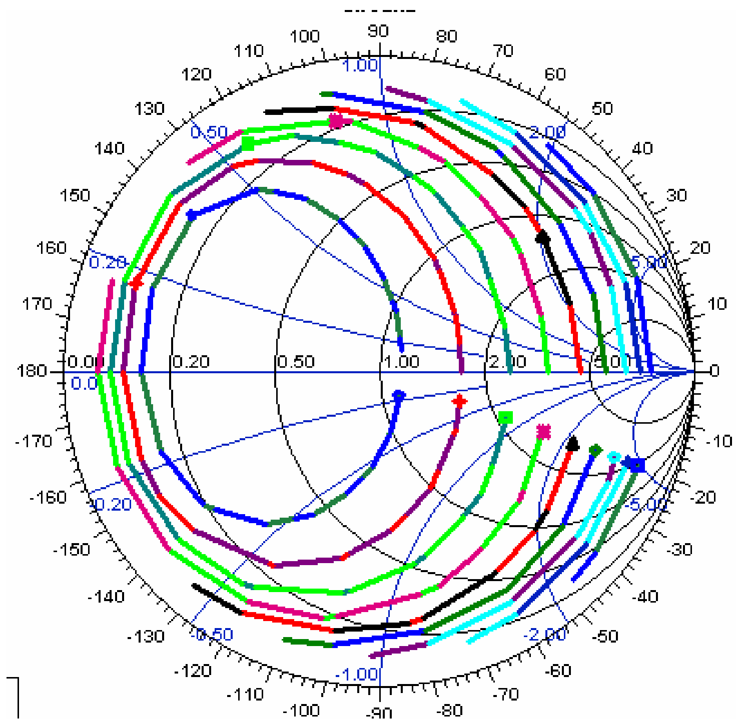


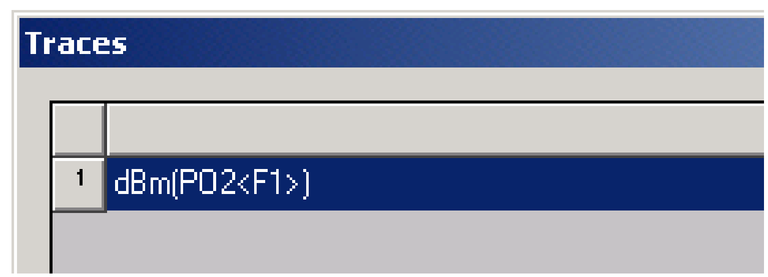
3. Select display type as smith chart (optional rectangular or polar plots also allowed)

Plotting Output Power Contours

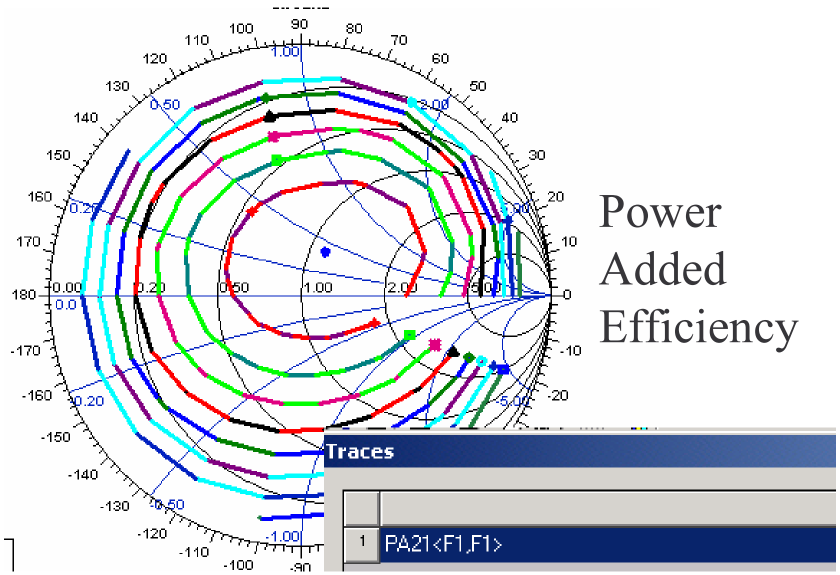




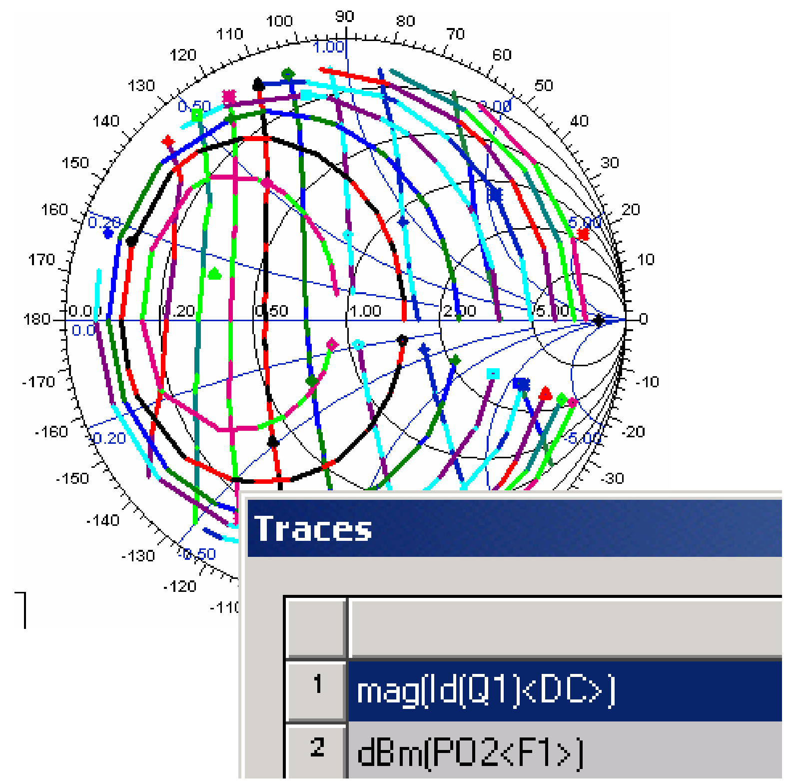


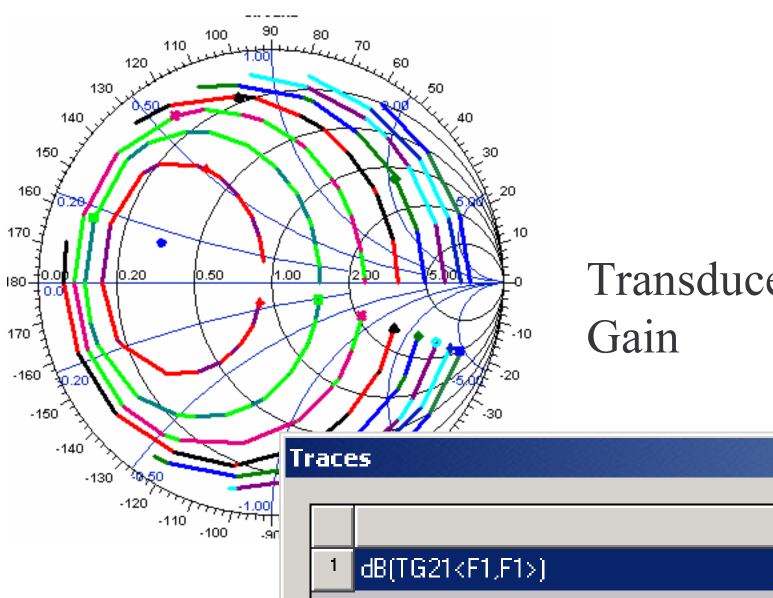


Other Contour Examples



Power
Added
Efficiency





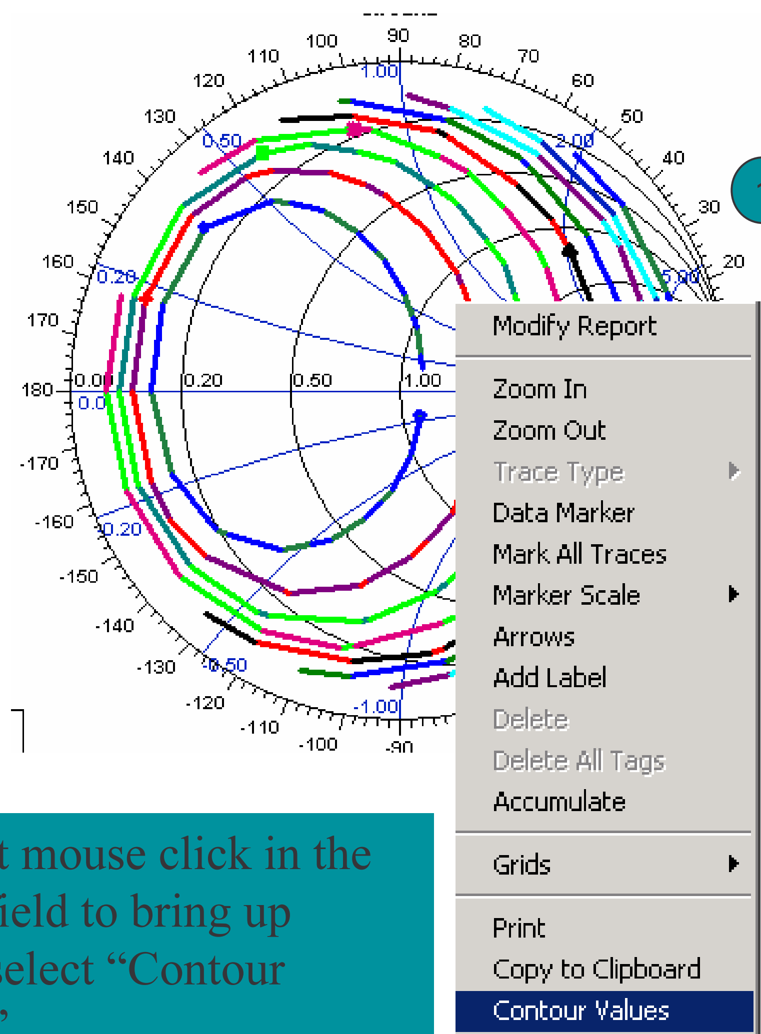
Transducer
Gain

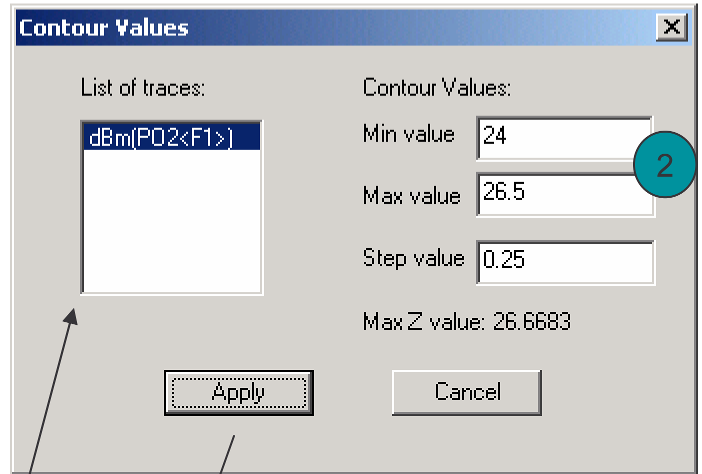
Multiple plots

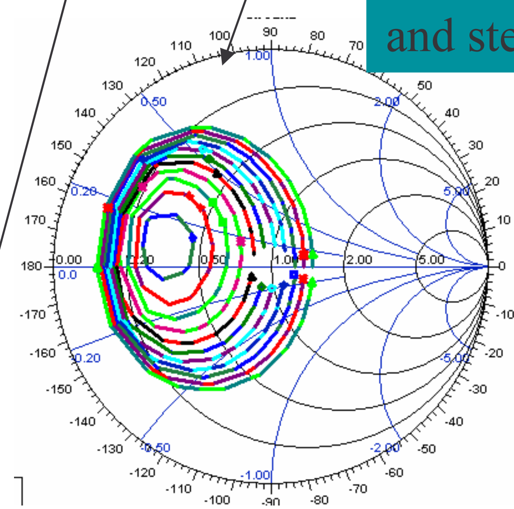
Drain current

Output power

Changing Contour Values







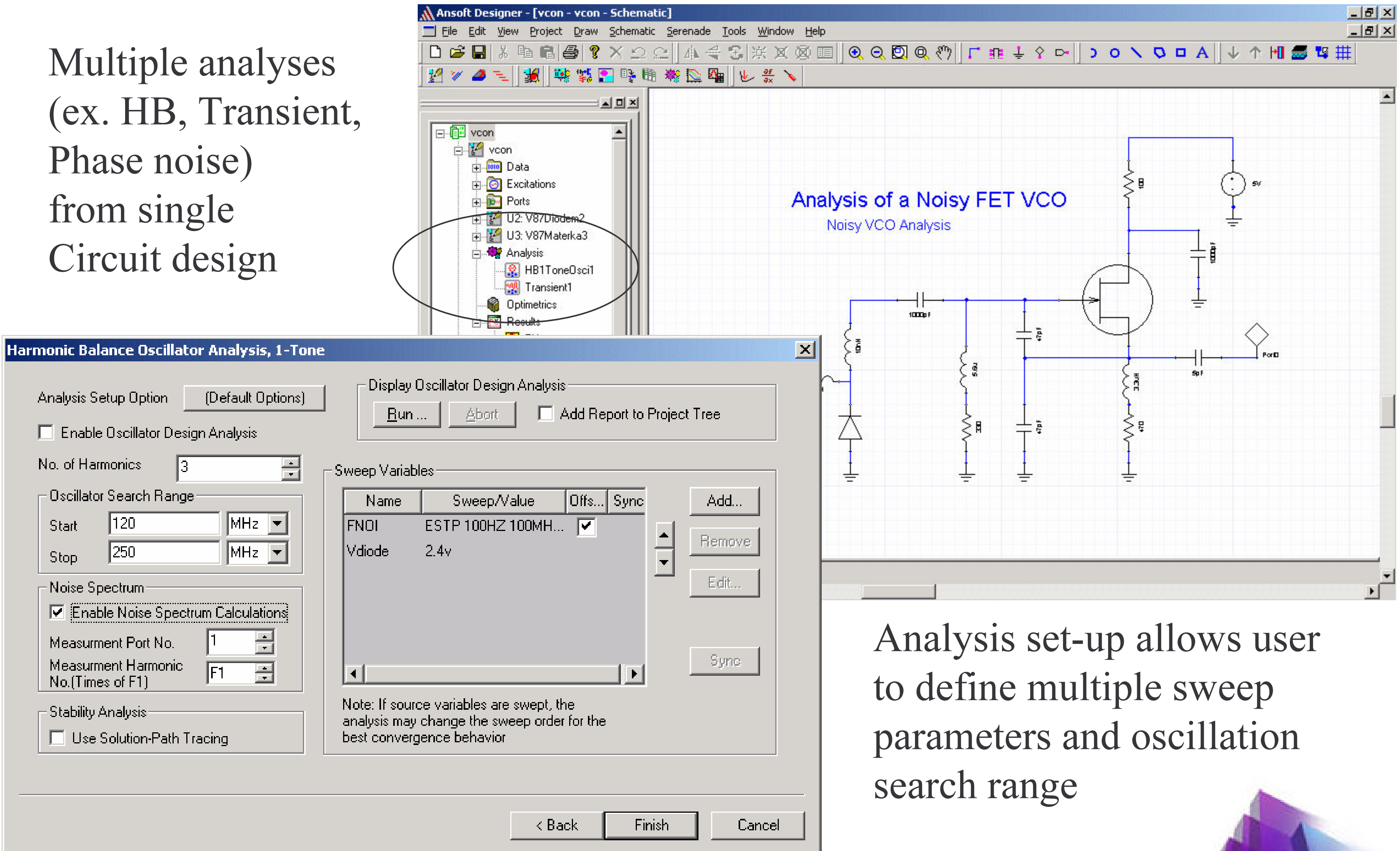
1. Right mouse click in the report field to bring up menu, select "Contour Values"

2. Set new min, max and step values

Oscillator Analysis

Oscillator Analysis

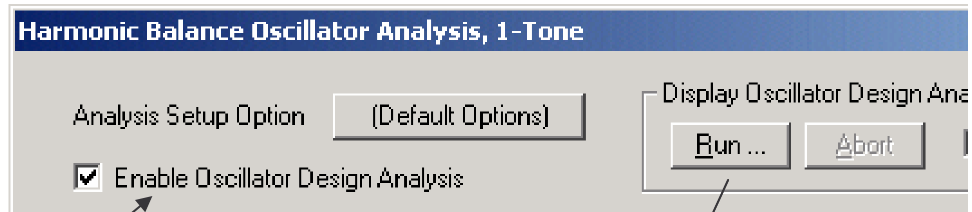
Multiple analyses (ex. HB, Transient, Phase noise) from single Circuit design

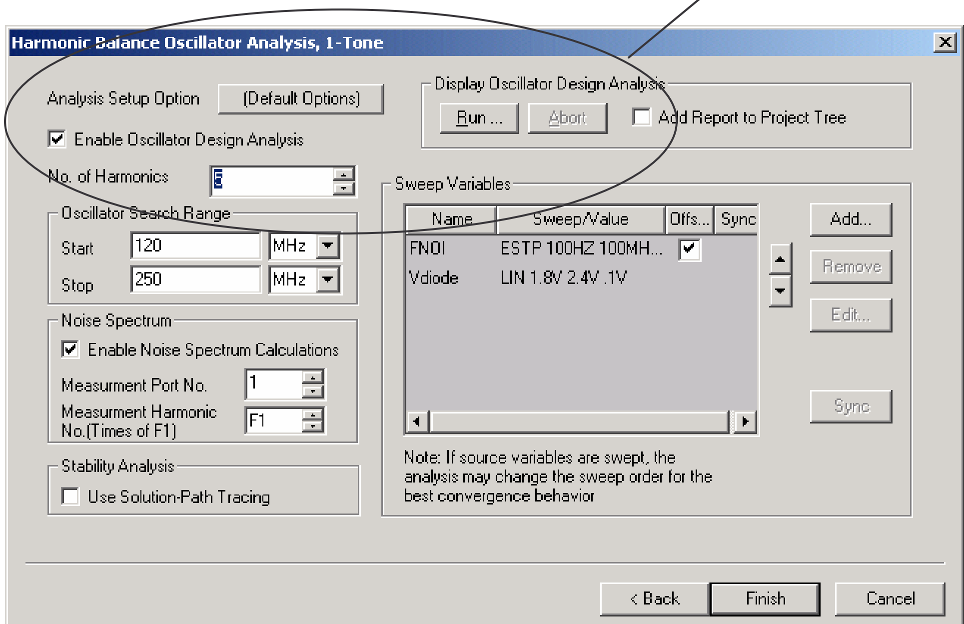


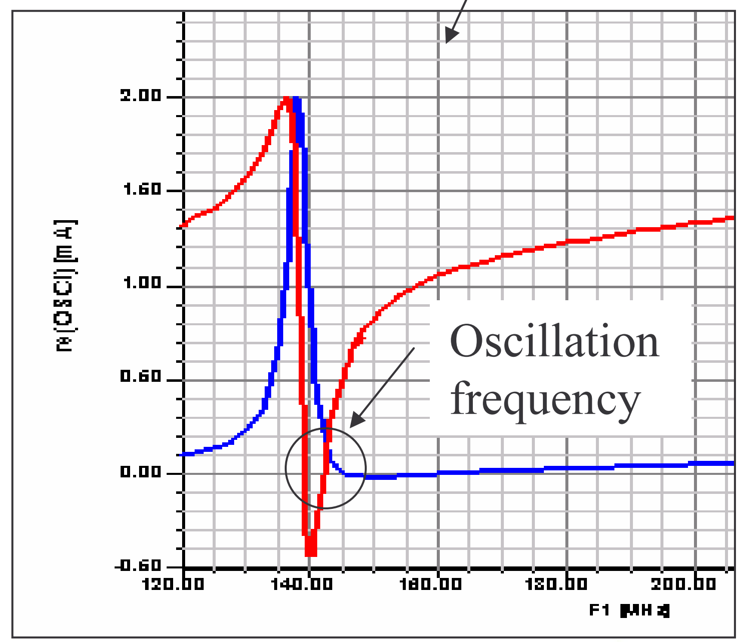
Analysis set-up allows user to define multiple sweep parameters and oscillation search range

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.



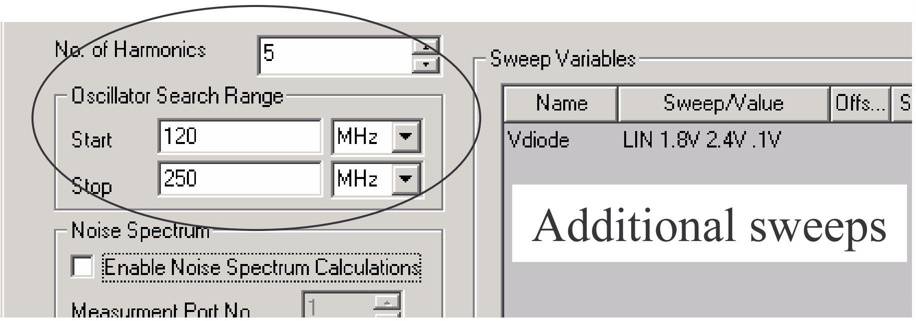


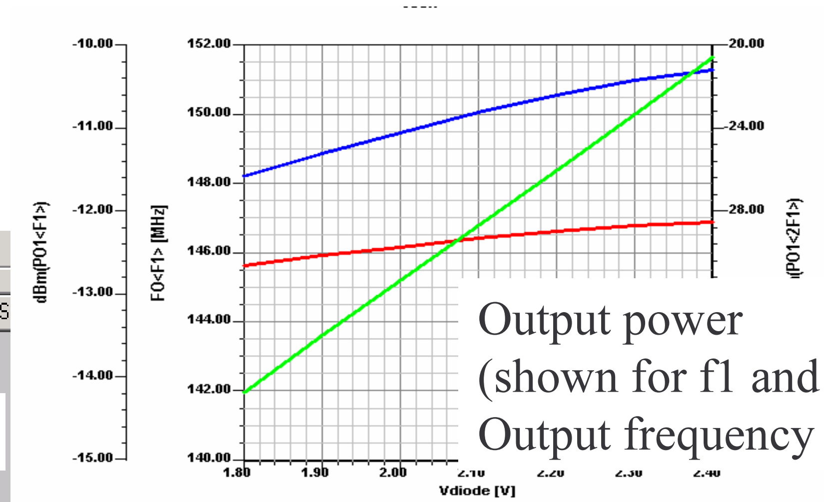


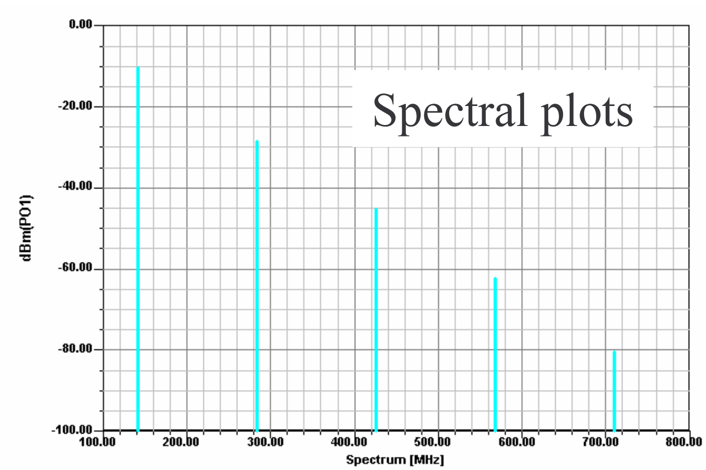
Oscillator Analysis

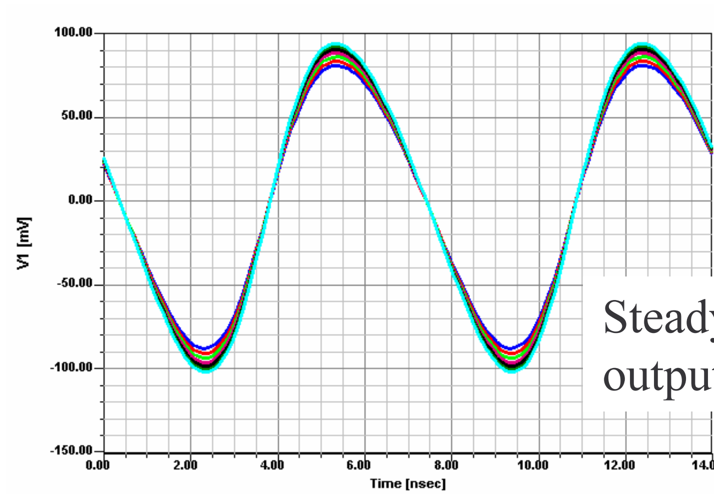
Harmonic Balance Oscillator analysis provides nonlinear circuit performance

Search range and harmonics



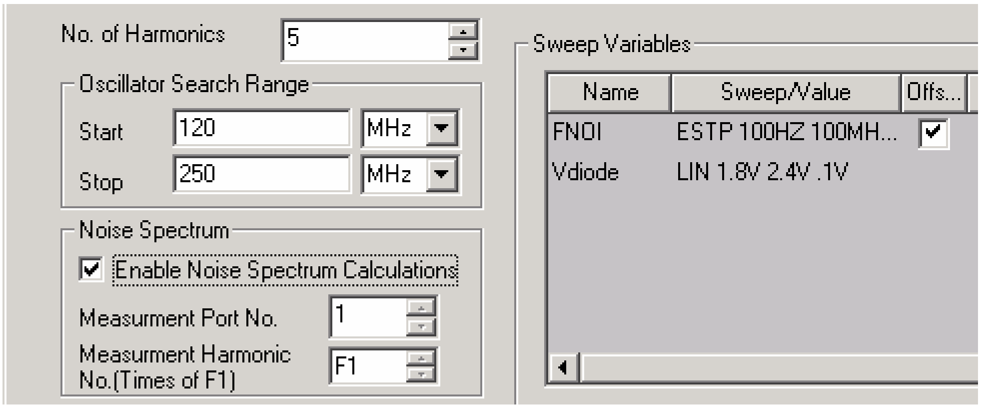




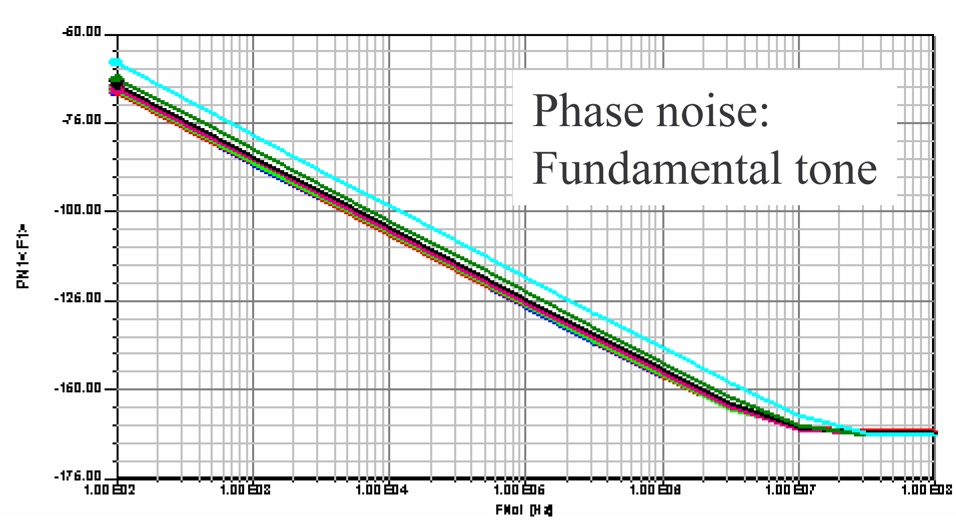


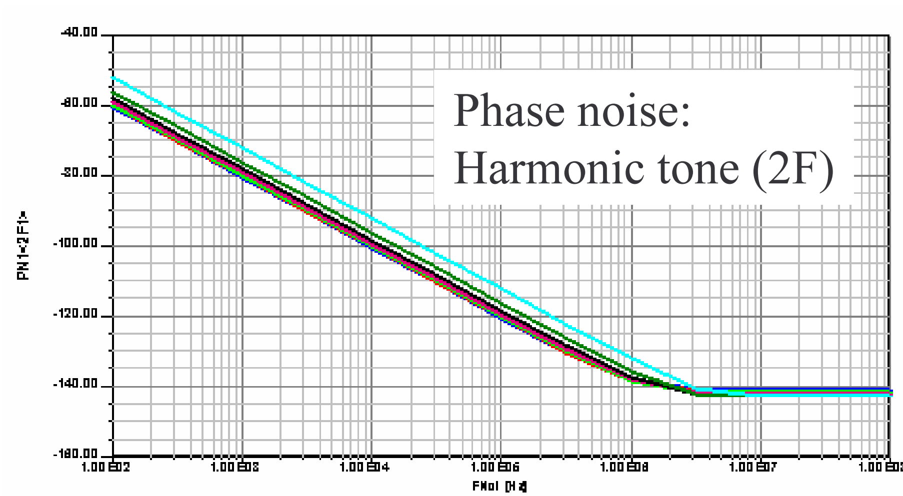
ANSOFT CORPORATION

Nonlinear Noise Analysis

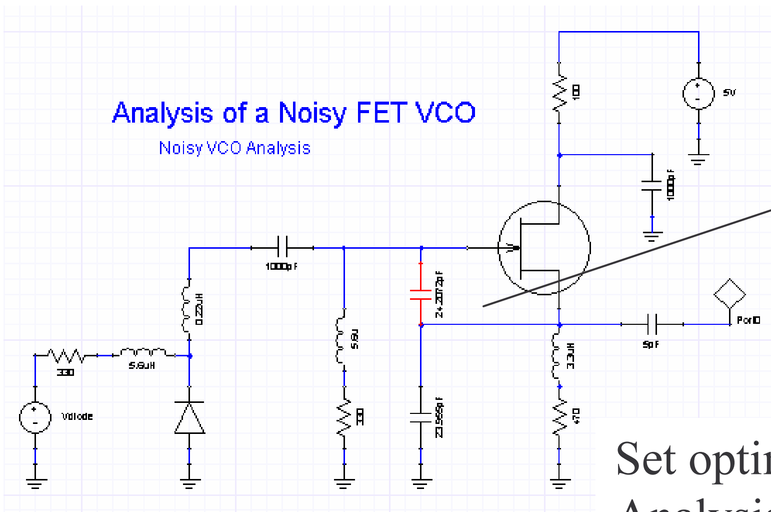


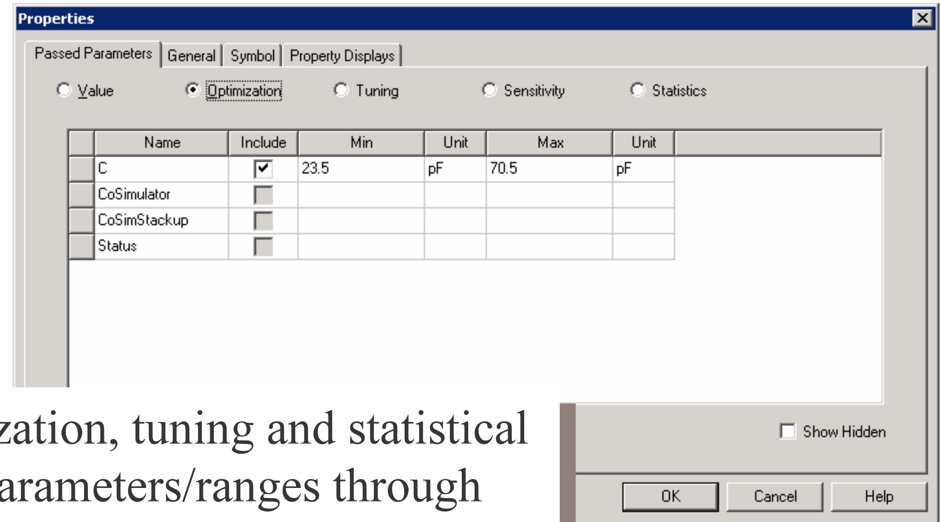
Noise Spectrum analysis provides nonlinear circuit noise spectrum data such as phase noise and amplitude noise





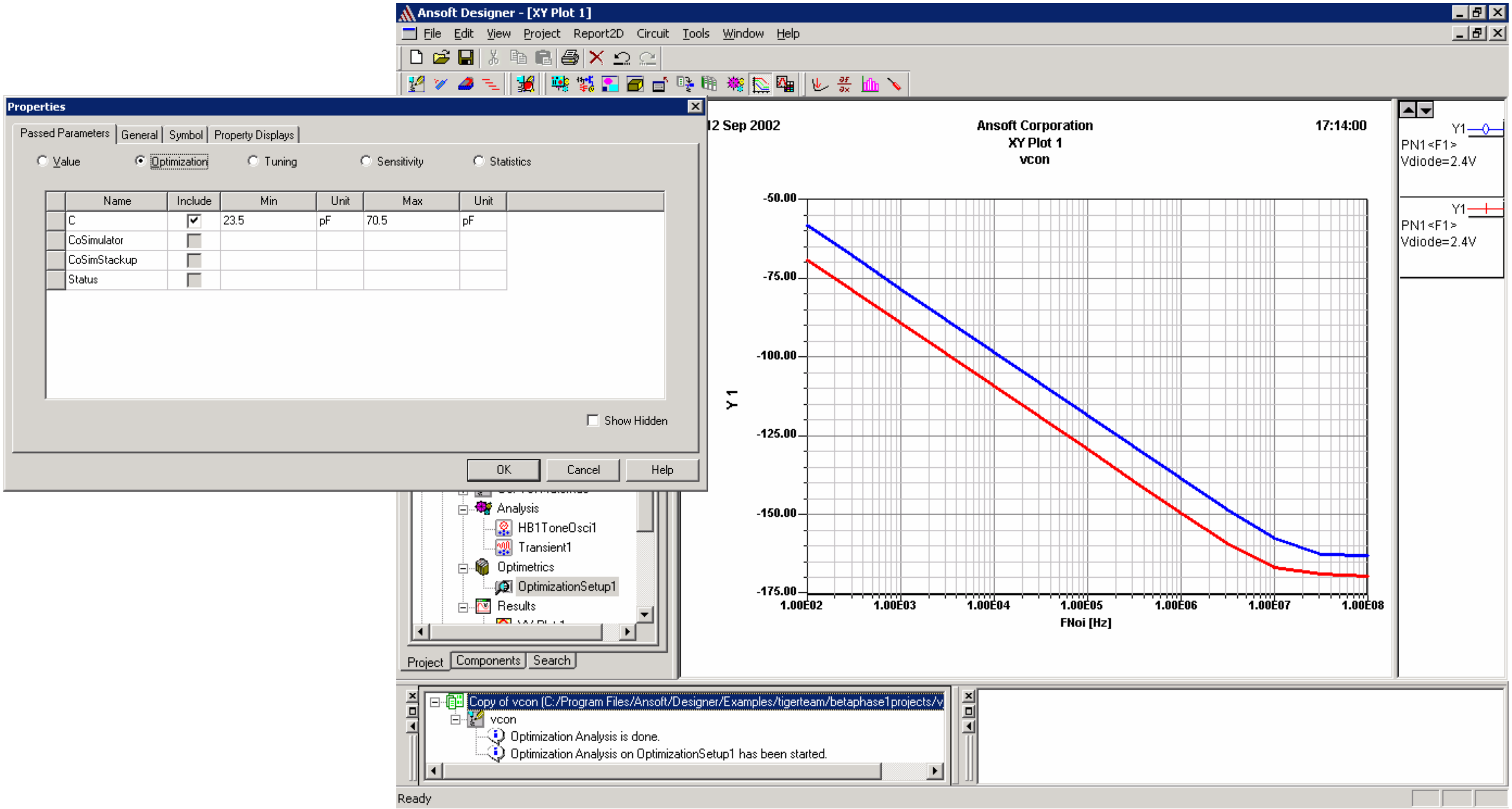
Optimization of Noise Performance



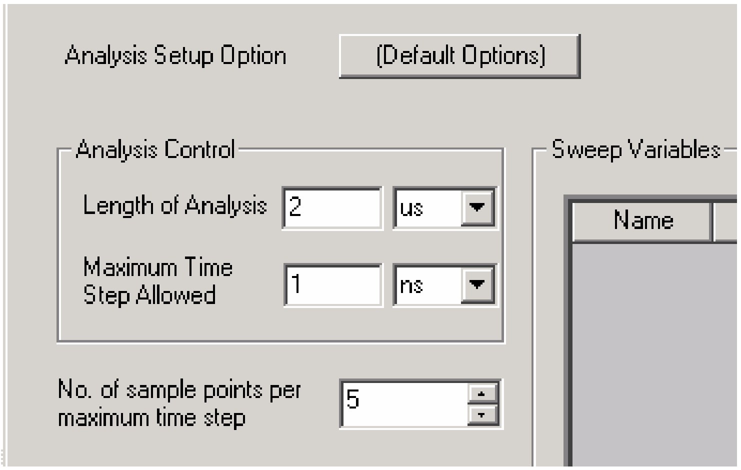


Set optimization, tuning and statistical Analysis parameters/ranges through component properties

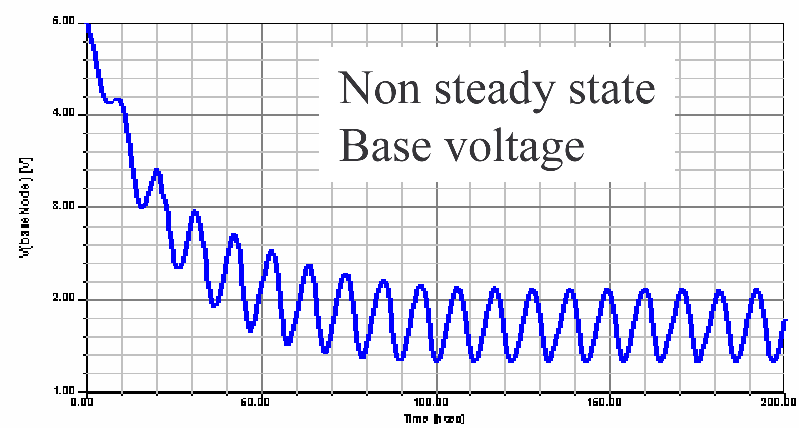
Optimization of Noise Performance

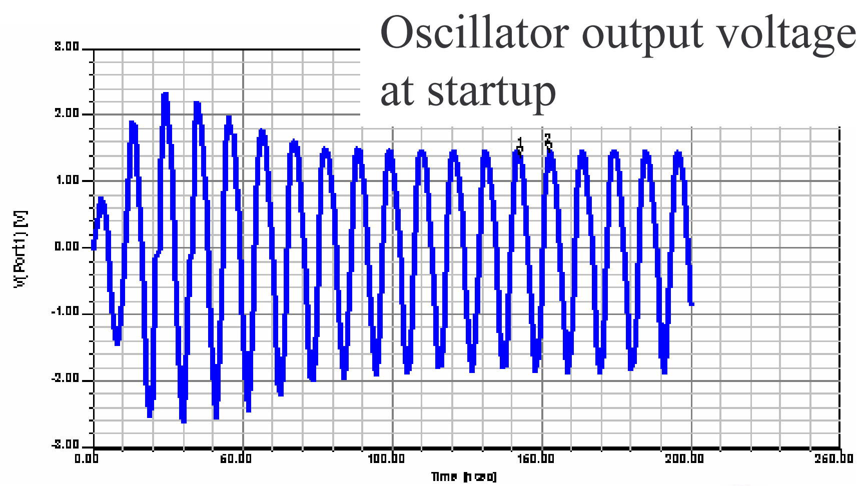


Transient Analysis



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

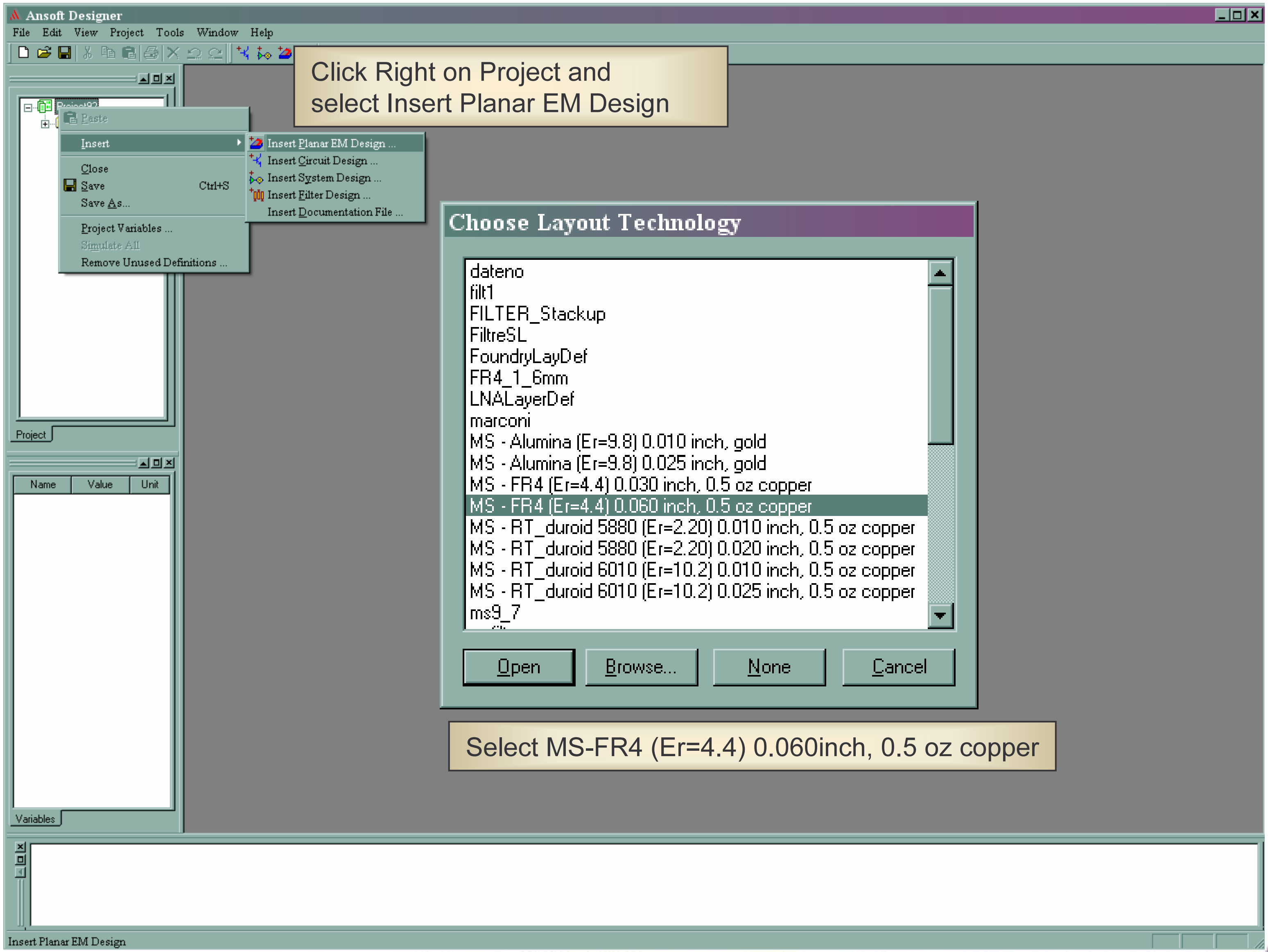




Field Solver Design Basics
(.avi File)

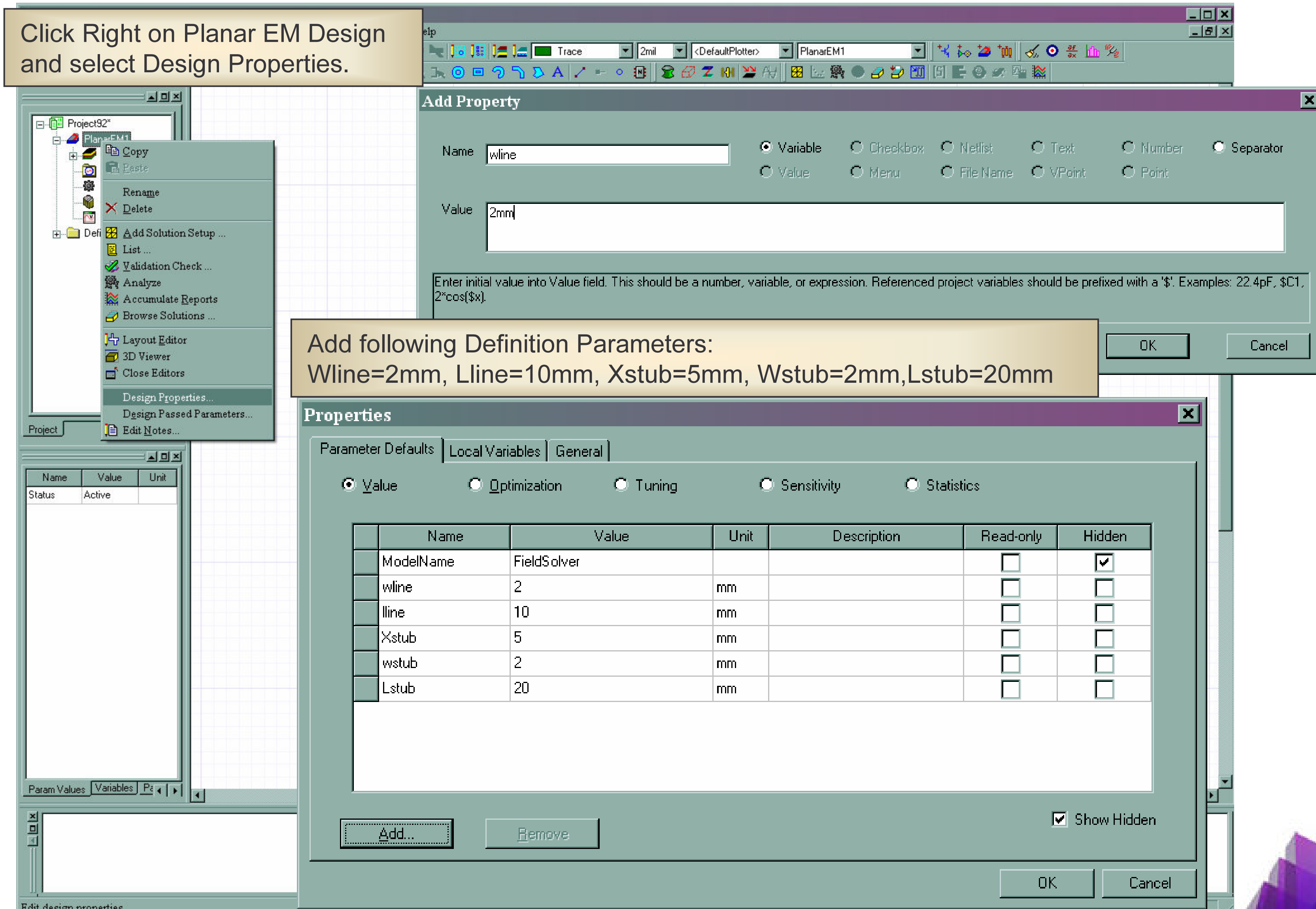
Exercise:
Create a Planar EM Design

Insert Planar EM Design



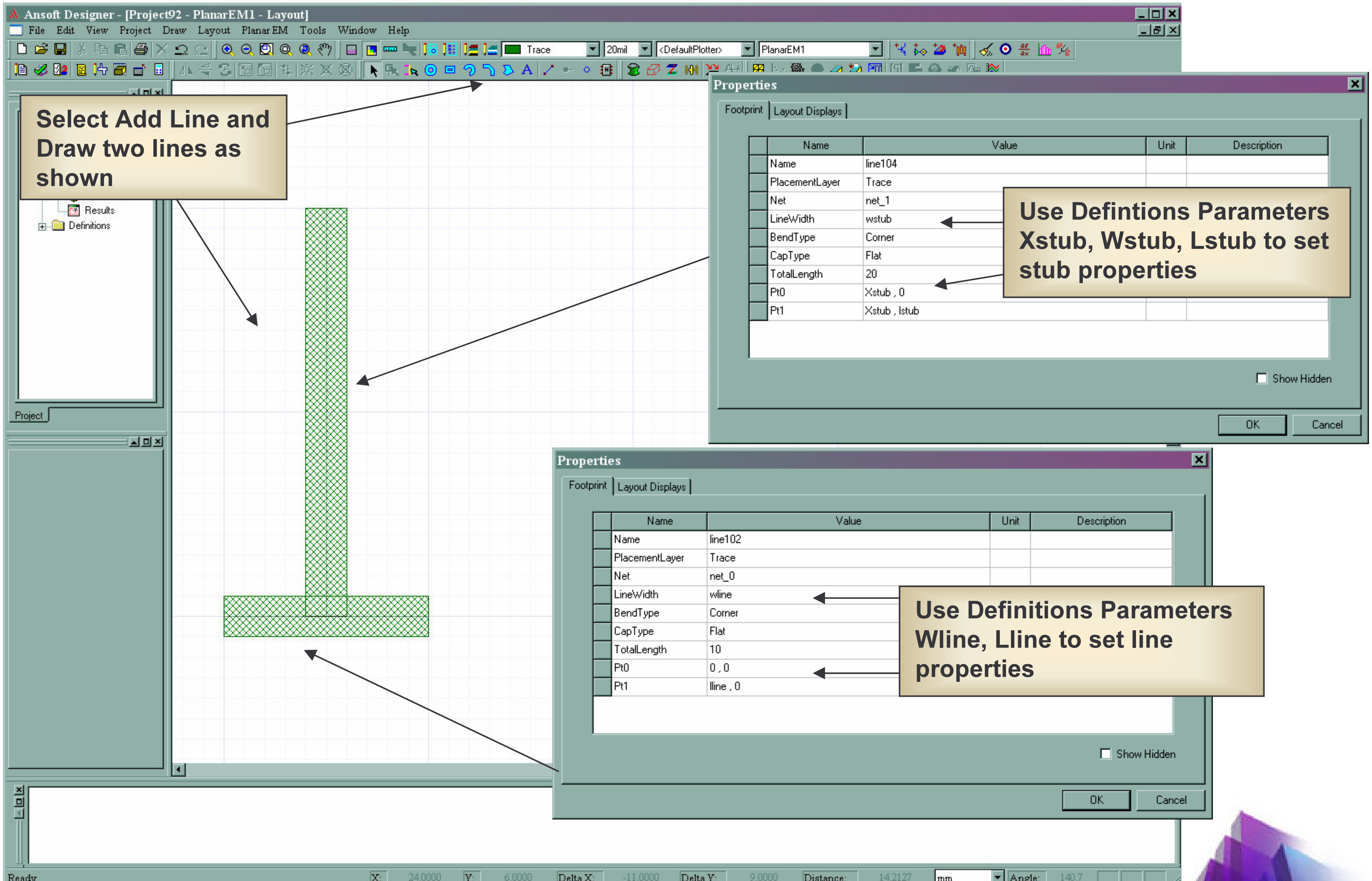
Add Definitions Parameters

Click Right on Planar EM Design and select Design Properties.

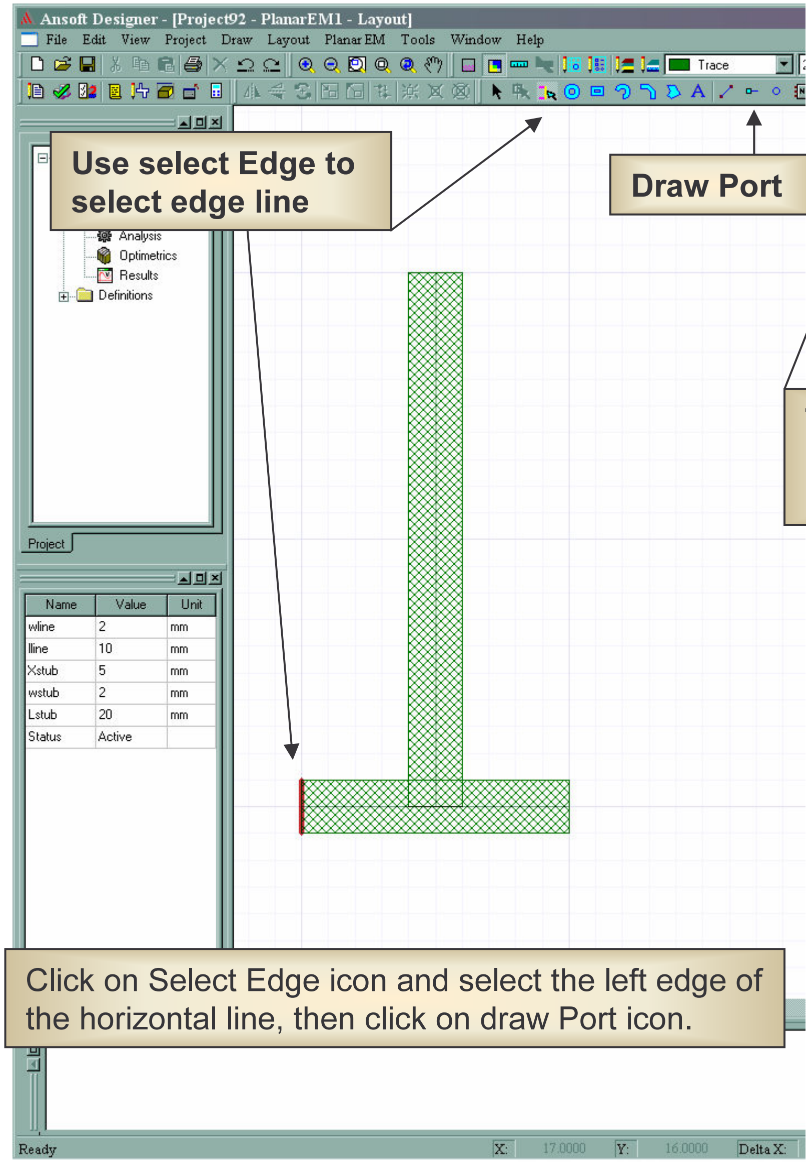


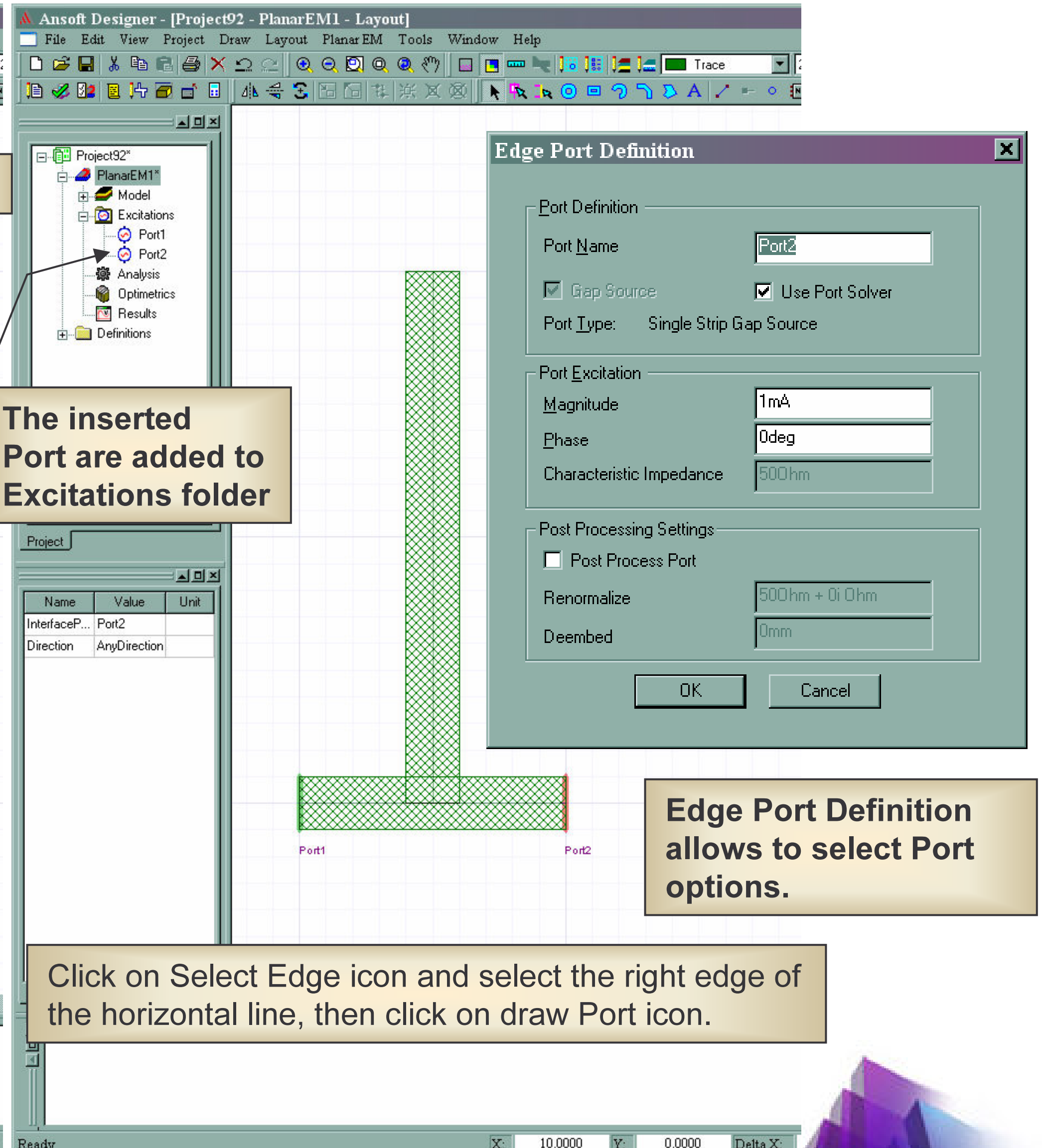
Add following Definition Parameters:
Wline=2mm, Lline=10mm, Xstub=5mm, Wstub=2mm,Lstub=20mm

Draw A Simple Open Stub

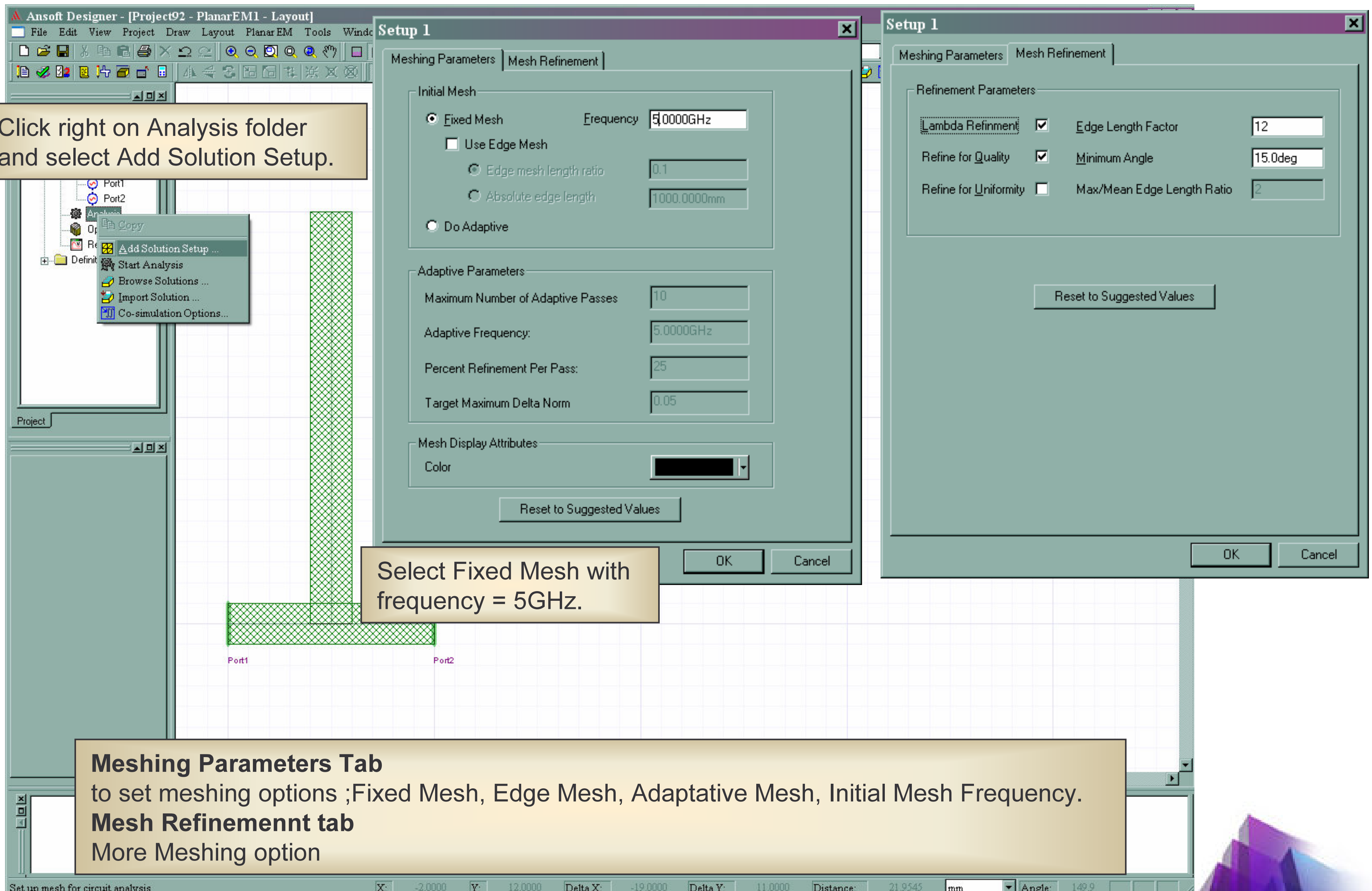


Add Ports


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


The inserted Port are added to Excitations folder
Edge Port Definition
Port Definition
Port Name
Port2
Gap Source
Use Port Solver
Port Type:
Single Strip Gap Source
Port Excitation
Magnitude
1mA
Phase
0deg
Characteristic Impedance
50Ohm
Post Processing Settings
Post Process Port
Renormalize
50Ohm + 0i Ohm
Deembed
0mm
OK
Cancel
Edge Port Definition allows to select Port options.
Click on Select Edge icon and select the right edge of the horizontal line, then click on draw Port icon.

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

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.

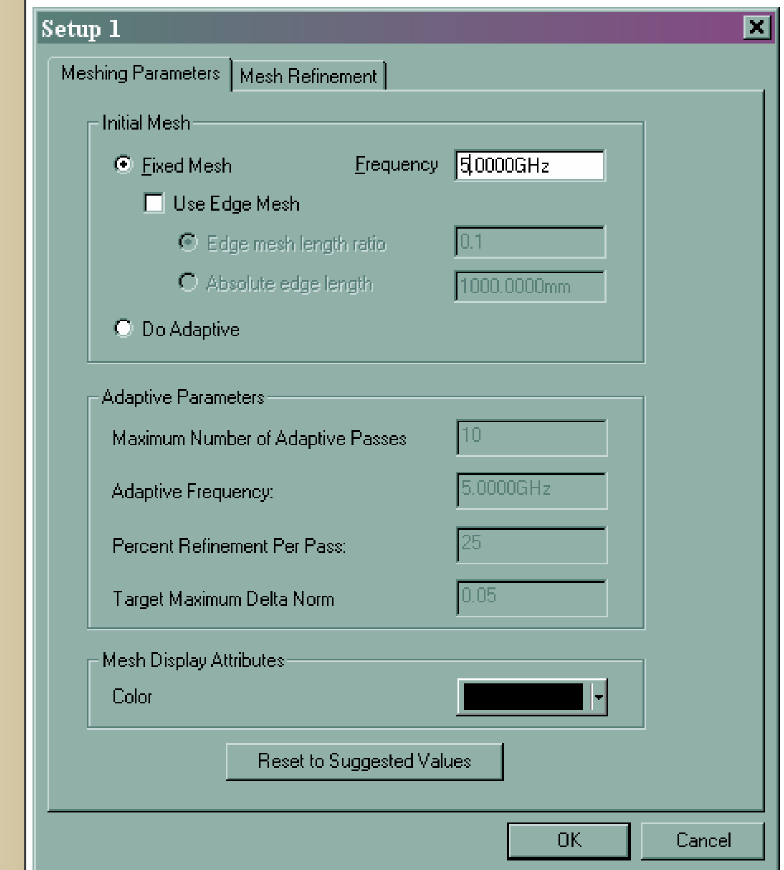
When Ansoft Designer refines the fixed mesh, it determines the length of the rectangles (the longer edges, which lie parallel to the model edge) by making them smaller than a fraction of the guided wavelength at the frequency you specified. You determine the width of the rectangles by either specifying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

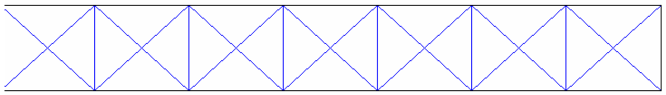
1. In the setup dialog box, click the **Meshing Parameters** tab.
2. Select **Fixed Mesh**, and then specify a frequency at which to generate the fixed mesh in the **Frequency** text box.
3. Select **Use Edge Mesh**.
4. Specify how the edge mesh is created by selecting one of the following:
   - **Edge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

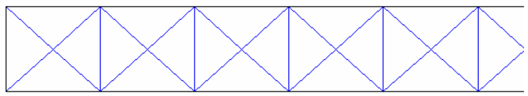
After refinement, the width of the rectangles will be nominally equal to ratio x length. The ratio value should be between 0.02 and 0.2 to prevent extremely narrow rectangles and extremely wide triangles.

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

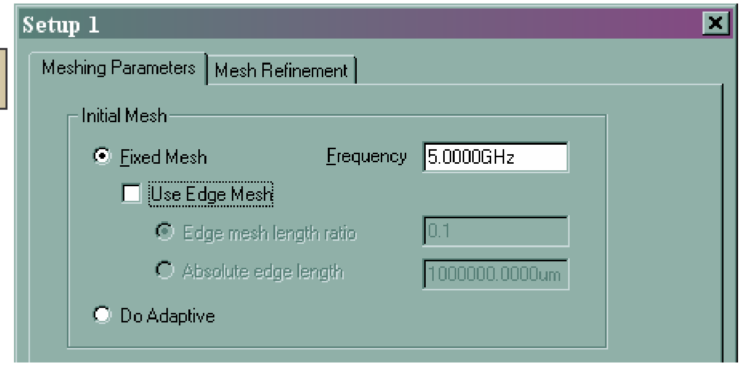


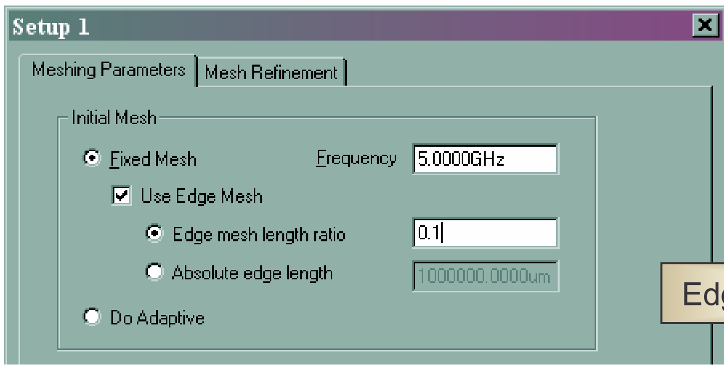
Example of Edge Meshing

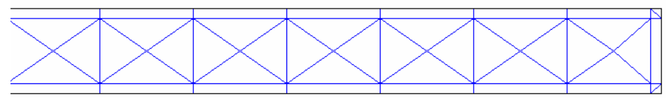


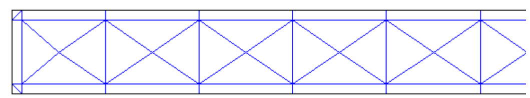


No edge mesh

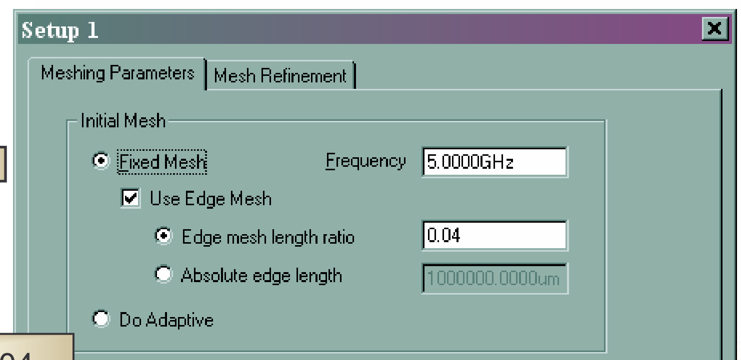


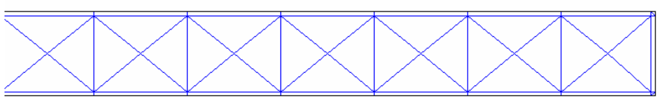


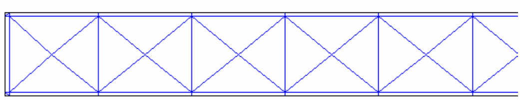




Edge mesh length ratio=0.1







Edge mesh length ratio=0.04

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:

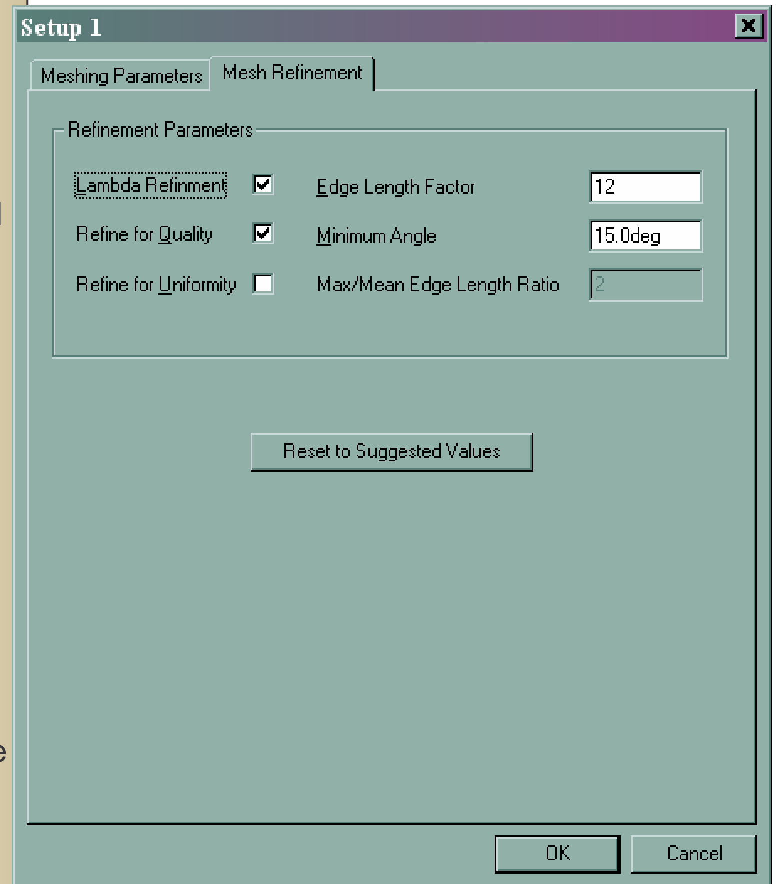
1. Under **the Mesh Refinement** tab in the solution setup dialog box, select **Refine for Quality**.
2. Type a value in the Minimum Angle text box in degrees.

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.

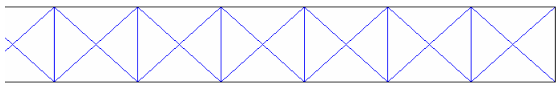
To refine for uniformity:

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.



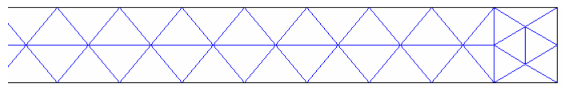
Example of Mesh refinement

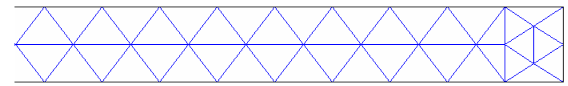
Lambda Refinement



Fixed Mesh Frequency= 5GHz
Edge Length factor = 12

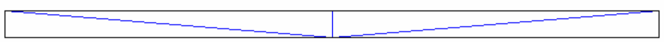
Fixed Mesh Frequency= 5GHz
Edge Length factor = 16



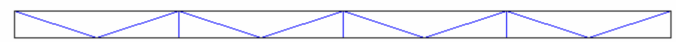


Fixed Mesh Frequency= 7.5GHz
Edge Length factor = 12

Refine for Quality

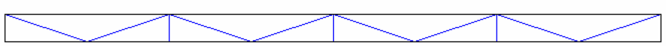


Fixed Mesh Frequency= 5GHz
No



Fixed Mesh Frequency= 5GHz
Minimum Angle =15deg

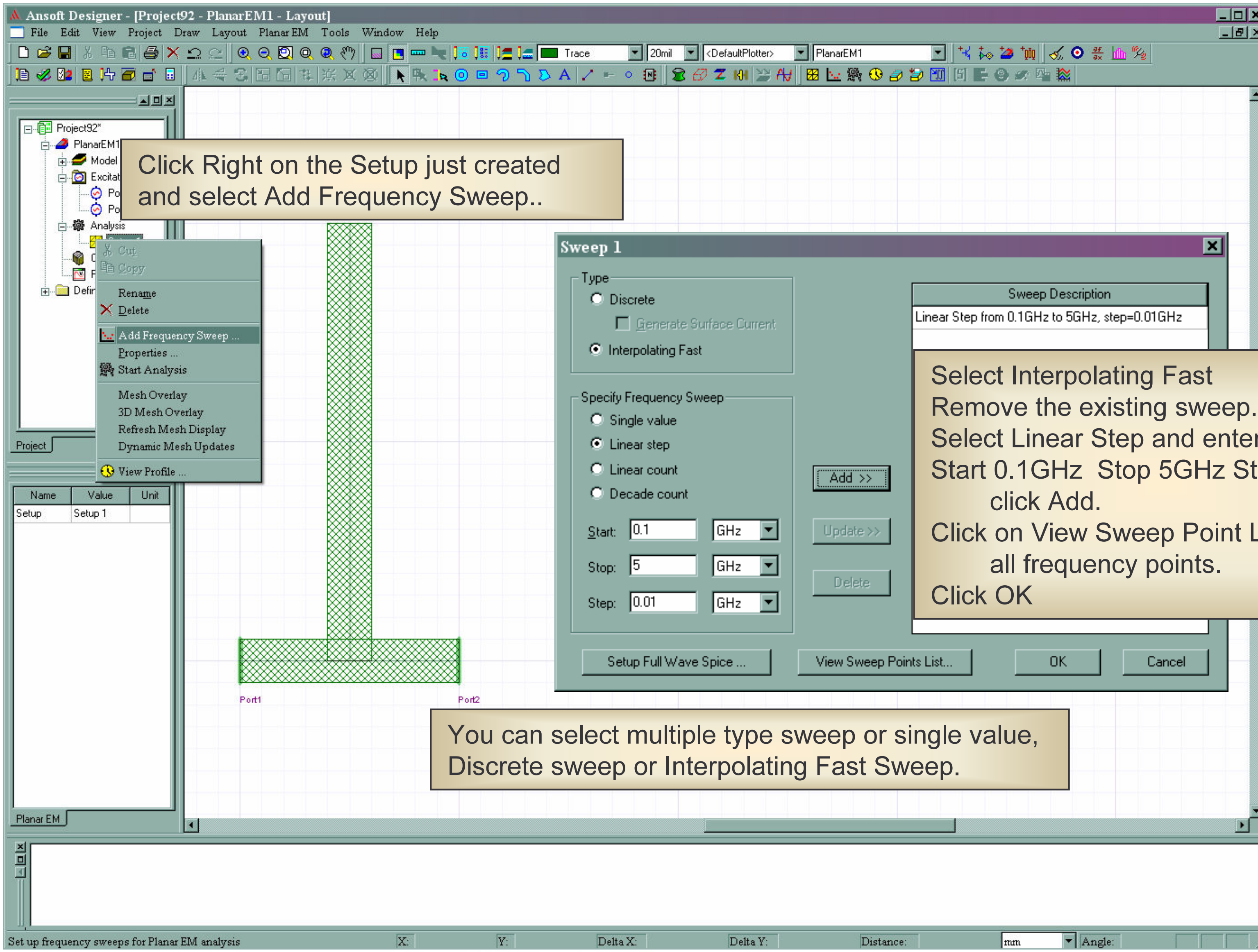
Refine for Uniformity



Fixed Mesh Frequency= 5GHz
Refine for Quality Minimum Angle =15deg
No

Fixed Mesh Frequency= 5GHz
Refine for Quality Minimum Angle =15deg
Max/Min Edge Length Ratio= 1

Add Frequency Sweep

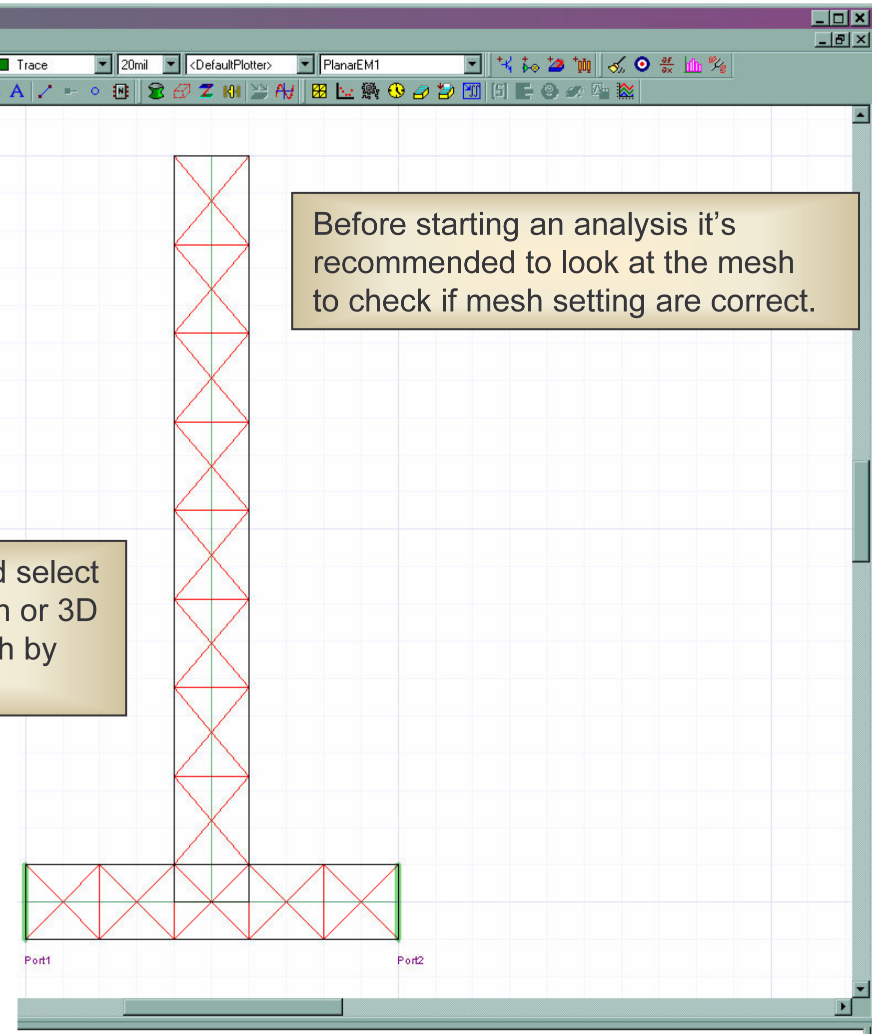


Click Right on the Setup just created and select Add Frequency Sweep..

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

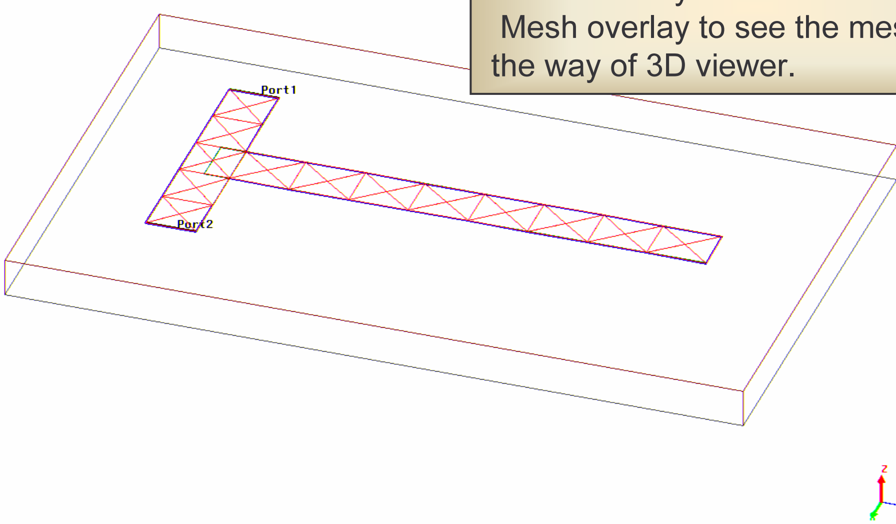
You can select multiple type sweep or single value, Discrete sweep or Interpolating Fast Sweep.

View Mesh

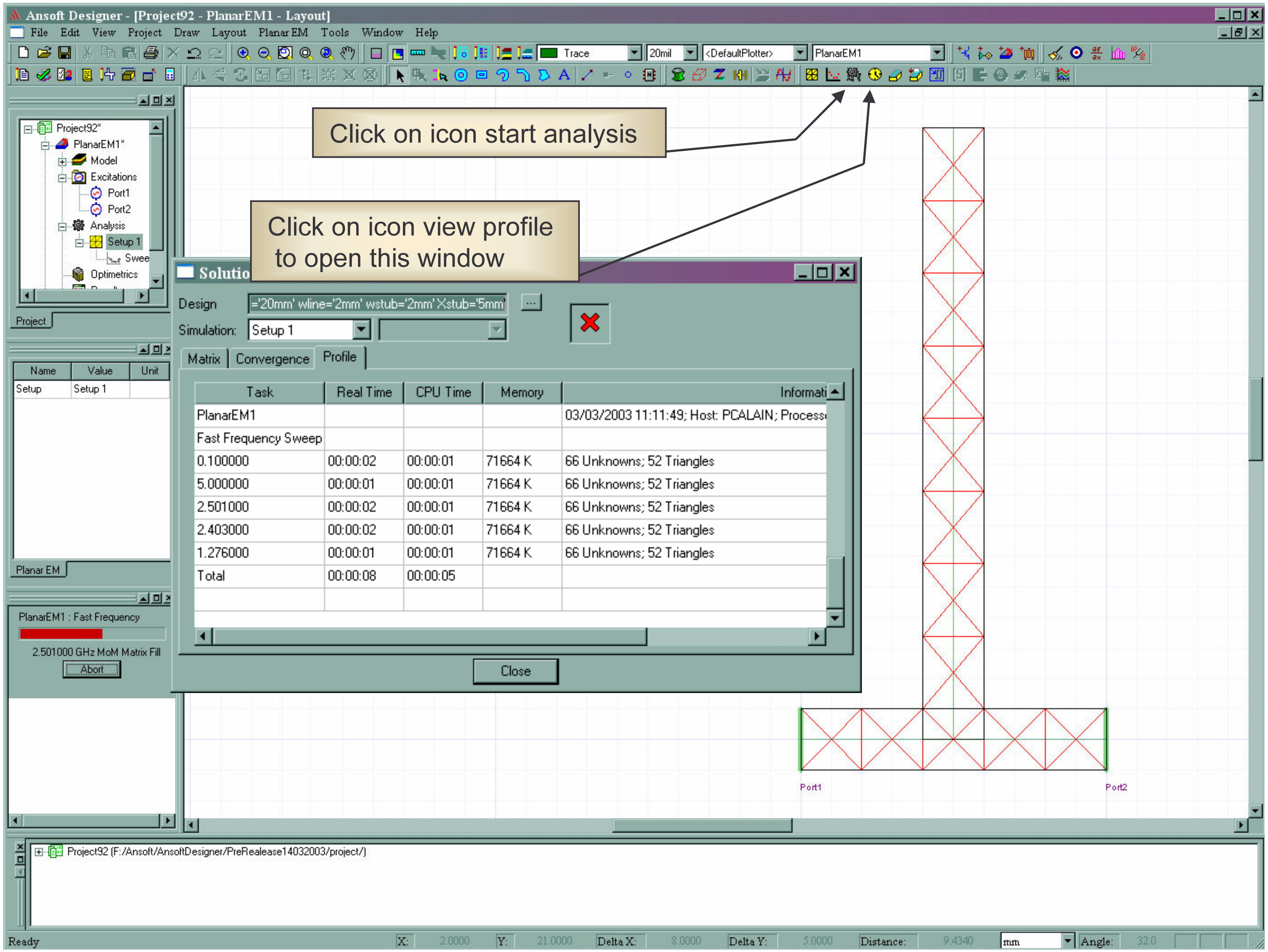


Before starting an analysis it's recommended to look at the mesh to check if mesh setting are correct.

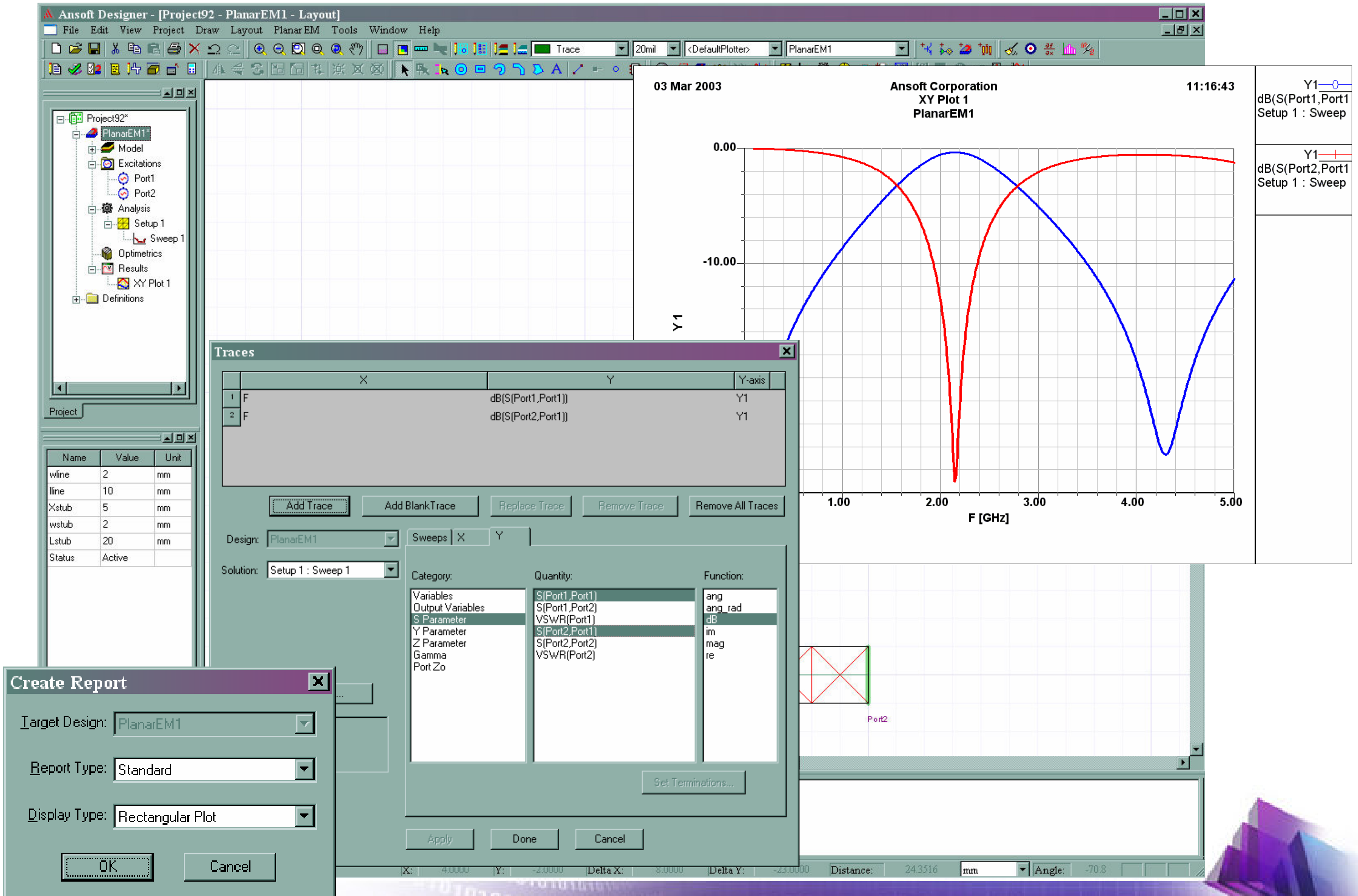
Click right on Setup folder and select Mesh overlay to see 2D mesh or 3D Mesh overlay to see the mesh by the way of 3D viewer.



Run Analysis



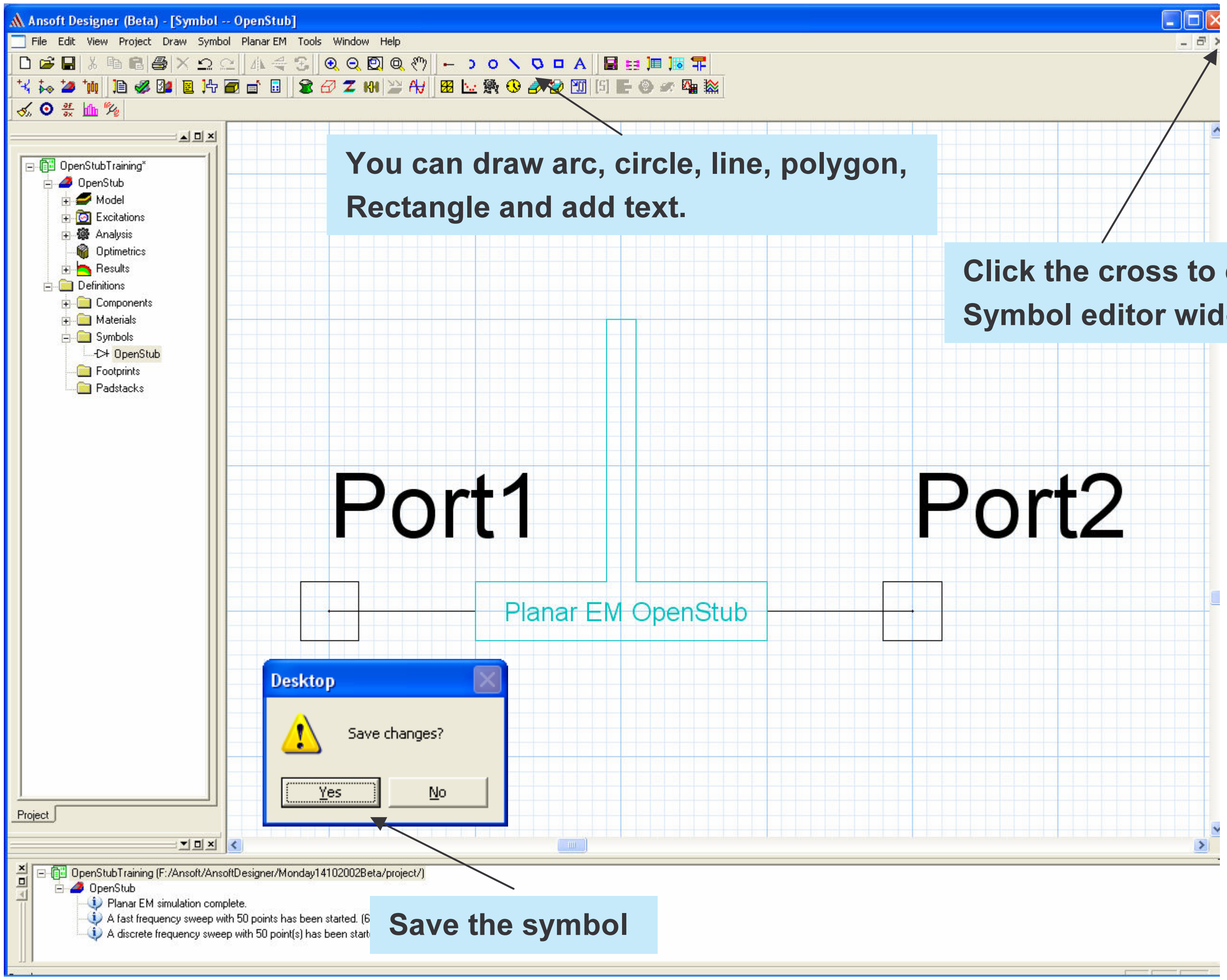
Create results: DB S11&S21



ANSOFT CORPORATION

Optional Exercise:
Planar EM Co-Simulation

Edit The Symbol and Modify It

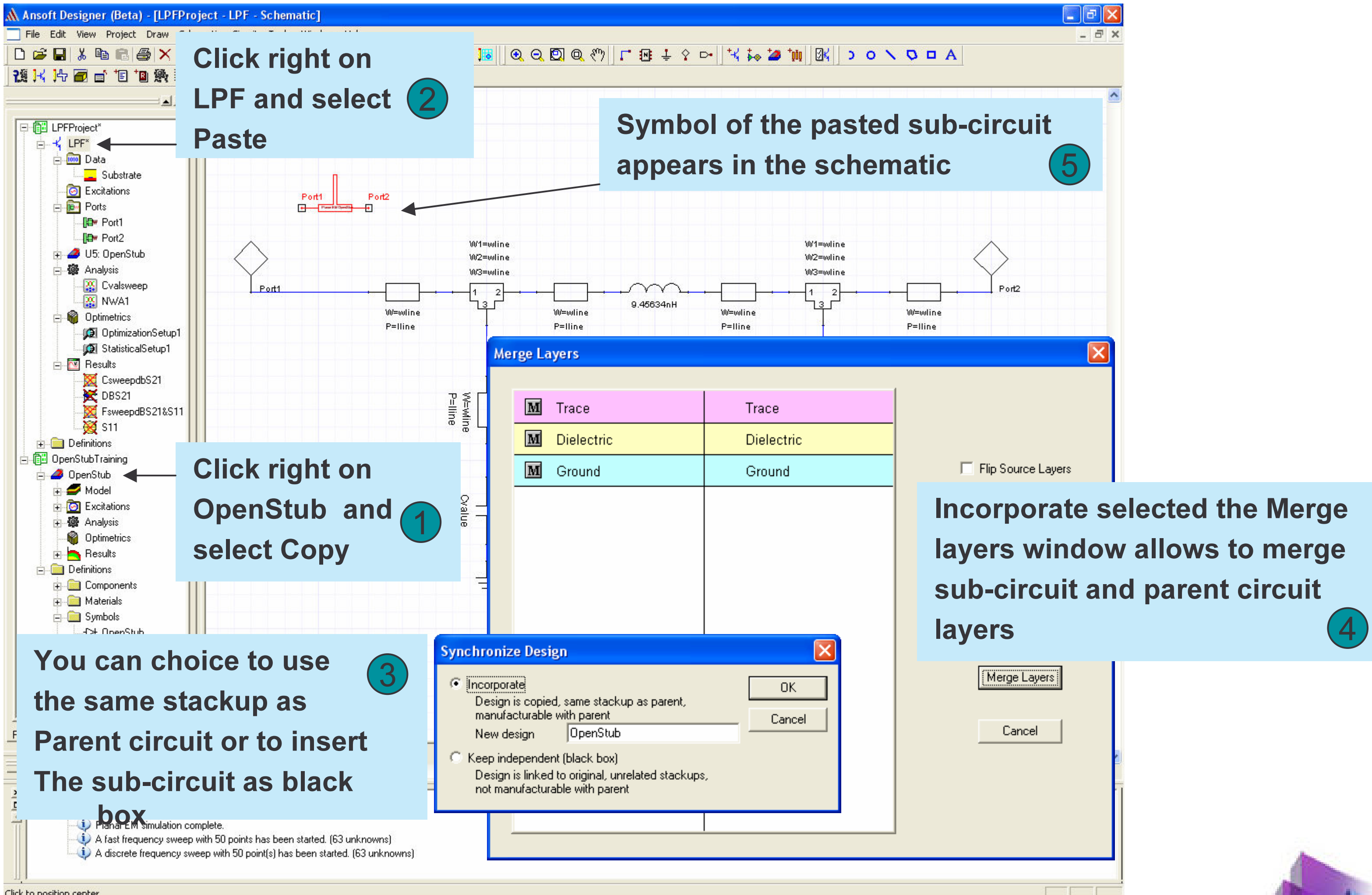


You can draw arc, circle, line, polygon, Rectangle and add text.

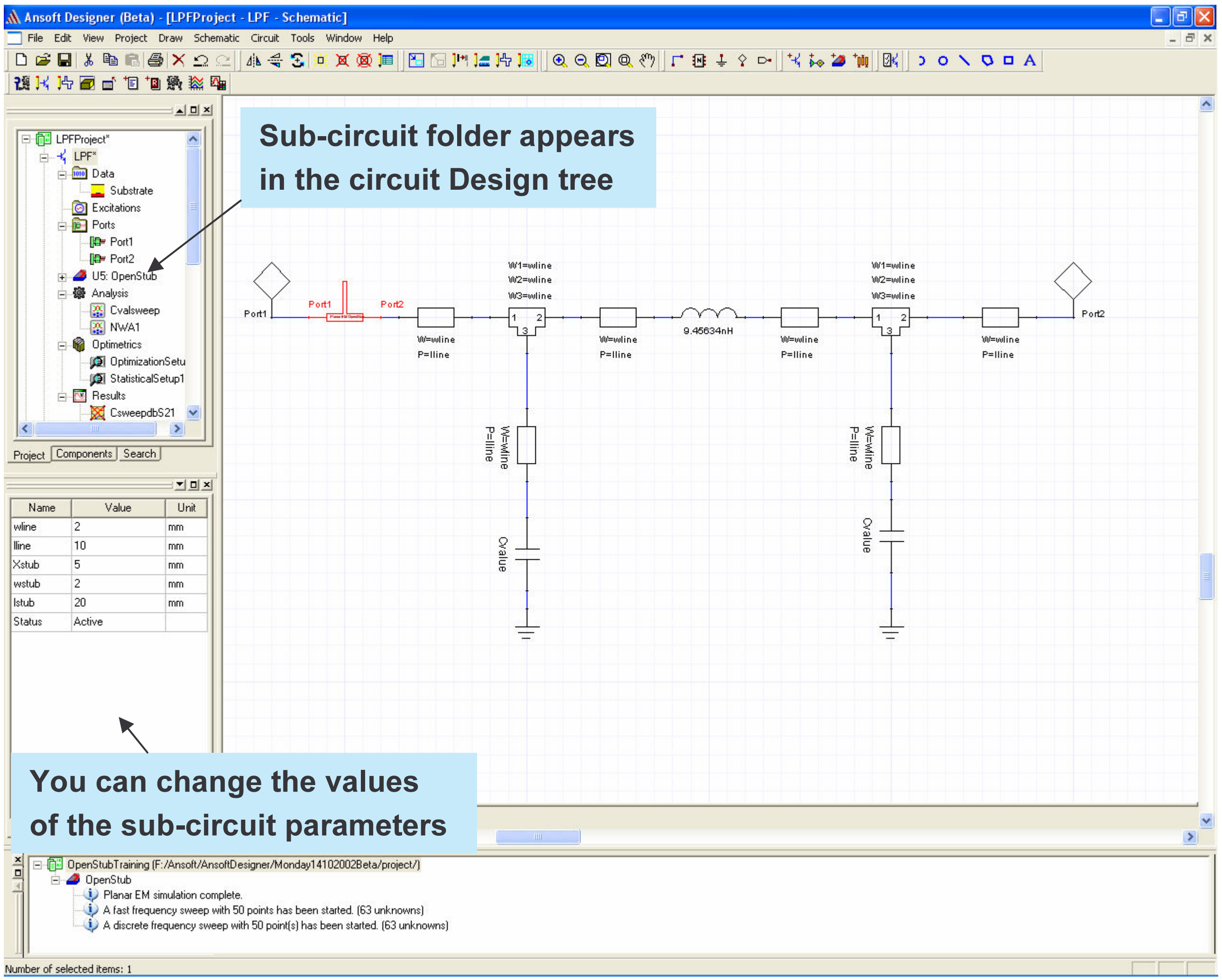
Click the cross to close the Symbol editor widow

Save the symbol

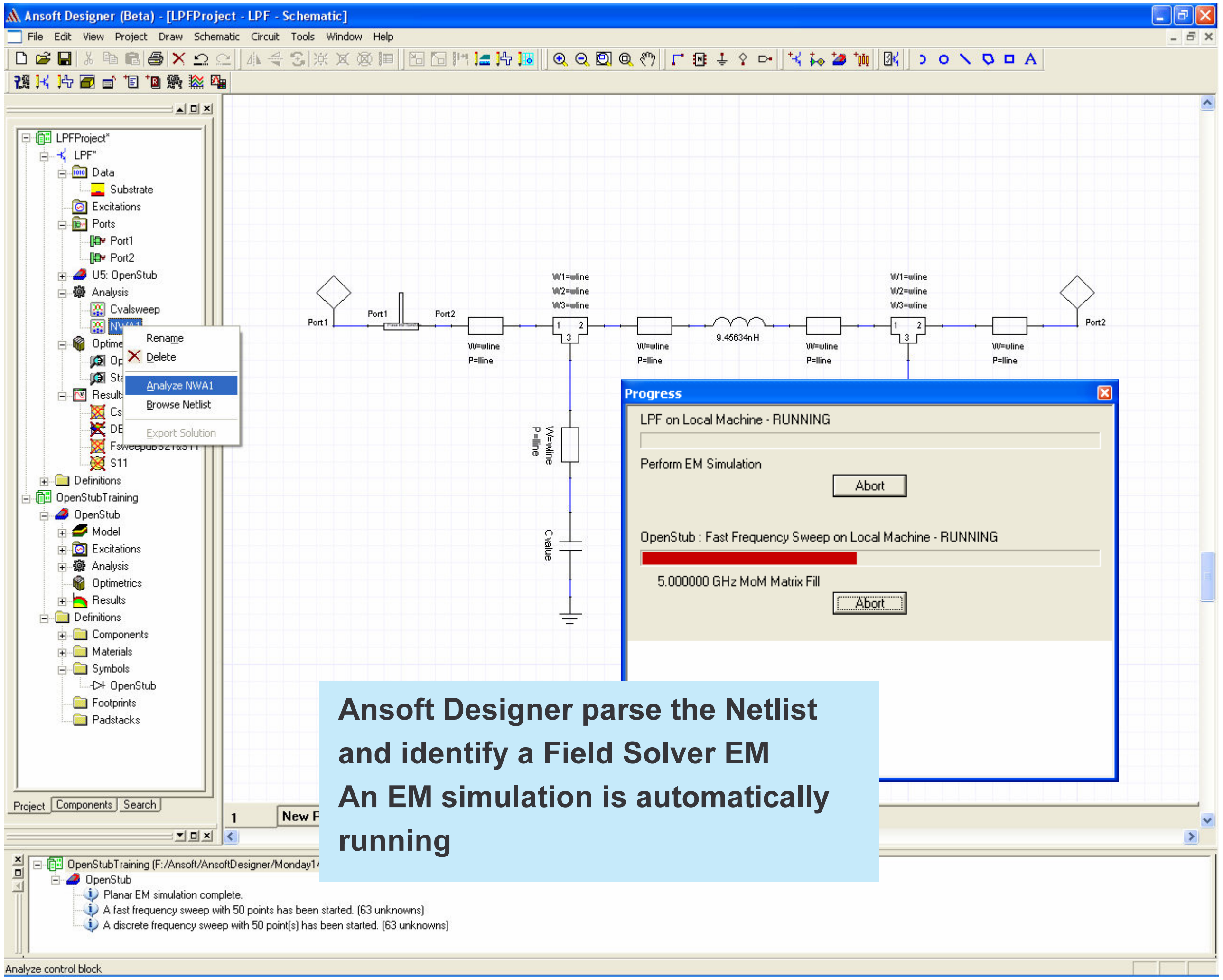
Copy and Paste the Circuit Design



Connect the Sub-Circuit Symbol

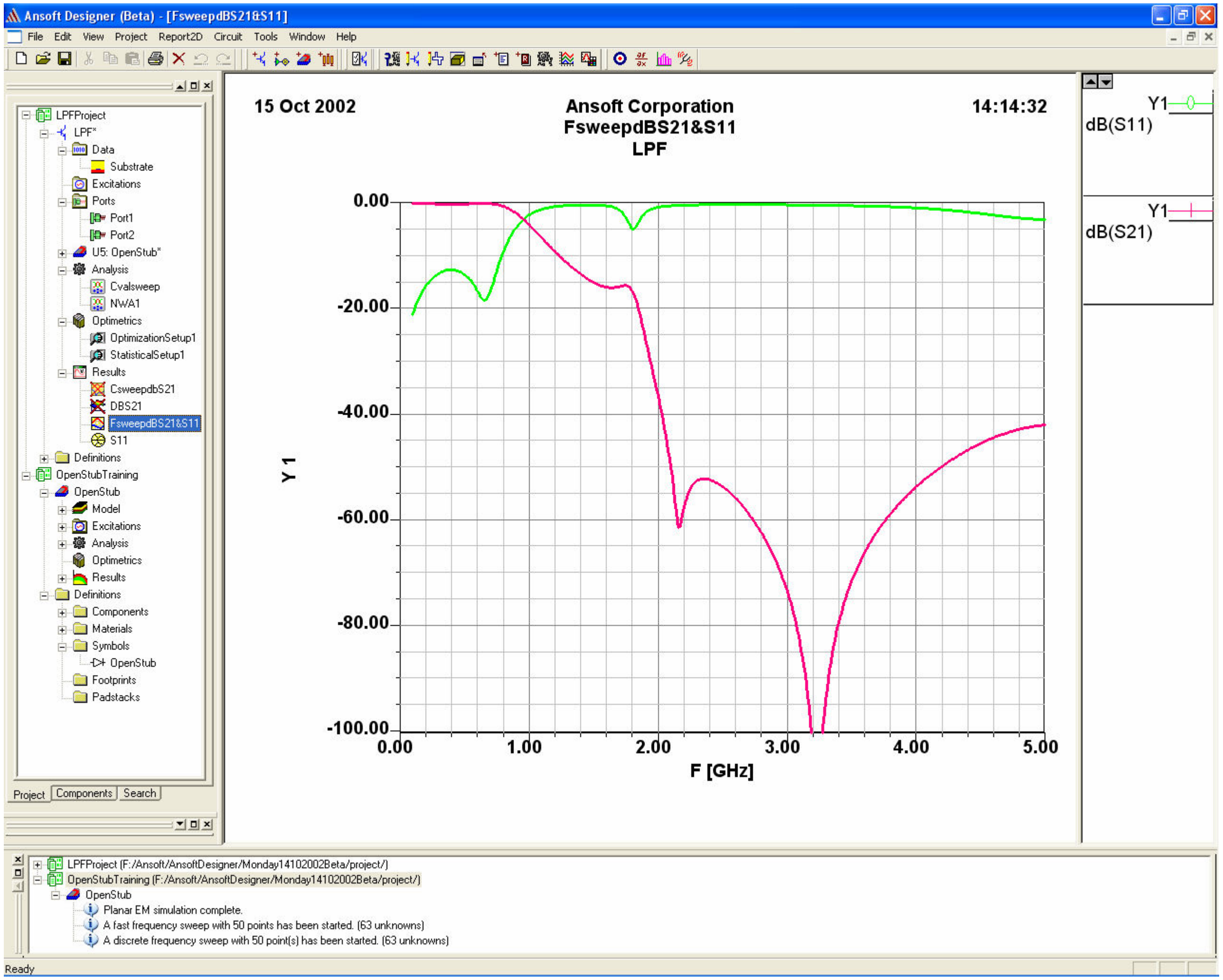


Run Analysis

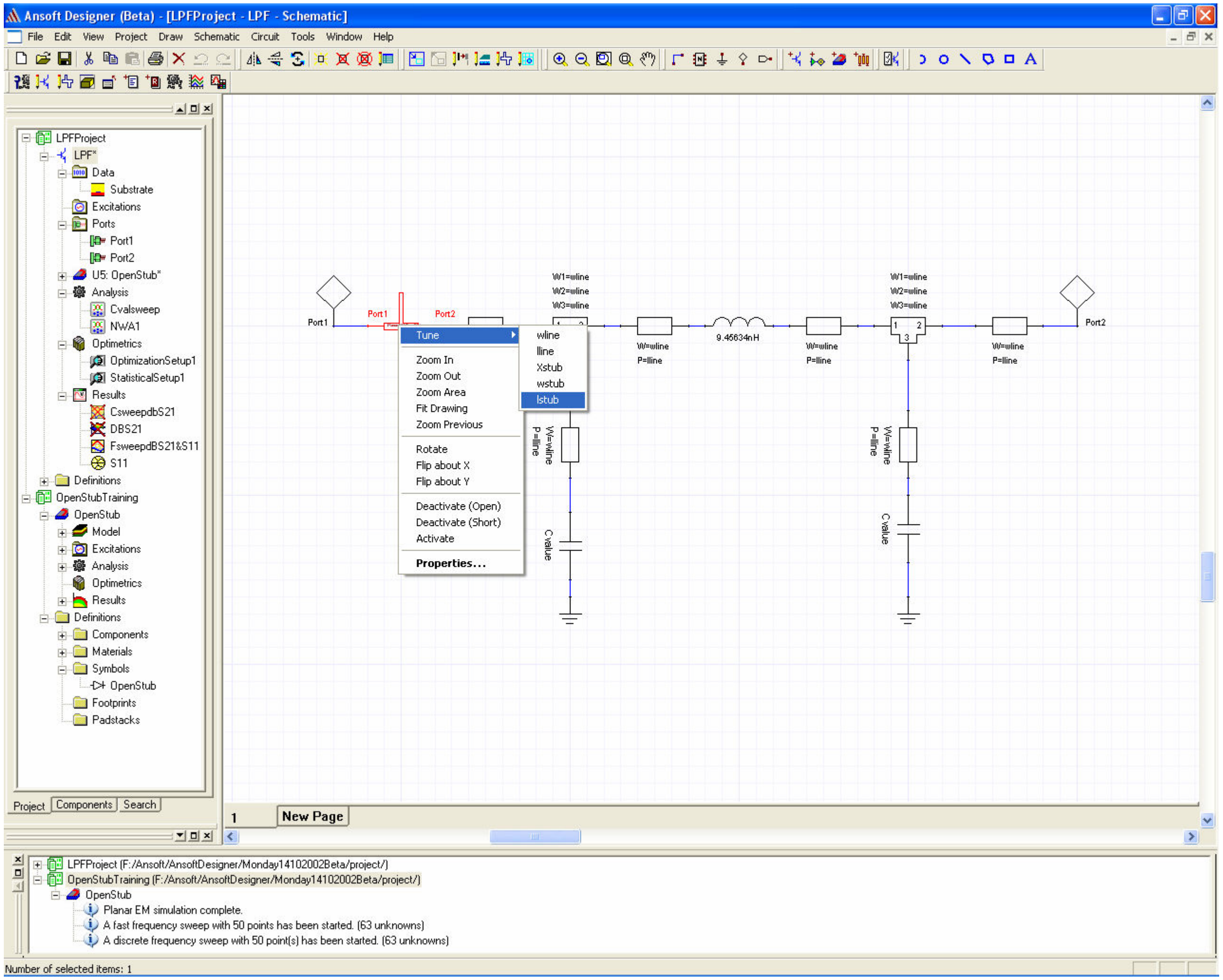


Ansoft Designer parse the Netlist and identify a Field Solver EM
An EM simulation is automatically running

Create Results: DBS21&S11

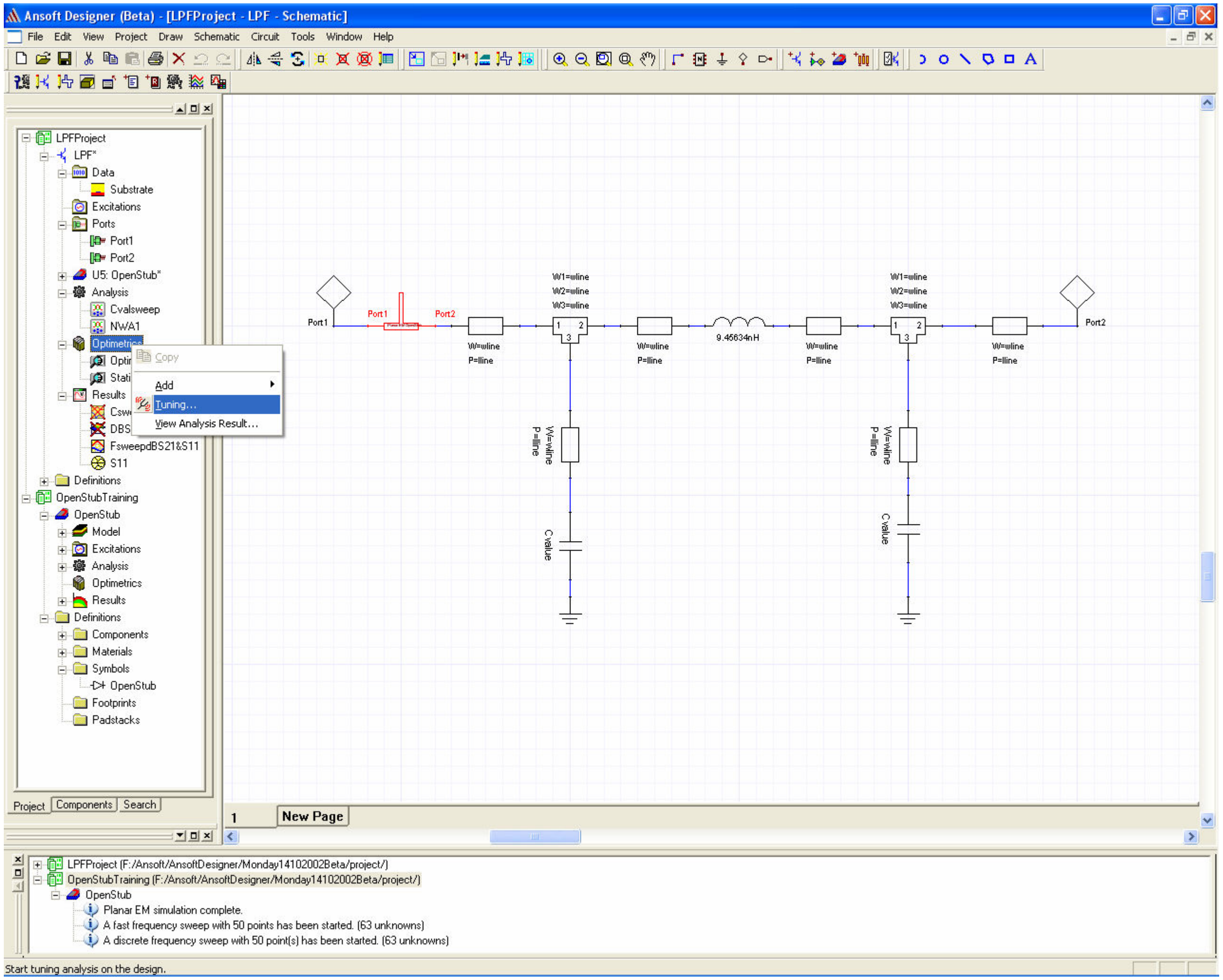


Define Lstub as Tunable

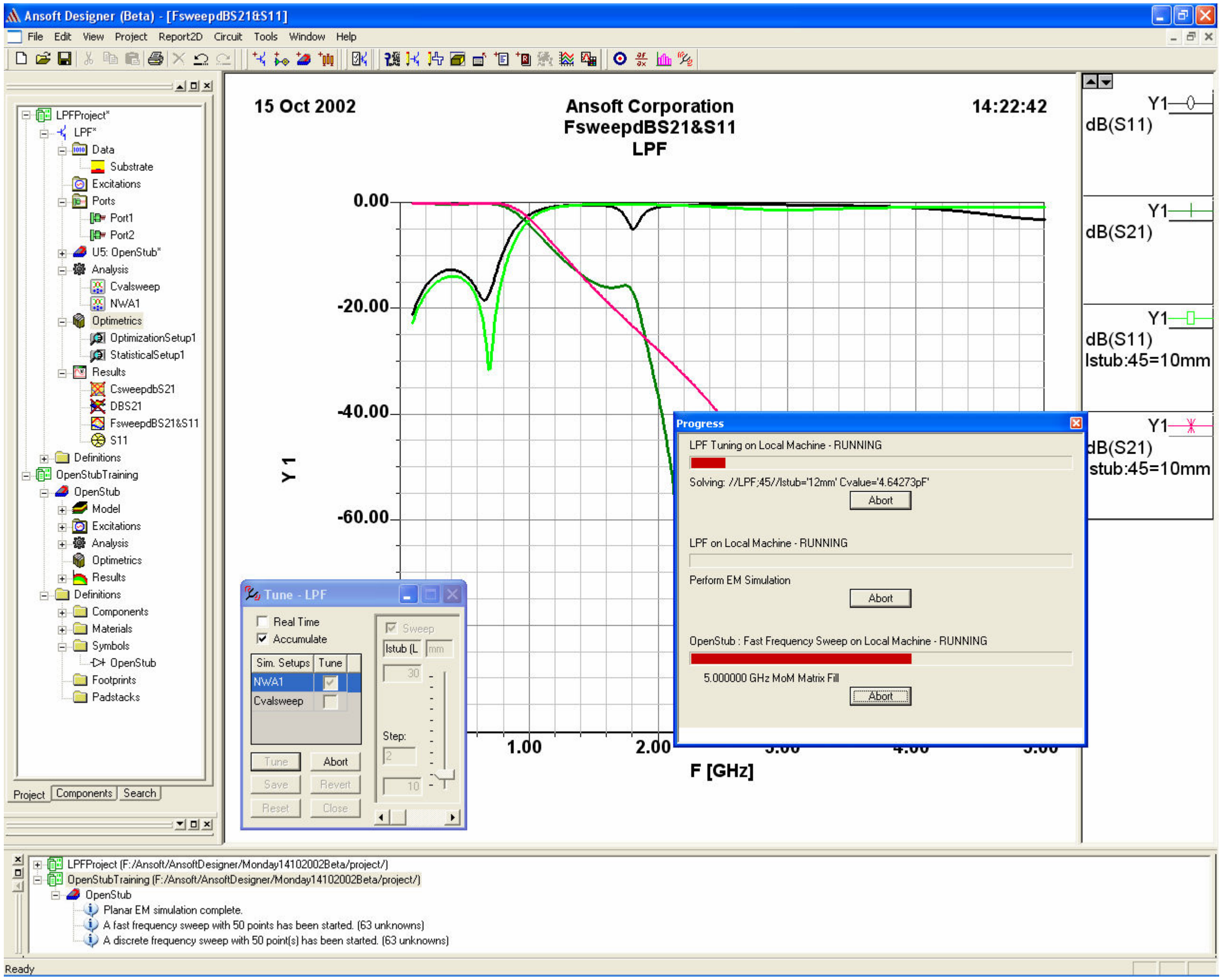


ANSOFT CORPORATION

Run Tuning


Ansoft Designer (Beta) - [LPFProject - LPF - Schematic]
Tuning...
View Analysis Result...
Start tuning analysis on the design.

Run Accumulate Sweep Tuning



Optional Exercise:
Planar EM Antenna Design

Configure Layout

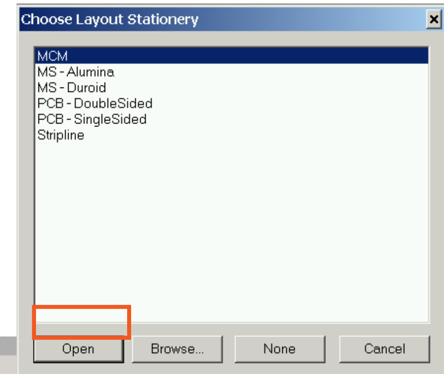
- Creating an Planar EM circuit will open layout.
- Click on ‘None’ in ‘Choose Layout Stationery’
- The grid and snaps for layout need to be configured.
  - Layout -> Grid & Snap
  - Set:

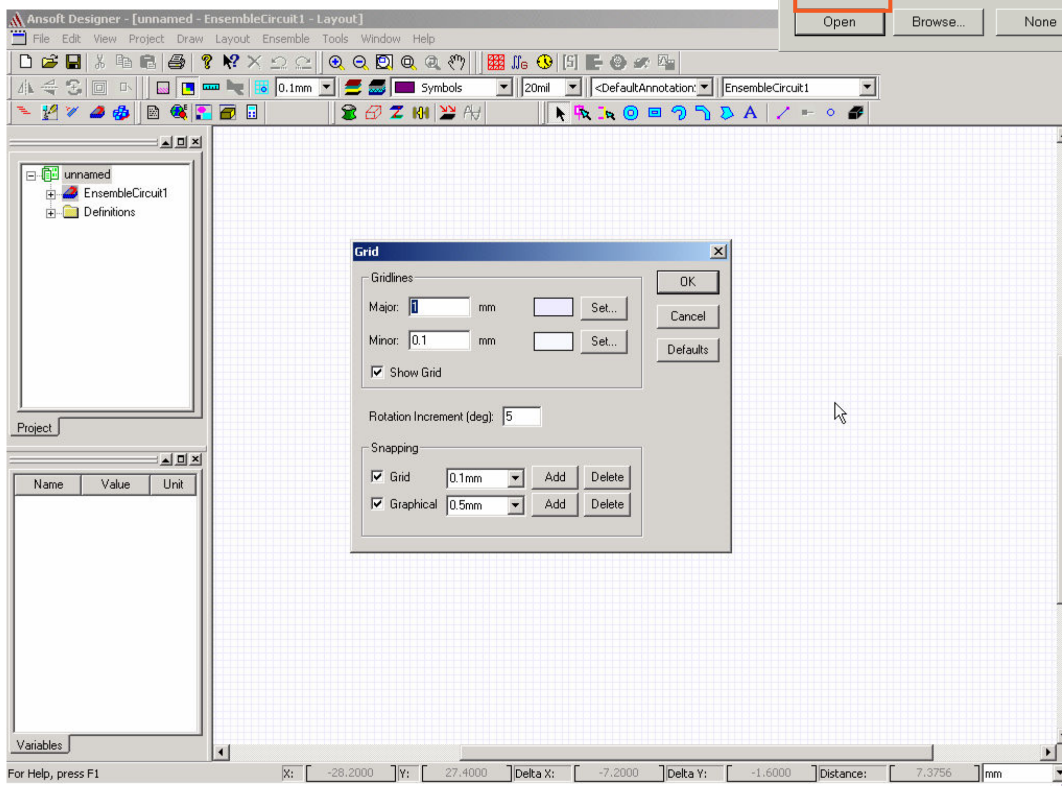
    Major = 1

    Minor = 0.1

    Grid = 0.1mm

    Graphical = 0.5mm
  - OK





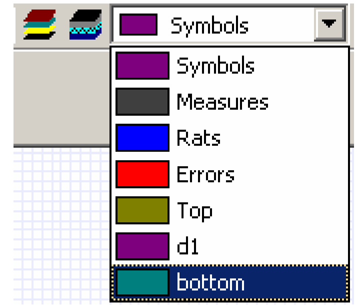
Insert Layers

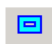
- Open the Stackup Editor in one of three ways:
  - Layout -> Layers. Select Stackup tab.
  - Click icon. Select Stackup tab.
  - Click icon.
- Insert an infinite ground layer
  - Add Layer -> Name = “top” -> Type = “signal”
- Insert a dielectric layer
  - Add Layer -> Name = “d1” -> Type = “dielectric”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

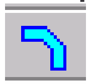
Drawing the model (1)



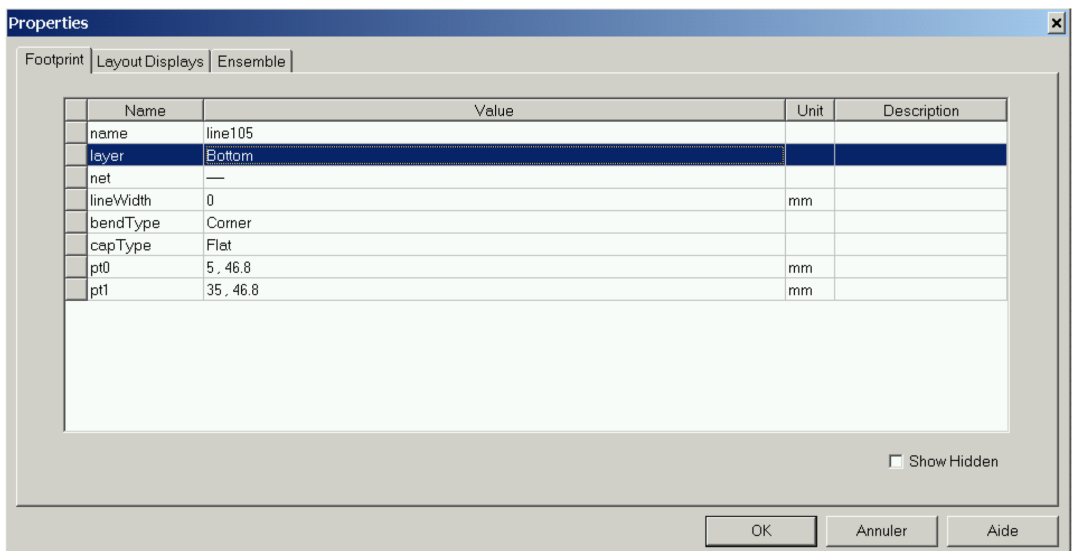
- Change the active layer to "bottom"
  - Select the layer pull-down menu and highlight "bottom"
- Begin drawing a rectangle object in one of two ways:
  - Draw -> Primitive -> Rectangle
  - Select the  icon.
- Enter the lower left-hand coordinates for the rectangle.
  - In the Status Bar, enter X=0.0, 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
  - deltaX = 30 , deltaY = 0

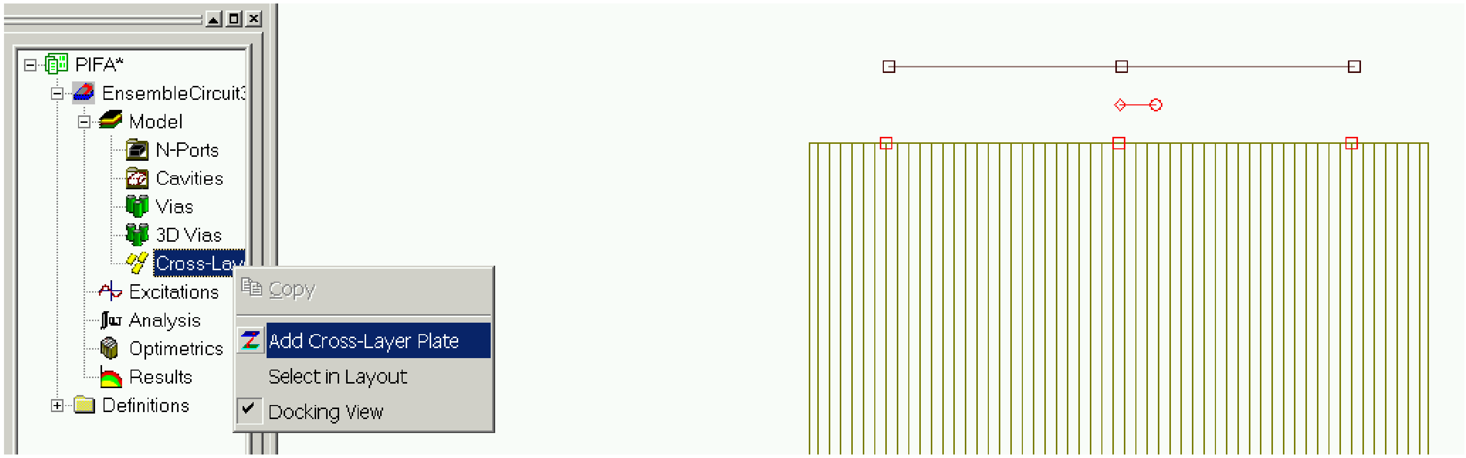
  - Use the TAB key to move between entries and press ENTER when finished.
  - Verify the following properties



Name	Value	Unit	Description
name	line105		
layer	Bottom		
net	—		
lineWidth	0	mm	
bendType	Corner		
capType	Flat		
pt0	5 , 46.8	mm	
pt1	35 , 46.8	mm	

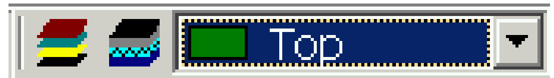
Drawing the model (3)

- 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

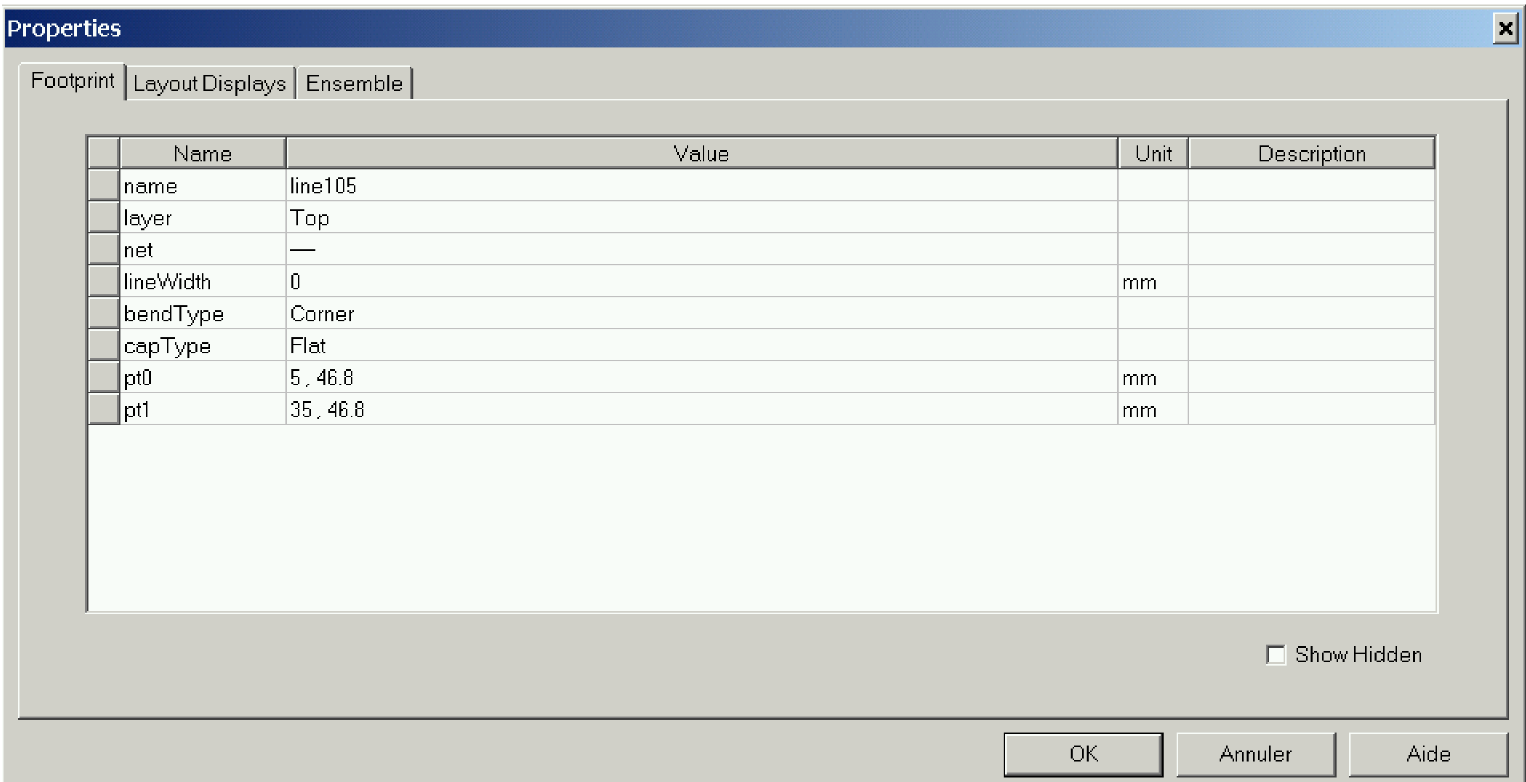


Drawing the model (4)

- Change the active layer to "Top"



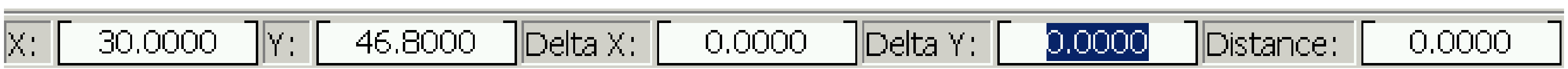
  - Select the layer pull-down menu and highlight "Top"
  - Double Click on the line and adjust the properties



Drawing the model (5)

- Begin drawing a rectangle object in one of two ways:
  - Draw -> Primitive -> Rectangle
  - Select the  icon.

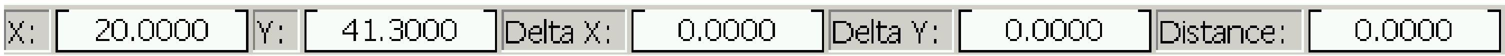
- Enter the lower left-hand coordinates for the rectangle.
  - In the Status Bar, enter X = 5, Y = 46.8. Use the TAB key to move between entries and press ENTER when finished.



- 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

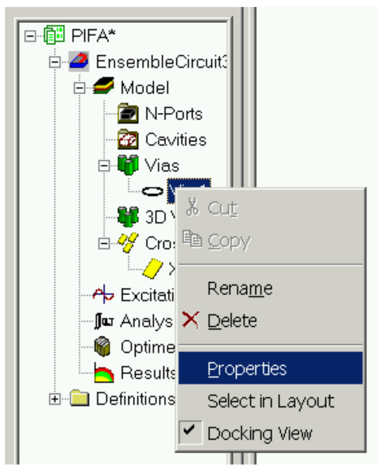
Define Excitation

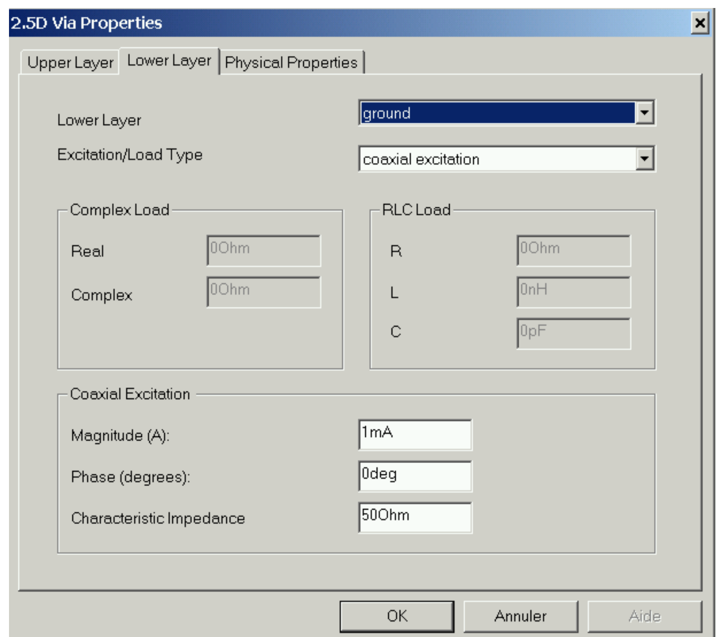
- Insert a hole 
- Double Click on the hole and change the properties



- 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.

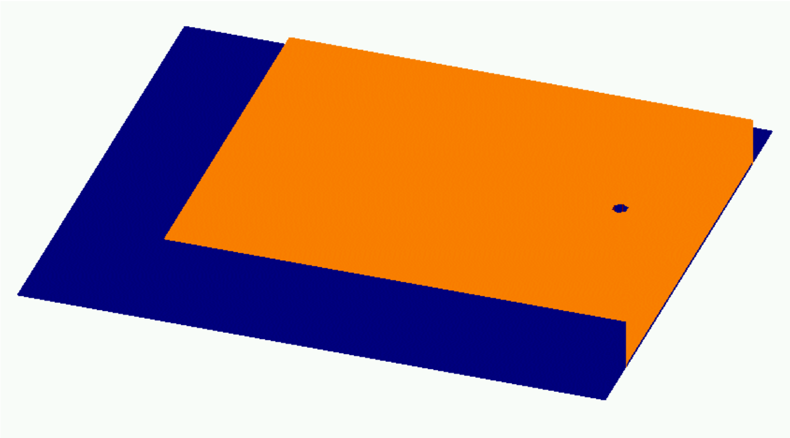
- Click on via1 and choose Properties





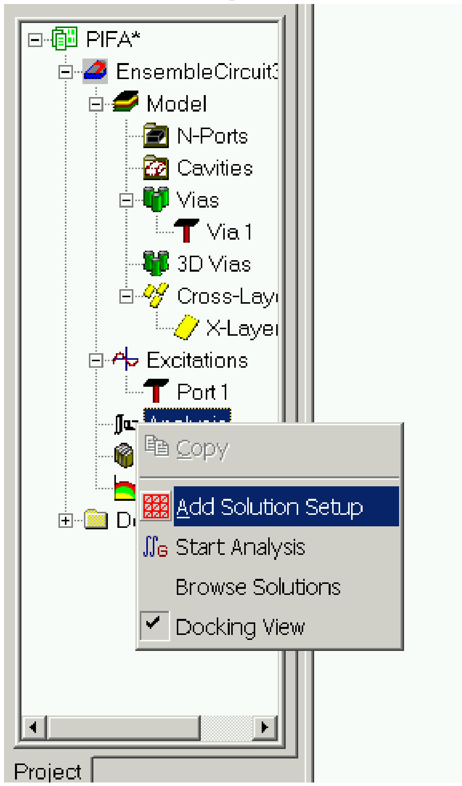
3D View

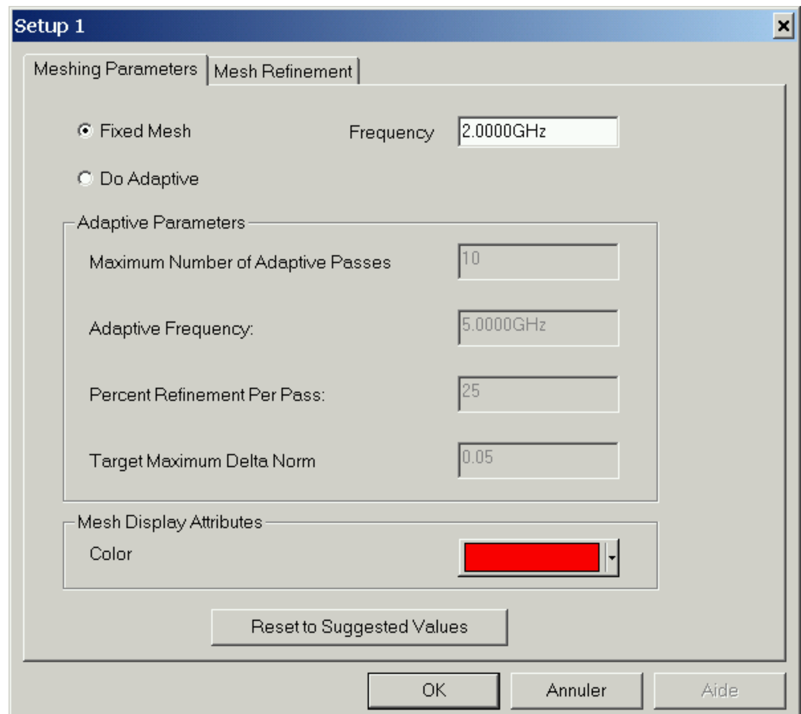
- Open a 3D view
  - Select Planar EM -> 3D Viewer   OR
  - Right-click on PlanarEM1 in the Project Manager and select “3D Viewer”
- Explore 3D view functionality
  - Right-click in the 3D view window
  - Many options are available here….
- Change the 3D lighting
  - View -> modify attributes -> lighting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

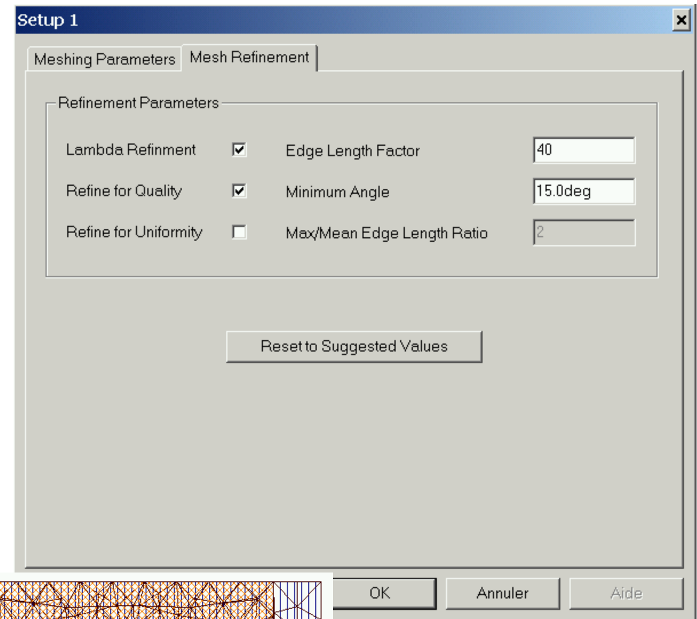




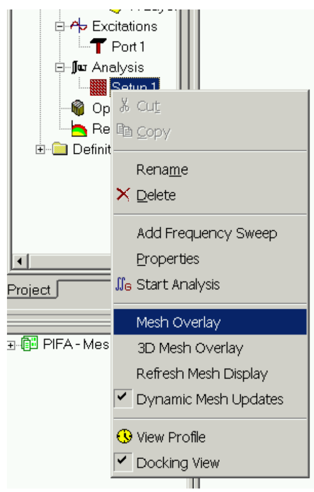
- Use a fixed mesh frequency of 2GHz. You can also change the mesh display color.

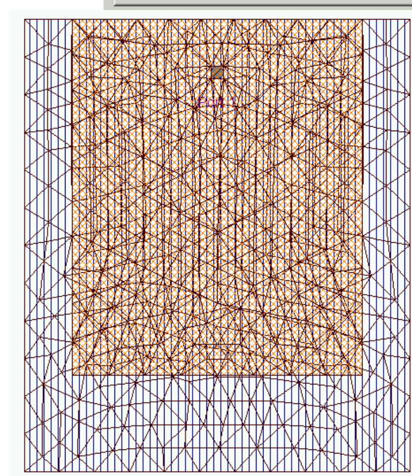
Setup Analysis (2)

- Modify Mesh refinement



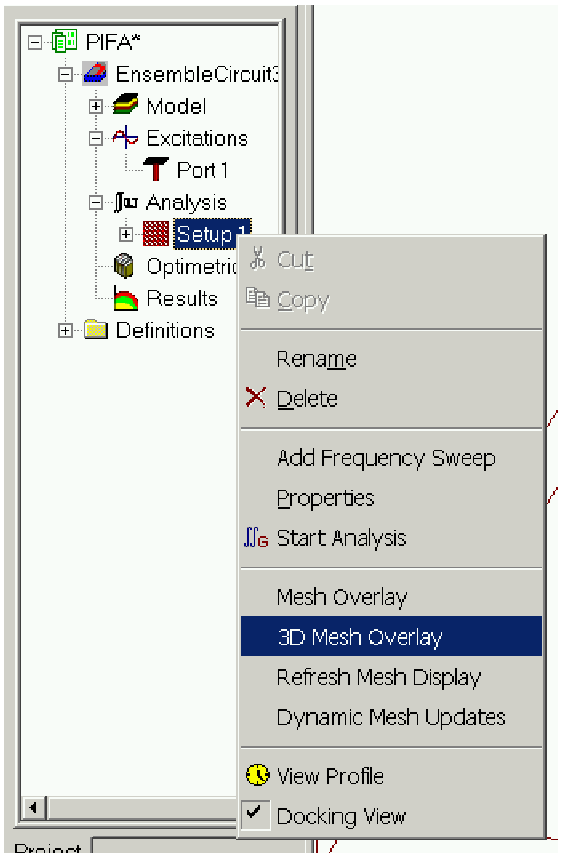
- Visualize the mesh

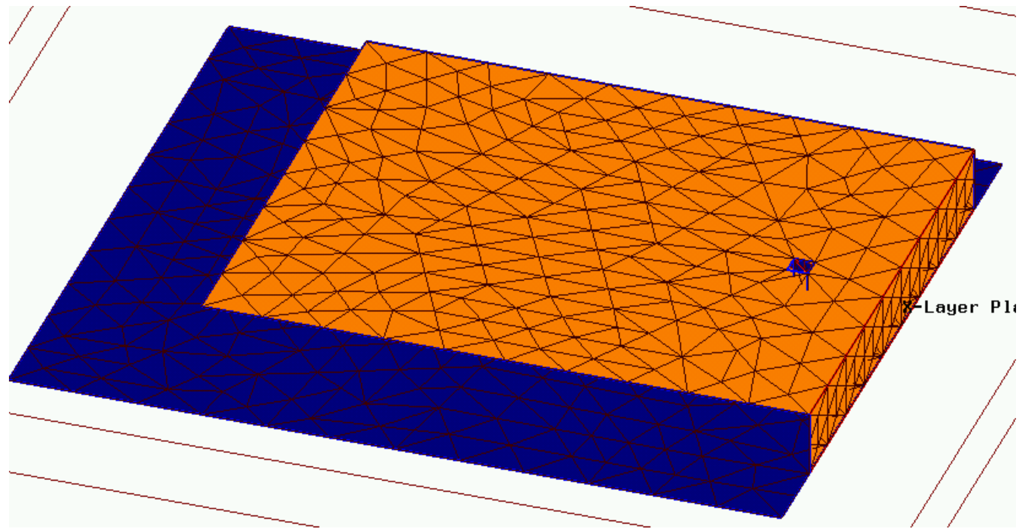




3D Mesh

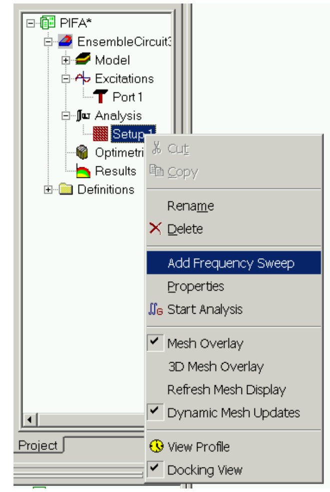
- Visualize 3D mesh



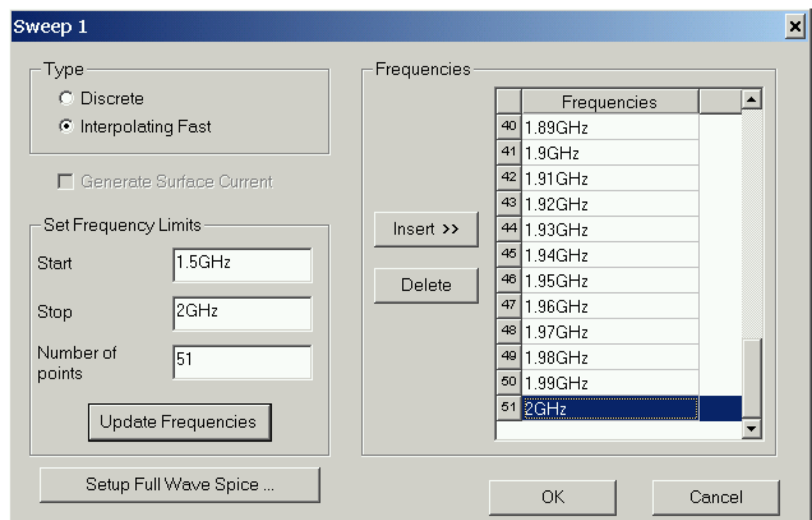


Setup Analysis (3)

- 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”

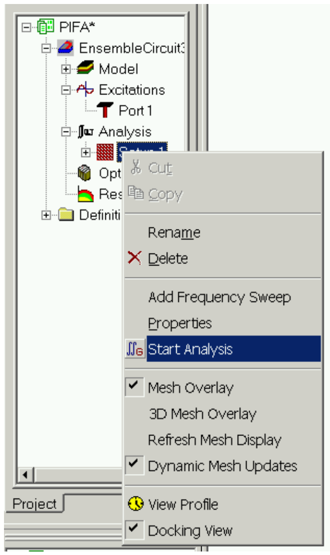


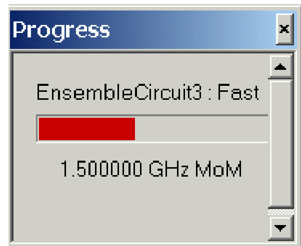
- Fill in the Sweep dialogue as seen below. Enter “Update Frequencies” to add the points to the sweep table.
  - Interpolating Fast Sweep
    - Fstart = 1.5GHz
    - Fstop = 2GHz
    - Number of points = 51



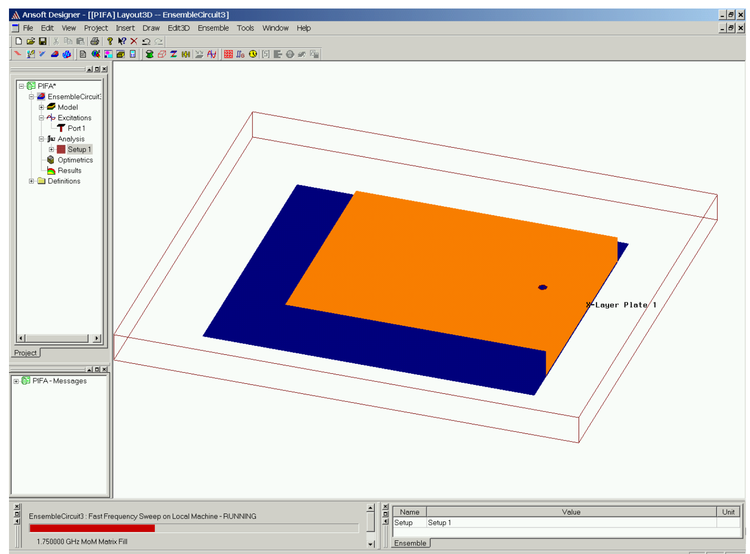
Setup Solution

- Right click on Sweep1 and select Start Analysis.



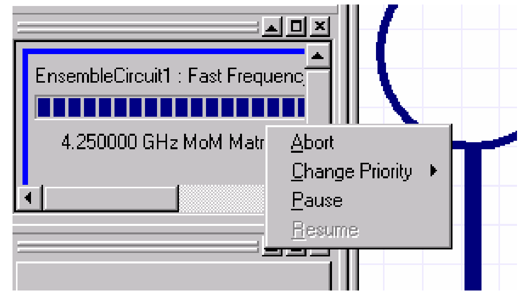


- Click on 3D Editor. 

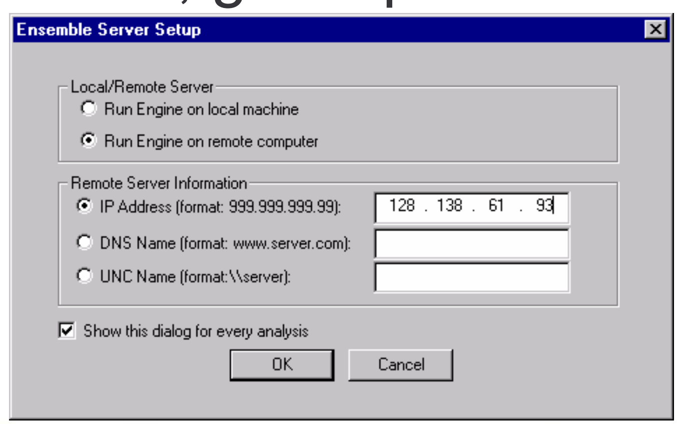


Simulation Odds and Ends

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

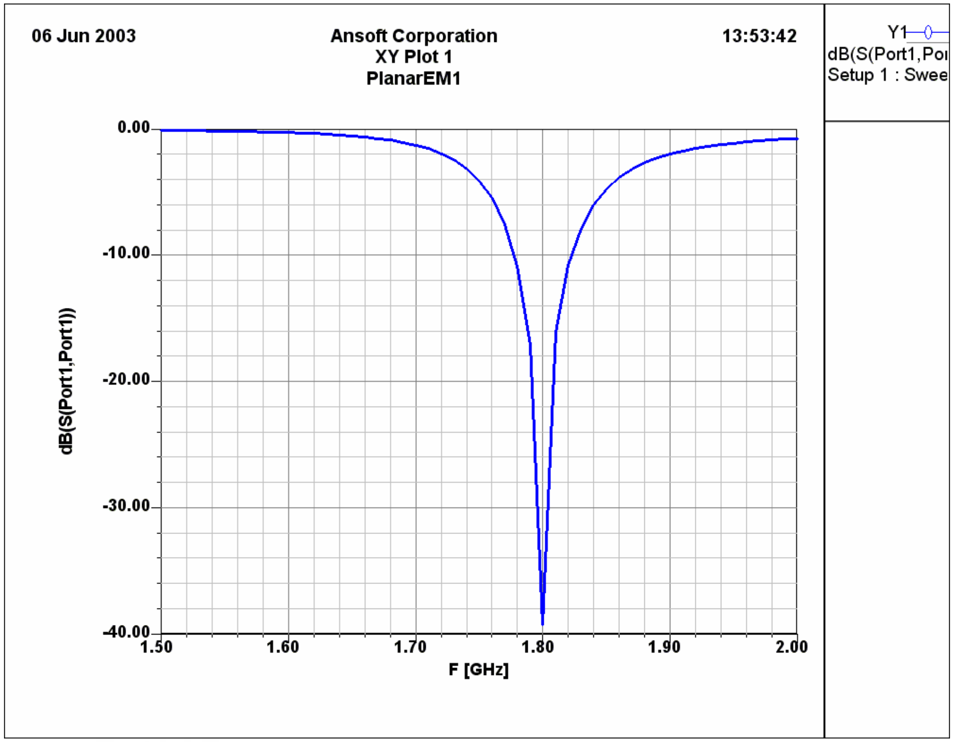


- The simulation engines have been turned into COM/DCOM objects. This allows a simulation to be run on a remote machine, given permissions and an IP address.



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



ANSOFT CORPORATION

Generate Reports (2)