Introduction of Keysight ADS (EE540)

Table of Contents

1 Introduction

ADS is capable of many different types of circuit analysis, electromagnetic models, and active device models.

A typical process flow of ADS begins with an insert the circuit components of your design, setup simulation, visualize result, and Optimization of design. In this document we will discuss a tutorial, ideal circuit simulation of a matching circuit, implemented with ideal lumped elements and ideal transmission-lines. The ideal transmission-lines and the lumped components are then replaced by microstrip transmission-lines. While dealing with these cases, the details of S-parameter simulation, line calculator, parameter sweeping, and design optimization tools will be introduced.

At the beginning it can seem quite difficult to use when faced with a blank schematic. This brief quick start guide will get you started using the simplest and most basic operations. For better understanding you will use your project topics to implement this in ADS and learn by doing.

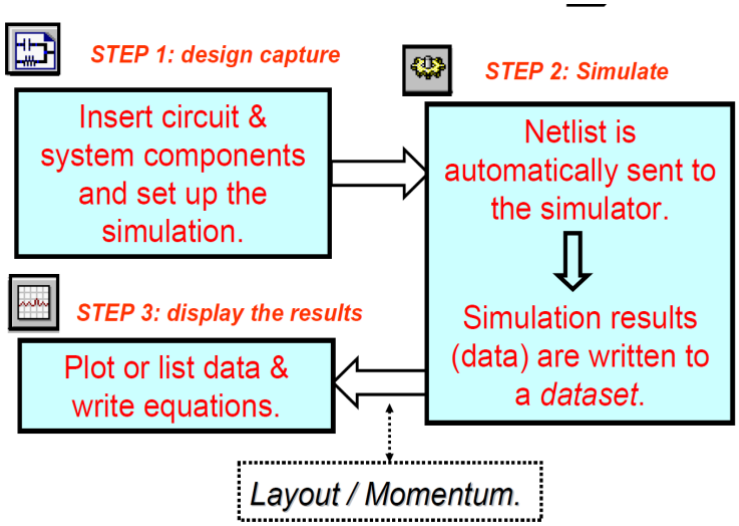
1.1 Download and Installation

Use the following request form to get the software and install at your PC. Contact instructor if you need help.

https://connectlp.keysight.com/StudentLicense

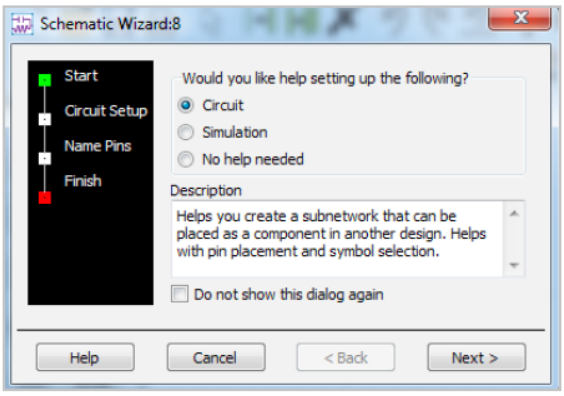
1.2 Design Flow in ADS:

There are three main steps in ADS design flow:

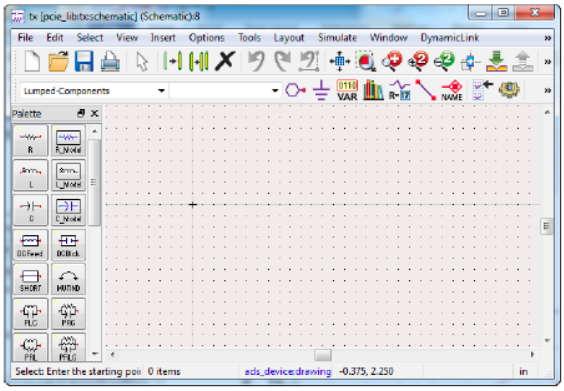


The list of work can be summarized as follows.

Close the Schematic Wizard by clicking Cancel.



A blank schematic window appears:



From this point, there are only three steps between you and answers you need in ADS. We call them:

The A B C's of ADS

Schematic Simulation Display

Summarized in 3 easy steps:

A) Place components, connection lines (called nodes), and a simulation controller onto the schematic. (The icon for schematic is . Later, we will explore the option to associate layout artwork with a schematic. The layout icon is .)

B) Create a dataset by clicking on the "run simulation" icon .

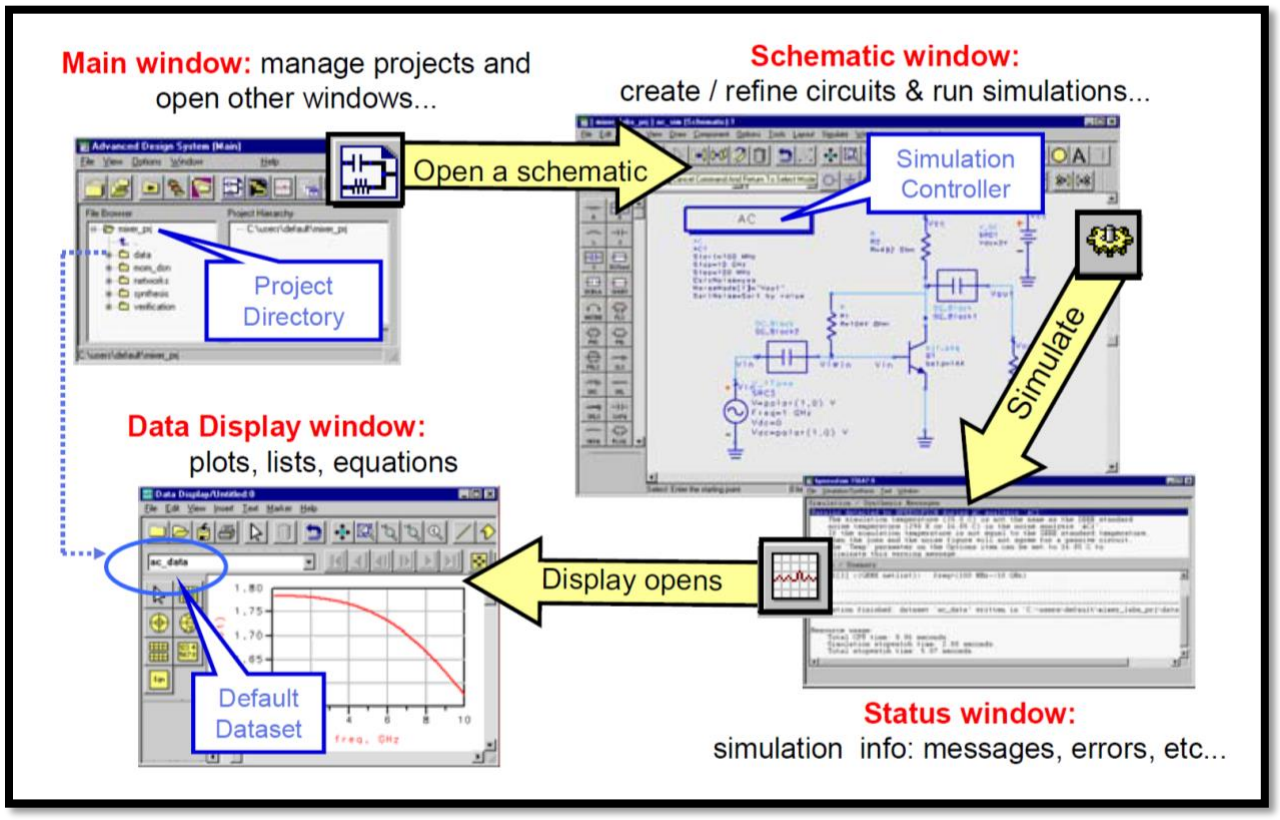
C) Create a data display to view your results in. The icon for the data display is .

That is it! That's all there is to it!

The ABC's will quickly become second nature to you. But for this first pass, let's walk through each one in detail by building a simple design to look at the eye diagram that we will be transmitting later on.

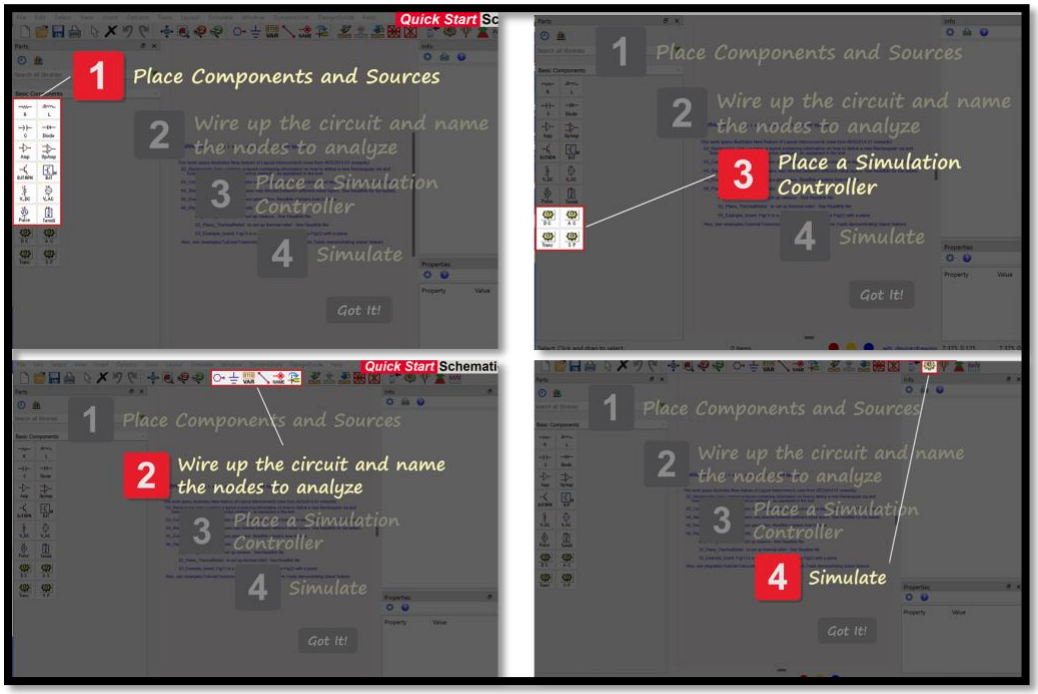
1.3 ADS Important Windows

The main window in the ADS are follows. The latest version also have similar number of windows however the windows outlook is changed.



1.4 Quick Start Guideline

The latest version can also help you to complete your design step by step. See the figure below.



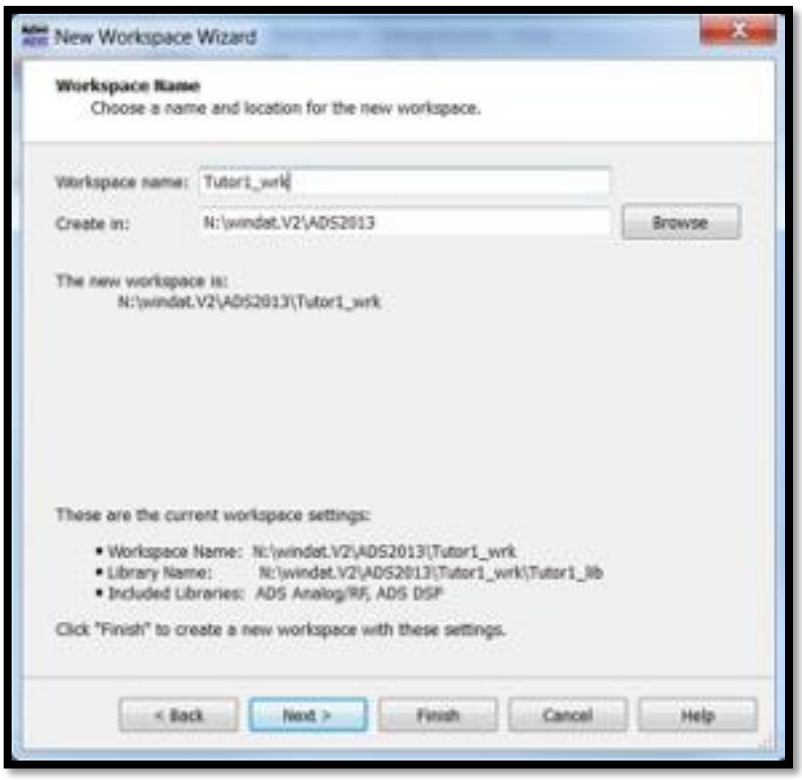
2 Step 1: Creating Workspace and Library

The first step is to setup the workspace and library tpo start your Schematic project

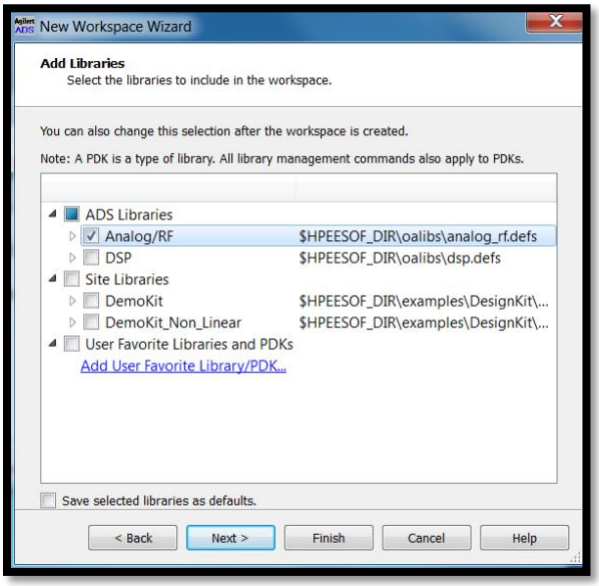
From the ADS main window,

- Select **File**>**New**>**Workspace…** The **New Workspace Wizard** opens.

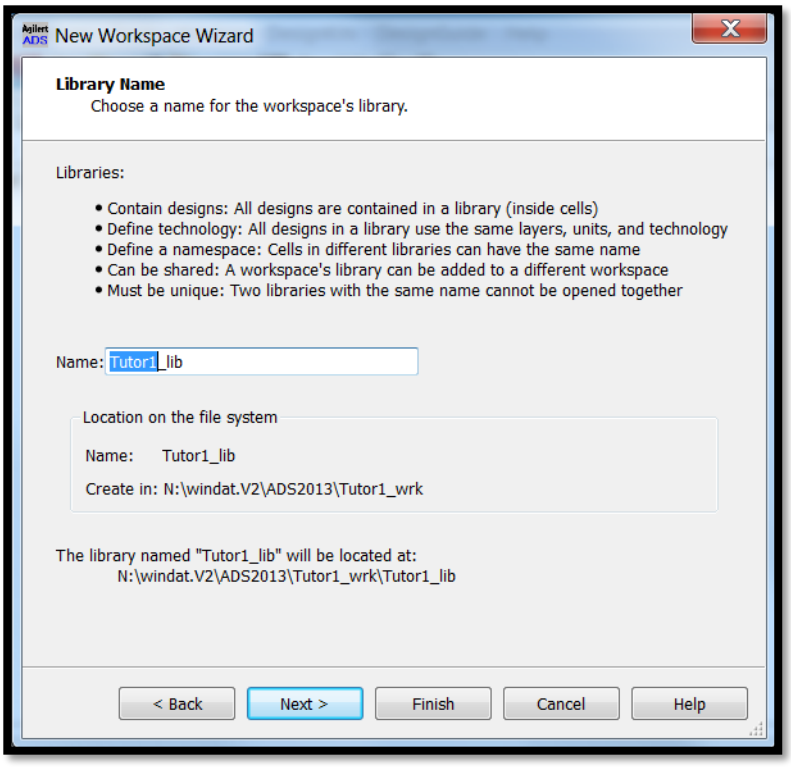
- Click the **Next button** at the bottom. Type an appropriate name (for this lecture:
- **Tutor1_wrk**) in the "Workspace name". **Create the directory where ADS files will be stored**.



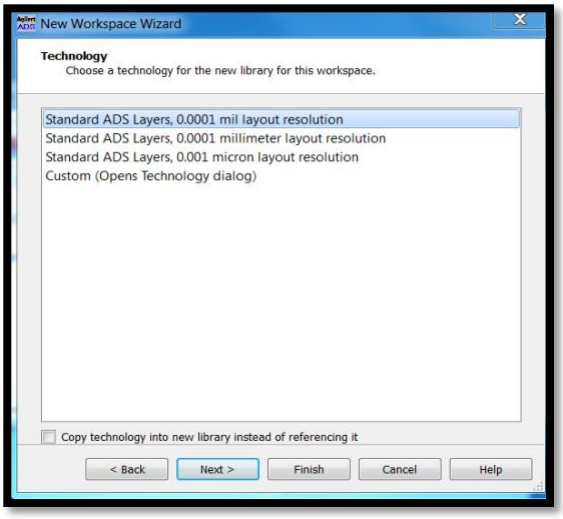
- Click **Next** and select the libraries you will use. We will use the ADS **Analog/RF** libraries throughout this course.



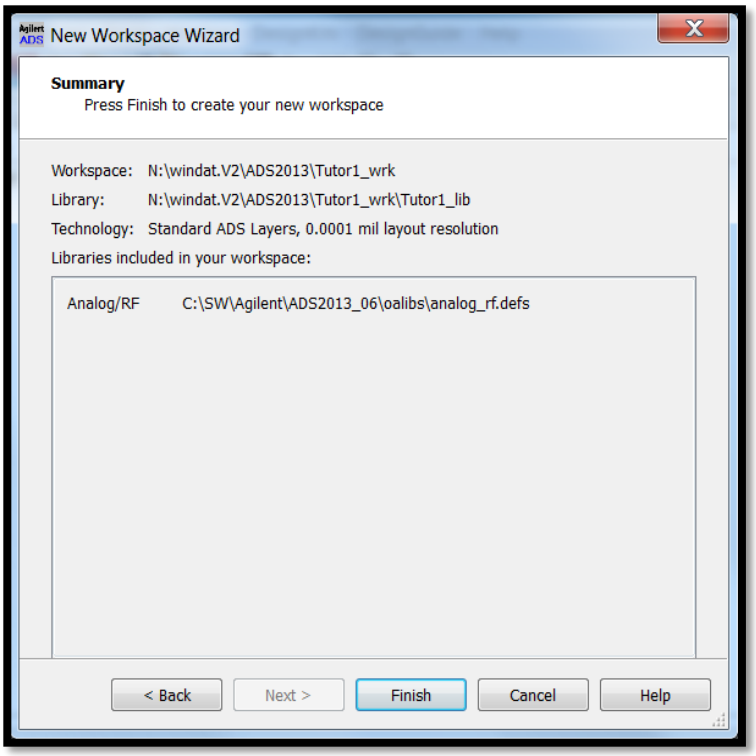
- Click **Next** and enter a library name for your work or use the default.



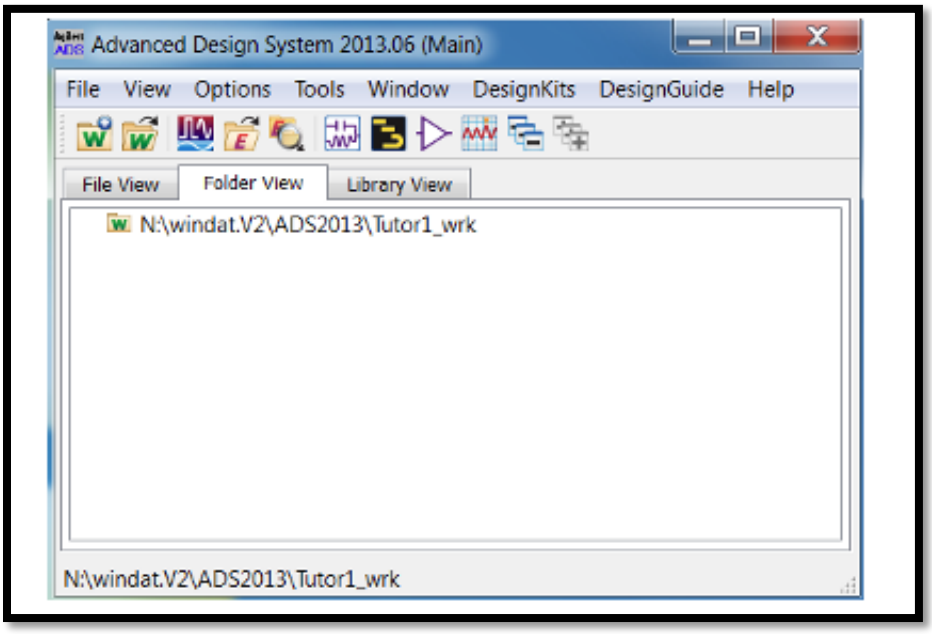
- Click **Next** to select the technology (0.0001 mil layout resolution).



- Click **Next** and review the **Summary**. Make changes if necessary or click **Finish**.

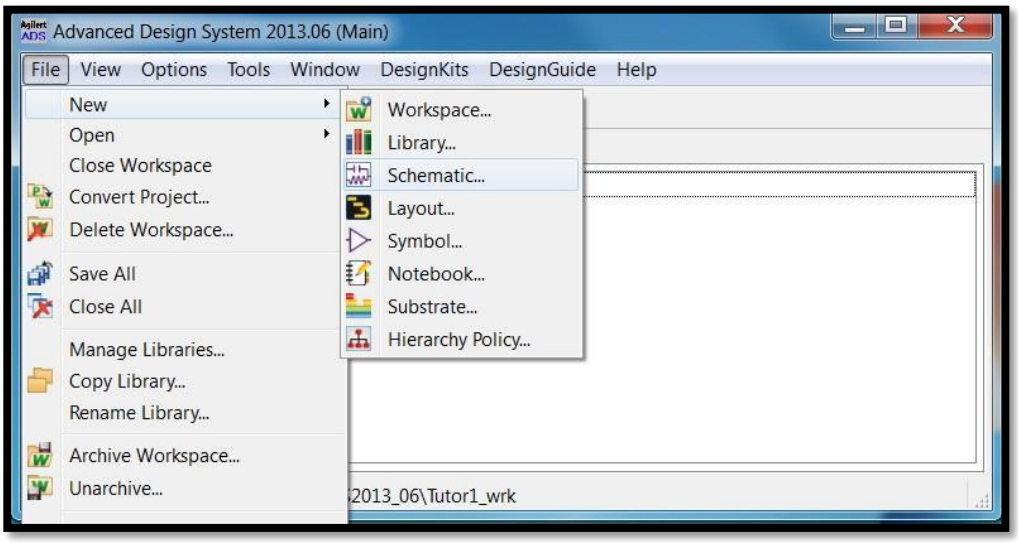


- The workspace and library are created.

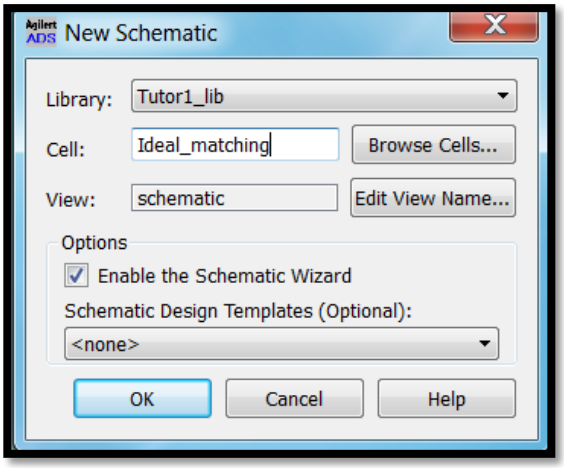


- Creating a New Schematic View

✓ Create a new schematic by selecting **File**>**New**>**Schematic**

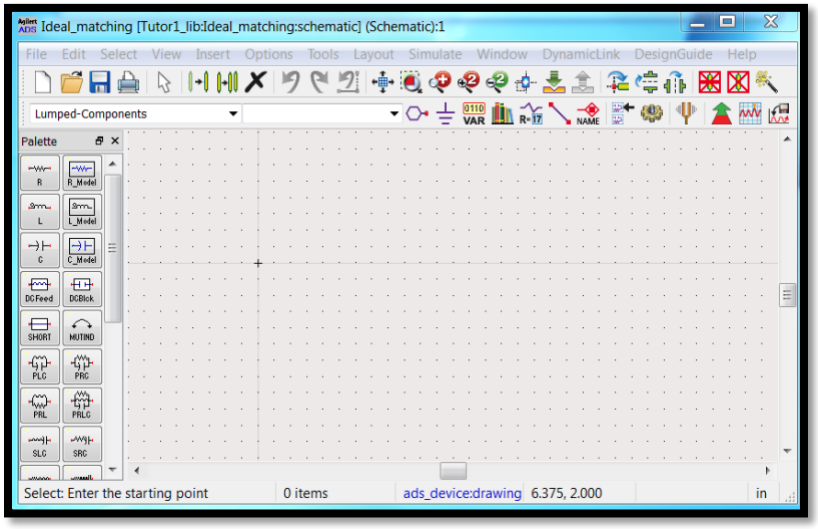


The **New Schematic** window opens. Type an appropriate name (e.g. **Ideal matching**) in the Cell.



- ✓ Click OK and cancel the "help" option.

An empty schematic named **schematic** opens.



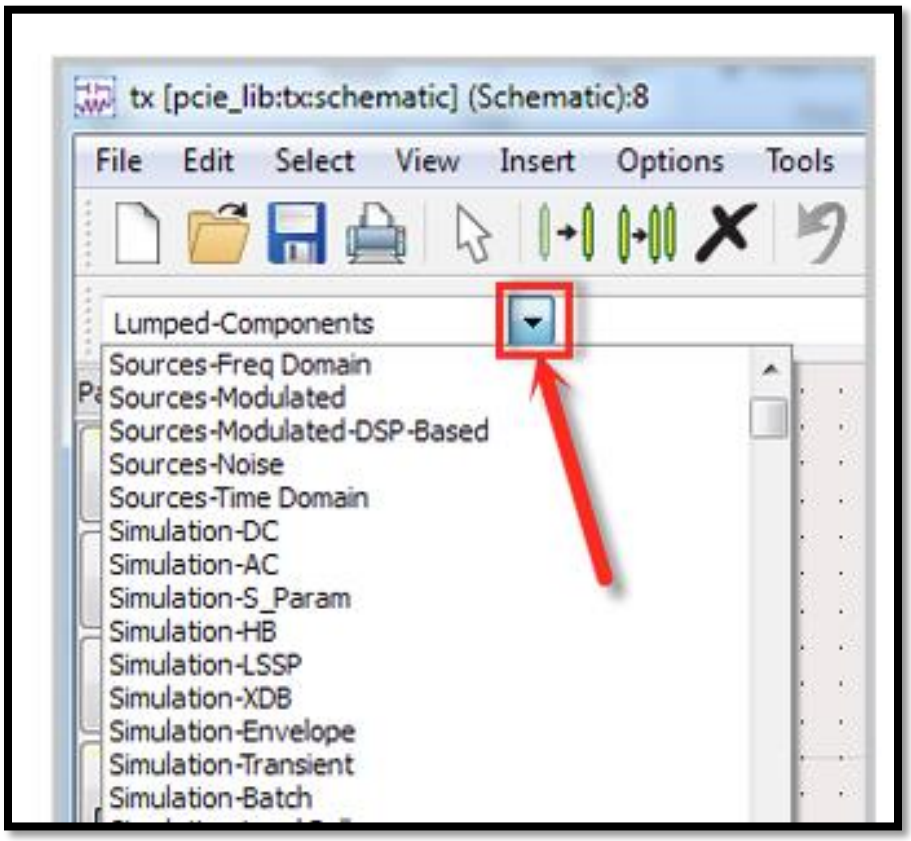
3 Step 2: Inserting and Editing Components

To place the components, we need to choose the relevant library from the ADS library list:

Mirroring the real world, there are thousands of components available in the ADS library  .

The palette on the left of the schematic is a tool to help organize them. By default, the Lumped-Components palette is displayed.

– Click on the down arrow  to show the list of



See an Example Below:

- To place the ideal transmission line

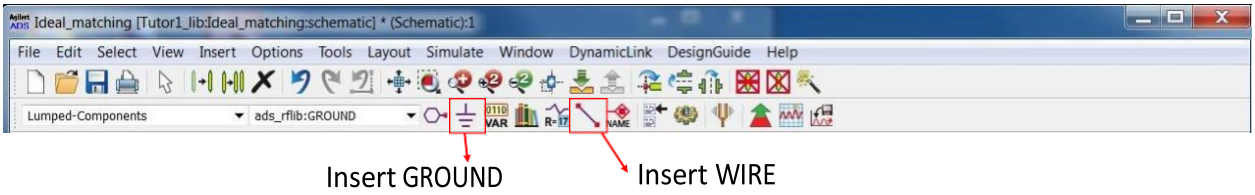
  ✓ Choose “TLines-Ideal” from Palette list and then select “TLIN” from Component Palette.

- ✓ Place it in the schematic window and set Z = 50 Ohm, E = 141.8 (degree) and F = 1 GHz.

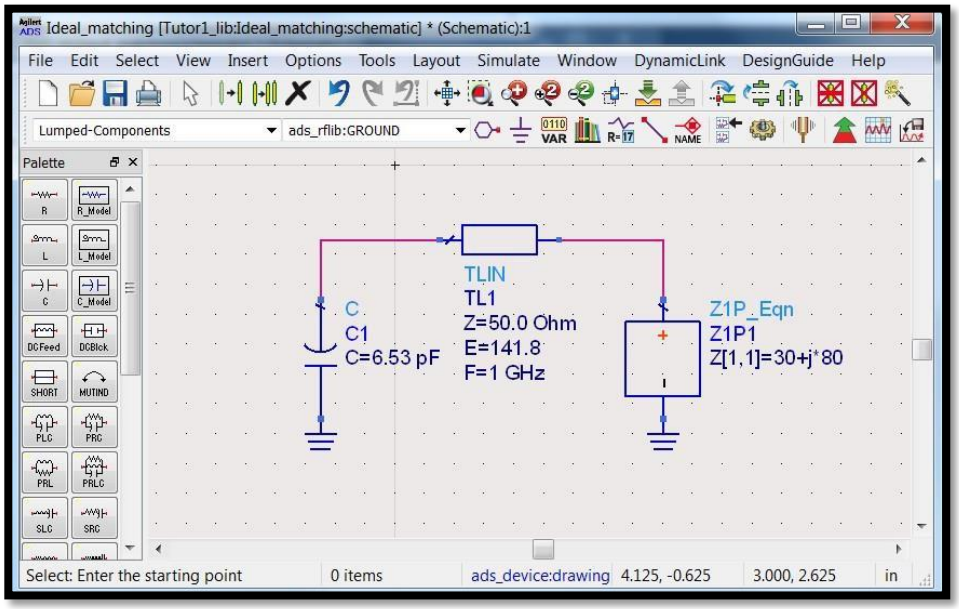
- To place the capacitor

  - ✓ Choose "Lumped-Components" from "Palette list" and then select "C" from Component Palette.
  - ✓ Place it in the schematic window and set C = 6.53 pF.

- To connect the components and place the ground, use "Insert GROUND" and "Insert WIRE" (Insert WIRE short cut: Ctrl + W) icons as shown below.



Now, your circuit should look like this.



4 Step 3: Simulation Setup and Optimization

ADS has several simulation methods. You need to specify the type of simulation. Here is a brief description of some simulation methods and their applications.

- ✓ **AC**: The AC simulation performs a small-signal, linear AC analysis. As part of the analysis, the DC operating point is calculated, and any nonlinear devices are linearized around that operating point. This analysis does not generate harmonics or exhibit compression. An AC simulation enables you to obtain small-signal transfer parameters, such as voltage gain, current gain, trans-impedance,

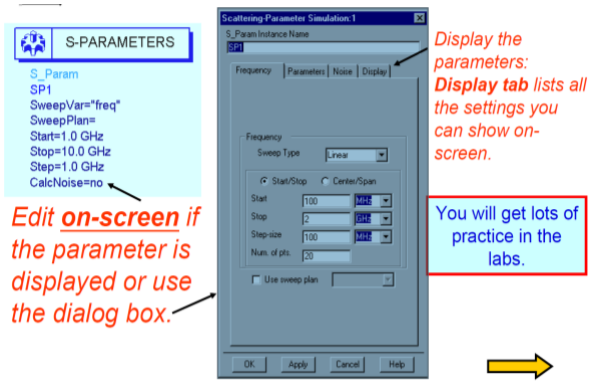
trans-admittance, and linear noise. Applications: Filter, Amplifier.

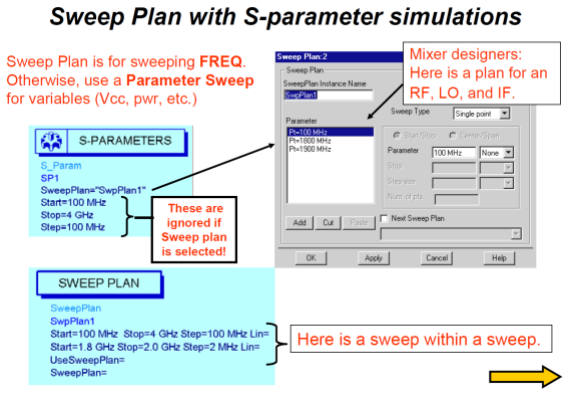
- ✓ **S-Parameter** provides linear S-, Y-, or Z- Parameters, linear noise parameters, trans- impedance, and trans-admittance. It can be used to achieve many goals of the AC simulator. Applications: Filters, couplers, amplifiers, matching networks. The S- parameter simulation can be used for both passive and active network small signal analysis. ***It is the most important simulation method for linear microwave circuits.***

- ✓ **HB (Harmonic Balance)** is a nonlinear mix domain (frequency and time domains) solver. Only the steady state results are obtained. Application: Large Signal Amplifiers, Mixers, Oscillators. **It *is the most popular simulation method for nonlinear circuits*. It does not provide start up and transient data.**

- ✓ **Transient** solves a nonlinear system entirely in the time domain through finite difference approximation of time-derivatives. It is used to simulate both the transient as well as the steady state behavior of circuits.

See example of adding S-parameter





See an Example Below:

In this tutorial and project, you will mostly use **S-parameters** analysis for linear circuits and **HB** analysis for nonlinear circuits. The matching network used in this tutorial is a linear passive circuit. Therefore, it is simulated through S-parameters analysis.

- ✓ Choose "Simulation-S_Param" from Palette list and then select "S_Param" and place it in the schematic.

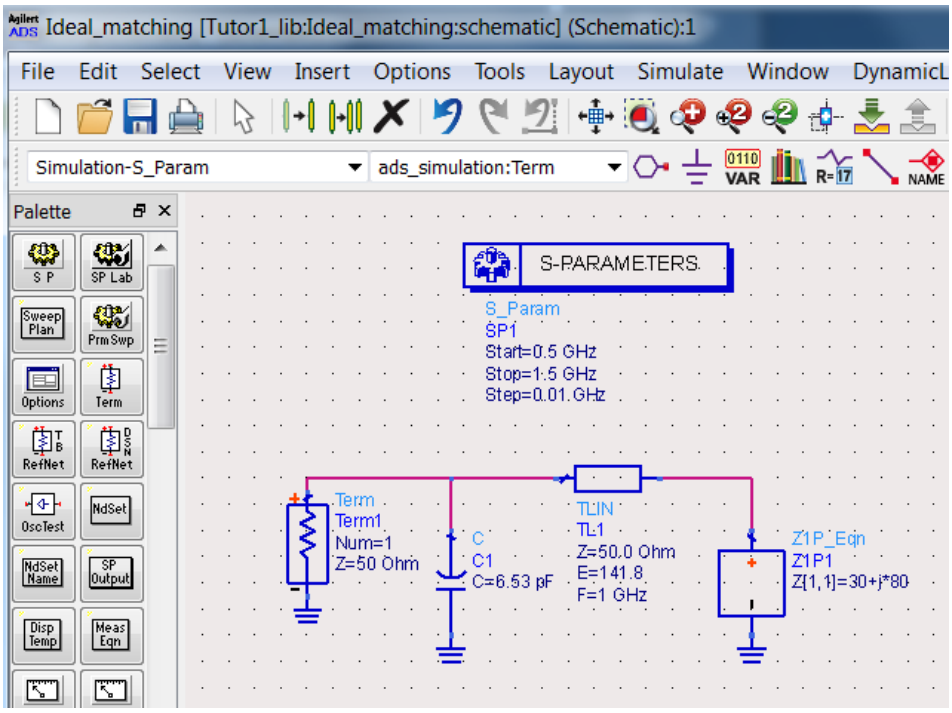
- ✓ Double click it and set the frequency sweep type to "linear" and set the frequency start from 0.5 GHz and stop to 1.5 GHz with 0.01 GHz steps and click "Apply".

- ✓ You can choose/change network parameters to be simulated (S-, Z-, and Y corresponding to scattering, impedance and admittance parameters, respectively) from the "Parameters" tab.

- ✓ Click “OK” to close the dialog box.

In S-parameters analysis, you are required to set terminations at the ports of your circuit. This is because the scattering parameters are calculated when the circuit ports are terminated (matched to 50 ohms).

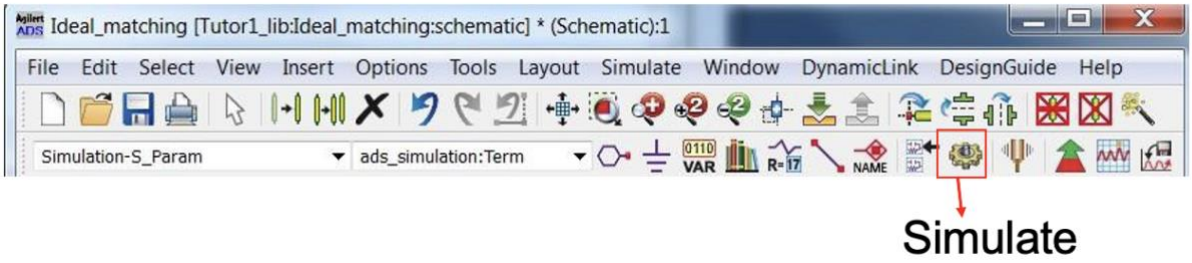
- ✓ Select “Term” from Palette and place it in to the schematic. Each “Term” represents a microwave port in the schematic.
- ✓ Connect the “+” and “-” nodes of “Term” to your matching circuit (see the following figure)
- ✓ By double clicking on the “Term” component you can set the termination impedance. It is by default 50 ohms. Leave it as default.



5 Step 4: Simulation and Data Analysis

5.1 Simulate

Click on the “simulate” icon (Shortcut: “F7”) to run simulation. A window is launched, which shows the simulation progress. After the simulation is done, a “data display” window pops up.

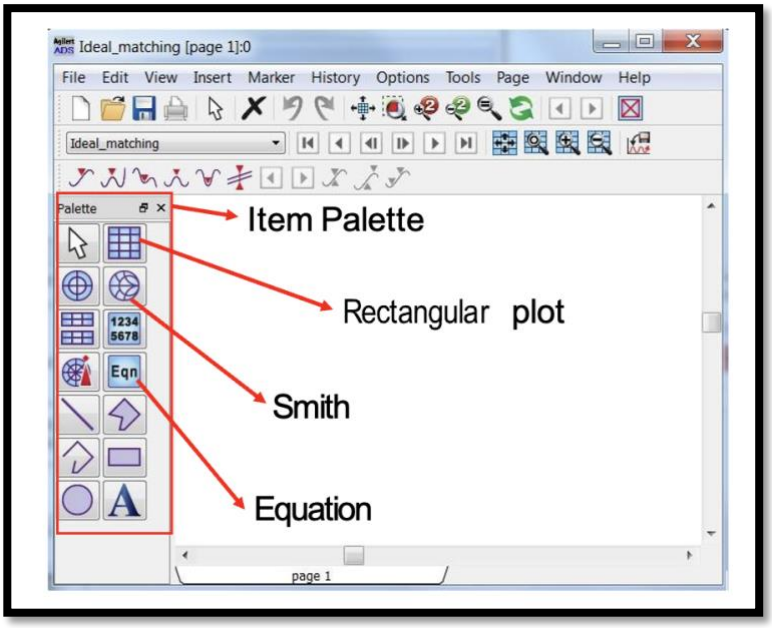


|

5.2 Simulation Result

A data display window is used to view and manipulate simulation results. In the data display window, you can write mathematical equations to post-process the simulation data.

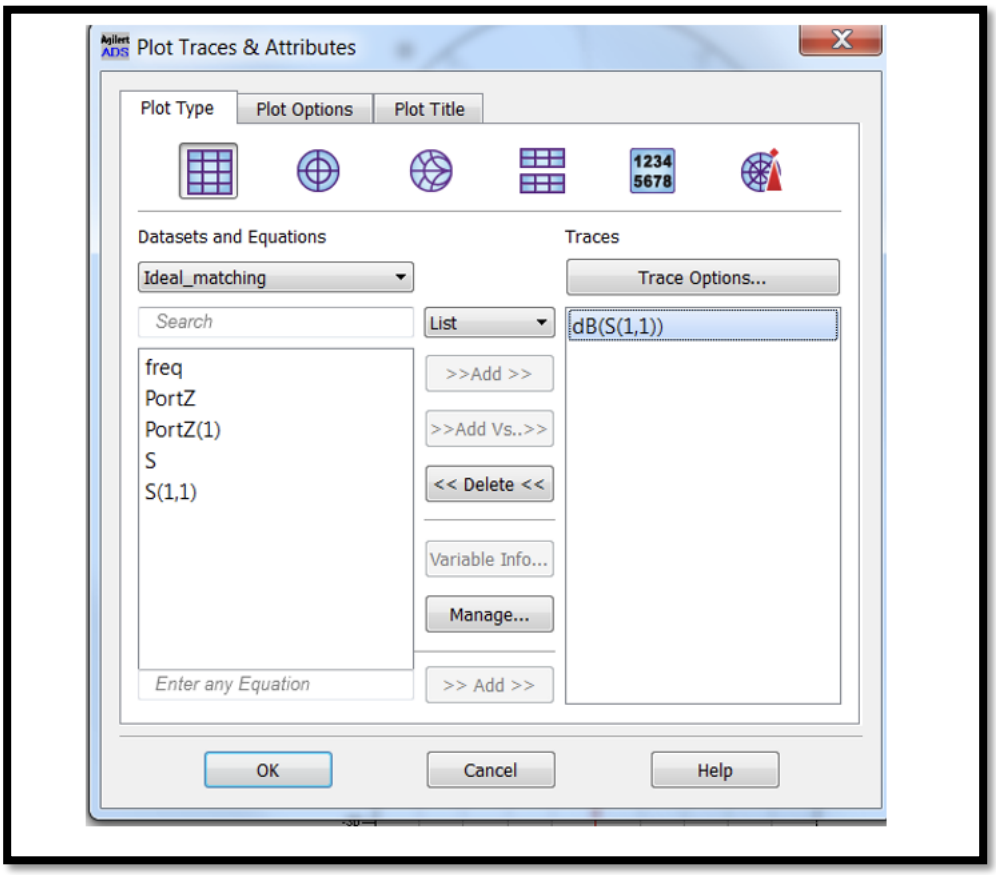
When your simulation is done, a data display window pops up with the same name as the simulated schematic design name.



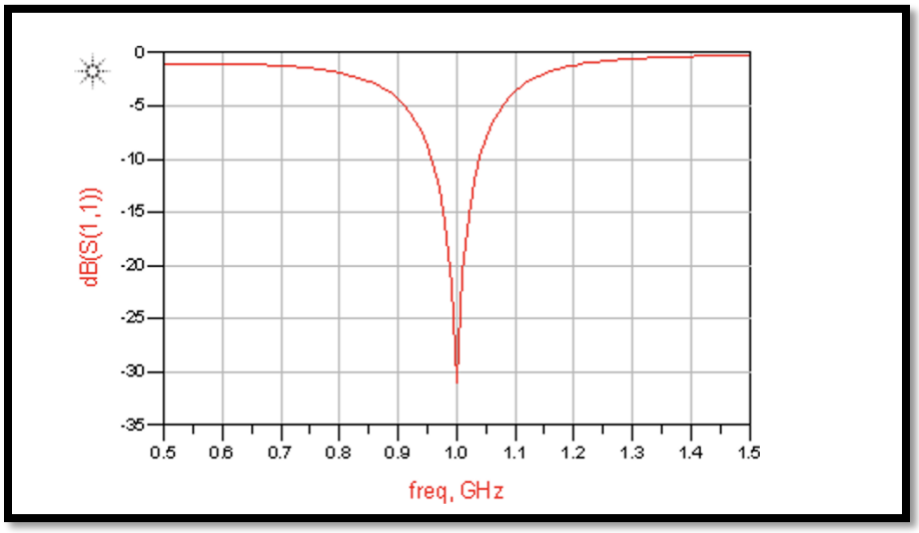
Select “Rectangular Plot” from Item Palette and place it in the window. A dialog box (Plot Traces & Attributes) appears asking you to select parameter(s) to plot. The parameters available to plot are listed on the left side.

To view the performance of your matching circuit, you should display the input reflection coefficient or S(1,1).

- ✓ Select S(1,1) and then press “Add” button.
- ✓ Select “dB” to view the data in dB format and then click “OK”.



The result of the plot look like as follows:

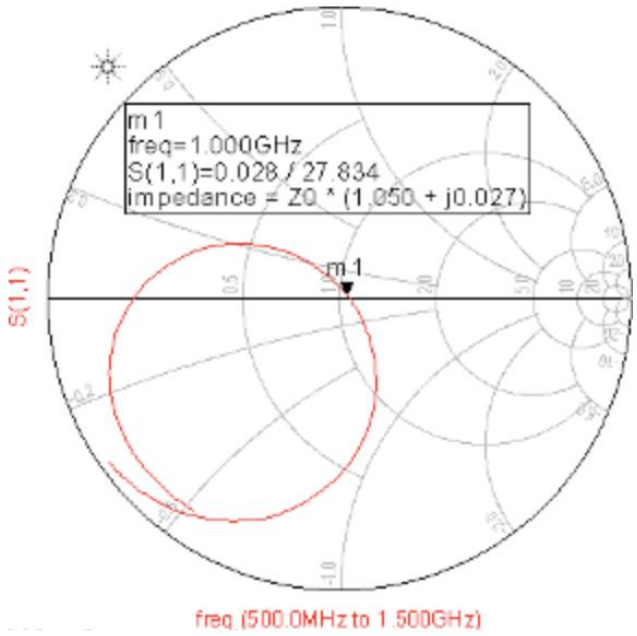


As it shown above, the reflection coefficient is around -30 dB, indicating a well- designed matching network at the design frequency of 1 GHz.

You can also plot the reflection coefficient on the smith chart.

- ✓ Select “Smith” from Item Palette and select S(1,1) to be plotted.
- ✓ To place a marker on the graph, use marker icons (Shortcut: “Ctrl + M”).

- ✓ Then, click on some point on the S(1,1) graph.
- ✓ Move the marker to the point at which the frequency is 1 GHz.



As you can see, the input impedance at 1 GHz is Z0*(1.05+j*0.027) ohms (by default, Z0 is 50 ohms), which is very close to 50 ohms.

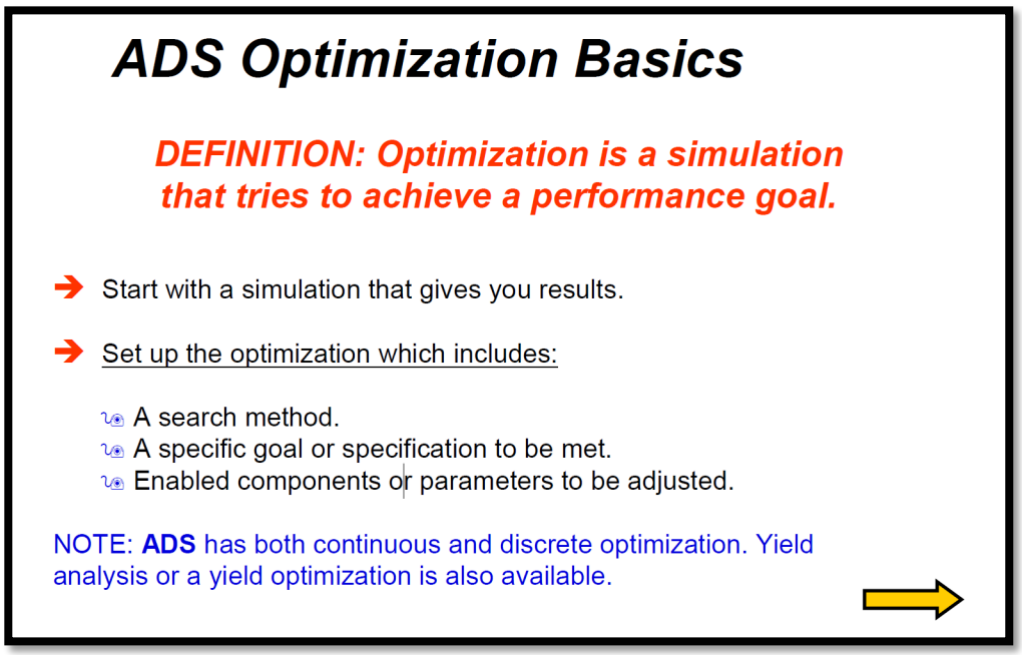
5.3 Tuning and Optimization

Tuning: Check the Optimization and Design useful resource link

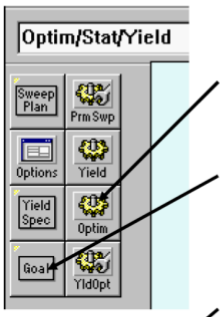
Tuning is used to check the effect of components values on the output performance.

Optimization:

Optimization is automatically reach at the design goal by defining specific performance goal.



Optimization palette: Controller and Goals



Four steps for optimization:

Optim *controller: set the type, etc.*

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

Enable *component (opt).*

Simulation *controller.*

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

Many Types are Available

Random & Gradient are often used together…

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

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

Goals and Error Function

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

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

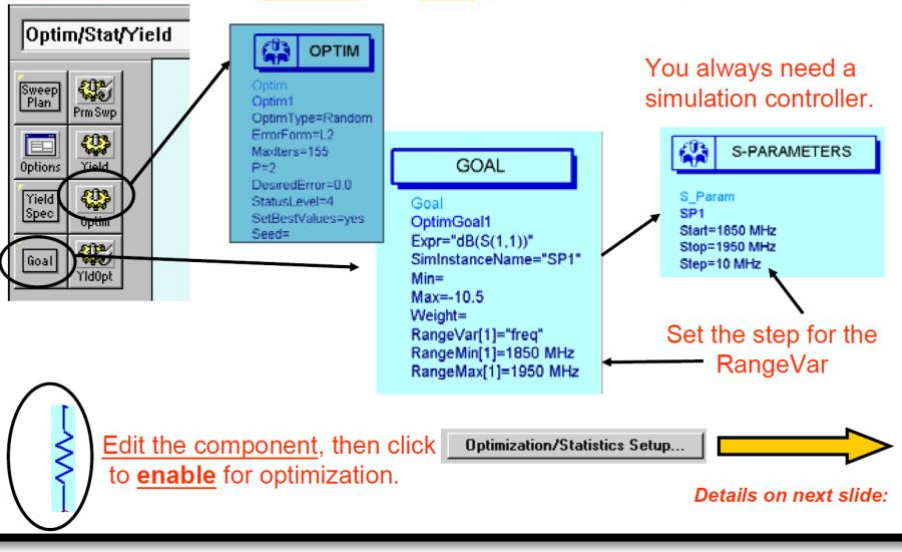
$$r_i = W_i \, | \, m_i - s_i \, |$$

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

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

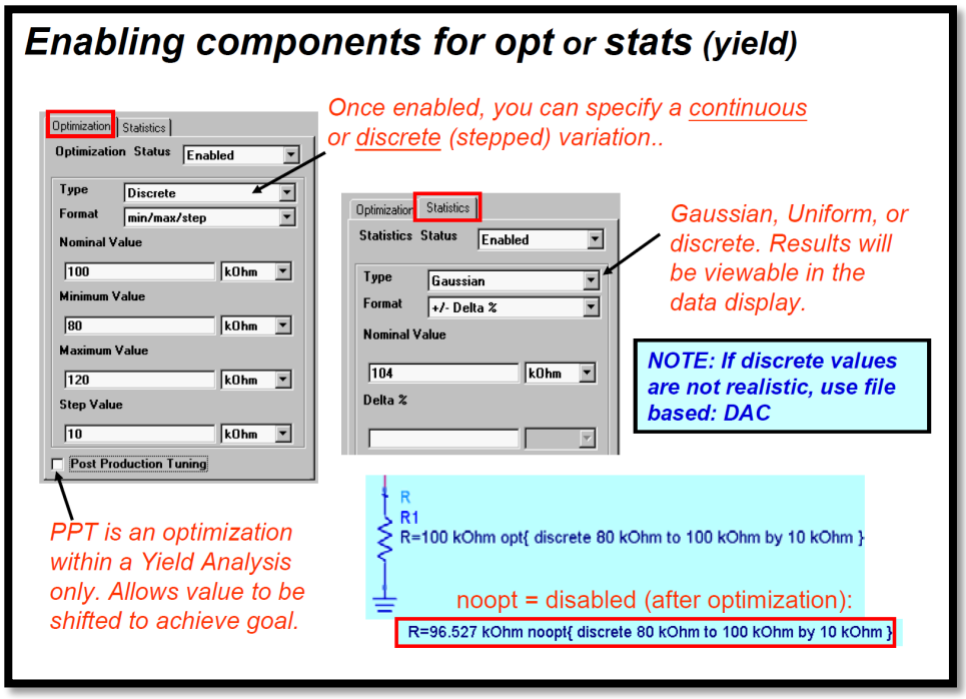
ADS Optimization Setup

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



Edit the component, then click Optimization/Statistics Setup... to **enable** for optimization.

Details on next slide:



6 Useful Resource

I. Start with ADS (video)
II. Filter Design (video)
III. Overview ADS (PDF)
IV. Tuning And Optimizations