Bern University of Applied Sciences
Engineering and Information Technology

Division of Electrical and Communication Technology

AgilentEESof Advanced Design System ADS 2012

Student Quick Reference Guide

Part 1

Part 1

Inhalt

1. Einleitung und Einstiegshilfen für ADS 2012

Dieses Dokument soll einen schnellen Einstieg in die EDA-Software ADS 2012 von Agilent erleichtern und Hilfe zu Themen im Zusammenhang mit dem Studentenprojekt im Modul „Telecom-Elektronik“ anbieten. Bei Teilen die der englischen Benutzerführung in ADS naheliegen oder Auszüge aus ADS-Help beinhalten, wird die englische Sprache verwendet um die Terminologie beizubehalten. Andere Teile werden in deutscher Sprache beschrieben.

ADS Help Documentation

„Advanced Design System Quick Start”: adstour.pdf
Dieses Dokument ist auf Boiler classes\ADS2012 abgelegt.

ADS Homepage

http://www.home.agilent.com/agilent/product.jspx?nid=-34346.0.00&cc=US&lc=eng

Agilent EEsof EDA Forums

http://www.home.agilent.com/owc_discussions/forum.jspa?forumID=104

ADS 2012 Quick Start for New Users of ADS:

(sehr zu empfehlen)

http://wireless.agilent.com/flash/eesof/ADS_QuickStart_2012_for_NEWUSERS/player.html

Agilent Kanal auf YouTube:

http://www.youtube.com/user/AgilentEEsof

ADS Overview:

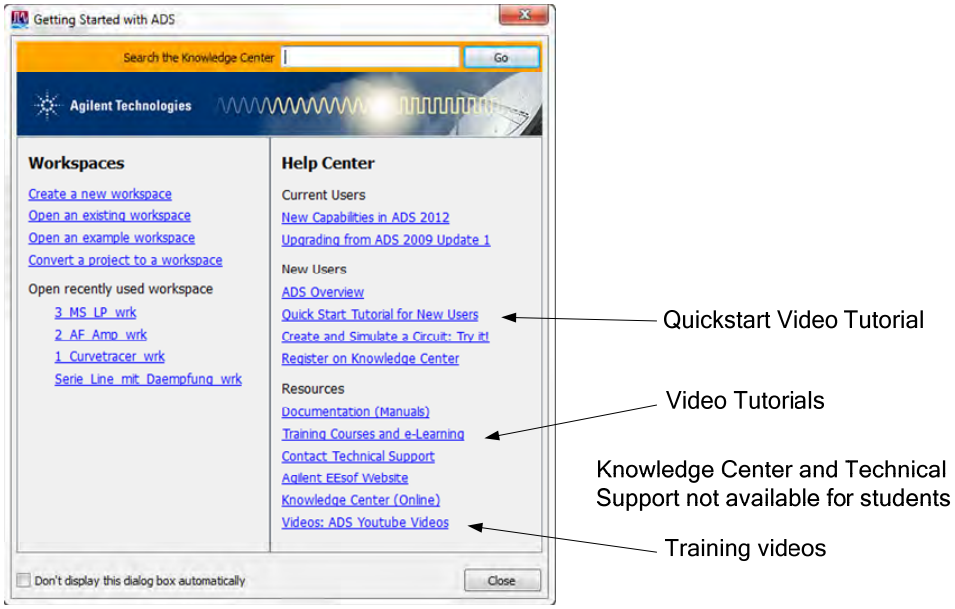
http://www.youtube.com/watch?v=i0IUAZ8GNyw&list=PLD918C8711CD75559&index=1

Agilent Video Homepage:

(Videos zu älteren ADS-Versionen)

“The Basics of Advanced Design System”:
http://wireless.agilent.com/vcentral/viewvideo.aspx?vid=338
weitere Videos:
http://wireless.agilent.com/vcentral/ suchen nach ADS oder Advanced Design

Direkte Links aus dem ADS Welcome-Window:



2. The ABC of ADS

A very short step by step Quickstart for first time users.

This gives you a first feeling of this powerful software package, without claiming completeness or details. For more detailed information see Help documentation.
After launching ADS and open or create a new workspace there are 3 main steps to get simulation results:

A Draw schematic

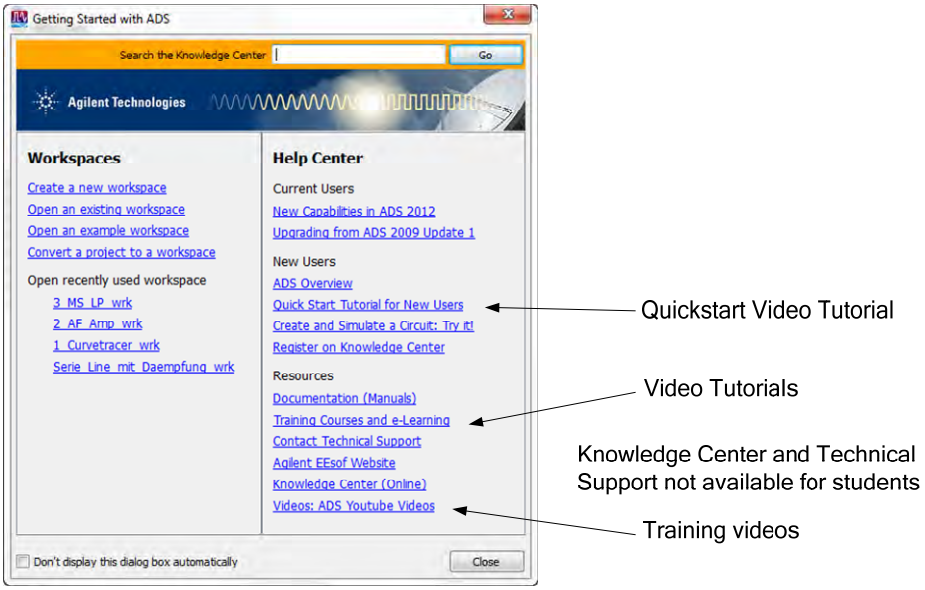
B Setup and run simulation

C Display data from simulation

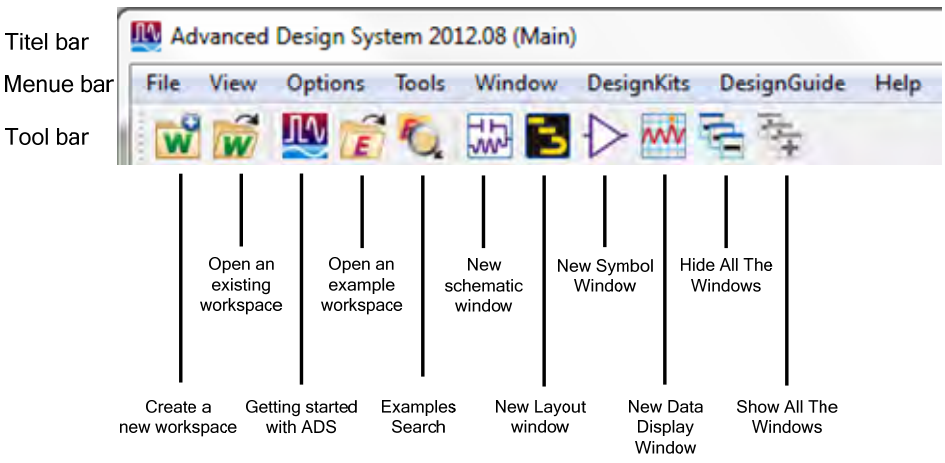
2.1. Main Window

After launching ADS 2012:

Startup Wizard:



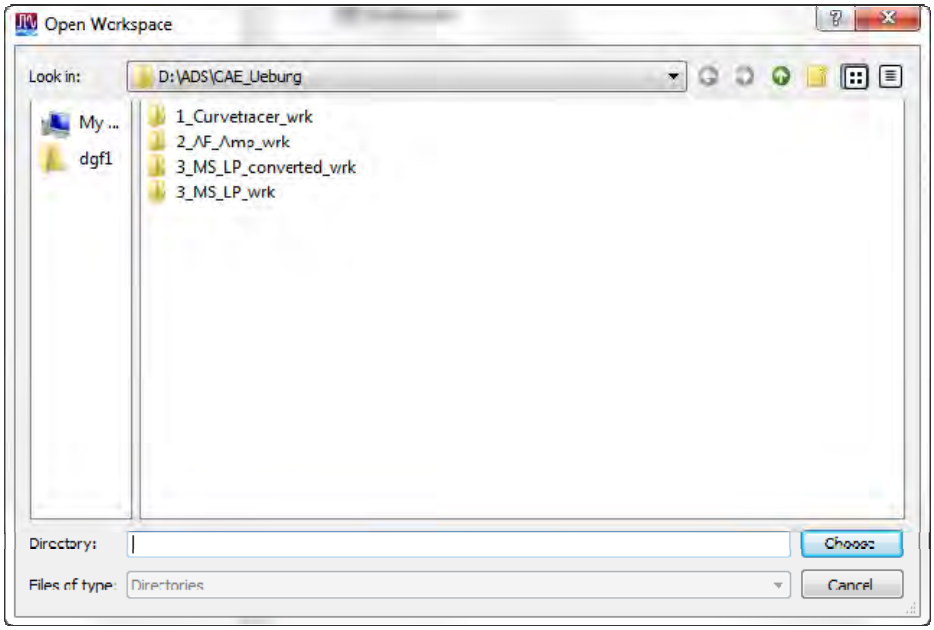
Main window:



2.1.1. Opening an existing workspace:

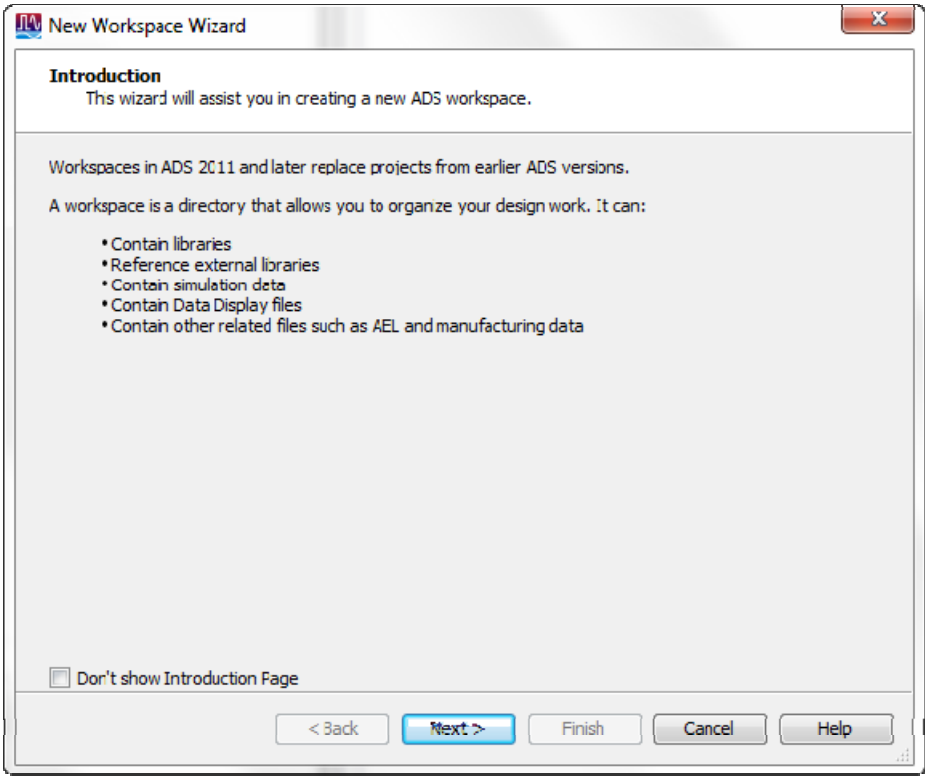
File > Open > Workspace or 

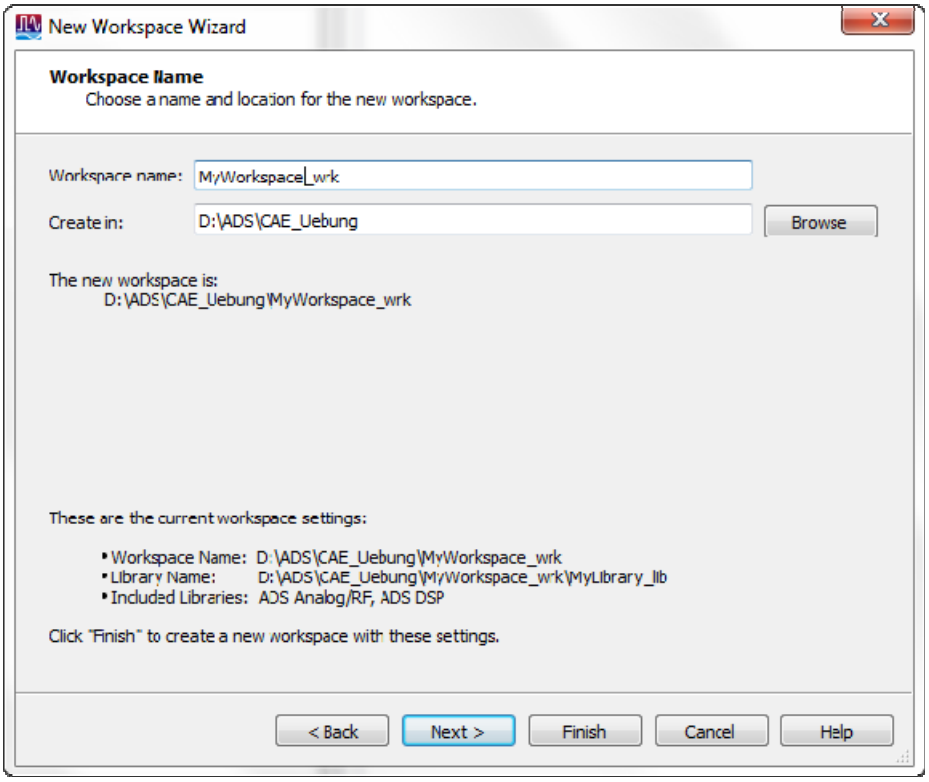
When the directory from which you launch the program contains workspaces, they are listed in the main window. Browse to the desired directory and choose workspace to open.

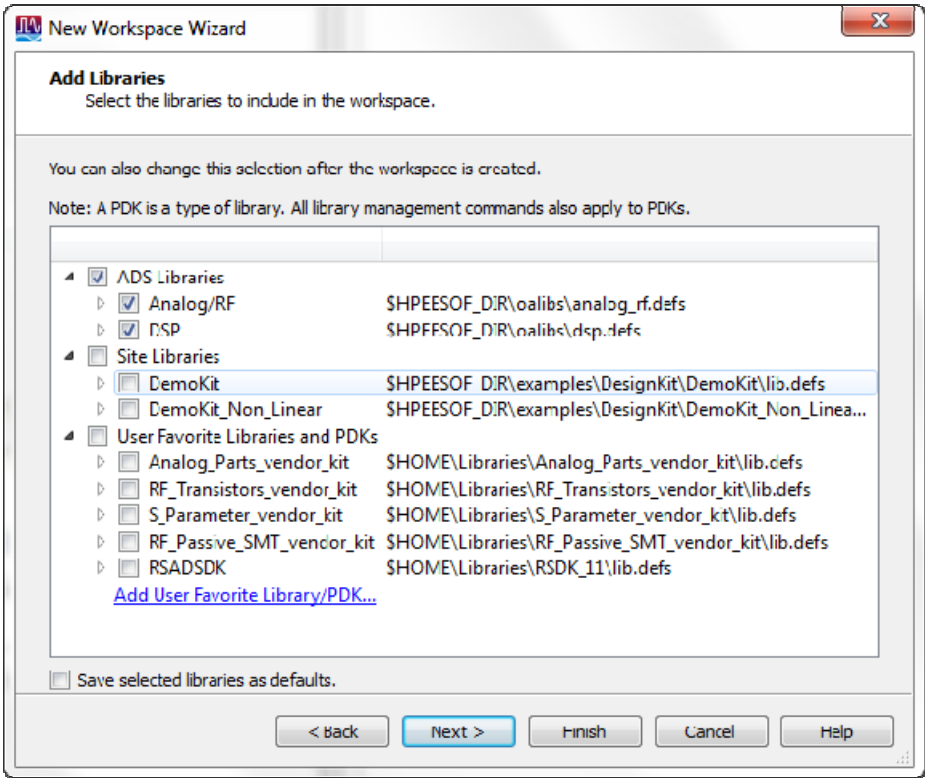


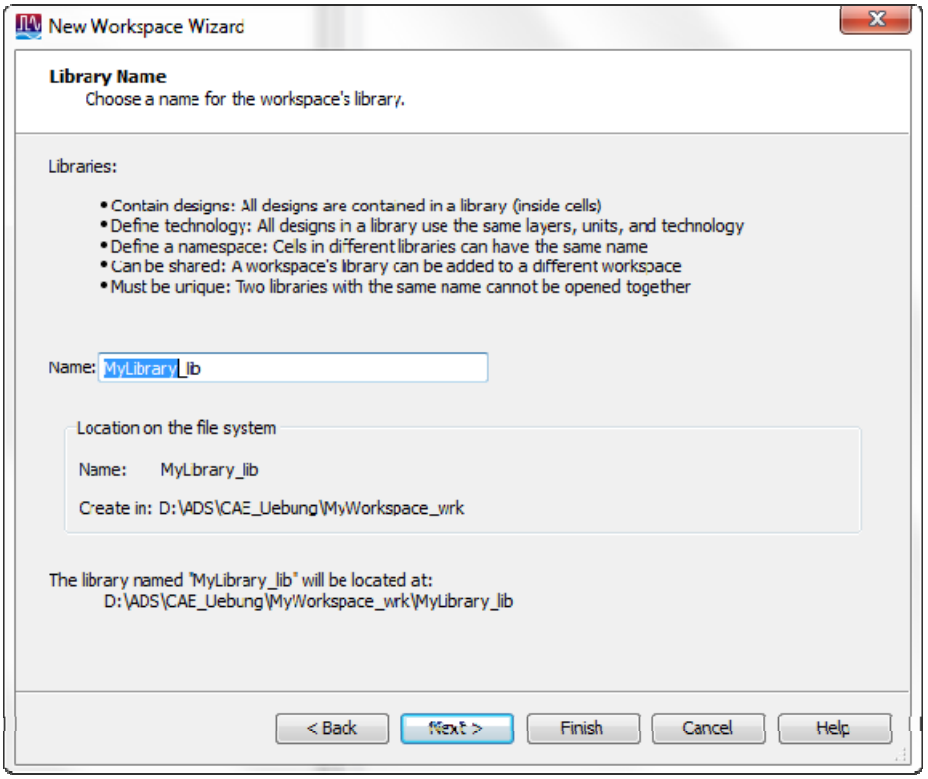
2.1.2. Creating a new workspace:

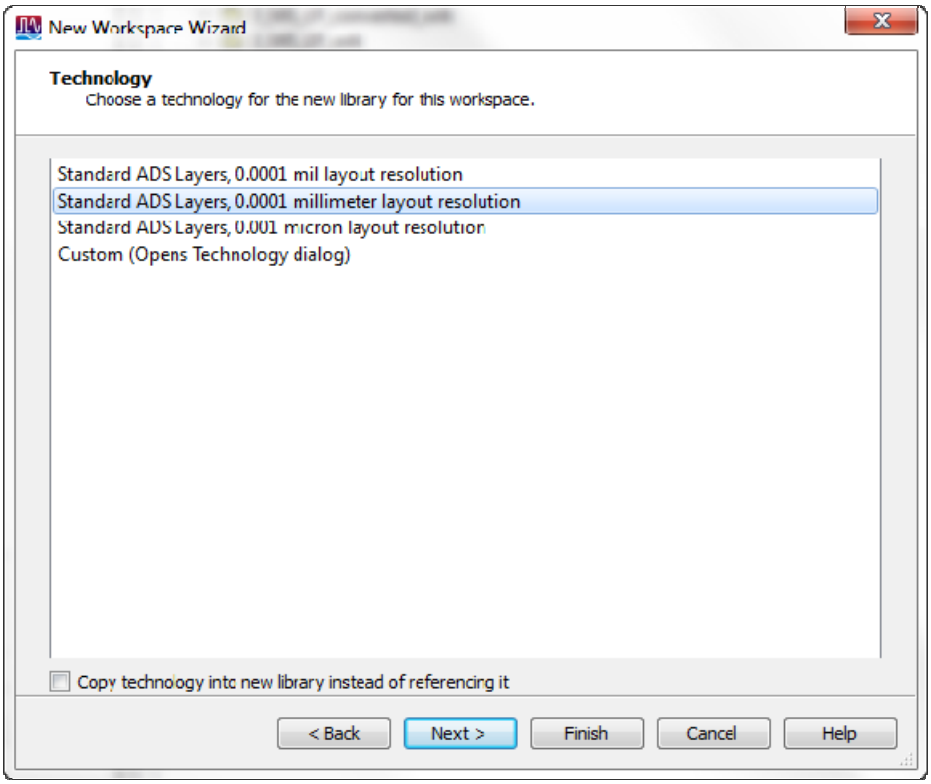
File > New > Workspace or 

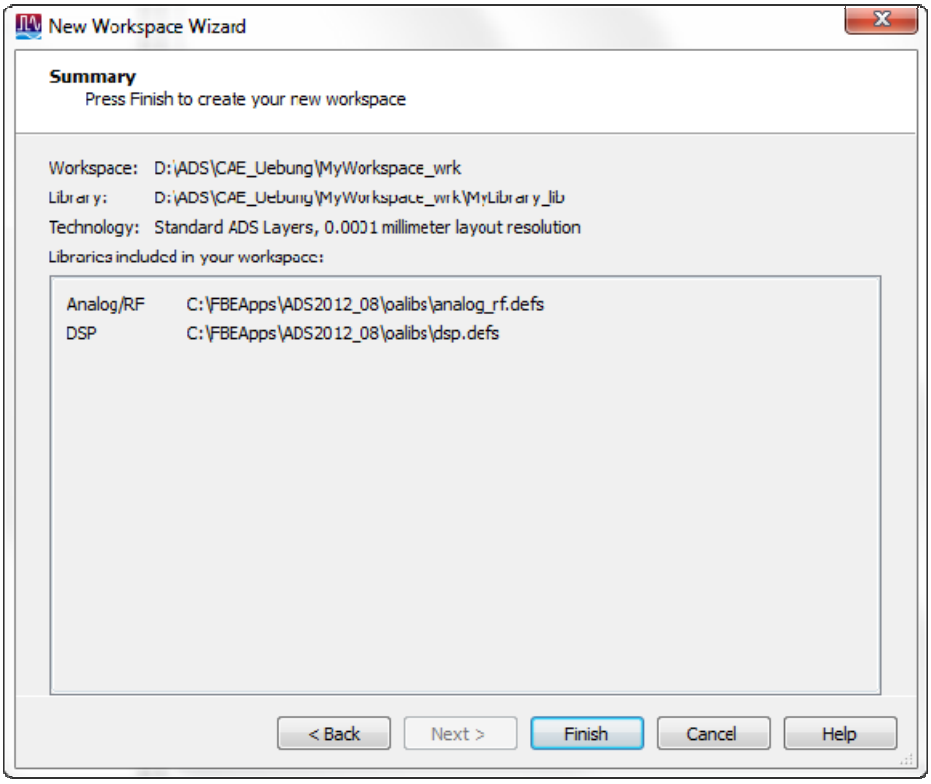


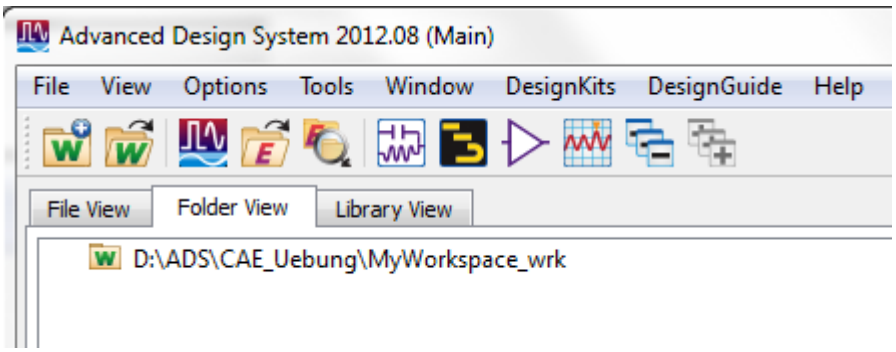

New Workspace Wizard
Workspace Name
Choose a name and location for the new workspace.
Workspace name: MyWorkspace_wrk
Create in: D:\ADS\CAE_Uebung
Browse
The new workspace is:
D:\ADS\CAE_Uebung\MyWorkspace_wrk
These are the current workspace settings:
Workspace Name: D:\ADS\CAE_Uebung\MyWorkspace_wrk
Library Name: D:\ADS\CAE_Uebung\MyWorkspace_wrk\MyLibrary_lib
Included Libraries: ADS Analog/RF, ADS DSP
Click "Finish" to create a new workspace with these settings.
< Back
Next >
Finish
Cancel
Help


New Workspace Wizard
Add Libraries
Select the libraries to include in the workspace.
You can also change this selection after the workspace is created.
Note: A PDK is a type of library. All library management commands also apply to PDKs.
ADS Libraries
Analog/RF $HPEESOF_DIR\oalibs\analog_rf.defs
DSP $HPEESOF_DIR\oalibs\dsp.defs
Site Libraries
DemoKit $HPEESOF_DIR\examples\DesignKit\DemoKit\lib.defs
DemoKit_Non_Linear $HPEESOF_DIR\examples\DesignKit\DemoKit_Non_Linea...
User Favorite Libraries and PDKs
Analog_Parts_vendor_kit $HOME\Libraries\Analog_Parts_vendor_kit\lib.defs
RF_Transistors_vendor_kit $HOME\Libraries\RF_Transistors_vendor_kit\lib.defs
S_Parameter_vendor_kit $HOME\Libraries\S_Parameter_vendor_kit\lib.defs
RF_Passive_SMT_vendor_kit $HOME\Libraries\RF_Passive_SMT_vendor_kit\lib.defs
RSADSDK $HOME\Libraries\RSDK_11\lib.defs
Add User Favorite Library/PDK...
Save selected libraries as defaults.
< Back
Next >
Finish
Cancel
Help


New Workspace Wizard
Library Name
Choose a name for the workspace's library.
Libraries:
• Contain designs: All designs are contained in a library (inside cells)
• Define technology: All designs in a library use the same layers, units, and technology
• Define a namespace: Cells in different libraries can have the same name
• Can be shared: A workspace's library can be added to a different workspace
• Must be unique: Two libraries with the same name cannot be opened together
Name: MyLibrary_lib
Location on the file system
Name: MyLibrary_lib
Create in: D:\ADS\CAE_Uebung\MyWorkspace_wrk
The library named 'MyLibrary_lib' will be located at:
D:\ADS\CAE_Uebung\MyWorkspace_wrk\MyLibrary_lib
< Back
Next >
Finish
Cancel
Help


New Workspace Wizard
Technology
Choose a technology for the new library for this workspace.
Standard ADS Layers, 0.0001 mil layout resolution
Standard ADS Layers, 0.0001 millimeter layout resolution
Standard ADS Layers, 0.001 micron layout resolution
Custom (Opens Technology dialog)
Copy technology into new library instead of referencing it
< Back
Next >
Finish
Cancel
Help





Names:
Do not use dash (-), use underscore (_) instead.
Do not use vowel mutation (Umlaut), use „ue“, „ae“, „oe“ instead.

Never use spaces or dashes in any names. Use underscore instead.

2.1.3. Copying a workspace:

Use Window Explorer to copy workspaces.

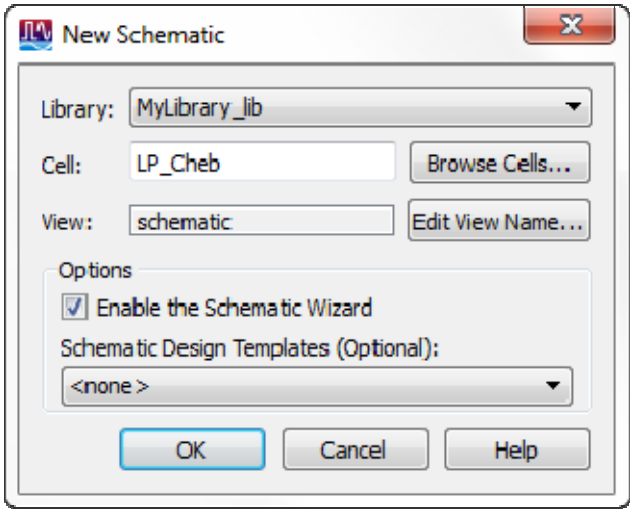
2.1.4. Covert a project from ADS 2009 or older to a workspace:

File > Convert Project to Workspace

2.2. Step A: Draw Schematic

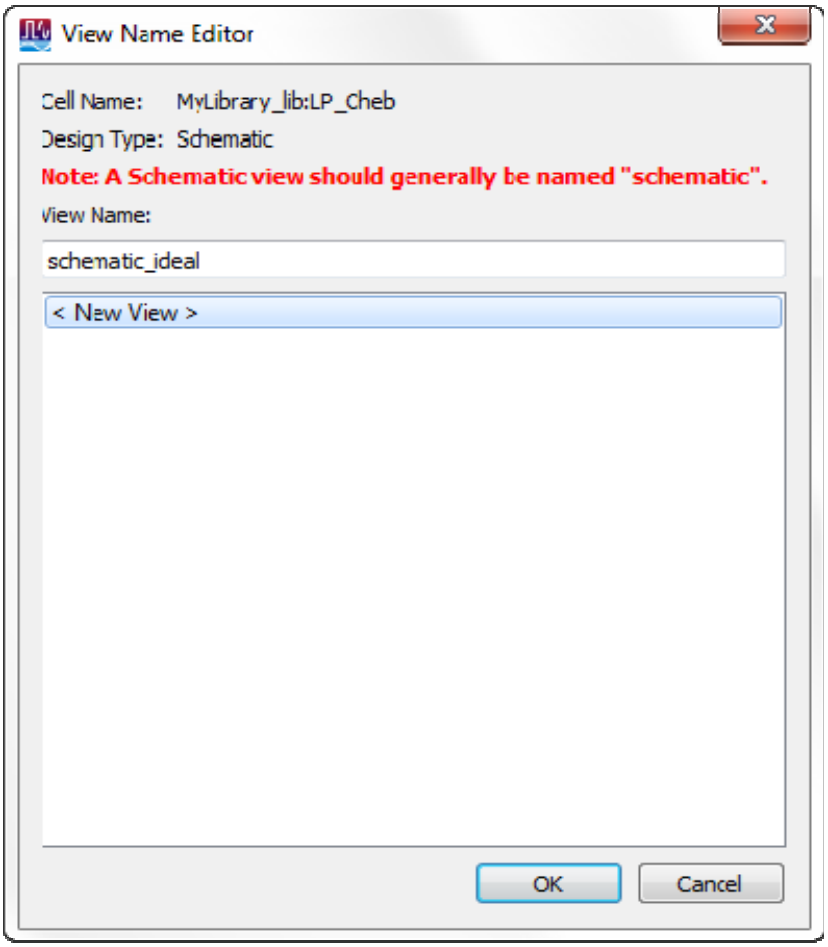
Creating a new schematic view

In Main window: **File > New > Schematic** or 

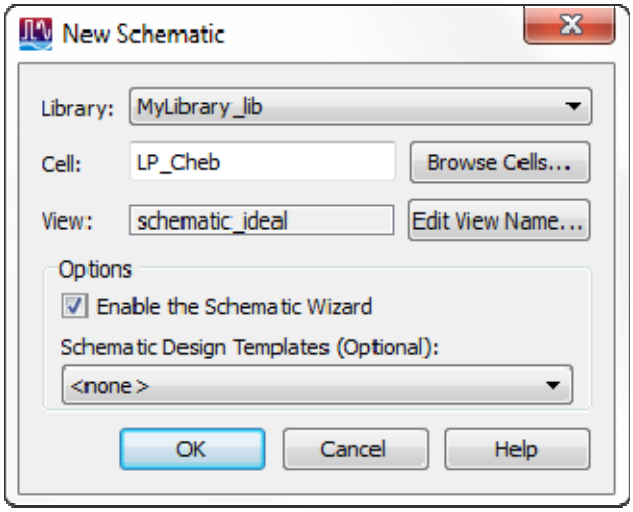


Enter an appropriate name for the cell.

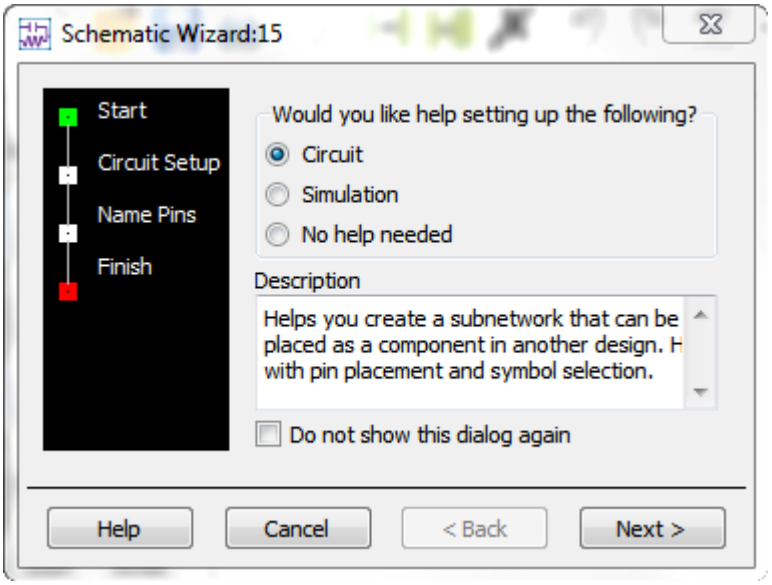
Edit View Name to “schematic_ideal”.



Click OK.



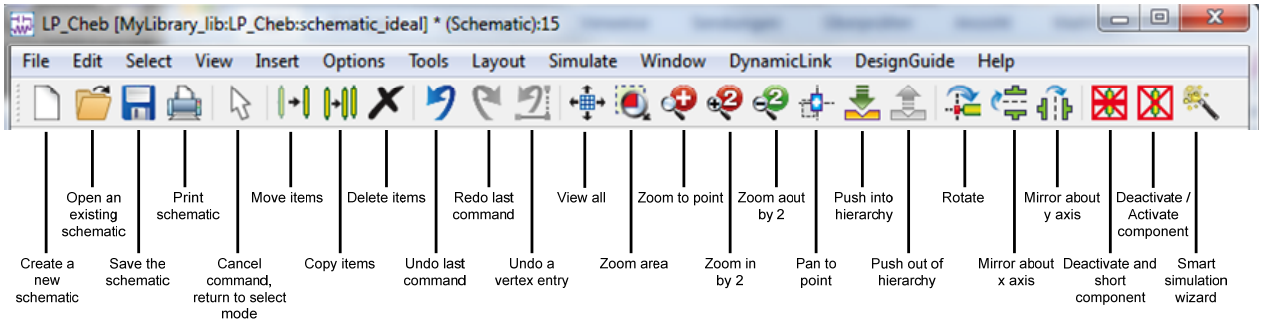
Click OK.

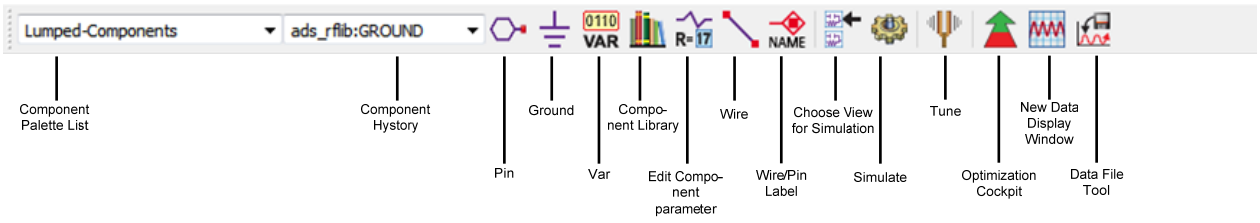


Cancel the schematic wizard.

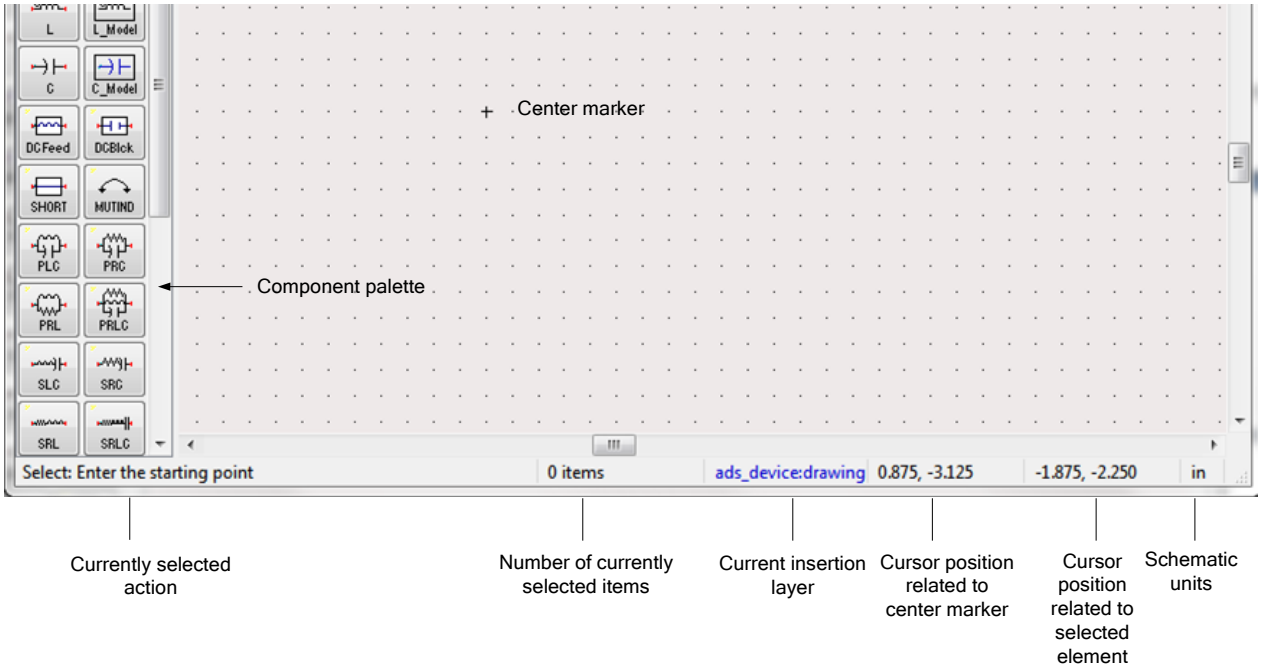
An empty schematic named **schematic_ideal** opens and is ready to draw the schematic.

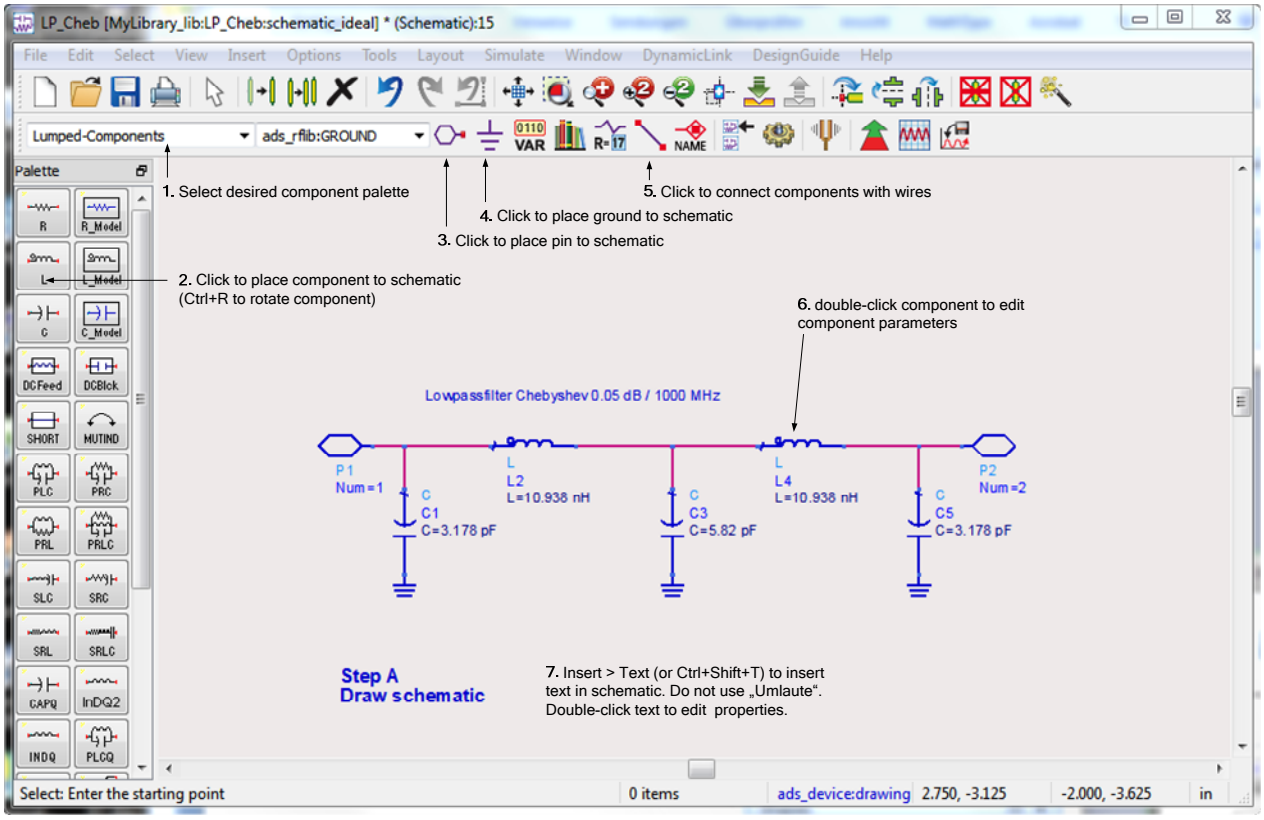
Schematic Window





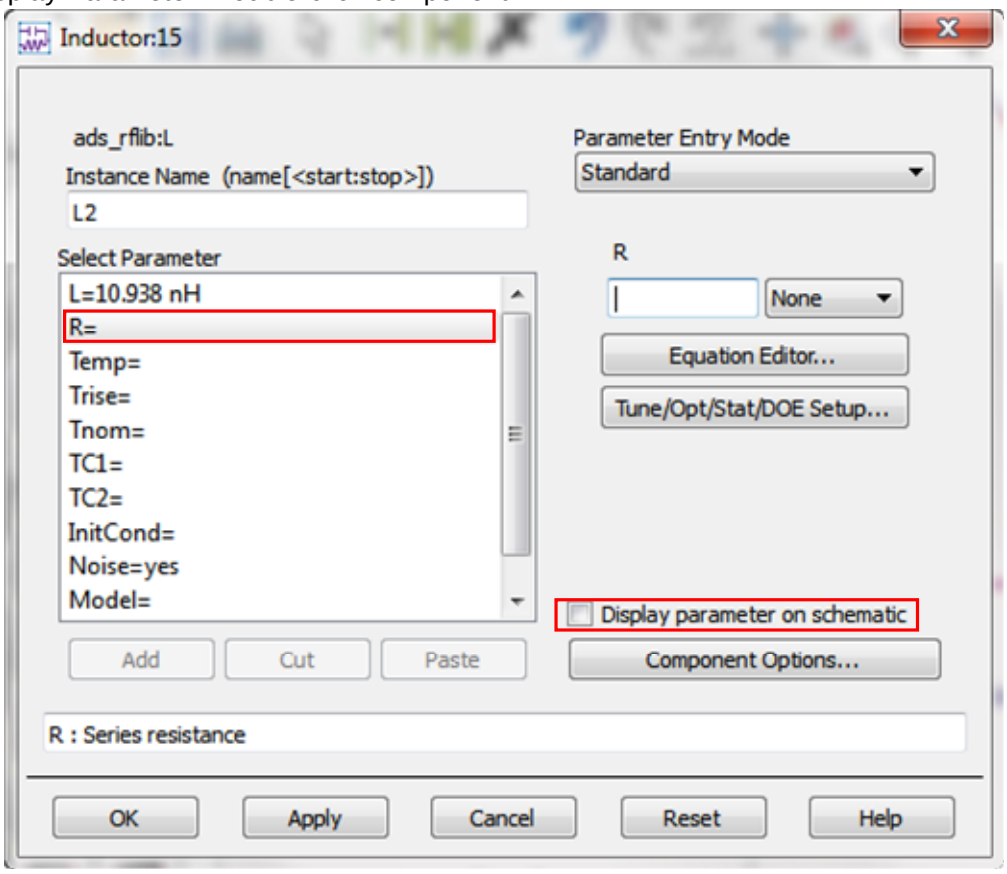
Toolbars can be customized by adding or deleting icons in menu "**Tools > Hot Key/Toolbar Configuration > Toolbar >Toolbar Groups**".





2.2.1. Component Parameters

Hide/Display Parameter: Double-click component



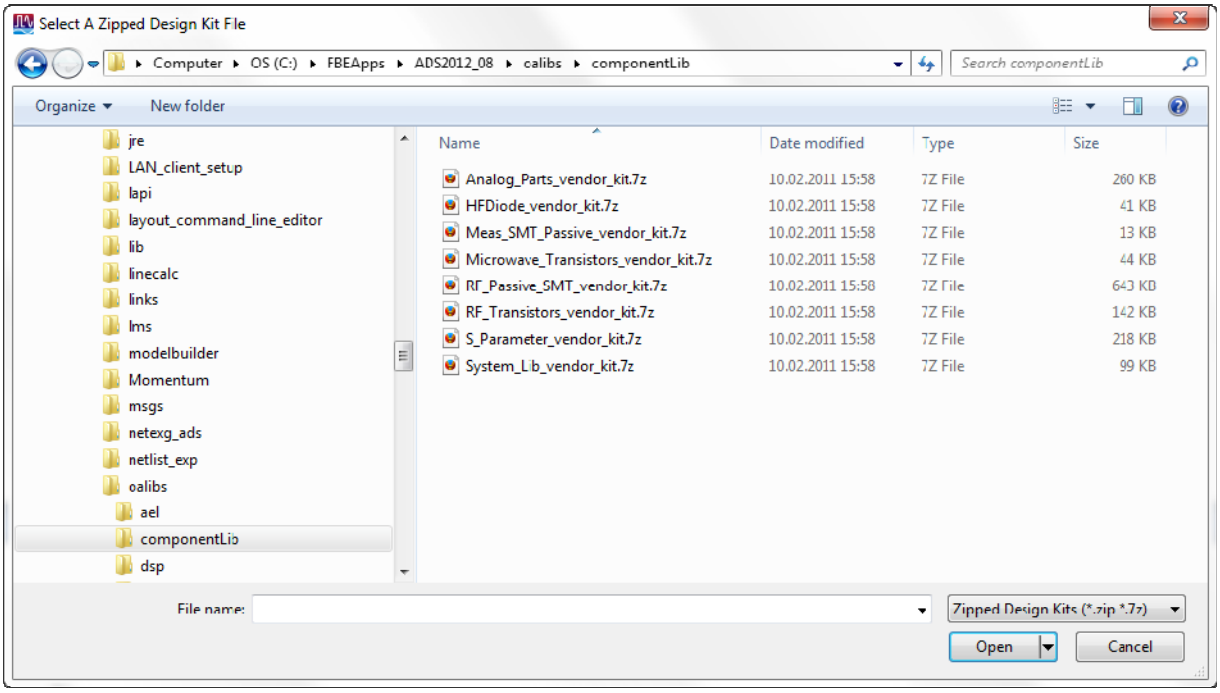
Move component text: **Edit > Text > Move Component Text**, or F5 or right click on component

2.2.2. Vendor Component Libraries

Supplied with ADS 2012. To find in Installation-directory of ADS2012 \oalibs\componentLib.

RF Passive SMT Library	SMT Inductors, Capacitors, Resistors
muRata Manufacturing Measurement-Based SMT Library	muRata SMT Inductors, Capacitors
RF Transistor Library	Packaged, BJTs, GaAsFETs, RF Power MOSFETs, nonlinear
Microwave Transistor Library	Chip BJTs, Chip GaAsFETs, Chip+Packaged HEMTs, nonlinear
High-Frequency Diode Library	Diodes, nonlinear
Analog Parts Library	Diodes, BJTs, MOSFETs, JFETs, no package, nonlinear
S-Parameter Transistor Library	Diodes, Transistors, no package, linear S-Par
System Library	Amplifiers, Filters, Cristals, Mixers, Switches, etc.

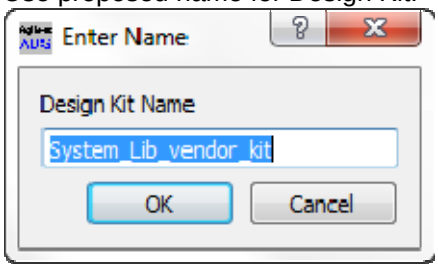
Add a vendor component library to your workspace:
Main window: **DesignKits > Unzip Design Kit**
Browse to installation-directory of ADS2012 \oalibs\componentLib and select desired library.



Click **Open**.

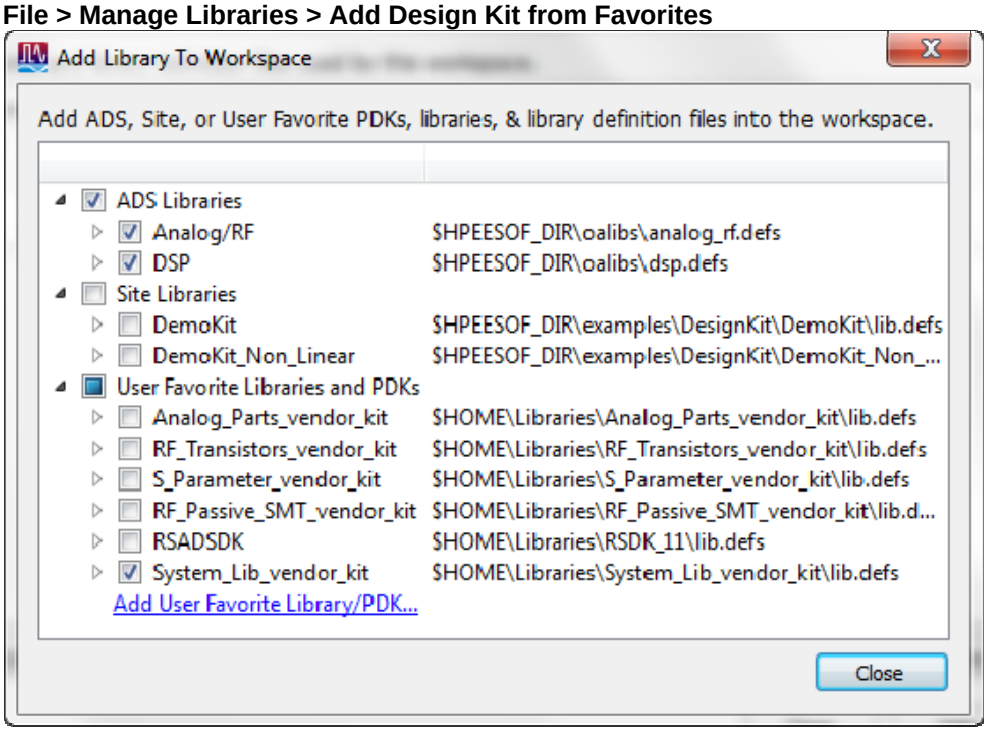
Directory to unzip file: Use directory “Libraries” in startup directory of ADS. If this directory does not exist, create it.

Use proposed name for Design Kit.



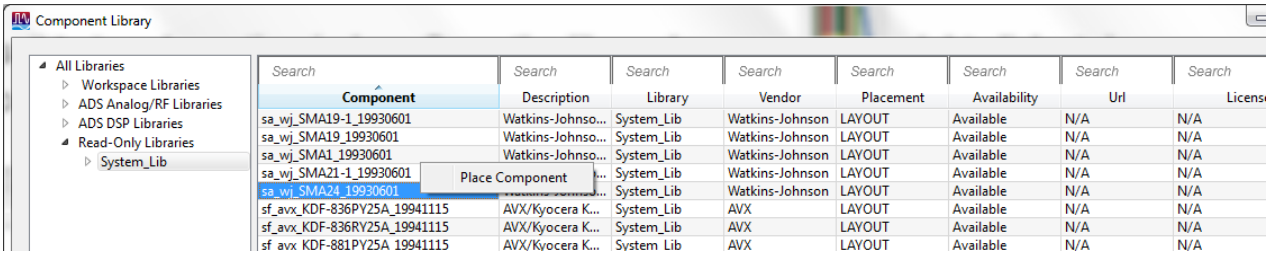
Check Libraries in Workspace:

File > Manage Libraries > Add Design Kit from Favorites



Close all.

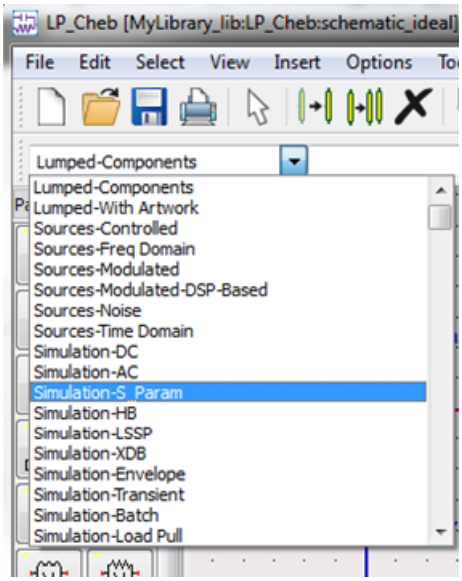
The components are accessible in schematic window. Open the library browser  , right click at desired component to place component in schematic.



2.3. Step B: Setting Up and Launching a Simulation

Setting Up a Circuit Simulation

A simulation is set up in a schematic window, using sources and simulators available in the Palette List. Simulator components are placed in the Schematic window. Simulation data are stored into a dataset.



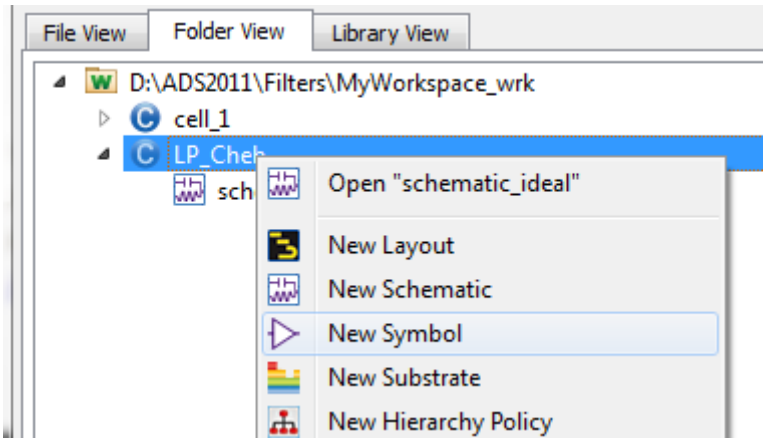
Source palettes

Simulation palettes

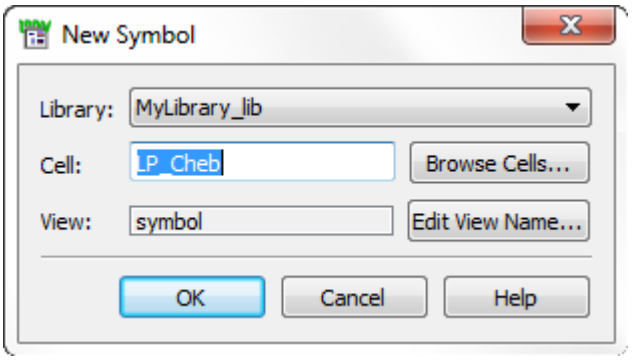
You could just add a simulation controller and terminations directly on this schematic and run a simulation. However, I think it is a better practice to instead treat this as a subcircuit that we place into a separate “test bench” schematic. This allows to use several test benches with different simulations on the same circuit. In order to do this, we must create a **Symbol** view.

Creating a symbol view

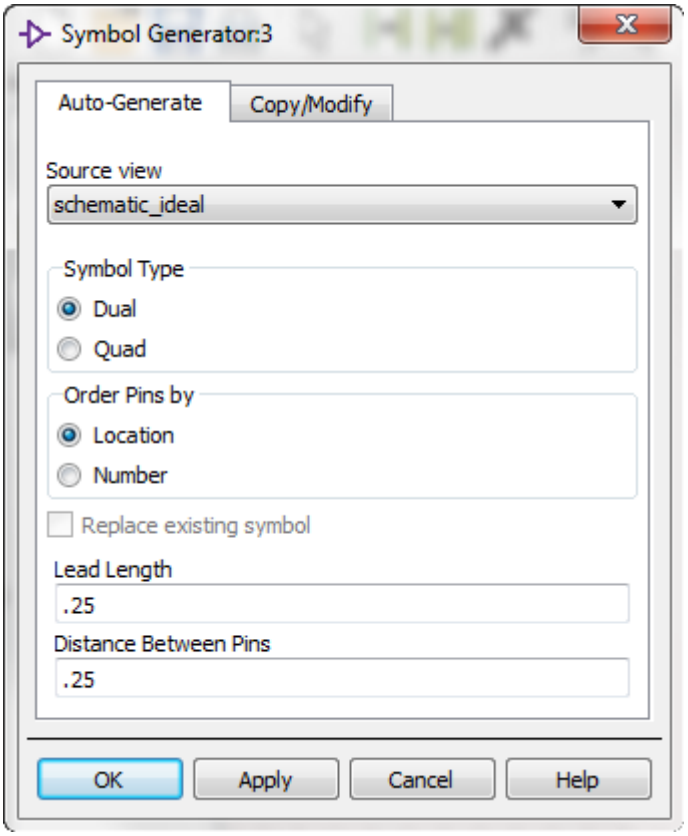
In the ADS Main Window, highlight the LP_Cheb cell and right click to select **New Symbol**.



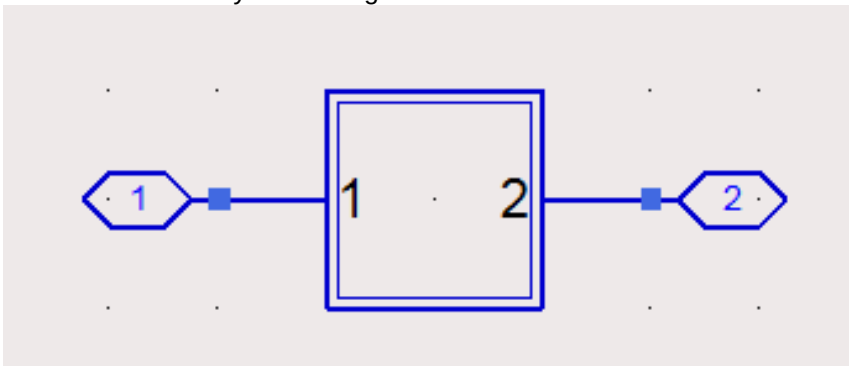
The defaults in the **New Symbol** dialog are good, so just click **OK**.



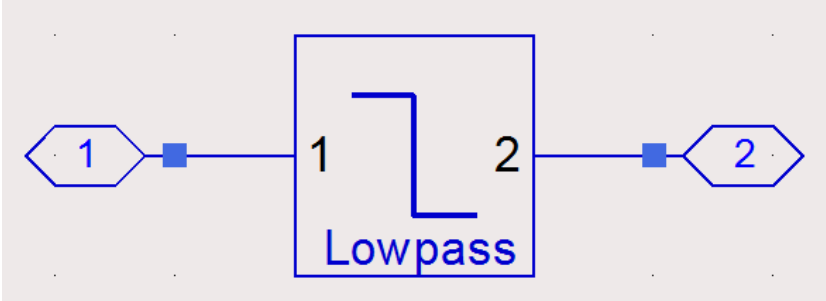
The defaults in the **Symbol Generator** are fine. Click **OK**.



This is the default symbol that gets created.



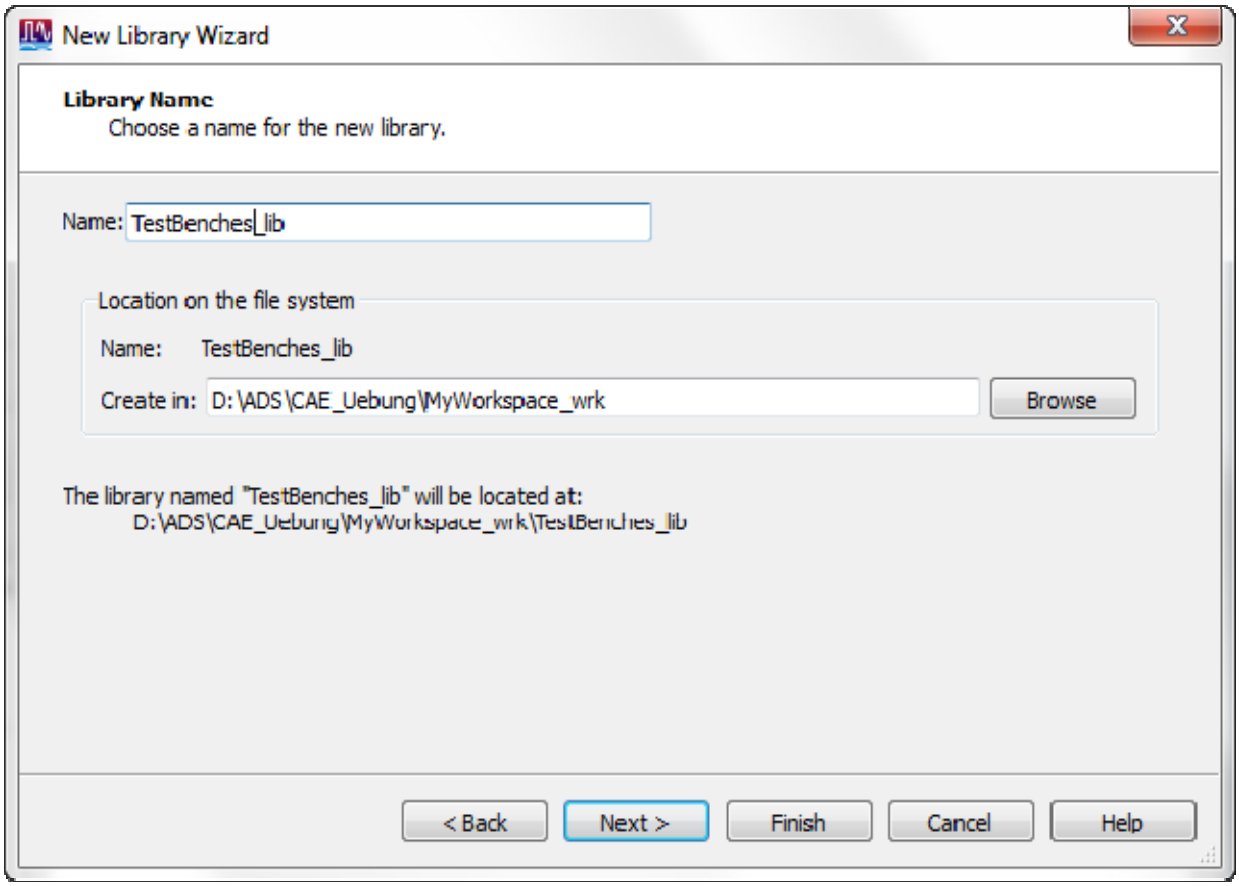
Edit the symbol to your choices or use one of the predefined symbols from ADS (see 5.4. Create New Symbol).



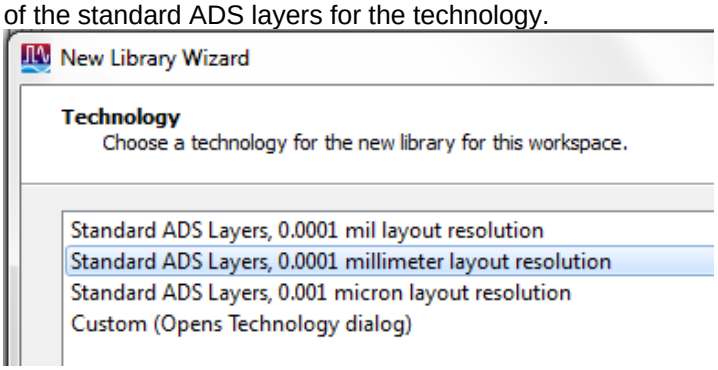
Now we have a design in a schematic and a symbol for it. Next, we will create a “test bench” to simulate its performance.

Creating a library of test benches

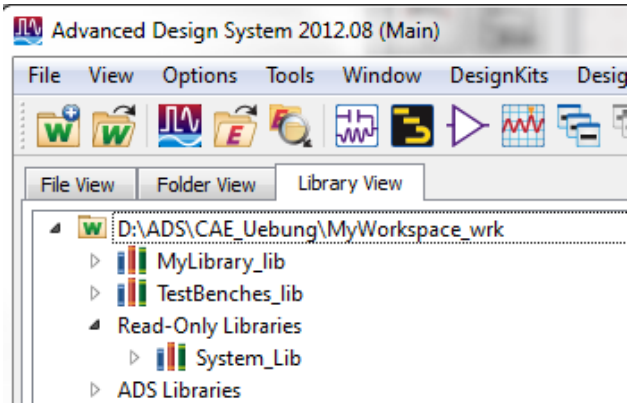
Assuming there may be many test benches, we will create a test bench library. From the ADS Main Window, select **File > New > Library…**



Click **Next >.** Since we intend for this library to only contain schematics for simulations, we will select one of the standard ADS layers for the technology.

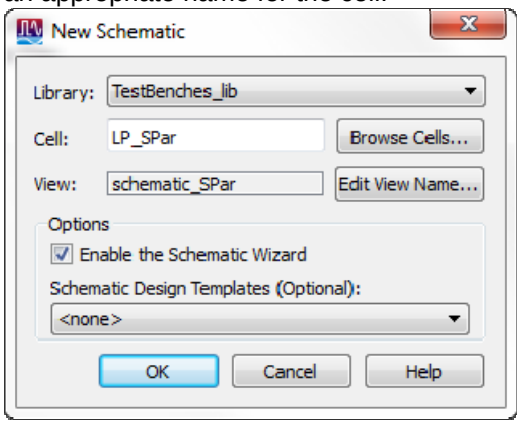


Click **Next >** then **Finish**.



Creating a new schematic view to run a simulation

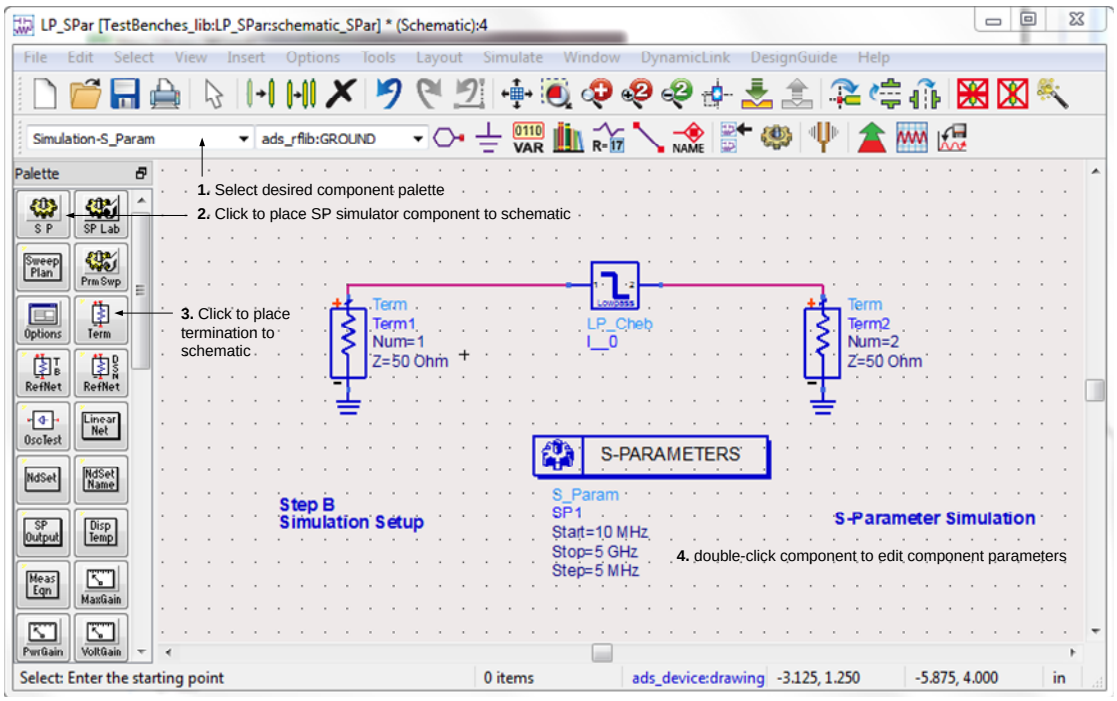
Now we will create a cell within this new library. In the ADS Main Window select the **schematic** icon enter an appropriate name for the cell.



Click **OK**, and a blank schematic and the schematic wizard opens. Cancel schematic wizard.

From the ADS Main Window, you can just drag the **symbol** view from **LP_Cheb-cell** into this schematic.

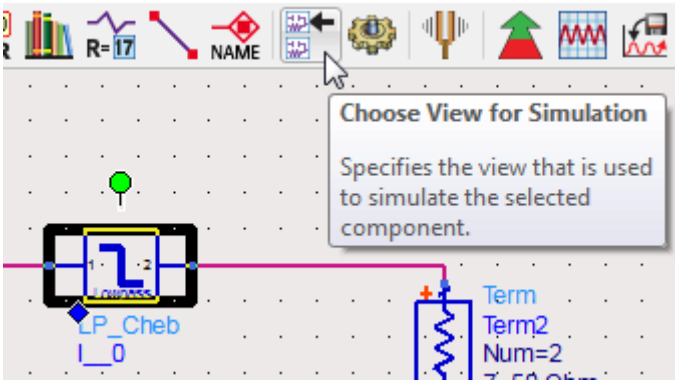
Place SP Simulation component and termination component to schematic. Set frequency sweep settings as desired.



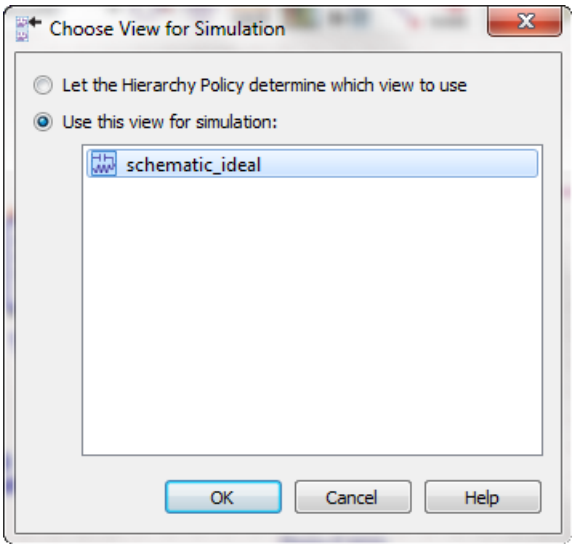
Alternatively to manual placing simulator and terminations you can use a template. From the schematic select **Insert > Template…** In the **Insert Template** dialog, highlight the **S_Params** template and click **OK.** Adjust the frequency sweep settings as desired.

Specifying the view to simulate

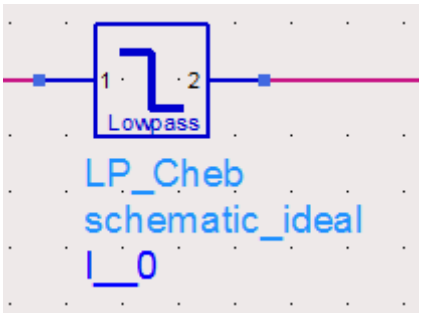
Click on the **LP_Cheb** symbol to select it. Next click on the **Choose View for Simulation** icon  .



In the **Choose View for Simulation** dialog, click on **Use this view for Simulation**.



Click **OK**. Note that **schematic_ideal** now appears under the **LPF** symbol in the schematic.



Launching the Simulation

You can launch the simulation from the schematic window **Simulate > Simulate**, or from the **Simulation Setup** dialog box, or **F7**, or with  .

The simulation runs quickly and opens a data display. Note that this data display file is not saved in your workspace unless you specifically save it.

Note By default, all simulations in the schematic window are run.

DC Bias Back Annotation

Immediately following a DC simulation:

Simulate > DC Solution displays current and voltage data at the ports of active devices and lumped elements.

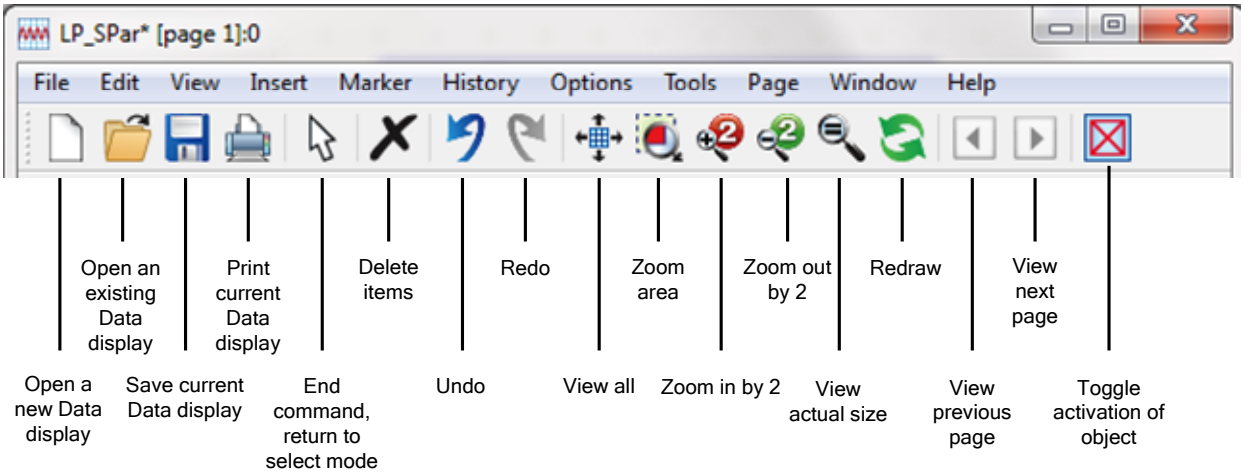
Simulate > Detailed Device Operating Point displays a detailed listing of operating points for all parameters of a selected device.

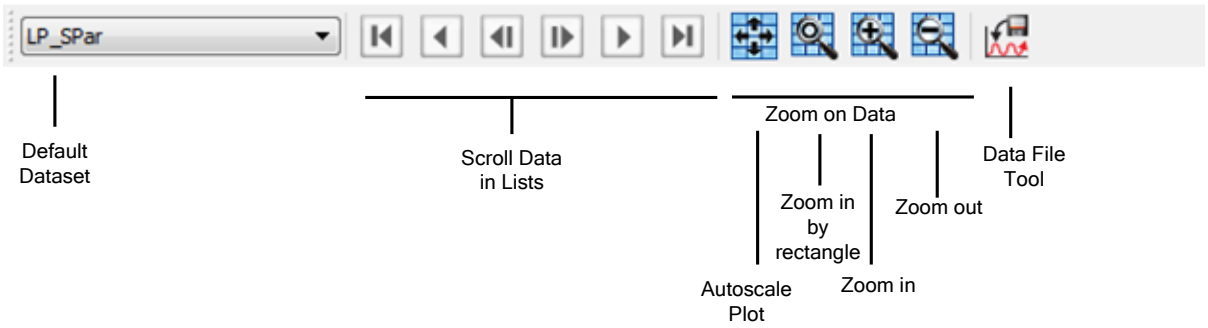
Simulate > Brief Device Operating Point displays a brief listing of operating points for common parameters of a selected device.

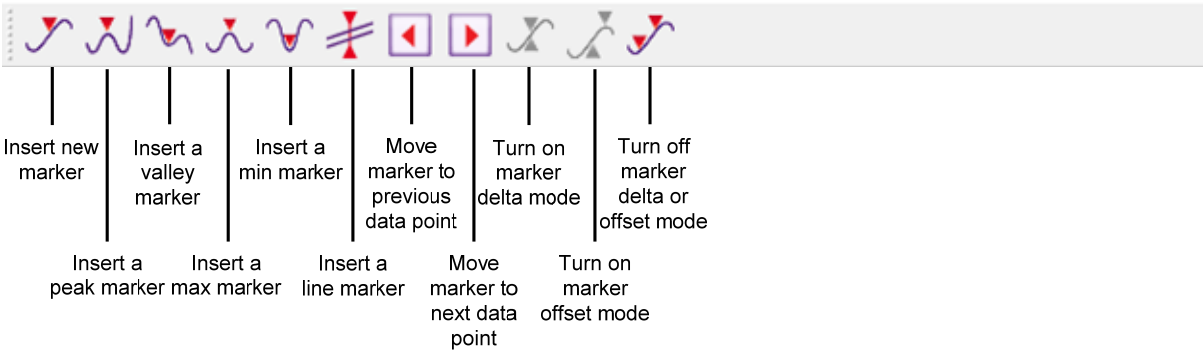
Simulate > Clear DC Annotation removes the current and voltage text produced by DC Solution.

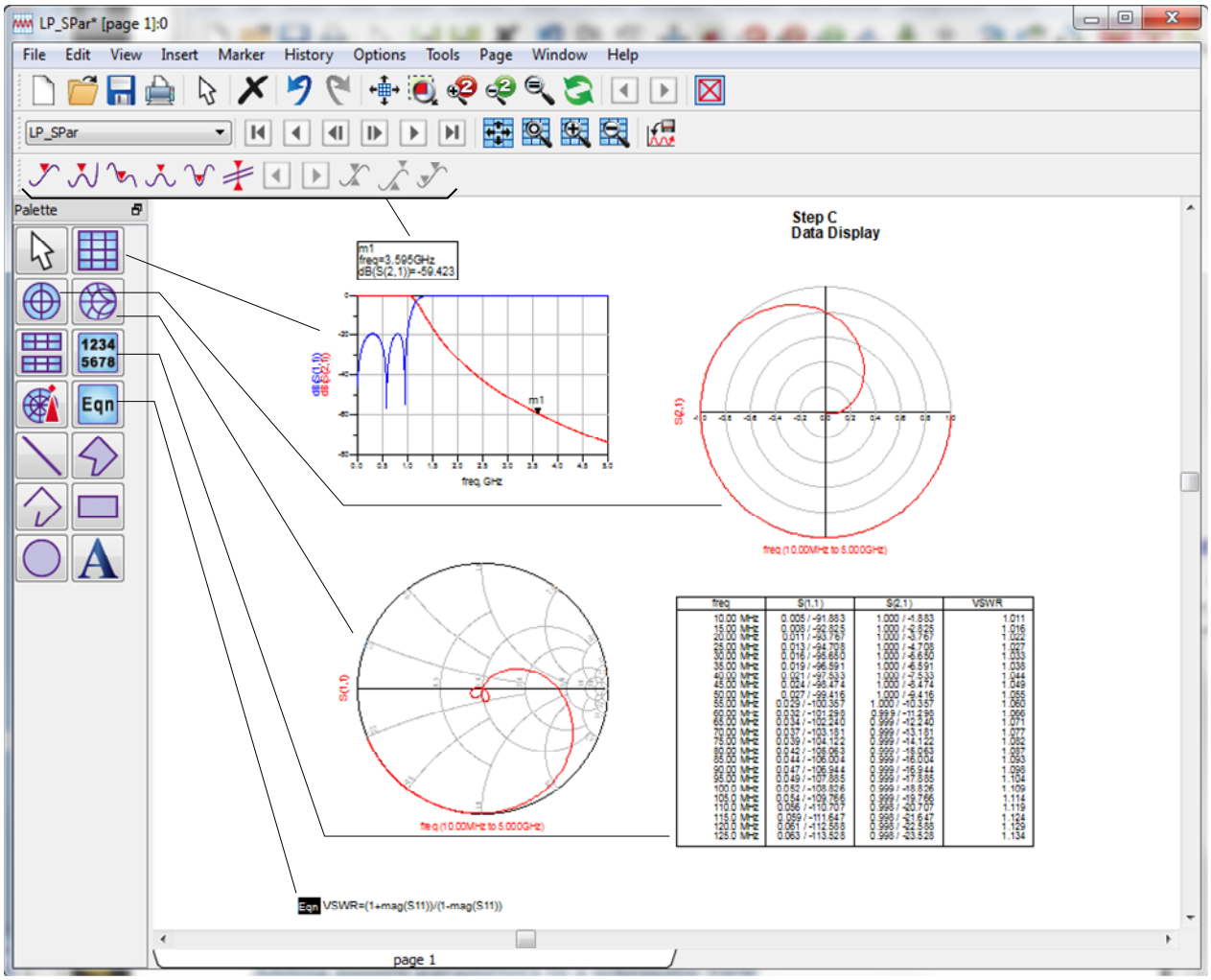
2.4. Step C: Display Simulated Data

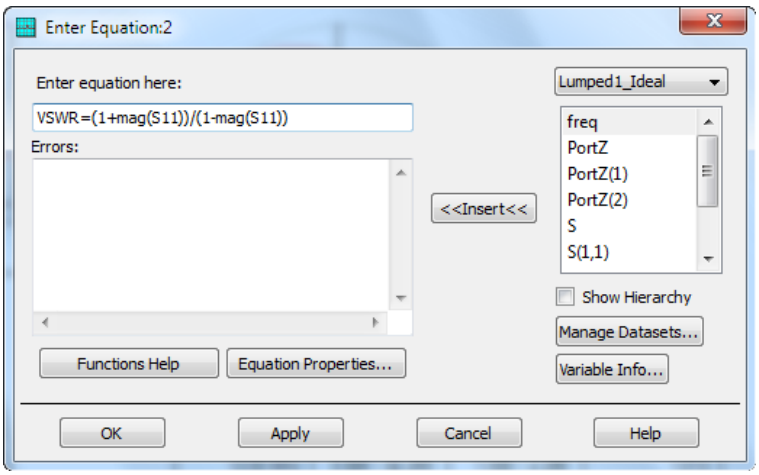
Data Display Window




LP_SPar
Default Dataset
Scroll Data in Lists
Zoom on Data
Autoscale Plot
Zoom in by rectangle
Zoom in
Zoom out
Data File Tool

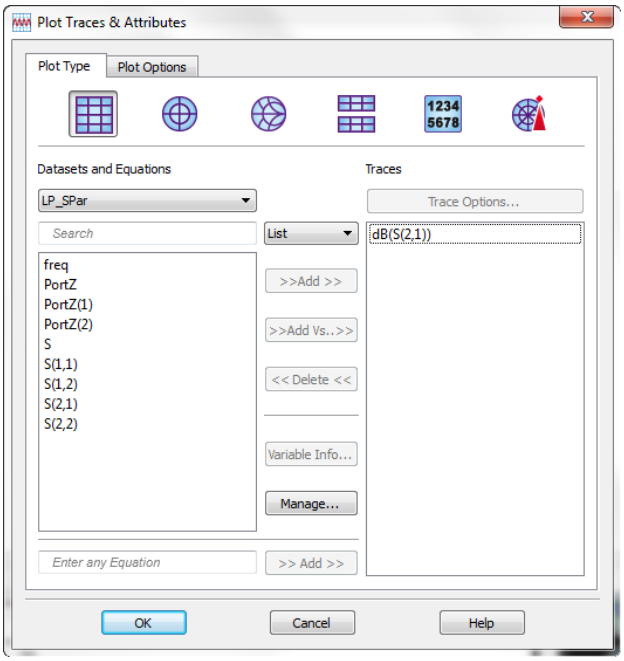

Insert new marker
Insert a peak marker
Insert a valley marker
Insert a max marker
Insert a min marker
Insert a line marker
Move marker to previous data point
Move marker to next data point
Turn on marker delta mode
Turn on marker offset mode
Turn off marker delta or offset mode


LP_SPar* [page 1]:0
File Edit View Insert Marker History Options Tools Page Window Help
LP_SPar
Palette
Step C
Data Display
m1
freq=3.595GHz
dB(S(2,1))=-59.423
freq, GHz
S(2,1)
S(1,1)
freq (10.00MHz to 5.000GHz)
freq
S(1,1)
S(2,1)
VSWR
Eqn VSWR=(1+mag(S11))/(1-mag(S11))
page 1

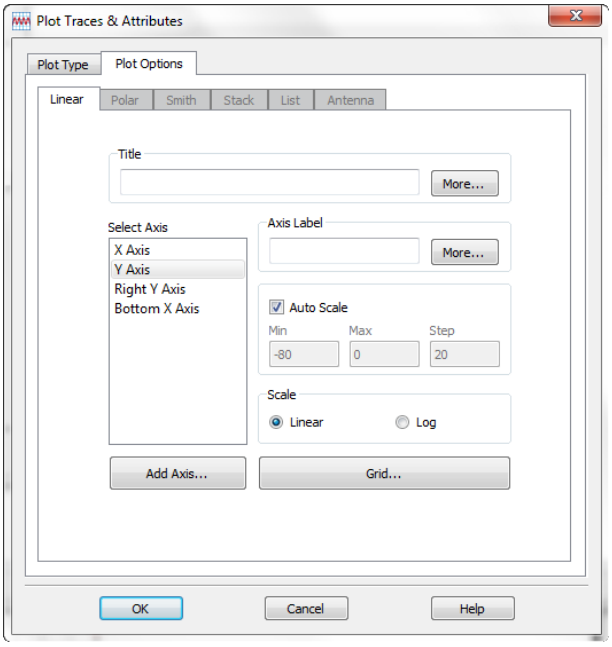


Doubleclick to Equation

S11, S(11) and S(1,1) are equivalent



Doubleclick to plot



Plot options

3. ADS 2012 Design Environment

Advanced Design System (ADS) 2012 provides a complete new *design environment* (see "Advanced Design System Quick Start" adstour.pdf) and terminology for existing ADS users. ADS Projects are replaced with workspaces, designs are now Cells and Views. To use your existing ADS Projects and designs (created using earlier versions of ADS) with ADS 2012, you must upgrade them to an *ADS Workspace* (see "ADS Quick Reference" oaqkref.pdf).

ADS Workspace
ADS 2012 uses workspaces to store and organize the design work. A Workspace is an organizer where you can group everything about a design within— such as libraries, technology, schematic, layout, simulation data, and Momentum data. The entire ADS user interface and simulation operates within a currently opened workspace.

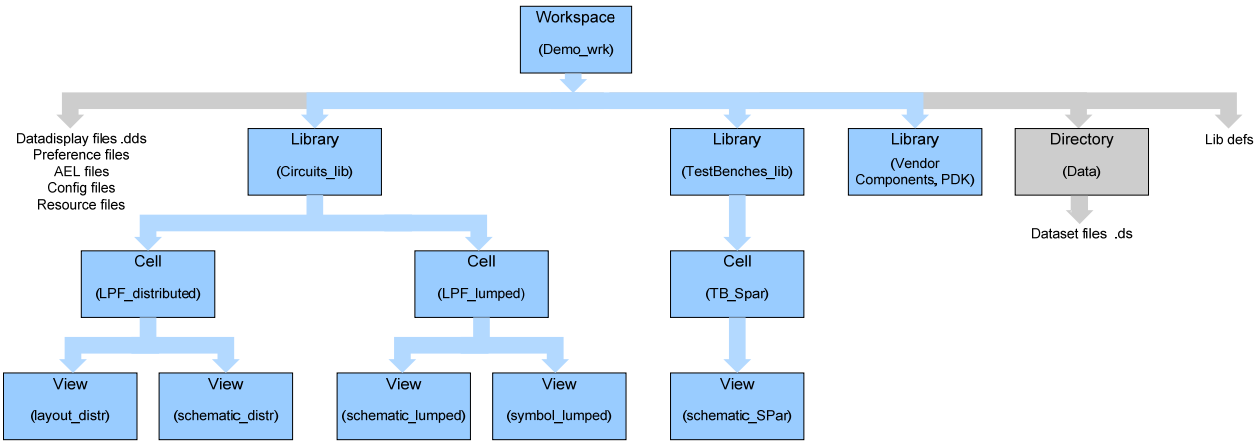
A Workspace includes:
Library: A Library is a directory that holds cells and a definition file, such as lib.defs. This file contains a summary of all the libraries selected into the given workspace and their mode of operation (Read-only, Non-Shared, or Shared). A Library also defines the technology (layers, resolution, and layout units) to be used by the Views created in that library.
Cell: A cell is a sub-directory that contains zero or more views.
.
View: A View is a sub-directory in a cell that stores the design work such as schematic, symbol, or layout, and can store other related data, such as an EM Setup.

Workspace Features
The following are the key features of an ADS workspace:

- Workspaces can have multiple libraries. Each Library can have multiple Cells, and each Cell can have multiple Views.
- Workspaces can be archived or un-archived.
- Workspaces contain data that includes simulation results, data display files, and other data files.

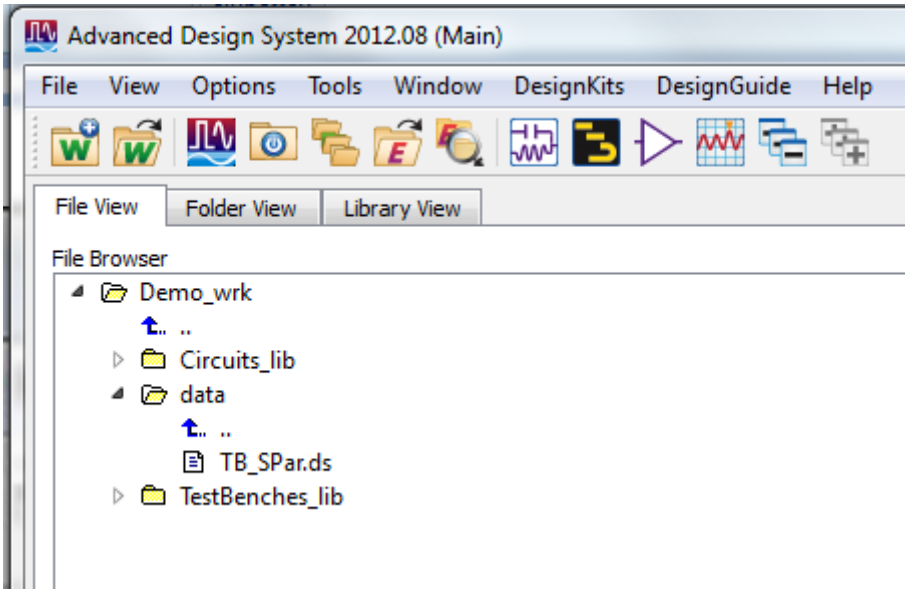
In an ADS Workspace,
1. A component name consists of the library name and cell name of the component.
2. A design name consists of the library name, cell name, and view name of the design.
3. Forms and Formsets are defined in a library.

The following figure displays the hierarchy of an ADS Workspace:



From the ADS Main window, the view options are provided for a workspace in three different tabs: "File View", "Folder View" and "Library View".
These tabs show a different view of the workspace and the loaded libraries. Changes made in any of these tabs (such as renaming or deleting a file or view) will affect all three tabs.

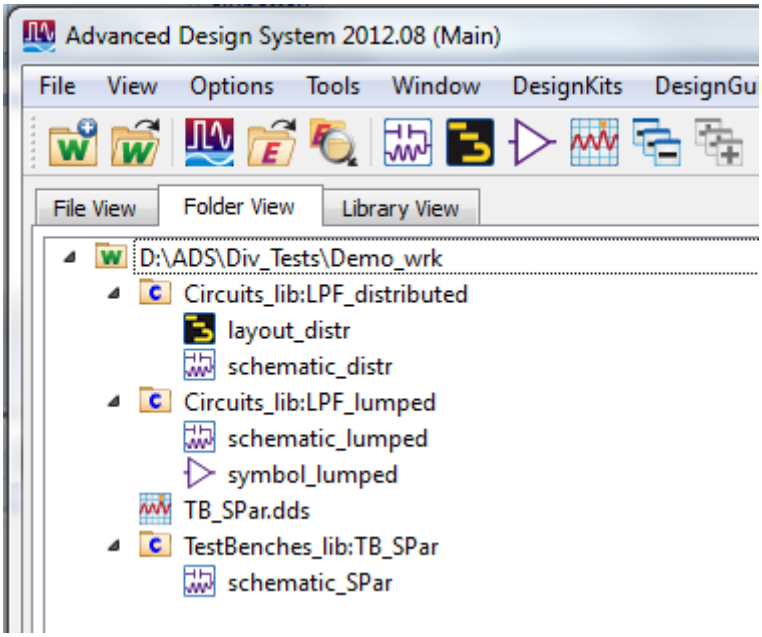
File View:



In the File View, you can:

- Browse to other directories (similar to previous releases)
- See the actual files that are stored in the file system

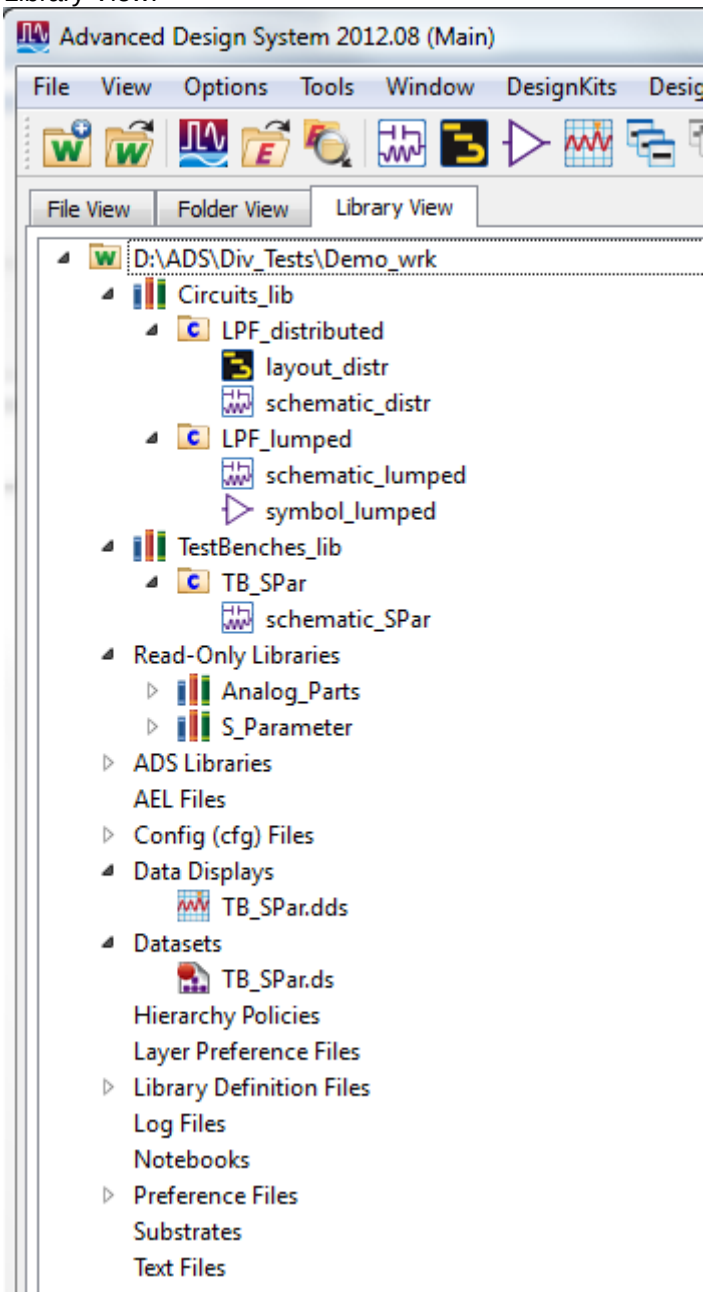
Folder View:



In the Folder View, you can:

- Create virtual folders to group related files (similar to previous releases)

Library View:



In the Library View, you can:

- View the system organization of a workspace
- Find files by type

Library

A Library is a subdirectory that includes cells. Libraries are used by specifying the subdirectory in a library definition file, such as lib.defs. This file defines name of the library and the mode of operation (Read Only, Non-Shared, or Shared). Files within the library itself define the technology (layers, resolution, and layout units) to be used by Views created in that library.

All types of designs are contained in libraries. Design kits in ADS 2012 contain libraries with cells. User workspaces contain libraries. Native ADS components themselves are also contained in Libraries.

The process information such as layer definitions, units, and substrates are defined in the technology of a library. You can create multiple libraries within a complete design hierarchy. Library usage is defined by a file lib.defs. A library may reside physically in the workspace directory.

The following are the key features of a Library:

- Each Library has a unique name and path, specified in a library definition file. (Note that the library itself does not specify its name. The name must be specified correctly in a lib.defs file.)
  Note
  Two libraries with the same name cannot be open at the same time.

- Associated with a Technology that defines physical layers, units, etc. The technology can be a native one created for a library, or it can be referenced to another library's.
- Contains zero (0) or more Cells.
- Loaded/unloaded independently or collectively loaded through a lib.defs file.
- Defines the technology (layers, resolution, layout units) that it will use. All views in a library uses the same technology.

Cell

A Cell is a container for *Views*. It is an equivalent of an ADS design (ADS 2009 Update 1 and earlier releases), as it may contain schematics, layouts, and symbols.

Each Cell:

- Must have a unique name in the library, although cells with the same name may exist in another library.
- Contains zero or more views.
- May have multiple views of the same type.
- May have a component definition, edited by choosing **File > Design Parameters...** while editing one of its views.

View

A View is a sub-directory in a cell that stores design information such as schematic, symbol, or layout. Views may also store an EM simulation setup or an EM Model. Each view is a container that stores a file or a database object. It is a specific representation of a cell. All views in a given library use the same technology.

Each view:

- is associated with a Cell.
- must have a unique name in the Cell.
- **Note**
  Schematic, Layout, and Symbol Views should generally be named as *schematic*, *layout*, and *symbol* unless you are using polymorphism.
- Has a type (schematic, layout, symbol, EM Setup, EM Model).

For more information see "Advanced Design System Quick Start" adstour.pdf, "ADS Quick Reference" oaqkref.pdf and product documentation.

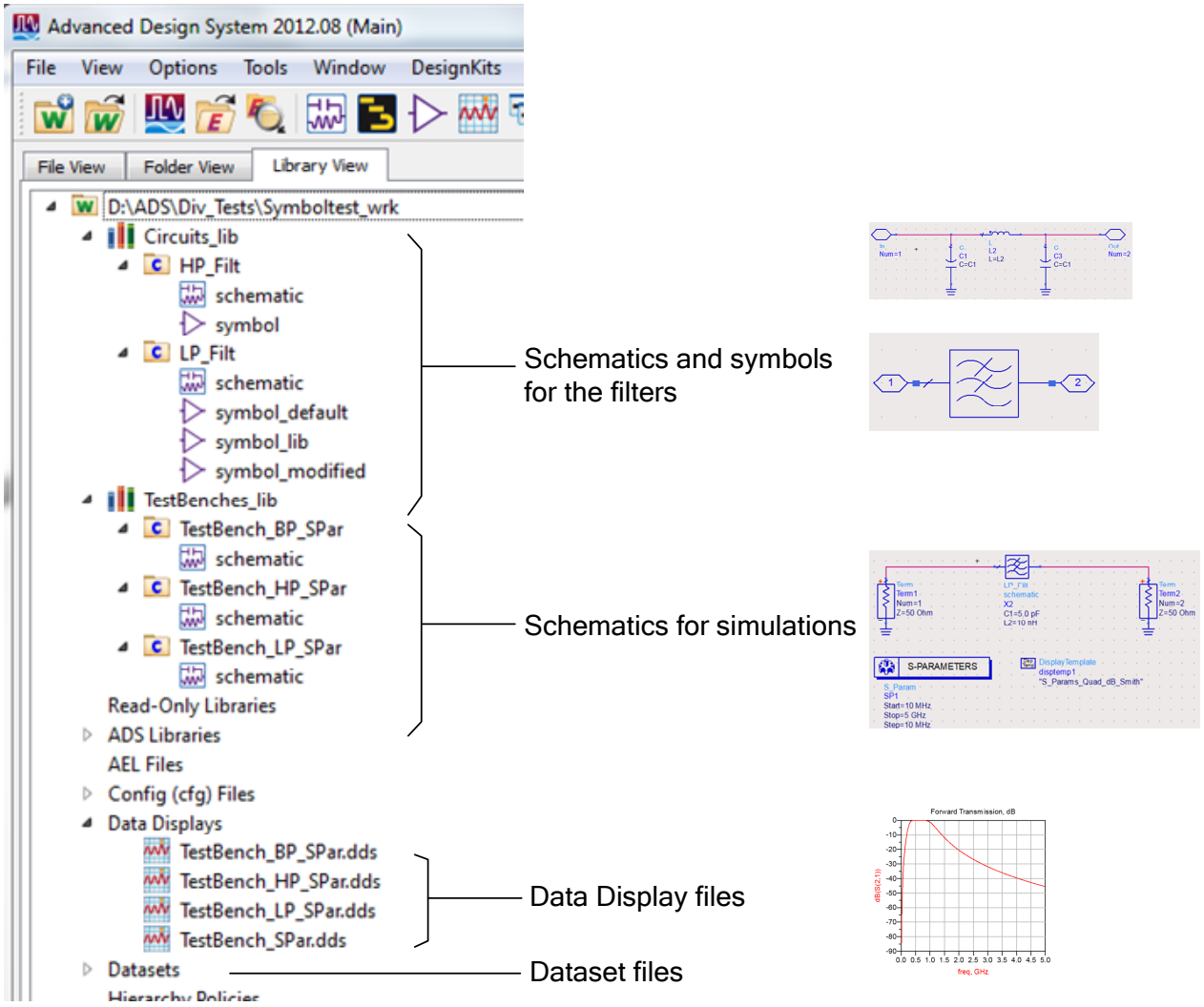
4. Simulations in ADS

Description	Typical Use
DC Fundamental to all RF/Analog simulations. It performs a topology check and an analysis of the DC operating point.	All RF/Analog designs
AC Obtains small-signal transfer parameters like voltage gain, current gain, and linear noise voltage and currents.	Filter Amplifier
S-Parameter Provides linear S-parameter, linear noise parameters, transimpedance, and transadmittance. Can be used to achieve many goals of the AC simulator.	Filter Oscillator Amplifier
Harmonic Balance Uses nonlinear harmonic-balance techniques to find the steady-state solution in the frequency domain.	Mixer Oscillator Power amplifier Transceiver
Circuit Envelope Uses a combination of frequency- and time-domain analysis techniques to yield a fast and complete analysis of complex signals such as digitally modulated RF signals.	Mixer Oscillator Power amplifier Transceiver Phase-locked loop
LSSP Performs large-signal S-parameter analyses to represent nonlinear behavior. The accompanying P2D simulator can be used to speed up subsequent analyses.	Power amplifier
Transient/Convolution Solves a nonlinear circuit entirely in the time domain using simplified models to account for the frequency-dependent behavior of distributed elements.	Mixer Power amplifier Switching circuits
Ptolemy Uses Digital Domain for simulation.	Digital Circuits DSP
Momentum Electro-magnetic Simulation of planar structures	Layouts
FEM Full 3D Finite-Element Electro-magnetic Simulation of 3-dimensional structures	Filters, Antennas, Couplers, Multilayer

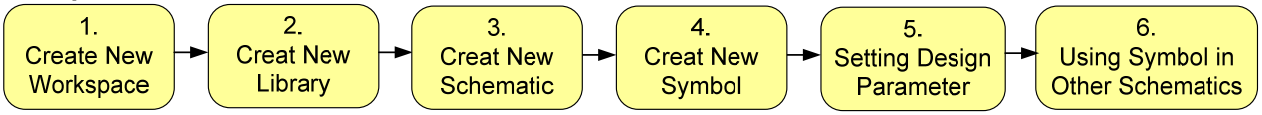
5. Working with Symbols in ADS2012

(Hierarchical Designs, Subnetwork, Subcircuit)

To explain working with symbols in hierarchical designs, we use a step-by-step **example** with a workspace **Symboltest_wrk**. The archived workspace is available as **Symboltest_wrk.7zap**.
Circuits (or Subcircuits) with their symbols are organized in a Library **Circuits_lib**. For each circuit we use a cell containing a schematic and its symbol.
Simulations are organized in a Library **TestBenches_lib**. For each simulation we use a cell containing a schematic with simulation-circuit.
In this step-by-step **example** we use schematics and symbols for a LC-lowpass filter and a LC-highpass filter in the library **Circuits_lib**. In the library **TestBenches_lib** we have schematics with S-parameter simulation for each filter and for cascaded filters.

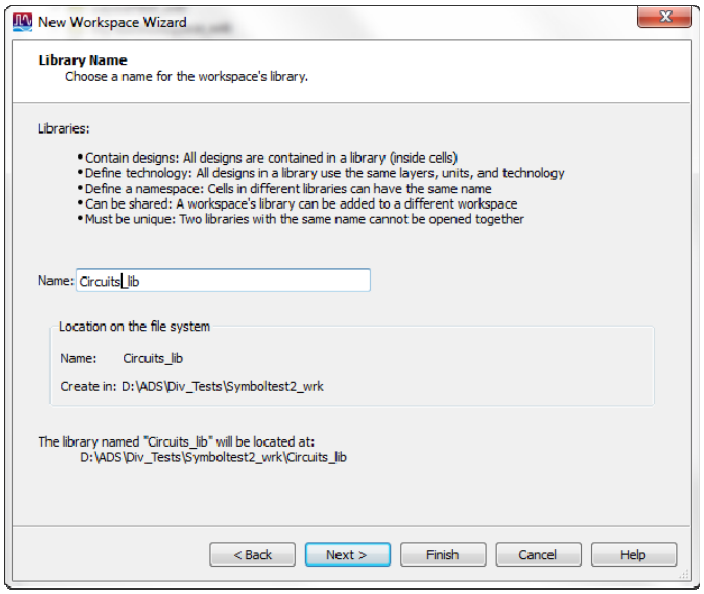


Example:



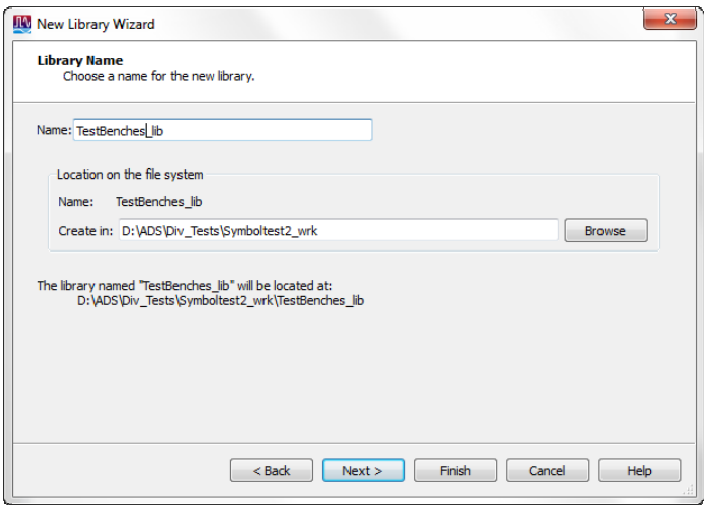
5.1. Create New Workspace

From the ADS Main Window, choose **File** > **New** > **Workspace.** Name the library **Circuits_lib**.



5.2. Create New Library

From the ADS Main Window, choose **File** > **New** > **Library.** Name the library **TestBenches_lib**.



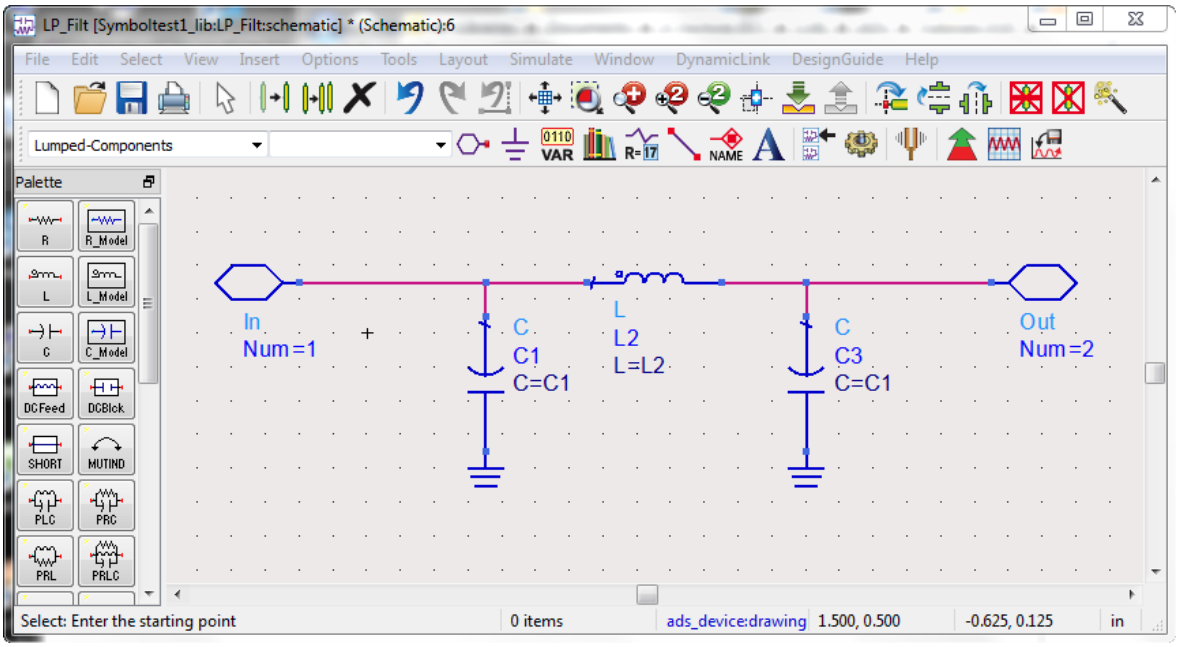
5.3. Create New Schematic

From the ADS Main Window, choose **File** > **New** > **Schematic** or click  and create the new schematic in library **Circuits_lib.**

Name the cell **LP_Filt** and accept the schematic default name **schematic.**

Insert components and complete schematic as shown below.

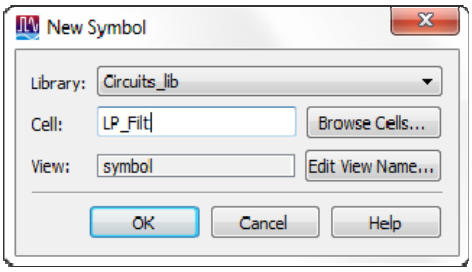
Save schematic  .

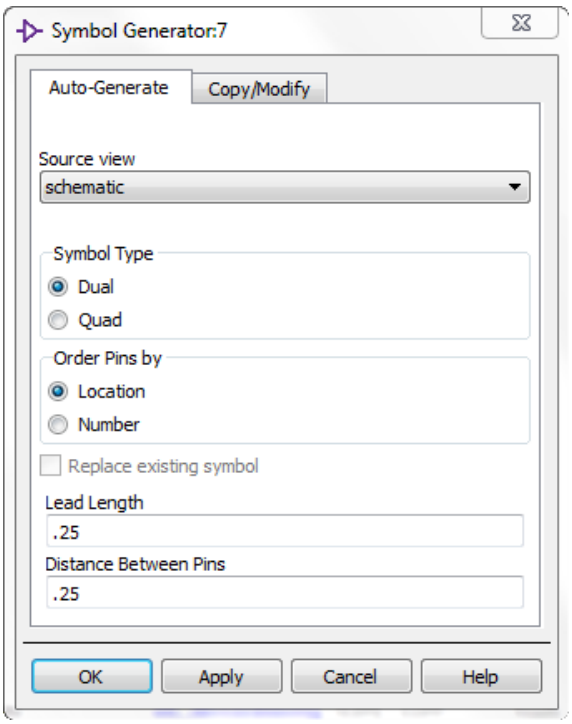


5.4. Create New Symbol

From the ADS Main Window, choose **File** > **New** > **Symbol** or click  and create the new symbol in library **Circuits_lib.**

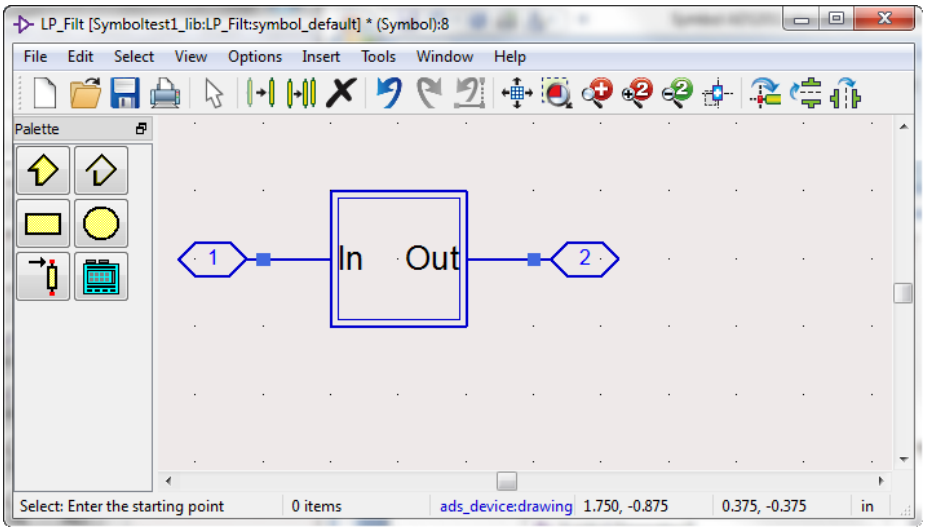
Browse Cells and select **LP_Filt** and accept the symbol default name **symbol.**



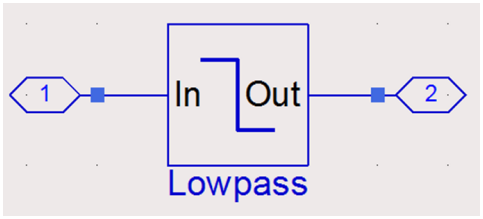


Symbol Type	Description
Dual	Restricts pins to two sides of symbol body
Quad	Allows pins on all four sides of symbol body

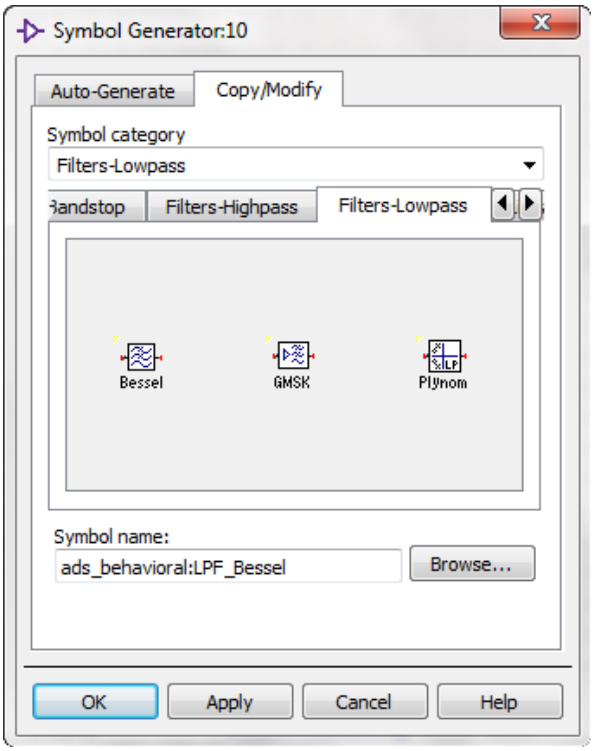
Order Pins by	Description
Location	Numbers pins in the same relative order as the ports on the schematic
Number	Numbers pins sequentially in a left-right, top-down order
Lead Length and Distance Between Pins	Be careful altering this values. The values have to fit the grid spacing in schematic.



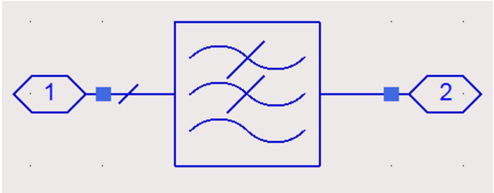
The generated default symbol can be edited as desired at any time.
Add text, change shape of symbol as desired.
Be careful moving pins. They have to fit the grid spacing in schematic.
See help “Working with Symbols”.



Alternatively one of the many symbols from ADS can be used:



Use this to select one of the many symbols from symbol library.

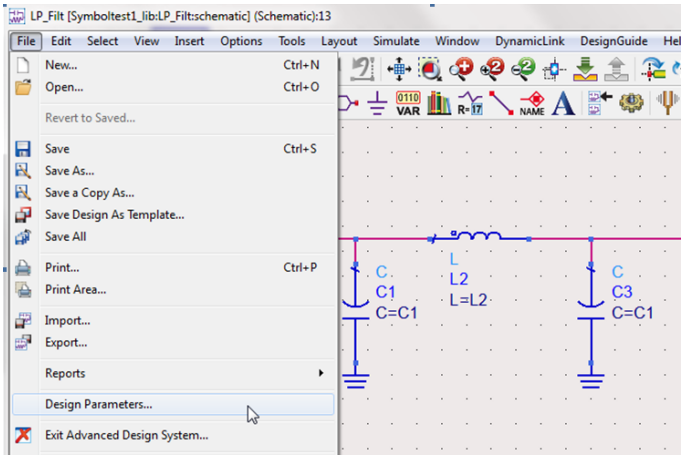


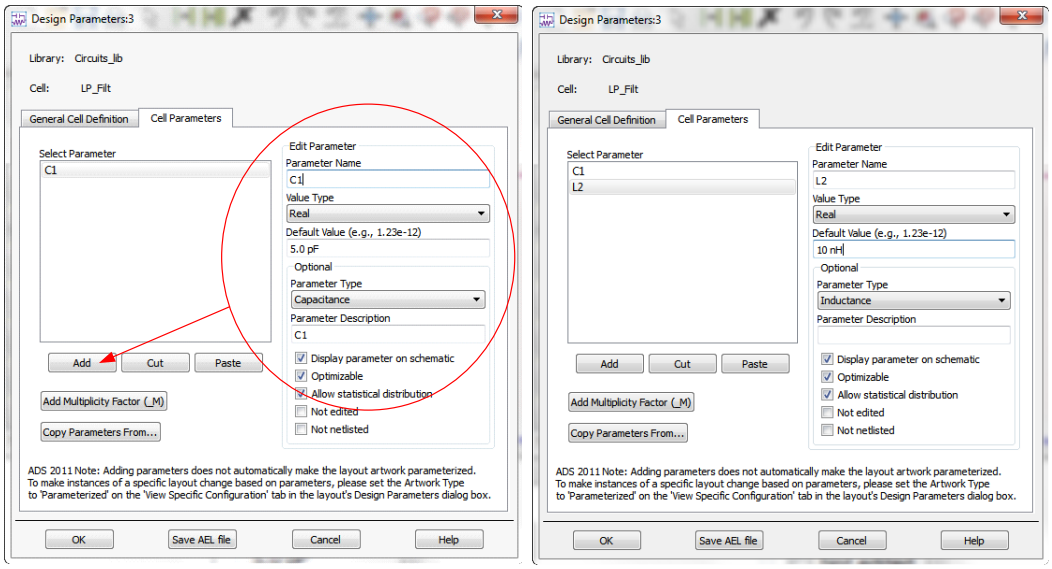
5.5. Setting Design Parameter

To pass component parameters from schematic to symbol view they have to be set in schematic. To define the parameters that should be passed to the upper-level symbol:

Open the **schematic** in the cell **LP_Filt**

Choose **File** > **Design Parameters**. In the **Design Parameters** dialog box, select *Cell Parameters* tab and enter variables and default values for the capacitor and inductor.





Click **OK** to close this dialog box, **save** and **close** the **schematic**.

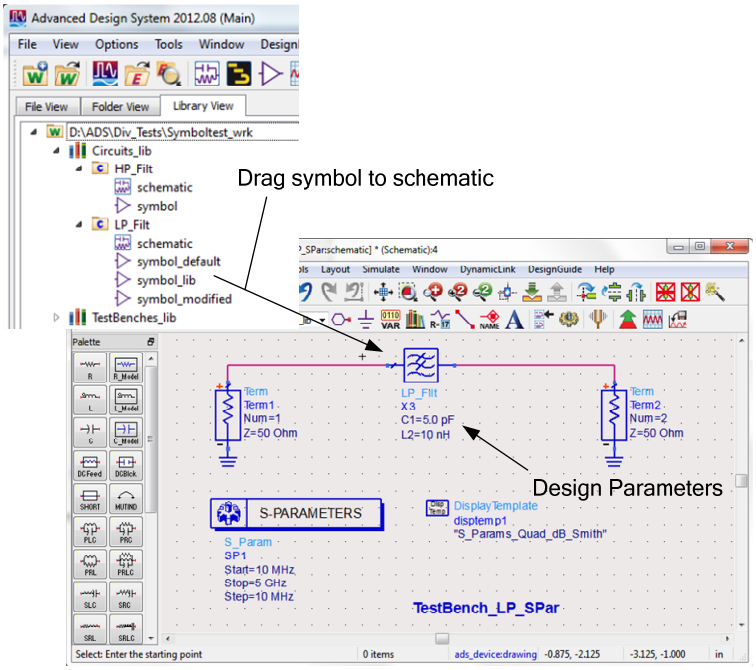
5.6. Using Symbol in Other Schematics

The symbol can be used in any other schematic. The parameter we assigned (capacitance and inductance in this example) will pass through to the network.

In this example we create a schematic in the TestBenches_lib with the cell name TestBench_LP_Spar to perform a S-parameter simulation on the lowpass filter.

From the ADS Main Window, choose **File** > **New** > **Schematic** or click  and create the new schematic in library **TestBenches_lib.** Name the cell **TestBench_LP_Spar** and accept the schematic default name **schematic.**

Insert the symbol by simply drag it from the Main window, Folder View or Library View to the schematic **TestBench_LP_Spar.**

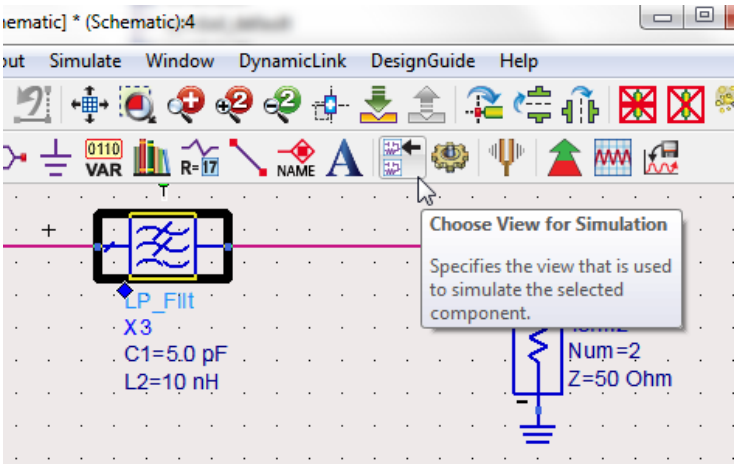


Complete schematic with S-parameter simulation template: **Insert > Templates > S_Params** and set frequencies to desired values.

Save schematic  .

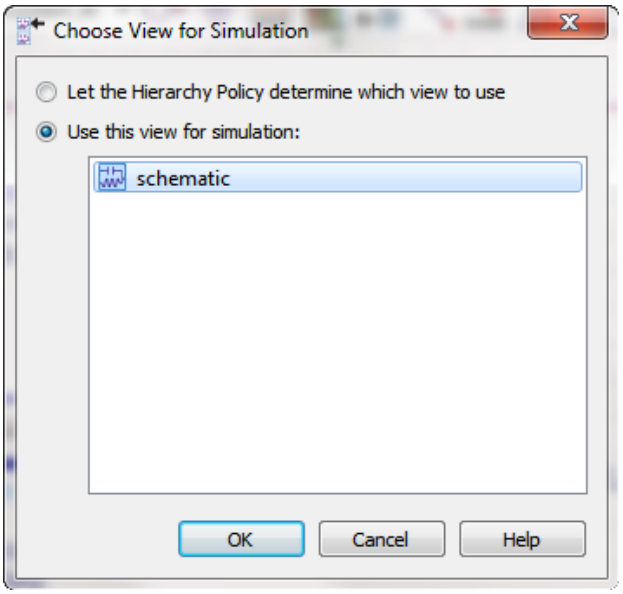
Specifying the View to Simulate

In the TestBench_LP_Spar schematic, select the LP_Filt symbol.

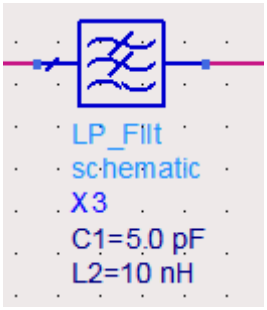


Click **Choose View for Simulation**  icon.

In the **Choose View for Simulation** dialog box, select **Use this view for Simulation**.



Click **OK**. Note that schematic now appears under the LPF symbol in the schematic.



Modifying the schematic of a symbol

If you make changes in the schematic that affect the component definition (any changes in the Design Parameters dialog box-information), that serves as a subnetwork in a higher-level design, you need to explicitly update the symbol to recognize those changes.

To update a symbol that contains a modified subnetwork:

Select the symbol and choose Edit > Component > Update Component Definitions.

Select the option **Update Component Definitions in Subnetwork** and click **OK**. Changes to any subnetwork designs are now reflected in the symbol.

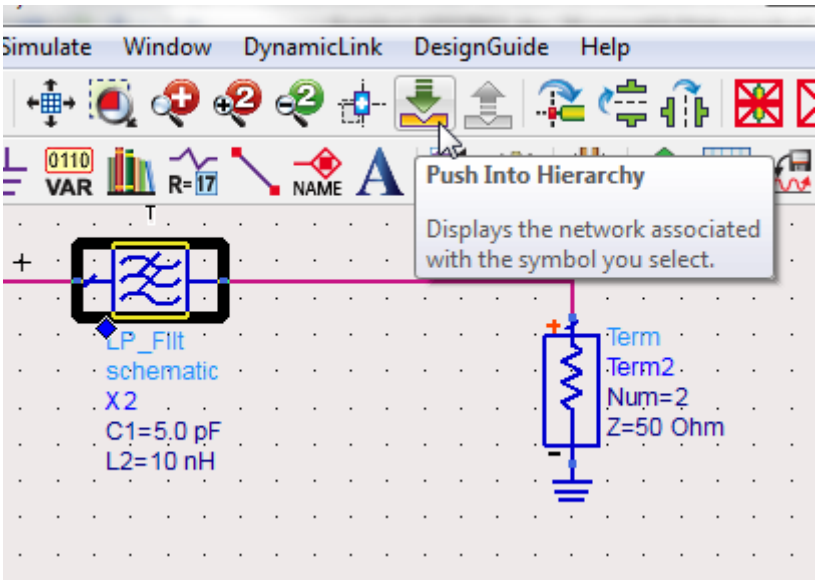
Viewing the Network Represented by a Symbol

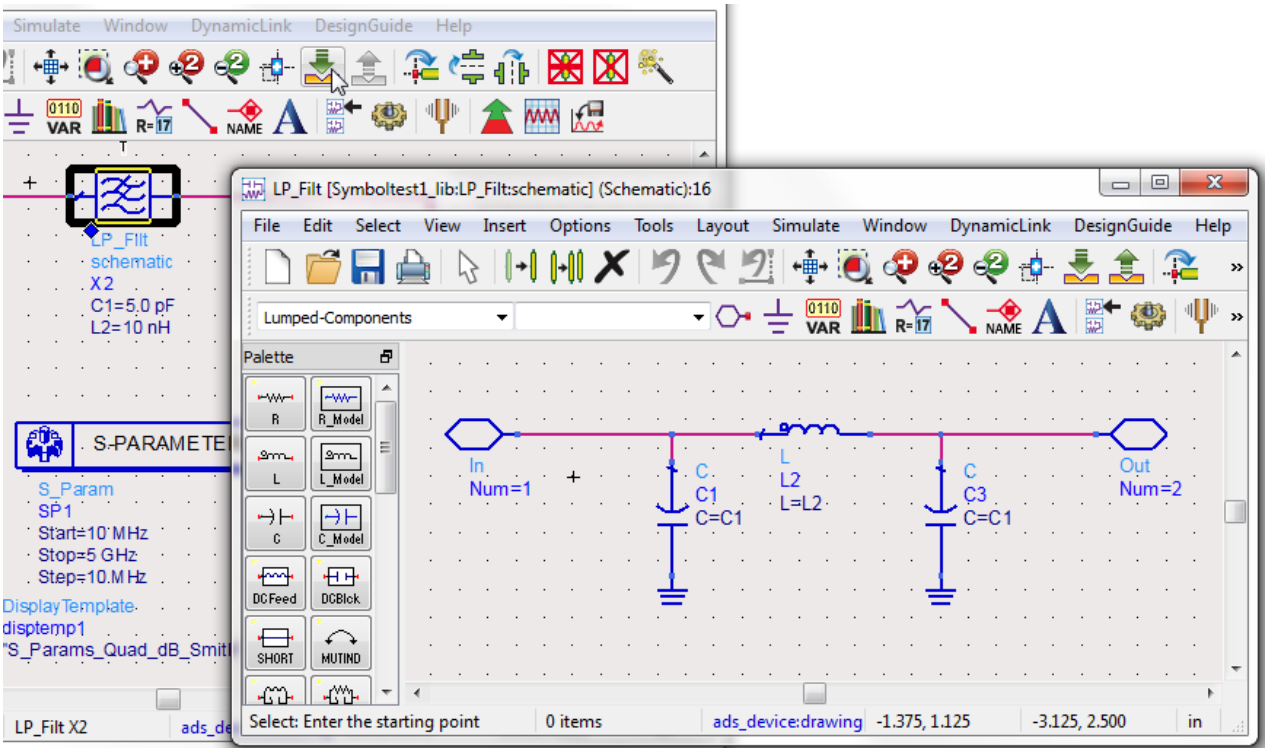
Whenever your design contains a symbol that represents a network, you can view the actual network being represented by the symbol by using the *Push Into Hierarchy* command.

To push into and then pop out of an item:

Select the item.

Choose **View > Push Into Hierarchy** or  . The network represented by the symbol is displayed.

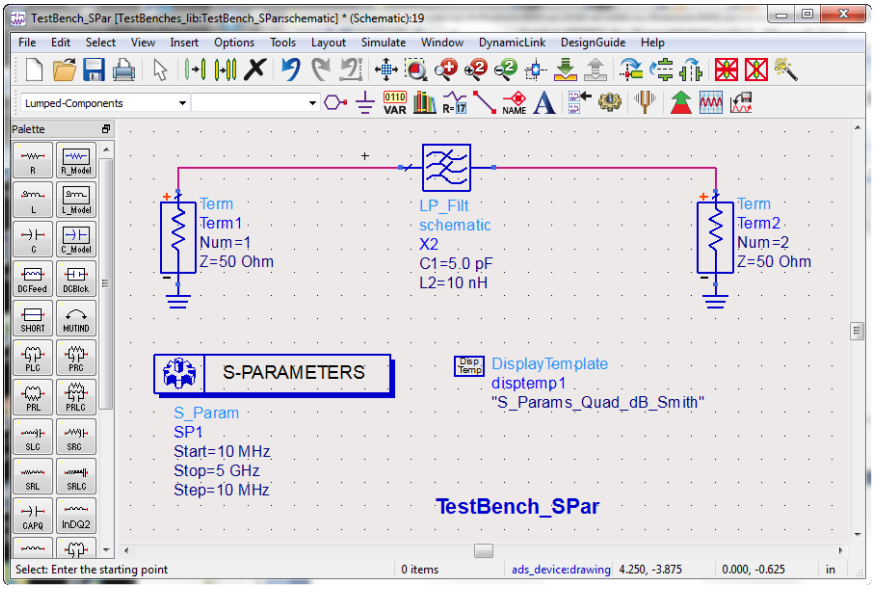




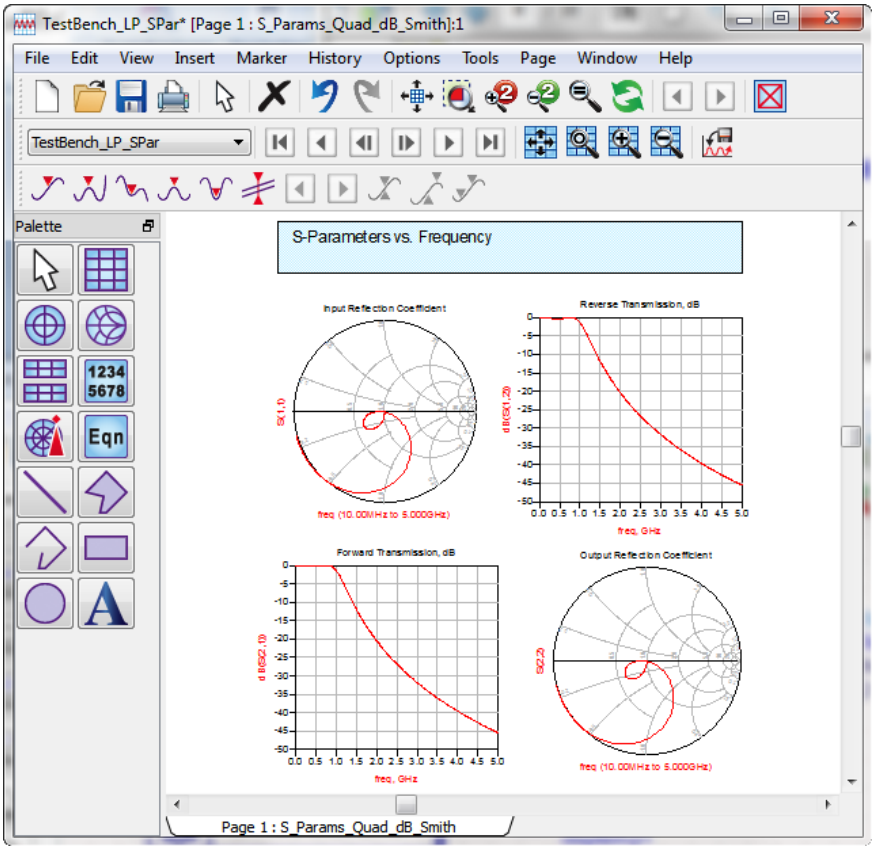
The *Pop Out of Hierarchy*  command is the reverse of pushing, and only works if a design has been pushed into.

When you are through viewing the network, choose **Pop Out of Hierarchy** and you are brought back to the item (or schematic containing the item).

5.7. Now everything is ready for simulation.

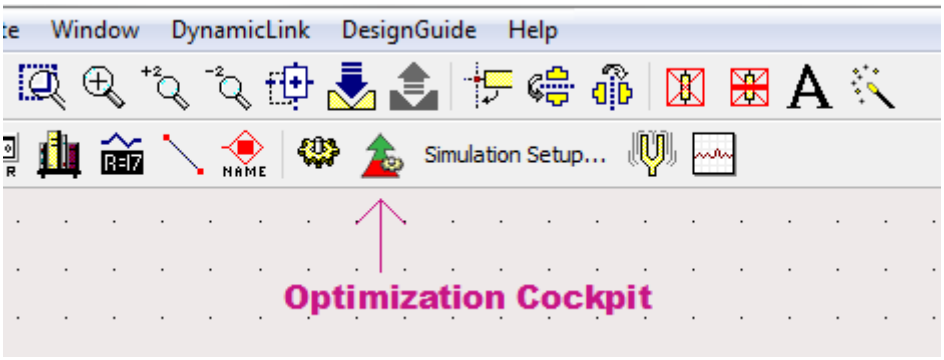


Start simulation  and display simulation results.

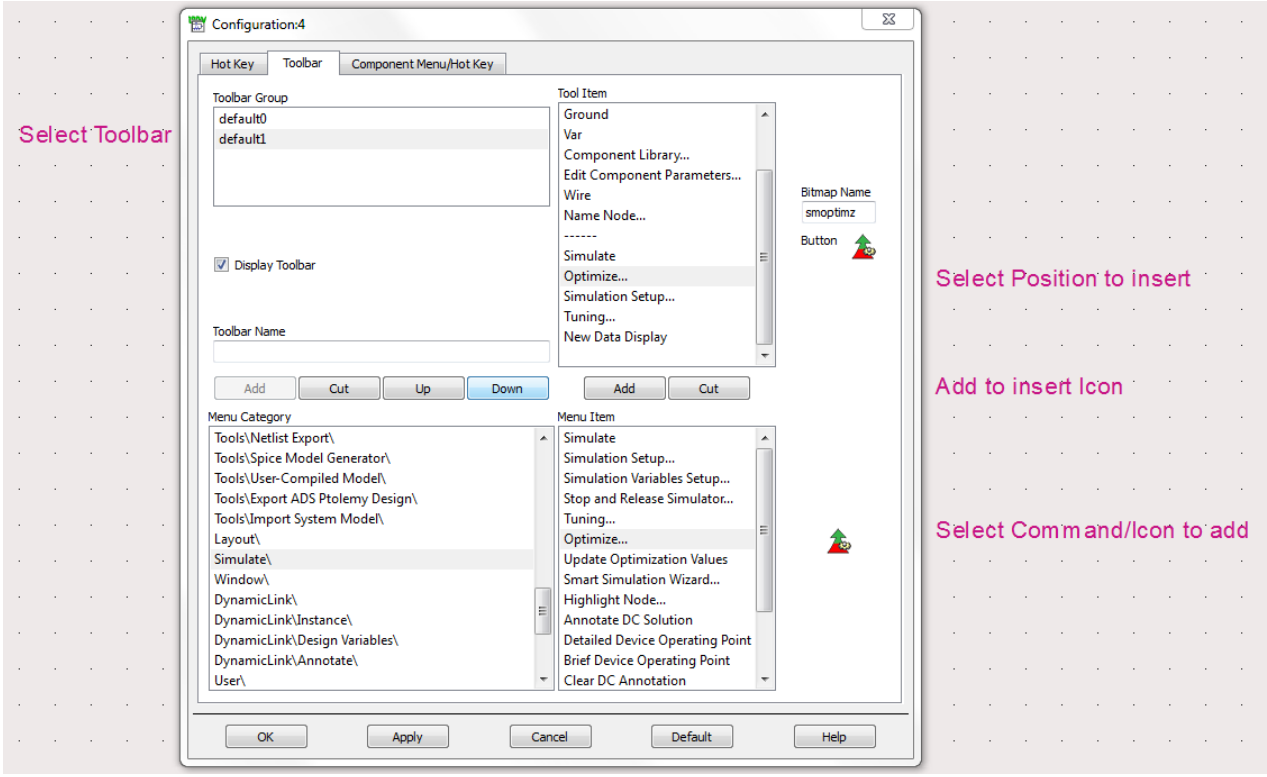


6. ADS Optimization Cockpit

For usage of Optimization Cockpit see:
http://www.youtube.com/user/AgilentEEsof#p/c/FF7051B3E314AD45/41/m0UkqaqVDoQ



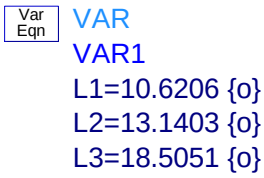
If you are missing icons in the toolbar you can customize your toolbar (add or delete icons):
Menu: Tools -> Hot Key/Toolbar Configuration

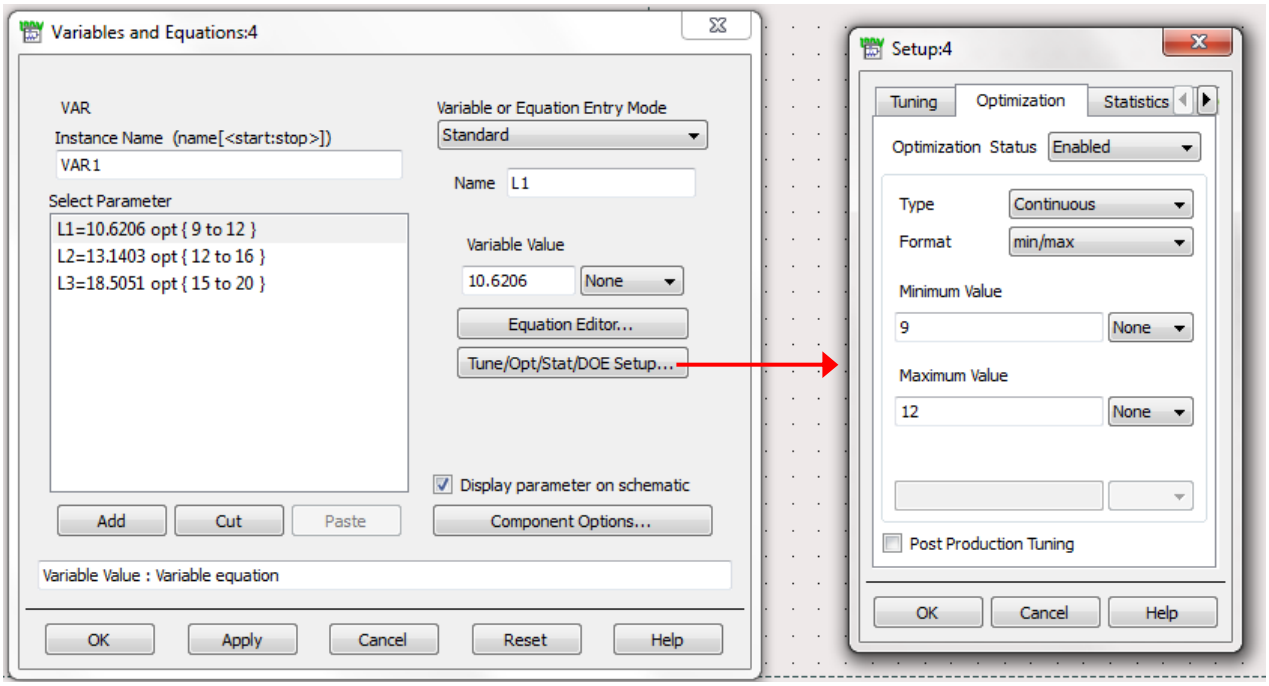


6.1. Setup for Optimization

- Place an Optimization-item and Goals to the schematic.

- Setup Variables or element values for Optimization



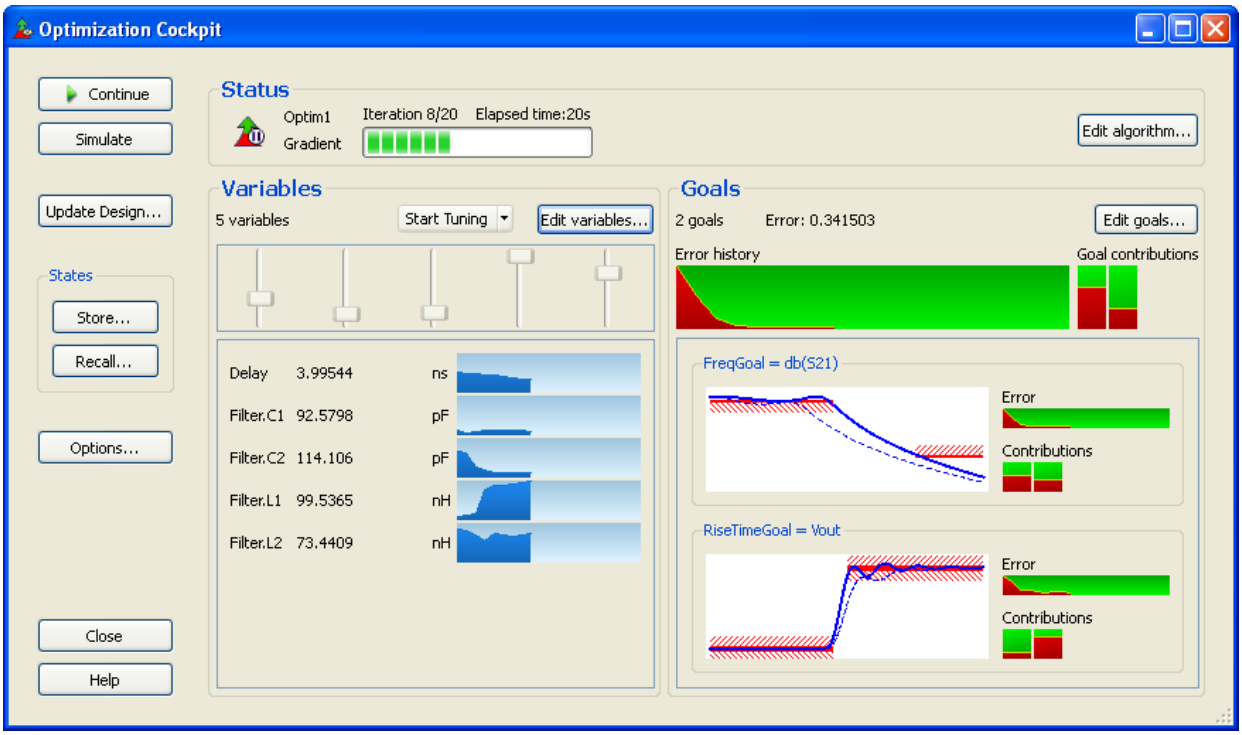


Optimization

The Optimization Cockpit is a live, graphical view of an optimization job. You see the cockpit data (error graphs, goal plots, and variable values) change in real time as the optimization progresses. In addition, you can use the cockpit to control the optimization while the optimization is running. For example, during the course of an optimization, you can increase the range of an optimization variable, modify the limit line of a goal, tune the optimization variables, and change the algorithm from Random to Gradient.

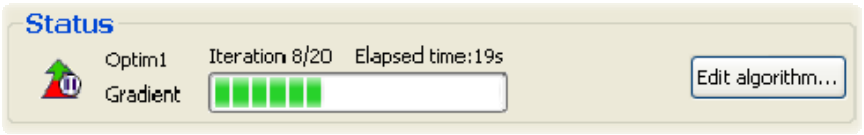
Cockpit Panels

The cockpit has three main panels: Status, Variables, and Goals. It also has a control panel on the left-hand side.



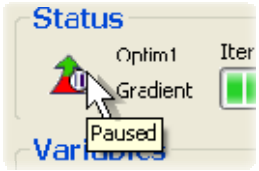
Status panel

The Status panel displays the optimizer's status, type, elapsed time, and progress. It also has a button for changing the optimization algorithm settings.



Activities

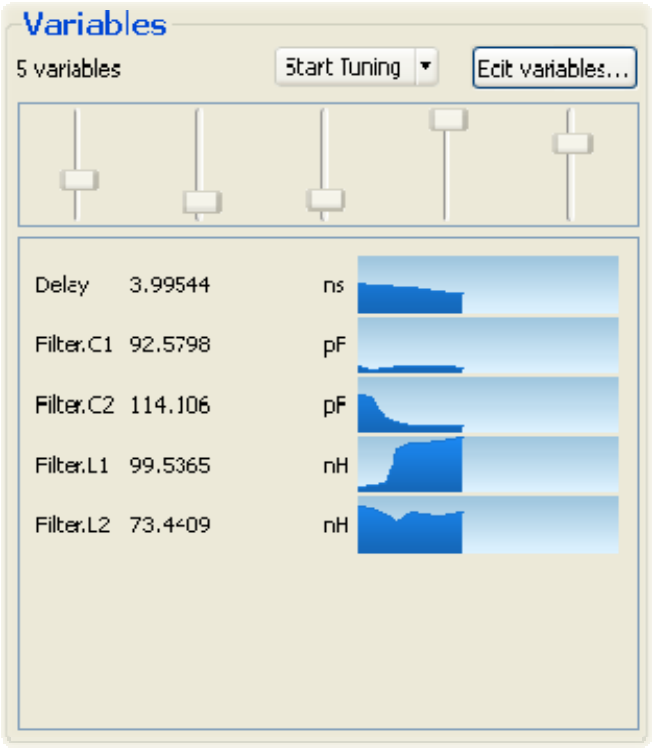
Roll the pointer over the status icon to know the state of the optimizer



While the optimizer is paused or towards the end of an optimization, click **Edit Algorithm** to modify the optimization algorithm. See “Modifying the algorithm” for more information.

Variables panel

The Variables panel displays the optimization variables as a row of sliders and as data in tabular format. It also has buttons for tuning and for editing the variable definitions.



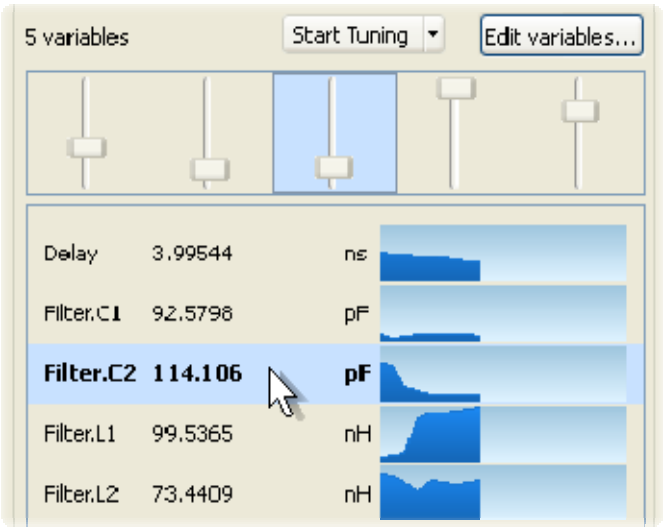
The sliders represent the current value of each optimization variable relative to its minimum and maximum values.

Each row of the table shows important information for each optimization variable:

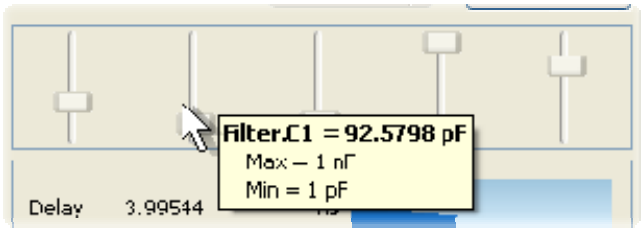
- Name
- Current value
- Units (if any)
- A history graph showing the values of the variable over the course of the optimization.

Activities

Click a row in the table to highlight its corresponding slider or vice-versa



Move the pointer over a slider to see information about that variable

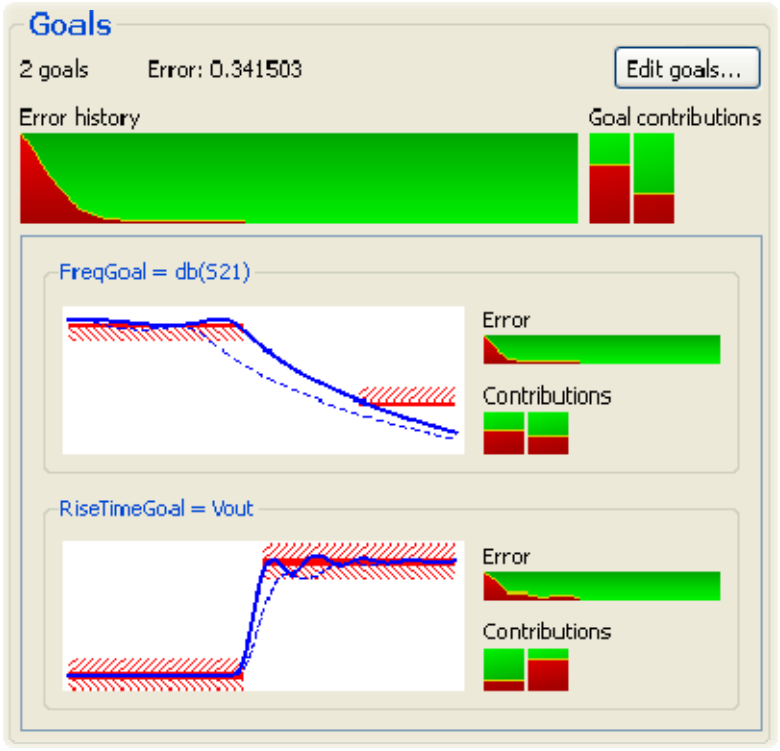


Click **Start Tuning** to switch to tune mode. See “Tuning” for more information.

While the optimizer is paused or towards the end of an optimization, click **Edit Variables** to modify the variable settings. See “Modifying variables” for more information.

Goals panel

The Goals panel displays the current error, the error history graph, the goal contribution histogram, and the goals table. It also has a button for editing the goal definitions.

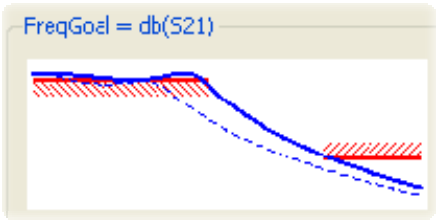


The *Error history* graph shows the overall error over the span of optimization.

The *Goal contributions* histogram is displayed when there are two or more goals. It represents each goal's contribution to the overall error.

The table contains a row for each goal. Each row represents the following information:

- A plot of the goal's response and limit lines. The limit lines are red and the response is blue.



The solid blue trace is the response for the current values of the optimization variables. The dashed blue trace is the response at the beginning of the optimization.

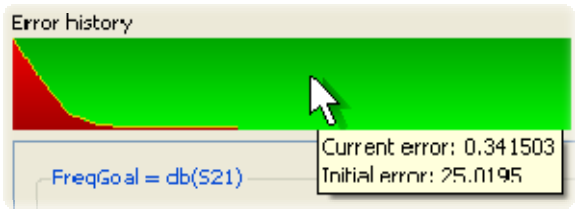
- A history plot of the goal's error over the span of optimization.

Sometimes you will notice the error for a goal increases. The optimizer is driving the overall error to zero, so the error for a particular goal can increase as long as the overall error decreases.

A Contributions histogram is displayed only when the goal has more than one limit line. The histogram shows each limit line's contribution to the goal's error.

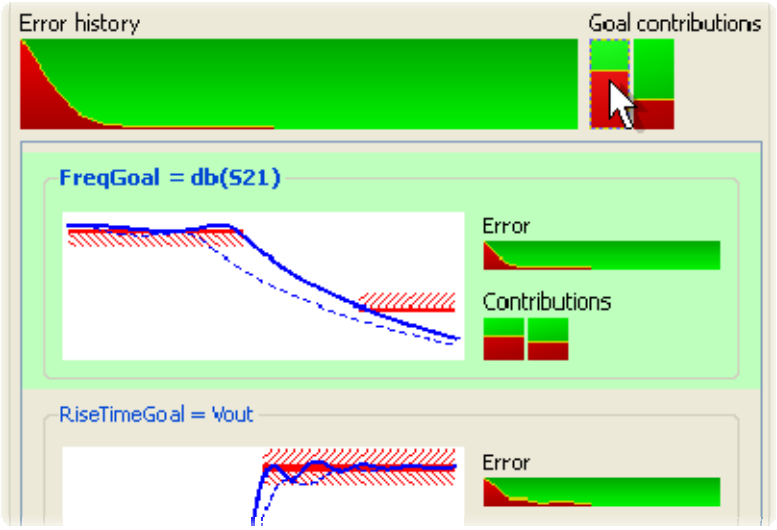
Activities

Move the pointer over the *Error history* graph to see information about the error:

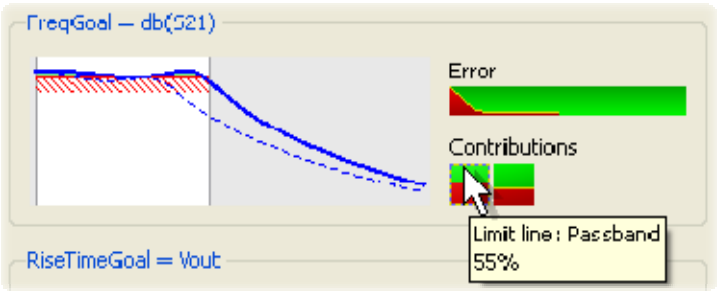


Move the pointer over a bar in the *Goal contributions* histogram to see information about that goal's contribution to the error:

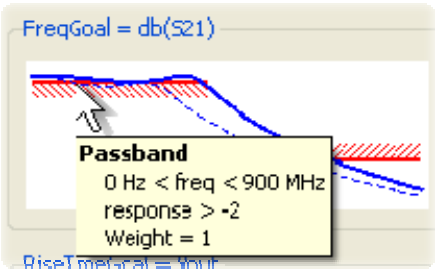
Click a bar in the Goal Contributions histogram to highlight the corresponding goal in the table or click a goal in the table to highlight the corresponding bar in the histogram:



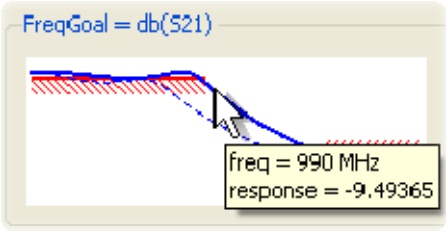
Move the pointer over a bar in the *Contributions* histogram to display the contribution percent and to highlight the corresponding limit line on the plot:



Move the pointer over a limit line to see information about that limit line:



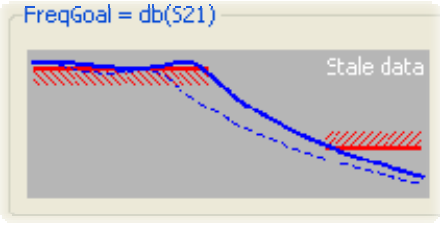
Move the pointer over a data point in the trace to see the coordinates of the point:



While the optimizer is paused or towards the end of an optimization, click **Edit Goals** to modify the goal settings. See “Modifying goals” for more information.

Stale plots

Goal plots go stale when the algorithm, goal, or variable settings change, but no simulation is performed.



This means that the simulation results (error value, the contribution graphs, and the goal plots) displayed in the Cockpit do not accurately reflect the current algorithm, goal, and variable settings. For example, click **Edit Goals** and modify one of the limit lines you will see the goals plot will go stale. Click **Simulate** to refresh the goals plot.

Control panel

This panel has several buttons on the left-hand side of the Cockpit that helps you to control the optimization while it is running.

Pause

Click **Pause** to pause a simulation while it is running. Once an optimization is paused, you can interact with the optimizer. It can take time for the optimizer to pause when the low-level simulation (for example, transient or harmonic balance) takes a while to run.
The **Pause** button changes to **Continue** when an optimization is paused or is completed.

Continue

Click **Continue** to continue the optimization. This button is available when the optimizer is paused or has finished.

If the optimizer has reached the maximum number of iterations and you click **Continue**, you can increase the maximum number of iterations.

Simulate

Click **Simulate** to re-evaluate the goals. This is useful after you have changed the settings (for example, modified a limit line on a goal) and you want to update the goals plots or see the new error value.
It is also useful in **Simulate after pressing Simulate** tuning mode. See "Tuning modes" for more information.

Data Display graphs do not get refreshed on clicking **Simulate**.

Update Design

Click **Update Design** to transfer variable values to the design.
This command can also transfer algorithm and goal data to the design. This is useful when you have used **Edit Algorithm** or **Edit Goal** to modify the algorithm or goal definitions. For more information on modifying the goal or algorithm settings, see "Controlling the Optimization".

Note: The options you specify while updating the design do not change from one optimization to other.

If the sweep panel is displayed (see "Sweep Panel") when you click **Update Design**, the design is updated using the optimization specified by *View*.

Store

This option allows you to store the current optimization state. For more information, see "Using Optimization states".

Recall

This option helps you to recall a state. For more information, see "Using Optimization states".

Options

It allows you to view the Cockpit options. For more information, see “Scaling the plots”.

Close

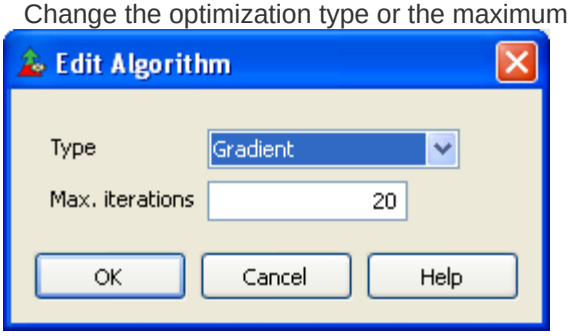
Click **Close** to close the Optimization Cockpit. If the optimization is still running, you have the option to stop it or to update the design.

Controlling the Optimization

Modifying the algorithm

Follow the steps below to modify the algorithm:

1. Click **Edit Algorithm**.
2. Change the optimization type or the maximum number of iterations.

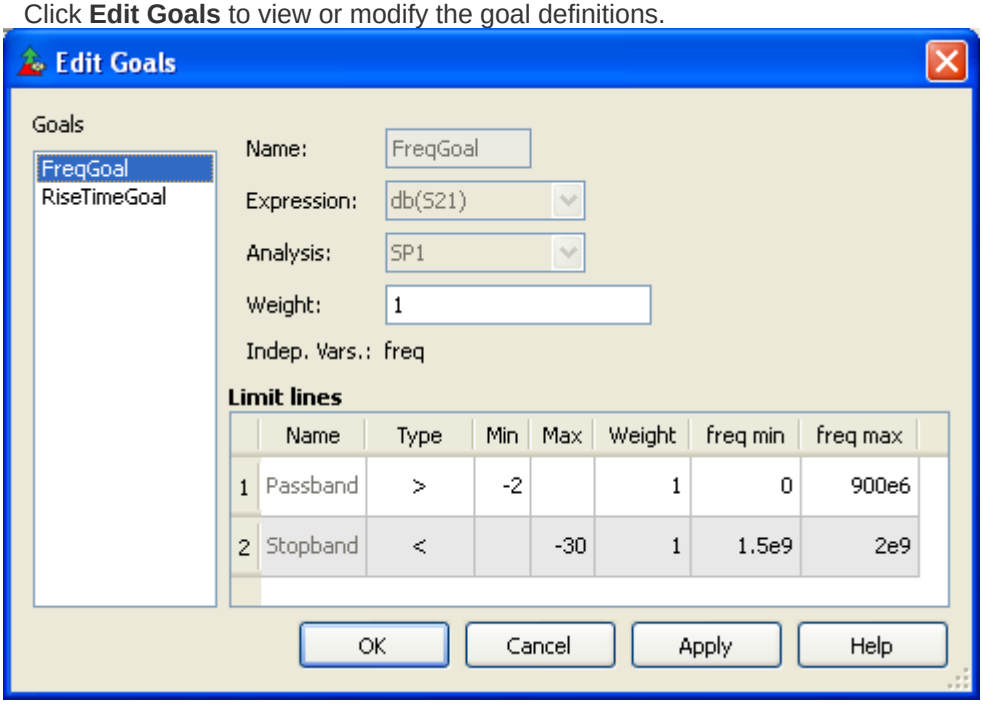


3. Click **OK**.
   Note: When you click **OK**, the Optimization component on the schematic is not updated.
4. Click **Update Design** to transfer the new settings to the schematic.

Modifying goals

Follow the steps below to modify the goals definition:

1. Click **Edit Goals** to view or modify the goal definitions.



2. Goals are listed in the *Goals* list. Click an entry from the list to view its properties. You can change the goal's weight and also the definition of each limit line. From the cockpit, you cannot change a goal's name, expression, or analysis. Also, it doesn't allows you to add a new goal, delete an existing goal, or change the name of a limit line.

Set *Weight* = 0 to disable a goal or a limit line.

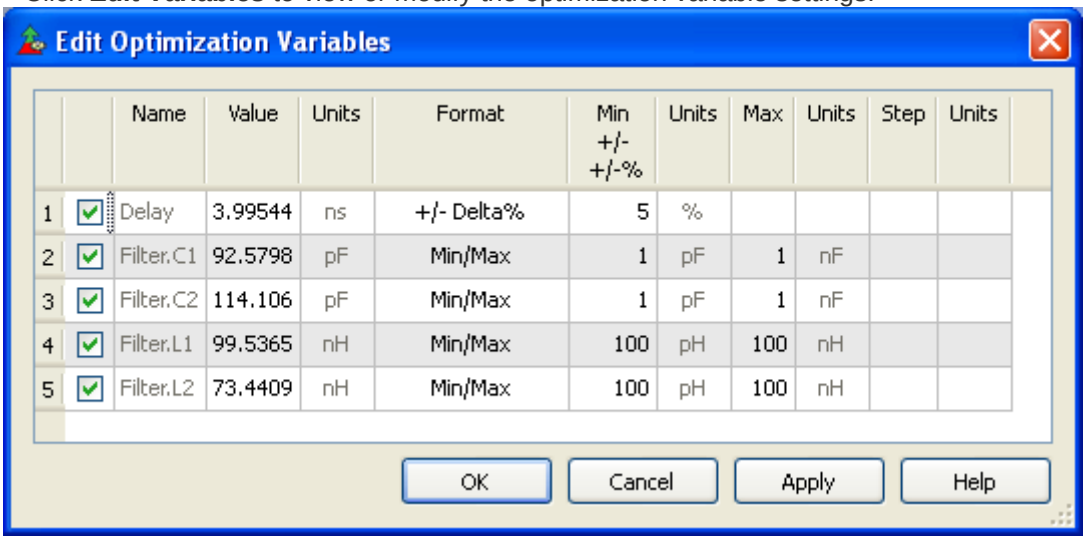
3. After changing the goals definition, Click **OK**.
Note: When you click **OK**, the Goal components on the schematic are not changed.

4. Click **Update Design** to transfer the new settings to the schematic.

Modifying variables

Follow the steps below to modify the variable settings:

1. Click **Edit Variables** to view or modify the optimization variable settings.



2. Select the check-box to enable that variable.
The edit variable dialog box allows you to change the nominal value, format, and format's min and max values. It doesn't allows you to change a variable's name, add a new variable, or delete an existing variable.

3. After modifying the variable settings, Click **OK**.
Note: When you click **OK**, the variables on the schematic are not changed.

4. Click **Update Design** to transfer the new settings to the schematic.

Note: Variables of **+/- Delta** or **+/- Delta%** *Format* are handled differently. For these variables, the minimum and maximum values for the variable are calculated as offsets from the nominal value. This calculation is performed at the beginning of the optimization and the minimum and maximum values do not change as the optimization progresses. In the above example, it seems as if **Delay** is defined as 3.99544 ns +/- 5%. This is not the case. At the beginning of the optimization, the nominal value for **Delay** was 4 ns, so the definition is actually 4 ns +/- 5%. When you edit any field for this variable and click **OK** or **Apply**, the variable's minimum and maximum are recalculated based on the value in the table; otherwise, the variable's minimum and maximum are not recalculated.

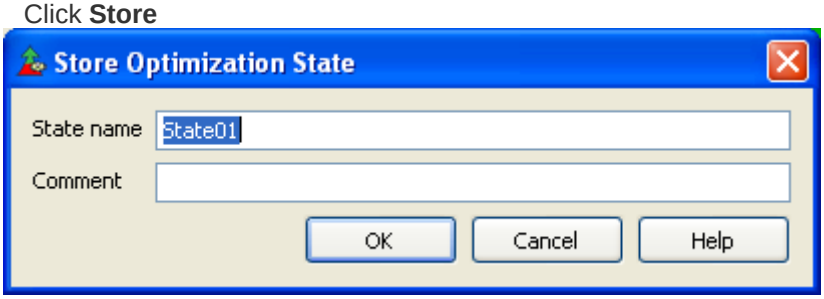
Using Optimization states

The cockpit has the ability to store and recall optimization states. This is useful when you explore the behavior of your design using the Optimization Cockpit. For example, before modifying some variable definitions or goal definitions, you can store the current optimization state and recall the saved state when required.
The optimization state contains the algorithm, goal, and variable settings.

Store

Follow the steps below to store an optimization state:

1. Click **Store**



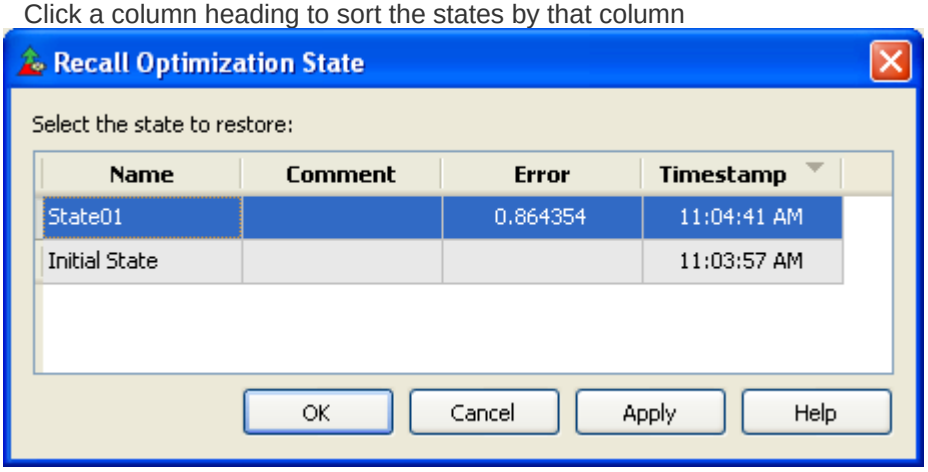
2. Specify the name of a state in *State name* field
3. Add a comment in the *Comment* field (optional)
4. Click **OK** to store the current optimization state

The Optimization Cockpit automatically stores the initial state.

Recall

Follow the steps below to recall an optimization state:

1. Click **Recall**
2. Click a column heading to sort the states by that column



3. Select a state
4. Click **OK** or **Apply** to restore the selected state

When you recall a state, the cockpit's algorithm, goal, and variable settings are replaced with those of the recalled state. It is similar to manually reverting back to previous settings using Edit Algorithm, Edit Goals, and Edit Variables to restore the state. This means that the history graphs and goal graphs do not change to what they were when you saved the state.

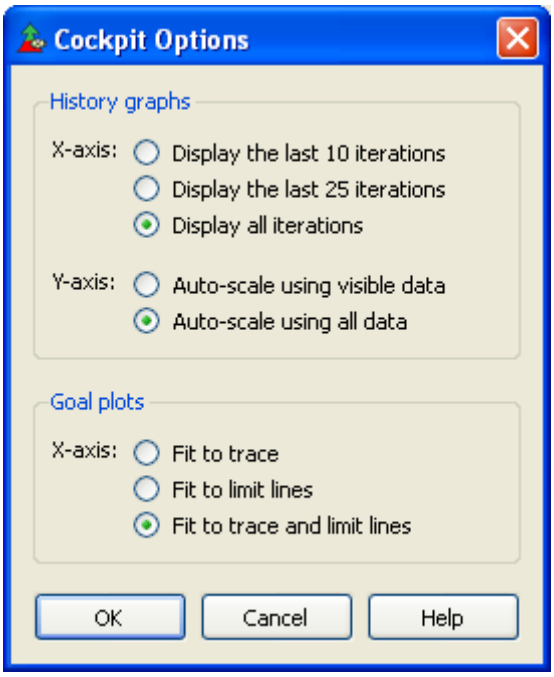
When you recall a state the goal plots will go stale (see “Stale plots”). Click **Simulate** to recalculate the error and update the goal graphs.

The automatically stored initial state has no Error value associated with it because the state is stored before any simulation is performed.

Note: Optimization states are discarded when the Optimization Cockpit is closed. If you want to use an optimization state in your next optimization, click **Recall** to recall the state and then click **Update Design** to update the schematic.

Scaling the plots

Click **Options** to view or modify the Cockpit options.



History graph X-axis options These options modify the X-axis scaling of all history graphs (both error and variable history graphs).

- **Display the last 10 iterations** The graph displays only the 10 most recent iterations.
- **Display the last 25 iterations** The graph displays only the 25 most recent iterations.
- **Display all iterations** Scales the X-axis from 0 to Max. iterations.

History graph Y-axis options These options modify the Y-axis scaling of the error history graphs.

- **Auto-scale using visible data** Consider only the iterations that are visible in the graph when scaling the Y-axis.
- **Auto-scale using all data** If there are iterations that are not visible on the graph (for instance, the graph shows the last 10 iterations, but the current iteration is greater than 10) then consider the invisible iterations when scaling the Y-axis.

Goal plots These options modify the X-axis scaling of all the goal plots.

- **Fit to trace** Set the X-axis scaling to show the entire trace.
- **Fit to limit lines** Set the X-axis scaling to show all the limit lines.
- **Fit to trace and limit lines** Set the X-axis scaling to show all the limit lines and the entire trace.

Note: All of these options are remembered from one optimization to the next.

Tuning

Click **Start Tuning** to start tuning. If the optimization is running, it will pause.

While tuning, the **Start Tuning** button is renamed as **Stop Tuning**.
To disable the tune mode:

- Click **Stop Tuning**, or
- Click **Continue** to resume the optimization.

You can modify the optimization variable values using the sliders or by entering values directly in the variables table.

Note: The minimum and maximum values of the variables are determined by the optimization variable definitions. The settings used by the standard tune mode are not used. To change the minimum and maximum values for a variable, click **Edit Variables**.

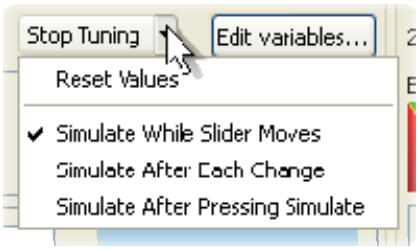
Data Display windows do not update while tuning.

Activities

- Click a row in the variables table to highlight the corresponding slider. Click a slider to highlight the corresponding row.
- On the keyboard, press *Tab* (or *Enter*) to move down in the variable table. Press *Shift+Tab* (or *Shift+Enter*) to move up.

Tuning menu

The **Start Tuning** (or **Stop Tuning**) button has a menu attached to it. Click the arrow to the right of the button to access the menu.



Reset values

Choose **Reset Values** to reset the optimization variable values to the values they had when you started tuning.

Use the **Store** and **Recall** buttons to store and recall intermediate tuning states. For more information, see “Using Optimization states”.

Tuning modes

Tuning has three different modes. Use the Tuning menu to change modes.

- **Simulate while slider moves** - In this mode, the simulator performs simulations while you are moving the slider. This is useful for faster simulations (less than 0.5s).
- **Simulate after each change** - In this mode, the simulator waits to perform a simulation until you release the slider. This is useful when the simulations are not as quick (0.5s to 5s).
- **Simulate after pressing Simulate** - In this mode, no simulations are performed until you click **Simulate**. This mode is useful when the simulation takes longer (greater than 5s) or you know in advance that you want to make several changes before simulating.

Note: This setting is remembered from one optimization to the next.

View-only mode

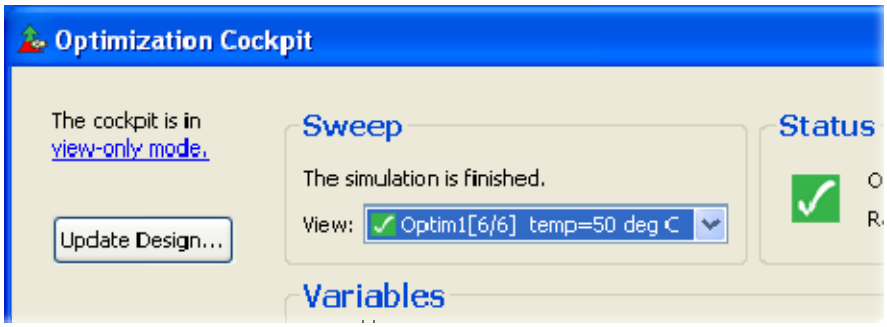
The Cockpit switches to view-only mode when either of the following is true:

- The top-level simulation controller is not an optimization. For example, there is a **ParamSweep** sweeping an optimization.
- There are two or more top-level simulation controllers. For example, there is a top-level harmonic balance controller and a top-level optimization.

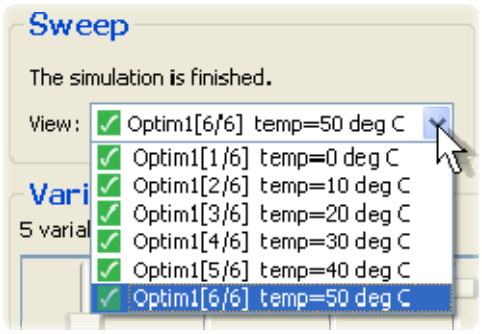
In view-only mode, the cockpit still displays the real-time optimization data, but none of the interactive features is available.

Sweep Panel

The sweep panel appears when there is more than one optimization in the simulation job. This happens when an optimization is swept (using a **ParamSweep** component) or when there is more than one optimization in the simulation job.



Use the *View* drop-down list to select an optimization to be displayed by the cockpit.



Changing the view has no affect on the simulation, so you can view the results of the first point in the sweep while the rest of the sweep is running.
When you click **Update Design**, the design is updated using the optimization specified by *View*.

A red icon in the *View* list indicates that the optimization was terminated because of a simulation error. A green icon indicates that no simulation error occurred.

> The green icon does not indicate that the error value for that optimization is zero. The error will be non-zero if the optimization was terminated for other reasons (for example, maximum iteration limit reached, gradient is zero).

Turning off the Optimization Cockpit

You can turn off the Optimization Cockpit by clearing the **Enable Optimization Cockpit** checkbox on the *Parameters* tab of the Optimization component.

7. Default Keyboard Shortcuts (hotkeys)

To customize the Keyboard/Toolbar, use the Main window command

Tools > HotKey/Toolbar Configuration

New Design	Ctrl+N
Open Design	Ctrl+0
Save Design	Ctrl+S
Print Design	Ctrl+P
Close Window	Ctrl+F4
Edit Undo	Ctrl+Z
End Command	ESC
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Delete	Del
Rotate	Ctrl+R
Mirror About X	Shift+X
Mirror About Y	Shift+Y
Move	Ctrl+M
Move Component Text	F5
Select all	Ctrl+A
View All	Ctrl+F
Restore Last View	Ctrl+L
Zoom In X2	Add (+)
Zoom Out X2	Sub (-)
Clear Highlighting	F8
Grid Display	Ctrl+G

Insert Wire	Ctrl+W
Insert Text	Ctrl+Shift+T
Insert Polygon	Ctri+shift+P
Insert Rectangle	Ctrl+Shift+R
Undo Vertex	Shift+Z
Snap Enabled (toggle)	Ctrl+E
Window Schematic	Ctrl+Shift+S
Window Layout	Ctrl+Shift+L
Simulate	F7
Help Whats this	Shift+F1
Help Topics and Index	F1

8. Built-in Constants, System Units and Prefixes

8.1. Built-in Constants

The following constants can be used in expressions:

Constant	Description	Value
PI (also pi)		**3.1415926535898**
e	Euler's constant	2.718281822
ln10	Natural log of 10	2.302585093
boltzman	Boltzman's constant	1.380658e-23 J/deg.K
qelectron	Electron charge	1.60217733e-19 C
plank	Plank's constant	6.6260755e-34 J-sec
c0	Speed of light in free space	2.99792e+8 m/sec
e0	Permittivity of free space	8.85419e-12 F/m
u0	Permeability of free space	12.5664e-7 H/m
i,j	Sqrt(-1)	1j

8.2. System Units

Type	Units
Angle	Deg
Capacitance	F
Concuctance	Sie
Current	A
Frequency	Hz
Inductance	H
Length	m, inch, mil
Linear Power	W
Power (dBm)	dBm
Resistance	Ohm
Time	sec
Voltage	V

8.3. System Prefixes

Factor	Meaning	System Prefix
10e12	Tera	t or T
10e9	Giga	g or G
10e6	Mega 1)	M (only)
10e3	Kilo	k or K
10e-2	Centi	c or C
10e-3	Milli 1)	m (only)
10e-6	Micro	u or U
10e-9	Nano	n or N
10e-12	Pico	p or P
10e-15	Femto 1)	f or F

1) The prefixes m, M and f (or F) must be followed by a unit description (MHz, fF, mOhm, MOhm) or they will be ignored.