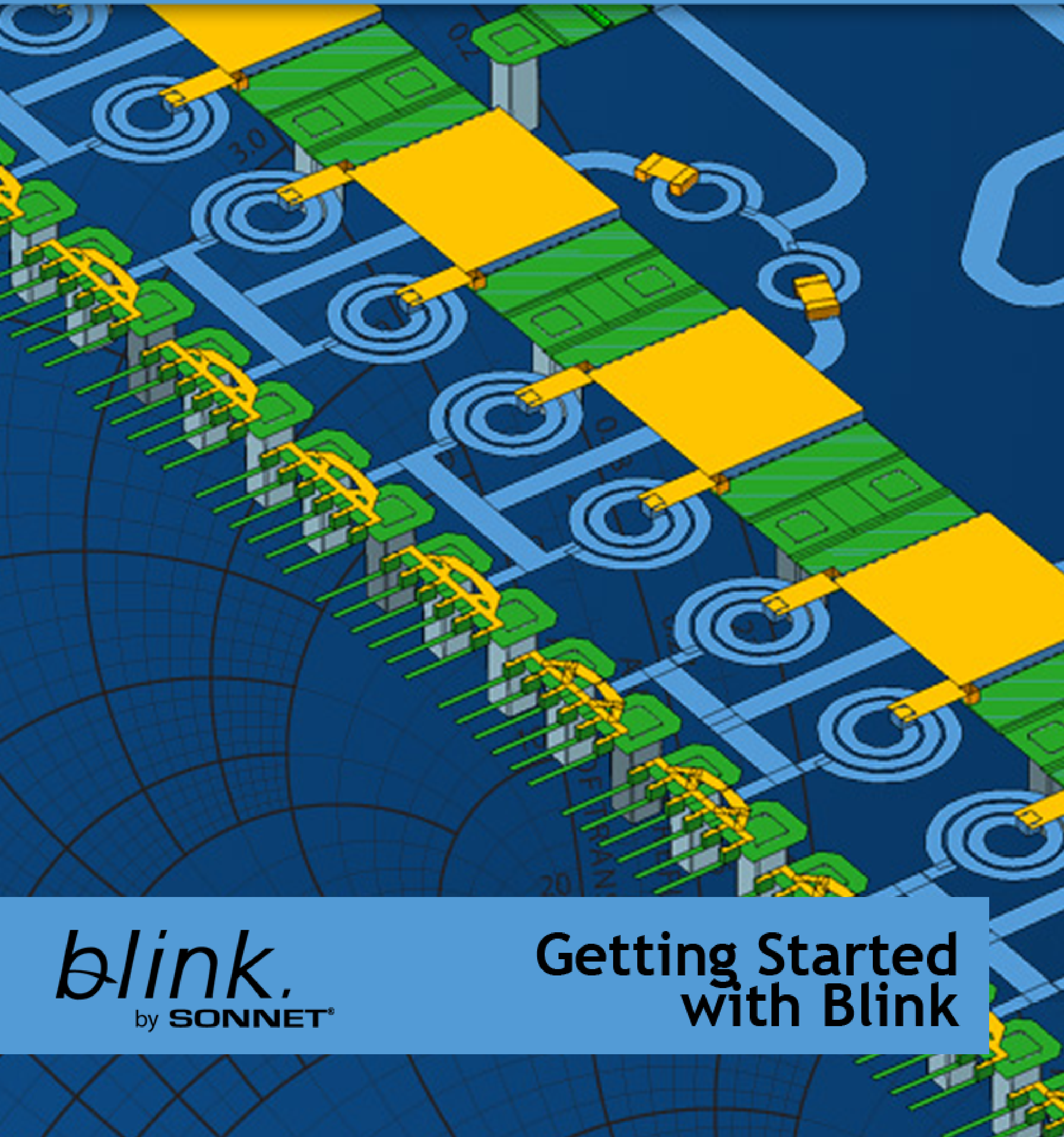
High Frequency Electromagnetic Software

SONNET® Blink 16



blink by SONNET®

Getting Started with Blink

At Sonnet, we've been developing 3D planar high frequency EM software since 1983, and our software has earned a solid reputation as the world's most accurate commercial planar EM analysis package for single and multi-layer planar circuits, packages and antennas.

Sonnet Software Inc., founded by Dr. James C. Rautio, is a private company, entirely dedicated to the development of commercial EM software. We take great pride in providing quality technical support for our products with timely response - which we believe to be very important for high-end technical software products.

Sonnet is based in Syracuse, NY, USA with representatives across the globe.

GETTING STARTED WITH BLINK

Published: November 2015

Release 16

Sonnet Software, Inc.
100 Elwood Davis Road
North Syracuse, NY 13212
Phone: (315) 453-3096
Fax: (315) 451-1694
Technical Support: support@sonnetsoftware.com
Sales Information: sales@sonnetsoftware.com
www.sonnetsoftware.com

Registration numbers: TX 2-723-907, TX 2-760-739

Rev 16.52

TABLE OF CONTENTS

CHAPTER 1 WELCOME

Welcome to Sonnet’s Blink. Blink is the first multi-solver passive device modeling suite that operates entirely from within the Cadence Virtuoso environment. Focus your efforts on speed or accuracy with Blink's two full-wave electromagnetic (EM) solvers. Using Blink allows you to choose the new Fast solver for speed, and Sonnet’s traditional shielded method-of-moments (MoM) solver, ***em***, for accuracy.

Complementing Sonnet’s traditional and highly Accurate solver, the new Fast solver uses advanced algorithms to accelerate your device modeling process. Both solvers operate with PDK-specific stackups, which can be customized for your implementation.

As an integrated circuit (IC) designer, you can now easily model new and unique passive devices within your normal Virtuoso-based IC design flow. Blink’s new Fast solver enables you to quickly characterize spiral inductors, baluns, transformers, MIM Caps, interdigitated capacitors, and transmission lines for radio frequency integrated circuit (RFIC) and mixed signal IC design.

Blink gives you the confidence and speed that you need from your modeled devices all within one package.

Documentation

Sonnet provides manuals for both the Blink interface and the native environment. The native environment refers to programs in Blink which are used when editing with the native editor, Sonnet's project editor, or accessing programs outside of the Cadence Virtuoso design flow. Some user's will work in the native environment at times to take advantage of advanced features not available in the Cadence Virtuoso Interface or to use another translator or interface to Sonnet Blink, such as the GDSII Translator. Context sensitive help for the Accurate solver's native environment is also included, as detailed below.

MANUALS: Blink comes with a set of manuals in PDF Format. PDF format manuals are accessed by selecting *Sonnet* ⇒ *Manuals* in the CIW window or *Help* ⇒ *Manuals* in the Cadence Virtuoso Interface main window. There is a tutorial on using Blink in this manual; please see Chapter 5 "Blink Tutorial" on page 29. The main reference document for Blink is the Blink User's Guide.

HELP: Only provided for the native environment. May be accessed in any of the following ways:

- Selecting *Help* ⇒ *Sonnet Help* from any native environment application
- Clicking on the Help button in any dialog box in the native environment
- In any dialog box in the native environment, press Shift+F1, then click on the desired control.

Sonnet Technical Support

- email: support@sonnetsoftware.com
- Voice: (315)453-3096
- Toll Free in US: (877) 776-6638
- Web site: www.sonnetsoftware.com/support

CHAPTER 2 INSTALLATION

Installation

Please refer to the Linux Installation PDF manual in your download. If you are installing from a CD-ROM, the PDF is available in the top level CD-ROM directory. If you are a new Blink user, once you have completed your installation return to the tutorial in this manual on page 29.

To set up remote ***em*** computing, please refer to the **Remote *em*** Computing manual available on the Administration page of the manual interface. The manual interface may be opened using the Manuals button on the Sonnet task bar after you have installed Sonnet.

System Requirements

For a complete list of requirements and testing updates, please see

www.sonnetsoftware.com/blink-requirements

Licensing

The Blink package includes licensing for the following:

- Accurate solver, ***em***
- Fast solver
- Cadence Virtuoso Interface
- Agilent Interface
- GDSII Translator
- DXF Translator
- Broadband Spice Extractor (bbextract)
- Remote EM computing

The Blink package also includes licensing for the following native environment programs:

- Project Editor
- Response Viewer
- Analysis Monitor
- Current Density Viewer

Loading the Cadence Virtuoso Interface in Cadence

The Cadence Virtuoso Interface must be loaded when you open Cadence. You may load the interface manually in the Command Interpreter Window (CIW) after launching Virtuoso or you may set up an initialization script that automatically loads the Cadence Virtuoso Interface each time the Cadence program is launched.

Automatic Loading

If you wish to have the Cadence Virtuoso Interface loaded each time you open the CIW, follow the instructions in the file "< Sonnet Directory>/sonnet_virtuoso_dk/virtuoso.txt" where < Sonnet Directory> is the directory in which the Sonnet software was installed.

NOTE: **You must have administrator privileges to make these changes.**

If the load is successful you will receive the following message in the CIW:

```
The Sonnet context file "< Sonnet Directory>/bin/
sonnet_virtuoso.cxt" was loaded and initialized.
```

If Sonnet has not been installed properly, the following error message appears:

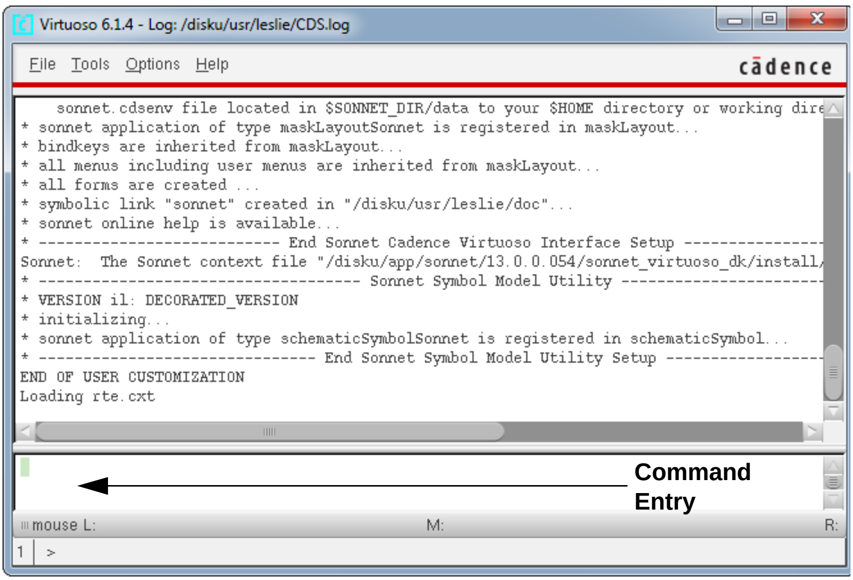
```
The Sonnet Directory is not set and the Sonnet Cadence
Virtuoso Interface is unable to be installed.
```

Manual Loading

To load the interface manually from the Command Interpreter Window (CIW), do the following:

1 **Invoke Cadence.**

The CIW appears on your display.



2 **Enter the command below at the prompt in the CIW.**

loadi("< Sonnet Directory>/sonnet_virtuoso_dk/install/sonnet_virtuoso_install.il")

where < Sonnet Directory > is the directory in which Sonnet is installed. For example, if you installed the software to "**`/home/sonnet`**", then type:

loadi("/home/sonnet/sonnet_virtuoso_dk/install/sonnet_virtuoso_install.il")

Again, note that you must substitute your pathname in place of "/home/sonnet."

If you choose to load the interface manually, you must enter this command each time Cadence Virtuoso is invoked.

Working Directories

The Cadence Virtuoso Interface creates several directories in your working directory which are used by the interface. The default working directory is $HOME/Sonnet. Two subdirectories are created in your working directory to store various files used in the interface. The two directories are:

$HOME/Sonnet/matl

$HOME/Sonnet/.sonnet_states

Both of these directories should have read and write permissions set. It is possible to change these directories to directories of your choice in the interface. For details about changing these directories from the default, see "Changing Working Directories."

Customizing Your Environment

Sonnet supplies a list of environmental variables for the Cadence Virtuoso Interface with your Sonnet installation which you may use to customize your Virtuoso environment. For more information please see "Customizing the Virtuoso Environment," page 35 in the **Blink User's Guide**.

Chapter 3 What's New in Blink Release 16

Version 16.52

Below is a summary of the major new features in release 16 of Blink. For a complete list of changes and bug fixes, please refer to the release notes, found at:

www.sonnetsoftware.com/release-notes

Increase in Threads for High Performance Solver: The thread count for the High Performance Solver has increased from 32 to 48 threads for release 16 improving processing time for larger circuits.

64-bit Support: The Cadence Virtuoso Interface now supports 64-bit.

Thick Metal Extrusion: When using the thick or rough metal type models for planar metal, you may now choose which direction from the metal layer, up or down, that you wish the metal to extend. For more information, see "material setup in the Cadence Virtuoso Interface" in the Index of Help.

Unified Ground Connections: This feature allows you to add an ideal short circuit to your circuit in locations other than the analysis box sidewall, and is especially useful for grounding internal shields and ground planes. For more information, please see "Using Port Number Zero for Ground Connections" on page 71 of the **Native Environment's User Guide** for more information.

ABS Preview: The ABS preview allows you to stop an adaptive sweep which has not yet converged in order to evaluate the data and possibly determine a new analysis band for your simulation. For more information, please see "simulate and release" in the Index of Help.

DC Point Extrapolation in ABS Sweep: Starting an ABS sweep at 0.0 GHz automatically generates data for a DC point by extrapolating from the ABS sweep data. Please see "Add Frequency Type" in the Index of Help for more information.

Enhanced Resonance Detection: Automatic Adaptive Band Synthesis (ABS) resolution adjustment to resolve extremely narrowband resonances as is typical for superconductor applications. Please see "Enhanced Resonance Detection" on page 151 of the **Native Environment's User Guide** for more information.

Robust Loss Metals: Improved loss models for multiple metal layers stacked on top of each other. See "Modeling Plated Metal" on page 305 and "Modeling Conductor-Via-Conductor Stackups" on page 305 of the **Native Environment's User Guide** for more information.

Resistance Per Via: The Volume and Array Via metal loss models now support a resistance per via definition for contact resistance dominated vias. For more information, please see "Via Metalization Loss" on page 52 of the **Native Environment's User Guide**

Automatic Bar Vias: There is a new via metal fill model that models the bar vias typically used in many RFIC processes in a more efficient manner. Bar vias may be automatically identified during translation. For more information, please see the "Via Simplification" appendix in the **Blink User's Guide**.

Polygon Edge Checking Support for Technology Layers: In previous releases, there was an option to have the analysis engine, ***em***, consider adjacent metal levels when computing the subsectioning. In this release, you may now consider adjacent technology layers. For more information, please see "polygon checking" in the Index of Help.

Via Simplification More Efficient for RFIC Circuits: Improvement in the array simplification feature now makes it possible to efficiently analyze circuits with 100,000+ microvias.

GDSII Translator 64-bit Support: The GDSII translator and via simplification now provide 64-bit support.

Keysight ADS Interface Enhancements: Supports the new advanced material settings in Release 16. There is also a new tutorial available for the Keysight ADS Interface. Please see Chapter 2 "Keysight ADS Interface Tutorial" on page 15 of the **Keysight ADS Interface** manual.

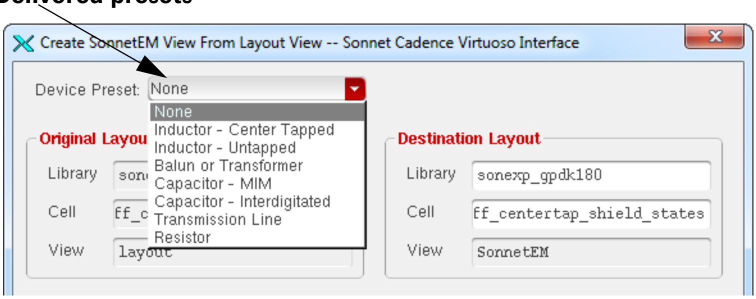
CHAPTER 4 THE BASICS

This chapter is intended to provide you with basic information about Sonnet Blink which will allow you to quickly and accurately model your circuit while avoiding some of the common mistakes made during EM simulations. The chapter which follows after contains a tutorial on using Blink.

Presets

To streamline the setup for simulation, designers can select a device preset when creating the SonnetEM view. The preset loads settings in the Cadence Virtuoso Interface that are typical for that type of device. A basic set of presets are delivered with your software, as pictured below. Additional presets may be created by a system administrator in order to customize your interface; therefore, what appears in your preset list may vary.

Delivered presets



Loading a preset changes any fields specified in the preset to the values specified in the preset. You may subsequently change those settings in the Cadence Virtuoso Interface without affecting the preset; it will still contain the original values the next time it is used. Presets may only be specified when creating a new SonnetEM view.

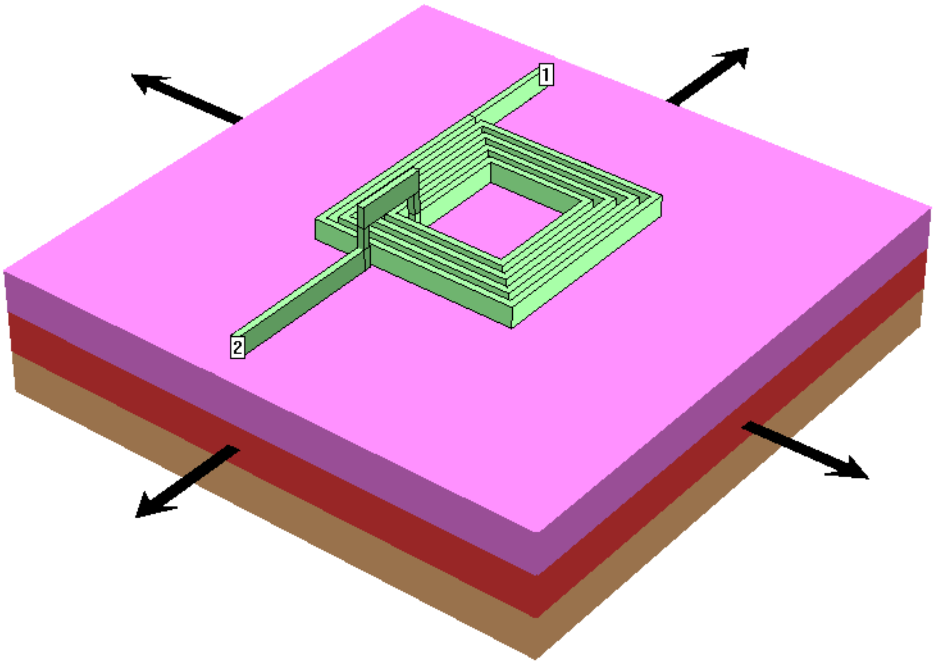
You may not edit a preset delivered with the software, but it is possible for your system administrator to create new customized presets. Please have your system administrator contact Sonnet Technical Support for more information on creating customized presets. A preset can contain all of the settings in the interface or any subset thereof.

Analysis Environment

The Fast solver and Accurate solver use different environments for analysis.

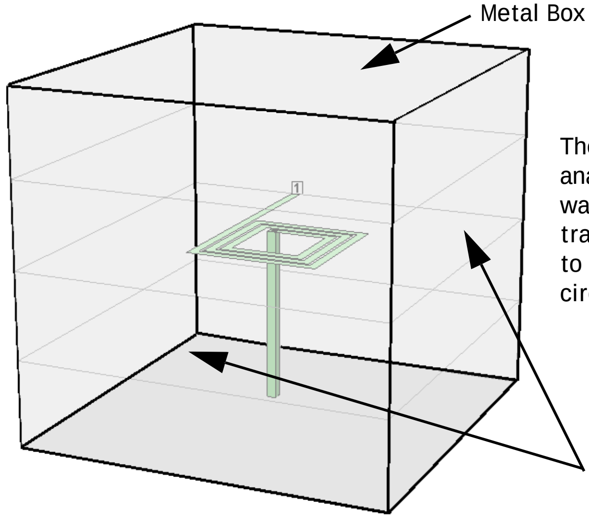
Fast Solver

The Fast solver simulates in an unshielded environment with a lossless ground plane underneath the substrate. The dielectrics and the ground are laterally infinite in extent.



Accurate Solver

The Accurate solver uses a shielded environment, the analysis box. The following section provides information about the Accurate solver's analysis box.



When using the Accurate solver, ***em***, the Sonnet EM analysis is performed inside a six-sided metal box as shown above. This box contains any number of dielectric layers which are parallel to the bottom of the box. Metal polygons may be placed on any or all of the dielectric layers, and vias may be used to connect the metal polygons on one level to another.

The four sidewalls of the box are lossless metal, which provide several benefits for accurate and efficient high frequency EM analysis:

- The box walls provide a perfect ground reference for the ports. Good ground reference is very important when you need S-parameter data with dynamic ranges that might exceed 50 or 60 dB, and Sonnet's sidewall ground references make it possible to provide S-parameter dynamic range that routinely exceeds 100 dB.
- Because of the underlying EM analysis technique, the box walls and the uniform grid allows the use of fast-Fourier transforms (FFTs) to compute all circuit cross-coupling. FFTs are fast, numerically robust, and map very efficiently to computer processing.
- There are many circuits that are placed inside of housings, and the box walls provide a natural way to consider enclosure effects on circuit behavior.

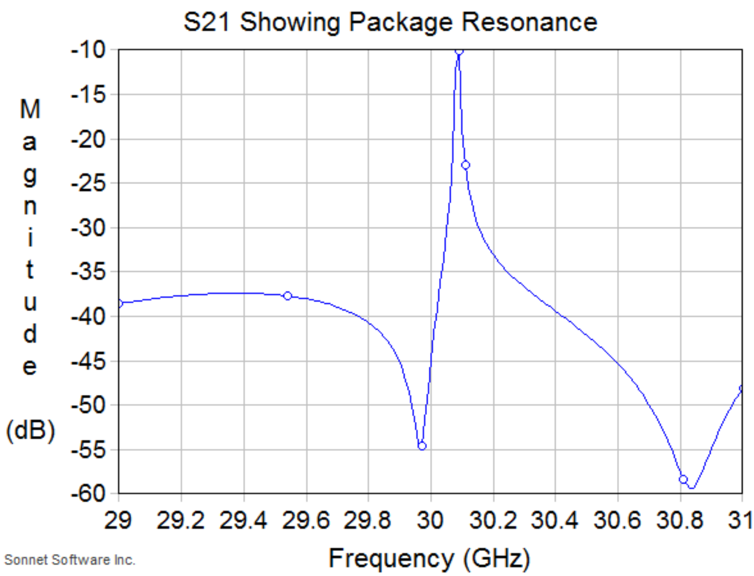
The top and bottom of the box may have any loss, allowing you to model ground plane loss.

Coupling to the Box

Since the four sidewalls of the box are lossless metal, any circuit metal which is close to these walls can couple to the walls - just like what would happen if you fabricated and measured a real circuit with the same box. If you do not want to model this coupling (for example, your real circuit does not have sidewalls), then you must keep the circuit metal far away from the box sidewalls. The presets provide a reasonable value for the analysis box size assuming your real circuit does not have sidewalls.

Box Resonances

It is possible for the six conducting sides of the analysis box to create a resonant cavity, just like you would see if you fabricated and measured the circuit in the same size box. You can use Blink to predict unwanted box resonances in the package or module housing the circuit. Box resonances usually appear as a "spike" in the S-Parameter magnitude and phase data, as shown below.



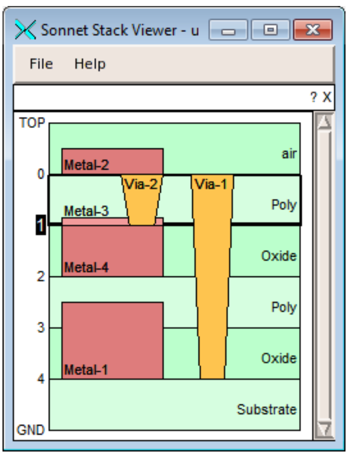
You can detect box resonances by observing your simulated results, through runtime warning messages or by using the Box Resonance Estimator. For more information, please see Step 11 on page 15 of the **Blink User's Guide**.

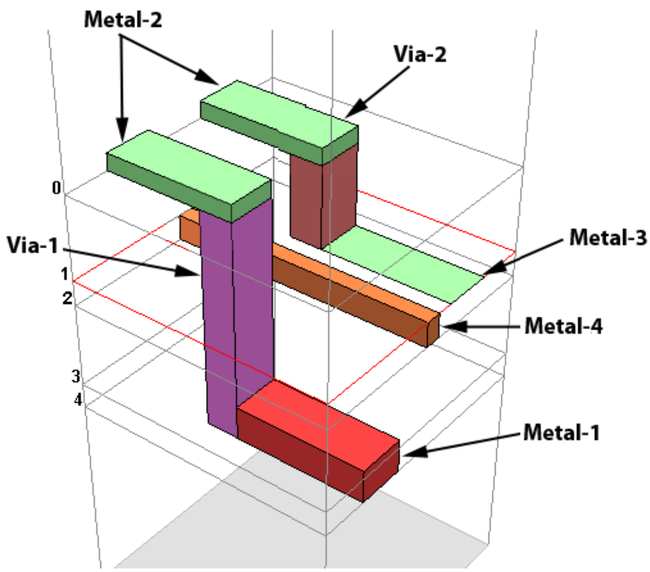
Dielectric Layers and Metal Levels

All geometry projects are composed of two or more dielectric layers. There is no limit to the number of dielectric layers in a Sonnet geometry project, but each layer must be composed of a single dielectric material. Metal polygons are placed at the interface between any two dielectric layers if they are using a zero-thickness model. When using Sonnet's thick metal model, then the metal extends upward into the dielectric above it. The Thick metal model provides more accuracy but may increase processing time and memory requirements. Vias may also be used to connect metal polygons on one level to metal on another level.

You can use the Stackup Viewer to view the dielectrics and metal levels in your circuit. Each time a new dielectric layer is added, a corresponding metal level is also added to the bottom of the new dielectric layer. The creation of the stackup and definition of materials is done in the Substrate Setup dialog box in the Cadence Virtuoso Interface. The stackup viewer is also accessed in this dialog box. Please see Step 10 on page 15 of the **Blink User's Guide** for more information.

The example below shows a 3-D drawing of a circuit (with the z-axis exaggerated). Please note that the pictured circuit is not realistic and is used only for purposes of illustrating the box setup.





If a Metal Technology Layer is assigned a thick metal (Metal-1, Metal-2 and Metal-4), then how it appears in the stackup viewer is based on its relationship to the dielectric layer above it. The thick metal height is not drawn to scale in the stackup viewer, but if its thickness is less than that of the dielectric above it, then the symbol covers half of the thickness of the dielectric layer (Metal-2). If it is the same thickness as the dielectric layer then the symbol is drawn the same height as the dielectric (Metal-4). If it is thicker than the dielectric then it also extends half of the height of the dielectric layer above (Metal-1). A zero-thickness metal (Metal-3) is represented with a thin rectangle in the stackup viewer. Note that all dielectric layers are represented as the same height in the stackup viewer regardless of their actual physical thickness. The actual physical thickness is drawn to scale in the 3D view on the right.

Dielectric Layers

A dielectric layer refers only to dielectrics, not metals. In the example above, there are six dielectric layers. There is an entry for each dielectric layer in the Sonnet Dielectrics dialog box. The dielectric layers are entered in the Sonnet Dielectrics dialog box in the Sonnet Cadence Virtuoso Interface or read in from an imported substrate file. For more information, please see Step 10 on page 15 of the **Blink User's Guide**.

Metal Levels

Metal levels are defined as the location directly below a dielectric layer. Metal levels are labeled in the stackup with a integer starting at 0 and increasing as you go down the stack. In the example above, there are five metal levels in addition to the box top and bottom. The bottom of the box is referred to as the GND level and is the lowest level in your stackup. The top and bottom of the box are lossless metal by default, but can be changed in the Analysis Box Options dialog box in the Cadence Virtuoso Interface. You can use as many different metal types as you wish on a single metal level; for instance, you may use a silver polygon and copper polygon on the same metal level.

As mentioned above, the metal level is associated with the dielectric layer above, such that when you delete a dielectric layer, the metal level directly below the layer is deleted.

You can place polygons on the GND level, but they have no effect because this level is already completely metalized. However, vias and dielectric bricks may be placed on the GND level.

Technology Layers

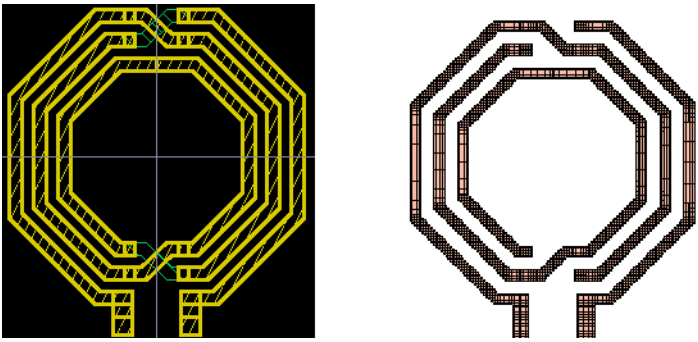
A Technology Layer allows you to define a group of objects with common properties including the metal level on which they are placed in your Sonnet project. This enables you to more easily control a large group of objects in your circuit and make changes to those objects as efficiently as possible. Technology Layers provide uses and advantages similar to using drawing layers in CAD and EDA programs. There are three types of Technology Layers: Metal, Via and Brick. Technology Layers are displayed in the Stackup Viewer. Metal Technology Layers are displayed in red, Via Technology Layers in yellow and Dielectric Technology Layers in blue. Technology Layers are automatically created in a translated Sonnet project using the mapping from the Cadence environment.

Cell Size and Meshing

Sonnet’s solvers calculate the response data for your circuit by dividing the metal into small “subsections” and analyzing the coupling between these subsections. However, determining the subsectioning, or meshing, is based on different algorithms for each solver.

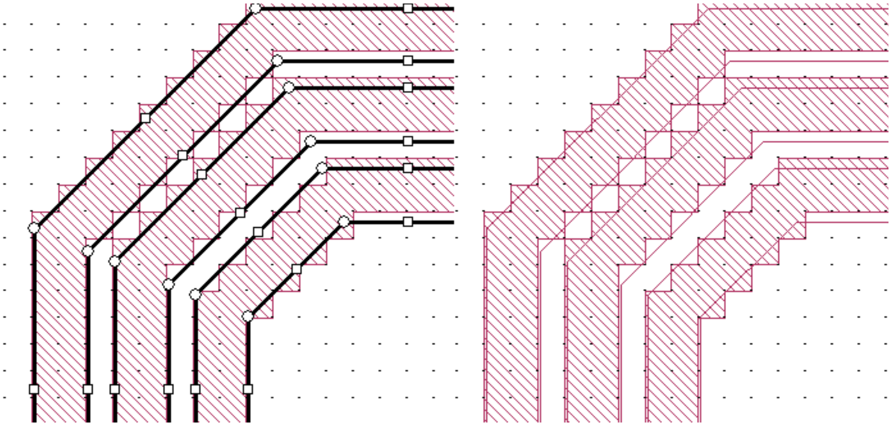
Accurate solver, *em*, Meshing

The ***em*** meshing is based on a uniform cell size. One or more cells are automatically combined together to create subsections. Cells may be square or rectangular (any aspect ratio), but must be the same over your entire circuit. Note that the cell dimensions do not have to be an integer nor do they have to be equal. For example, you may define a cell size of 1.687 by 2.453 microns.



The picture on the left shows the circuit as viewed in Cadence Virtuoso. On the right is shown the subsectioning used by the Accurate solver, *em*, in analyzing the circuit.

Metal polygons in a Sonnet project are represented with an outline and a cell fill pattern. The outline represents exactly what was translated from Cadence Virtuoso. The cell fill represents the actual metalization analyzed by the Accurate solver, ***em***. Therefore, the actual metalization analyzed may differ from that input by you as illustrated below.We recommend that you open your translated Sonnet project in the project viewer to check the metalization before analyzing. As you can see on the right, in the illustration below, the actual metalization does not provide a gap between the traces of the inductor. You turn the cell fill on and off in the project viewer using the *View* ⇒ *Cell View* command.



The cell size is specified in the main window of the Cadence Virtuoso Interface. The analysis solves for the current on each subsection. Since multiple cells are combined together into a single subsection, the number of subsections is usually considerably less than the number of cells. This is important because the analysis solves an N x N matrix where N is the number of subsections. A small reduction in the value of N results in a large reduction in analysis time and memory.

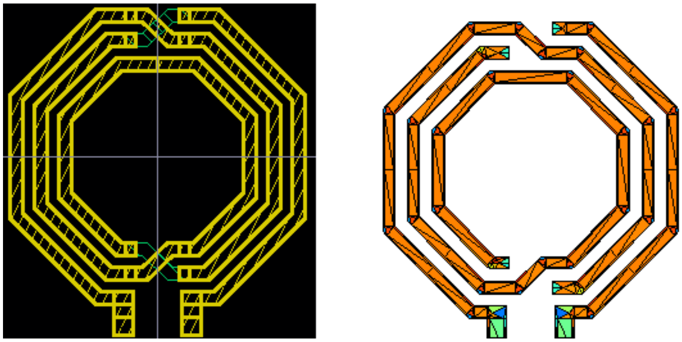
Care must be taken in selecting a cell size so that accuracy is not sacrificed. If the cell size is too large, then your results will not be accurate. If the cell size is too small, the analysis may become too costly in processing time. ***Em*** automatically places small subsections in critical areas where current density is changing rapidly, but allows larger subsections in less critical areas, where current density is smooth or changing slowly.

It is recommended that you run the Connectivity Checker (*Tools* ⇒ *Check Connectivity*) to ensure that no opens or shorts were caused by the cell size or placement of the geometry on the grid. For more information on the Connectivity Checker, please see “Creating the Models and Model Views,” page 17 in the **Blink User’s Guide**.

For a more detailed discussion of cell size and subsectioning, please refer to Chapter 2 "Subsectioning" on page 21 of the **Native Environment User’s Guide.**

Fast solver Meshing

The Fast solver analyzes in an unshielded environment and uses triangular subsections. You may choose to allow the software to automatically control the size of the subsections or manually enter a size. The Maximum Mesh Size is entered on the main page of the Cadence Virtuoso Interface. An example of Fast solver meshing is shown below.



The picture on the left shows the circuit as viewed in Cadence Virtuoso. On the right is shown the subsectioning used by the Fast solver in analyzing the circuit.

Chapter 5 Blink Tutorial

Introduction

The Cadence Virtuoso Interface allows you to use Sonnet's electromagnetic solvers from within the Cadence Virtuoso environment. This tutorial provides you with an example of a basic translation and analysis of a cell view in Blink using both the Fast and Accurate solvers. The whole process from creating a SonnetEM view to creating a model symbol with the analysis results is covered.

During this tutorial you will first set up and run an analysis using the Fast solver, then run an analysis using the Accurate solver to verify your results. However, the tutorial is structured so that you may perform an analysis using only the Fast solver, only the Accurate solver, or both. Instructions are provided during the tutorial.

The following topics are covered:

- Creating the initial SonnetEM cell view from your layout
- Importing a substrate file (.matl)
- Analysis Box setup (Accurate solver only)
- Ports
- Setting up analysis frequencies and run options
- Specifying output models
- Running a simulation
- Viewing simulation results

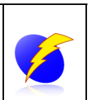
- Loading your model in the Analog Design Environment (ADE)

This tutorial assumes that you are already familiar with using Cadence Virtuoso and that Sonnet's Cadence Virtuoso Interface is already installed. The Cadence Virtuoso Interface is automatically installed when you perform your Blink Installation. Please refer to the Linux Installation Manual for instructions on installing Sonnet software. However, you do need to ensure that the Cadence Virtuoso Interface is loaded when you start Cadence Virtuoso. For instructions, please see "Loading the Cadence Virtuoso Interface in Cadence," page 10.

NOTE: This tutorial uses the IC6 version of Cadence Virtuoso. Some procedures, commands and dialog boxes in Cadence will differ slightly for IC5 users.

Solver Icons

Throughout the rest of the tutorial, if a section applies to only one of the solvers, the section heading will be preceded with an icon indicating to which solver the section is applicable. If the section applies to both solvers, then no icon appears. The icons are as shown below:

Fast solver	
Accurate solver	

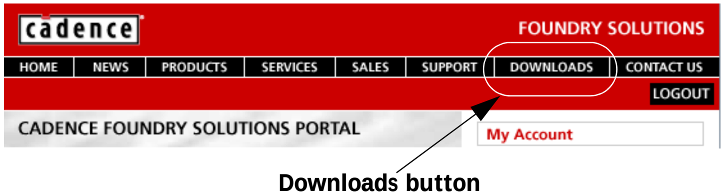
Copying Examples

This tutorial requires that you have the generic PDK gpdk180 installed in Cadence. If the gpdk180 PDK is already installed, continue the tutorial at Step 5 below. If you need to install the PDK, do the following:

1 Go to https://pdk.cadence.com/home.do and login.

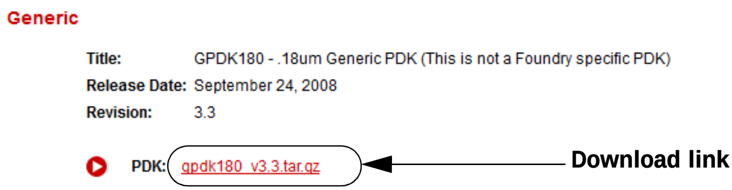
If you do not already have an account, then register and create one. Once you are logged in, continue.

2 **Click on the Downloads button in the banner across the top of the page.**



Your browser should now display the Downloads page.

3 **Scroll down and click on the "gpdk180 v3.3.tar.gz" link and download the file.**



You will have to read and agree to the license before downloading the PDK. This may take several minutes.

4 **Once the file has downloaded, open the file using your Archive Manager and extract the files to your PDK library location.**

If you need instructions on installing the PDK, please refer to the reference manual which was part of your download from Cadence. After extracting the files, you may find the manual at:

<Extraction location>/docs/gpdk180_referenceManual.pdf.

Creating the Library

5 **Copy the Sonnet Examples from the Sonnet Installation to your working directory.**

The examples may be found at the following location based on which version of Cadence Virtuoso you are using:"

Cadence's Virtuoso IC6.1x

<Sonnet Installation Directory>/sonnet_virtuoso_dk/examples/ic61

where <Sonnet Installation Directory> is the directory in which Sonnet is installed.

Copy the directory “sonexp_gpdk180” from the appropriate directory cited above to your working directory. The default for the working directory is <User home directory>/cadence/virtuoso/oa.

Cadence’s Virtuoso IC5.1x

<Sonnet Installation Directory>/sonnet_virtuoso_dk/examples/ic5141

where <Sonnet Installation Directory> is the directory in which Sonnet is installed.

Copy the directory “sonexp_gpdk180” from the appropriate directory cited above to your working directory. The default for the working directory is <User home directory>/cadence/virtuoso/cdb.

6 **If you have not yet started Cadence Virtuoso, do so now.**

7 **Select *Tools* ⇒ *Library Manager* from the main menu in the CIW.**

The Library Manager window appears on your display.

8 **Select *File* ⇒ *New* ⇒ *Library* from the main menu of the Library Manager window.**

The New Library dialog box appears on your display.

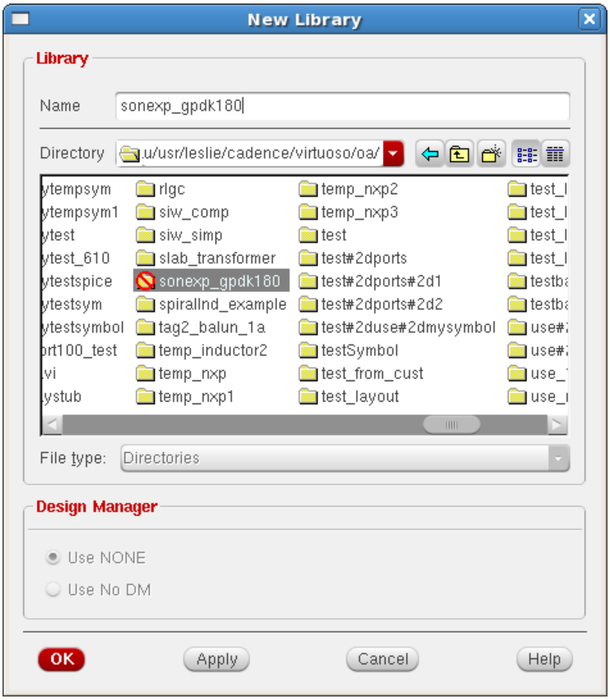
9 **Enter “sonexp_gpdk180” in the Name text entry box.**

This will be the name of the new library you are creating.

10 Select your working directory from the Directory drop list.

11 Select the "sonexp_gpdk180" directory from the Directory list below.

This should be the directory you copied in the previous step.



12 Click on the OK button in the New Library dialog box.

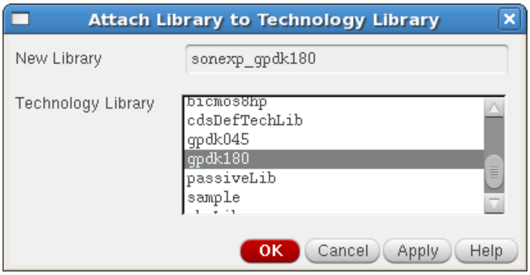
The Technology File for New Library dialog box appears on your display.

13 If it is not already selected, click on the "Attach to an existing technology library" radio button.

14 Click on the OK button in the Technology File for New Library dialog box.

The Attach Library to Technology Library dialog box appears on your display.

15 Select "gdpk180" from the list of technology libraries.



16 Click on the OK button in the Attach Library dialog box to close the dialog box and create the library.

The following message should appear in the CIW window:

```
INFO (TECH-180011): Design library 'sonexp_gpdk180' successfully attached to tech-
nology library 'gpdk180'.
```

Note that there are a number of cell views in the supplied library. You will be using only one of the cells for this tutorial, but the rest are also available for your use.

Creating a SonnetEM cell view

The first step in performing a Sonnet electromagnetic analysis is to create a SonnetEM cell view from your layout view of a cell in Virtuoso.

NOTE: **The definition of a "cell" in the Cadence Virtuoso environment is similar to the "circuit geometry" in Sonnet. When "cell" is used in the Sonnet environment, it is referring to the smallest part of the analysis grid used in subsectioning your circuit for the Accurate solver.**

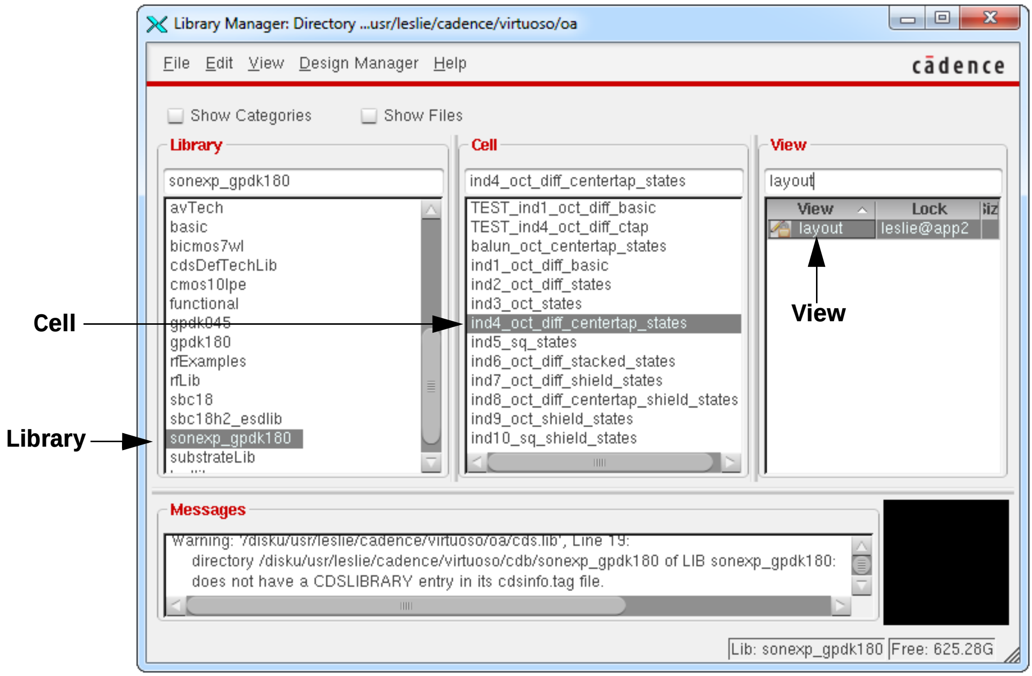
It is possible to use only part of a layout or the whole layout when creating a SonnetEM view. For this tutorial, we will be using the whole layout.

17 Click on "sonexp_gpdk180" in the Library list in the Library Manager window.

The Library Manager is updated with a list of cells. We will only be using one of the cells for this tutorial: ind4_oct_diff_centertap_states.

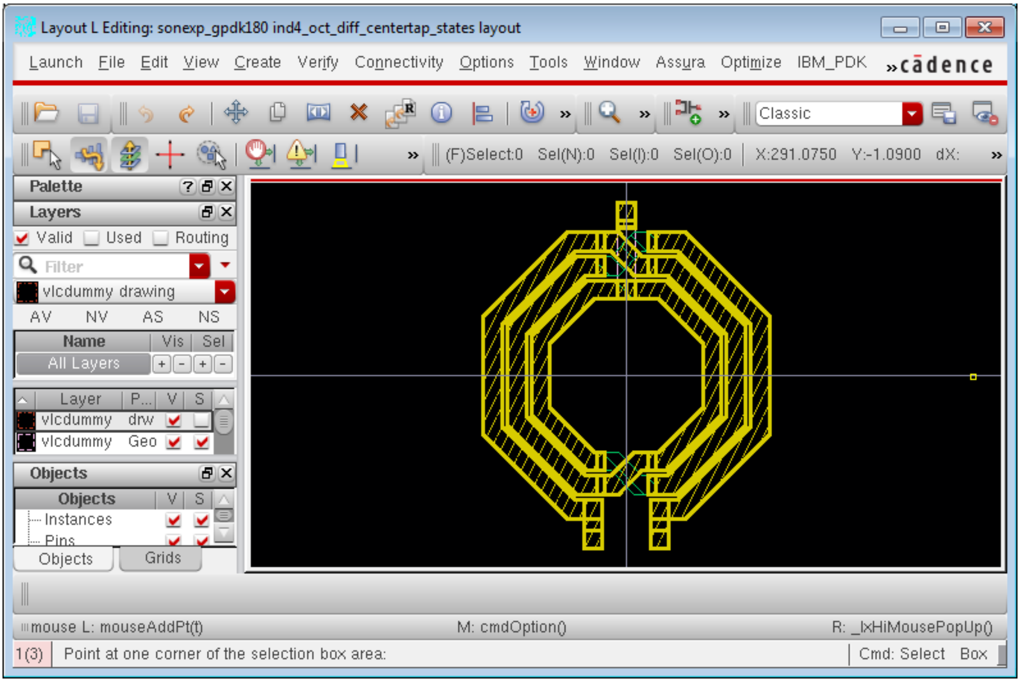
18 Click on the "ind4_oct_diff_centertap_states" cell in the Cell list.

The View list in the Library Manager is updated.



19 Double-click on “layout” in the View list to open the layout view of the ind4_oct_diff_centertap_states cell.

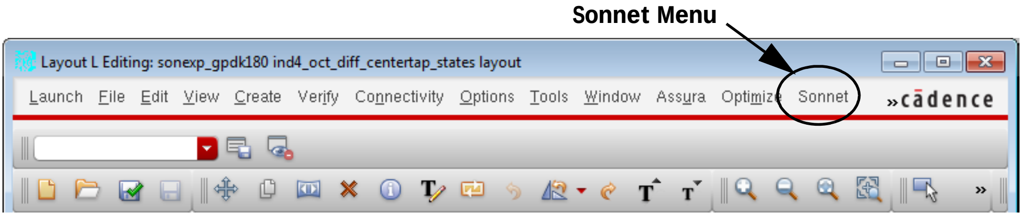
The layout view of the ind4_oct_diff_centertap_states cell appears on your display.



20 Select *Launch* ⇒ *Sonnet* from the main menu of the Layout window.

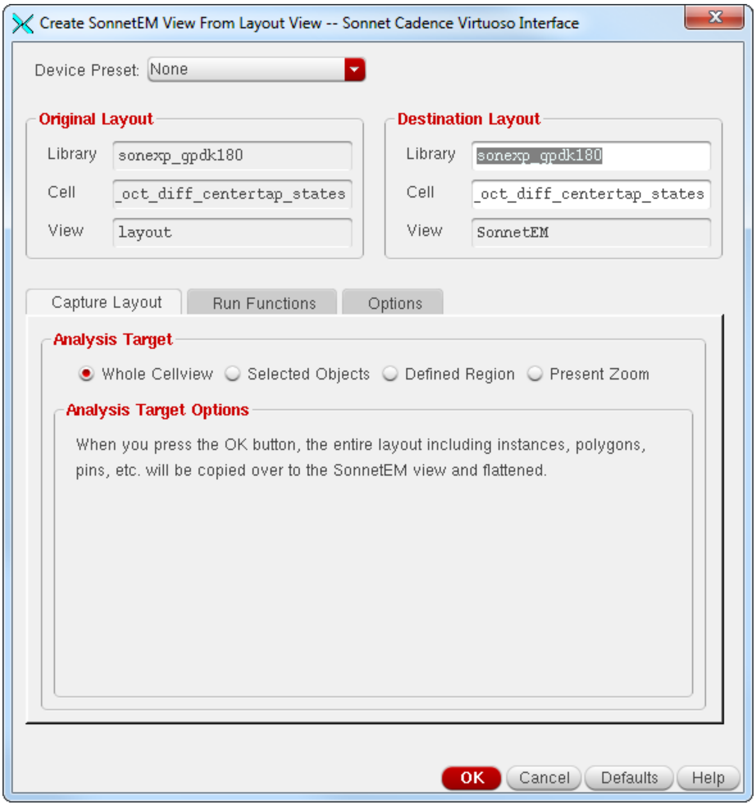
NOTE: **Note that if you are using IC5 the command to load the Sonnet menu is *Tools* ⇒ *Sonnet*.**

This command loads the Sonnet interface and needs to be used in any layout window in which you wish to use the Sonnet Cadence Virtuoso Interface. The Sonnet menu appears in the main bar of the layout window as shown below.



21 Select *Sonnet ⇒ Create SonnetEM view* from the main menu of the Layout window.

The Create SonnetEM View dialog box appears on your display. You must create a SonnetEM view in order to run a Sonnet simulation on your layout. This way the original layout is not affected. Also, when using only a part of your original layout, it is easier to determine exactly what you are analyzing by having a separate SonnetEM layout.



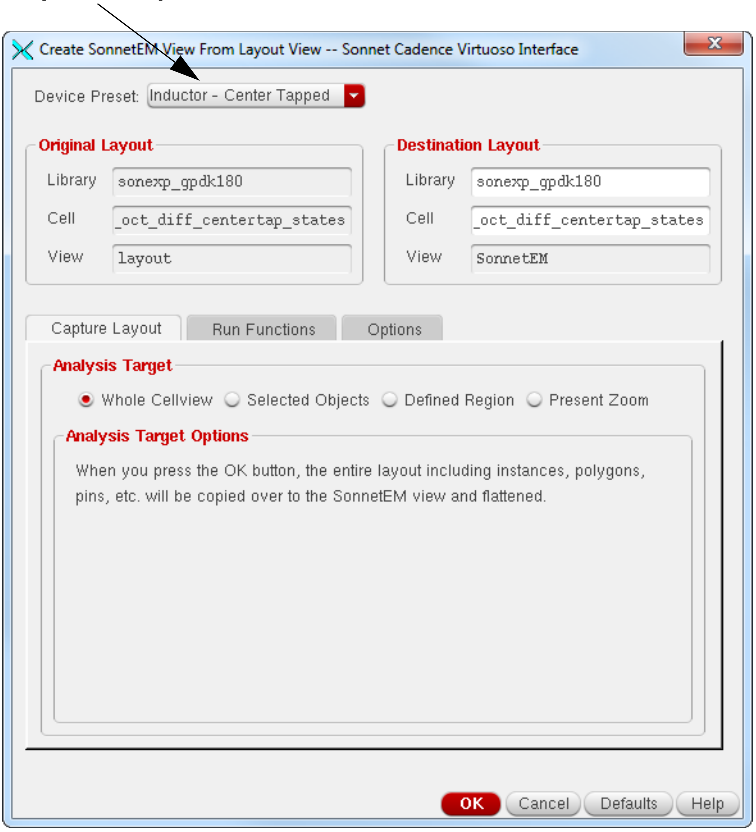
This dialog box displays the source used to create the SonnetEM view. The view of the resulting cell is also displayed. You may choose to use the same library and cell name or enter the desired Library name in the Library text entry box and the desired cell name in the Cell text entry box in the Destination Layout section of the dialog box. The default for both of these fields are the library and cell of the originating layout. We will be using these default settings.

The Analysis Target section of the dialog box allows you to select what part of the source cell you wish to include in the resulting SonnetEM view cell. The default is the whole cell view. In that case, everything in the source cell is translated into the SonnetEM view. Since we are using the default of the whole cell view, there is no need to change the radio button selection.

22 Select “Inductor - Center Tapped” from the Device Preset drop list.

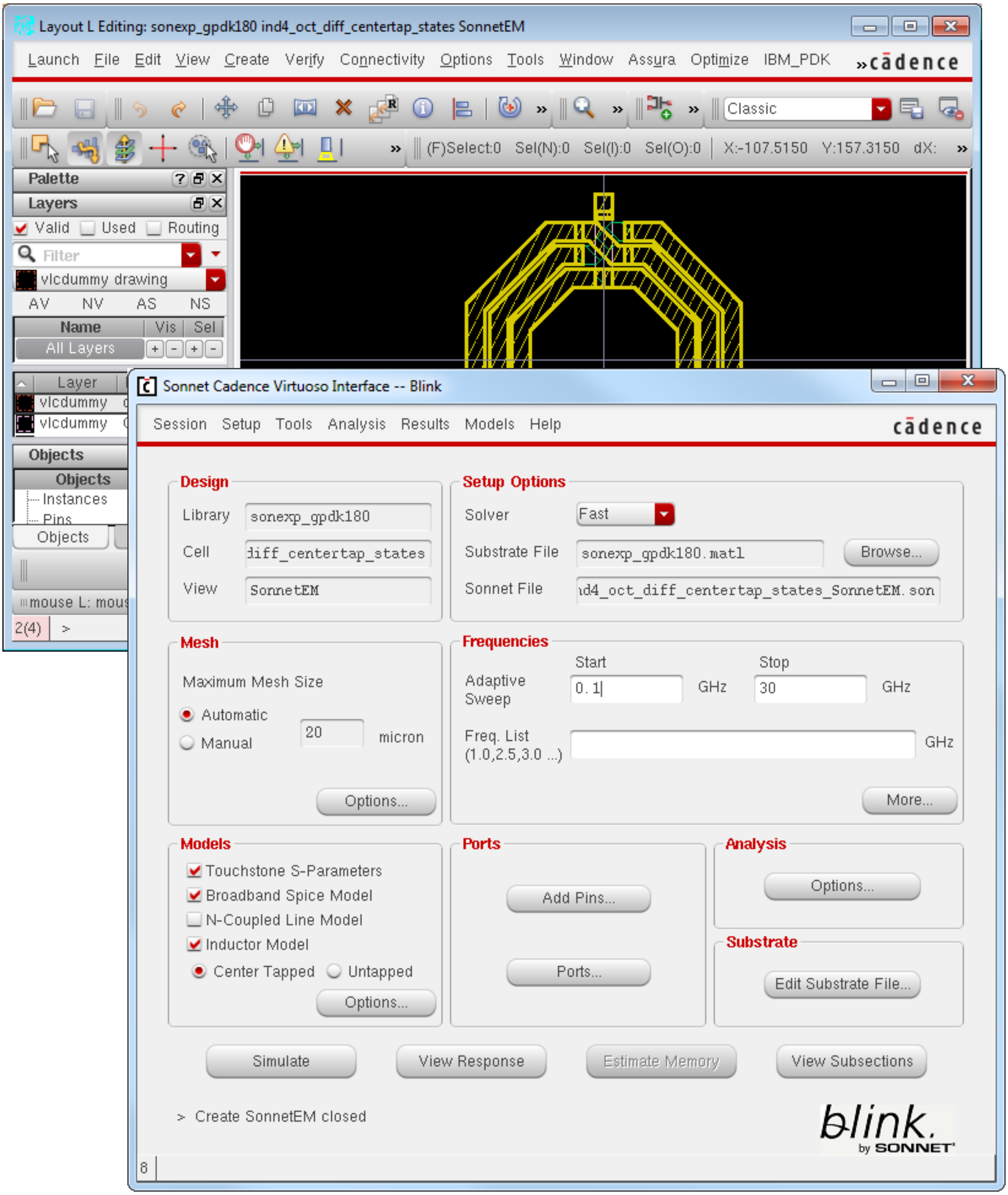
The example circuit we are using for this tutorial is a center tapped inductor. Choosing this preset defaults the settings in the Cadence Virtuoso Interface to the most common settings suitable for analyzing a center-tapped inductor. If any of the settings are not the values you wish to use for your analysis, they may be changed in the interface before running your simulation.

Device preset drop list



23 Click on the OK button to close the dialog box and create the SonnetEM view.

A new layout window appears which displays the SonnetEM view as well as the Sonnet Cadence Virtuoso Interface. The Sonnet Cadence Virtuoso Interface dialog box allows you to control all aspects of the analysis setup and simulation.



TIP

If you are not able to complete the tutorial in one sitting, use the *Session* ⇒ *Save State* command in the Cadence Virtuoso Interface before exiting. Then, you may access the Sonnet Cadence Virtuoso Interface by opening the SonnetEM view, launching the Sonnet menu (*Launch* ⇒ *Sonnet* for IC6 AND *Tools* ⇒ *Sonnet* for IC5) in the Virtuoso Layout window containing the SonnetEM view, then selecting *Sonnet* ⇒ *Open Interface Application*. Use the *Session* ⇒ *Load State* command to restore your previous progress.

Choosing the Solver

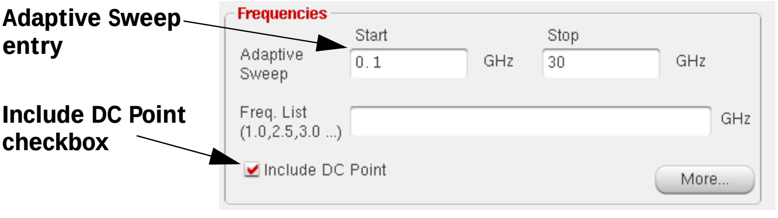
You have a choice between two solvers for your analysis: the Fast solver and the Accurate solver, ***em***. The default solver is dependent upon your Sonnet Cadence Environment variable settings. In this case, the Fast solver was selected in the preset used to create the SonnetEM view. Many of the setup tasks apply to either solver; however, some settings are only used for one of the solvers. For the purposes of this tutorial, we are going to perform the common setup tasks, then run a Fast solver analysis, followed by an Accurate solver analysis. If you wish to use only the Accurate solver, you should continue to follow the steps below, but will be directed later in the tutorial to skip over the parts unique to the Fast solver. For a discussion of solver differences and when each should be used, please see “Choosing the Solver,” page 10 of the Blink User’s Guide.

Analysis Frequencies

You may set up the analysis frequencies in the Sonnet Cadence Virtuoso window. For this circuit, we will do a broadband sweep by performing an Adaptive sweep from 0.1 GHz to 30 GHz and analyze at a DC Point.

An adaptive sweep provides a fine resolution frequency sweep by performing an EM simulation at a few discrete points, and then using a rational fitting algorithm to calculate the fine resolution frequency data. You enter the desired frequency band by entering a beginning and ending frequency.

The Presets for an inductor have already input an ABS sweep from 0.1 GHz to 30 GHz and selected the “Include DC Point” checkbox. If you had wished to analyze a different frequency band, you would have entered the desired values in the Start and Stop Text entry boxes.



If there are specific frequency points at which you wish to analyze, you may enter these frequencies in the Freq. List entry box; entries may be separated by a comma. It is also possible to define other types of frequency sweeps, by clicking on the More button to open the Frequencies dialog box. For details on this dialog box and other frequency sweep types, please click on the Help button in the Frequencies dialog box.

It is highly recommended that you use the automatic setting for the DC Point sweep, especially if switching between the Accurate and Fast solvers. The criteria for determining a good DC point changes based on which solver is selected, so entering a manual value may not produce good results for either solver. The Automatic setting is the default so you do not need to take any action. If you wish to set the DC frequency point manually, you would do so in the Edit Frequency Type dialog box. For details please click on the Help button in the Frequencies dialog box

Please note when using the Fast solver, a DC Point sweep may only be done following a previous ABS sweep, as is the case here. Sweep data is extrapolated to determine the response at DC, so a previous sweep must be run to provide the response data from which to extrapolate.

Because of the selected presets, the settings in the Frequencies section of the Cadence Virtuoso Interface are already set up correctly, so no action is required. This completes the discussion of the analysis frequencies, next we will set up the models we wish created at the end of the analysis.

Specifying Models

You select the models you wish created at the end of the analysis in the Models section of the Cadence Virtuoso Interface main window. Usually, there is no need to change the default values, but there are advanced settings available for each of the models. You access these by clicking on the Options button which opens the Advanced Model Options dialog box. For details, please click on the Help button in the Advanced Model Options dialog box.

The inductor presets have already selected the Touchstone S-Parameters, Broadband Spice, and Center Tapped Inductor models. If you are certain what models you will need for later use, you should select only those types of models.Be aware that for the Fast solver, if you did not create one of the models, it is necessary to run your simulation again to generate that model.

Since the desired models were all selected by the presets, you do not need to change any settings.

Setting up the substrate

A Sonnet substrate file contains all the information in a Tech Process file that is required to define the stackup and materials for your translated Sonnet project. You create a substrate file by editing the settings in the Substrate File Setup dialog box. Once a substrate file has been created, it may be imported into other SonnetEM views so that circuits which use the same stackup may be analyzed more efficiently without having to perform a manual setup each time. So, although it is possible to completely define your substrate in the Cadence Virtuoso Interface, including mapping the layers and objects in your Cadence cell to your Sonnet project and defining all the materials, normally all of this is accomplished by importing a substrate file (.matl) or by importing a Technology File and converting it to a substrate file. You may import and convert the following formats: Keysight, Assura, Helic, Integrand, or Sonnet project. Click on the Help button in the Substrate File Setup dialog box for more information on converting from other formats.

For this tutorial, we have supplied a substrate file, sonexp_gpdk180.matl, to import which defines your materials and dielectrics and contains the proper mapping from your Cadence cell to the Sonnet project.

To import the substrate file, do the following:

24 Copy the supplied .matl file to your Sonnet matl directory.

The example file may be found at:

<Sonnet Installation Directory>/sonnet_virtuoso_dk/examples/sonexp_gpdk180.matl

where <Sonnet Installation Directory> is the directory in which Sonnet software is installed.

You should copy this file to:

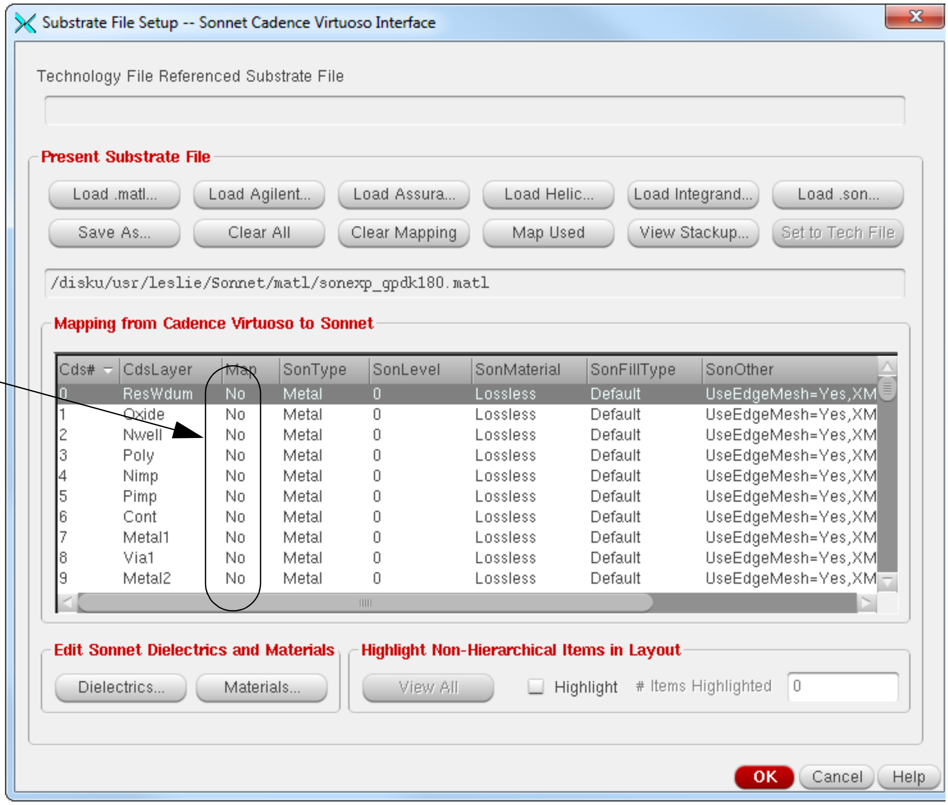
<Home Directory>/Sonnet/matl

where <Home Directory> is your home directory. For example, if your home directory is /diska/usr/johndoe then you should copy the file to /diska/usr/johndoe/Sonnet/matl/sonexp_gpdk180.matl. Note that you will overwrite the default substrate file that was created based on the songpdk180 library when you opened your SonnetEM view. This file was not copied with the rest of the example files at the beginning of the tutorial to prevent it from being overwritten.

25 **Click on the “Edit Substrate File” button in the Sonnet Cadence Virtuoso Interface Window.**

The Substrate File Setup dialog box appears on your display. Note that since you have not yet imported a substrate file that there is no mapping set in the window.

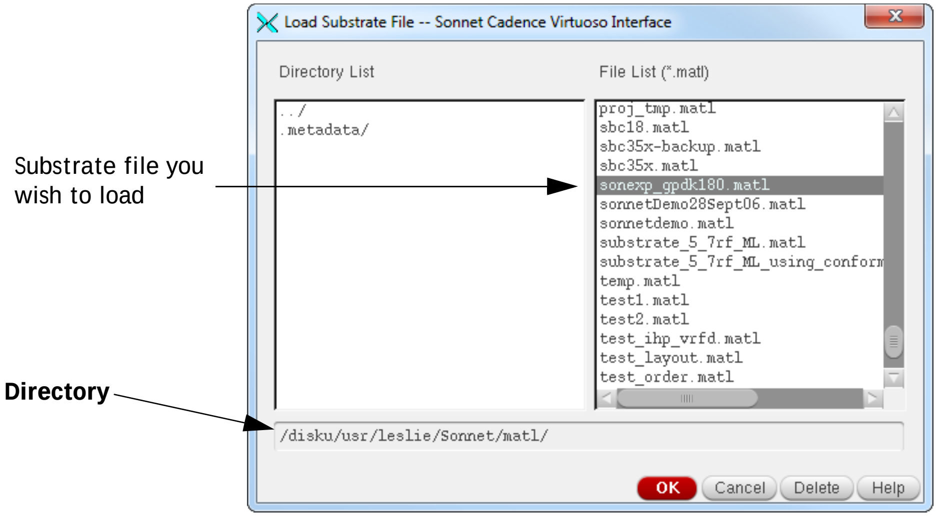
No mapping set



26 **Click on the “Load .matl” button in the Substrate File Setup dialog box.**

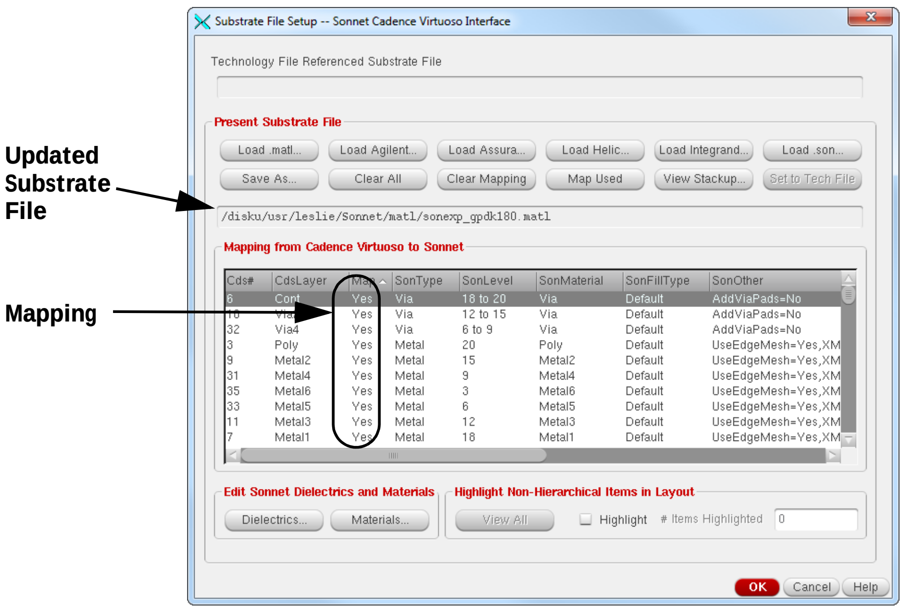
The Load Substrate dialog box appears on your display. The directory should default to your Sonnet matl directory, so there is no need to change directories.

27 Click on “sonexp_gpdk180.matl in the File List to select it.



28 Click on the OK button to load the selected .matl file.

The Load Substrate File dialog box is closed and the Substrate File Setup dialog box is updated with the new .matl file. Note that this file is now displayed as the present substrate file and the mapping output window has also been updated.



29 **Click on the OK button to close the Substrate File Setup dialog box.**

This completes setting up the substrate. Next, we will set up the ports.

Ports

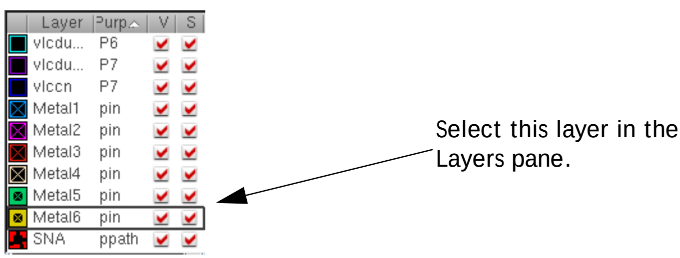
You need to specify the parameters for the ports to be used in Sonnet. You define the Sonnet ports by adding Virtuoso pins to your SonnetEM view which are converted to Sonnet ports. All pin types, with the exception of symbolic, may be used in Virtuoso. Pins can be converted to standard ports, co-calibrated ports, autogrounded ports, via ports or internal ports when using the Accurate solver. The Fast solver supports only standard ports.

For this tutorial, you your pins will be converted to standard ports. For a discussion of the other port types, please see "Setup ports.," page 12 in the **Blink User's Guide**.

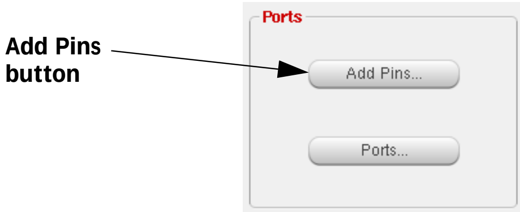
You may wish to zoom in on the feed lines of the inductor before adding the pins. To define the ports for your design, do the following:

30 **In the Layers pane on the left side of the layout window, select the "Metal6" layer whose purpose is set to "pin."**

You want to draw the pin on the same drawing layer where you wish to attach the Sonnet port to the metal trace. The metal traces for the inductor are drawn on "Metal6" so we also want to place the pins on Metal6.



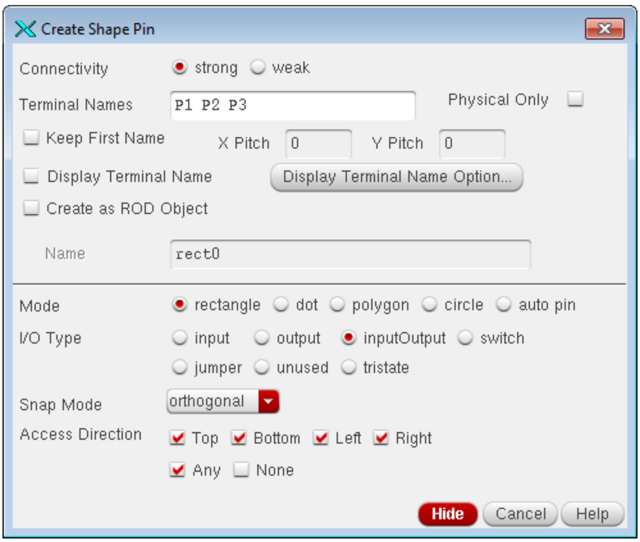
31 Click on the Add Pins button in the Ports section of the Cadence Virtuoso Interface.



The Create Shape Pin dialog box appears on your display.

32 Enter “P1 P2 P3” in the Terminal Names text entry box.

These names identify the pins. You may enter any character string for the terminal names. We are using P1, P2 and P3 since these pins are going to be converted to Port 1, Port 2, and Port 3 in the translated Sonnet project.



33 If it is not already selected, click on the “rectangle” radio button for Mode.

For the best translation of pins into the Sonnet environment, we recommend using a shape pin of the type “rectangle.”

34 Select the “Display Terminal Name” checkbox.

Displaying the terminal name in the layout window is optional. For the tutorial, we will display the terminal name. Selecting this checkbox allows you to also place a label on the pin when you are adding it in the layout window.

35 **Click on the "Display Terminal Name Option" button in the Create Shape Pin dialog box.**

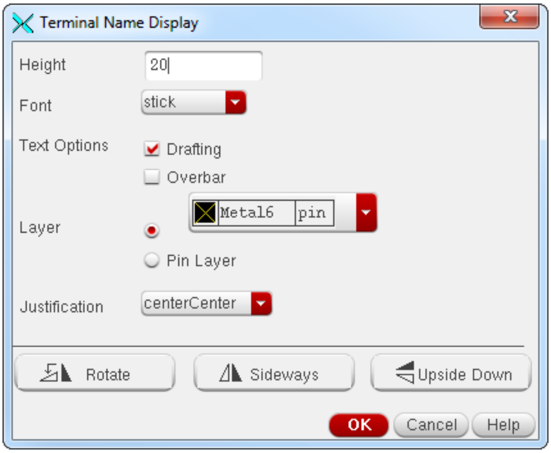
This opens the Terminal Name Display dialog box which allows you to control the appearance of the terminal name in the layout window.

36 **Click on the Layer drop list in the Terminal Name Display dialog box and select the "Metal6 pin."**

This ensures that label appears on the same drawing layer as the pin.

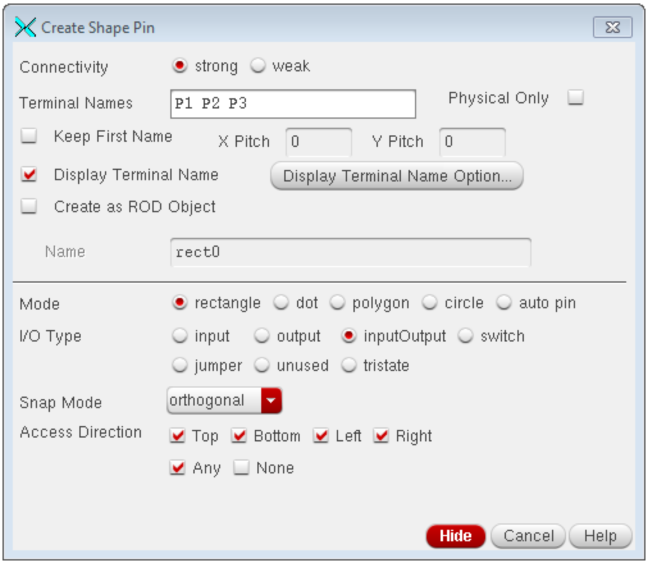
37 **Enter "20" in the Height text entry box.**

This ensures that the label is large enough to see in the layout window.



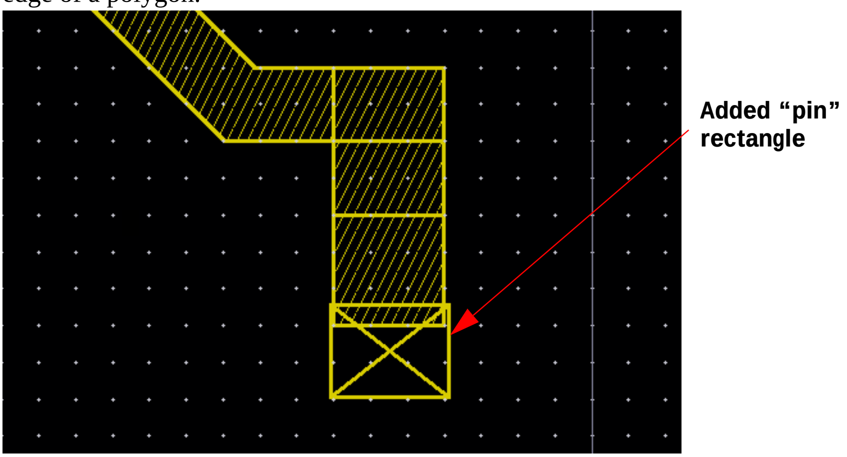
38 Click on the OK button to close the dialog box and apply the changes.

The Create Shape Pin dialog box should appear similar to the one shown below.



39 In the layout window, click on the lower left hand corner of the trace on the left and drag to form a rectangle that overlaps the bottom edge of the trace.

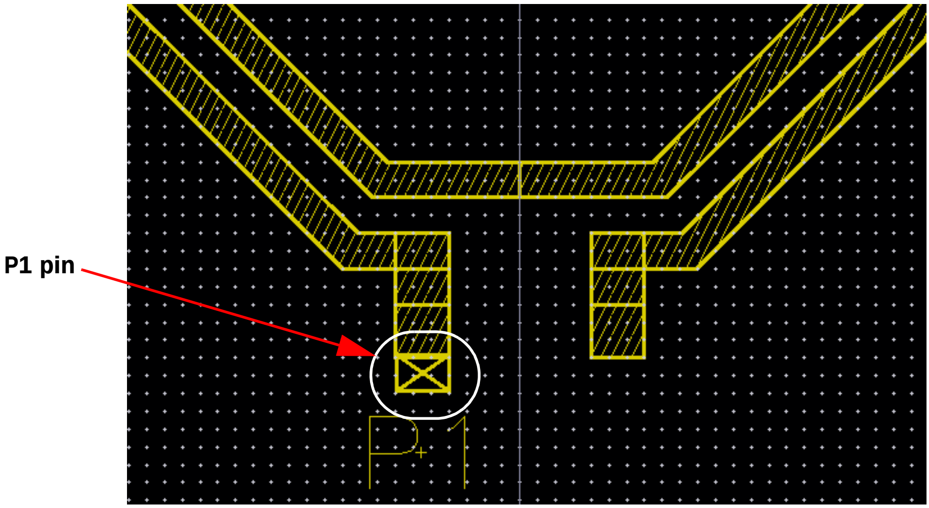
The pin should be placed across the edge of the metal trace to which you wish the Sonnet port attached. It is not necessary to be overly precise in placing your pin; during the translation, the center point of the pin is snapped to the closest available edge of a polygon.



When you complete adding the pin, the label appears.

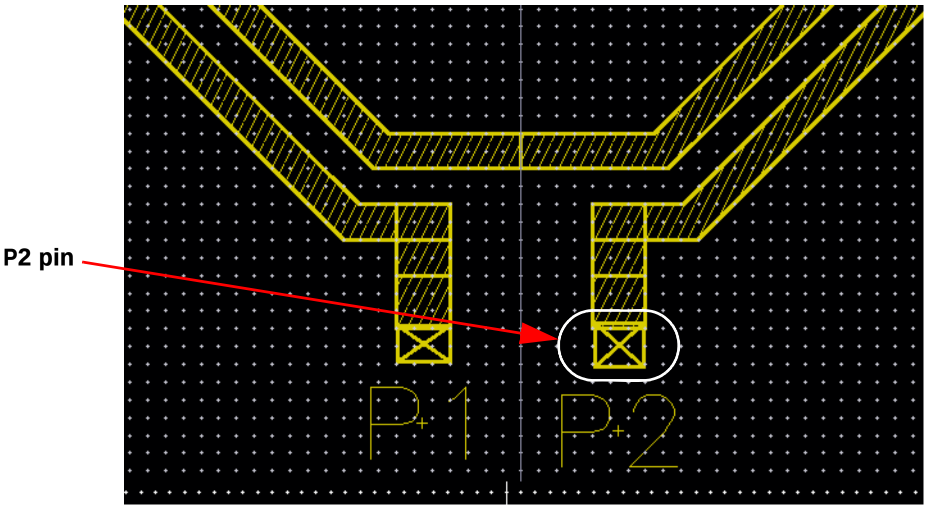
40 Click in the layout window where you wish to place the label.

Your circuit should appear similar to the picture below.



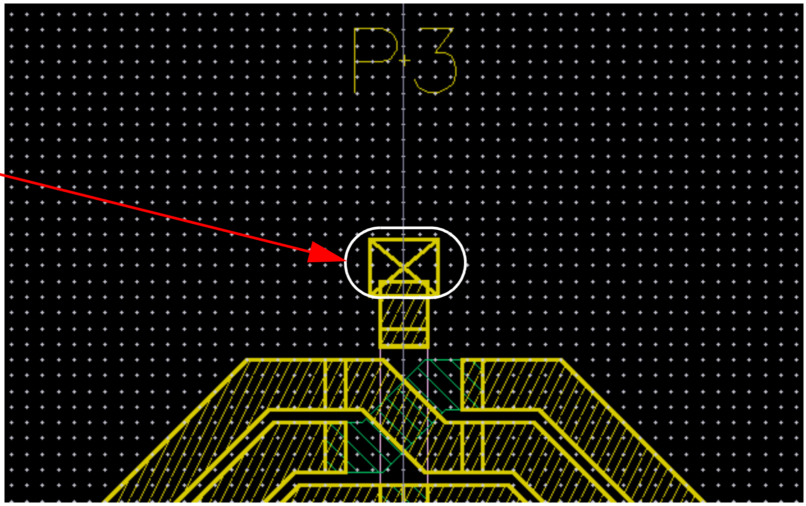
41 In similar fashion, add another pin to the trace on the right.

Your circuit should now appear similar to this:



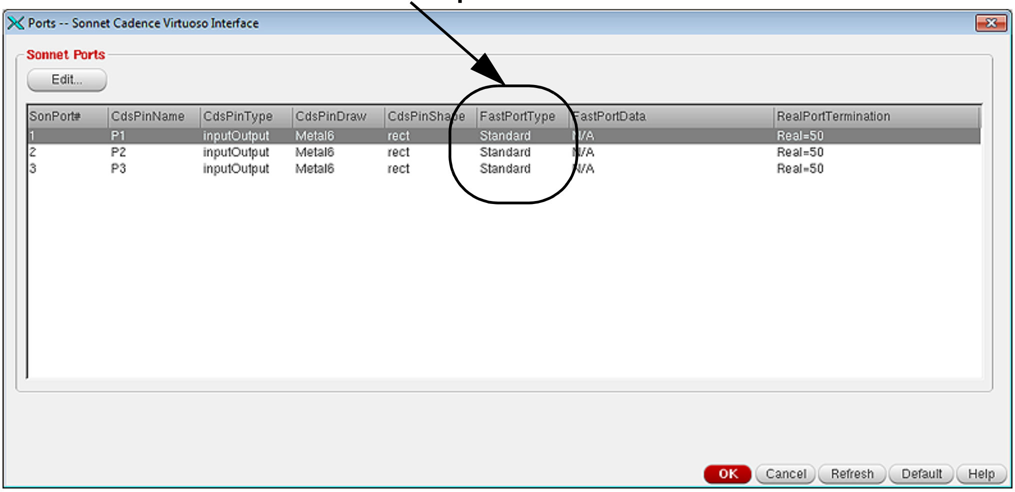
42 In similar fashion, add another pin to the trace for the centertap at the top of the inductor.

Your circuit should now appear similar to this:



43 In the Sonnet Cadence Virtuoso Interface window, click on the “Ports” button.

The Ports dialog box appears on your display.



As you can see, the Cadence pins were automatically mapped to Sonnet standard ports since the Fast solver is selected. When you switch to the Accurate solver, later in the tutorial, these ports are converted to standard box-wall ports since they were placed on the bounding box of the circuit.

44 Click on the Cancel button to close the Ports dialog box.

Since no changes were made in this dialog box, you may use the Cancel button to close it, ensuring that nothing was changed.

45 Click on the SonnetEM layout window to select it, then press the ESC key to exit "Create Pin" mode.

The appearance of the cursor changes.

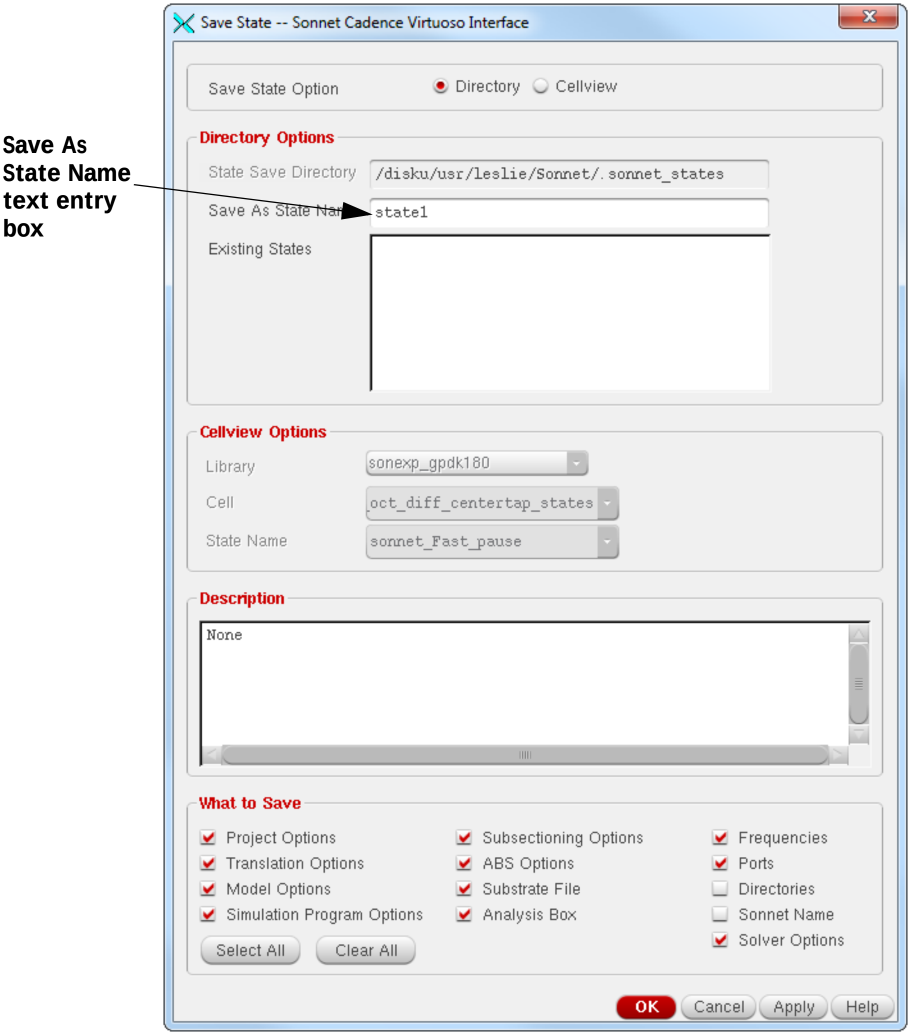
This completes setting up the ports. Next, we will save the changes we have made thus far in the tutorial using the Save State command.

Save State

Once you have completed the setup of your Sonnet simulation environment, you may wish to save all these settings for use with this or other SonnetEM views. You may use the Save State command to save all of the settings in the Sonnet Cadence Virtuoso Interface so that you may load them later.

46 Select *Session* ⇒ *Save State* from the main menu of the Sonnet Cadence Virtuoso Interface.

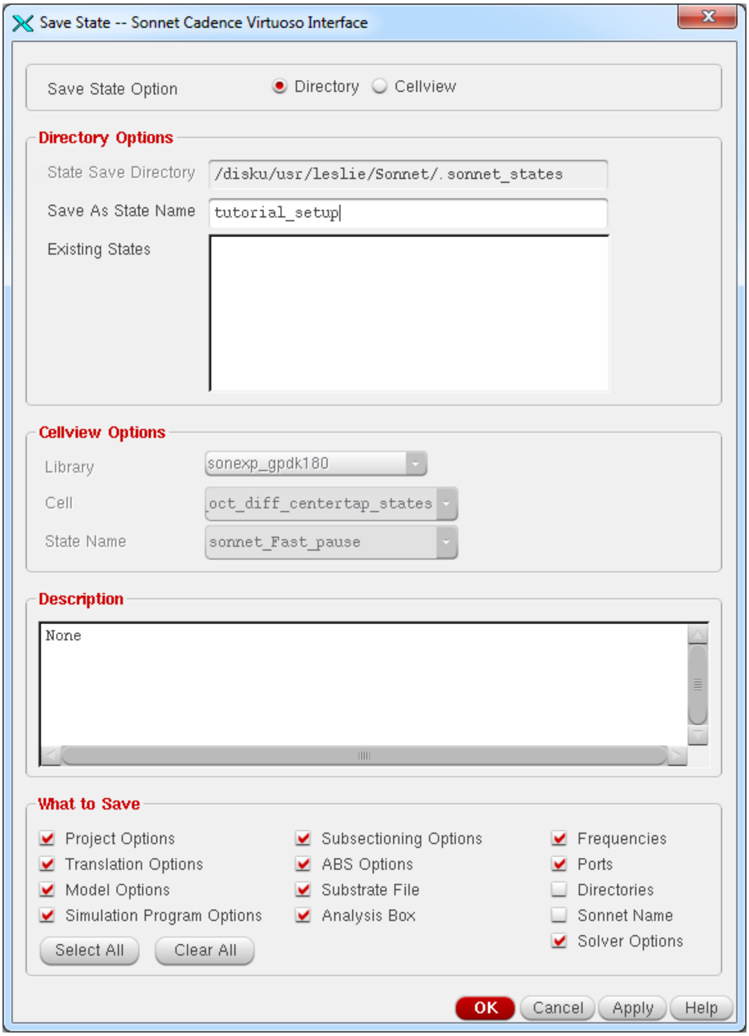
The Save State dialog appears on your display.



47 Enter “tutorial_setup” in the Save As State Name text entry box.

This identifies this save state as containing the setup you have accomplished so far in the tutorial.

The checkboxes in the What to Save section of the dialog box allow you to control what properties you wish to include in the save state. We will save all of the properties to ensure that we save both changes made so far in the tutorial and settings applied from the preset. The dialog box should appear similar to that shown below:



48 Click on the OK button to close the dialog box and save the state.

The message “State ‘tutorial_setup’ saved successfully” appears at the bottom of the Interface window. If you later wish to load these setting, you would use the *State* ⇒ *Load State* command.

This completes the setup items that are common to both the Fast and Accurate solvers. If you wish to perform the tutorial for only the Accurate solver, please continue at "Analysis Box Setup for Accurate Solver" on page 64. If you wish to perform the analysis using both solvers, or only the Fast solver, continue below.

Fast Solver Analysis Options

Analysis options allow you to specify run options for the Fast solver. You may also change the Sonnet project name if you wish. The present settings for the analysis options are correct for the tutorial, so no action needs to be taken. For more information about Fast solver analysis options, please click on the Help button in the Fast Solver Options dialog box.

Please note that current density data is only calculated for the discrete data points at which a full analysis is performed, not for data output from the calculated polynomial curve fit.

Fast Solver Simulation

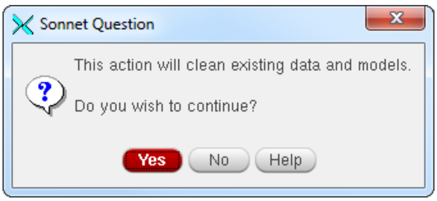
When you run a simulation, your circuit is translated into a Sonnet project and the Sonnet solver you have chosen is evoked.

To analyze your circuit, do the following:

49 Click on the Simulate button in the Sonnet Cadence Virtuoso Interface.

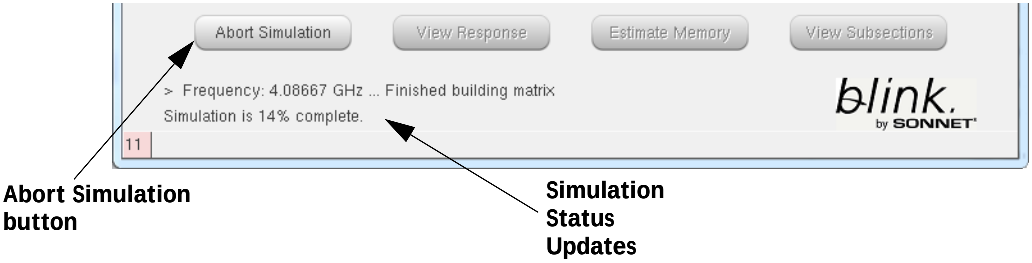
If your circuit needs to be translated before analysis, a query window appears on your display. If this happens, click on the Yes button to continue.

Another query box may appear, warning you that all existing data and models will be deleted if you continue. Your cell view is always translated before performing a simulation using the Fast solver, so the data and models are deleted.



50 Click on the "Yes" button in the query window to translate your circuit and begin the simulation.

The simulation is launched. Note that the "Simulate" button has been changed to "Abort Simulation." If you wish to stop the simulation, you would click on this button. Progress updates are displayed at the bottom of the Cadence Virtuoso Interface main window and in the CIW window.



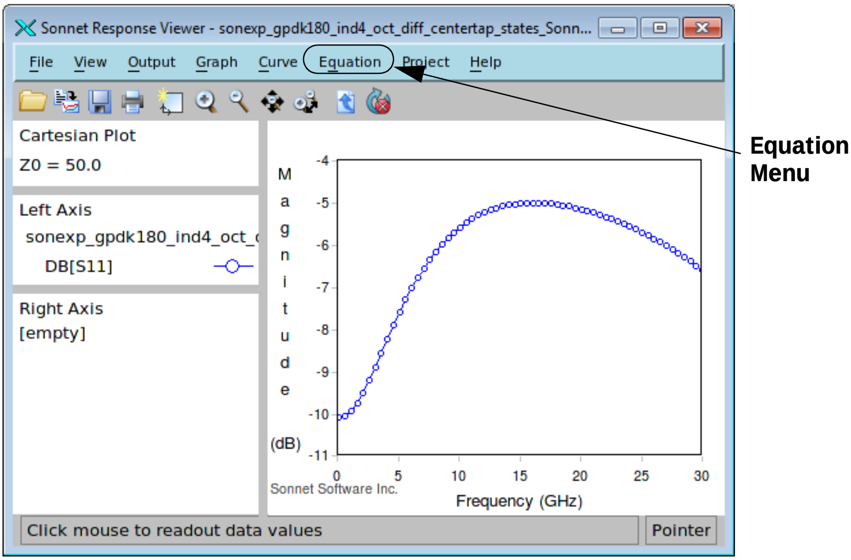
The simulation may take several minutes, depending on the speed of your computer. Once the simulation is complete, the Cadence Virtuoso Interface displays messages which indicate the progress of model creation. When that is done the message "The Inductor Model was created" is displayed.

View Results in Sonnet

You may examine the results of your simulation in Sonnet's response viewer. In this section of the tutorial, we will plot the differential inductance and differential Q by using equations. You will also learn how to add a marker to your plot. The equations we will use allow you to look at the results differentially.

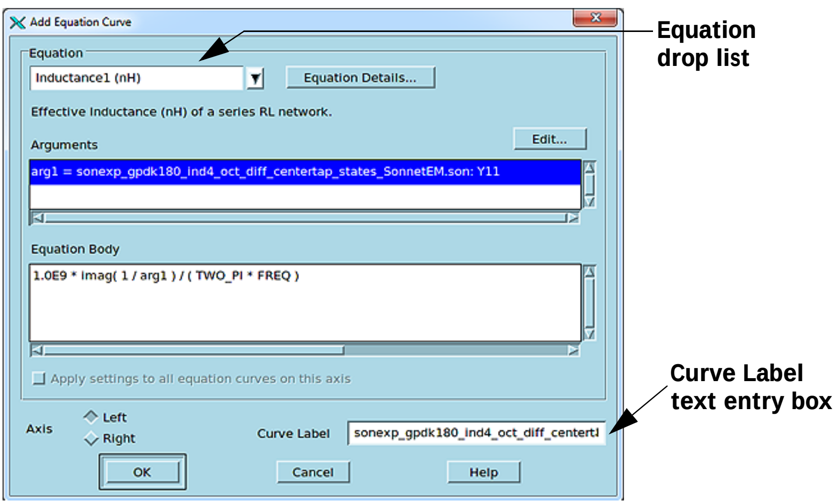
51 Click on the View Response button at the bottom of the Sonnet Cadence Virtuoso main window.

Sonnet’s response viewer appears on your display showing DB[S11]. This is the factory default curve group displayed the first time the response viewer is opened for a project. You may also select *Results* ⇒ *View Response* from the main menu of the Sonnet Cadence Virtuoso Interface window to open the response viewer.



52 Select *Equation* ⇒ *Add Equation Curve* from the response viewer main menu.

The Add Equation Curve dialog box appears on your display.

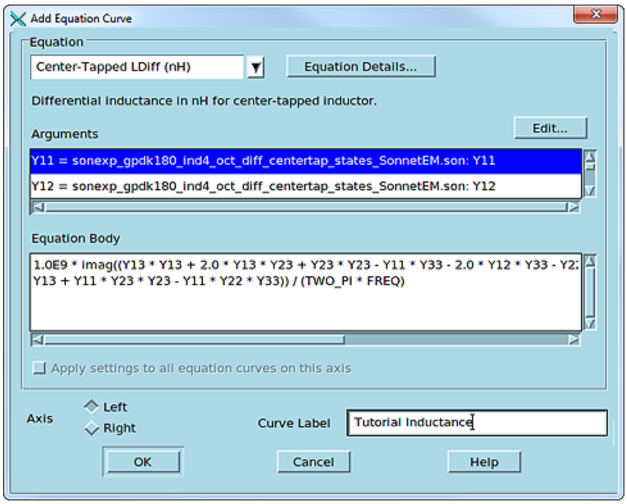


53 Select “Center-Tapped LDiff (nH)” from the equation drop list.

The dialog box is updated and now displays the equation for the differential inductance.

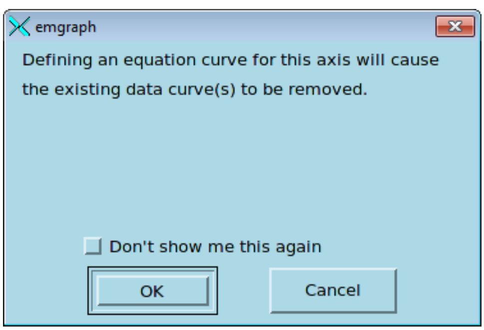
54 Enter “Tutorial Inductance” in the Curve Label text entry box.

The automatically generated label in this case is quite long, so using the shorter label makes the plot easier to understand. The dialog box should now appear similar to that shown below.



55 Click on the OK button to close the dialog box and add the curve to your plot.

Because a curve of DB[S11] already appears on the left axis of the plot, a query box warning you that the DB[S11] curve will be deleted may appear on your display.



56 Click OK in the query box.

The curve is displayed on the left axis of your plot. Now we will add the differential Q to the right axis of the plot.

57 Select *Equation* ⇒ *Add Equation Curve* from the response viewer main menu.

The Add Equation Curve dialog box appears again on your display.

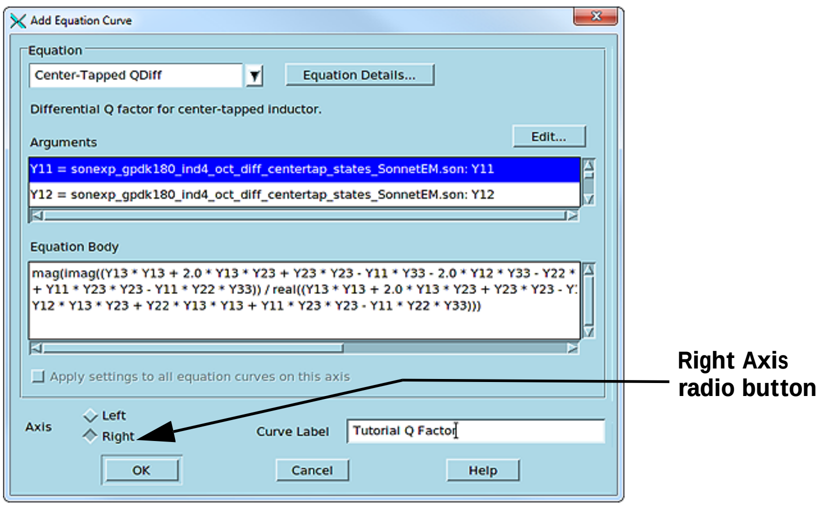
58 Select “Center-Tapped Q Diff” from the equation drop list.

The dialog box is updated and now displays the equation for the differential inductance.

59 Enter “Tutorial Q Factor” in the Curve Level text entry box.

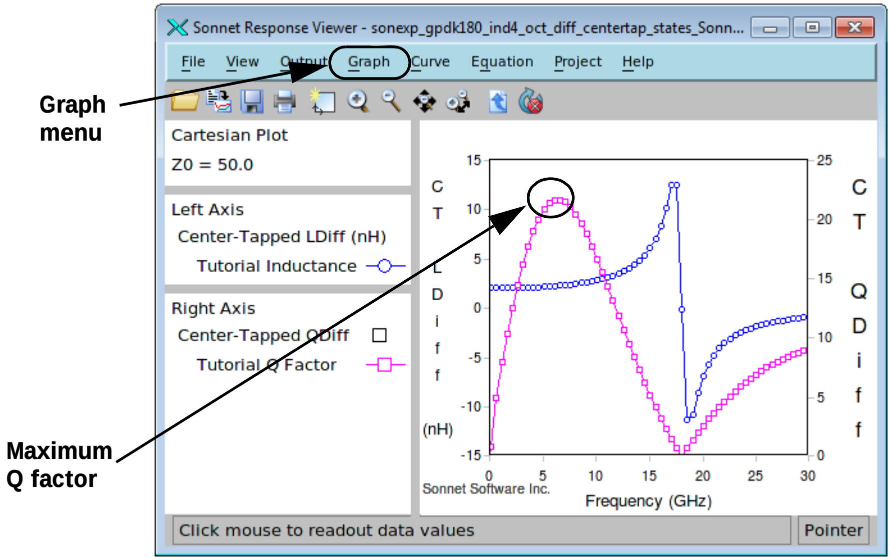
60 Click on the “Right” axis radio button.

This will add the equation curve to the right axis of your plot. The dialog box should now appear similar to that shown below.

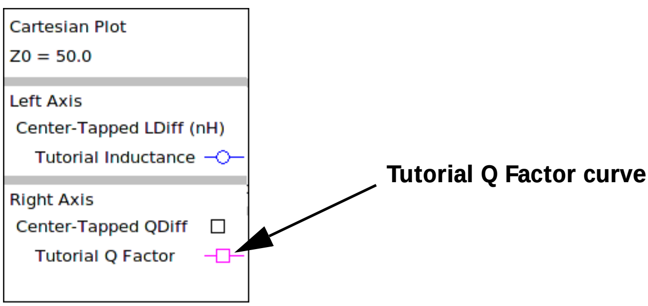


61 **Click on the OK button to close the dialog box and add the equation curve to your plot.**

The plot is updated and now displays the Differential Q Factor on the right axis as shown below.



In the plot legend shown below, you can see that the Tutorial Q factor curve is displayed in your plot using a magenta line and square data point symbol,



Adding a Marker

Sonnet's response viewer allows you to place a marker to label important data points in your plot. In this section, you will add a marker to identify the maximum Q factor and the frequency at which it occurs.

62 **Select *Graph* ⇒ *Marker* ⇒ *Add* ⇒ *Data Marker* from the main menu of the response viewer.**

The appearance of your cursor changes:

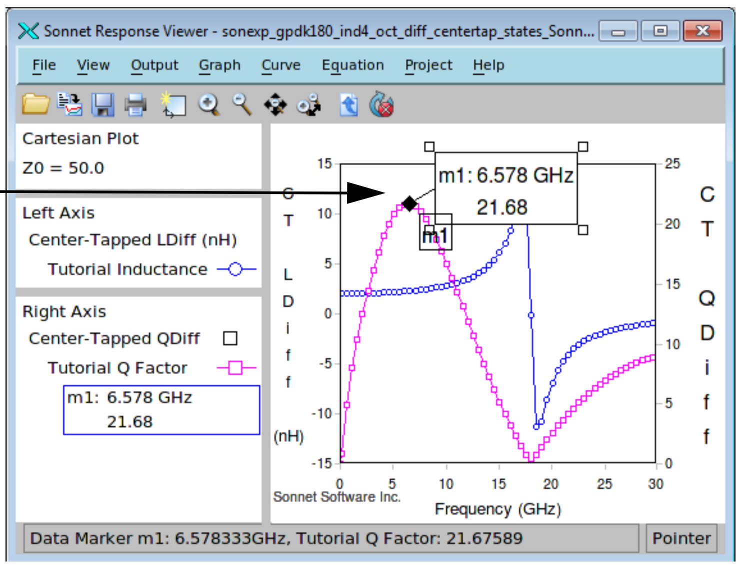
TIP

Note that when you are over a data point in your plot, the box that is part of the cursor appears in reverse video (black with a white "A.")

63 **Click on the maximum value of the Q factor in your plot.**

When you select the data point, a label M1 appears as well as a another label which displays the value of the data point and the frequency at which it occurs. Note that once you add the marker, you may drag the data display box to any location on your plot. The maximum Q whose value is 21.68 occurs at 6.578 GHz.

Data Marker



NOTE: If you are not sure if you have selected the maximum value, you may right click on the data marker and select "Go to Max" from the pop-up menu which appears.

64 **Select *File* ⇒ *Exit* from the main menu of the response viewer.**

The response viewer window is closed.

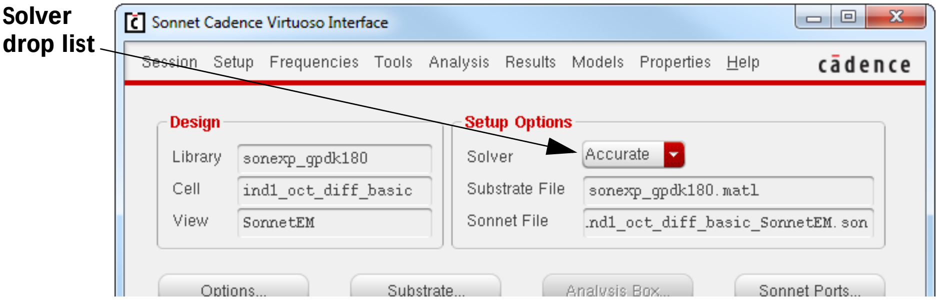
Once you have completed your analysis, if you wish to continue the tutorial by running the simulation using the Accurate solver, continue below. If you do not wish to run a simulation with the Accurate solver, then continue the tutorial at "View Current Density Data," page 78.

Selecting the Accurate Solver, *em*

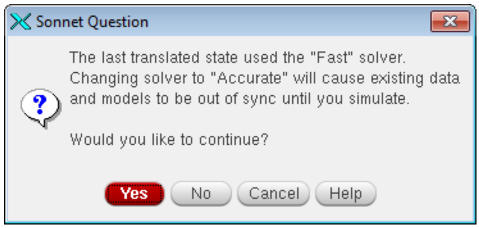
For the next section of the tutorial, you will perform an analysis using the Accurate solver, ***em***. ***Em*** performs electromagnetic analysis for arbitrary 3-D planar geometries, maintaining full accuracy at all frequencies. The Accurate solver is a "full-wave" solver which takes into account all possible coupling mechanisms.This is a time-tested engine whose results are highly accurate, although it will typically require more processing time and memory than the Fast solver. This solver should be used when accuracy in your results is critical, such as when you are finalizing a design.

The analysis is performed in a shielded enclosure, a six-sided metal box, and uses a fixed grid based on a cell size which forms the basis for the rectangular meshing. You will be changing the cell size used for this tutorial.

65 **Select "Accurate" from the Solver drop list in the Cadence Virtuoso Interface.**



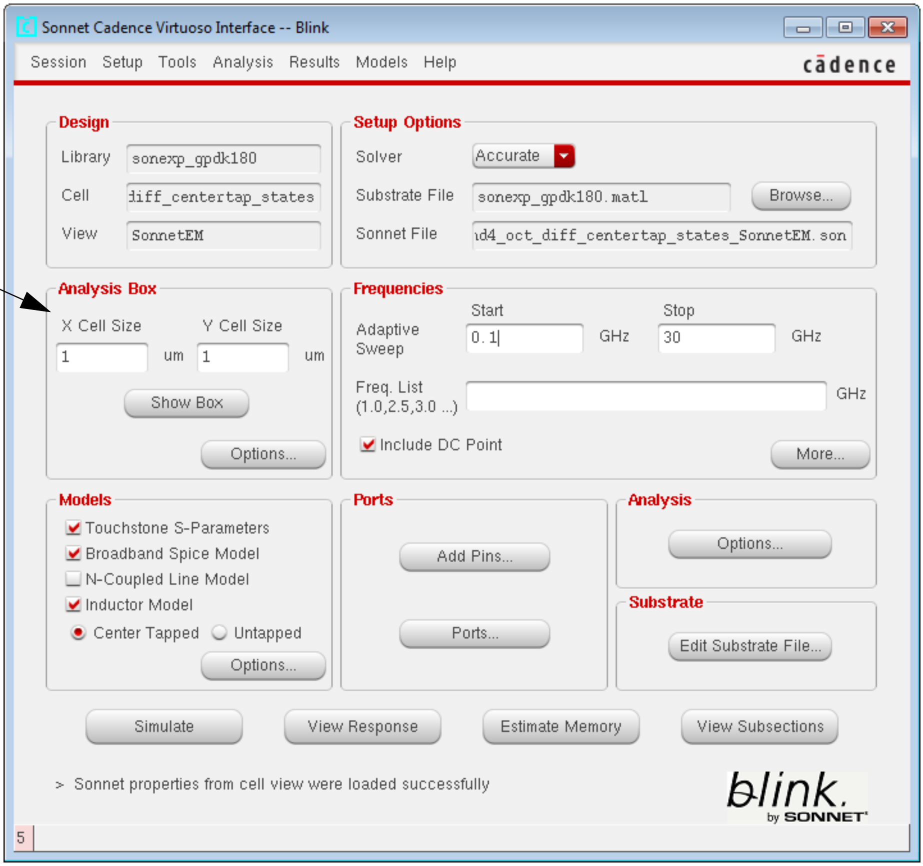
A query window appears on your display. Since you are switching solvers, the analysis data and models contained in your Sonnet project are out of sync, since the results were created using the Fast solver.



66 Click on the “Yes” button in the query box.

The appearance of the Cadence Virtuoso Interface main window is updated. The Solver drop list displays “Accurate” and the Maximum Mesh size section has been replaced with an Analysis Box section.

Analysis Box section



Analysis Box Setup for Accurate Solver

Blink’s Accurate solver, ***em***, performs the analysis in a six-sided metal box. The sides of the metal box are always lossless, but you may define the top and bottom covers as lossy metal. In order to prevent coupling with the box walls, we set a margin to increase the box size. The margin is also used to set the reference plane lengths for any ports. Blink’s Accurate solver has an automatic de-embedding capability. When invoked, the solver removes the port discontinuity and a desired length of transmission line. The reference planes instruct the Accurate solver as to the desired length of the transmission line to be removed. One reference plane length per box side may be specified but a single reference plane can be used for multiple parallel lines on the same box side.

For a discussion of the analysis box, please see "Accurate Solver" on page 20.

Cell Size

You may also control the resolution of the meshing used by the Accurate solver for your analysis by setting the cell size. The smaller the cell size in Sonnet, the finer the resolution and more accurate the results. However, there is a trade off in that smaller cell sizes increase the amount of memory and processing time for your analysis.

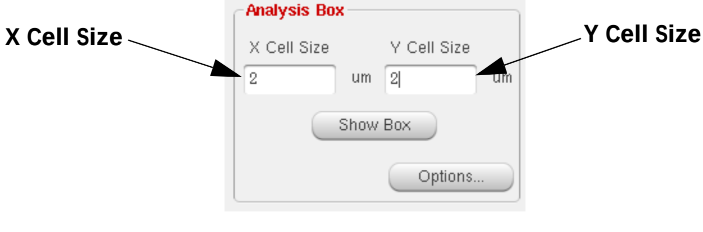
The Accurate solver subsectioning is based on a uniform grid made up of cells. One or more cells are automatically combined together to create subsections. Cells may be square or rectangular (any aspect ratio), but must be the same over your entire circuit. Your cell size should be set to the largest size that still provides a sufficiently accurate answer.

The conductor width of the inductor used in this tutorial is 10 microns and the spacing is 10 microns. Setting your cell size to 2 microns provides five cells across the inductor width which is reasonable for this circuit.

To set the cell size, do the following:

67 **Enter a value of "2" in the X Cell Size and Y Cell Size text entry boxes in the Analysis Box section of the Cadence Virtuoso Interface main window.**

This sets your cell size to 2 X 2 microns. In this case, we are using a square cell but note that a cell may also be a rectangle; the values do not need to be the same.



Setting the Box Size using Margins

Adding a margin moves the box wall out that distance from the bounding box of the geometry being translated. A feedline from the Virtuoso pin (Sonnet port) is added to the circuit and a reference plane is set for the length of the feedline.

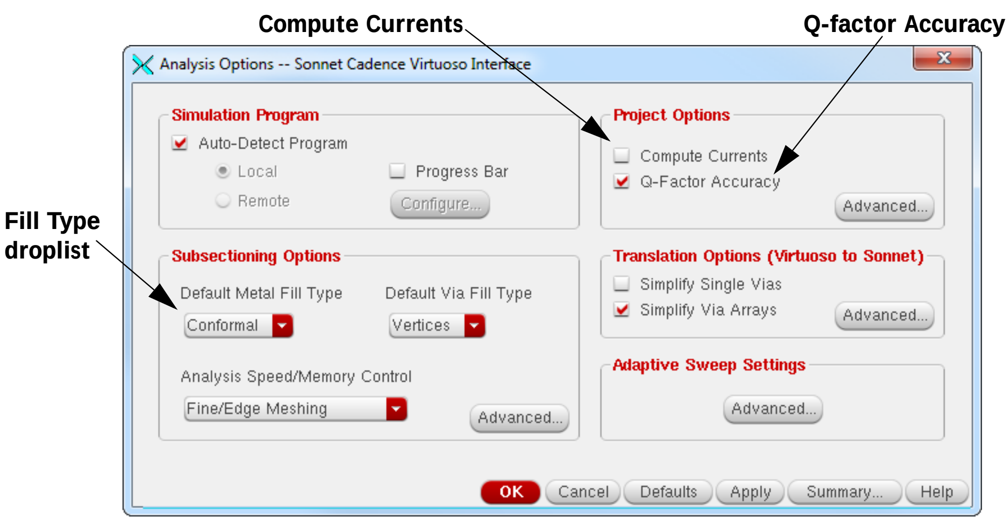
For most devices, a good rule of thumb is to set the box size to three times the device circuit size. Since the inductor is in the center with a margin on either side, setting the margin to the width of the inductor provides the proper box size. Selecting the preset for an inductor set all of your margins to 100% of the circuit size, so the margin settings do not need to be changed. If you had needed to change them, you would have clicked on the Options button in the Analysis Box section to open the Analysis Box Options dialog box. For more information, click on the Help button in the Analysis Box Options dialog box.

Accurate Solver Analysis Options

Analysis options allow you to specify various advanced run options. To set the run options, do the following:

68 **Click on the Options button in the Analysis section of the Sonnet Cadence Virtuoso Interface window.**

The Analysis Options dialog appears on your display.



Conformal meshing is more efficient when analyzing curved or diagonal metal with the Accurate solver. Since this is an octagonal spiral, conformal mesh provides a faster, more accurate analysis. This option was already selected by the Inductor preset. For a detailed discussion of conformal meshing, please see Chapter 10, "Conformal Mesh" in the **Native Environment User's Guide**.

69 Click on the "Compute Currents" checkbox in the Options dialog box.

Current density data is calculated for your circuit when the Compute Currents option is enabled. Please note that for an ABS sweep, current density data is only calculated for the discrete data points, not for the adaptive data. Calculating the current density data does increase analysis time, so only select this option if you need to view the current density data.

The "Q-factor Accuracy" project option was also selected by the Inductor preset. Selecting this checkbox increases the accuracy of the Q-factor when ABS is used. Normally ABS uses S-parameters to determine convergence. When this option is used, ABS uses both the S-parameters and Q-factor for convergence criteria.

70 Click on the OK button in the Analysis Options dialog box to close the dialog box and apply the changes.

This completes the set up of analysis options.

Pre-Analysis Checks

Once you have completed the setup of your Sonnet environment and specified your analysis frequencies, there are some easy sanity checks you can run to gain confidence that your circuit is correct and that the analysis will run in an acceptable amount of time.

Estimate Memory

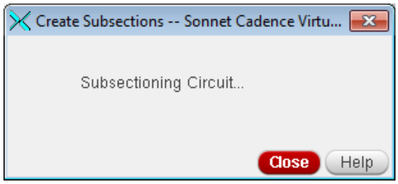
The Accurate solver, ***em***, allows you to subsection your circuit without running an analysis to get an estimate of memory use.

71 To estimate memory, select *Analysis ⇒ Estimate Memory* from the main menu in the Cadence Virtuoso Interface window.

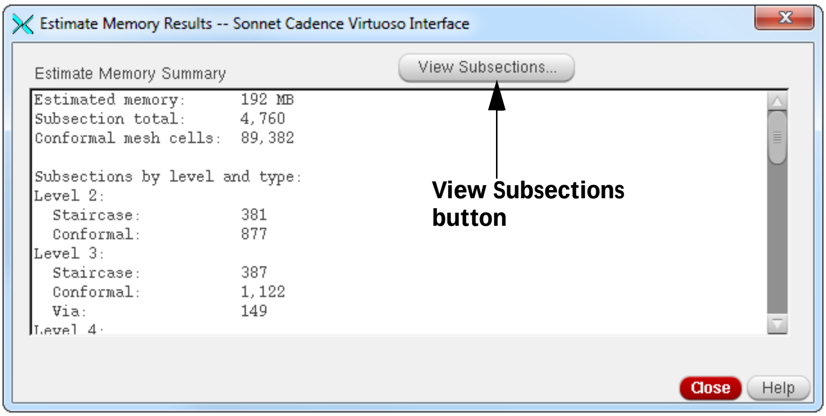
You may also estimate memory by clicking on the "Estimate Memory" button in the Cadence Virtuoso Interface main window.

72 If a query window appears asking if you wish to translate the circuit, click on yes.

A simple status popup appears to indicate that the circuit is being subsectioned.

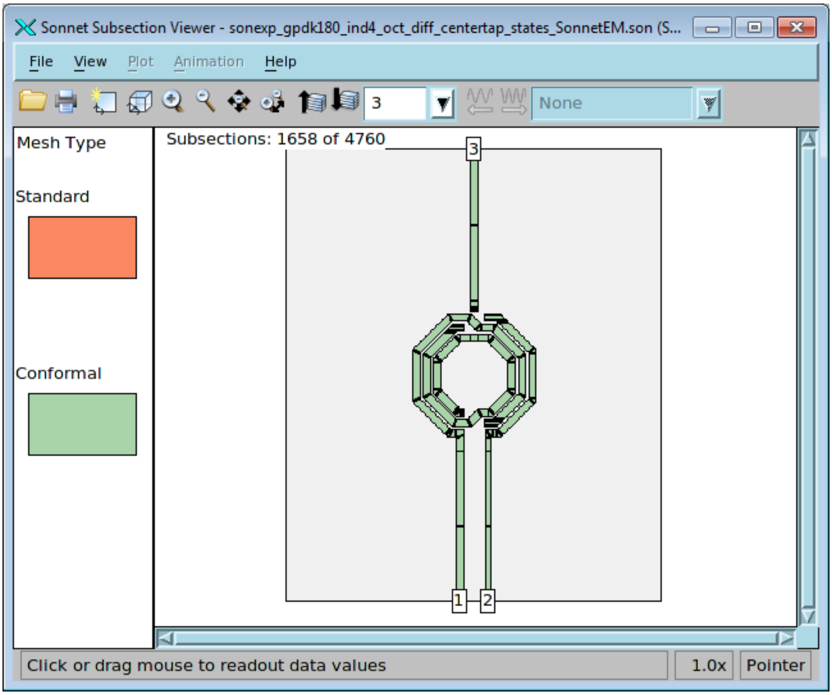


When the subsectioning is complete, the memory use estimate and the subsectioning information, listed by level and total amounts, is displayed.



73 Click on the View Subsections button in the Estimate Memory Results window.

Sonnet’s subsection viewer appears on your display. Note that since the default metal fill type was conformal, the majority of the subsectioning is green indicating Conformal Mesh.



74 Select *File* ⇒ *Exit* from the main menu of the Sonnet Subsection Viewer to close the window.

The Subsection Viewer is closed.

75 Click on the Close button in the Estimate Memory Results window.

The Estimate Memory Results window is closed.

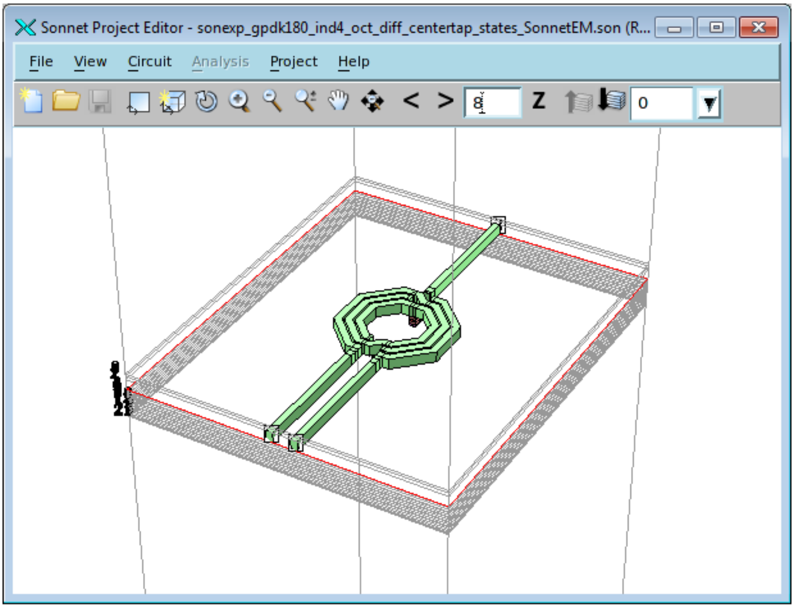
3D View

You may open your circuit to a 3D view in the native environment project editor, which allows you to visually inspect if your drawing layers and vias are mapped correctly.

To see the 3D view of your circuit, do the following:

76 **Select *Tools* ⇒ *Project Viewer/Editor* ⇒ *View 3D* from the main menu of the Sonnet Cadence Virtuoso Interface window.**

If a query window appears asking if you wish to translate the cell view, click on yes. The Sonnet project viewer is opened on your display with the 3D view open, as shown below. If any problems were noticed, you would need to go back and edit the mapping in the Substrate File Setup dialog box.



The 2D view of your circuit is also displayed.

77 **Select *File* ⇒ *Exit* from the main menu of the Sonnet project viewer.**

The project viewer is closed.

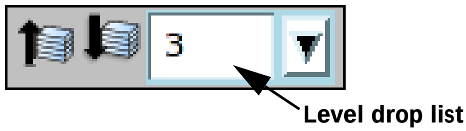
Connectivity Checker

Blink has a tool, the Connectivity Checker, available in the project viewer which allows you to visually check for opens or shorts in your circuit. You may invoke the tool directly from the Sonnet Cadence Virtuoso Interface window. This tool is especially useful after translating to make sure that the metal placement has not created any opens or shorts.

78 Select *Tools* ⇒ *Check Connectivity* from the main menu of the Sonnet Cadence Virtuoso Interface window.

Sonnet's project viewer appears on your display with the Connectivity Checker on.

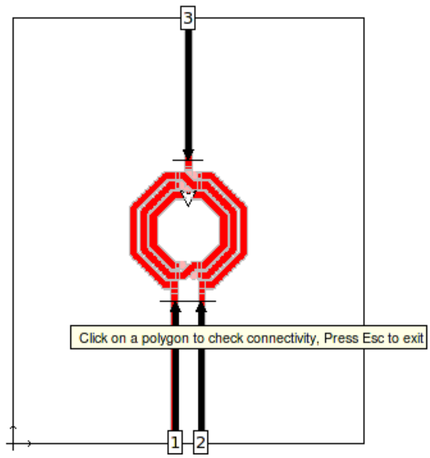
79 If Level 3 is not already displayed, select "3" from the Level drop list in the Sonnet tool bar.



Since most of the circuit is on Sonnet metal level 3, you need to view that level.

80 Click on any metal in the circuit.

The polygon you select is highlighted in red, as well as any metal it is electrically connected to. In our case, all of the metal in the circuit should be highlighted. If some part of the circuit was not highlighted this would indicate there was an open.



81 Press the Escape key to exit the Connectivity Checker.

The view is updated to show the 2D view.

82 Select *File* ⇒ *Exit* from the main menu of the Sonnet project viewer.

The project viewer is closed.

Now that setup is complete and you have checked your translated circuit, you are ready to run the simulation.

Accurate Solver Simulation

When you run a simulation, your circuit is translated into a Sonnet project and the Sonnet solver you have chosen is evoked.

For an ABS sweep, the Accurate solver analyzes the circuit at the beginning and end frequencies. Using an iterative process, the solver then analyzes at other discrete frequencies and determines a rational polynomial fit to the S-parameter data within the frequency band. Once a rational polynomial fit is achieved with an acceptable error, the frequency response across the specified bandwidth is calculated.

The output data consists of the discrete data points, frequencies at which the solver performs a full electromagnetic analysis, and the adaptive data, which is data calculated using the rational polynomial.

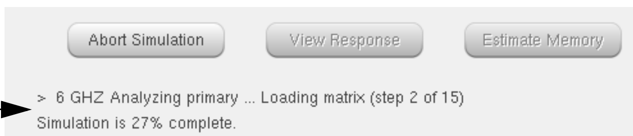
To analyze your circuit, do the following:

83 Click on the Simulate button in the main window of the Sonnet Cadence Virtuoso Interface.

If your circuit needs to be translated before analysis, a query window appears on your display. Click on the "Yes" button in the query window to translate your circuit before performing the simulation.

The simulation is launched. Updates of the simulation appear in the bottom of the Sonnet Cadence Virtuoso Interface window and the CIW, as shown below.

Simulation Status Updates



After the simulation is complete, continue the tutorial.

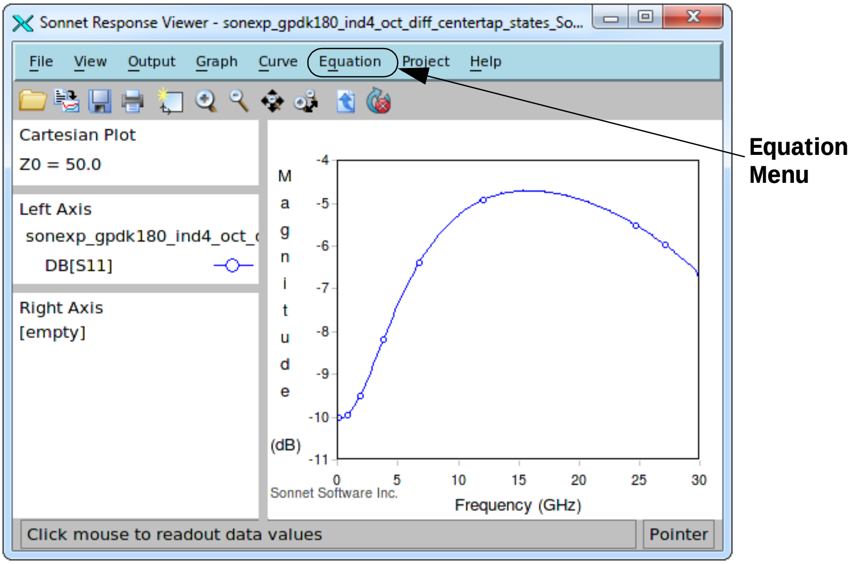
View Results in Sonnet

You may examine the results of your simulation in the native environment response viewer. In this section of the tutorial, we will plot the differential inductance and differential Q by using equations. You will also learn how to add a marker to your plot. Note that there are also equations available to plot the single-ended inductance and Q factor of your circuit if you wished to see those values. The equations we will use allow you to look at the results differentially.

If you have performed the Fast Solver part of the tutorial, this next section will be familiar. If you wish to skip this part of the tutorial, continue at "View Current Density Data," page 78.

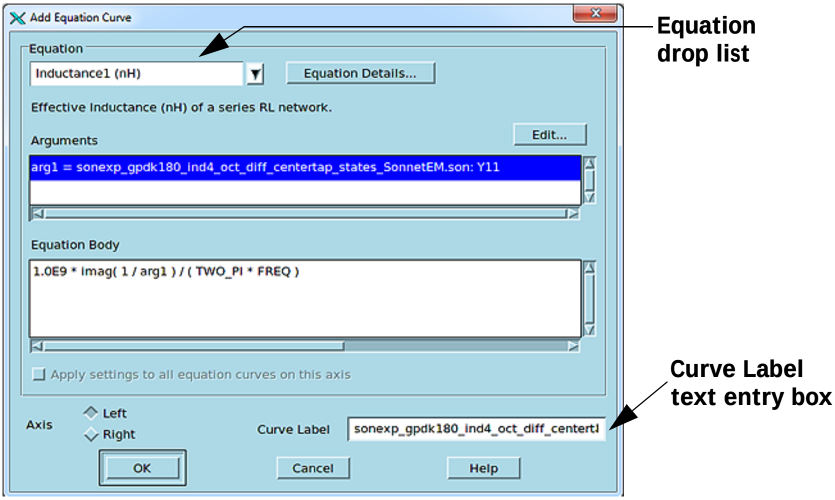
84 Click on the View Response button at the bottom of the Sonnet Cadence Virtuoso Interface main window.

Sonnet's response viewer appears on your display showing DB[S11]. This is the factory default curve group displayed the first time the response viewer is opened for a project. You may also select *Results* ⇒ *View Response* from the main menu of the Sonnet Cadence Virtuoso Interface window to invoke the response viewer.



85 Select *Equation* ⇒ *Add Equation Curve* from the response viewer main menu.

The Add Equation Curve dialog box appears on your display.

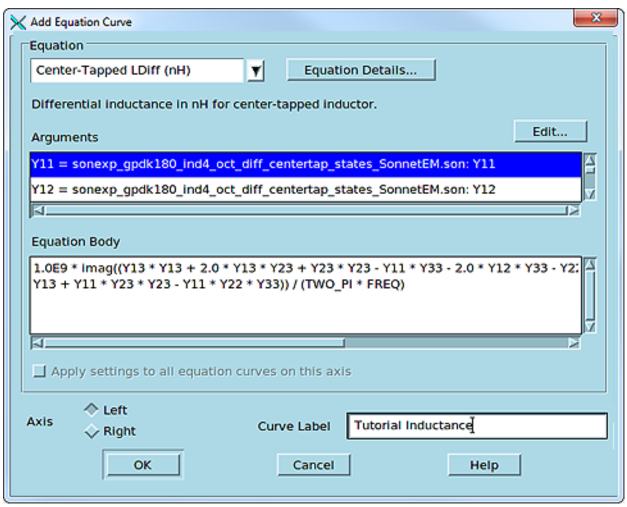


86 Select "Center-Tapped LDiff (nH)" from the equation drop list.

The dialog box is updated and now displays the equation for the differential inductance.

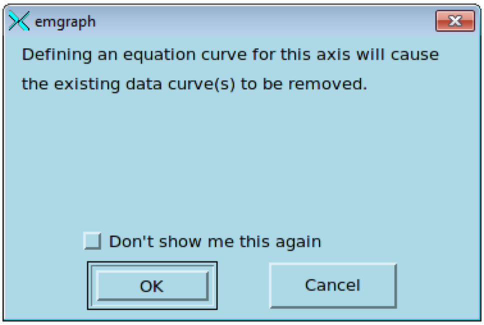
87 Enter "Tutorial Inductance" in the Curve Level text entry box.

The automatically generated label in this case is quite long, so using the shorter label makes the plot easier to understand. The dialog box should now appear similar to that shown below.



88 Click on the OK button to close the dialog box and the curve to your plot.

Because a curve of DB[S11} already appears on the left axis of the plot, a query box warning you that the DB[S11] curve will be deleted if you proceed may appear on your display.



89 Click OK in the query box.

The curve is displayed on the left axis of your plot.

90 Select *Equation* ⇒ *Add Equation Curve* from the response viewer main menu.

The Add Equation Curve dialog box appears again on your display.

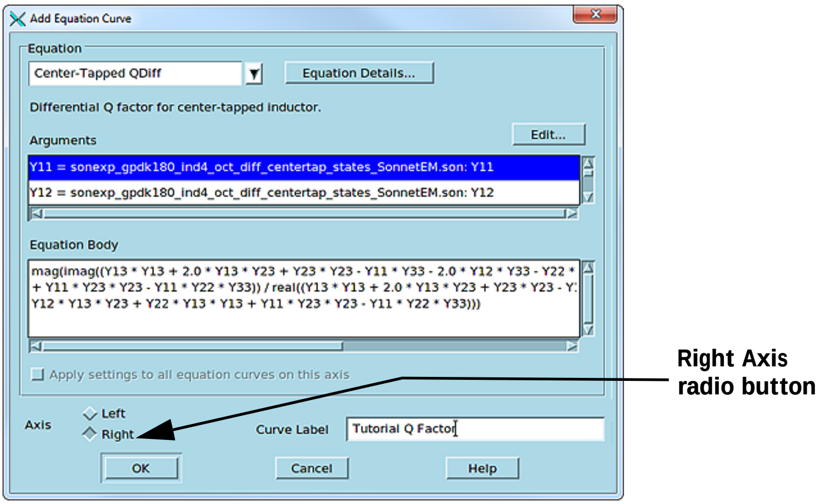
91 Select "Center-Tapped QDiff" from the equation drop list.

The dialog box is updated and now displays the equation for the differential inductance.

92 Enter “Tutorial Q Factor” in the Curve Level text entry box.

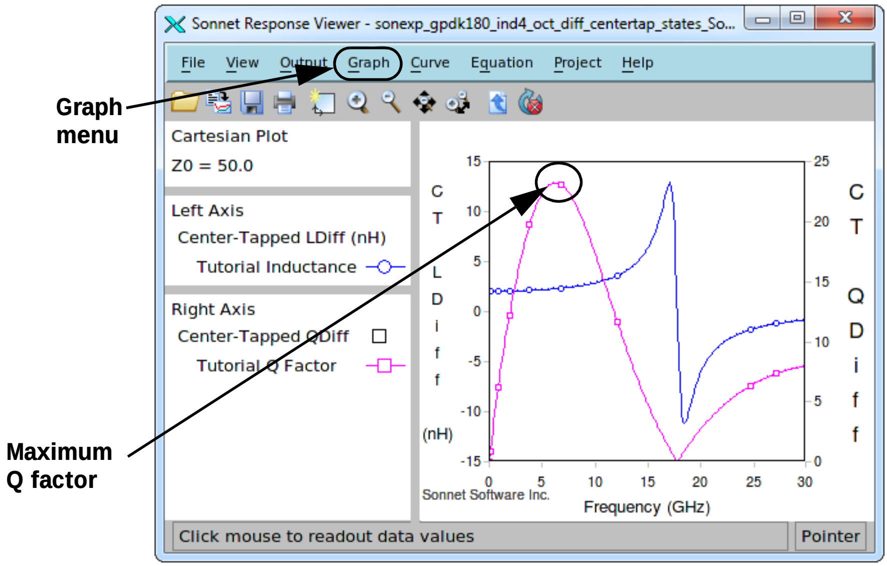
93 Click on the “Right” axis radio button.

This will add the equation curve to the right axis of your plot. The dialog box should now appear similar to that shown below.



94 Click on the OK button to close the dialog box and add the equation curve to your plot.

The plot is updated and now displays the Differential Q Factor on the right axis as shown below.



Adding a Marker

Sonnet’s response viewer allows you to place a marker to label important data points in your plot. In this section, you will add a marker to identify the maximum Q factor and the frequency at which it occurs.

95 Select *Graph* ⇒ *Marker* ⇒ *Add* ⇒ *Data Marker* from the main menu of the response viewer.

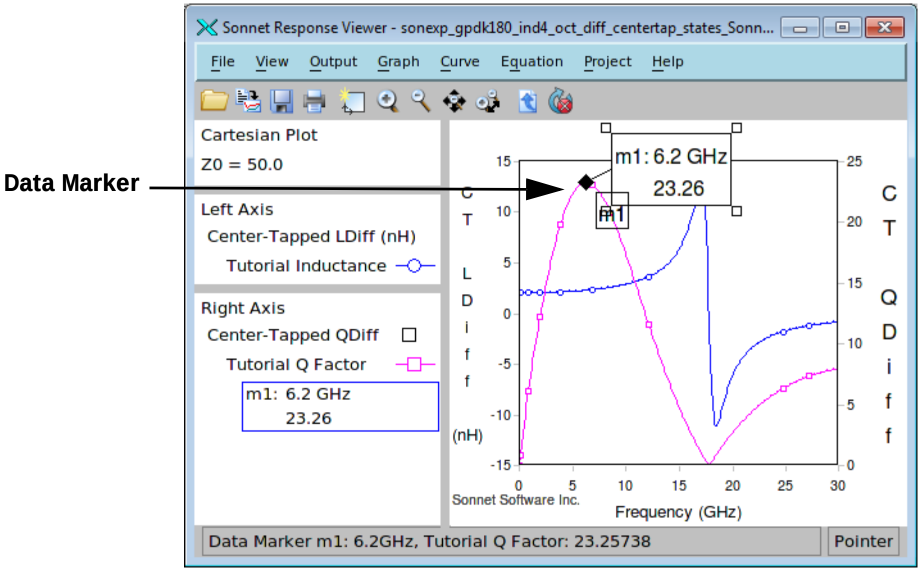
The appearance of your cursor changes:

TIP

Note that when you are over a data point in your plot, the box that is part of the cursor appears in reverse video (black with a white “A.”)

96 Click on the maximum value of the Q factor in your plot.

When you select the data point, a label M1 appears as well as a another label which displays the value of the data point and the frequency at which it occurs. Note that once you add the marker, you may drag the data display box to any location on your plot. The maximum Q whose value is 23.26 occurs at 6.2 GHz.



TIP

If you are not sure if you have selected the maximum value, you may right click on the data marker and select “Go to Max” from the pop-up menu which appears.

97 Select *File* ⇒ *Exit* from the main menu of the response viewer.

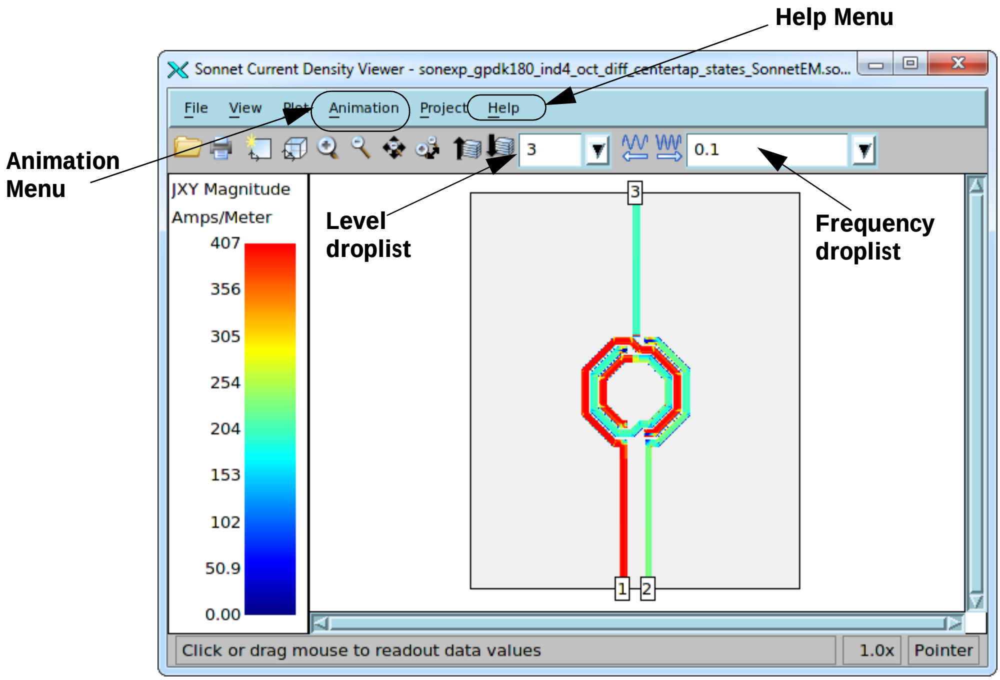
The response viewer is closed.

View Current Density Data

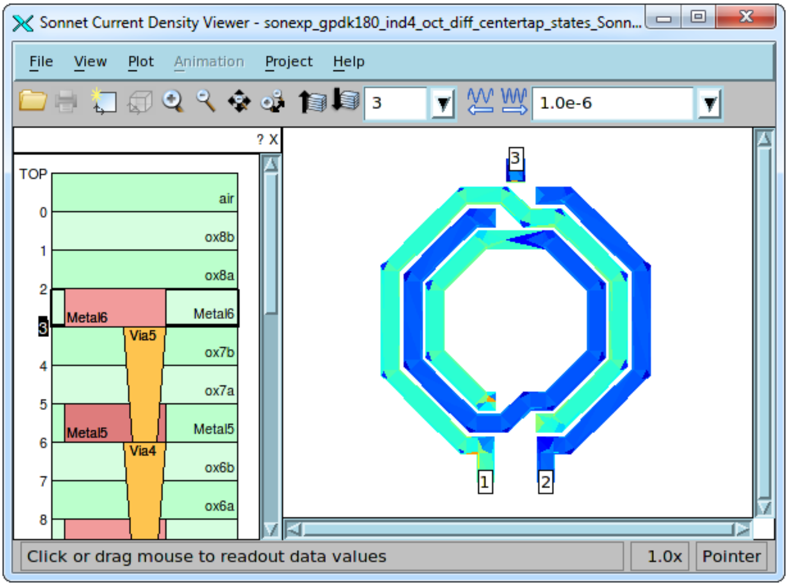
During the setup of the analysis for the Accurate solver, we specified that current density data should be calculated. Current density data is calculated by default for the Fast solver. The current density viewer can be used to view this data.

98 Select *Results* ⇒ *View Currents* from the main menu of the Sonnet Cadence Virtuoso Interface window.

Sonnet's current density viewer appears on your display showing metal level 3. Depending on your previous settings, the stackup viewer may be displayed, instead of the scale. If so, select *View* ⇒ *Scale* from the currently density viewer main menu to display the scale. Note that if you are viewing Fast Solver results, the *View* ⇒ *Scale* command is not enabled. The current density viewer allows you to view the current density data at any frequency that was used in calculating the solution by selecting the frequency from the Frequency drop list on the tool bar. You may also animate your results as a function of frequency or time. For more information on using the current density viewer, please select *Help* ⇒ *Sonnet Help* from the main menu of the current density viewer.



If you have come directly here after completing the Fast Solver analysis, then the current density viewer will appear similar to the picture below.



99 When you are done examining your current density data, select *File* ⇒ *Exit* from the main menu of the current density viewer.

The current density viewer is closed.

Using a Model

When you specify any model, a schematic view of the model and a symbol view for the model are created. For all models, the symbol view is automatically created along with the model. You use the symbol view created by the Cadence Virtuoso Interface in your schematic so that your models can be used in the Cadence simulation.

In order to use a model in the Analog Design Environment, it is recommended that you create a config view which references the Sonnet symbol. A config view has been provided as part of your example files. The next section of the tutorial shows you how to set up the environment to use the Broadband Spice model created by the simulation.

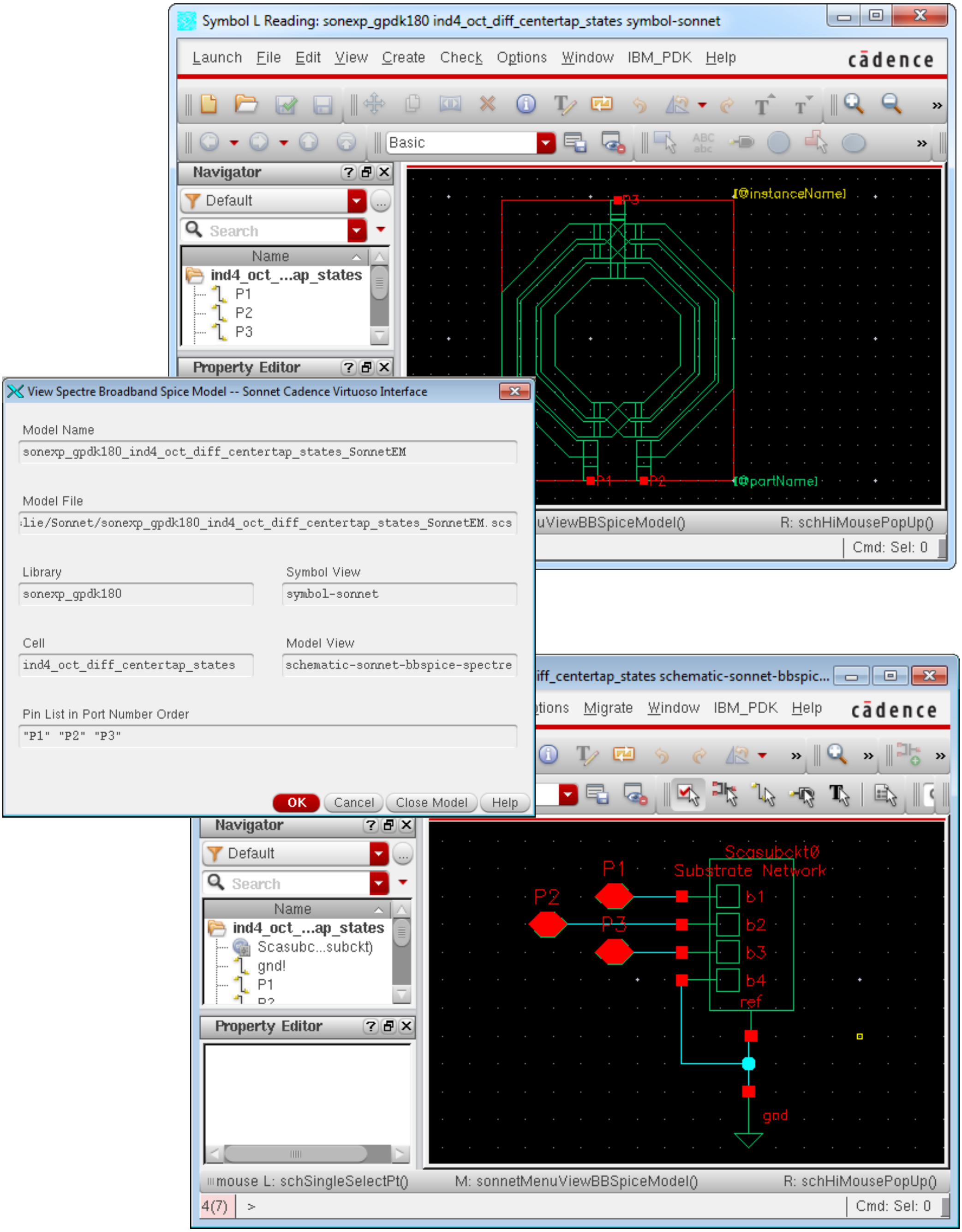
The Broadband Spice model is a lumped element model based on a rational polynomial fit. In order to create a Spice model which is valid across a broad band, the Sonnet broadband SPICE Extractor feature finds a rational polynomial which "fits" the S-Parameter data. This polynomial is used to generate the equivalent lumped element circuits in either PSpice or Spectre format. Since the S-Parameters are fitted over a wide frequency band, the generated models can be used in circuit simulators for AC sweeps and transient simulations.

For a detailed discussion of the Broadband Spice model, please see "Broadband SPICE Model," page 337 in the **Native Environment User's Guide.**

100 Select *Models* ⇒ *Broadband Spice* ⇒ *Show View* from the main menu of the Sonnet Cadence Virtuoso Interface window.

Three windows are opened on your display: the schematic view with the scasubckt component that references the inductor model raw data file, the Symbol view with the layout look-alike of the inductor layout, and the View Spectre Broadband

Spice Model dialog box. The symbol-sonnet view is created by the Cadence Virtuoso Interface as part of the model creation. This symbol can be placed in schematics to represent the model results. The three windows are shown below.



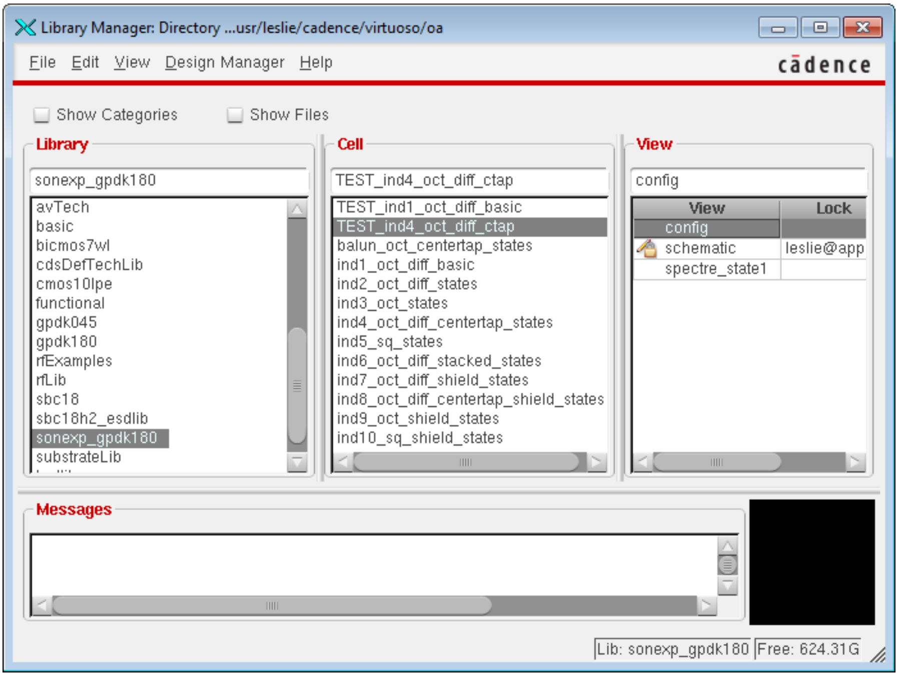
101 Close all three windows.

The view was opened to show the resulting model, but no action needs to be taken.

Config View

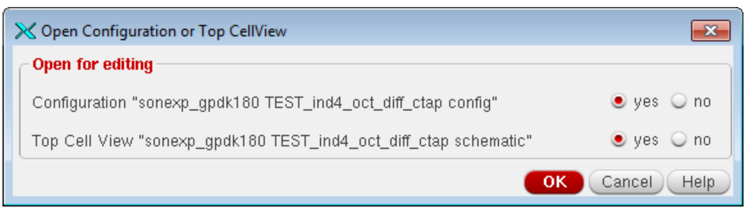
A config view uses the model symbol to refer to a specific model produced by a Sonnet simulation. That view can then be used in running simulations in Cadence Virtuoso, allowing you to use your Sonnet EM results in your netlist simulation. The config view also allows you to easily switch between models, if more than one model was created in your Sonnet simulation.

102 Select *Tools* ⇒*Library Manager* in the CIW.



103 Open the config view of the test bench schematic, TEST_ind4_oct_diff_ctap, that was provided as part of your example files.

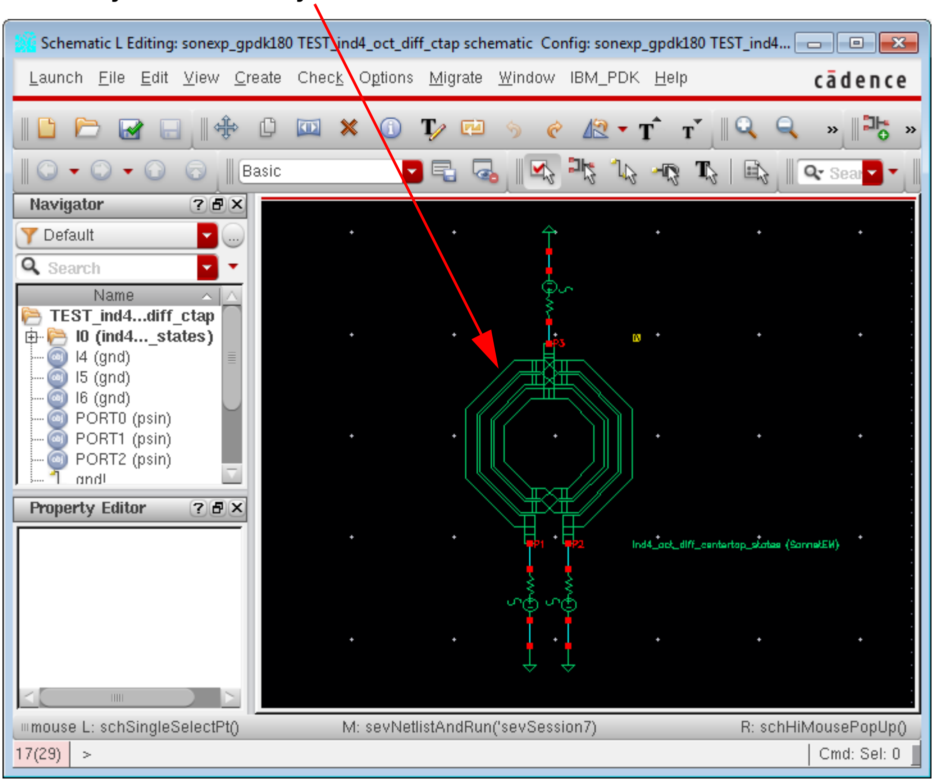
The Open Configuration or Top Cellview dialog box appears on your display. There are two elements to the configuration view, the schematic and the configuration. The dialog box defaults to opening both.

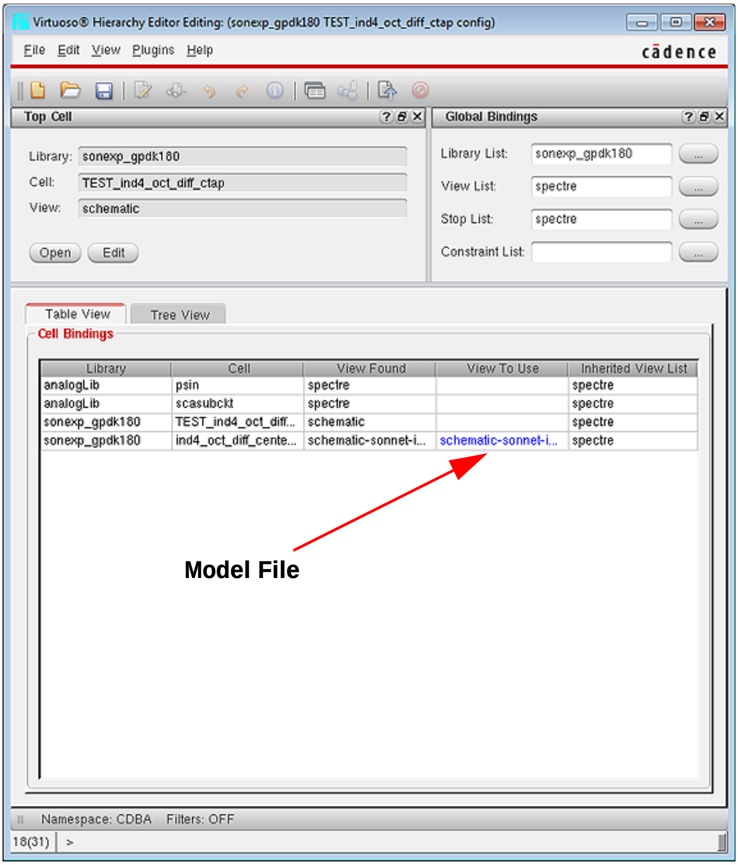


104 Click on the OK button to close the dialog box and open the schematic and configuration.

The schematic is opened for the test bench and the Hierarchy Editor is opened for the configuration.

Layout look-alike symbol





The Hierarchy Editor defines which view to use for the symbol in the test bench schematic. You created three different models during your simulation. By using the symbol in the test bench, you may easily switch between model types by modifying the View to Use. The inductor model is presently selected (this was done for the purposes of the tutorial in order to demonstrate switching models.)

105 Right-click on the bottom line in the Hierarchy Editor, then select *Set Cell View* ⇒ *schematic-sonnet-bbspice-spectre* from the pop-up menu.

When the symbol is used in the test bench, the model used will be the Broadband Spice model created during your Sonnet analysis.

106 Select *File* ⇒ *Save* from the main menu of the Hierarchy Editor dialog box.

This saves which model will be used in the config view. With the broadband spice model selected in the configuration, you are ready to simulate the test bench circuit.

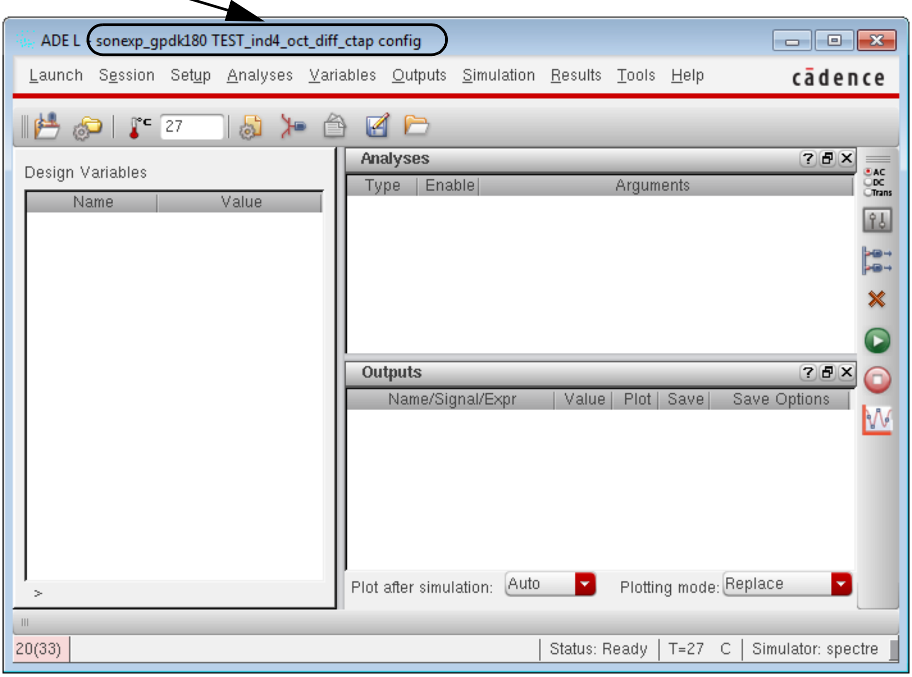
107 In the schematic window, select *File* ⇒ *Check and Save* from the main menu.

This updates the schematic with the new information from the Hierarchy Editor so that the test bench schematic is referencing the proper model.

108 Select Launch ADE L from the main menu of the schematic window.

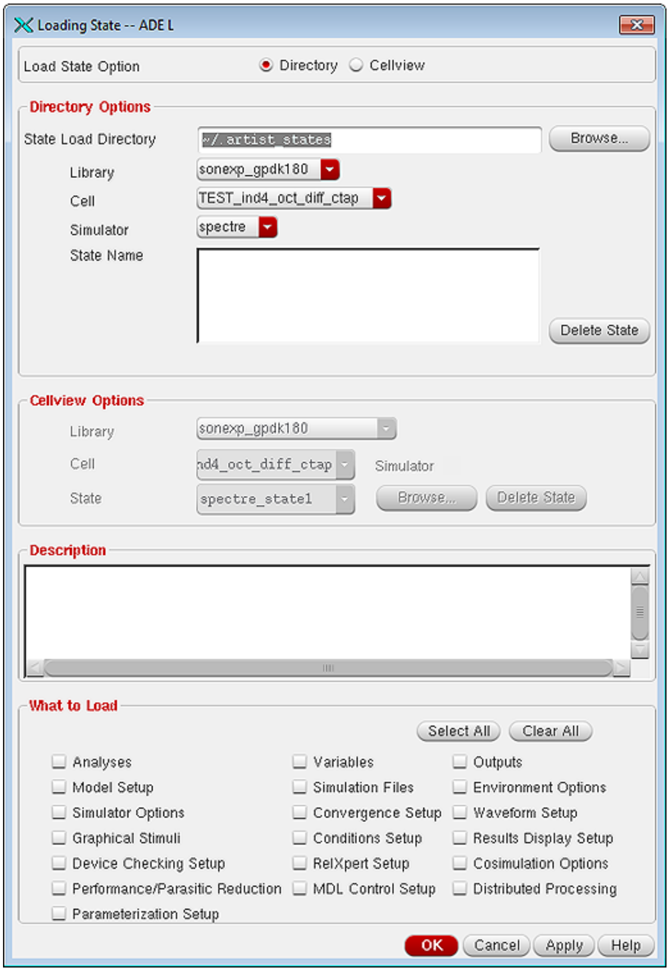
The Virtuoso Analog Design Environment appears on your display with the test bench loaded as the design. The name of the config view appears in the title bar.

Config View



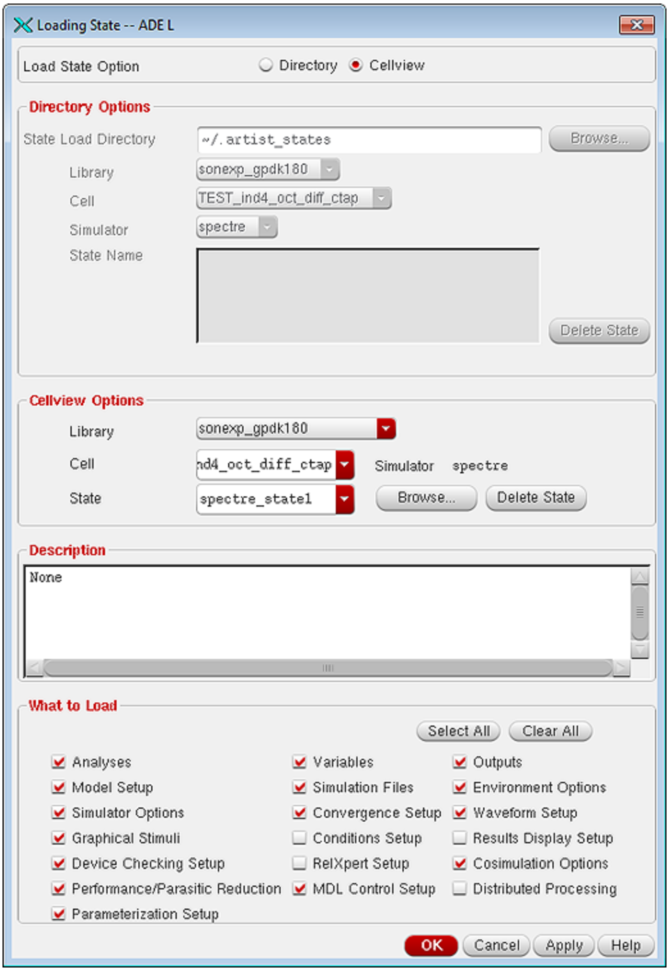
109 Select *Session* ⇒ *Load State* from the main menu of the ADE window.

Loading the state performs the setup for the simulation which was included in the example file. The Loading State dialog box appears on your display.



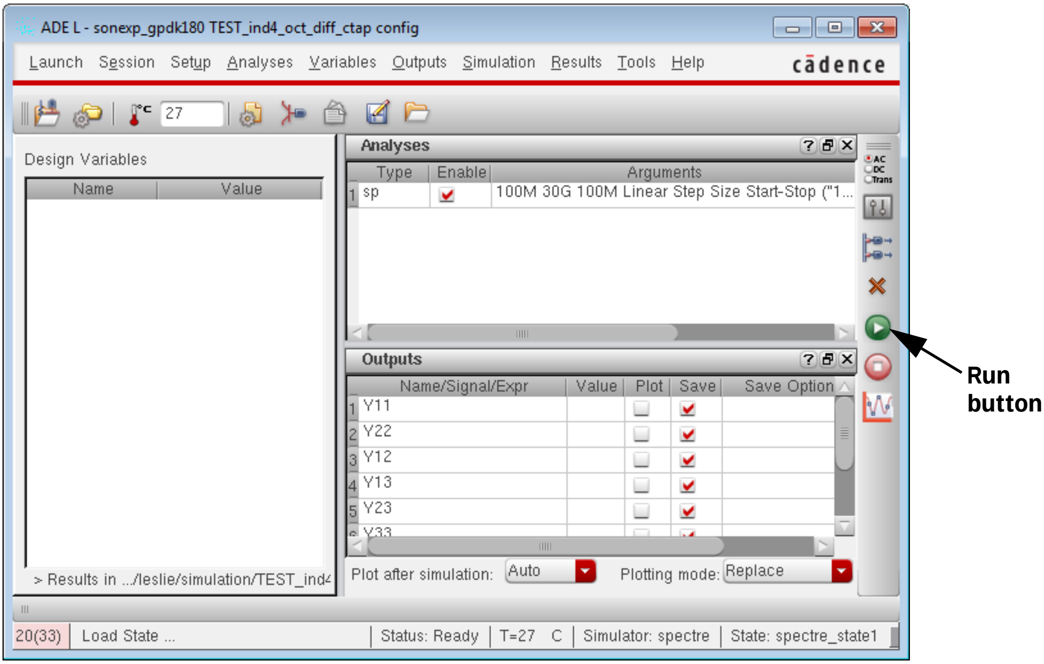
110 Click on the Cellview radio button in the Load State section of the dialog box.

The Cellview Options section of the dialog box is enabled, with the state spectre_state1 already selected. The bottom of the dialog box lists the elements included in the state.



111 Click on the OK button to close the dialog box and load the state.

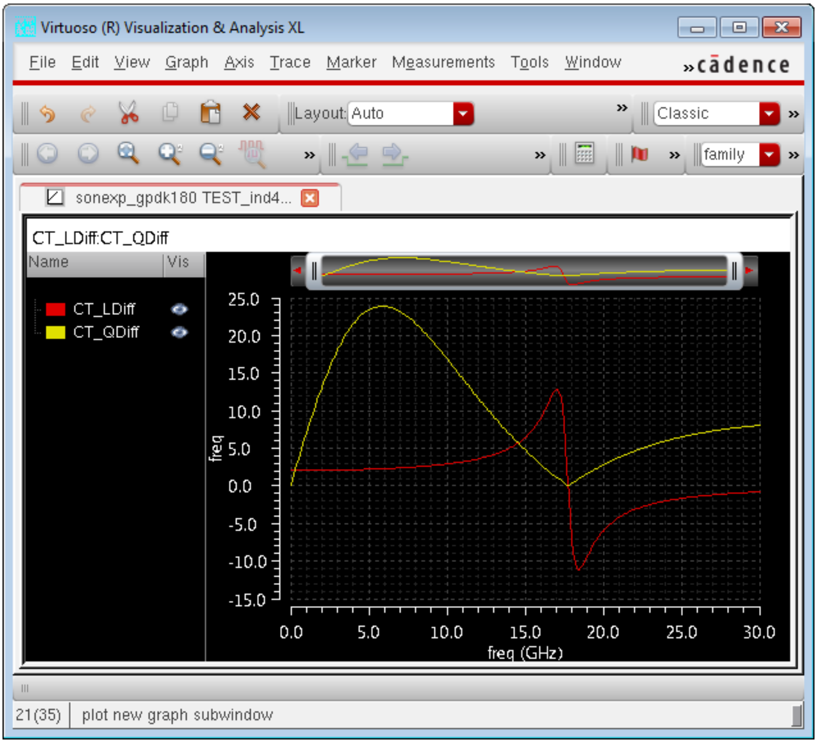
The ADE L window is updated and should appear similar to the picture below.



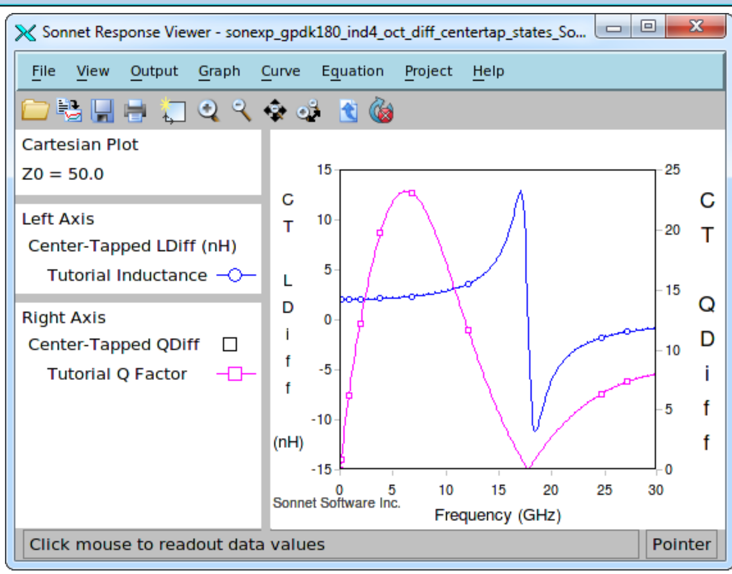
112 Select *Simulation* ⇒ *Netlist and Run* from the ADE L main menu to start the simulation.

You can also click on the green Run button on the right edge of the ADE L window to start the simulation. An output window appears on your display and is updated while the simulation runs. Once the simulation is complete, a plot of the results is opened. Similar to the plot you set up at the end of the EM simulation earlier in the tutorial, this plot also displays the inductance and the Q factor. However, this plot represents the broadband spice model, whereas the previous plot represented the raw S-parameters. If you compare the two plots, you can see that they are virtually identical, showing the broadband spice model accurately represents the original S-parameters.

Shown to the right is the plot using the broadband spice model resulting from the Accurate solver simulation. If you skipped the Accurate solver section of the tutorial, your plot will represent the model resulting from the Fast solver simulation and may appear slightly different.



Shown to the right is the plot generated earlier in the tutorial which represents the raw S-parameter data resulting from the Accurate solver simulation



This completes the Sonnet Cadence Virtuoso Interface tutorial.

Index