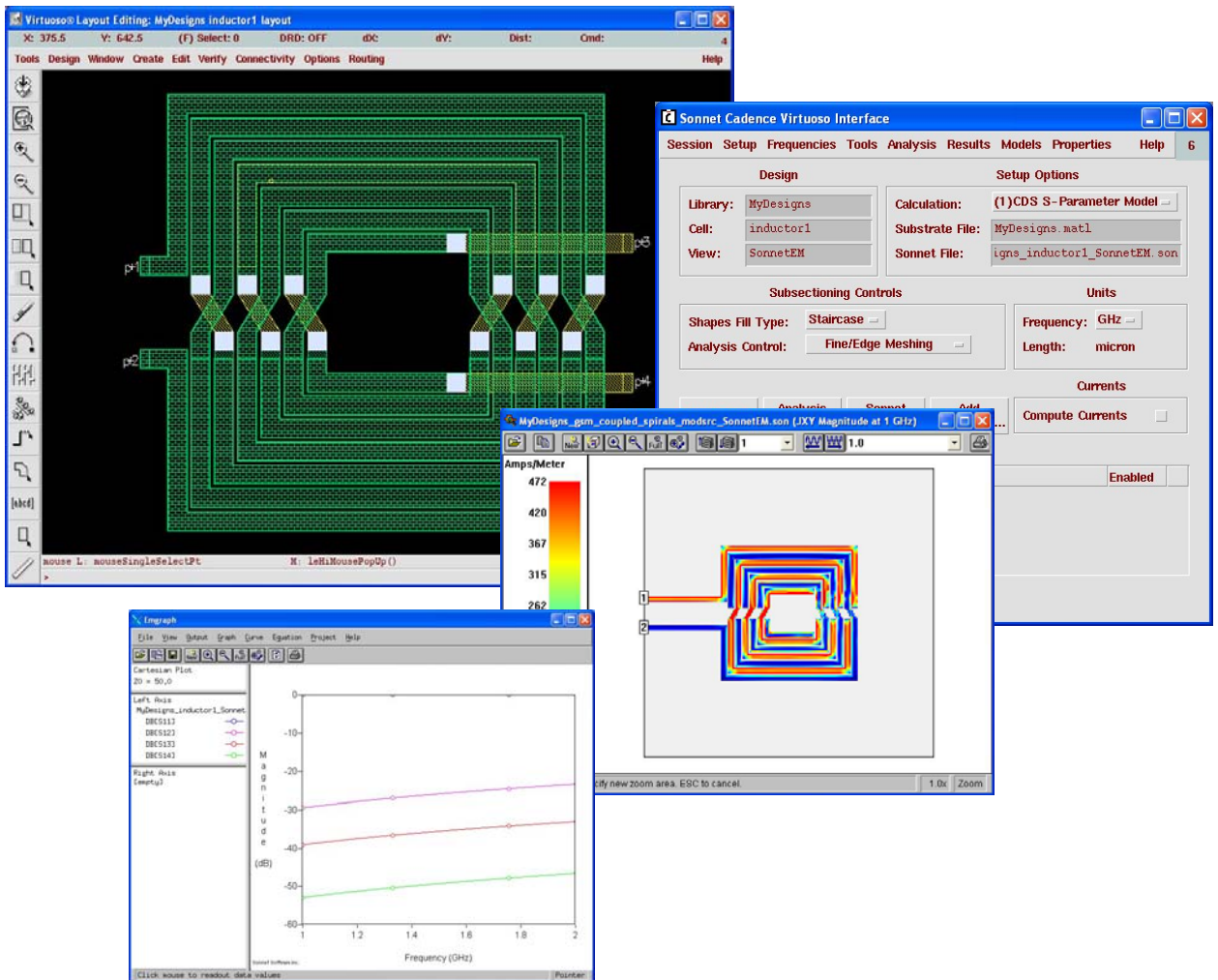


Using Sonnet EM Analysis with Cadence® Virtuoso® in RFIC Design

Sonnet Application Note: SAN-201A

May 2005



Description of Sonnet Suites Professional

Sonnet Suites Professional is an industry leading full-wave 3D Planar Electromagnetic (EM) field simulation software based on the Method of Moment (MoM) technique which accounts for all coupling and radiation effects from DC to THz. It also takes full advantage of mathematically robust and reliable FFT formulation which avoids time consuming, error prone numerical integration. Thus, combined with both MoM and FFT, Sonnet provides an added assurance that it will give outstanding accuracy every time on problems that have traditionally been difficult to solve. Problems with high dielectric constant, thin dielectric layers and/or small dimensions with respect to the wavelength are all handled especially well with Sonnet. Sonnet continues to be an indispensable tool for designers involved in RF/Microwave circuits such as distributed filters, transitions, Low Temperature Co-fired Ceramics, multi-layer RF packages, coplanar waveguides, and antennas. In addition, Sonnet has proven successful in mm-wave designs as well as in EMC and EMI analysis.

Sonnet Professional Key Benefits

- Accurately model passive components (inductors, capacitors, resistors) to determine values like RLC and Q-factor
- Accurately model multi layer interconnects and via structures
- Generate a technology accurate electrical model for arbitrary layout shapes
- Quantify parasitic coupling between components, interconnects and vias
- Include substrate induced effects like substrate loss and eddy currents
- Visualize the current flow in components, interconnects and vias

Sonnet Professional Key Features

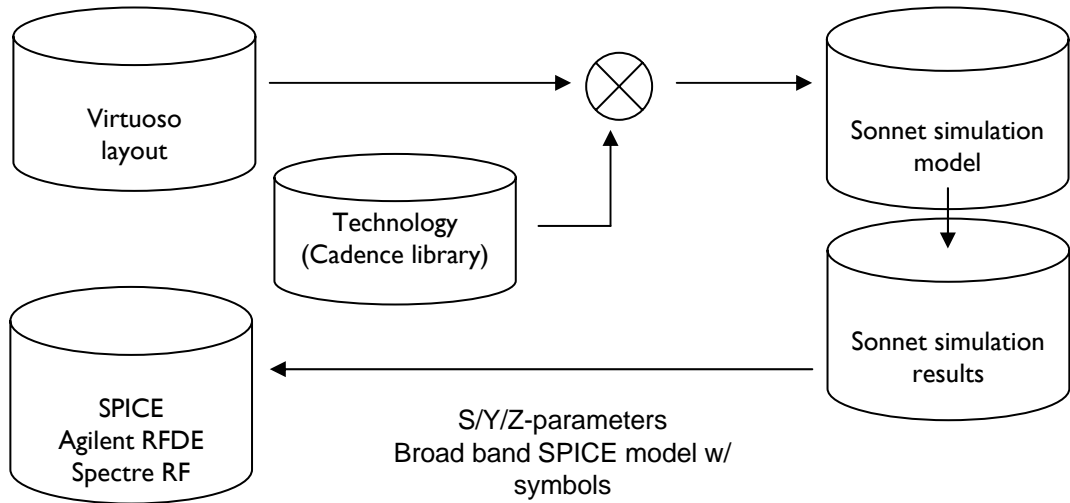
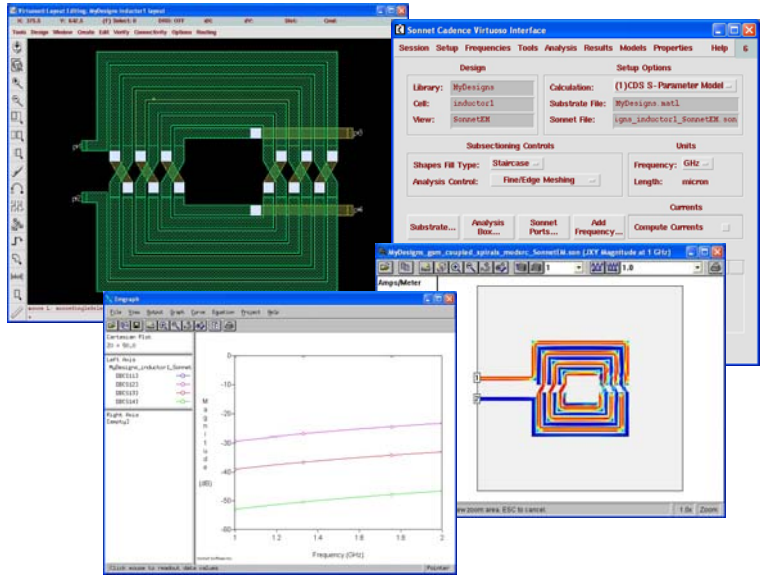
- FFT based Method of Moments analysis for ultimate reliability and accuracy
- Easy to learn, easy and efficient to use
- Only one high precision analysis engine – no need to switch between solvers
- Patented Conformal Meshing strategy for very efficient high accuracy meshing of curved structures
- Finite thickness modelling (including advanced N-sheet model)
- Dielectric bricks for truncated dielectric materials (e.g. MIM capacitor)
- Adaptive Band Synthesis for fast and reliable frequency sweeps with a minimum number of EM samples - more efficient than traditional approaches
- Easy to use data display for analysis results, including R, L, C, Q evaluation
- Equation capability for pre-defined or customized calculation on simulated data
- All configuration and technology setup is menu / dialog based– no need to edit configuration text files
- Remote simulation capability
- Compatible with the LSF cluster and load balancing system
- Seamless integration with Cadence® Virtuoso®, Agilent EEsof EDA's ADS, AWR® Microwave Office® and Analog Office™ and Eagleware-Elanix GENESYS™ design environments
- Sonnet Software Inc. is a Cadence Connections partner

Introduction

The new plug-in integration of the Sonnet Professional analysis program into the Cadence Virtuoso layout environment allows Cadence users to perform accurate EM modelling on selected Virtuoso cells, for physical verification of critical components and interconnections. This plug-in solution, which was released in Q4/2004, allows designers to extract a Sonnet EM analysis cell view directly from the layout view. Simulation results can be easily back annotated to the Cadence schematic.

High precision electromagnetic (EM) analysis with Sonnet complements the traditional RC extraction tools. With Sonnet, simulation results include all physical effects like coupling, skin effect, proximity effects, surface waves and radiation. While traditional RC extraction tools like Cadence Assura are optimised to handle full chip extraction, Sonnet analysis is most suitable to rigorously analyze individual components, interconnects and smaller parts of a layout with very high confidence.

With Sonnet, you can replace legacy 3D finite element solvers, which are often not integrated into the Cadence design flow and require cumbersome and error prone manual modelling and data transfer.



When to Use Sonnet Professional Analysis

The use of electromagnetic analysis with Sonnet Professional is especially valuable in the following design situations:

When parasitic coupling is present.

Parasitic coupling is not always easy to predict without using electromagnetic analysis. Even elements which are "sufficiently" far apart can suffer from parasitic coupling: inductive or capacitive coupling, resonance effects due to grounding and surface waves that might propagate at the substrate boundary under certain conditions. Sonnet Professional analysis is based on the physical properties of your technology and will account for such physical effects.

When accurate circuit models are not available or circuit model parameters are out of range.

Model based circuit simulators are based on models for a specific application, with limited parameter range. For example, only selected geometries, substrate types and substrate parameters are available. It is difficult to estimate the error induced by parameter extrapolation, so using models outside their designed parameter range is not suitable for critical applications.

Whenever a layout feature cannot be described by a circuit model, due to its geometry or technology, the physics based analysis with Sonnet Professional will provide the answer. An example for this could be a special inductor, capacitor or transformer which is not included in the design kit. Sonnet can be used to analyze those components "on the fly", or generate a full library of component models with trustworthy electrical results.

Sonnet Professional Design Process

The following steps describe a typical process for creating and simulating a design with Sonnet Professional for Cadence Virtuoso.

1-Define the stack-up and material properties.

A simulation model for Sonnet Professional contains a number of layered dielectrics, conductor levels at the interface between dielectric layers and vias to connect different conductor levels. To simulate a layout in Sonnet, these material properties and definitions ("technology") are required.

Typically, the EDA support group will define and qualify a technology file for a Cadence library and release that to the RFIC designers. This is an easy and straightforward process once the material properties are known, because that definition is fully driven by menus and dialog boxes.

When a designer creates a Sonnet simulation view of a cell, that pre-defined Sonnet technology file will be automatically loaded from the Cadence library.

The screenshot shows the Sonnet Cadence Virtuoso Interface. The main window has a menu bar (Session, Setup, Frequencies, Tools, Analysis, Results, Models, Properties, Help) and a toolbar. The 'Setup Options' panel includes fields for Calculation (set to '(1) CDS S-Parameter Model'), Substrate File (MyDesigns.matl), and Sonnet File (igns_inductor1_SonnetEM.son). Below this is a 'Substrate File Setup' dialog box with a table for 'Mapping from Cadence to Sonnet' and an 'Edit Mapping Layer' dialog box for the 'metall' layer.

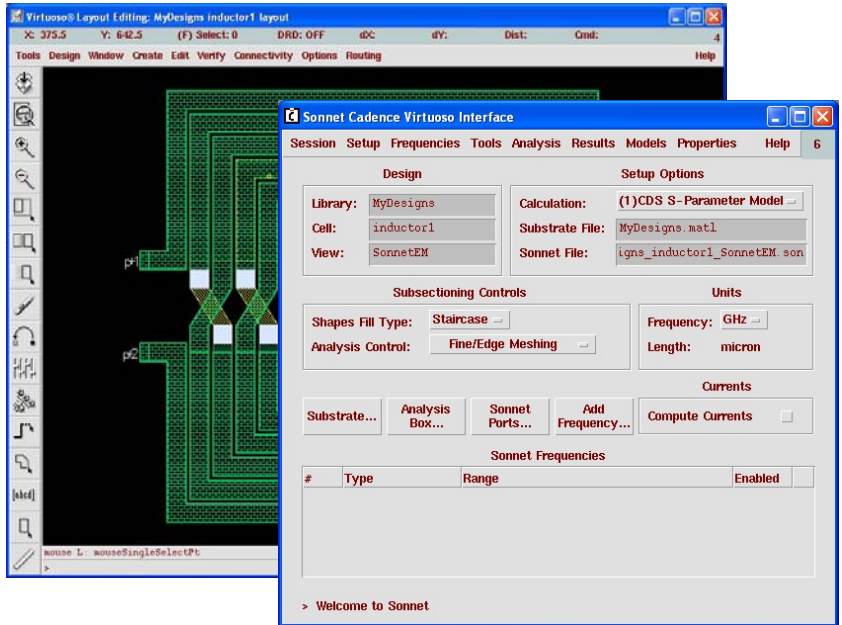
| # | Cds Layer | Map | Son Type | Son Level | Son Material | Son Fill Type |
|---|-----------|-----|----------|-----------|--------------|---------------|
| 7 | metal3 | Yes | Metal | 1 | Metal3 | Default |
| 6 | metall | Yes | Metal | 3 | Metall | Default |
| 3 | via2 | Yes | Via | 1 to 3 | Metal2 | Default |
| 9 | cont | No | Metal | 0 | Lossless | Default |
| 8 | metal2 | No | Metal | 0 | Lossless | Default |
| 5 | poly1 | No | Metal | 0 | Lossless | Default |
| 4 | via | No | Metal | 0 | Lossless | Default |
| 2 | diff | No | Metal | 0 | Lossless | Default |
| 1 | ndiff | No | Metal | 0 | Lossless | Default |

| Cadence | Sonnet |
|---------------|---|
| Layer: metall | Type: Metal, Fill Type: Default, Material: Metall, Level: 3 |

Substrate Definition

2-Make Sonnet View.

Based on the layout view of a cell, the designer can easily create the Sonnet analysis view for that cell. If the cell layout contains hierarchy, the hierarchy will be resolved to create a flattened Sonnet EM view.



Make Sonnet View

3-Define the analysis box.

With Sonnet's FFT based approach to Method of Moments (MOM), it is necessary to define a finite analysis area ("box") and a uniform sampling resolution ("cell size"). This defines the granularity for the analysis, and usually requires a trade off between analysis time and accuracy. The user defined cell size is very useful to control the meshing density of very complex shapes, to avoid over-meshing the circuit without any need for manual clean-up.

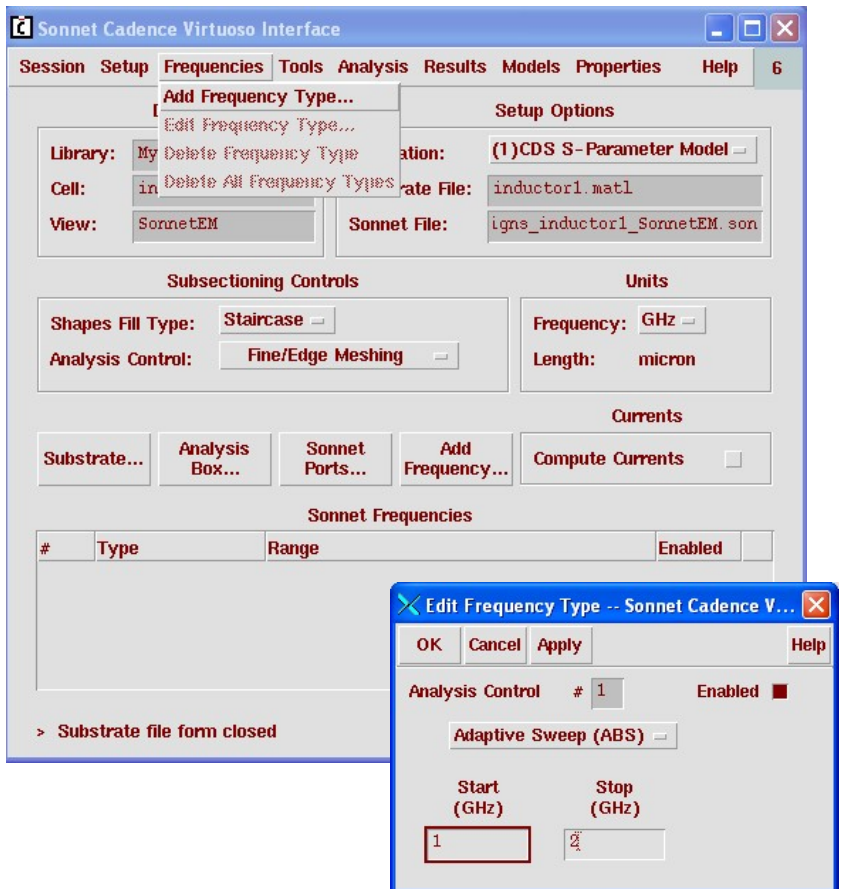
4-Assign port numbers and properties.

In Sonnet, circuit ports connect the analysed structure with the outer circuit. Different port types are available for different purposes. Cadence pins can be converted to Sonnet ports fully automatically, based on the defined mapping. Where needed, feed lines will be extended to the boundary of the analysis area automatically, and de-embedded from the analysis results.

5-Define analysis frequencies.

Sonnet uses the Method of Moments where the circuit is analyzed at different frequencies of interest. With the unique Adaptive Band Synthesis (ABS) in Sonnet, an accurate wide band response can be obtained from a minimum number of EM simulated data points within a given frequency range.

Using ABS, the designer only needs to enter the start and stop frequency. Of course, composite frequency sweeps are also possible, as well as a "DC" point.



Define Analysis Frequencies

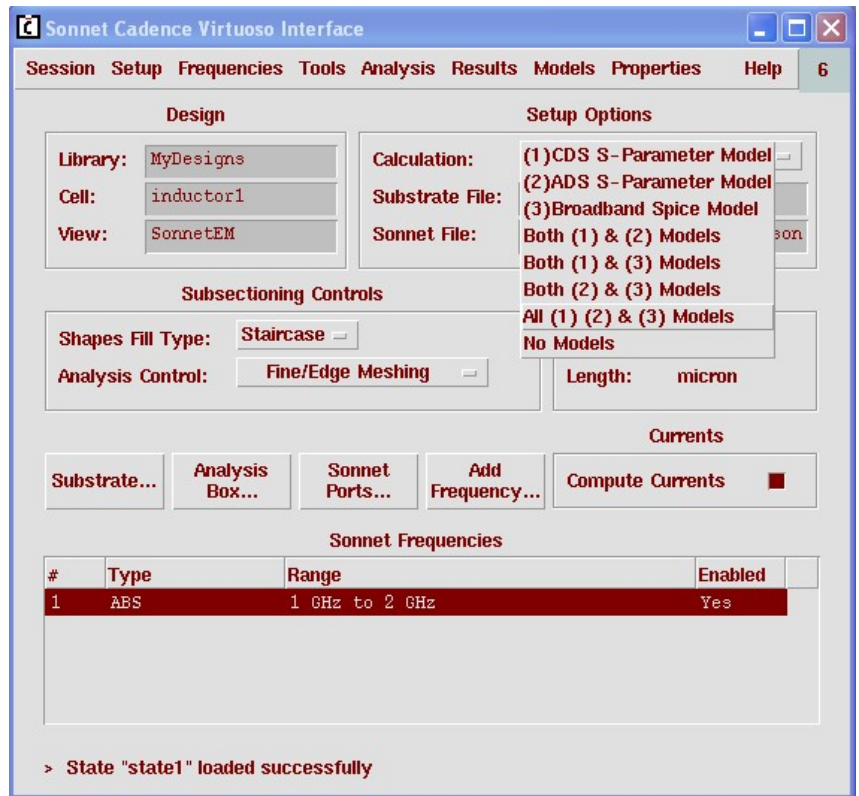
6-Define desired output format.

The analysis results can be in many different formats such as S/Y/Z parameters, a schematic symbol with ADS or Spectre frequency domain data or a schematic symbol for broad band SPICE extracted data. The designer can define the desired output data before starting the Sonnet EM simulation, or generate the data later as a post processing task after EM analysis has finished.

Note: The Broadband Spice Model can only be generated if you have purchased a Broadband Spice Extractor license from Sonnet. Please see your system administrator if you are unsure of the availability of this option.

The Broadband Spice feature allows you to create a circuit model which is valid over broad band. The Broadband Spice Extractor generates models that can be used in Spice as a "black box" representing the high frequency behavior of your project. This feature supports PSpice and Cadence Spectre formats.

The Broadband Spice model is a direct result of a rational polynomial fit to the S-Parameter data. This polynomial is used to generate the equivalent lumped element circuits which may be used as an input to either PSpice or Spectre. 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. Sonnet has optimized these models to be causal. The models used are resistors, capacitors, inductors and dependent Voltage and Current sources.



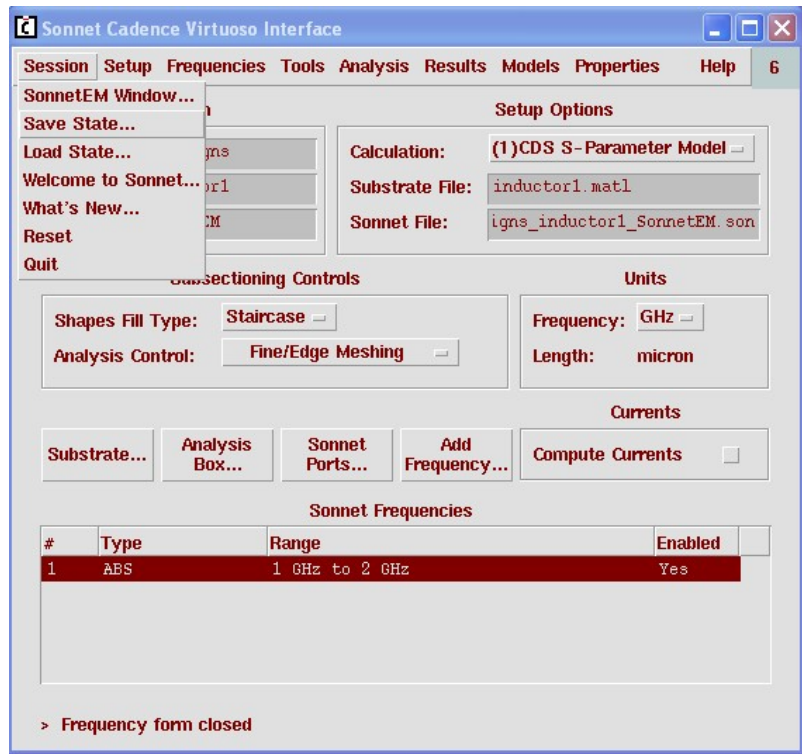
Choosing Output Formats including the Broadband Spice Model

7-Save state.

Save the current "state" of the analysis settings for later use. This way, all settings can be loaded into a similar project easily.

8-View simulation model in Sonnet (optional).

If desired, the designer can take the simulation model as a 2D or 3D view to Sonnet, and review it in the Sonnet model editor. This is especially useful for complex multi layer circuits. The Sonnet interface will keep track of any manual changes, to makes sure that the Virtuoso view and the Sonnet view stay in sync.



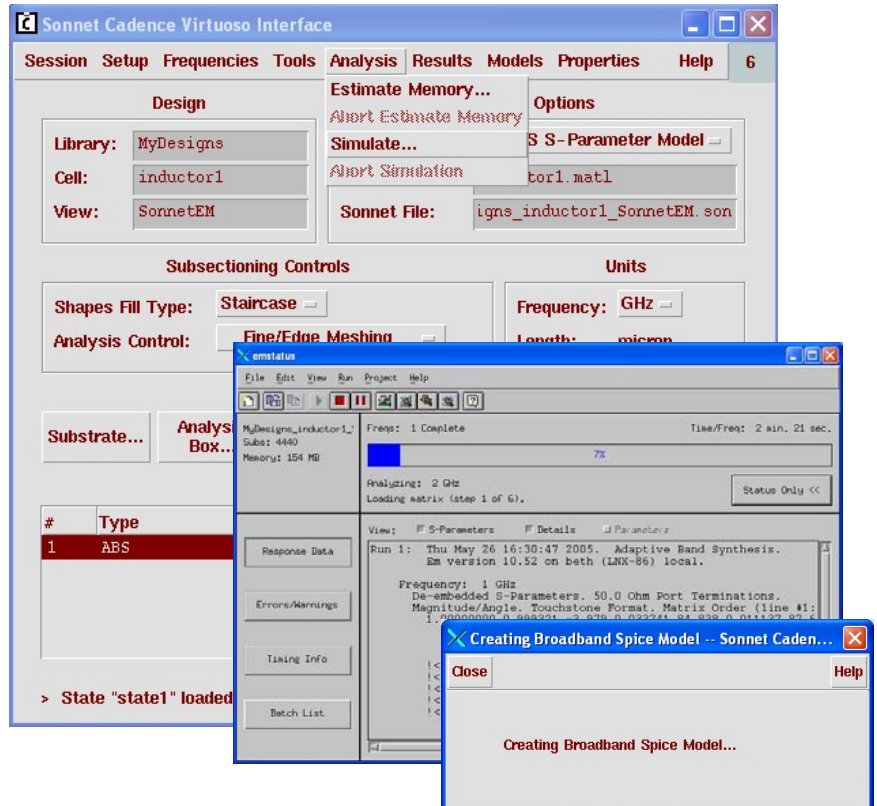
Save the Analysis Settings

9-Simulate the circuit.

The simulation can be started from within Cadence Virtuoso. The Sonnet analysis monitor will show analysis progress and an estimate of the memory requirement and total simulation time.

If the Broadband Spice Model was chosen as one of the desired output formats (Step 6) then Creating Broadband Spice Model form will appear upon completion of the EM Analysis.

Note: The Broadband Spice Model can only be generated if you have purchased a Broadband Spice Extractor license from Sonnet. Please see your system administrator if you are unsure of the availability of this option.



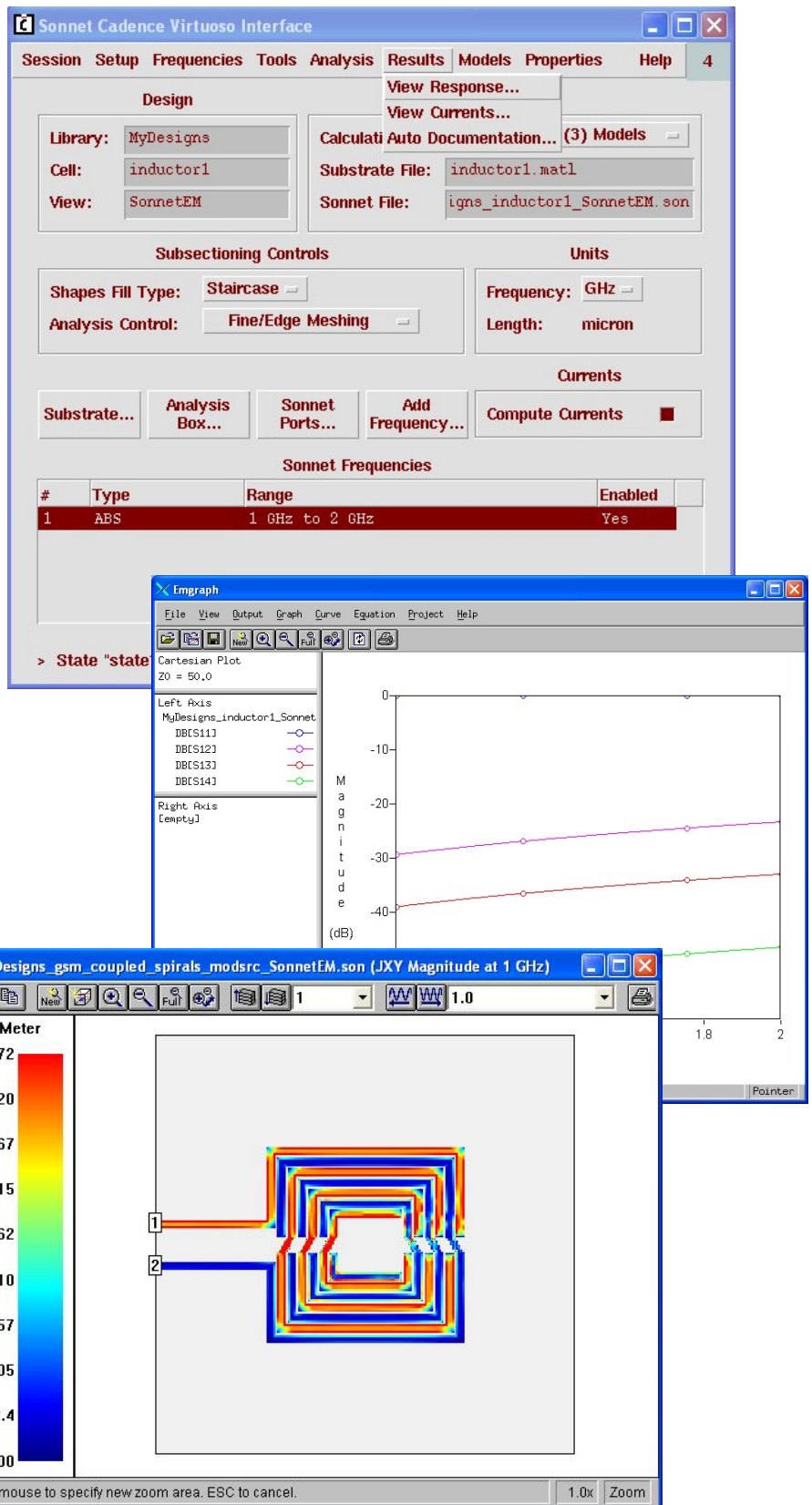
Simulate the Circuit

10-View simulation results.

When the simulation is done, simulation results can be viewed with the Sonnet plotting tool. It is possible to plot any available results while the analysis engine is still computing more frequencies. Parameters like inductor L and Q or capacitor C and Q can be plotted immediately by using pre-defined equations.

When the current distribution is of interest, the simulation can be configured to store current density data. That data can be visualized in various ways as 2D or 3D graphs. For a better understanding of the circuit, the current density or charge visualization can also be animated over time or frequency. Current density results can be exported to a spread sheet for post processing.

In the current density image shown here, note the smooth display of the current distribution, which shows high edge current (required to calculate the conductor losses properly) and current crowding. This is one indication of the high resolution and high precision of the Sonnet analysis result.



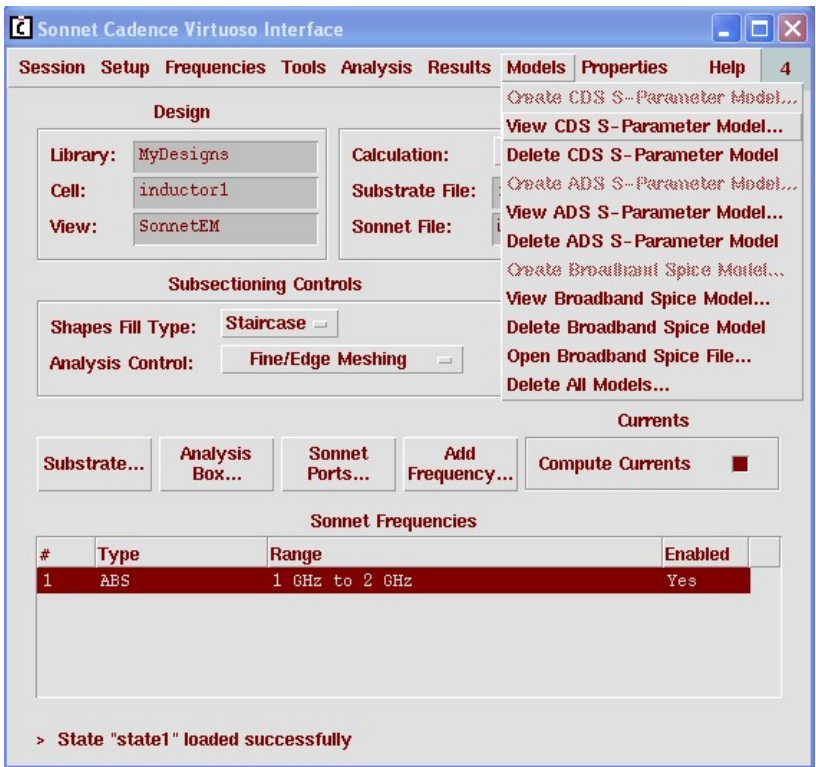
View Simulation Results

11-Back annotate results to the Cadence schematic.

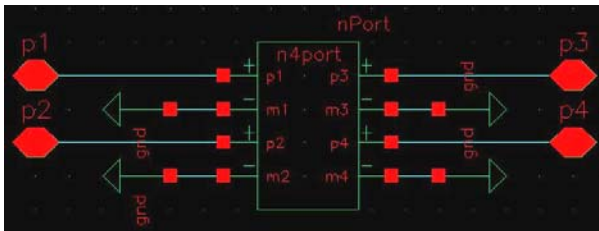
When the Sonnet analysis is finished, results are available in different formats. This includes both S/Y/Z-parameters for frequency domain simulation, as well as a broad band SPICE extraction for time domain (SPICE) simulation.

Note: The Broadband Spice Model can only be generated if you have purchased a Broadband Spice Extractor license from Sonnet. Please see your system administrator if you are unsure of the availability of this option.

Cadence schematic symbols that hold the Sonnet analysis results are created automatically, with all the required mapping files. A sub-circuit schematic for the analyzed cell is also created automatically.



Create Cadence models from simulation results



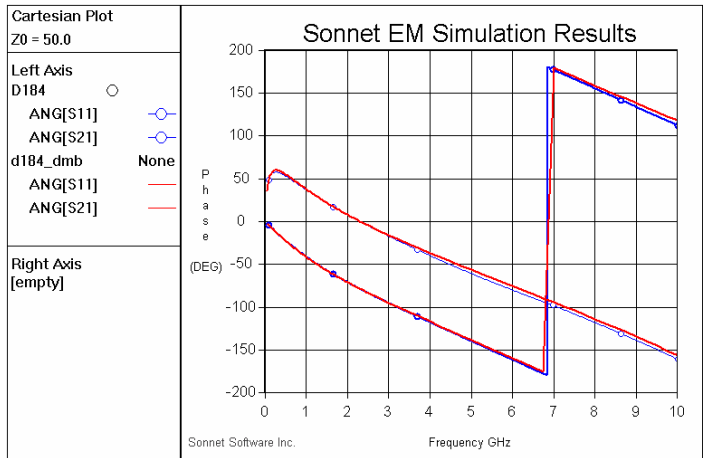
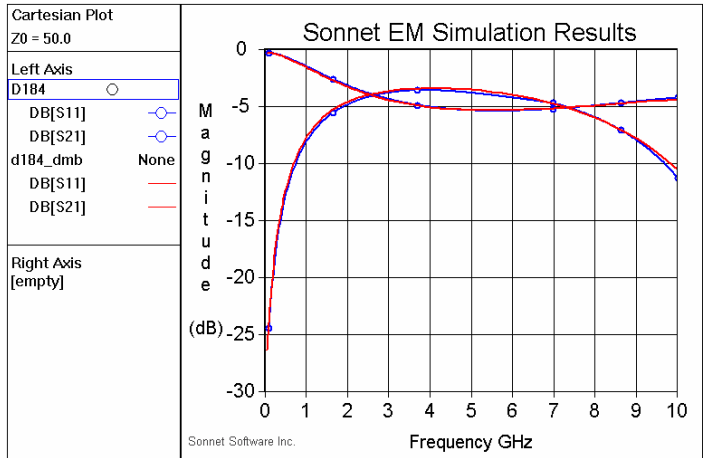
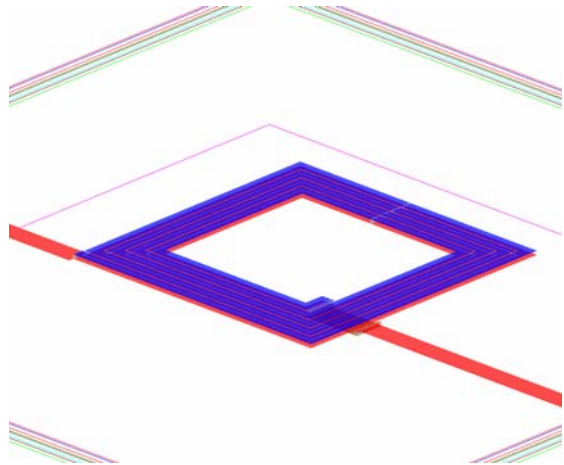
Symbol and schematic for the cell

Example 1

Spiral Inductor on 100 um Silicon (step-graded conductivity in substrate)

5 insulating layers between 1 um and 3 um, thick metal effects modeled.

Data and design courtesy of Motorola SPS/WISD



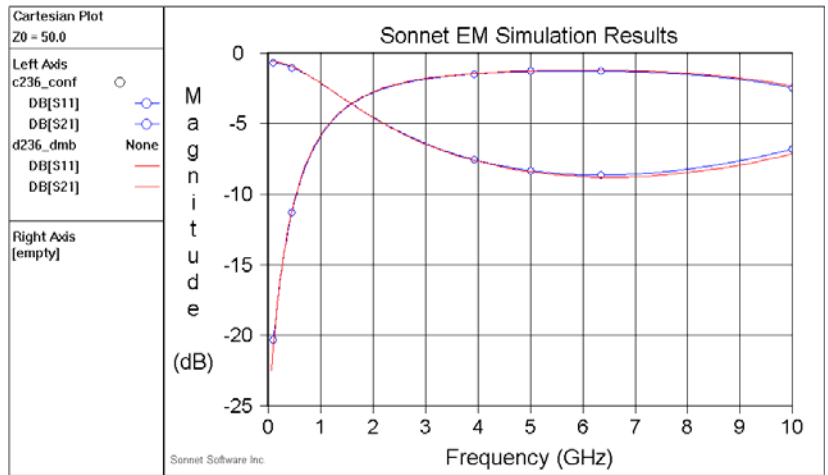
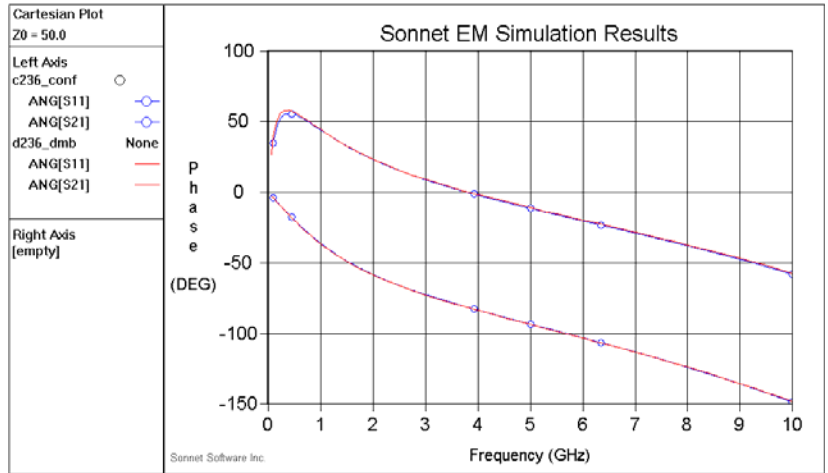
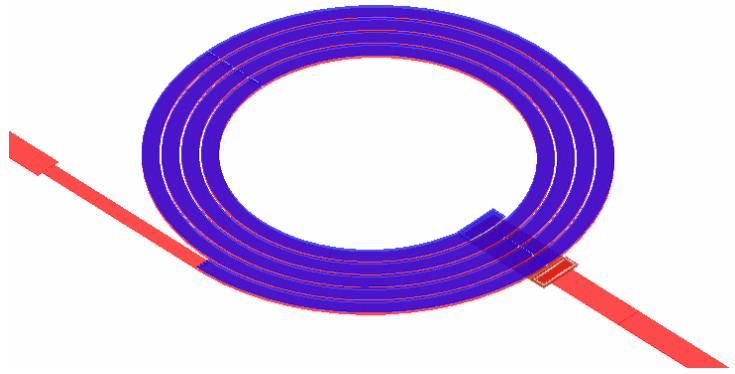
Measured vs. Calculated
red = measured
blue = Sonnet result

Example 2

Spiral Inductor on 100 μm Silicon (step-graded conductivity in substrate)

5 insulating layers between 1 μm and 3 μm , thick metal effects modeled.

Data and design courtesy of Motorola SPS/WISD



Measured vs. Calculated

red = measured

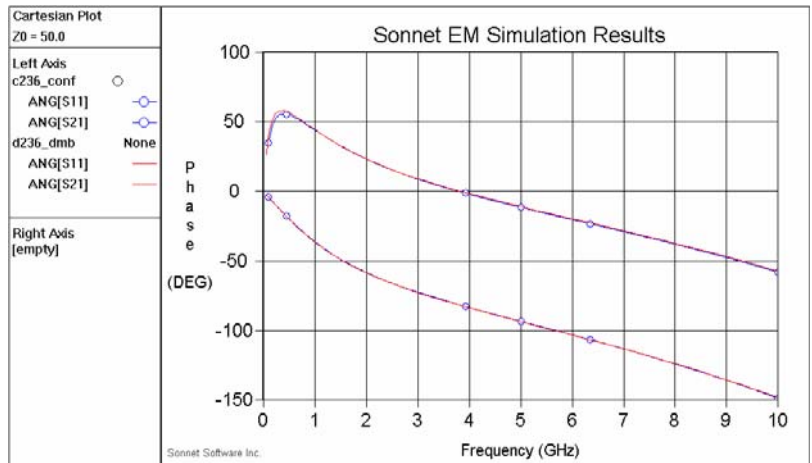
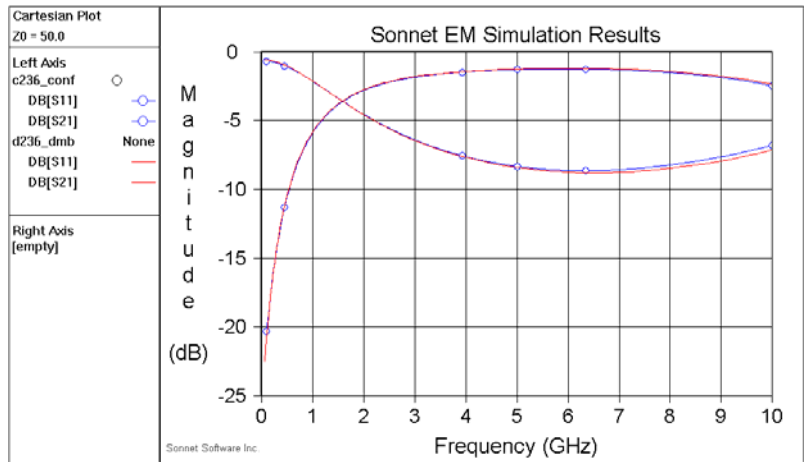
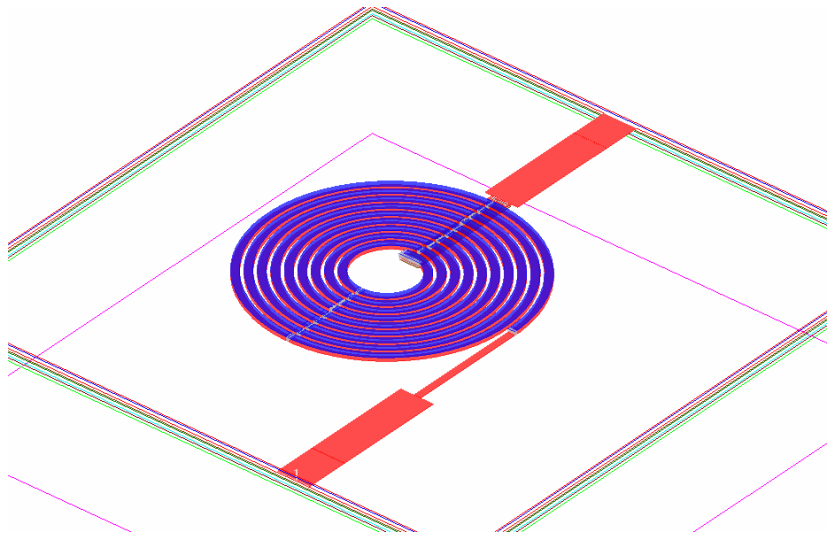
blue = Sonnet result

Example 3

9.25-turn Circular Spiral Inductor on 100 um Silicon (step-graded conductivity in substrate).

5 insulating layers between 1 um and 3 um, thick metal effects modeled.

Data and design courtesy of Motorola SPS/WISD



Measured vs. Calculated

red = measured

blue = Sonnet result

By in large, this application note is based on the original work of
Dr.-Ing. Volker Muehlhaus of
Muehlhaus Consulting & Software GmbH
www.muehlhaus.com

Sonnet Software, Inc.

Phone: 315-453-3096

Toll Free in North America: 877-7SONNET

Email: info@sonnetsoftware.com

Web: www.sonnetsoftware.com

United States Regional Sales:

Eastern US Sales Office: 203.439.0815

Western US Sales Office: 303.443.2646

Japan:

Sonnet Giken Co., Ltd.

Phone: 43-463-6663

Email: info@mail.sonnetsoftware.co.jp

Web: www.sonnetsoftware.co.jp

Germany:

Muehlhaus Consulting & Software GmbH

Phone: +49 (2302) 91438-0

Email: info@muehlhaus.com

Web: www.muehlhaus.com

