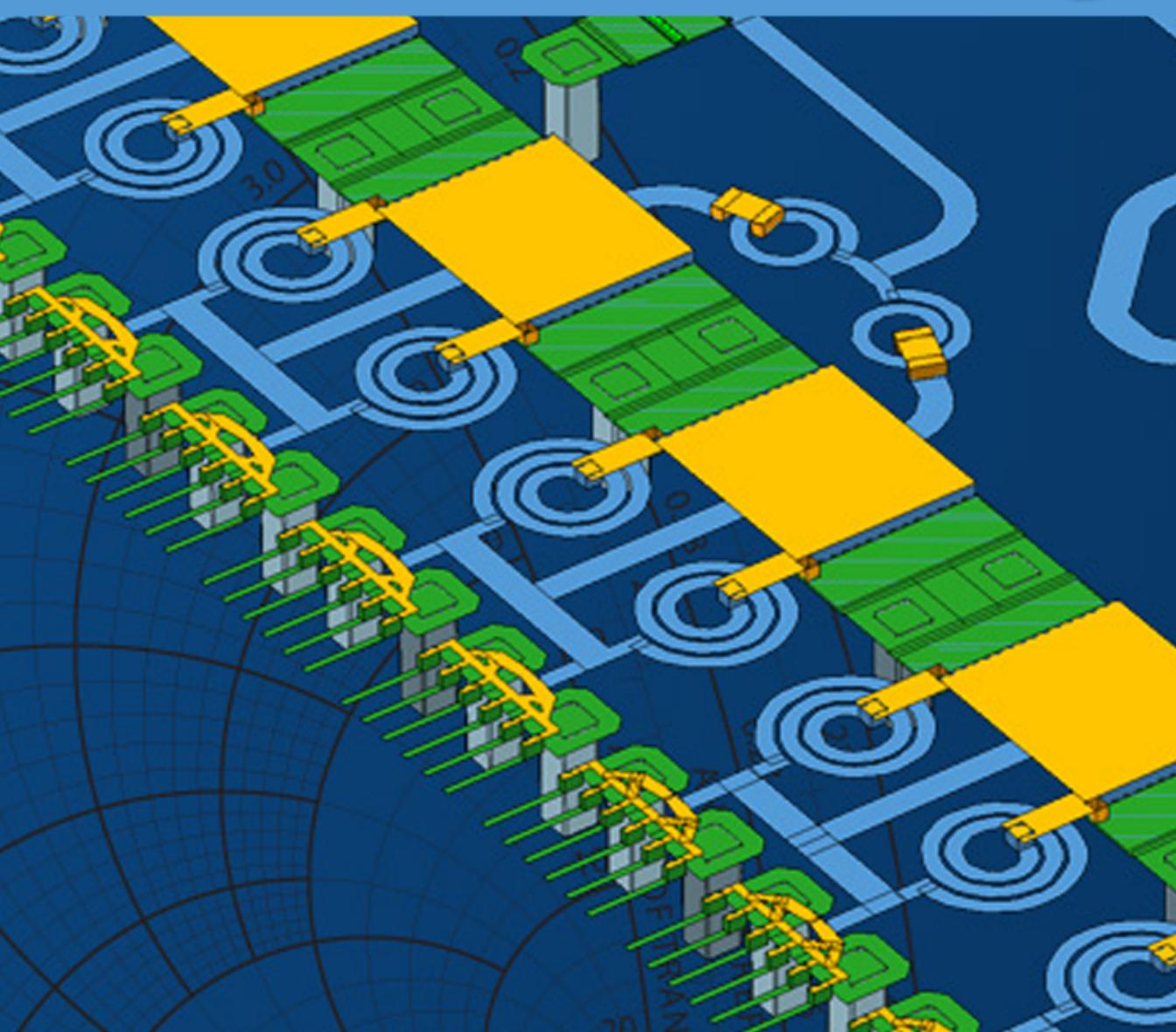


High Frequency Electromagnetic Software

# SONNET® Suites 16



User's Guide

*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.*

# **SONNET® USER'S GUIDE**

*Published: May 2018*

## **Release 16**

Sonnet Software, Inc.  
126 N. Salina Street  
Syracuse, NY 13202  
Phone: (315) 453-3096  
Fax: (315) 451-1694

Technical Support: support@sonnetsoftware.com  
Sales Information: sales@sonnetsoftware.com  
[www.sonnetsoftware.com](http://www.sonnetsoftware.com)

© Copyright 1989,1991,1993, 1995-2018 Sonnet Software, Inc. All Rights Reserved  
Registration numbers: TX 2-723-907, TX 2-760-739

Rev 16.56

---

## Copyright Notice

---

Reproduction of this document in whole or in part, without the prior express written authorization of Sonnet Software, Inc. is prohibited. Documentation and all authorized copies of documentation must remain solely in the possession of the customer at all times, and must remain at the software designated site. The customer shall not, under any circumstances, provide the documentation to any third party without prior written approval from Sonnet Software, Inc. This publication is subject to change at any time and without notice. Any suggestions for improvements in this publication or in the software it describes are welcome.

---

## Trademarks

---

The program names, *xgeom*, *emstatus*, *emu*, *patvu*, *dxfgeo*, *ebridge*, *emgraph*, *gds*, *cvbridge*, *emserver*, *emclient*, *sonntcds*, and *sonntawr*, *sonntawr64*, Blink, Co-calibrated, Lite, LitePlus, Level2 Basic, Level2 Silver, and Level3 Gold are trademarks of Sonnet Software, Inc.

Sonnet®, *em*®, and *emCluster*® are registered trademarks of Sonnet Software, Inc.

Windows XP, Windows Vista, Windows 7, Windows 8, Windows 10 and Internet Explorer® are U.S. registered trademarks of Microsoft Corporation.

AutoCAD and Drawing Interchange file (DXF) are trademarks of Auto Desk, Inc.

Cadence® and Virtuoso® are registered trademarks of Cadence Design Systems, Inc.

GLOBALFOUNDRIES® is a registered trademark of GlobalFoundries, Inc.

Agilent, ADS, and Touchstone are trademarks of Keysight Technologies.

NI AWR and Microwave Office are registered trademarks and EM Socket is a trademark of National Instruments, Inc.

HSPICE is a registered trademark of Synopsys, Inc.

GDSII is a trademark of Calma Company.

Flexera Software, Flexlm, FlexNet, InstallShield, are trademarks of Flexera Software, Inc. and/or InstallShield Co.Inc. in the United States of America and/or other countries.

OSF/Motif is a trademark of the Open Software Foundation.

X Window System is a trademark of The Open Group

Linux® is a registered trademark of Linus Torvalds.

Red Hat® is a registered trademark of Red Hat, Inc.

SUSE®, openSUSE® and SLES® are registered trademarks of SUSE LLC.

OpenGL® is a registered trademark owned by Silicon Graphics, Inc.

MATLAB is a registered trademark of The MathWorks, Inc. in the United States and/or other countries.

Acrobat® is a registered trademark of Adobe Systems Incorporated.

Xpeedic® and IRIS® are registered trademarks of Xpeedic Technology.

ODB++® is a registered trademark of Mentor Graphics Corporation.

Modelithics® is a registered trademark of Modelithics, Inc.

---

---

TABLE OF CONTENTS

TABLE OF CONTENTS . . . . .	5
1 INTRODUCTION . . . . .	15
The Sonnet Design Suite . . . . .	15
The Analysis Engine, <i>em</i> . . . . .	20
<i>Em</i> Origins . . . . .	22
2 WHAT'S NEW IN SONNET SUITES RELEASE 16 . . . . .	23
3 SUBSECTIONING . . . . .	27
Metal Fill . . . . .	29
Tips for Selecting A Good Cell Size . . . . .	29
Viewing the Subsections . . . . .	31
Using the Speed/Memory Control . . . . .	32
Changing the Subsectioning of a Polygon . . . . .	35
Default Subsectioning of a Polygon . . . . .	35
X Min and Y Min with Edge Mesh Off . . . . .	37
X Min and Y Min with Edge Mesh On . . . . .	39
Using X Max and Y Max . . . . .	40
Setting the Maximum Subsection Size Parameter . . . . .	41
Defining the Subsectioning Frequency . . . . .	41
Diagonal Fill Subsectioning . . . . .	42
Conformal Mesh Subsectioning . . . . .	43
Conformal Mesh Subsectioning Control . . . . .	45
4 METALIZATION AND DIELECTRIC LAYER LOSS . . . . .	47
Planar Metalization Loss . . . . .	48
Sonnet's Loss Model for Planar Metalization . . . . .	48
Problems In Determining Planar Metal Loss . . . . .	49
Determining Good Input Values . . . . .	50
Creating Planar Metal Types . . . . .	50
Via Metalization Loss . . . . .	58
Creating Via Metal Types . . . . .	59
Metal Libraries . . . . .	65
Dielectric Layer Loss . . . . .	66

# Sonnet User's Guide

---

Dielectric Layer Parameters . . . . .	66
Frequency Dependent Loss . . . . .	68
How to Create a New Dielectric Layer. . . . .	69
Dielectric Libraries . . . . .	69
<b>5 PORTS . . . . .</b>	<b>71</b>
Port Type Overview . . . . .	71
Port Normalizing Impedances . . . . .	73
Changing Port Impedance . . . . .	74
Special Port Numbering . . . . .	75
Ports with Duplicate Numbers . . . . .	75
Ports with Negative Numbers . . . . .	76
Using Port Number Zero for Ground Connections. . . . .	77
Changing Port Numbering . . . . .	78
Renumbering All Ports . . . . .	79
Port Placement with Symmetry On . . . . .	79
Independent Reference Planes and Diagonal Ports . . . . .	80
Applying the Diagonal Port Feature to a Port . . . . .	81
Diagonal Port Feature Limitations . . . . .	87
Standard Box-wall Port . . . . .	87
Adding Box-wall Ports . . . . .	88
Ref. Planes and Cal. Lengths for Box-wall Ports . . . . .	88
Co-calibrated Internal Ports . . . . .	89
Calibration Group Properties. . . . .	89
Adding Co-calibrated Ports . . . . .	93
Rules for Co-Calibrated Ports . . . . .	94
Ref. Planes and Cal. Lengths for Co-calibrated Ports . . . . .	97
Use in Components . . . . .	97
Via Ports. . . . .	98
Adding Via Ports . . . . .	99
Automatic-Grounded Ports. . . . .	99
Special Considerations for Auto-Grounded Ports . . . . .	100
Adding Auto-grounded Ports . . . . .	101
Ref. Plane and Cal. Length for Autogrounded Ports . . . . .	101
Ungrounded Internal Ports . . . . .	101
<b>6 DE-EMBEDDING . . . . .</b>	<b>103</b>
Introduction . . . . .	103

## Table of Contents

---

Port Discontinuities . . . . .	105
Shifting Reference Planes . . . . .	106
Single Feed Line . . . . .	107
A Common Misconception . . . . .	108
Rules for Reference Planes . . . . .	109
Feed Lines Must Be Transmission Lines. . . . .	109
De-embedding Without Reference Planes . . . . .	110
Short Reference Planes for Boxwall Ports . . . . .	110
Reference Planes Longer than the Feedline . . . . .	112
Calibration Standards. . . . .	114
Modifying Calibration Standard Lengths . . . . .	118
Coupled Transmission Lines. . . . .	120
Shared reference planes . . . . .	121
Independent reference planes . . . . .	122
De-embedded Output Format . . . . .	125
De-embedding Codes . . . . .	126
Plotting Z <sub>0</sub> and E <sub>eff</sub> . . . . .	126
Calculating Z <sub>0</sub> and E <sub>eff</sub> for Boxwall Ports . . . . .	126
Lengths at Multiples of a Half-Wavelength . . . . .	129
Lengths Much Greater than One Wavelength. . . . .	129
Single Transmission Line . . . . .	130
Coupled Transmission Lines . . . . .	134
<b>7 COMPONENTS. . . . .</b>	<b>137</b>
Introduction . . . . .	137
Component Assistant . . . . .	138
Anatomy of a Component . . . . .	139
Component Types . . . . .	140
Data File . . . . .	140
Ideal Component . . . . .	141
Ports Only . . . . .	141
Sonnet Project . . . . .	142
User Model . . . . .	142
Component Properties . . . . .	143
Ground Node Connection. . . . .	143
Terminal Width . . . . .	145
Reference Planes. . . . .	148

# Sonnet User's Guide

---

Calibration Lengths . . . . .	148
Physical Size . . . . .	149
Rules for Using Components . . . . .	149
Analysis of a Component . . . . .	152
Data File Frequencies . . . . .	152
Rerunning an Analysis . . . . .	153
<b>8 ADAPTIVE BAND SYNTHESIS (ABS) . . . . .</b>	<b>155</b>
DC Point Extrapolation . . . . .	156
ABS Resolution . . . . .	156
Q-Factor Accuracy . . . . .	157
Enhanced Resonance Detection . . . . .	157
Running an Adaptive Sweep . . . . .	157
ABS Caching Level . . . . .	159
Multiple ABS Sweeps and Subsectioning . . . . .	159
Multi-Sweep Caching Scenarios . . . . .	160
Find Minimum and Find Maximum . . . . .	162
Parameter Sweep . . . . .	163
Analysis Issues . . . . .	165
Multiple Box Resonances . . . . .	165
De-embedding . . . . .	165
Transmission Line Parameters . . . . .	166
Current Density Data . . . . .	166
Ripple in ABS S-Parameters . . . . .	166
Output Files . . . . .	166
Viewing the Adaptive Response . . . . .	167
ABS Preview . . . . .	168
<b>9 PARAMETERIZING YOUR PROJECT . . . . .</b>	<b>171</b>
Variables . . . . .	172
How to Create a Variable . . . . .	173
Equations . . . . .	175
Dependent Variables . . . . .	176
Dimension Parameters . . . . .	177
Anchored Dimension Parameters . . . . .	179
Symmetrical Dimension Parameters . . . . .	180
Radial Dimension Parameters . . . . .	185
Reference Planes . . . . .	187

## Table of Contents

---

Dependent Dimension Parameters . . . . .	187
Circular Dependencies in Parameters . . . . .	188
Parameter Sweep . . . . .	189
Optimization . . . . .	192
<b>10 PARAMETER SWEEP AND OPTIMIZATION TUTORIAL . . . . .</b>	<b>197</b>
Setting Up Dimension Parameters . . . . .	198
Anchored Parameters . . . . .	198
Symmetric Parameters . . . . .	203
Parameter Sweep . . . . .	206
Setting Up a Parameter Sweep . . . . .	207
Executing the Parameter Sweep . . . . .	210
Observing the Parameter Sweep Data . . . . .	210
Optimization . . . . .	213
Entering New Nominal Values . . . . .	214
Setting Up an Optimization . . . . .	214
Running an Optimization . . . . .	219
Observing your Optimization Data . . . . .	220
Accepting the Optimized Values . . . . .	223
<b>11 CONFORMAL MESH . . . . .</b>	<b>227</b>
Introduction . . . . .	227
Use Conformal Meshing for Feed Lines, Not Patches . . . . .	229
Applying Conformal Meshing . . . . .	229
Conformal Meshing Rules . . . . .	231
Single Precision Option . . . . .	233
Using Conformal Meshing Effectively . . . . .	233
Use Conformal Meshing for Non-Manhattan Polygons . . . . .	233
Boundaries Should Be Vertical or Horizontal . . . . .	235
Cell Size and Processing Time . . . . .	235
Current Density Viewing . . . . .	236
<b>12 NETLIST PROJECT ANALYSIS . . . . .</b>	<b>239</b>
Networks . . . . .	240
Example of Node Assignments in a Complex Circuit . . . . .	241
Creating a Netlist . . . . .	244
Netlist Example Files . . . . .	244
Cascading S-, Y- and Z-Parameter Data Files . . . . .	245

A Network File with Geometry Project . . . . .	246
Inserting Modeled Elements into a Geometry . . . . .	249
Using Ungrounded-Internal Ports . . . . .	252
<b>13 CIRCUIT SUBDIVISION . . . . .</b>	<b>255</b>
Introduction . . . . .	255
Circuit Subdivision in Sonnet . . . . .	257
Choosing Subdivision Line Placement . . . . .	260
Good and Bad Placements of Subdivision Lines . . . . .	261
Subdivision Line Orientation . . . . .	265
Setting Up Circuit Properties . . . . .	267
Setting Up the Coarse Step Size Frequency Sweep . . . . .	268
Subdividing Your Circuit . . . . .	269
Analyzing Your Subdivided Circuit . . . . .	269
<b>14 CIRCUIT SUBDIVISION TUTORIAL . . . . .</b>	<b>271</b>
Obtaining the Example File . . . . .	272
Adding the Subdivision Lines . . . . .	272
Setting Up Circuit Properties . . . . .	275
Subdividing Your Circuit . . . . .	277
Analysis of the Network File . . . . .	280
Additional Improvements . . . . .	282
<b>15 VIAS . . . . .</b>	<b>285</b>
Introduction . . . . .	285
Creating Vias . . . . .	286
Via Direction . . . . .	287
Via Polygons . . . . .	288
Via Subsections . . . . .	289
Meshing Fill for Vias . . . . .	290
Adding a Rectangular Via to Ground . . . . .	296
Via Loss . . . . .	298
Via Pads . . . . .	299
Via Ports . . . . .	299
Auto Height Vias . . . . .	299
Edge Vias . . . . .	300
Edge Via Loss . . . . .	301
Simple Via Example . . . . .	302

---

## Table of Contents

---

<b>16 THICK METAL . . . . .</b>	<b>303</b>
Thick Metal Type . . . . .	303
Creating a Thick Metal Polygon . . . . .	305
Viewing Thick Metal in the Project Editor . . . . .	308
Stackup Manager . . . . .	309
Modeling . . . . .	310
Modeling Plated Metal . . . . .	311
Modeling Conductor-Via-Conductor Stackups . . . . .	311
Bar Via Configuration . . . . .	312
Via Array Configuration . . . . .	313
Modeling an Arbitrary Cross-Section . . . . .	314
Thick Metal in the Current Density Viewer . . . . .	315
<b>17 DIELECTRIC BRICKS . . . . .</b>	<b>317</b>
Applications of Dielectric Bricks . . . . .	319
Guidelines for Using Dielectric Bricks . . . . .	319
Subsectioning Dielectric Bricks . . . . .	319
Using Vias Inside a Dielectric Brick . . . . .	319
Air Dielectric Bricks . . . . .	320
Limitations of Dielectric Bricks . . . . .	320
Diagonal Fill . . . . .	320
Antennas and Radiation . . . . .	320
Interfaces . . . . .	320
Dielectric Brick Concepts . . . . .	321
Creating a Dielectric Brick . . . . .	321
Viewing Dielectric Bricks . . . . .	321
Defining Dielectric Brick Materials . . . . .	322
Changing Brick Materials . . . . .	324
Z-Partitioning . . . . .	325
<b>18 ANTENNAS AND RADIATION . . . . .</b>	<b>327</b>
Background . . . . .	328
Modeling Infinite Arrays . . . . .	328
Modeling an Open Environment . . . . .	329
Validation Example . . . . .	333
Far Field Viewer . . . . .	334
Analysis Limitations . . . . .	335

Spherical Coordinate System . . . . .	336
Normalization . . . . .	339
Polarization . . . . .	340
References . . . . .	340
<b>19 FAR FIELD VIEWER TUTORIAL . . . . .</b>	<b>341</b>
Creating an Antenna Pattern File . . . . .	342
Running the Far Field Viewer . . . . .	343
Calculating the Response . . . . .	344
Selecting Phi Values . . . . .	345
Selecting Frequencies . . . . .	345
Selecting the Response . . . . .	346
Zooming . . . . .	349
Probing the Plot . . . . .	350
Re-Normalizing the Plot . . . . .	351
Changing to a Polar Plot . . . . .	355
Turning Off the Legend . . . . .	355
Changing the Radius Axis . . . . .	356
Selecting a Frequency Plot . . . . .	358
Viewing a Surface Plot . . . . .	361
Saving the Far Field Viewer File . . . . .	361
Exiting the Far Field Viewer Program . . . . .	362
References . . . . .	362
<b>20 SPICE MODEL SYNTHESIS . . . . .</b>	<b>363</b>
PI Model . . . . .	364
Using The PI Model Spice Option . . . . .	364
A Simple Microwave Example . . . . .	368
Topology Used for PI Model Output . . . . .	368
N-Coupled Line Model . . . . .	369
Broadband SPICE Model . . . . .	371
Class of Problems . . . . .	372
Creating a Broadband Spice Model . . . . .	373
Checking the Accuracy of the Broadband Spice Model . . . . .	376
Improving the Accuracy of the Broadband Spice Model . . . . .	378
Broadband Spice Extractor Stability Factor . . . . .	379
Inductor Model Extractor . . . . .	380
Model Topologies . . . . .	380

## Table of Contents

---

Model Output . . . . .	383
Using the Inductor Model Extractor (IME) . . . . .	383
<b>21 BOX RESONANCES . . . . .</b>	<b>389</b>
Introduction . . . . .	389
Initial Indicators of a Box Resonance . . . . .	390
Bandpass Filter Example with Resonances . . . . .	391
Is It a Box Resonance? . . . . .	392
The Box Resonance Estimator . . . . .	392
Runtime Warning Messages . . . . .	394
Removing artificial box resonances . . . . .	396
Change box size . . . . .	397
Use a free space cover . . . . .	397
Suppressing Real Resonances . . . . .	399
Changing the Box Size . . . . .	399
Adding Vias . . . . .	399
Adding Absorbing Material . . . . .	400
Using a Probe Circuit for a Quick Analysis . . . . .	403
Viewing Electric Fields . . . . .	405
Summary . . . . .	406
<b>22 ACCURACY BENCHMARKING . . . . .</b>	<b>407</b>
An Exact Benchmark . . . . .	407
Residual Error Evaluation . . . . .	409
Using the Error Estimates . . . . .	411
<b>APPENDIX I EM AND XGEOM COMMAND LINE FOR BATCH . . . . .</b>	<b>413</b>
<i>em</i> Command Line . . . . .	413
Causal Dielectrics . . . . .	416
<i>xgeom</i> Command Line . . . . .	417
Example of <i>xgeom</i> Command Line . . . . .	419
<b>APPENDIX II SONNET REFERENCES . . . . .</b>	<b>421</b>
<b>INDEX . . . . .</b>	<b>429</b>

## Sonnet User's Guide

---

# Chapter 1                  Introduction

---

The Sonnet User’s Guide is intended to provide in depth discussions of features of Sonnet’s software. There is a short exposition of the theory behind Sonnet’s analysis engine, *em*, followed by discussions of geometry elements and features available in Sonnet. This manual also contains tutorials demonstrating how to use some features in Sonnet. The tutorials follow chapters discussing that topic. Please refer to the [Table of Contents](#) to see what tutorials are available.

For installation instructions and the basics of using Sonnet Suites, please refer to the **Getting Started** manual. To learn about new features in this release, please refer to [Chapter 2, “What’s New in Sonnet Suites Release 16”](#) on page 23.

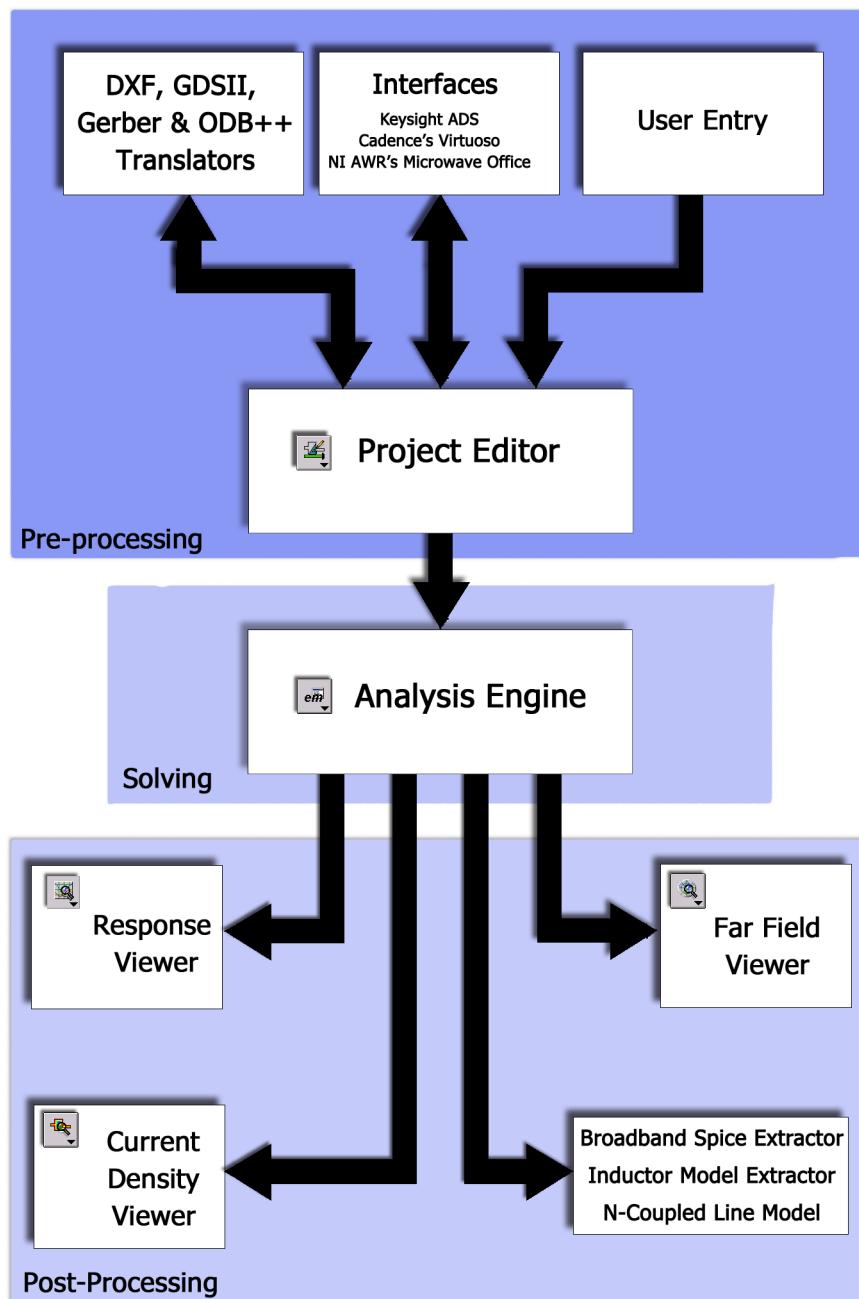
## The Sonnet Design Suite

The suite of Sonnet analysis tools is shown on page 19. Using these tools, Sonnet provides an open environment to many other design and layout programs. The following is a brief description of all of the Sonnet tools. Check with your system administrator to find out if you are licensed for these products:

Project Editor	The project editor is a user-friendly graphical interface that enables you to input your circuit geometry or circuit netlist for subsequent <b>em</b> analysis. If you have purchased the DXF, GDSII and/or the Gerber translator, the translator interface is found in the project editor. You also set up analysis controls for your project in the project editor.  Program module: <b>xgeom</b>
Analysis Engine	<b>Em</b> is the electromagnetic analysis engine. It uses a modified method of moments analysis based on Maxwell's equations to perform a true three dimensional current analysis of predominantly planar structures. <b>Em</b> computes S, Y, or Z-parameters, transmission line parameters ( $Z_0$ , Eeff, VSWR, GMax, Zin, and the Loss Factor), and SPICE equivalent lumped element networks. Additionally, it creates files for further processing by the current density viewer and the far field viewer. <b>Em</b> 's circuit netlist capability cascades the results of electromagnetic analyses with lumped elements, ideal transmission line elements and external S-parameter data.  Program module: <b>em</b>
Analysis Monitor	The analysis monitor allows you to observe the on-going status of analyses being run by <b>em</b> as well as creating and editing batch files to provide a queue for <b>em</b> jobs.  Program module: <b>emstatus</b>
Response Viewer	The response viewer is the plotting tool. This program allows you to plot your response data from <b>em</b> , as well as other simulation tools, as a Cartesian graph or a Smith chart. You may also plot the results of an equation. In addition, the response viewer may generate Spice lumped models.  Program module: <b>emgraph</b>
Current Density Viewer	The current density viewer is a visualization tool which acts as a post-processor to <b>em</b> providing you with an immediate qualitative view of the electromagnetic interactions occurring within your circuit. The currents may also be displayed in 3D.  Program module: <b>envu</b>

<b>Far Field Viewer</b>	The far field viewer is the radiation pattern computation and display program. It computes the far-field radiation pattern of radiating structures (such as patch antennas) using the current density information from <i>em</i> and displays the far-field radiation patterns in one of three formats: Cartesian plot, polar plot or surface plot.  Program module: <b><i>patvu</i></b>
<b>GDSII Translator</b>	The GDSII translator provides bidirectional translation of GDSII layout files to/from the Sonnet project editor geometry format.  Program module: <b><i>gds</i></b>
<b>DXF Translator</b>	The DXF translator provides bidirectional translation of DXF layout files (such as from AutoCAD) to/from the Sonnet project editor geometry format.  Program module: <b><i>dxfgeo</i></b>
<b>Gerber Translator</b>	The Gerber translator provides bidirectional translation of Gerber single layer and multi-layer files to/from the Sonnet project editor geometry format.
<b>Keysight ADS Interface</b>	The Keysight ADS Interface provides a seamless translation capability between Sonnet and Keysight's ADS. From within ADS Layout package you can directly create Sonnet geometry files. <i>Em</i> simulations can be invoked and the results incorporated into your design without leaving the ADS environment.  Program module: <b><i>ebridge</i></b>
<b>NI AWR Microwave Office Interface</b>	The NI AWR Microwave Office Interface provides a seamless incorporation of Sonnet's world class EM simulation engine, <i>em</i> , into the Microwave Office environment using Microwave Office's EM Socket. You can take advantage of Sonnet's accuracy without having to learn the Sonnet interface. Although, for advanced users who wish to take advantage of powerful advanced features not presently supported in the integrated environment, the partnership of AWR and Sonnet has simplified the process of moving EM projects between Microwave Office and Sonnet.  Program Module: <b><i>sonntawr</i></b>
<b>Cadence Virtuoso Interface</b>	This Sonnet plug-in for the Cadence Virtuoso suite enables the RFIC designer to configure and run the EM analysis from a layout cell, extract accurate electrical models, and create a schematic symbol for Analog Artist and RFDE simulation.  Program Module: <b><i>sonntcds</i></b>
<b>Broadband Spice Extractor</b>	A Broadband Spice extraction module is available that provides high-order Spice models. 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 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.



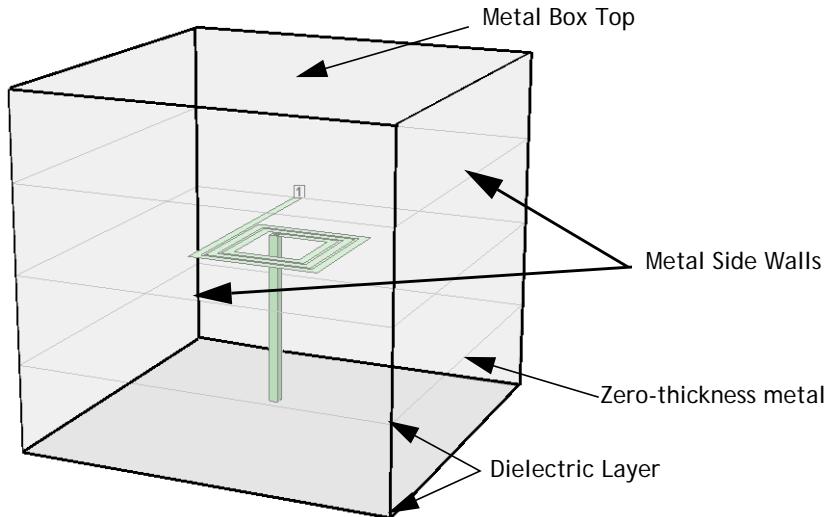
**Em** performs electromagnetic analysis [87, 88, 90] for arbitrary 3-D planar [62] (e.g., microstrip, coplanar, stripline, etc.) geometries, maintaining full accuracy at all frequencies. **Em** is a “full-wave” analysis in that it takes into account all possible coupling mechanisms. The analysis inherently includes dispersion, stray coupling, discontinuities, surface waves, moding, metalization loss, dielectric loss and radiation loss. In short, it is a complete electromagnetic analysis. Since **em** uses a surface meshing technique, i.e. it meshes only the surface of the circuit metallization, **em** can analyze predominately planar circuits much faster than volume meshing techniques.

## The Analysis Engine, **em**

**Em** analyzes 3-D structures embedded in planar multilayered dielectric on an underlying fixed grid. For this class of circuits, **em** can use the FFT (Fast Fourier Transform) analysis technique to efficiently calculate the electromagnetic coupling on and between each dielectric surface. This provides **em** with its several orders of magnitude of speed increase over volume meshing and other non-FFT based surface meshing techniques.

**Em** is a complete electromagnetic analysis; all electromagnetic effects, such as dispersion, loss, stray coupling, etc., are included. There are only two approximations used by **em**. First, the finite numerical precision inherent in digital computers. Second, **em** subdivides the metalization into small subsections made up of cells. A Simple Outline of the Theory

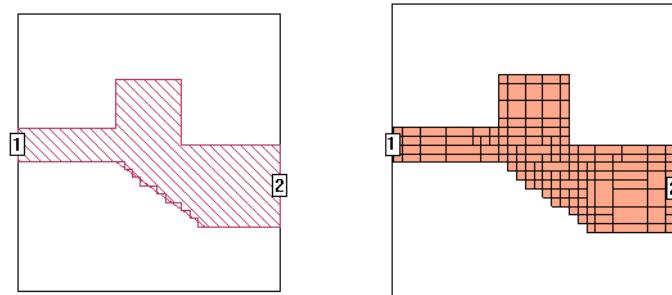
**Em** performs an electromagnetic analysis of a microstrip, stripline, coplanar waveguide, or any other 3-D planar circuit by solving for the current distribution on the circuit metalization using the Method of Moments. The metalization is modeled as zero-thickness metal between dielectric layers.



*Em* analyzes planar structures inside a shielding box. Port connections are usually made at the box sidewalls.

### Subsectioning the Circuit

**Em** evaluates the electric field everywhere due to the current in a single subsection. **Em** then repeats the calculation for every subsection in the circuit, one at a time. In so doing, *em* effectively calculates the “coupling” between each possible pair of subsections in the circuit.



The picture on the left shows the circuit as viewed in the project editor. On the right is shown the subsectioning used in analyzing the circuit.

### Zero Voltage Across a Conductor

Each subsection generates an electric field everywhere on the surface of the substrate, but we know that the total tangential electric field must be zero on the surface of any lossless conductor. This is the boundary condition: no voltage is allowed across a perfect conductor.

The problem is solved by assuming current on all subsections simultaneously. ***Em*** adjusts these currents so that the total tangential electric field, which is the sum of all the individual electric fields just calculated, goes to zero everywhere that there is a conductor. The currents that do this form the current distribution on the metallization. Once we have the currents, the S-parameters (or Y- or Z-) follow immediately.

If there is metalization loss, we modify the boundary condition. Rather than zero tangential electric field (zero voltage), we make the tangential electric field (the voltage on each subsection) proportional to the current in the subsection. Following Ohm's Law, the constant of proportionality is the metalization surface resistivity (in Ohms/square).

Sonnet is designed to work with your existing CAE software. Since the output data is in Touchstone or Compact format (at your discretion), ***em*** provides a seamless interface to your CAE tool.

## *Em* Origins

The technique used in ***em*** was developed at Syracuse University in 1986 by Rautio and Harrington [87, 88, 90]. It was originally developed as an extension of an analysis of planar waveguide probes [92]. The technique expresses the fields inside the box as a sum of waveguide modes and is thus closely related to the spectral domain approach.

The complete theory has been published in detail in peer reviewed journals. A full list of relevant papers is presented in [Appendix II "Sonnet References" on page 421](#).

## Chapter 2

## What's New in Sonnet Suites Release 16

---

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

[www.sonnetsoftware.com/release-notes](http://www.sonnetsoftware.com/release-notes)

**Custom User Model Components:** You may now create a component which allows you to add a model from a library that has been purchased, or developed in-house, to your geometry. For more information, please see ["User Model" on page 142](#).

**Modelithics Integration:** You may use the User Model Component to add a model from the Modelithics library to your circuit. For more information, please see ["User Model" on page 142](#).

**Project Components:** You may now create a component which represents another Sonnet project. For more information, please see ["Sonnet Project" on page 142](#).

**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.

**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 the Planar Metal Editor dialog box in 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 77 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 "[ABS Preview](#)" on page 168.

**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 "[DC Point Extrapolation](#)" on page 156 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 157 for more information.

**New Boolean Editing Operations:** There are four new commands for editing the metal in your geometry available in the project editor: Union, Intersect, Trim and Subtract. For more information see "Boolean operations" in the Index of Help.

**Robust Loss Metals:** Improved loss models for multiple metal layers stacked on top of each other. See "[Modeling Plated Metal](#)" on page 311 and "[Modeling Conductor-Via-Conductor Stackups](#)" on page 311.

**ODB++ Translator:** This release introduces an ODB++ translator allowing you to import an ODB++ file archive or directory and translating the contents to a Sonnet project. For more details, please see the "[Gerber/ODB++ Translator](#)," page 3.

**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 58.

**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 any Interface or Translator manual.

**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.

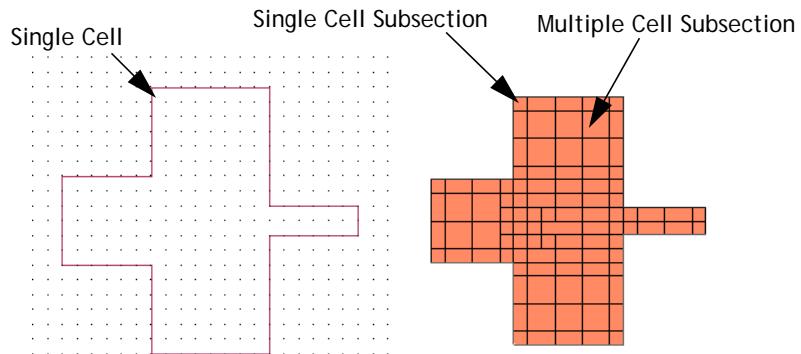
**Cadence Virtuoso Interface 64-bit Support:** There are a number of new features which have been incorporated into the Cadence Virtuoso Interface including support for 64-bit. Also new with this release, Help is available in all dialog boxes in the Cadence Virtuoso Interface. For a detailed list of new features in the Cadence Virtuoso Interface, please see [What's New in Sonnet Suite's Cadence Virtuoso Interface](#).

**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 17](#) of the [Keysight ADS Interface](#) manual.



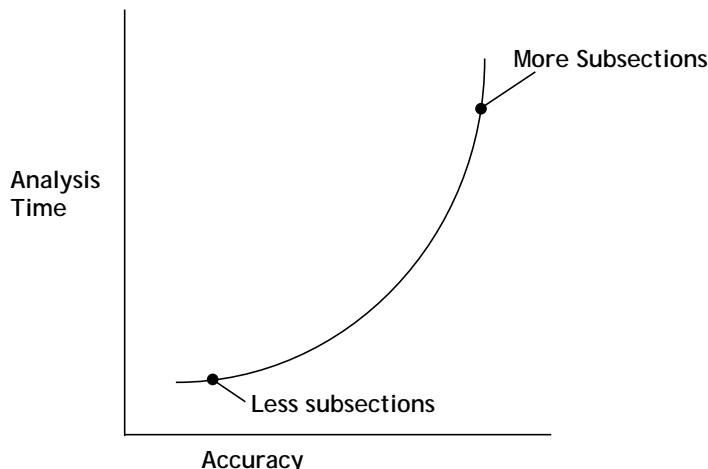
# Chapter 3 Subsectioning

The Sonnet subsectioning is based on a uniform grid indicated by the small dots in the project editor window. The small dots are placed at the corners of a “cell”. 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.



A metal polygon with the cell grid is shown on the left. The same metal polygon's subsectioning is shown on the right. Note that some subsections are comprised of only one cell, while other subsections are made up of multiple cells.

The cell size is specified in the project editor in the Box Settings dialog box which is opened by selecting *Circuit*  $\Rightarrow$  *Box*. Since multiple cells are combined together into a single subsection, the number of subsections is usually considerably smaller than the number of cells. This is important because the analysis solves an  $N \times N$  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. All EM simulators have an inherent trade off between speed and accuracy. Decreasing the subsection size typically increases the accuracy but also causes an increase in analysis time. The figure below shows the typical trade off between analysis time and accuracy.



**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. However, in some cases you may wish to modify the automatic algorithm because you want a faster, less accurate solution, or a slower, more accurate solution, than is provided by the automatic algorithm.

This chapter explains how **em** combines cells into subsections and how you can control this process to obtain an analysis time or the level of accuracy you require. There is also a discussion of selecting the cell size and how that may affect the **em** analysis.



### TIP

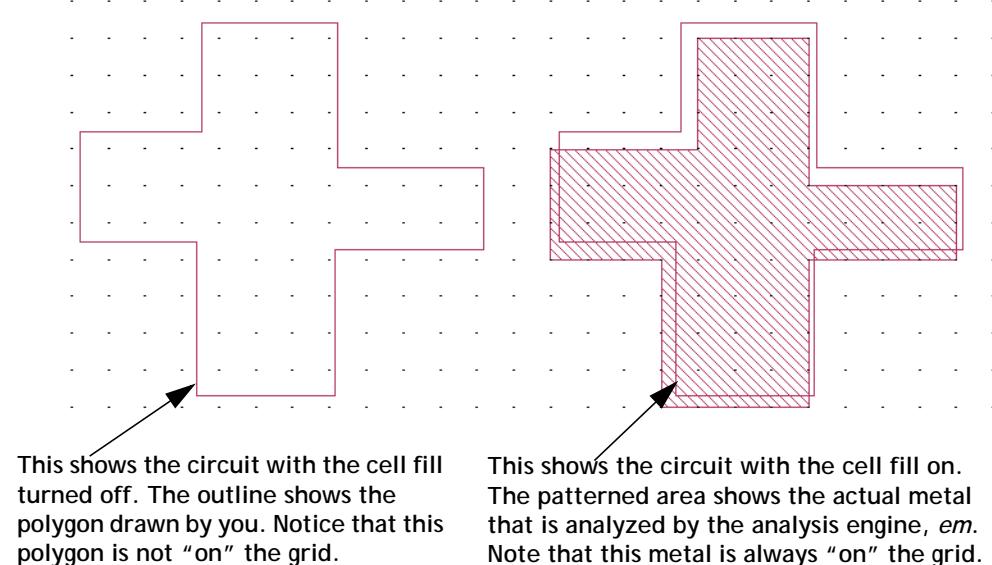
---

For a discussion of efficient subsectioning practices with application examples, please refer to the app note “Efficient Meshing in Sonnet” by Dr. Volker Mühlhaus. This app note is available in help by looking under “subsectioning” in the index.

---

## Metal Fill

Metal polygons are represented with an outline and a cell fill pattern. The outline represents exactly what you entered or imported. The cell fill represents the actual metalization analyzed by **em**. Therefore, the actual metalization analyzed by **em** may differ from that input by you as illustrated below. You turn the cell fill on and off using the control-M key.



## Tips for Selecting A Good Cell Size

As discussed above, **em** subdivides the circuit into subsections which are made up of "cells," the building block in the project editor. The following discussion describes how to select a cell size. To access the Cell Size calculator, click on the Cell Size Calculator button in the Box Settings dialog box, which is invoked when you select *Circuit*  $\Rightarrow$  *Box* from the project editor menu.



### TIP

Select a cell size that is smaller than 1/20 of a wavelength.

Before calculating a cell size, it is important to calculate the wavelength at your highest frequency of analysis. An exact number is not important. If you know the approximate effective dielectric constant of your circuit, use this in the wavelength calculation; otherwise, use the highest dielectric constant in your structure.

Most circuits require that your cell size be smaller than 1/20 of a wavelength. Larger cell sizes usually result in unacceptable errors due to incorrect modeling of the distributed effects across the circuit.



### TIP

---

When possible, find a common factor for your most critical dimensions.

---

Since your circuit geometry is snapped to the nearest cell, you should set your cell size to be a common factor of your most critical dimensions. For example, if your circuit has dimensions of 30 microns, 40 microns and 60 microns, possible cell sizes are 10 microns, 5 microns, 2.5 microns, 2 microns, etc. Large cell sizes result in more efficient analyses, so choosing 10 microns is probably best.

Sonnet provides a cell size calculator utility which helps you find a good common factor for your critical dimensions. Using the Cell Size Calculator is detailed in [Chapter 6, “Spiral Inductor”](#) of the **Getting Started** manual.



### TIP

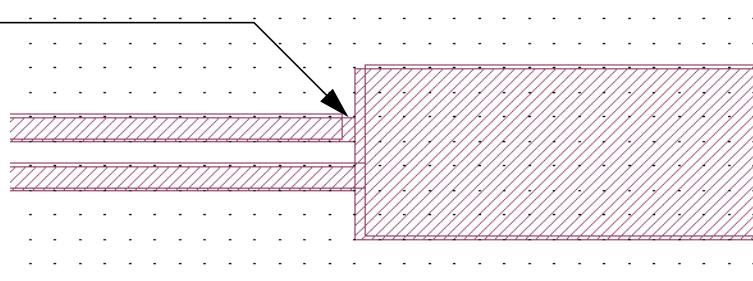
---

After setting your initial cell size, visually inspect the metal fill (indicated by the “hatched” pattern) for potential problems.

---

Once a common factor is found, some of your less critical dimensions may not be a multiple of this factor. You should inspect your circuit for potential problems caused by polygons which are not exact multiples of this cell size. An example is shown below.

Notice that there is a gap between the two polygons as drawn by the user, but the actual on-grid metal “shorts” the two polygons together.



**TIP**

---

Calculate the X cell size and the Y cell size independently.

---

The X cell size and Y cell size do not have to be the same number. Calculate the X cell size based on just your dimensions in the X direction, and your Y cell size based on just your dimensions in the Y direction.

## Viewing the Subsections

You can see the subsections used by Sonnet by following the instructions below. Be aware that your dielectric layers and analysis frequencies must be defined and at least one port must be added to your circuit before you may use the Estimate Memory command.

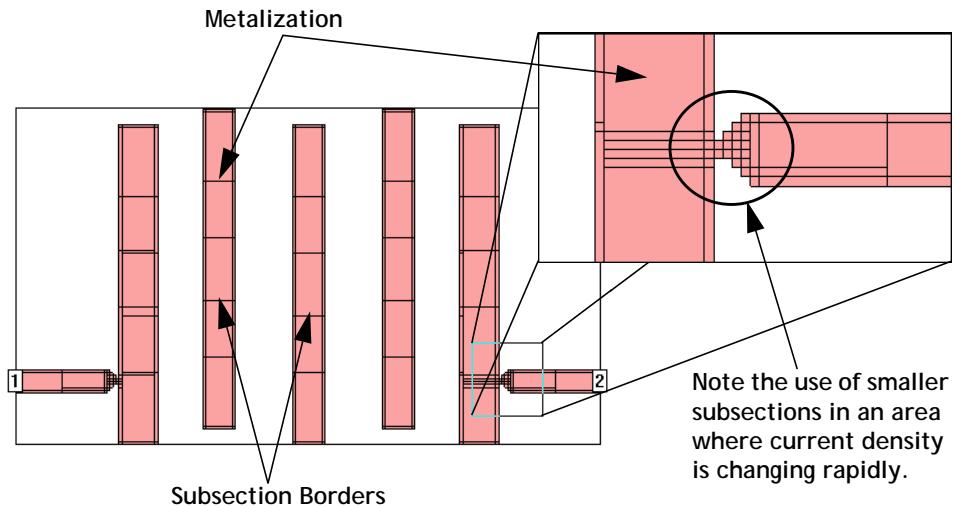
To view the subsectioning, do the following:

- 1 **From the project editor, select Analysis => Estimate Memory.**

This calculates the amount of memory required for your analysis.

### 2 Click the View Subsections button.

A picture of your circuit will appear. The metal will show up as red, and the subsection borders will show up as black lines as shown in the illustration below:



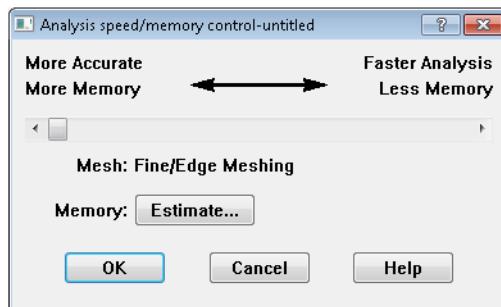
## Using the Speed/Memory Control

The Speed/Memory Control allows you to control the memory usage for an analysis by controlling the subsectioning of your circuit. The high memory settings produce a more accurate answer and usually increase processing time. Conversely, low memory settings run faster but do not yield as accurate an answer.

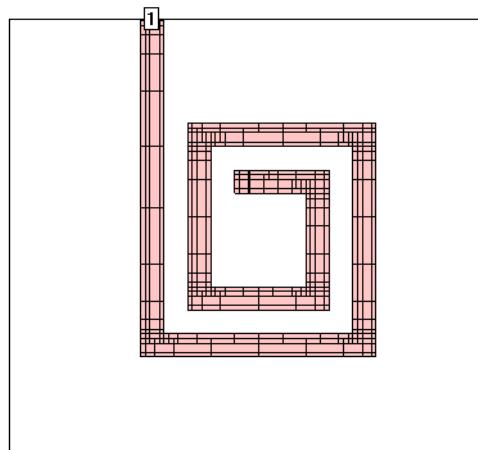
To access the Speed/Memory Control, follow the instructions below.

### 1 Select *Analysis* $\Rightarrow$ *Setup* from the project editor main menu.

- 2 In the Analysis Setup dialog box which appears, click on the Speed/Memory button.
- 3 In the Analysis Speed/Memory Control dialog box which appears, select the desired setting.

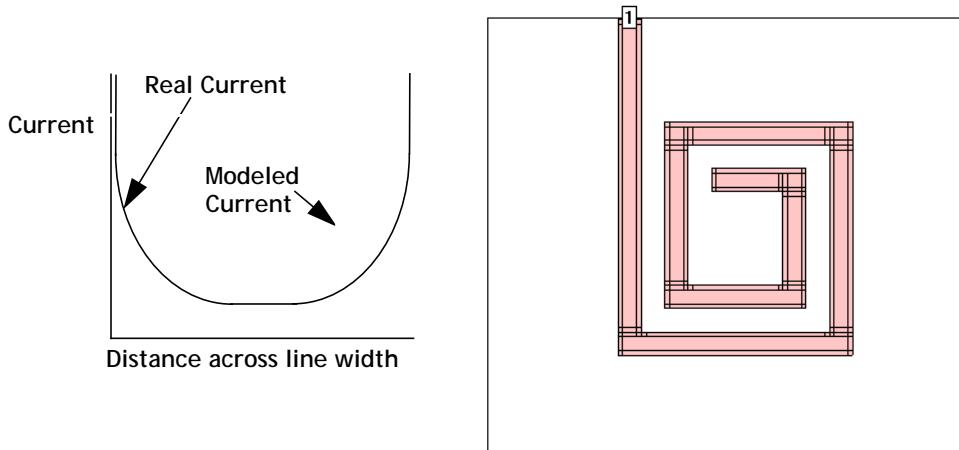


There are three settings for the Speed/Memory Control: Fine/Edge Meshing, Coarse/Edge Meshing, Coarse/No Edge Meshing. Fine/Edge Meshing is the default setting and is described in “Default Subsectioning of a Polygon,” page 35. An example is shown below. Again, note that this setting provides the most accurate answer but demands the highest memory and processing time.

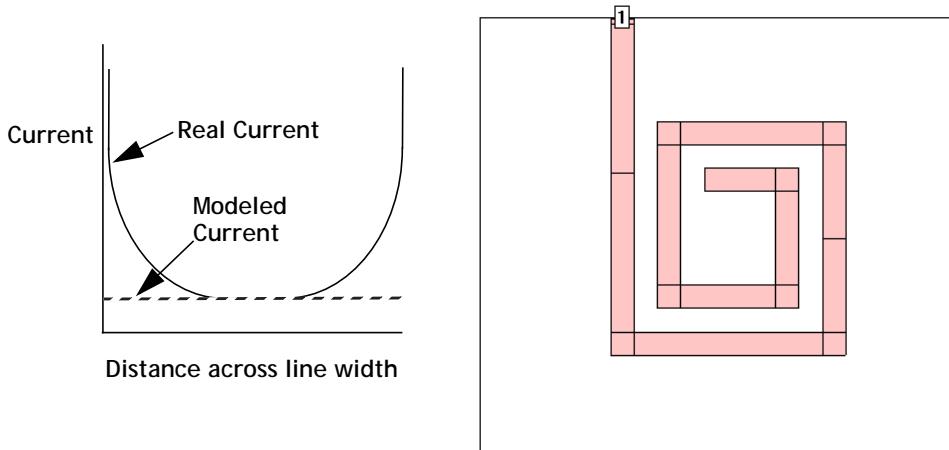


The second option is Coarse/Edge Meshing. This setting is often a good compromise between speed and accuracy. Shown below is a typical circuit with this setting. Notice the edges of the polygons have small subsections, but the inner

portions of the polygons have very large subsections. This is a good compromise between speed and accuracy because the currents on most RF traces change most rapidly on the metal edge.



The last option is Coarse/No Edge Meshing. For this setting, all polygons are set to use large subsections and edge mesh is set to “off.” This yields the fastest analysis, but is also the least accurate because the current across the width of the line is constant. Shown below is the subsectioning of a typical circuit using this option.



# Changing the Subsectioning of a Polygon

Sonnet allows you to control how cells are combined into subsections for each polygon. This is done using the parameters “X Min”, “Y Min”, “X Max” and “Y Max”. These parameters may be changed for each polygon, allowing you to have coarser resolution for some polygons and finer resolution for others. You use the *Modify ⇒ Metal Properties* command to change these setting.

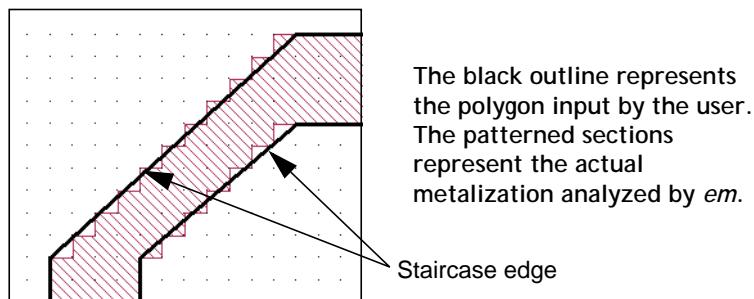
Please note that these controls are not used if conformal meshing is applied to the metal polygon. There are separate controls when conformal mesh is applied to a polygon. For more information about conformal meshing subsection control, please refer to “Conformal Mesh Subsectioning Control,” page 45.

Before discussing how to make use of these parameters, we need to first understand *em*’s automatic subsectioning for a polygon when the parameters are set to their default settings.

## Default Subsectioning of a Polygon

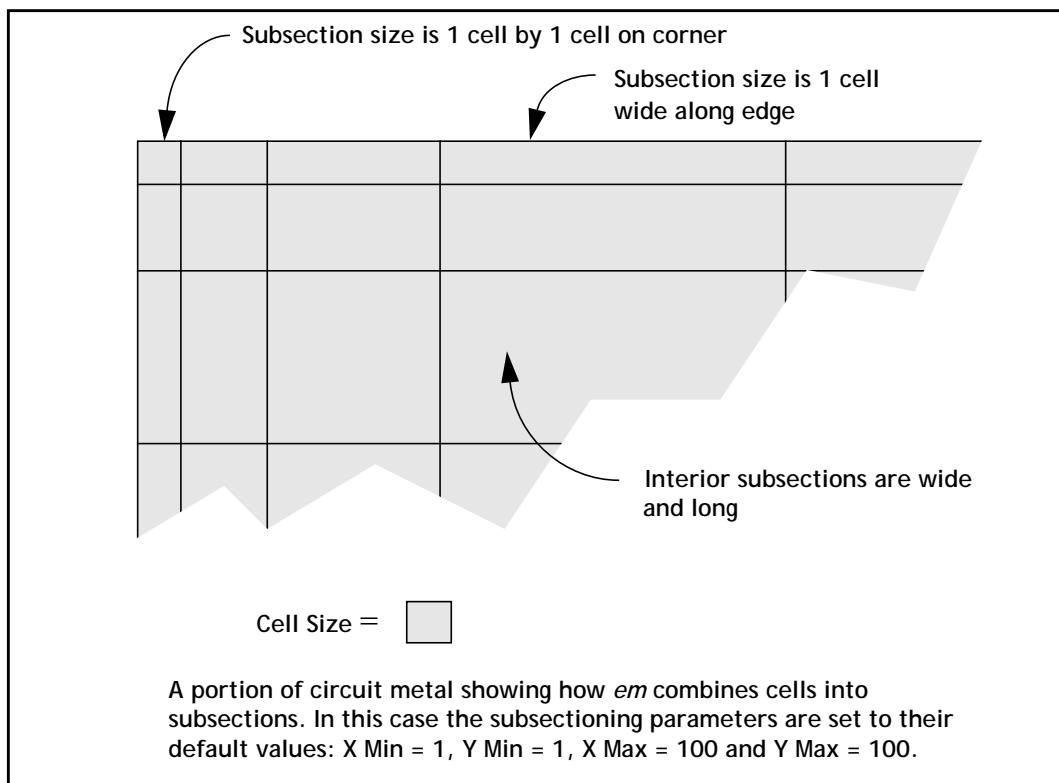
By default, Sonnet fills a polygon with “staircase” subsections. Other, more advanced fill types (diagonal and conformal) can be used. For diagonal subsections, see “Diagonal Fill Subsectioning” on page 42. For conformal mesh, see “Conformal Mesh Subsectioning” on page 43. This discussion applies to staircase fill.

This fill type is referred to as staircase because when using small rectangular subsections to approximate a diagonal edge, the actual metalization takes on the appearance of a staircase, as in the example shown below.



The default values for the subsectioning parameters are X Min = 1, Y Min = 1, X Max = 100 and Y Max = 100. These numbers specify the smallest and largest allowed dimensions of the subsections in a polygon. With X Min = 1, the smallest subsection in the X dimension is one cell. With X Max = 100, subsections are not allowed to go over 100 cells in length.

The illustration below shows how these default subsectioning parameters are used. Notice in the corner, the subsection size is just one cell. The current density changes most rapidly here, thus, the smallest possible subsection size is used.



As we go away from the corner, along the edge, the subsections become longer. For example, the next subsection is two cells long, the next one is four cells long, etc. If the edge is long enough, the subsection length increases until it reaches X Max (100) cells, or the maximum subsection size parameter, whichever comes first, and then remains at that length until it gets close to another corner, discontinuity, etc.

Notice, however, that no matter how long the edge subsection is, it is always one cell wide. This is because the current density changes very rapidly as we move from the edge toward the interior of the metal (this is called the edge singularity).

In order to allow an accurate representation of the very high edge current, the edge subsections are allowed to be only one cell wide. (See the current plots on page 34.) However, the current density becomes smooth as we approach the interior of the metal. Thus, wider subsections are allowed there. As before, the width goes from one cell to two cells, then four, etc.



### TIP

---

If two polygons butt up against each other or have a small overlap, the modeling of the edge singularity will require a larger number of subsections at the boundary between the two polygons. Using the Union command (*Edit*  $\Rightarrow$  *Boolean Operations*  $\Rightarrow$  *Union*) to join the two polygons into one will reduce the number of required subsections and speed up your analysis.

Conversely, if you have an area of your circuit at which you desire greater accuracy, using the Divide (*Edit*  $\Rightarrow$  *Divide Polygons*) command at the point of interest to create two polygons, forces the analysis to use smaller subsections in order to model the edge singularities.

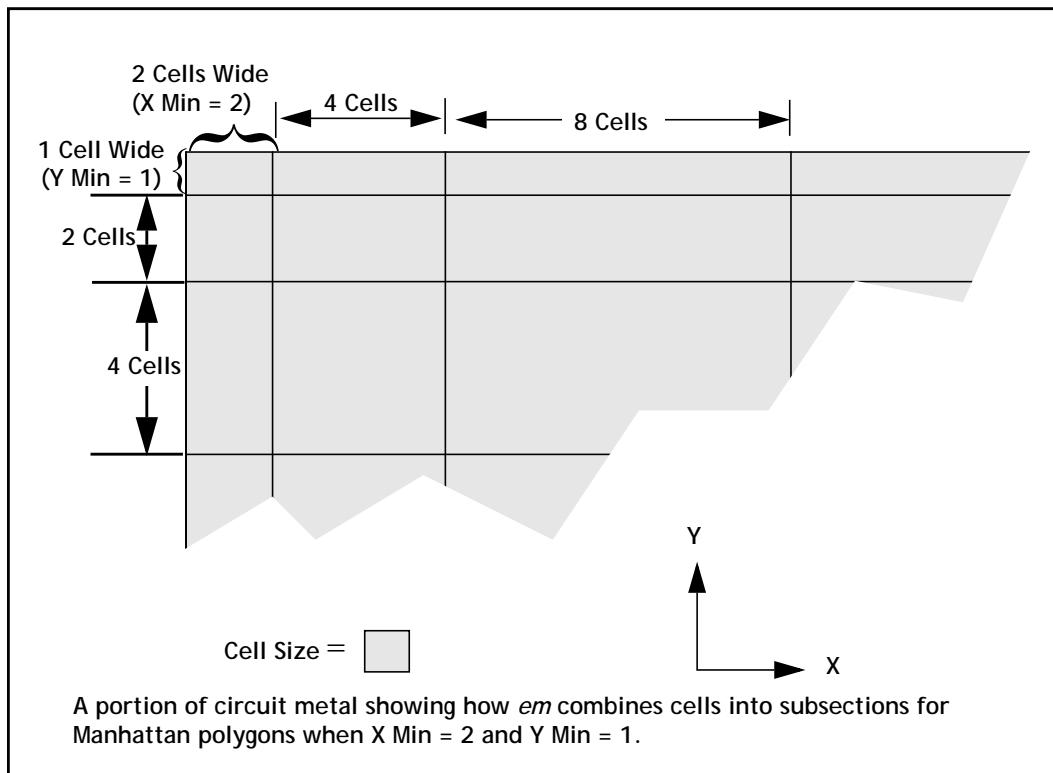
---

## X Min and Y Min with Edge Mesh Off

On occasion, you may wish to change the default subsectioning for a given polygon. You can do this using the subsectioning parameters X Min, Y Min, X Max and Y Max.

Having the edge mesh option “on” is the default state for Sonnet projects; however, examining the case where edge mesh is “off” first makes understanding the concept easier. This part of the discussion only applies to Manhattan polygons, which is a polygon that has no diagonal edges. Turning edge mesh off for non-manhattan polygons has no effect.

For Manhattan polygons with edge mesh off, X Min and Y Min set the size of the edge subsections. By default, X Min and Y Min are 1. This means the edge subsections are 1 cell wide. When X Min is set to 2, the subsections along vertical edges are now 2 cells wide in the X direction (see the figure on page 38). This reduces the number of subsections and reduces the matrix size for a faster analysis. However, accuracy may also be reduced due to the coarser modeling of the current density near the structure edge or a discontinuity.



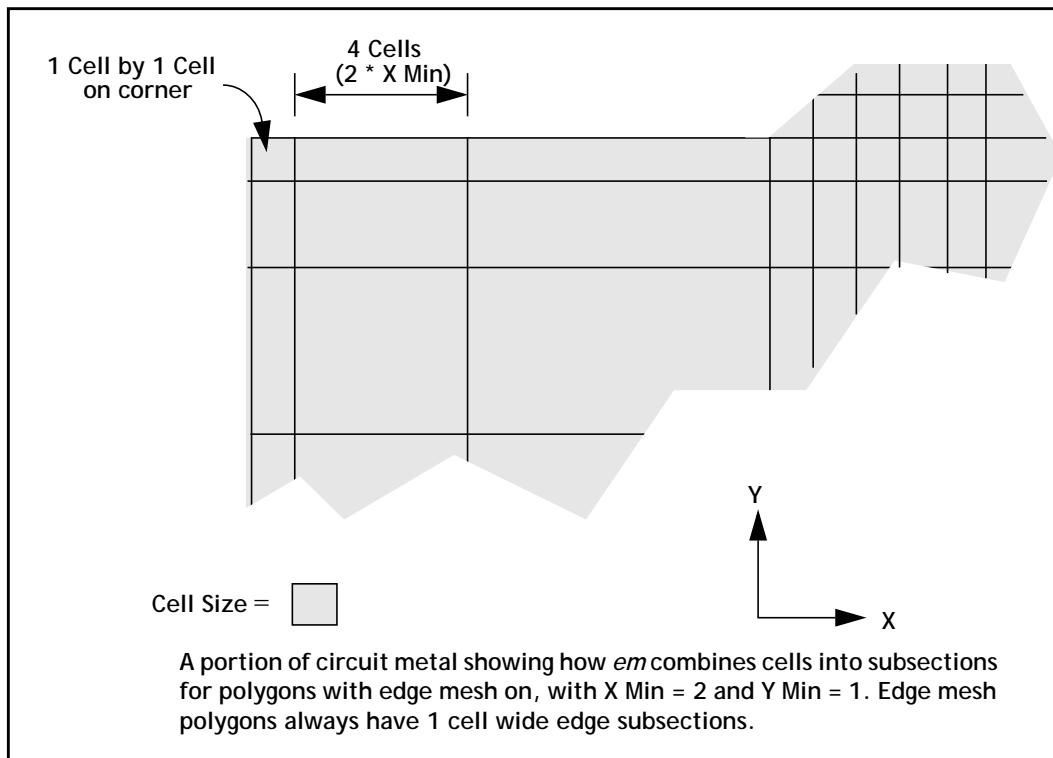
If X min or Y min are greater than your polygon size, *em* uses subsections as large as possible to fill the polygon.

**NOTE:** The subsection parameters, X Min, Y Min, X Max and Y Max are specified in *cells* (not mils, mm, microns, etc.). For example, X Min = 5 means that the minimum subsection size is 5 cells.

Although the X Min and Y Min parameters are very useful options, it is not a substitute for using a larger cell size. For example, a circuit with a cell size of 10 microns by 10 microns with X Min = 1 and Y Min = 1 runs faster than the same circuit with a cell size of 5 microns by 5 microns with X Min = 2 and Y Min = 2. Even though the total number of subsections for each circuit may be the same, *em* must spend extra time calculating the value for each subsection for the circuit with the smaller cell size.

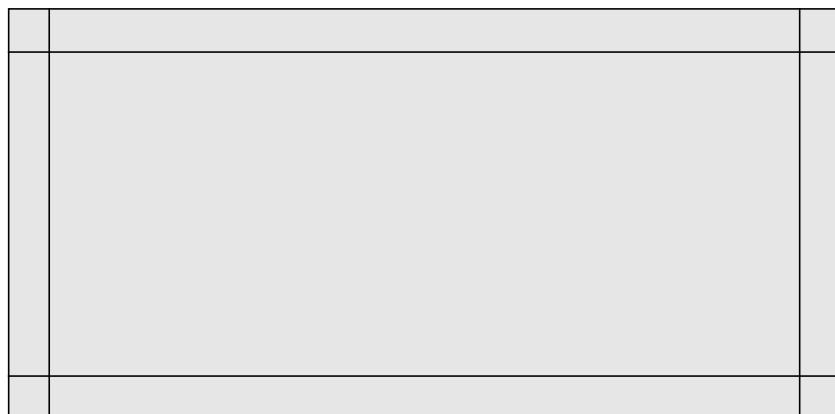
## X Min and Y Min with Edge Mesh On

Having the edge mesh option “on” is the default state for Sonnet projects because it provides a more accurate analysis. Having edge mesh “on” for a polygon changes how the subsections on the very edge are handled. Starting from the left side of the previous example with edge mesh off, the subsections were 2 cells, 4 cells and 8 cells wide. With edge mesh on, the subsections for the same polygon would be 1 cell, 4 cells, and 8 cells as shown in the illustration below. Notice only the outermost edge is affected.



As mentioned in the previous section, the edge mesh setting only affects Manhattan polygons (i.e. those with no diagonal or curved edges). Edge mesh is always “on” for non-Manhattan polygons, regardless of the edge-mesh setting for that polygon.

When used in conjunction with large X Min or Y Min values, the edge mesh option can be very useful in reducing the number of subsections but still maintaining the edge singularity, as shown in a simple example below. This is very often a good compromise between accuracy and speed.



In the case pictured above, X Min and Y Min are set to be very large, and the frequency is low enough so that the Max. Subsection size parameter corresponds to a subsection size that is larger than the polygon. This is exactly what is being done when you set the Speed/Memory setting to "Coarse/Edge Meshing."

### Using X Max and Y Max

You may control the maximum subsection size of individual polygons by using the X Max and Y Max parameters. For example, if X Max and Y Max are decreased to 1, then all subsections will be one cell. This results in a much larger number of subsections and a very large matrix which are the cause of increased analysis time. Thus, this should be done only on small circuits where extremely high accuracy is required or you need a really smooth current density plot.

---

**NOTE:**

If the maximum subsection size specified by X Max or Y Max is larger than the size calculated by the Max. Subsection Size parameter, the Max. Subsection Size parameter takes priority. The Max. Subsection Size is discussed in "Setting the Maximum Subsection Size Parameter" section which follows.

---

## Setting the Maximum Subsection Size Parameter

The parameter Max. Subsection Size allows the specification of a maximum subsection size, in terms of subsections per wavelength, where the wavelength is approximated at the beginning of the analysis. By default, the highest analysis frequency is used in the calculation of the wavelength. This value is a global setting and is applied to the subsectioning of all polygons in your circuit.

The default of 20 subsections/ $\lambda$  is fine for most work. This means that the maximum size of a subsection is 18 degrees at the highest frequency of analysis. Increasing this number decreases the maximum subsection size until the limit of 1 subsection = 1 cell is reached.

You might want to increase this parameter for a more accurate solution. For example, changing it from 20 to 40 decreases the size of the largest subsections by a factor of 2, resulting in a more accurate (but slower) solution. Keep in mind that using smaller subsections in non-critical areas may have very little effect on the accuracy of the analysis while increasing analysis time.

Another reason for using this parameter is when you require extremely smooth current distributions for the current density viewer. With the default value of 20, large interior subsections may make the current distribution look “choppy.” Setting this value to a large number forces all subsections to be only 1 cell wide, providing smooth current distribution. Again, analysis time is impacted significantly.

The Max. Subsection Size parameter is specified in the Advanced Subsectioning Controls which are opened by selecting *Analysis*  $\Rightarrow$  *Advanced Subsectioning* from the project editor main menu.

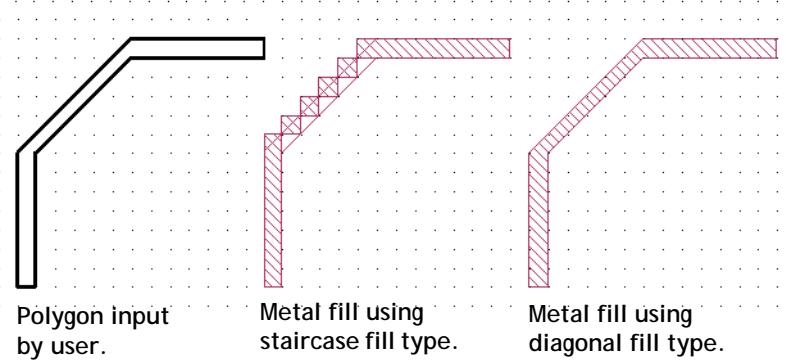
## Defining the Subsectioning Frequency

The subsectioning parameter, Max. Subsection Size, applies to the subsectioning of all polygons in your circuit, and is defined as subsections per wavelength. Normally, the highest analysis frequency is used to determine the wavelength. However, this may be changed by using the Subsectioning Frequency options in the Advanced Subsectioning Control dialog box in the project editor. This dialog box is opened by selecting *Analysis*  $\Rightarrow$  *Advanced Subsectioning* from the project editor main window. For details on what options are available to define the subsectioning frequency, click on the Help button in the Advanced Subsectioning Control dialog box.

The frequency defined by the selected option is now used to determine the maximum subsection size instead of the highest frequency of analysis. Thus, the same subsectioning can be used for several analyses which differ in the highest frequency being analyzed.

## Diagonal Fill Subsectioning

Diagonal fill subsectioning is a technique which allows you to more accurately model straight diagonal edges by using triangular subsection as well as rectangular. When modeling a circuit with one to two cell wide traces which contain a diagonal miter corner or a diagonal line, the metal fill using a staircase fill type sometimes causes the actual metal being analyzed to have opens which do not allow current to flow (see the graphic below). In these cases, applying the diagonal fill type to a polygon creates a metal pattern which more accurately models the polygon input by the user.

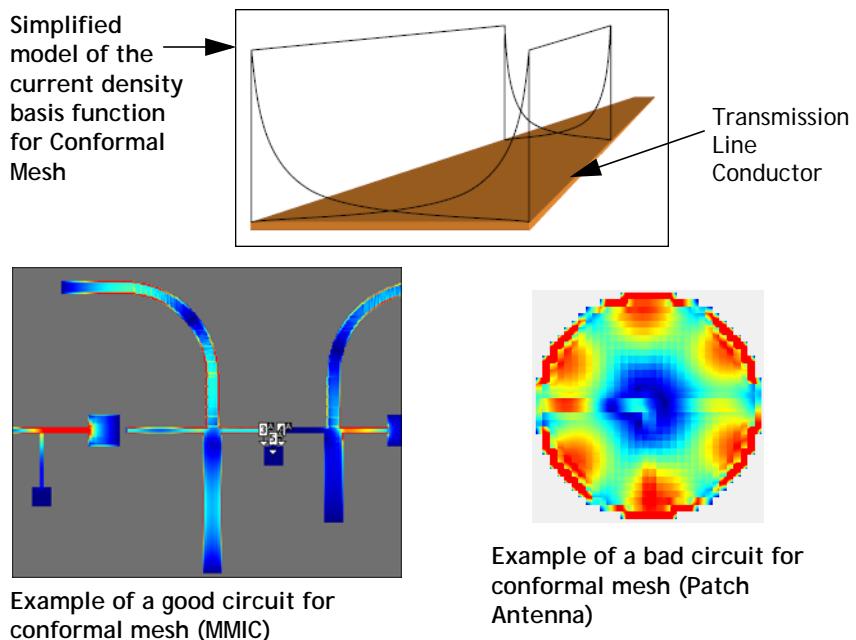


The first polygon on the left shows the polygon input by the user. The middle polygon shows the metal fill pattern when the staircase fill type is used for the polygon. Note that the metal fill is a series of polygons which only share a corner hence not allowing any current to flow. The third polygon on the right shows the metal fill pattern using the diagonal fill type. As you can see, the metal fill now more closely matches the polygon input by the user and allows the current to flow.

Note that while diagonal fill yields more accurate answers, it increase processing time and should only be used when necessary to correctly model your circuit.

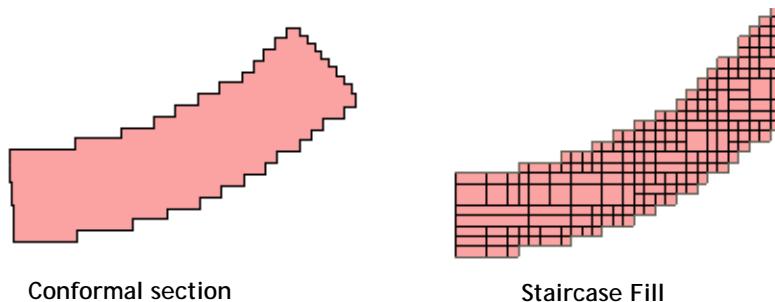
## Conformal Mesh Subsectioning

Conformal meshing is a technique which can dramatically reduce the memory and time required for analysis of a circuit with diagonal or curved polygon edges. For a detailed discussion of conformal mesh and its rules of use, please refer to “Conformal Mesh,” page 227. Only the effect of conformal mesh on subsectioning is discussed in this chapter. Conformal mesh assumes that most current flows on the edges of circuit conductors, as shown below, and should only be applied when this holds true.



This technique groups together strings of cells following diagonal and curved metal contours to form long subsections along those contours. Whereas staircase fill results in numerous small X- and Y-directed subsections, conformal mesh re-

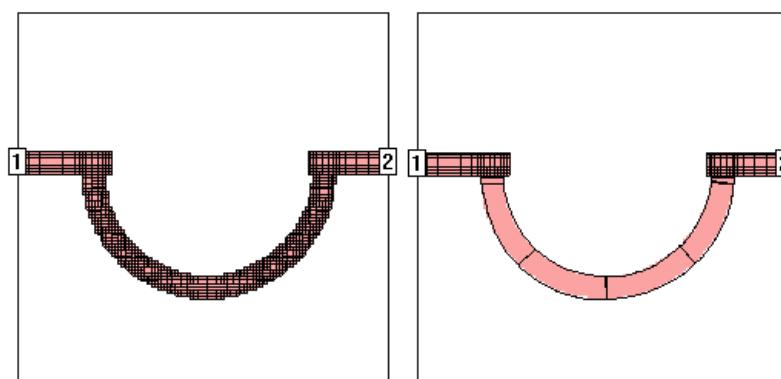
sults in a few long conformal subsections. The illustration below shows the actual metalization of a conformal section in close up alongside the same section using staircase fill.



Conformal sections, like standard subsections, are comprised of cells, so that the actual metalization still shows a "jagged" edge when the polygon has a smooth edge. However, the sections can be much larger due to conformal meshing. These larger sections yield faster processing times with lower memory requirements for your analysis.

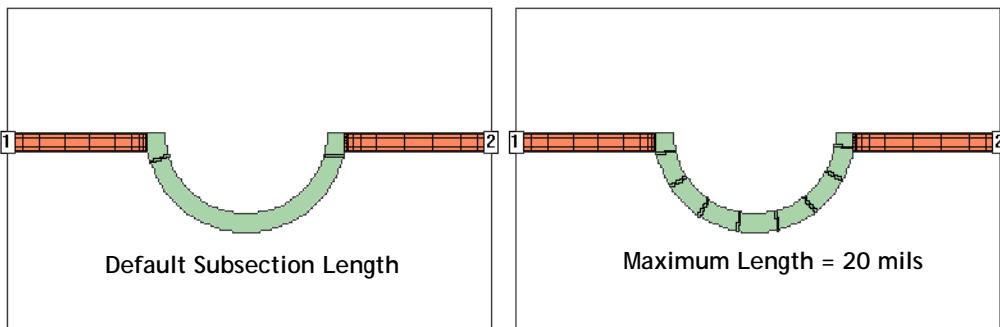
Standard subsectioning requires a lot of subsections to model the correct current distribution across the width of the line. Conformal subsections have this distribution built into the subsection. Sonnet conformal meshing automatically includes the high edge current in each conformal section. This patented Sonnet capability is unique. (U.S. Patent No. 6,163,762 issued December 19, 2000.)

An example of a circuit using both standard subsectioning and conformal mesh is shown below. The circuit shown at the left is displayed using standard subsectioning (staircase fill). Conformal meshing for the curved part of the circuit is shown on the right. Note that for the curved part of the geometry, conformal mesh uses substantially fewer subsections than the number used in the standard subsectioning.



## Conformal Mesh Subsectioning Control

When you apply conformal mesh to a polygon, it is possible to limit the maximum length of a conformal section in order to provide a more accurate simulation. The default length of a conformal section is 1/20 of the wavelength at the subsectioning frequency. For more information on the subsectioning frequency, see “Defining the Subsectioning Frequency,” page 41.



Setting the maximum length to 20 mils limits the length of the subsections. As you can see pictured above, the default subsection length uses one subsection for most of the curve. When the maximum length is set to 20 mils, multiple subsections are used to model the same metal providing a more accurate answer.

To set the maximum length for a conformal section, do the following:

- 1 Select the desired polygon(s).**

The selected polygons are highlighted.

- 2 Select *Modify*  $\Rightarrow$  *Metal Properties***

This opens the Metalization Properties dialog box.

- 3 Click on the Maximum Length checkbox in the Conformal Mesh Subsectioning Controls section of the dialog box.**

This will enable the Length text entry box to the right. Note that this checkbox is only enabled when Conformal is chosen as the Fill Type.

- 4 Enter the desired Maximum Length in the text entry box.**

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

For a more detailed discussion of Conformal Mesh, please refer to **Chapter 11, “Conformal Mesh”** on page 227. There is also an application note on conformal mesh available in Help; look up conformal mesh in the index.



# Chapter 4

# Metalization and Dielectric Layer Loss

---

This chapter is composed of three parts: planar metalization loss, via metalization loss and dielectric layer loss. For information on dielectric brick loss, see **Chapter 17, “Dielectric Bricks”** on page 317. Both the theoretical aspect of how Sonnet models loss and the practical how to’s of assigning loss in your circuit are covered, including the use of metal and dielectric material libraries. The discussion of metalization loss begins below. For the discussion of dielectric loss, see “Dielectric Layer Loss,” page 66.

There is also a paper available by the president and founder of Sonnet Software, Dr. James Rautio which contains a detailed discussion of metal losses; see reference [[15](#)] in Appendix II, “Sonnet References” for a detailed description. This paper is also available on the Sonnet [website](#).

There are two general metal types in Sonnet, planar metal types which are assigned to metal polygons, and via metal types which are assigned to via polygons. In metal polygons, the current flow is in the x and y direction, while in via polygons, the current flows in the z direction between metal levels. The underlying theory behind both types is the same, but the parameters you enter are based on the physical properties of the object for which you will use the metal. For example, a planar metal can be defined with a conductivity and a thickness, while a via metal can be defined with a conductivity and solid cross-sectional area. Planar metals are discussed first followed by via metals.

## Planar Metalization Loss

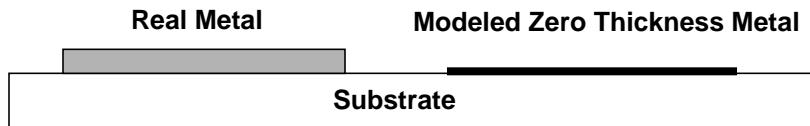
Metalization loss is specified in the project editor in the Metal Types dialog box which is opened by selecting *Circuit*  $\Rightarrow$  *Metal Types*. You click on the Add Planar button to add a planar metal. Planar metal types may be assigned to circuit metal, top cover and ground plane. Sidewalls are always assumed to be perfect conductors.

A common misconception is that only one type of planar metalization is allowed on any given level. In fact, different metalizations (i.e., different losses) can be mixed together on any and all levels. For example, it is possible to have a thin film resistor next to a gold trace on the same level.

Sonnet allows you to use pre-defined planar metals, such as gold and copper, using the global library. The global library allows you to define your own metal types as well. There is also a local metal library which can be created for an individual or to share between users. There are both planar and via metals available in the metal libraries.

### Sonnet's Loss Model for Planar Metalization

The Sonnet model of planar metal loss uses the concept of surface impedance, measured in Ohms per square. This concept allows planar EM Simulators, such as Sonnet *em*, to model real 3-dimensional metal in two dimensions.



If you are unfamiliar with this concept, please refer to any classic textbook such as **Fields and Waves in Communication Electronics** by Simon Ramo, John R. Whinnery and Theodore Van Duzer, John Wiley & Sons, New York, 1965.

It is important to note that this technique models the loss of the true 3-dimensional metal fairly accurately, but does not model any change in field distribution due to the metal thickness. This approximation is valid if the metal thickness is small with respect to the width of the line, the separation between lines, and the thickness of the dielectric. If the true 3-dimensional affect of the metal is important, then you should consider using the Thick Metal Model metal type as discussed in **Chapter 16, “Thick Metal”** on page 303.

Some electromagnetic analyses use a “perturbational” approach for loss. This means that they assume the current flowing everywhere is the same as the lossless case. This approximation works for low loss metals (good conductors) at most RF frequencies. However for thin film resistors and good conductors at low frequencies, the lossless current is not the same as the lossy current and a perturbational approach fails. *Em*'s loss analysis is not perturbational. It works just as well for a 100 Ohms/square resistor as it does for a 0.004 Ohms/square good conductor. The Sonnet loss analysis also properly models the transition between electrically thin (low frequency) and electrically thick (high frequency) conductors. See reference [26] in reference Appendix II listed on page 421 for a detailed description of the theory used by Sonnet. See reference [93] listed on page 428 for the equations actually used in the Sonnet model.

Another aspect of loss is that the surface impedance of a good conductor has an imaginary part which is equal to the real part. This reactive surface impedance is physically due to the increased surface inductance caused by the current being confined closer to the surface of the conductor. This surface reactance is included in the Sonnet loss model. The effect is small, but potentially significant in certain cases.

Keep in mind that a circuit running with lossless metal and dielectrics requires about one-half the amount of memory and runs about twice as fast. Therefore, the simplest approximation is to run a lossless simulation. This can be quite useful in the initial design phase.

### Problems In Determining Planar Metal Loss

Sonnet's planar metal loss model is very accurate if accurate values are used. In practice, however, there are many aspects of metal loss that cannot easily be accounted for. For example, surface roughness, metal purity, metal porosity, etc. cannot easily be measured and included in an all-encompassing loss model. In addition, most software programs, Sonnet included, do not allow you to enter all of the parameters that determine metal loss. Many users like to use the ideal values as a starting point and add a little of their own “real-world” loss. But how much should be added to the ideal models? This question is not easily answered, but is addressed in the next section.

An additional loss problem exists with planar EM analysis tools such as Sonnet. The problem stems from the fact that planar EM tools treat the planar metal conductor as zero thickness. This means that there is no difference between the top of the conductor and the bottom of the conductor. In some circuits, stripline for ex-

ample, the current is symmetrical with half of the current flowing on the top of the conductor and half flowing on the bottom of the conductor. The zero thickness model (Normal loss model) works well in these cases.

In other circuits, such as microstrip, the current can be unequally distributed, resulting in higher loss than the equivalent stripline circuit. If you know the ratio between the top and the bottom currents, you can obtain a better loss model. All planar solvers must either estimate this value in order to calculate metal loss, or the information must be input by the user. For this class of circuits, it is difficult for the user to know an exact value of the current ratio without obtaining measured data on the circuit. For these cases, assuming all the current flows on one side of the conductor gives a “worst case” loss result. This tends to compensate for the “best case” loss caused by ignoring the other aspects of loss (metal porosity, etc.) mentioned earlier.

## Determining Good Input Values

The best method to determine proper loss values is to build and measure a simple structure of the desired metalization. The structure should be sensitive enough to loss so that measurement errors do not significantly affect the results. Then analyze the same structure in Sonnet and adjust the loss values until the calculated loss matches the measured loss. This may take several iterations before success, but then you can use these values for similar circuits. You are now effectively using measured values for the loss parameters.

## Creating Planar Metal Types

To assign loss to a polygon in Sonnet, you first define a planar metal type by inputting its loss parameters and then assign that planar metal type to the polygon drawn in your circuit.

The Planar Metal Editor dialog box allows you to enter a loss definition for the planar metal type. There are seven different methods for entering loss: Normal, Resistor, Rdc/Rrf, General, Rough, Sense and Thick Metal Model. The different planar loss models are discussed below followed with a procedure for entering new planar metal types. The discussions assume simple, single conductor microstrip and stripline geometries where mentioned.

You do not need to read the details of each loss model. Instead, you should make yourself familiar with the loss models you are likely to use. Most users only need concern themselves with “Normal.”

### Conductivity, Resistivity, or Sheet Resistance at DC

Some of the loss models allow you to choose which parameter you wish to use to calculate loss: Conductivity, Resistivity or Sheet Resistance at DC. The three parameters are defined as follows:

- Conductivity is the bulk conductivity of the metal. The default unit is S/m.
- Resistivity = 1.0 / conductivity. Resistivity has a default unit of Ohm-cm.
- Sheet Resistance at DC = 1.0 / (conductivity \* thickness)

where “thickness” refers to the metal thickness in meters. Sheet resistance at DC has a default unit of Ohms/square.

#### Normal

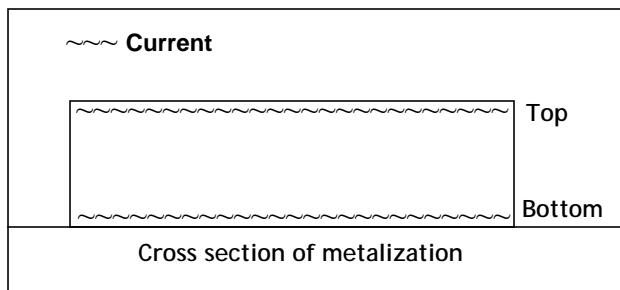
For the Normal planar metal type, you determine the loss using the metal thickness, the current ratio and a third parameter. For the third parameter you may choose bulk conductivity, resistivity or sheet resistance at DC.

---

**NOTE:** Metal thickness is used only in calculating loss; it does not change the physical thickness of metalization in your circuit. The metalization in your circuit is still modeled as zero-thickness.

---

**Current Ratio:** Normal metal models your circuit using zero thickness metal, but your real circuit possesses a metal thickness. At higher frequencies current flows in a thin skin around the edge of the metal, as pictured below. The current ratio is the ratio of the current flowing on the top of the metal to the current flowing on the bottom of the metal.



There are no sides in a zero thickness model; therefore when translating from these parameters, the current which flows on the sides is ignored. In some circuits, stripline for example, the current is symmetrical with half of the current

flowing on the top of the metal and half flowing on the bottom of the metal. In this instance, the current ratio is 1. If you had half as much current flowing on the top as on the bottom, the current ratio is 0.5. This could occur for some microstrip circuits, for example. It is difficult to know an exact value of the current ratio without obtaining measured data on your circuit. All planar solvers not using a thick metal model must estimate this value in order to calculate metal loss; the Normal Metal model and Rough Metal model allow you to enter the value. If you do not have access to measured data for your metal type, the default values (0 for Normal Metal and 1 for Rough Metal) are reasonable for most microstrip.

### Thick Metal

For thick metal, you input four parameters: metal thickness, number of sheets, direction and a fourth parameter. For the fourth parameter, you may choose bulk conductivity, resistivity, or sheet resistance at DC. The loss is calculated in the same manner as for the Normal metal type, except that the thickness in this case represents a physical thickness which eliminates the need to enter the current ratio. For more information about conductivity, resistivity, and sheet resistance at DC, please see “Conductivity, Resistivity, or Sheet Resistance at DC,” page 51. The direction setting determines if the thick metal extends upward into the dielectric layer above, or down into the dielectric layer below. Up is the default.

For a detailed discussion of thick metal, see Chapter 16, “Thick Metal” on page 314.

### Resistor

To define a planar metal which you wish to use as a resistor, enter the DC resistance in Ohms/sq in the Rdc text entry box which appears when you select this metal type. To use this loss model, the surface resistance should be constant over the frequency range of interest.

### Rdc/Rrf

This method allows you to enter two values:  $R_{DC}$  and  $R_{RF}$ . The first parameter,  $R_{DC}$ , determines loss at low frequency (where the conductor is much thinner than the skin depth). Surprisingly, other electromagnetic analyses often predict zero loss at low frequency because they assume  $R_{DC}$  is zero.

The second parameter is the skin effect coefficient,  $R_{RF}$ . **Em** multiplies this number by the square root of the frequency (in Hertz) to yield the Ohms/square value at high frequency.

The equations for  $R_{DC}$  and  $R_{RF}$  are:

$$R_{DC} = 1/(\sigma t)$$

$$R_{RF} = \text{Skin effect coefficient} = \sqrt{(\pi\mu)/\sigma}$$

where

$\sigma$ : Bulk conductivity in Siemens/meter

t: Metalization thickness in meters

$$\mu = \mu_0 * \mu_r$$

$\mu_r$  = relative permeability of your metal, which is typically 1.0 for most metal materials. Magnetic materials, such as nickel, have a  $\mu_r$  greater than 1 and therefore significantly decrease the skin depth.

Typical values for  $R_{DC}$  and  $R_{RF}$  are 0.004 and 3e-7. If you start getting very strange loss results, check  $R_{RF}$ , paying special attention to the exponent.



### TIP

---

Use this planar metal type if you wish to include the  $\mu_r$  effects in the metal type. Sonnet allows you to do this by adjusting the  $R_{RF}$  value.

---

**Em** also properly models the transition between electrically thin (low frequency) and electrically thick (high frequency) conductors. The transition frequency from  $R_{DC}$  to  $R_{RF}$  is the square of  $R_{DC}/R_{RF}$ . At this frequency, and a relatively narrow band around it, both coefficients are important. If the skin effect coefficient ( $R_{RF}$ ) is set to 0.0, then the value of  $R_{DC}$  is used over all frequencies. This is the usual case for resistors.

The above equation for  $R_{RF}$  assumes that all of the current travels on just one side of the conductor. This is a good approximation for some microstrip circuits. However, if the current really travels on both sides, this gives a pessimistic value for the loss. The equation should be modified for other structures. Stripline, for example, has current of equal amplitude on both the top and bottom of the conductor. In this case, you should divide the  $R_{RF}$  value by two, while maintaining  $R_{DC}$ .

As an example, the conductivity ( $\sigma$ ) for copper is 5.8E+7 Siemens/m, giving  $R_{DC} = 0.006$  Ohms/square ( $t = 3 \mu\text{m}$ ) and a microstrip  $R_{RF} = 2.6\text{E-}7$ . In reality, the bulk conductivity of copper, or any other given metal, may not equal the laboratory value, so the figures as calculated above are likely to be lower than actual results. The table below provides calculated results of commonly used metals using the equations above.

## Properties of Commonly Used Metals

Metal	$\sigma$ (S/M)	RDC ( $\Omega/\text{sq}$ ) $t = 1 \mu\text{M}$	RDC ( $\Omega/\text{sq}$ ) $t = 1 \text{ mil}$	RRF ( $\Omega\text{Hz}^{-1/2}/\text{sq}$ ) “Skin Effect” (microstrip)
Aluminum	3.72e7	0.027	1.1e-3	3.3e-7
Brass	1.57e7	0.070	2.5e-3	5.0e-7
Copper	5.80e7	0.017	6.8e-4	2.6e-7
Gold	4.09e7	0.024	9.6e-4	3.1e-7
Nichrome	1.00e6	1.000	3.9e-2	2.0e-6
Silver	6.17e7	0.016	6.4e-4	2.5e-7
Tantalum	6.45e6	0.155	6.1e-3	7.8e-7
Tin	8.70e6	0.115	4.5e-3	6.7e-7

### General

This loss model includes the metalization surface resistance described above in Rdc/Rrf. It also includes the metalization reactance, composed of the surface reactance, Xdc and the surface inductance, Ls. Surface reactance, Xdc, is specified, in Ohms/square and uses the same reactance at all frequencies. The surface inductance, L<sub>s</sub>, is specified in pH/square and models the reactance as being linearly proportional to frequency.

A superconductive effect known as “kinetic inductance” slows the velocity of the electrons with no loss of energy. This can be modeled as a surface inductance.

There are three recommended approaches to obtaining a value for Ls in superconductors. A first order approximation is to assume the metal is a perfect conductor.

$$R_{DC} = 0 \quad R_{rf} = 0 \quad X_{DC} = 0 \quad L_s = 0$$

This model works well for moderate frequencies (less than 150 GHz) and moderate circuit dimensions which are much greater than the London depth of penetration.

The second approach is a model which is still valid at moderate frequencies, but includes effects due to kinetic inductance. The kinetic inductance is a function of temperature and can be approximated in the following manner:

$$R_{DC} = 0 \quad R_{rf} = 0 \quad X_{DC} = 0 \quad L_s = \mu_0 \lambda_L(T)$$

where

$$\mu_0 = 4\pi(10^{-7}) \text{ H/m}$$

$$\lambda_L(T) = \lambda_0 / (\sqrt{1 - (T/T_c)^4}) \quad \text{London depth of penetration at temp.}$$

$\lambda_0$  = London depth at T = 0 meters

$T_c$  = Critical (Transition) Temperature in degrees Kelvin

The third model should be used to account for high frequency effects or effects due to small circuit dimensions. In these cases, the surface resistance's proportionality to  $\omega^2$  begins to dominate and the following model is suggested. The surface resistance is a function of frequency-squared, so you should use the equation below as the entry for  $R_{DC}$  and use the FREQ function in the equation so that the value is valid for all analysis frequencies. For more information about equation syntax, please refer to Help. Use the following equations for this model:<sup>1</sup>

$$\begin{aligned} R_{rf} &= 0 \\ R_{DC} &= \frac{1}{2}\omega^2 \mu_0^2 (\lambda_L(T))^3 \sigma_N (\eta_n / (\eta_n + \eta_s)) \\ X_{DC} &= 0 \\ L_s &= \mu_0 \lambda_L(T) \end{aligned}$$

where

$$\omega = 2\pi f \text{ radians/sec}$$

$\sigma_N$  = Conductivity of the superconductor in its normal state (Siemens/m<sup>2</sup>)

$\eta_n$  = Normal state carrier density (1/m<sup>3</sup>)

$\eta_s$  = Superconducting state carrier density (1/m<sup>3</sup>)

$\mu_0$  and  $\lambda_L(T)$  are as defined above

<sup>1</sup>Shen, Z. Y., "High-Temperature Superconducting Microwave Circuits," Boston, 1994, Artech House.

The Surface Impedance,  $Z_s$ , is modeled using the following equation:

$$Z_s = R_{DC} + j\omega L_s$$

where  $R_{DC}$  and  $L_s$  are defined in the superconductor models given above.

### Rough Metal

Many circuit manufacturing processes produce planar metallization with rough surfaces. Depending on how the metallization is formed, one, or both, of the trace metal surfaces could be relatively rough. The Rough Metal type allows you to capture this surface roughness effect in your model.

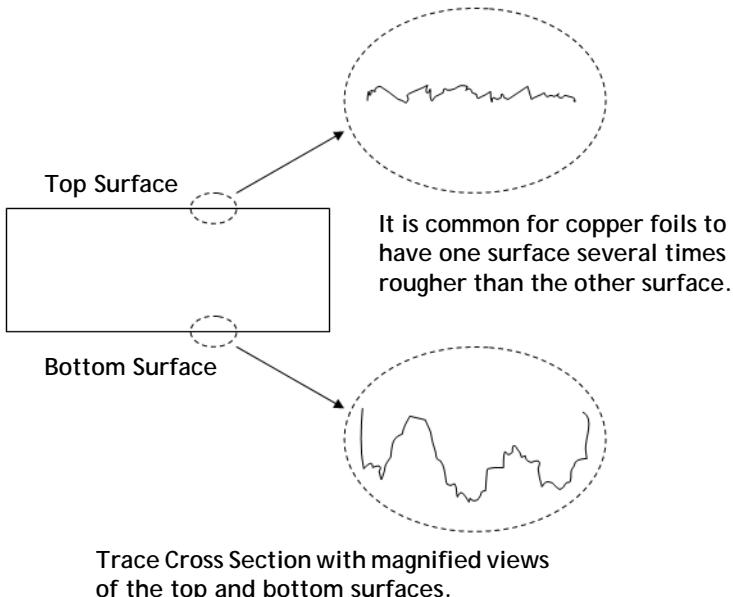
At frequencies where the skin depth is comparable or smaller than the surface roughness dimension, the current flow is forced into the surface roughness region and this introduces excess resistance and excess inductance in your traces. This in turn increases the insertion loss and insertion phase of your planar transmission lines.

You do not necessarily need to use the Rough Metal type as you create models in Sonnet. However, when you are working with circuits that are very sensitive to these transmission line characteristics, it will improve your model accuracy. Generally, models with rough metal surfaces and narrow trace widths, that operate at high frequency (above 5 GHz) are good candidates for the Rough Metal model.

The Sonnet Rough Metal Model is based on an extensive study of surface roughness effects in printed circuit board conductors. See reference [1] in Appendix II, "Sonnet References" for a detailed description.

It may be valid to apply Rough Metal to conductors from other manufacturing processes, as long as the surface roughness effect that you are trying to capture is a function of the trace top and bottom surfaces.

To define a Rough Metal type you must first obtain the metal trace top and bottom surface roughness in RMS microns for the metal. Often circuit material suppliers provide this information directly or you can ask the manufacturing group producing your physical parts.



### TIP

---

Surface roughness can also exist on ground plane metal. If you are using a Sonnet box cover to represent a ground plane, you can define a Rough Metal type and apply it to the cover. Rough Metal types assigned to a box cover are limited to a thin cross section.

---



### WARNING

---

Care should be taken when using the results in a time domain simulation, as the Rough Metal results are not guaranteed to be causal.

---

There are two variations of the Rough Metal: Thin and Thick. The Thin option models the metal as a single “zero thickness” (2D sheet), like the Normal loss model described on page 51 but with the surface roughness effect. With this option, the user must specify a Current Ratio.

The Thick option models the metal as two sheets of current; one for the top surface and one for the bottom surface of the planar metal trace. This is very similar to the Thick Metal loss model (with two sheets), with the surface roughness effect. See Chapter 16 "Thick Metal" on page 303 for additional details on modeling metal thickness.

For additional details about rough surface metal, please see "Rough metal model" in the index of Help. You may access help by selecting *Help* ⇒ *Sonnet Help* from any Sonnet application.

### Sense

**Em** solves for the current distribution; however, on occasion, you may want to view the fields, not the current. You do this with what is called a "sense metal". The sense metal is a patch of conductor placed where you want to see the tangential electric field. (You cannot view the normal direction of the field with Sonnet.)

For further discussion of sense metal, see "View Tangential Electric Fields" under Tips and App notes in Help.

### How to Create a Planar Metal Type

For detailed instructions on creating a planar metal type, please refer to "metal types" in the Index of Help. You may access help by selecting *Help* ⇒ *Sonnet Help* from the main menu of any Sonnet application or by clicking on the Help button in any dialog box.

## Via Metalization Loss

Via metalization loss is specified in the project editor in the Metal Types dialog box which is opened by selecting *Circuit* ⇒ *Metal Types*. You click on the Add Via button to add a via metal type. Via metal types may be assigned to via polygons only. Please note that edge vias are a special case which use the planar metal type of the polygon to which they are attached. For a discussion of loss in edge vias, please see "Edge Vias" on page 300.

Sonnet allows you to use pre-defined via metals, such as gold and copper, using the global library. The global library allows you to define your own metal types as well. There is also a local metal library which can be created for an individual or to share between users.

## Creating Via Metal Types

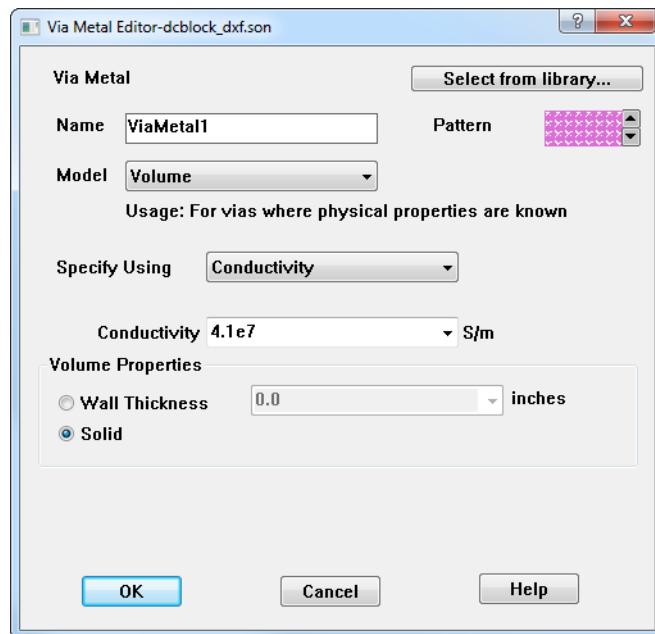
To assign loss to a via in Sonnet, you first define a via metal type by inputting its loss parameters and then assign that via metal type to the via drawn in your circuit. This section provides instructions for creating a via metal type and discusses the loss models used in the Via Metal Editor dialog box.

The Via Metal Editor dialog box allows you to enter a loss definition for the via metal type. There are three different loss models: Volume, Surface, and Array. The different via loss models are discussed below.

You do not need to read the details of each loss model. Instead, you should make yourself familiar with the loss models you are likely to use. Most users only need concern themselves with “Volume.”

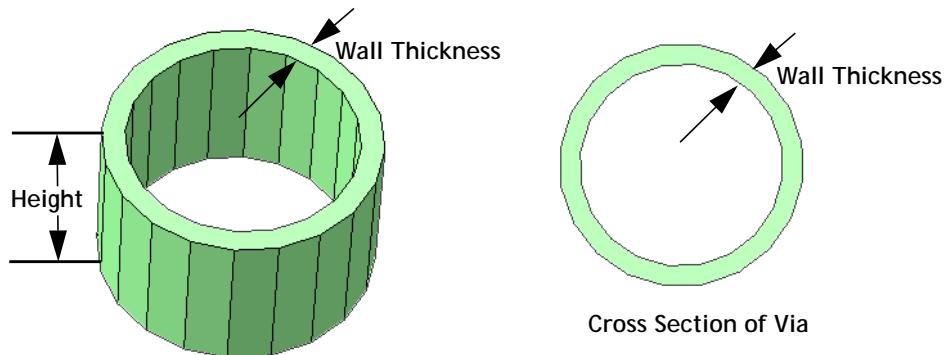
### Volume Loss Model

The Volume via loss model is the most commonly used model because the values you specify are directly related to the physical properties of the via. With the volume model, you enter a conductivity, resistivity, resistance per via, or sheet resistance at DC and Volume Properties. An example using “Solid” Volume Properties is shown below:



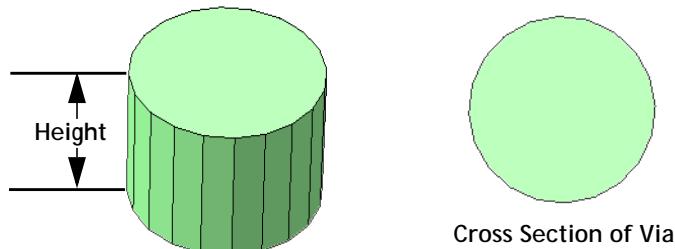
The Volume Properties section allows you to model both hollow and solid vias. Examples are depicted below:

**Hollow:** If your vias are hollow, you enter the wall thickness of your vias. The wall thickness and the shape of each via polygon are used to determine the cross-sectional area used for the loss calculations. In the illustration below, a 3D view of the via is shown on the left, with the cross sectional area shown on the right.



3D Volume model of via with finite wall thickness (hollow via)

**Solid:** If your via is solid, you select the “Solid” radio button. The complete area of the via polygon is used for the loss calculations. In the picture below, the cross sectional area is the area of the circle.



3D Volume model of via with solid cross-sectional area

---

**NOTE:**

Note that defining your Volume Properties as either Wall Thickness or Solid does not affect the subsectioning of the via or how the via appears in the project editor.

---

### Loss Using Conductivity, Resistivity and Sheet Resistance at DC

If you specify your via loss using conductivity, resistivity, or sheet resistance at DC, the Volume loss model is frequency-dependent with both a DC loss component and an RF loss component. At DC, the vertical loss of a single via polygon is a resistance given by the equation below:

$$Z_{Loss}(0) = H / (\sigma \cdot A) \text{ (ohms)}$$

where

H: Height of via in meters.

$\sigma$ : Conductivity of via metal in Siemens/meter.

A: Cross-sectional area of via based on wall thickness or solid setting.

Note that you may choose to use either the resistivity or sheet resistance at DC, instead of the bulk conductivity as in the equation above to define your loss. If you do so, this equation is modified based on the chosen parameter. For the relationship between conductivity, resistivity, and sheet resistance at DC, please see “Conductivity, Resistivity, or Sheet Resistance at DC,” page 51.

At RF, the equation is too complex to show here, but is the volumetric equivalent of the surface impedance equation used by Sonnet for planar metal, and is described in detail in [reference \[15\] listed on page 422](#). This equation includes the skin effect, resulting in a smooth transition from DC to high frequencies.

### Loss Calculations for Volume Model Using Resistance Per Via

The loss may also be defined by using the Resistance per Via. Resistance per Via for the volume model represents the total resistance in Ohms of the via polygon in the vertical (Z) direction. This resistance is constant with frequency. If the via polygon extends over more than one dielectric layer in the Z-direction, then the total specified resistance is apportioned to each layer according to the thickness of each layer. For example, if the specified RPV for a via polygon is 3.0 Ohms, and if the via polygon extends from the top of dielectric layer 1 (10 um thick) to the bottom of dielectric layer 2 (20 um thick), then 1 Ohm is apportioned to dielectric layer 1 and 2 Ohms is apportioned to dielectric layer 2.

## Horizontal Loss for the Volume Model

When you are using the Volume Loss model (solid) for bar vias (mesh is set to Bar), then the horizontal DC loss in ohms/square is calculated as follows:

$$Z_{loss}(0) = 1.0 / (\sigma \times H)$$

where

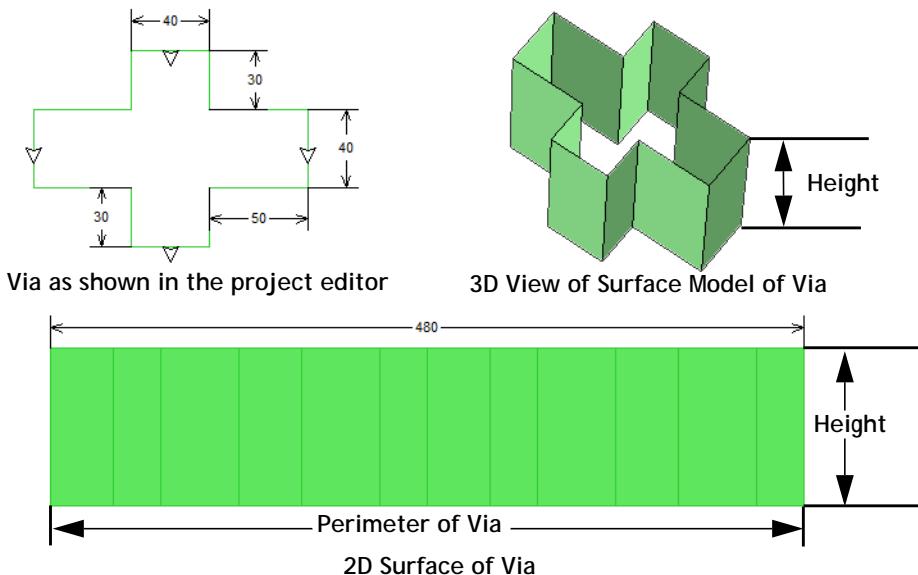
H: Height of via in meters.

$\sigma$ : Conductivity of via metal in Siemens/meter.

The horizontal DC loss for other types of fill is geometry and meshing dependent so it is not possible to provide an equation.

## Surface Loss Model

The surface via loss model treats the via as an infinitely thin hollow tube giving it a 2D surface as pictured below.



## Loss Calculations for Surface Model

This method allows you to enter three values:  $R_{DC}$ ,  $R_{RF}$  and  $X_{DC}$ . The first parameter,  $R_{DC}$ , determines loss at low frequency (where the conductor is much thinner than the skin depth).

The equation for  $R_{DC}$  is:

$$R_{DC} = 1/(\sigma t)$$

where

$\sigma$ : Conductivity of via metal in Siemens/meter.

t: Metalization thickness in meters

The second parameter is the skin effect coefficient,  $R_{RF}$ . At frequencies above DC, **em** uses the same surface impedance equation used by Sonnet for planar metal [15]. This equation includes the skin effect, resulting in a smooth transition from DC to high frequencies. The equation for  $R_{RF}$  is:

$$R_{RF} = \text{Skin effect coefficient} = \sqrt{(\pi\mu)/\sigma}$$

where

$\sigma$ : Bulk conductivity in Siemens/meter

$$\mu = \mu_0 * \mu_r$$

$\mu_r$  = relative permeability of your metal, which is typically 1.0 for most metal materials. Magnetic materials, such as nickel, have a  $\mu_r$  greater than 1 and therefore significantly decrease the skin depth.

The third parameter is  $X_{dc}$ , the surface reactance, and is normally set to zero. This parameter is provided for advanced users who want to customize the surface impedance of the metal.

The horizontal loss at DC and RF for the Surface model is infinite.

### Array Loss Model

The Array via loss model is used to represent an array of small solid vias by approximating the entire array as a single large via. This is usually done to reduce the processing resources necessary to analyze the circuit, as having many small vias is computationally expensive. The Array via loss model is most often used for computing the loss of via arrays which have been simplified when translating a circuit into a Sonnet project. In those cases, the Via metal type is automatically created. You may also use this loss model to manually create a via metal type if you wish to model a single via in the project editor that represents an array in your final design.

---

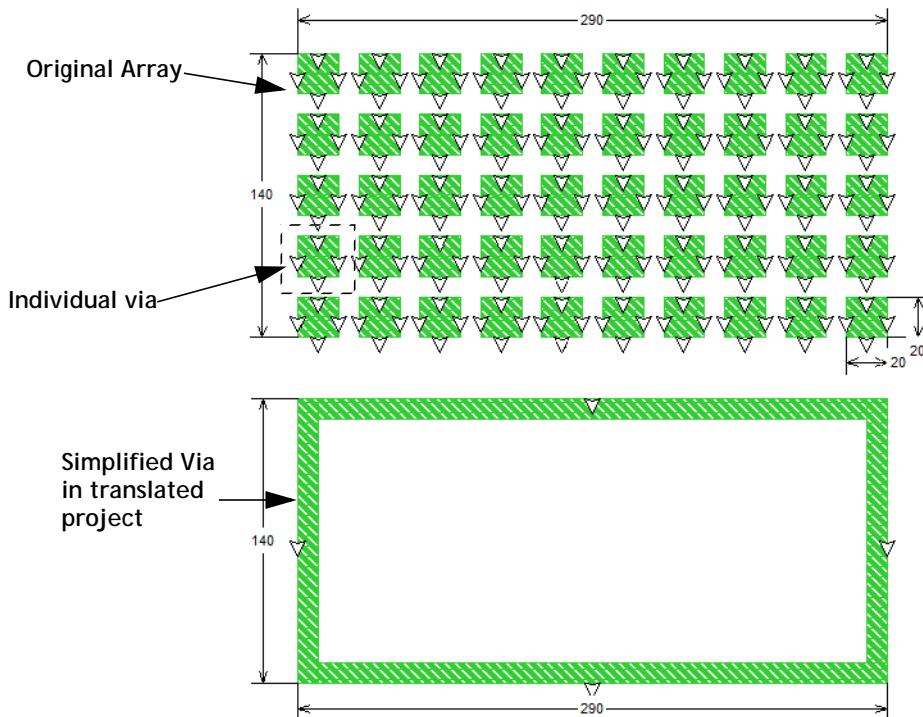
NOTE:

For a detailed discussion of array simplification, please see the **Via Array Simplification Appendix** in any interface or translator manual.

---

Like the Volume model defined above, it is a frequency-dependent model which takes into account the 3D characteristics of the via array. The only difference between the Volume model and this model is the method by which the cross-sectional area is computed. With the Volume model, the cross-sectional area is computed based on the wall thickness or solid setting. For the Array model, the cross-sectional area is computed using the Fill Factor or Density which is based on the original via array.

The Fill Factor is a loss parameter when using conductivity or resistivity to define the loss. The Fill Factor is a percentage indicating how much of the cross-sectional area of the original via array is metal-filled. The Fill Factor is calculated by summing the areas of all the individual vias in the array and dividing that by the total area of the array, defined as the bounding box which surrounds all the vias in the array. The result is multiplied by 100 to convert to a percentage. An example of an array and how to compute its fill factor is shown below.



The array, shown on top, consists of a 5 by 10 array of 50 individual vias whose cross section is a 20 micron by 20 micron square. Therefore the sum of the areas of all the individual vias is  $50 * 400 = 20,000$  square microns. The total cross-sectional area of this array is 40,600 square microns ( $290 \text{ microns} * 140 \text{ microns}$ ). This provides a Fill Factor of 49.3% ( $((20,000/40,600) * 100$ ). The converted simple via modeled in Sonnet is shown on the bottom. Although the total cross sec-

tional area of the simplified via is 40,600 microns, the Array loss model is set such that the calculated DC loss for the via is based on the loss of the actual metal in the array, ( $40,600 * 0.493 = 20,015.8$ ) i.e., the 20,000 square microns.

---

**NOTE:** When translating circuits into Sonnet, if there are two or more arrays which use the same metal but whose fill factors differ significantly, two or more via metal types are created in the Sonnet project. For example, there are two arrays, one with a fill factor of 50% and another of 75% both using a metal named "viametal1." The translated project will contain two via metal types called "viametal1\_1" and "viametal1\_2."

---

The loss for the Array model may also be defined using the Resistance Per Via. Resistance Per Via for the array model represents the vertical (Z) resistance in Ohms of an individual via within the via array. As discussed above, the array of vias is modeled as a single, merged via polygon, but the loss for that simplified polygon is based on the actual metal present in the array.

The loss parameters of this simplified via polygon are Resistance Per Via ( $R_{per\_via}$ ) in Ohms/via and Via Density ( $D_{via}$ ) in vias per square micron. The total resistance in Ohms of the simplified via polygon is constant with frequency and is computed as follows:

$$R_{simplified\ -\ via} = \frac{R_{per\ -\ via}}{(D_{via} \times A_{via})}$$

where  $A_{via}$  is the cross-sectional area of the simplified via polygon in square microns.

The horizontal loss at DC and RF for the Array model is infinite.

## Metal Libraries

There are two types of metal libraries available in Sonnet: global and local. The metal libraries contain standard definitions for metal types, both planar and via, which may be used in your projects. There is no real difference between the global and local library. The names refer to how they are used. The global library would usually be used as a group wide library of standard metals for a group of designers.

There is a default global library supplied by Sonnet which contains definitions for some common metals, both planar and via. The default location for the global library is in <Sonnet Data Directory>/data/libraries where <Sonnet Data Directory> is the directory in which your Sonnet software data is installed. You may choose to use this location or can save this library in another location.

The local library would usually be used as the user's own library of metal definitions. This library may be stored in a location of the user's choice. You use either the Planar Metal Editor dialog box or the Via Metal Editor dialog box to add, edit and delete entries to these libraries.

For instructions on adding planar and via metals to a metal library, editing metals in a metal library or using metals defined in a metal library, please refer to Sonnet Help available from the Help menu in any Sonnet application.

## Dielectric Layer Loss

The dielectric layer loss calculations in Sonnet are virtually exact. Our web site has a lossy conductivity benchmark you can perform on any electromagnetic solver (or measurement system). See Benchmarking on the Products section of our web site, [www.sonnetsoftware.com/products/sonnet-suites/benchmarking.html](http://www.sonnetsoftware.com/products/sonnet-suites/benchmarking.html).

The dielectric loss is calculated in Sonnet at the beginning stages of the analysis. The method Sonnet uses starts with the calculation of a sum of waveguide modes. The exact solution requires an infinite sum of modes, but Sonnet truncates this sum to some reasonable value (the truncation has never been a source of error). So, for each mode, if there is a lossy dielectric, the calculation involves complex numbers instead of just real numbers. This is NOT a discretized function - it is a continuous function. Therefore, the dielectric loss calculation can be thought of as exact (only limited to the precision of the machine).

## Dielectric Layer Parameters

You can set the dielectric constant and loss of a dielectric layer by changing the following parameters in the project editor by selecting *Circuits*  $\Rightarrow$  *Dielectric Layers*, then clicking on the Above, Below or Edit button in the Dielectric Layers dialog box. This opens the Dielectric Editor dialog box which allows you to edit the parameters below.

- **E<sub>rel</sub>:** The relative dielectric constant ( $\epsilon_r$ ). The ratio ( $\epsilon/\epsilon_0$ ), where  $\epsilon$  is the real part of the permittivity of the dielectric layer material, and  $\epsilon_0$  is the permittivity of free space. The ratio is dimensionless.
- **Dielectric Loss Tan:** The dielectric loss tangent. The ratio ( $\epsilon''/\epsilon'$ ), where  $\epsilon = \epsilon' - j\epsilon''$ , and  $\epsilon$  is the complex permittivity of the dielectric layer material. The ratio is dimensionless.
- **Conductivity:** Dielectric conductivity,  $\sigma$ , where  $\sigma$  is the conductivity of the dielectric. The default unit is S/m. You may define your dielectric loss using conductivity or resistivity. To use conductivity, you should select “Conductivity” from the Specify Using drop list.
- **Resistivity:** Resistivity = 1.0 / conductivity. Resistivity has a default unit of Ohm-cm. You may define your dielectric loss using conductivity or resistivity. To use resistivity, you should select “Resistivity” from the Specify Using drop list.
- **M<sub>rel</sub>:** The relative magnetic permeability ( $\mu_r$ ) of the dielectric layer material.
- **Magnetic Loss Tan:** The magnetic loss tangent of the dielectric layer material.

One last parameter that may be specified for a dielectric layer is the Z Partitioning. This value may be changed in the Z Partitions dialog box which is opened when you click on the Z Parts button in the Dielectric Layers dialog box.

- **Z-Partitions:** The z-partitioning parameter is the setting which controls the number of partitions which the dielectric layer is divided into in the z direction. for the dielectric layer. While this parameter is specified in the dielectric layer window, it only has an effect on the dielectric bricks on that layer. Changing this value for a particular layer will have absolutely no affect on the analysis if there are no bricks on the layer. If there are multiple bricks on the layer, the Z subsectioning for all of those bricks will be identical.

The more partitions (better resolution) used in the Z-dimension, the more accurate the analysis; however, analysis time and memory requirements also increase dramatically.

## Frequency Dependent Loss

Sonnet allows you to write your own frequency-dependent equations, or use a lookup table for any of the parameters defining a dielectric. For example, you could write a frequency-dependent equation for the conductivity of your dielectric. For more information about using equations, please refer to the topic ‘Equation Syntax’ in Help.

Most common dielectrics have either a fairly constant loss tangent or conductivity as a function of frequency. Thus, it is normally not necessary to specify an equation for your dielectric loss. You can even enter both a loss tangent and a conductivity (or resistivity) for a dielectric and two values are combined to provide the total loss for your analysis. The following equation can be used to calculate the total effective  $\tan\delta$  used by **em** for this situation:

$$\tan\delta = (\text{Loss Tan}) + \frac{(\text{Diel Cond})}{\omega(E_{\text{rel}})\epsilon_0}$$

where,  $\omega$  is the radian frequency ( $\omega = 2\pi f$ , where  $f$  is frequency in hertz). Note that  $\tan\delta$  has both a frequency-dependent term and a frequency-independent term

The above equation for  $\tan\delta$  can also be expressed in terms of conductivity as follows:

$$\text{Total Effective Cond} = (\text{Loss Tan})\omega(E_{\text{rel}})\epsilon_0 + (\text{Diel Cond})$$

Both equations are equivalent. Each describes how **em** uses the input dielectric parameters to compute loss in the dielectric material.

---

**NOTE:**

Note that if resistivity is input, instead of conductivity, the equation used is altered accordingly.

---

See “Circuit  $\Rightarrow$  Dielectric Layers” in the project editor’s Help for information on setting these parameters.

### How to Create a New Dielectric Layer

For detailed instructions on creating a new dielectric layer, please refer to “dielectric layers:adding” in the Index of Help. You may access help by selecting *Help* ⇒ *Sonnet Help* from the main menu of any Sonnet application or by clicking on the Help button in any dialog box.

### Dielectric Libraries

The dielectric libraries contain standard definitions of dielectric materials which may be used for your dielectric layers. There are two types of dielectric libraries available in Sonnet: global and local. There is no real difference between the two libraries. The names refer to the way in which they are used.

---

**NOTE:** **The Dielectric Libraries materials may not be used for dielectric bricks but only for dielectric layers.**

---

The global library would usually be used as a group wide library of standard dielectrics for a group of designers. There is a default global library supplied by Sonnet which contains definitions for dielectric materials. The default location for the global library is in <Sonnet Data Directory>/data/libraries where <Sonnet Data Directory> is the directory in which your Sonnet software data is installed. You may choose to use this location or can save this library in another location.

The local library would usually be used as the user's own library of dielectric material definitions. This library may be stored in a location of the user's choice. You use the Dielectric Editor dialog box to add, edit and delete entries to these libraries.



# Chapter 5

# Ports

---

This chapter describes the five different types of ports used in Sonnet, how they are modeled and how to place or delete them in your circuit.

The five types of ports used in Sonnet are the standard box-wall port, the co-calibrated internal port, the via port, the auto-grounded port and the ungrounded internal port.

## Port Type Overview

As mentioned above, the five types of ports available in Sonnet are the standard box-wall, co-calibrated internal port, via port, autogrounded port and internal ungrounded port. All ports in Sonnet are two-terminal devices. The box-wall and co-calibrated port types are those used the majority of the time. Each type of port is described below.

### **Box-wall:**

- Most common type of port.
- Positive terminal is attached to a metal polygon and the negative terminal is attached to the box-wall (ground).
- De-embedding is the most accurate for this type.
- Used for connections to the periphery of your geometry.

- Reference planes may be used.

For more information on box-wall ports, see “Standard Box-wall Port,” page 87.

### **Co-calibrated internal port:**

- Used in the interior of a circuit.
- By default, automatically identified as part of a calibration group with a common ground node connection. The ground node connection can be defined as floating or the Sonnet box. Please note that you can use a single co-calibrated port if the ground node connection is set to the Sonnet box.
- When *em* performs the electromagnetic analysis, the co-calibrated ports within a group are simultaneously de-embedded.
- Highly accurate de-embedding.
- Often used by a circuit simulation tool to connect some type of element into your geometry at a later time outside the Sonnet environment.
- Reference planes may be used.

For more information on co-calibrated internal ports, see “Co-calibrated Internal Ports,” page 89.

### **Via port:**

- Used in the interior of a circuit.
- Negative terminal is connected to a polygon on a given level and the positive terminal is connected to a second polygon on another level above.
- Cannot be de-embedded.
- Most commonly used to attach a port between two adjacent levels in your circuit or when you want a port to go up to the box cover rather than down to ground.
- Reference planes cannot be used.

For more information on via ports, see “Via Ports,” page 98.

### **Auto-grounded port:**

- Used in the interior of a circuit.
- The positive terminal is attached to the edge of a metal polygon and the negative terminal is attached to the ground plane through all intervening dielectric layers.
- Used in place of a co-calibrated port to reduce the de-embedding processing time at the cost of less accuracy.
- Reference planes may be used.

---

**NOTE:** When using auto-grounded ports, be aware that any coupling between auto-grounded ports is not accounted for when performing the de-embedding.

---

For more information on auto-grounded ports, see “Automatic-Grounded Ports,” page 99.

**Ungrounded Internal Ports:**

- Used in the interior of a circuit.
- Each terminal is attached to one of two adjacent polygons.
- Used in place of a co-calibrated port when you do not want any space between the two polygons.
- Will have a different ground reference from the other ports in the circuit.
- Commonly used to add a series element in post *em* processing.
- Reference planes cannot be used.

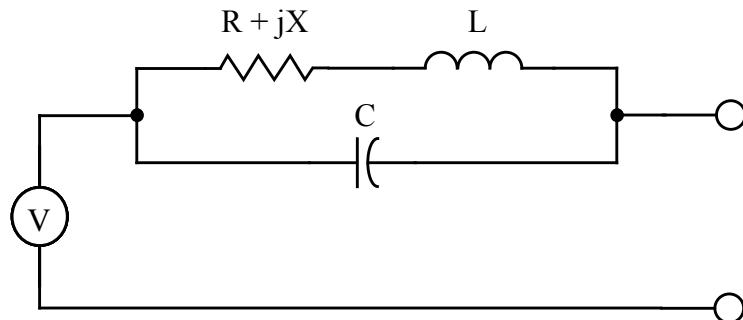
For more information on ungrounded internal ports, see “Ungrounded Internal Ports,” page 101.

## Port Normalizing Impedances

The default normalizing impedance for a port is 50 ohms. This is done since 50 ohms is an industry standard; some analysis tools only accept the value of 50 ohms as the normalizing impedance.

In rare cases, you may wish to have S-parameters normalized to some other impedance. The normalizing impedance in Sonnet is represented by four numbers as shown in the diagram below. First is the real part in ohms. Next comes the reactive part in ohms. Third is the inductive part in nanohenries (nH). The last number is

the capacitive part in picofarads (pF). The inductive and capacitive parts modify only the reactive portion of the load, they are included so you do not have to manually re-calculate the reactive part at each frequency.



Equivalent circuit of an *em* port.

---

**NOTE:**

The normalizing impedances are ignored if Y- or Z-parameters are specified for output. Y- and Z-parameters are always normalized to one ohm.

---

This capability should be used only by the most advanced user. This feature should never be applied to data which is to be used in a standard circuit theory program other than Sonnet's. Many programs assume S-parameters normalized to exactly 50 ohms; S-parameters normalized to another value would introduce error into the analysis.

## Changing Port Impedance

There are two methods for changing the impedance of a port. If you wish to change the impedance of a given port, and do not need to see the impedance values of other ports, take the following steps using the project editor:

- 1 Select the desired port(s). This will enable the *Modify* menu option.
- 2 Select *Modify*  $\Rightarrow$  *Port Properties* to open the Port Properties dialog box.
- 3 The impedance values can be changed by typing the desired values in the *Resistance*, *Reactance*, *Inductance* and *Capacitance* text boxes in the dialog box. This changes the parameters on all ports selected and all ports with the same number as the ports selected.

If you wish to change the impedance of a given port, and wish to see the impedance values of other ports while doing so, proceed as follows:

- 1 Select *Circuit*  $\Rightarrow$  *Ports* from the main menu to open the Port Impedance dialog box.
- 2 The impedance values for any port can be changed by typing the desired values in the *Resistance*, *Reactance*, *Inductance* and *Capacitance* fields in the row labeled with the desired port number.



### TIP

---

Note that the impedance of multiple ports may be changed at the same time through the first method by selecting multiple ports before selecting *Modify*  $\Rightarrow$  *Port Properties*, and by the second method, by modifying all the desired port values while the Port Impedance dialog box is open.

---

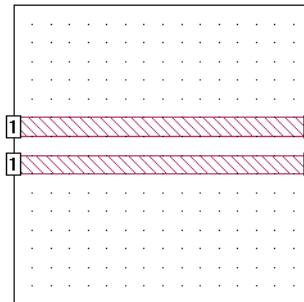
## Special Port Numbering

All ports are assigned a number at the time they are created in the project editor. There is no limit on the number of ports and the number of ports has a minimal impact on the analysis time needed for de-embedding. By default, the ports are numbered by the order in which they are created (i.e. first port created is assigned the number 1, second port created is assigned the number 2, etc.). With this default method, all ports are positive and unique. However, there are some applications that require the ports to have duplicate, and even negative, numbers. There is also a special case of Port 0 which is used for a ground connection.

### Ports with Duplicate Numbers

All ports with the same number, as pictured below, are electrically connected together. As many physical ports as desired may be given the same numeric label. Such ports are common mode ports (sometimes referred to as “even-mode” or

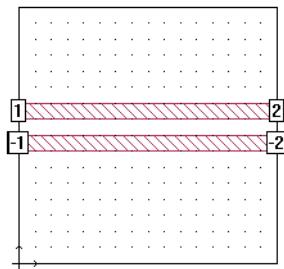
“push-push” ports) and have many uses, including simulating the even-mode response of a circuit. See “Modeling an Arbitrary Cross-Section,” page 314, for an example of using common mode ports.



Ports with identical port numbers are electrically connected together.

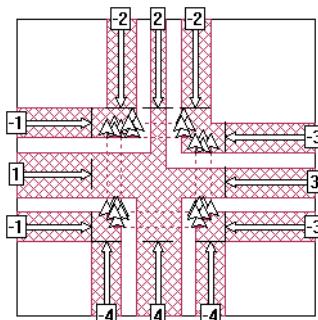
## Ports with Negative Numbers

Ports may also have negative numbers as shown in the figure below. This feature can be used to redefine ground. Strictly speaking, **em** sums the total current going into all the positive ports with the same port number and sets that equal to the total current going out of all the ports with that same negative port number. For example, for a circuit with a +1 port and a -1 port, **em** sets current flowing into port +1 to be equal to the current flowing out of port -1. These are differential mode ports (sometimes called “balanced”, “push-pull” or “odd-mode” ports). An example is shown below.



The total current going into port 1 is equal to the total current going out of port -1.

Coplanar lines can be represented with balanced ports. See below for an example of differential ports.



An example of a coplanar waveguide cross junction.

Ports are required to be in sequential order with no numbers missing. If the ports are not in sequential order, you will receive an error message when you attempt an analysis. If you are editing your circuit in the project editor, you may renumber all of the ports in your circuit to ensure that they are sequential in the Port Impedance dialog box (*Circuit*  $\Rightarrow$  *Ports*). The port order for the S, Y, or Z-parameters will be listed in increasing numeric order. For the example of a two-port, the output would be S11, S12, S21, and S22. For a four-port, it would be S11, S12, S13, S14, S21, S22, etc.

For a discussion of using ports to model coplanar waveguides (CPW), please see “Modeling Co-Planar Waveguide (CPW) in Sonnet” under Tips and App Notes in Help. You can find this topic by selecting “CPW” in the help Index.

---

**NOTE:**

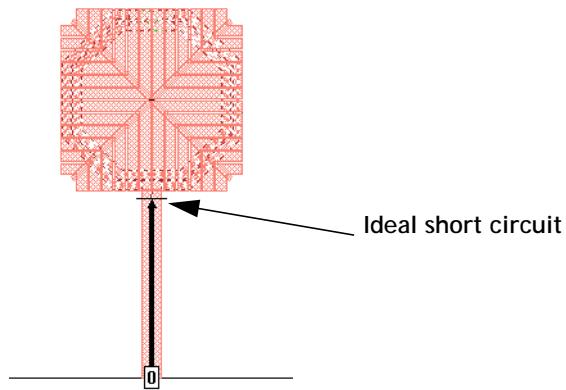
**When you are referred to Sonnet’s Help, you may access Help by selecting *Help*  $\Rightarrow$  *Sonnet Help* from the menu of any Sonnet application or by clicking on the Help button in any dialog box.**

---

## Using Port Number Zero for Ground Connections

The Port 0 feature allows you to add connections to ground that are de-embedded during the em simulation. Ports can be assigned a number “0” as shown in the

picture below.



This feature allows you to add an ideal short circuit to your circuit in locations other than the analysis box sidewall. In the example above, a boxwall port with a reference plane was used to create an ideal short circuit at the edge of a patterned ground shield. Assigning the port number 0 sets the port to be an ideal short circuit, and the reference plane is used to shift the short circuit to the desired location.

All ports with the number “0” are treated as a normal port during the main part of the EM simulation. These ports are de-embedded in exactly the same manner as all other ports. When the simulation for a particular frequency is completed, **em** then automatically shorts out all ports with the number 0. This means that the effects of the port are de-embedded without creating extra ports in your S-parameter results.

All port types: box-wall, co-calibrated, auto-grounded and ungrounded-internal can be set to 0. Multiple ports within a circuit can be set to number 0 just as multiple ports can use the same number as discussed in ["Ports with Duplicate Numbers" on page 75](#).

## Changing Port Numbering

You can change the port number of any port after it has been added to your circuit. Any integer, zero, negative or positive, is valid. To change the number on a port or ports, do the following:

- 1 Select the port(s) whose number you wish to change.
- 2 Select *Modify* ⇒ *Port Properties* from the main menu to open the Port Properties dialog box.

- 3 The number for the selected port(s) can be changed by typing the desired port number in the dialog box. Note that if multiple ports are selected, all are set to the number input in the dialog box.

## Renumbering All Ports

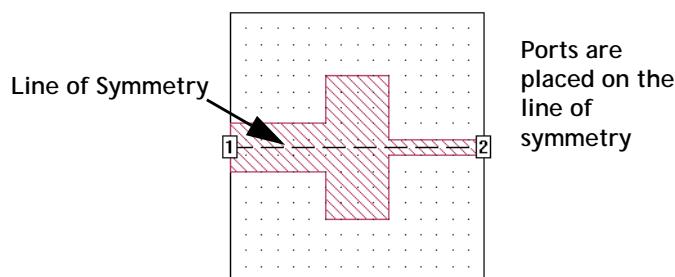
Ports in your circuit must be sequentially numbered prior to analysis. If you have deleted some ports such that your ports are no longer sequentially numbered, you may automatically renumber all of your ports. Duplicate port numbers can be changed or kept. If you wish to renumber your ports, proceed as follows:

- 1 Select *Circuit*  $\Rightarrow$  *Ports* from the project editor's main menu to open the Port Impedance dialog box.
- 2 Click on the Renumber button in the Port Impedance dialog box. If you have ports with duplicate numbers, you will be asked if you wish to keep the duplicates or renumber them. Ports numbered 0 are not affected by this command, all other ports are renumbered.

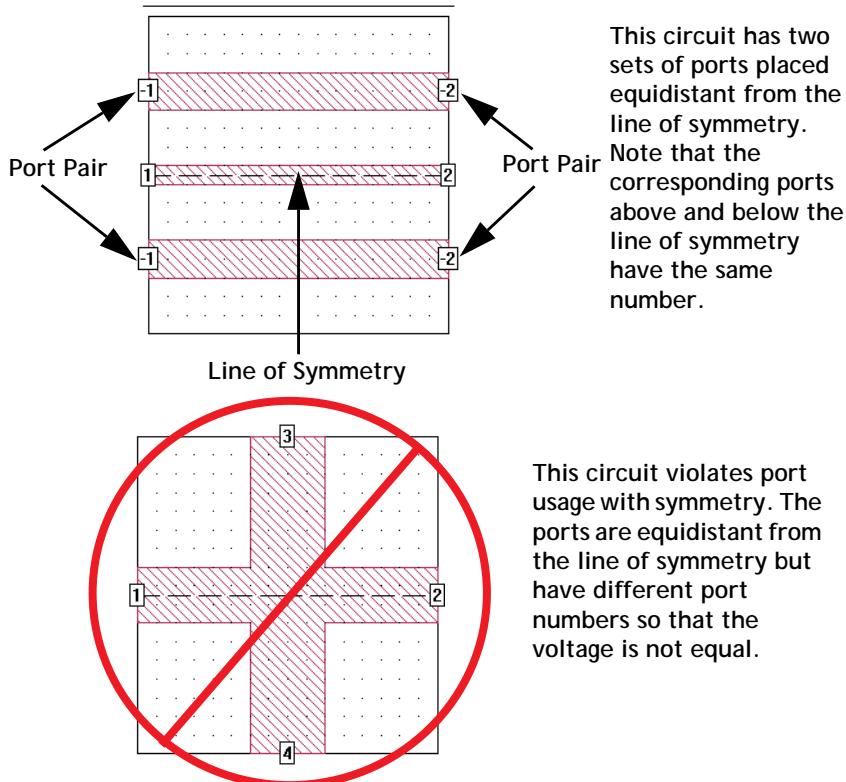
## Port Placement with Symmetry On

Symmetry can be used to considerably reduce the amount of memory and processing time required to analyze a circuit which is symmetrical about the midline of the substrate. When symmetry is turned on in the Box Settings dialog box (*Circuit*  $\Rightarrow$  *Box*), everything below the line of symmetry is ignored, and all metal above the line of symmetry is “reflected” about the symmetry line. Special care should be given to ports when using symmetry.

Normally, if you are using symmetry, ports are placed on the line of symmetry as pictured below.



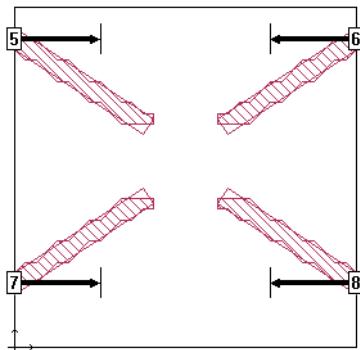
Ports can be placed off the line of symmetry, but the port placed above the line of symmetry must have another port equidistant from and below the line of symmetry. These two ports must also have the same port type, port number and properties. The basic premise of symmetry is that the voltage at any given point above the line of symmetry must be equal to the voltage at the corresponding location below the line of symmetry. Using the same port number, port type and properties ensures that the voltage is the same at both required points since all ports with the same number, as pictured below, are electrically connected together.



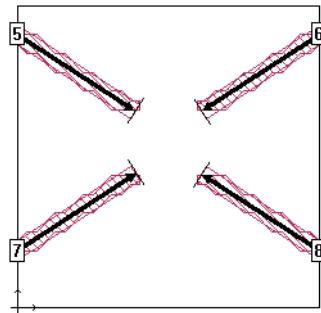
## Independent Reference Planes and Diagonal Ports

There are two types of reference planes: shared or independent. By default, all reference planes on a box side are shared and the same reference length is used for all ports on that side. When independent reference planes are used, any coupling between feed lines is not removed.

For some designs, feed lines do not lie in either the x or y axis, but extend diagonally from the port. In this case, if the port is orthogonal, the reference plane does not extend along the feed line, so that the feed line effects are not removed in the de-embedding process. An example is shown below.



The solution to this problem is to apply the diagonal port feature to your port. A reference plane must be defined as independent in order to use the diagonal feature. The diagonal feature may be applied to box-wall ports, co-calibrated ports and components. This feature is not available for auto-grounded ports or via ports. Once a port is designated as diagonal, the reference plane will extend along the polygon to which the port is attached as shown below.

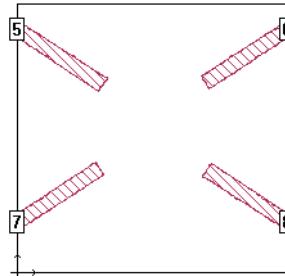


## Applying the Diagonal Port Feature to a Port

In order to apply the diagonal feature to a port, you must first add the port to circuit. As mentioned earlier, the port defaults to being an orthogonal port. After the port has been added, you may use the Port Properties dialog box to select the diagonal feature and specify the reference planes. Detailed instructions are provided below.

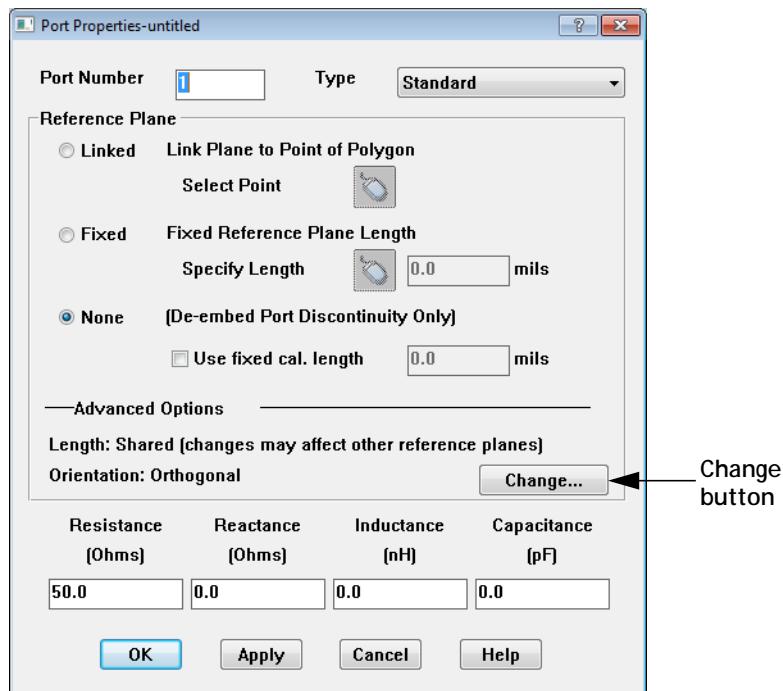
## 1 Add the desired ports to your circuit.

For instructions on adding a port, please see the appropriate section later in this chapter for the type of port you wish to add. This example uses standard box-wall ports. For instructions on adding a box-wall port, see "Adding Box-wall Ports" on page 88. An example circuit with four box-wall ports is shown below.



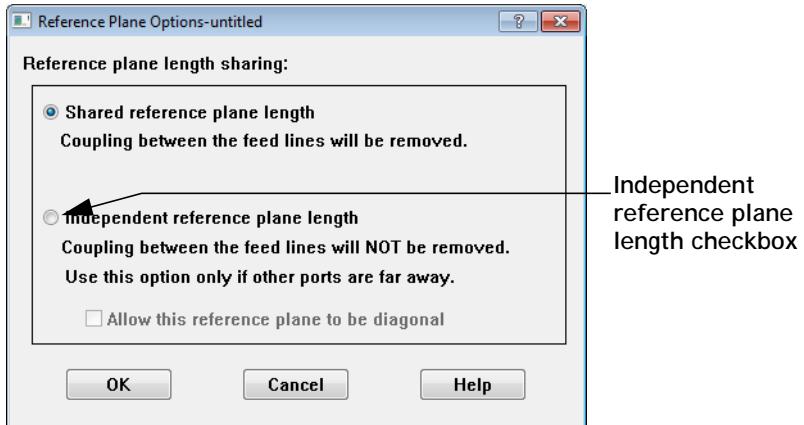
## 2 Double click on a port to open the Port Properties dialog box.

You can also select a port and then use the *Modify*  $\Rightarrow$  *Port Properties* command in the project editor's main menu to open this dialog box.



- 3 Click on the Change button in the Advanced Options section of the dialog box.

This opens the Reference Plane Options dialog box which allows you to specify reference plane sharing and apply the diagonal feature.

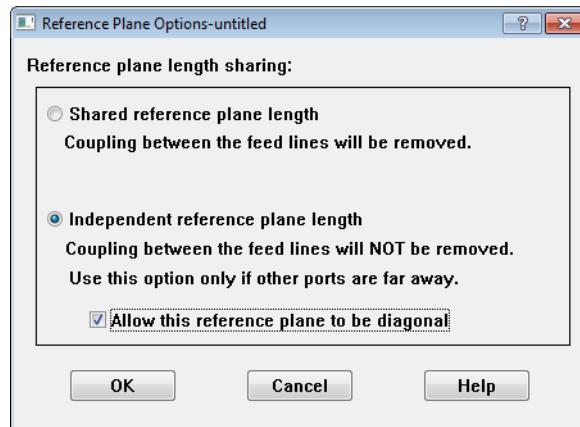


- 4 Click on the “Independent reference plane lengths” radio button.

This option is used to make the reference plane for this port independent of other reference planes for other ports. As mentioned previously, by default, all ports on the same box-wall (or same side for components or co-calibrated port groups) share the same reference plane length. This feature allows you to use a different reference plane length for this port, regardless of reference plane lengths for other ports, even ports which share a box-wall. A port cannot use the diagonal feature unless it is using an independent reference plane. When you select this option, the “Allow this reference plane to be diagonal” checkbox is enabled.

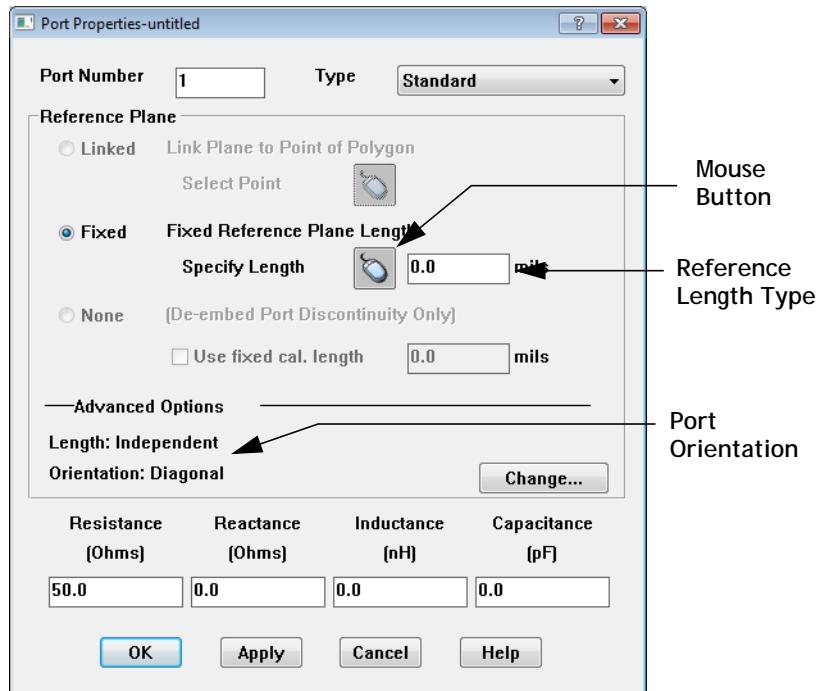
**5 Click on the “Allow this reference plane to be diagonal” checkbox.**

This option allows your reference plane to occur at any angle in your circuit, instead of being in x or y axis. The polygon to which the port is attached is used to determine the direction of the reference plane instead of the box-wall.



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

The Port Properties dialog box is updated and the Length field is updated to read “Individual” and the Orientation field to read “Diagonal.” The options for the reference plane have been reduced. For an independent port, you may only specify a fixed length reference plane.



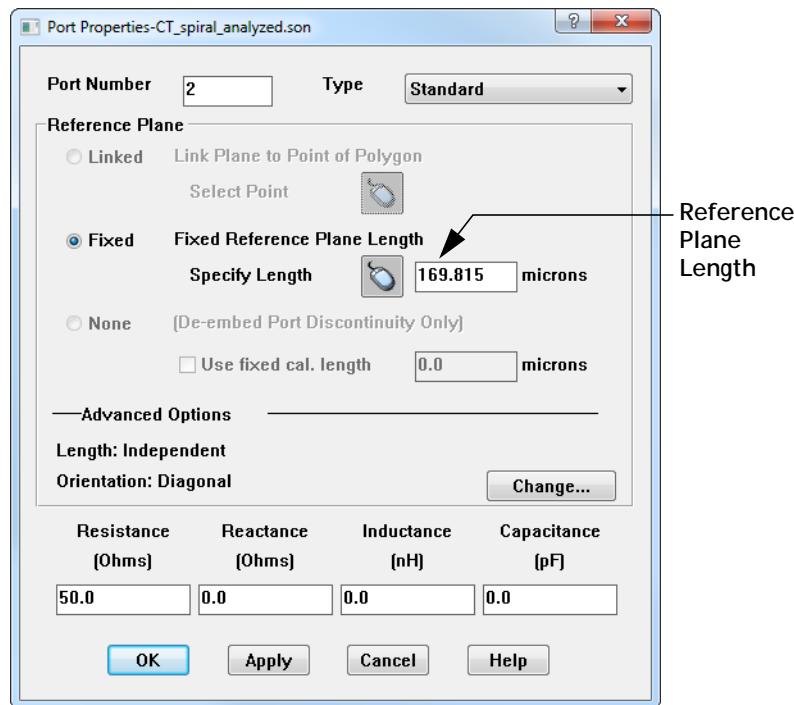
 **7 Click on the Mouse button to set the reference plane length for the port.**

Clicking on the mouse icon allows you to click in your circuit to define the reference plane. When you click on the icon, the dialog box disappears from your display so that you may access the window.

**8 Click at the desired location for the reference plane.**

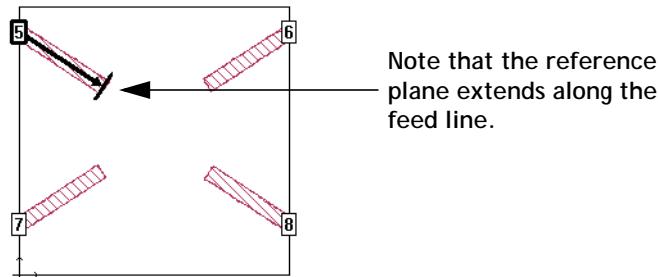
For this example, the reference plane is set at the end of the polygon to which the port is attached so that the transmission line effects for the whole feed line are removed during de-embedding. After clicking at the end of the polygon, the Port

Properties dialog box reappears with a length entered in the Ref. Plane text entry box. Note that you could have set the reference plane by entering a length into this text entry box.



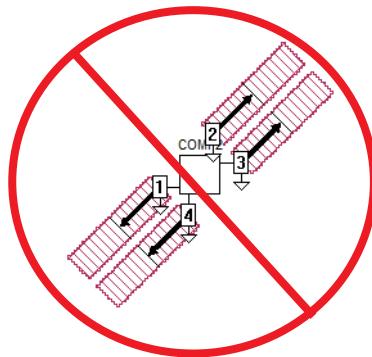
**9 Click on the OK button to close the dialog box and apply the changes.**

The reference plane is drawn, along the polygon, as shown below. This completes applying the diagonal feature to your port. To complete this design, you would follow the same procedure for the other three ports.



## Diagonal Port Feature Limitations

There are several limitations that the user should be aware of when applying the diagonal port feature. Once you select a reference plane as an individual reference plane any coupling between feed lines is no longer removed during the de-embedding process. Therefore, when using the diagonal port feature, care should be taken to place ports and their feedlines far enough apart that coupling effects are not significant. An example of the feedlines being too close to use diagonal ports is shown below.

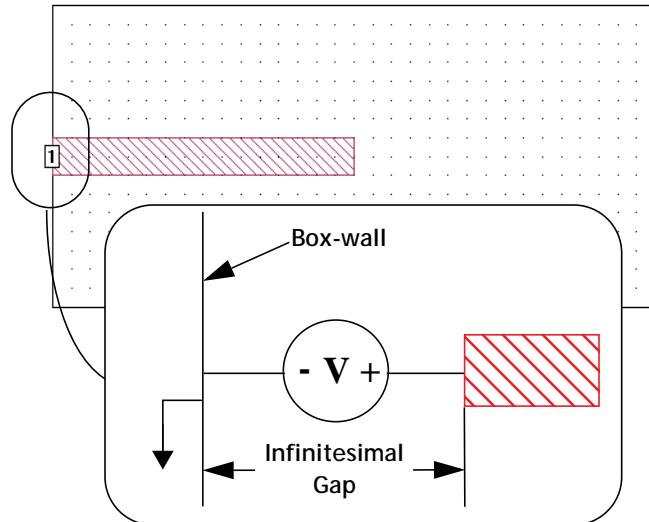


In addition, the *em* analysis cannot calculate the transmission line parameters  $E_{\text{eff}}$  and  $Z_0$  for a diagonal port since the port is not separated from the feed lines when de-embedded. However, the resulting response data is still accurate.

## Standard Box-wall Port

A standard box-wall port is a grounded port, with one terminal attached to a polygon edge coincident with a box-wall and the second terminal attached to ground. The port, therefore, breaks the connection between the polygon and the wall and is inserted in an “infinitesimal” gap between the polygon and the wall. The signal (plus) portion of the port goes to the polygon, while the ground (negative) portion

of the port is attached to the box-wall (ground). An example of a standard box-wall port is shown below. Standard box-wall ports can be de-embedded and can also have reference planes. This type of port is the most commonly used.



### Adding Box-wall Ports

You add a standard box-wall port to your circuit by selecting the command *Tools*  $\Rightarrow$  *Add Ports* and clicking on the polygon edge on the box-wall where you wish to place the port. Ports are numbered automatically, in the order in which they are added to your circuit, starting at the number one. You may change the properties of a port after it has been added to the circuit by selecting the port and using the *Modify*  $\Rightarrow$  *Port Properties* command. For detailed instructions for these tasks, please click on the Help button in the Modify Port Properties dialog box which appears when you select the command.

### Ref. Planes and Cal. Lengths for Box-wall Ports

Reference planes and calibration lengths are both used during the de-embedding process in which the analysis engine, *em* removes the port discontinuity and a desired length of transmission line. For details of how these values are used during the de-embedding process, please refer to **Chapter 6, “De-embedding”** on page 103.

For a box-wall port, you may set either a reference plane or a calibration length; both values cannot be set at the same time for box-wall ports. Ports can be shared in which case all shared ports on the same box-wall use the same reference length or calibration standard. If the port is set to individual, then a reference plane or calibration standard may be set that applies only to that port. Please note that if the port is individual, then the coupling between ports is not de-embedded; coupling is only removed between shared ports. To set either of these values for box-wall ports, use the *Modify*  $\Rightarrow$  *Port Properties* command. For details, please click on the Help button in the Port Properties dialog box which appears when you select the command.

## Co-calibrated Internal Ports

Co-calibrated ports are used in the interior of a circuit. These ports are often used by a circuit simulation tool to connect some type of element into your geometry at a later time outside the Sonnet environment. Co-calibrated internal ports are identified as part of a calibration group with a common ground node connection. When **em** performs the electromagnetic analysis, the co-calibrated ports within a group are simultaneously de-embedded using a high accuracy de-embedding technique<sup>1</sup>; thus, coupling between all the ports within a calibration group is removed during de-embedding. This type of port is the most commonly used internal port.

### Calibration Group Properties

A co-calibrated port is always associated with a calibration group. Co-calibrated internal ports are automatically added to a calibration group made up of ports in reasonable proximity who also share the same type of calibration group properties: ground node connection and terminal width. By default, a new co-calibrated port uses the Sonnet Box as the ground node connection and the Feedline Width as the terminal width. If these properties are subsequently changed, then this affects the ports with which the modified port is grouped. If you wish to observe the grouping determined by the software before running your analysis, you may select the command *View*  $\Rightarrow$  *View Subsections* from the project editor main menu.

It is also possible to manually define a calibration group which overrides the automatic settings. For more information, please refer to “calibration group” in Help’s index.

<sup>1</sup> “Deembedding the Effect of a Local Ground Plane in Electromagnetic Analysis,” by James C. Rautio, president and founder of Sonnet Software, Inc. The article is available in PDF format in the Support section of our web site.

For a detailed explanation of the rules used in grouping co-calibrated ports, please see "Rules for Co-Calibrated Ports" on page 94.

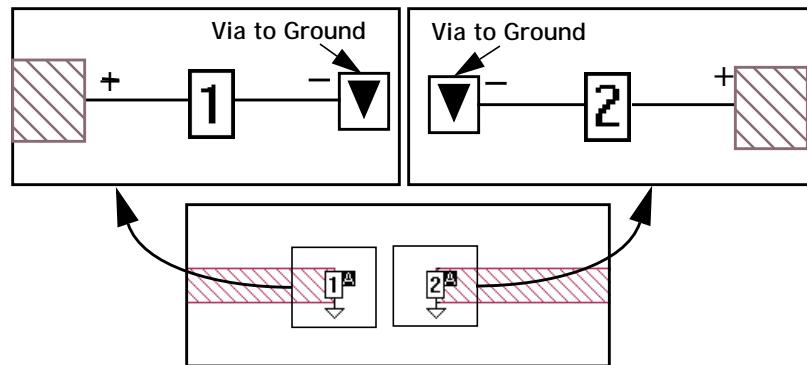
### Ground Node Connection

You must define how the common ground of your calibration group is connected to your circuit. There are two types of ground node connection: Sonnet Box and Floating.

**Sonnet Box:** When the ground node connection is defined as the Sonnet box, all co-calibrated ports in the group are globally grounded to the Sonnet box. To do this, a via is automatically created which connects to the top or bottom of the Sonnet box. You should choose this option if the element model or measured data to be connected to these ports includes shunt elements. Examples are:

- S-parameter data that includes pads or other coupling to ground.
- Transistor data that includes a via in the model/measurement.

This type of co-calibrated port is illustrated below. The positive terminal is attached to a polygon edge of a feedline and the negative terminal is attached to a via to ground which is removed during de-embedding.



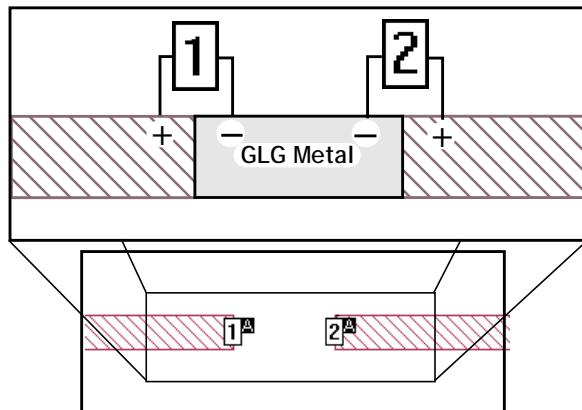
When the Sonnet box is selected as the ground node connection, the analysis engine automatically determines the most efficient direction the ground via extends taking into consideration both the distance and the loss of the box top or bottom box cover. When using this type of ground, you must make sure that there is a clear path with no metal on other levels interfering with the path to the box top or bottom.

In addition, either the box top or bottom must have loss less than 50 ohms/sq. For example, you should not use “Free Space” for both your box top and bottom definition. If the loss is too high on both the box top and bottom for a ground via from the calibration group to be attached, the analysis engine will issue an error message.

**Floating:** When the ground node connection is defined as Floating, all co-calibrated ports in the group are connected to a common ground but not to the Sonnet box. Instead, the ground node is left unconnected to the rest of your circuit. You should choose this option if the element model or measurement data that will be connected to these ports does not have a ground reference, or does not have shunt elements. Examples of this are:

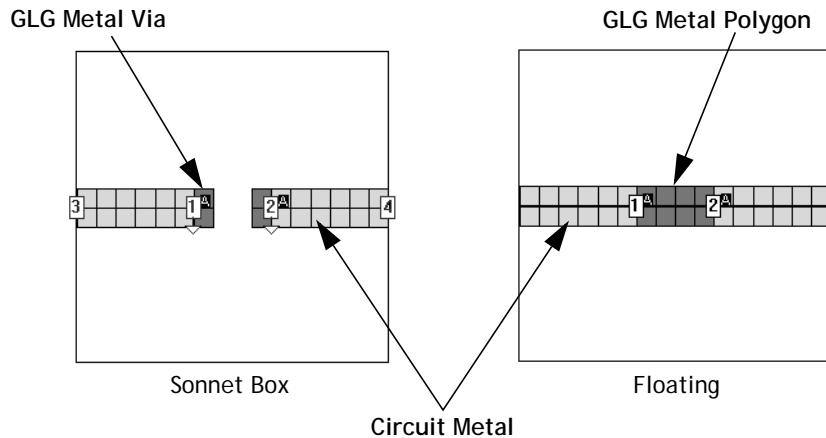
- A series RL equivalent circuit
- S-Parameter data that was measured without any pads

This type of co-calibrated port is illustrated below. Sonnet automatically adds extra metal which connects the co-calibrated ports in a calibration group. This extra metal is defined as Generalized Local Ground (GLG) metal since it acts as a local ground for the ports in the calibration group. This metal is removed during the de-embedding process. The positive terminal of a co-calibrated port is attached to a polygon edge of a feedline and the negative terminal is attached to GLG metal.



To observe the GLG metal for either type of ground node connection, you may select the *View*  $\Rightarrow$  *View Subsections* command in the project editor. When the Subsection Viewer appears, select *View*  $\Rightarrow$  *GLG Metal* from the subsection viewer’s main menu. Views of two calibration groups, as viewed in the subsection viewer

with the GLG metal displayed, are shown below. A calibration group which is grounded to the Sonnet box is shown on the left and one with a floating ground is shown on the right.

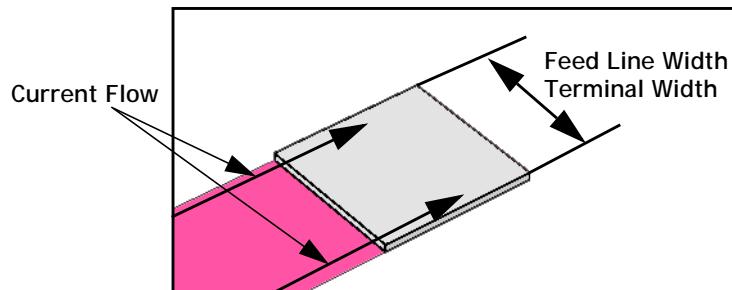


Note that in the case of the Sonnet box ground node connection, the GLG metal does not connect the two ports. Instead, there is a via composed of GLG metal which extends to ground.

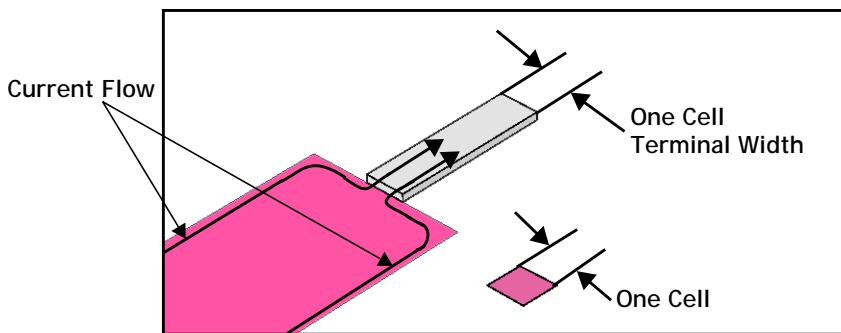
### Terminal Width

You must also determine how you will define the terminal width for your calibration group. Terminal width is the electrical contact width of the component which is to be attached. This allows the current flow from the circuit geometry into the component to be accurately modeled. There are three settings for terminal width: Feed Line Width, One Cell, and User Defined. Please note that the terminal width is not shown in the project editor.

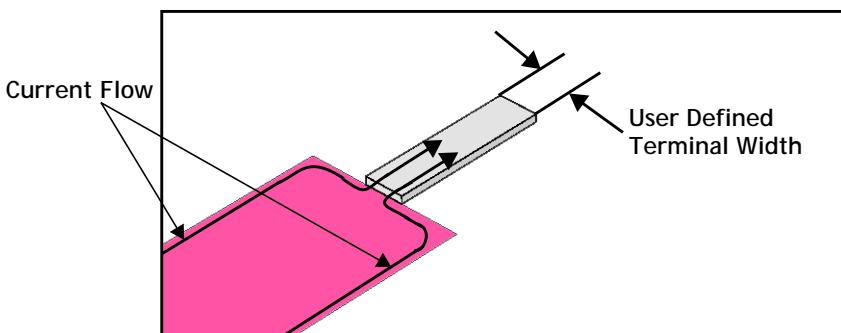
**Feed Line Width:** This defines the terminal width as equal to the feed line to which the co-calibrated port is attached. This is illustrated below.



**One Cell:** This defines the terminal width to the smallest possible size as pictured below.



**User Defined:** This allows you to enter any value desired for the terminal width. This type of Terminal Width is shown below.



## Adding Co-calibrated Ports

You add co-calibrated ports to your circuit by selecting the command *Tools*  $\Rightarrow$  *Add Ports* and clicking on an open polygon edge where you wish to place the port. Once the ports have been added to the circuit, you select the ports, then use the *Modify*  $\Rightarrow$  *Port Properties* command which allows you to select the co-calibrated port type. For detailed instructions on this process, please refer to “co-calibrated internal port” in Help’s index.

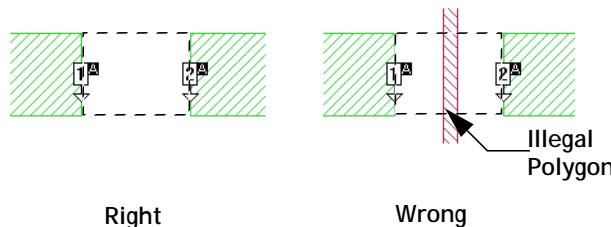
## Rules for Co-Calibrated Ports

The set of criteria used by the software when autogrouping co-calibrated ports is as follows:

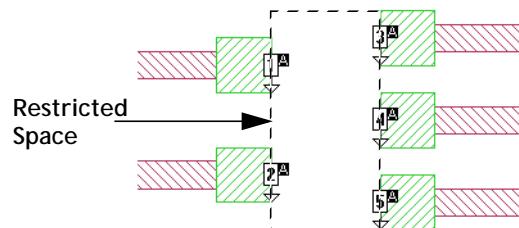
- Proximity of the ports
- Calibration Group Properties: ground node connection and terminal width (see "Calibration Group Properties" on page 89 for more details)
- Physical layout rules

The physical layout rules are detailed in this section. Note that these rules should also be followed when manually defining a calibration group. For detailed instructions on manually creating a calibration group, please refer to "co-calibrated internal port" in Help's index.

1. **No objects may be placed within the "restricted space" in the interior of the co-calibrated port group:** Metal polygons, vias, or dielectric bricks may not be present in the rectangular area defined by the port locations and the terminal width, as illustrated below. The bottom circuit shows the restricted area in a multiport Component. For the best possible accuracy, no objects should be present on any layer in this restricted space, even if the ports are not on these layers.



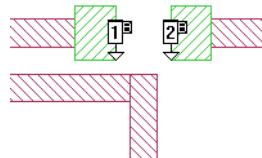
The dashed boxes identify the "restricted space" for each co-calibrated port group. On the circuit labeled "Wrong" a metal polygon passes through the middle of the restricted space.



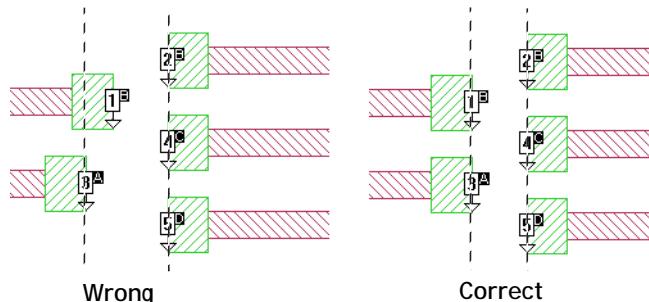
2. Co-calibrated ports whose ground node connections are defined as the Sonnet box require direct access to the Sonnet top or bottom cover: The analysis engine determines the most efficient direction the ground via extends taking into consideration both the distance and the loss of the box top or bottom. When using this type of ground, you must make sure that there is a clear path with no metal on other levels interfering with the path to either the box top or box bottom.

In addition, the box top or bottom should not have loss greater than 50 ohms/sq. If the loss is too high on both the box top or bottom for a ground via from the co-calibrated ports to be attached, the analysis engine issues an error message.

3. Nearby objects should be placed so that coupling between the co-calibrated ports and the object does not occur: Metal polygons, vias, or dielectric bricks which couple to the co-calibrated ports may decrease the accuracy of the analysis as shown below.

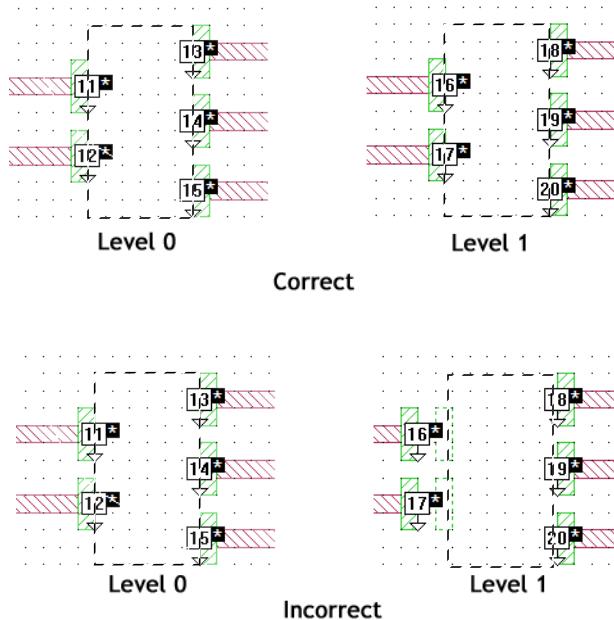


4. Multiple ports on the same side of the restricted area should be aligned: In the illustration below, the co-calibrated ports on the left are placed incorrectly and will produce an error during an analysis. The ports on the right are placed correctly.



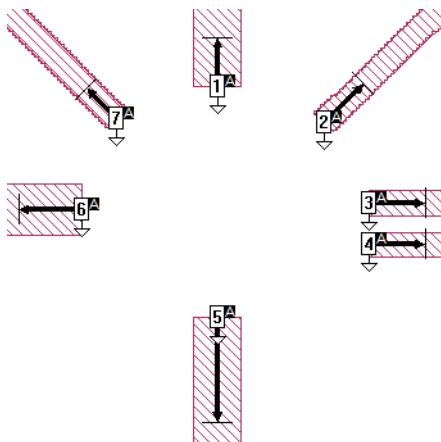
The dashed lines represent the alignment plane.

For multi-level co-calibrated ports, ports on different metal levels need to be aligned in the x, y plane. In the illustrations below, the multi-level calibration group on top is correct. The one on the bottom is incorrect since the ports on the left side of metal level 0 are not aligned with the ports on the left side of metal level 1



5. Reference planes for each side of a co-calibrated port group are shared and therefore use the same reference plane length: However, reference planes can be set independently for each side of the “port rectangle” as shown below. If a co-calibrated port is attached to a polygon which does not lie in the x or y plane, then the port defaults to an independent, diagonal port and the reference plane is defined only for that port. In the graphic below, ports 3 and 4 are shared orthogonal ports which use the same reference plane length since they are on the same orthogonal side of the port group. Ports 1, 6, and 5 are all shared orthogonal ports but may use different reference lengths since they are all on different sides of the port group. If a port was added to the left side of the port group, it would share a reference plane length with port 6. Both port 2

and port 7 are independent diagonal ports whose reference lengths may be different from each other and different from any of the shared ports on an orthogonal side of the port group.



## Ref. Planes and Cal. Lengths for Co-calibrated Ports

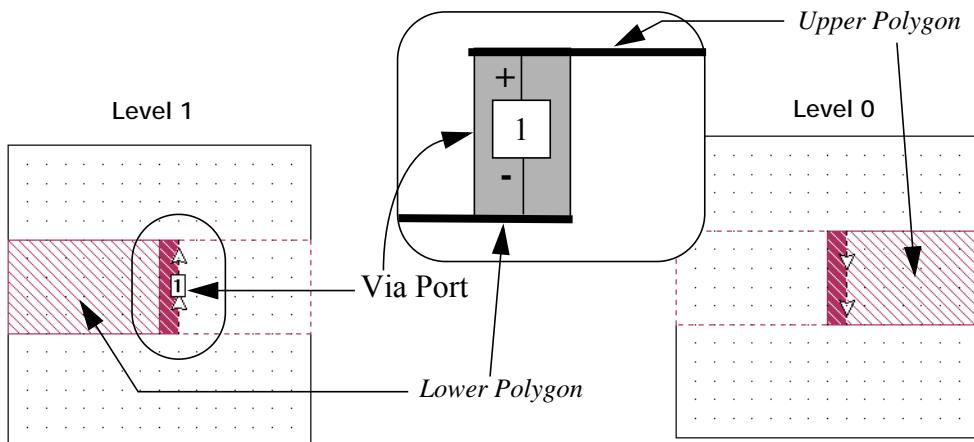
You may specify reference planes for ports in a co-calibrated port group as discussed in rule 5 above. Calibration lengths are automatically calculated by the software. For a detailed discussion of how reference lengths and calibration lengths are used during the de-embedding process, please refer to Chapter 6 "De-embedding" on page 103. For directions on how to set a reference plane and/or calibration length for a co-calibrated port, please refer to "calibration group" in the index of Help.

## Use in Components

Co-calibrated ports are used to implement the Component feature. Many of the considerations which apply to the setup of a Component also apply to the setup for a calibration group. To gain a deeper understanding of the co-calibrated ports implementation and uses, we highly recommend that you also read Chapter 6 "De-embedding" on page 103.

## Via Ports

A via port has the negative terminal connected to a polygon on a given circuit level and the positive terminal connected to a second polygon on another circuit level. A via port can also have the negative terminal connected to the top or bottom of the box. An example of this port type on an edge via is shown below.



An example of a circuit with a standard via port. A side view of the enclosed area on the circuit is shown in the middle.

Unlike co-calibrated ports, **em** cannot de-embed via ports. However, in a circuit which contains a combination of via ports and other port types, the other port types can still be de-embedded. **Em** will automatically identify all of the other ports present in the circuit and de-embed them, but leave the via ports un-de-embedded.

The example file “stackedpatch” has an example of a via port used in a patch antenna. This example file can be found in the Sonnet example files. You may access the Sonnet Example Browser by selecting *Help* ⇒ *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window.

In most cases where you need grounded ports, your first choice would be to use co-calibrated ports (as discussed earlier), especially since it is possible to accurately de-embed co-calibrated ports. The most common case where a via port would be used is when you wish to attach a port between two adjacent levels in your circuit. Another application is when you wish to connect a port to the interior of a polygon, which is not allowed for co-calibrated ports which must be added to an open polygon edge.

You can not apply reference planes to via ports, since it is not possible for *em* to de-embed them.

## Adding Via Ports

To add a via port, you must first create a via. The via can be an edge-via or a via polygon. For more details on how to create vias, see “vias” in Help’s index.

Once the via is in place, click on the Add a Port button in the tool box and click on the via to add a port.

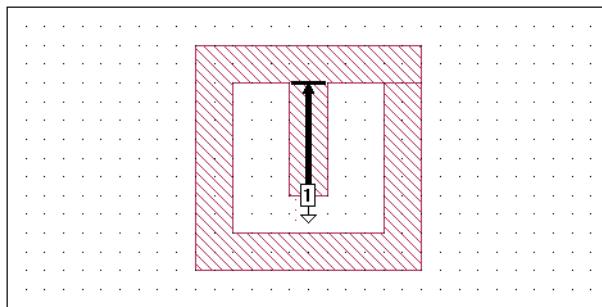
---

**NOTE:** The via port only appears in the project editor on the bottom level of the via; if you are adding the port to any other level of the via, the port is not displayed in the project editor. To see the port, go to the metal level on which the bottom of the via is placed.

---

## Automatic-Grounded Ports

An automatic-grounded port is a special type of port used in the interior of a circuit similar to a co-calibrated internal port. This port type has the positive terminal attached to the edge of a metal polygon located inside the box and the negative terminal attached to the ground plane through all intervening dielectric layers. An auto-grounded port with a reference plane is shown below. Auto-grounded ports can be de-embedded by the analysis engine.



In most circuits, the addition of co-calibrated ports has little influence on the total analysis time of the *em* job. However, for some circuits, co-calibrated ports may require more time in the de-embedding phase of the analysis. In these cases, you

may wish to use auto-grounded ports, which are more efficient but less accurate than co-calibrated ports, since the de-embedding for auto-grounded ports does not take into account the coupling between the auto-grounded ports.

## Special Considerations for Auto-Grounded Ports

### Metal Under Auto-Grounded Ports

Similar to co-calibrated ports which use the Sonnet box as the ground node connection, you cannot have metal directly beneath an auto-grounded port in a multi-layer circuit. Auto-grounded ports are two-terminal devices with the positive terminal connected to an edge of a metal polygon and the negative terminal connected to the ground plane. When **em** detects the presence of an auto-grounded port, it automatically connects the two terminals in this manner. This includes circuits which have multiple dielectric layers between the polygon and the ground plane. However, in order for **em** to accomplish this, there must be a direct path from the edge of the metal polygon to the ground plane. When an auto-grounded port is used in a circuit where there is more than one dielectric layer between the port and the ground plane, **em** checks to make sure that there is no metal directly beneath the auto-grounded port. If metal is found, **em** prints an error message and stops.

### Edge of Metal Polygon is Lossless

Auto-grounded ports can attach to the edge of any metal polygon in the interior of a circuit. There are no restrictions on the loss parameters of the metal used in the polygon. However, along the edge of the metal polygon where the port is attached, **em** does force the cells to be lossless. For most circuits, this should have little or no effect on the results. If, however, the port is attached to a highly lossy metal polygon, such as a thin-film resistor, the edge cell(s) of that polygon will be made lossless, and the output results may be affected.

### Using Negative Port Numbers with Autogrounded Ports

You are able to assign negative port numbers to autogrounded ports but should be aware of some special considerations when doing so. The two autogrounded ports (+n and -n), although associated by port number, are still de-embedded separately so the coupling between the ports is not de-embedded which can affect the accuracy of your data. Because these ports are de-embedded separately, the port impedance,  $Z_0$  only accounts for the impedance of the single line attached to the autogrounded ports since the calibration standard does not include both lines of the differential pair. Lastly, using reference planes in this case may also affect your data, since reference planes are used to de-embed a single transmission line while this configuration creates a differential transmission line.

## Adding Auto-grounded Ports

You add an auto-grounded port to your circuit by selecting the command *Tools* ⇒ *Add Ports* and clicking on the open polygon edge where you wish to place the port. Once the port has been added to the circuit, you select the port, then *Modify* ⇒ *Port Properties*. In the Port Properties dialog box which appears, you select the auto-grounded type. For detailed instructions on this process, please refer to “auto-grounded ports” in Help’s index.

## Ref. Plane and Cal. Length for Autogrounded Ports

To set a reference plane for an auto-grounded port, use the *Modify* ⇒ *Port Properties* command. Calibration standards are automatically calculated by the software. You enter the desired Reference Plane in the appropriate field. For detailed instructions please refer to Help. Note that if you enter a reference plane, it applies only to the selected port(s). For details on how reference planes and calibration lengths are used in the de-embedding process, please refer to Chapter 6 "De-embedding" on page 103.



### TIP

---

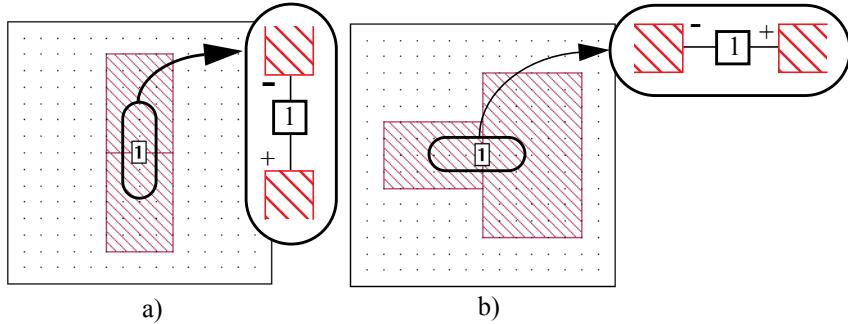
Changing a port to an autoground type and setting up a reference plane for the port can be accomplished at the same time in the Port Properties dialog box. It is also possible to set reference planes for multiple auto-grounded ports by selecting the desired ports, selecting *Modify* ⇒ *Port Properties* and inputting a value in the Ref. Plane text entry box in the Port Attributes dialog box.

---

## Ungrounded Internal Ports

An ungrounded internal port is located in the interior of a circuit and has its two terminals connected between abutted metal polygons. An ungrounded internal port is illustrated below. Note that internal ports have their negative terminals on the left or top side of the port depending on how the port is oriented. Ungrounded internal ports can be de-embedded by *em*, however, you may not set a reference

plane or calibration length. Ungrounded internal ports are not as accurately de-embedded as co-calibrated internal ports, but they do not require any space between the two polygons as is required for a co-calibrated port.



Ungrounded internal ports are not allowed on the edge of a single polygon because this would leave one terminal of the port unattached. Also, care should be taken in interpreting the results for circuits which use these ports since the ports do not all access a common ground.

You add an ungrounded-internal port in the same manner that you add a standard box-wall port; for details, see “Adding Box-wall Ports,” page 88.

# Chapter 6

# De-embedding

---

## Introduction

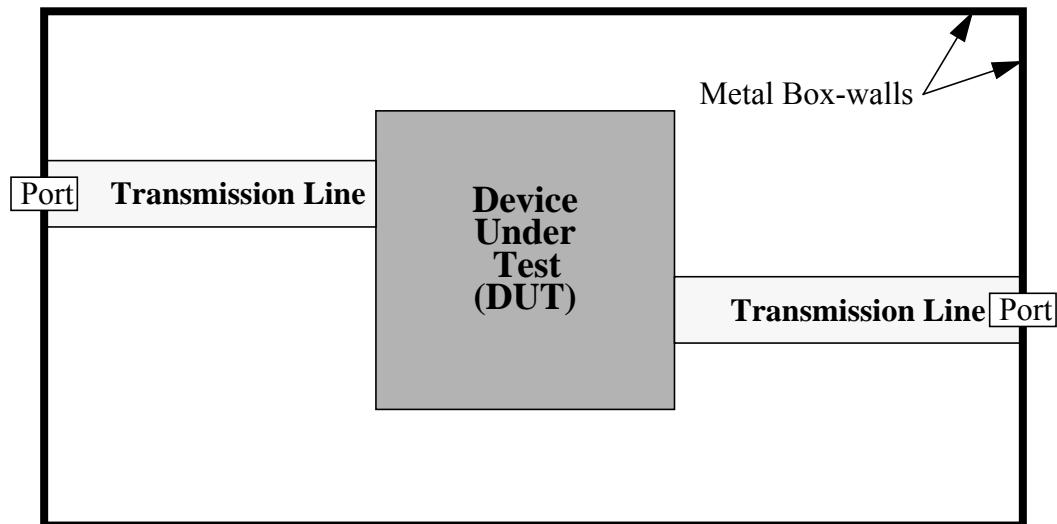
De-embedding is the process of calibrating the ports, similar to a calibration of an RF measurement system with the probes or connectors used in the measurement. Sonnet de-embedding is performed automatically, behind the scenes, and in most cases, is not a concern for the user. However, just as in an RF measurement, understanding of the inner workings of the process can sometimes allow you to design your circuits in such a way as to produce more accurate results or analyze your circuit more efficiently.

Each port in a circuit analyzed by *em* introduces a discontinuity into the analysis results. In addition, any feed transmission lines that might be present introduce phase shift, and possibly, impedance mismatch and loss. Depending upon the nature of your analysis, this may or may not be desirable. De-embedding is the process by which the port discontinuity and feed transmission lines are removed from the analysis results.

The figure on page 104 illustrates the general layout of a circuit to be analyzed with *em*. The device under test (DUT), shown as a box in the figure, is the circuitry for which we wish to obtain analysis results. The DUT is located inside the metal box and is connected to one or more ports. The ports may be located on box-walls, as in the figure, or in the interior of the metal box (see [Chapter 5](#) for a description of port types available in *em*). Typically, feed lines are necessary to connect the ports to the DUT.

When de-embedding is enabled, **em** performs the following sequence of steps:

- 1 Calculates port discontinuities.
- 2 Removes effects of port discontinuities from analysis results.
- 3 Optionally shifts reference planes (negates effects of feed transmission lines from analysis results).
- 4 For most port types, calculates transmission line parameters  $E_{eff}$  and  $Z_0$  for the feed transmission lines. For more information, please see "[Plotting Z<sub>0</sub> and E<sub>eff</sub>](#)" on page 126.



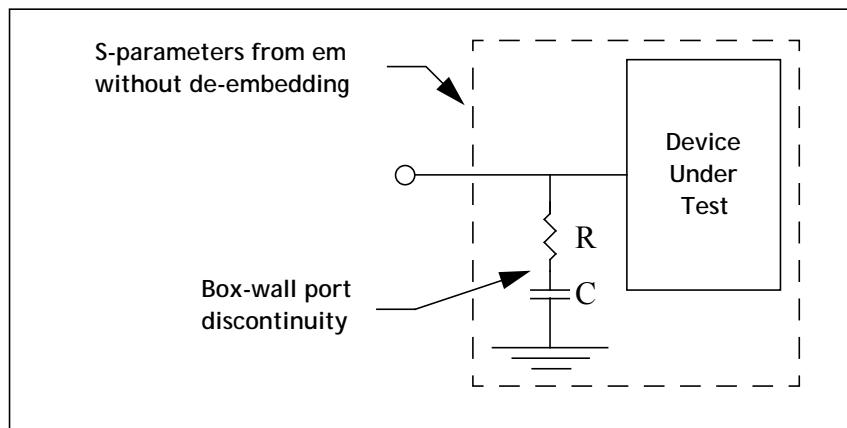
General layout of a circuit to be analyzed with **em**.

Upon completion of the de-embedding process, **em** outputs de-embedded S-parameter results.

The de-embedding algorithm uses a Short Open Calibration (SOC) technique. For an in-depth discussion of the calibration technique, please see reference [4] and for a discussion of the de-embedding technique used for co-calibrated ports see reference [6] in Appendix II, “Sonnet References.” Both of these articles are available in PDF format on the Sonnet web site.

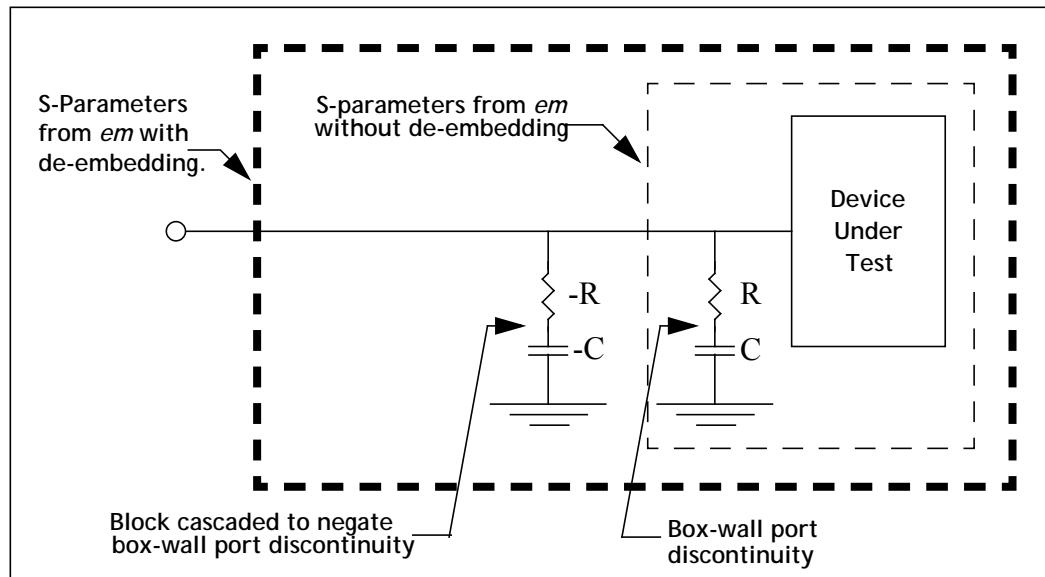
## Port Discontinuities

The port discontinuity for box-wall ports is described in the section that follows. The discontinuity for the other types of ports may be different, but the process outlined below is similar for all port types. Box-wall ports have one port terminal connected to a polygon inside the metal box, and the second port terminal connected to ground (see the figure on page 88). The port discontinuity is modeled as a series resistor, R, and capacitor, C, shunted to ground as shown below. If the circuit being analyzed is completely lossless, the resistor value, R, is zero. Even with loss in the circuit, the capacitive reactance is normally very large compared to the resistance.



Port discontinuity associated with a box-wall port.

When enabled, *em*'s de-embedding feature automatically calculates the values of R and C for each box-wall port present in the circuit. The port discontinuities are then removed by cascading a negative R and C as illustrated below.



De-embedding automatically cancels the discontinuity associated with a box-wall port.

## Shifting Reference Planes

Feed lines are required in many circuits to connect ports to the device under test (DUT). If the length of a transmission line is more than a few degrees relative to a wavelength, unwanted phase (and possibly loss) will be added to the results. If the impedance of the transmission line differs from the normalizing impedance of the S-parameters (usually 50 ohms), an additional error in the S-parameters results. Thus, if we are only interested in the behavior of the DUT, any “long” feed lines connecting the ports to the DUT should be negated during de-embedding. The process of negating lengths of transmission line during de-embedding is known as “shifting reference planes”.

Reference planes may be specified in the project editor for box-wall, co-calibrated, and auto-grounded ports, but not ungrounded-internal and via ports. When your circuit contains a reference plane, and de-embedding is enabled, **em** automatically builds and analyzes the calibration standards necessary to de-embed the port and shift the reference plane by the specified length.

---

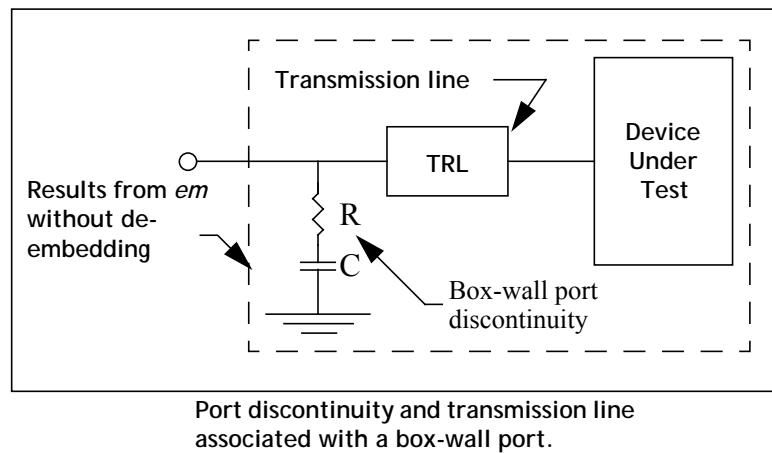
NOTE:

Reference planes are not necessary for de-embedding. If you do not specify a reference plane in the project editor, the reference plane length defaults to zero. This means that *em* will not shift the reference plane for that port when de-embedding is enabled. However, *em* will negate the discontinuity for that port.

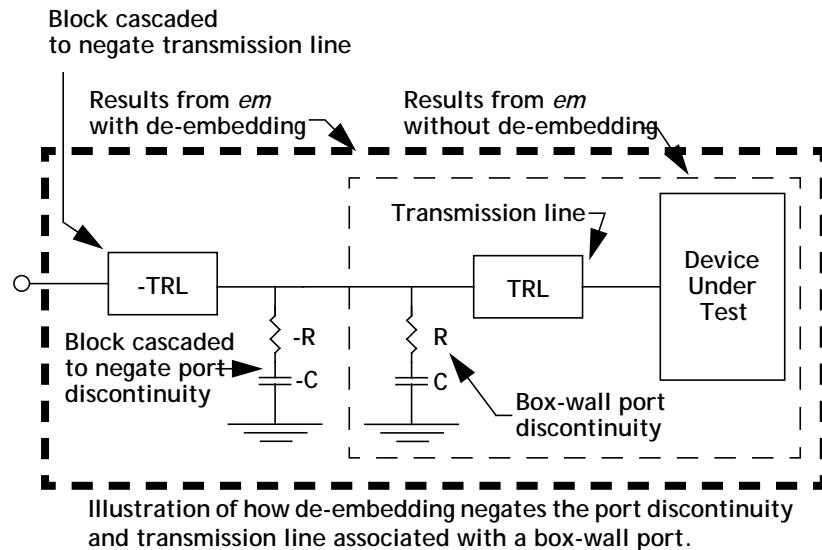
---

## Single Feed Line

Continuing with the box-wall port example, the figure below shows a circuit with a length of transmission line, TRL, inserted between a box-wall port and the device under test.

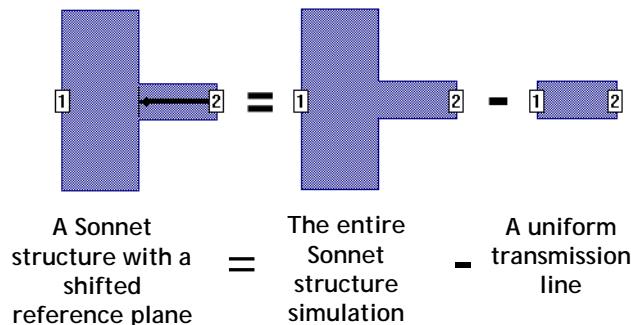


When the circuit has a reference plane, ***em*** negates the transmission line in a manner similar to that used to remove the port discontinuity. ***Em*** calculates S-parameters for the line alone, and then cascades a “negative” line along with negative R and C as illustrated in the next figure.

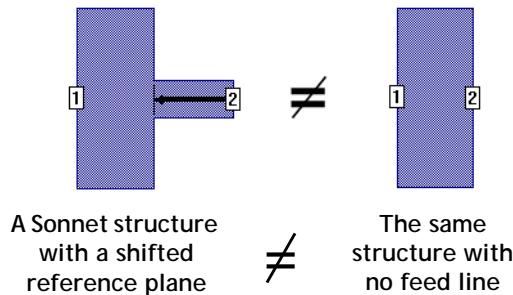


## A Common Misconception

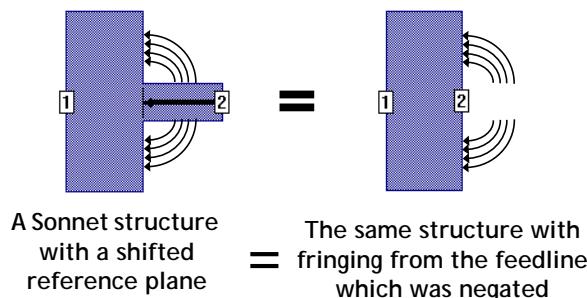
As explained above, a shifted reference plane negates the effects of one or more transmission lines using circuit theory after the simulation of the entire circuit. This concept is shown in the picture below:



Notice that reference planes do not remove the transmission line metal, but only negate it. Shifting a reference plane is not the same as removing the metal from the simulation:



A shifted reference plane does not remove the fringing fields due to the step in width. This concept is illustrated below. Note that the fringing fields still exist in the final Sonnet results after de-embedding.



## Rules for Reference Planes

The Sonnet project editor places very few restrictions on the reference planes which may be defined for a given circuit. This is done intentionally so as to provide maximum flexibility for all users. However, there are some basic guidelines concerning reference planes that should almost always be followed. These guidelines are discussed below.

### Feed Lines Must Be Transmission Lines

The first guideline is to ensure that your feed lines are true TEM or quasi-TEM transmission lines. This means the following:

- Feed lines must be of uniform width across the entire distance which the reference plane covers.
- Feed lines must not over-mode.
- The feed line(s) must not have an unusually high amount of loss. The metal must be conductive enough to allow power to reach the other side of the line. In addition, the dielectric must be resistive enough so the power is not shorted to ground before it reaches the other side.

## De-embedding Without Reference Planes

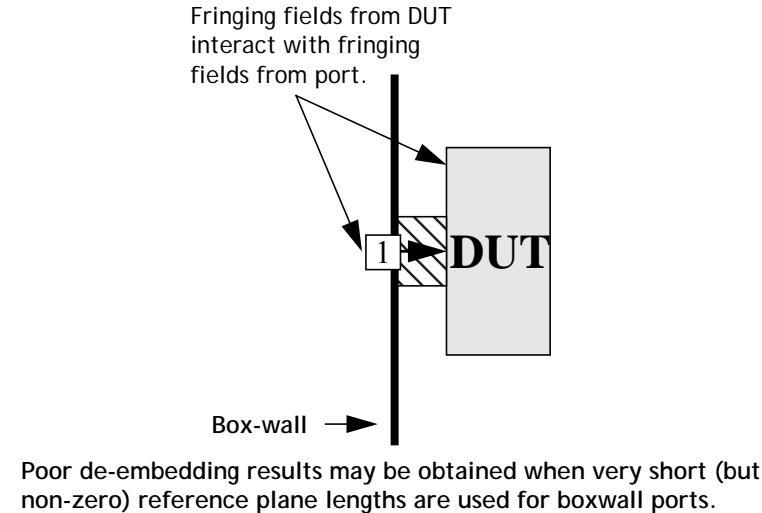
Reference planes are optional for co-calibrated, auto-grounded, and orthogonal box-wall ports but are required for diagonal boxwall ports. If you do not specify a reference plane for a particular port in the project editor, **em** will assume a zero-length reference plane for that port. This means that de-embedding will remove the discontinuity associated with that particular port, but will not shift the reference plane for it.

As discussed in the next section, **em** may generate bad de-embedded results if you attempt to remove a very short (but greater than zero) reference plane for a box-wall port. However, if you de-embed without a reference plane, **em** will not attempt to negate any length of transmission line at all. As a result, de-embedding without a reference plane does not lead to any error. Therefore, we recommend that you de-embed without reference planes rather than specify very short, non-zero, reference plane lengths for orthogonal boxwall ports.

## Short Reference Planes for Boxwall Ports

While auto-grounded and co-calibrated ports do not have a minimum length for their reference planes, short reference planes for boxwall ports, both orthogonal and diagonal, can cause problems with your de-embedded results. If the reference

plane for a boxwall port is very short relative to the substrate thickness or the width of the transmission line, *em* may generate poor de-embedded results. This is due to one or both of the reasons discussed below.



As shown above, the port is too close to the device under test (DUT). There are fringing fields associated with the port and separate fringing fields associated with the DUT. If the port and DUT are too close, the fringing fields interact. The de-embedding algorithm (which is virtually identical to algorithms used in de-embedding measured data) is based on circuit theory and cannot handle fringing field interaction. See reference [68] in Appendix II, “Sonnet References” for a detailed description of the problem.

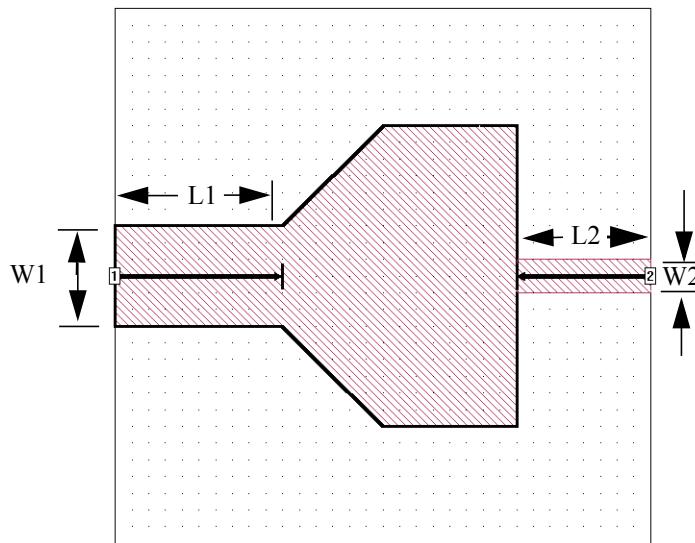
The other possible problem is that the calibration standard is too short: The calibration standard must be long enough such that the currents on the line(s) behave like standard transmission line(s). Since the calibration standard length for a boxwall port is based on the reference plane length, a short reference plane can result in inaccurate results. For more details about this problem, see ["Modifying Calibration Standard Lengths" on page 118](#).

There is no precise rule as to how long a boxwall port’s reference plane must be made in order to prevent the above effects from corrupting the de-embedded results. The required length is dependent upon the circuit geometry and the nature of the analysis. However, we recommend that you use reference planes equal to or greater than several times the distance to the nearest ground metal. This is sufficient for most types of analyses.

## Reference Planes Longer than the Feedline

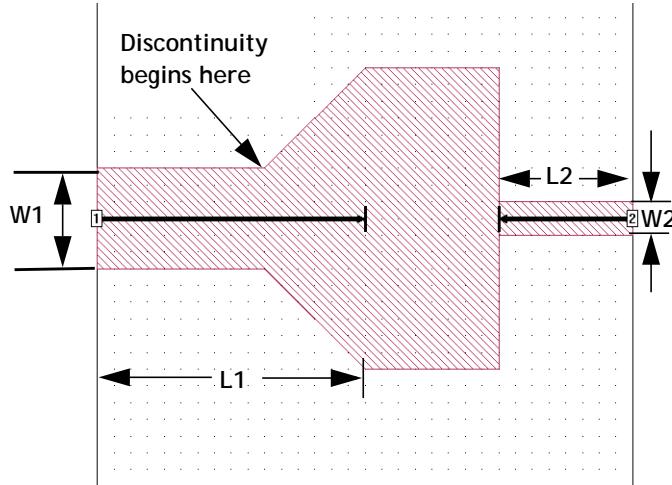
Generally, reference planes should not be longer than the feed line(s) attached to the port. This is because a reference plane is “negating” a transmission line. Thus if your reference plane is longer than your feedline, you are effectively attempting to negate a part of a transmission line which does not exist. Doing so may result in non-physical S-parameters.

To illustrate this problem, consider the valid circuit shown below. In this circuit, the reference planes are the same length as the feedlines. When de-embedding is enabled, the port 1 discontinuity is negated along with a transmission line of width  $W_1$  and length  $L_1$ . Similarly, the port 2 discontinuity is negated along with a transmission line of width  $W_2$  and length  $L_2$ . The de-embedded result is a set of 2-port S-parameters for the block in the middle of the circuit which is usually the desired data.



Now, consider the figure on page 113. This circuit is identical to the circuit shown above except that the length of the reference plane associated with Port 1 has been increased. If **em** is run with de-embedding enabled on this circuit, it “negates” a length of transmission line equal to the specified reference plane length. This occurs even though the actual feed transmission line is shorter than the reference

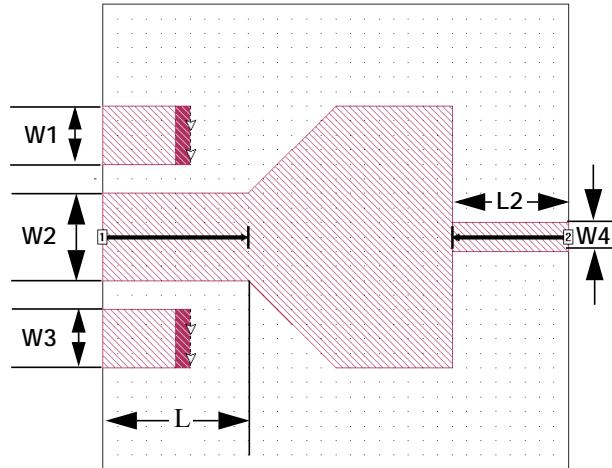
plane length. As a result, the de-embedded S-parameters may be non-physical (i.e., the results may not obey the laws of physics by being non-passive or non-causal).



Example circuit for which non-physical results may be obtained when *em* is run with de-embedding on.

A second de-embedding example leading to non-physical S-parameter results is shown in the next figure. In this example, the circuit has two via pads on each side of the port transmission line. The via pads are grounded to the box-wall.

When **em** is run with de-embedding enabled on this circuit, it “negates” three coupled feed lines with a length equal to the reference plane length. Since the reference plane extends from the box-wall beyond the vias, the de-embedded results may be non-physical.

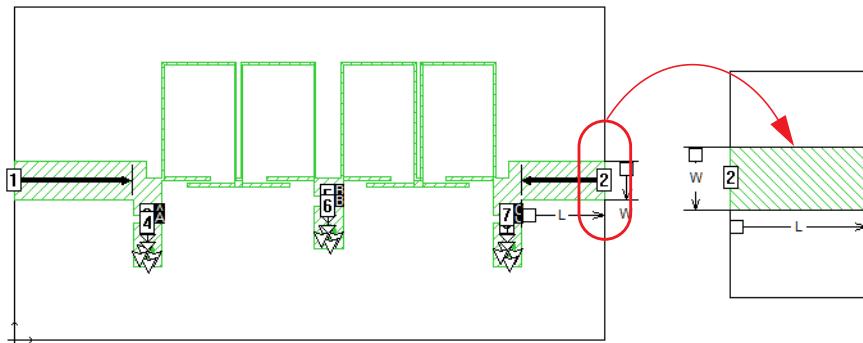


Analyzing this may result in non-physical results because the reference plane is longer than the coupled feedline.

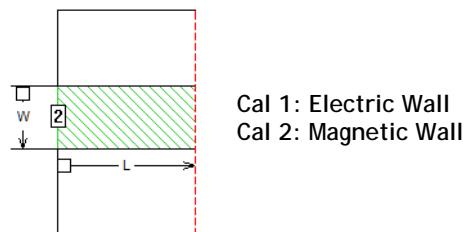
## Calibration Standards

A user will rarely need to set their own calibration standard, normally using the lengths automatically calculated by **em**. However, understanding calibration standards can help you simulate your circuit in the most efficient way while ensuring the accuracy of your response data.

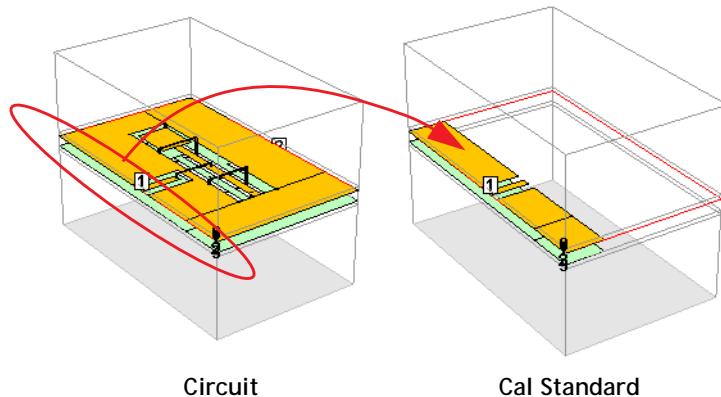
In order to de-embed a port discontinuity, Sonnet must electromagnetically analyze several calibration standards which include the same port discontinuity as is in the primary circuit. For example, to de-embed port 2 in the circuit below, the calibration standards contain a transmission line with the same line width as the primary circuit.



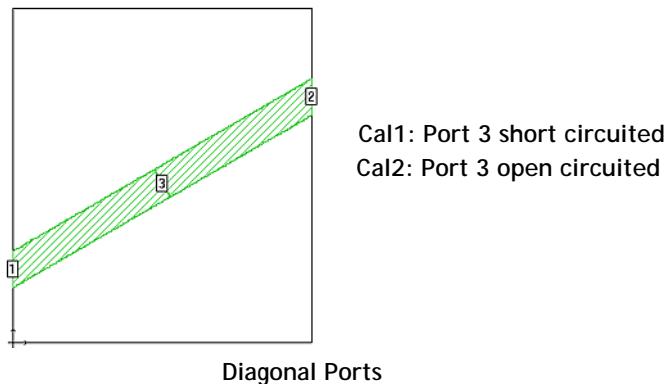
For box-wall ports, Sonnet uses the Short Open Calibration (SOC) technique which means a short and open standard must be analyzed. For orthogonal ports (ports which are connected to transmission lines which are perpendicular to a Sonnet box-wall), the two calibration standards look like the picture below, where the dashed line is replaced with an electric wall (for the short circuit) and a magnetic wall (for the open circuit).



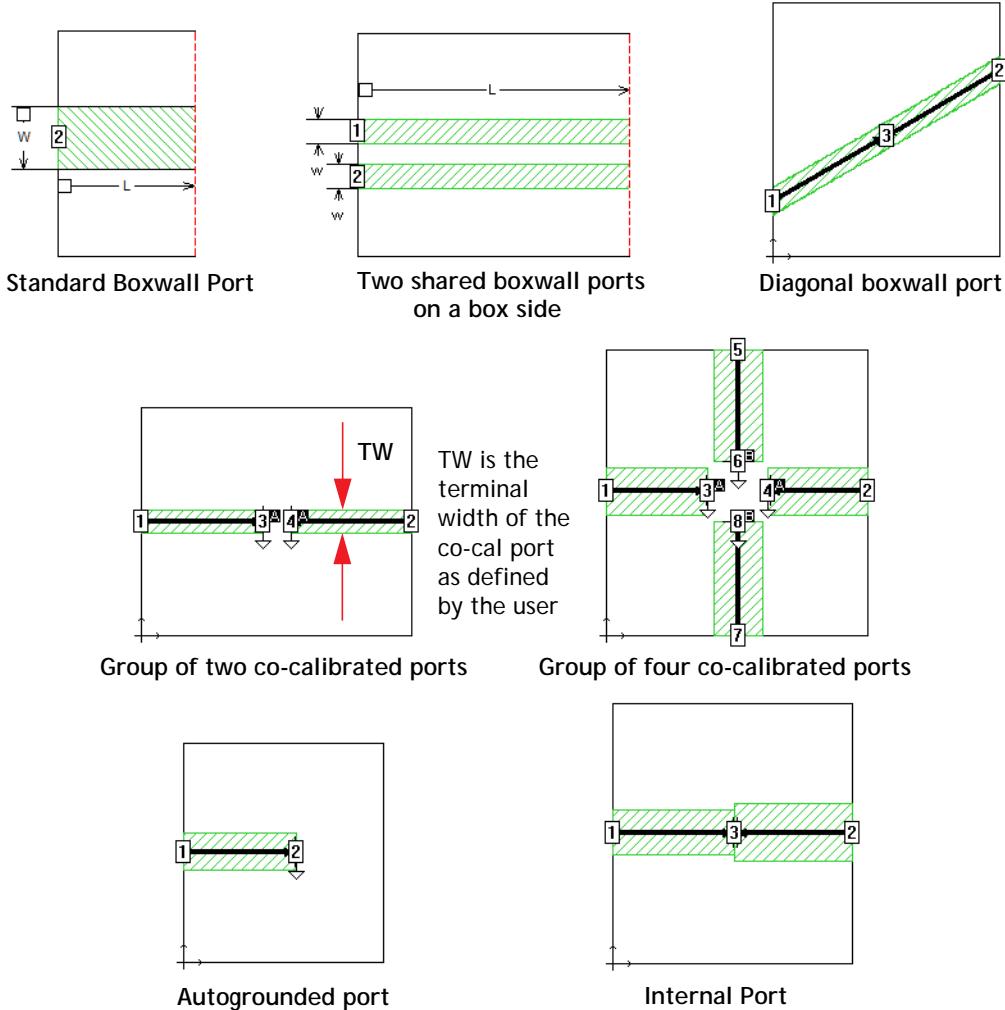
For box-wall ports, any object that touches the same wall as the port will be included in the calibration standard. This rule applies to vias, dielectric bricks, and metal on other levels as shown in the example below. This metal is included in the calibration standard.



For diagonal ports, Sonnet uses the single three-port calibration standard shown below, where the short and open data are generated by a single analysis, and by short-circuiting and open-circuiting port 3.



Shown below are some examples of the calibration standards used for various port types. The calibration length for the boxwall ports is “L” and for all other ports the calibration length is the same length as the reference plane.

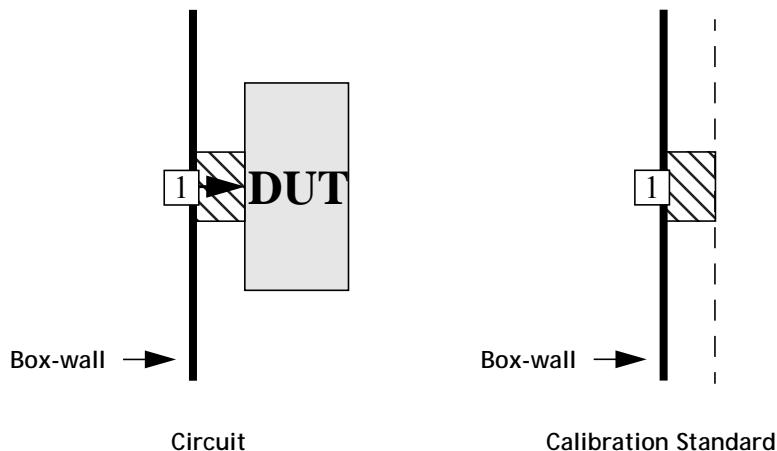


For any of the pictures above which show a reference plane, **em** first analyzes the structure without the reference planes, then simulates additional standards (not shown) to remove the port discontinuity and shift the reference plane, just as it would if this were your primary structure. For example, in the two-port co-calibrated port example above, Sonnet first analyzes the structure shown in the picture, and then analyzes the boxwall port calibration standards in order to de-embed up to the co-calibrated ports.

## Modifying Calibration Standard Lengths

Sometime the calibration standard automatically determined by *em* is longer than necessary, which can increase the analysis time. In most cases, de-embedding is not a large part of the analysis, so the effect of a longer calibration standard is negligible. However, where the calibration standard is comprised of a significant amount of metal, as for example, coplanar waveguide circuits, longer calibration standards can impact analysis time.

There are also cases in which the automatically determined calibration standard is too short. This can cause errors in your de-embedded results since the line must be long enough to allow the currents flowing on the line to exhibit a traditional TEM mode (see "[Feed Lines Must Be Transmission Lines](#)" on page 109). A box-wall port with a calibration standard which is too short is illustrated below. Notice the length of the calibration standard is even shorter than the width.

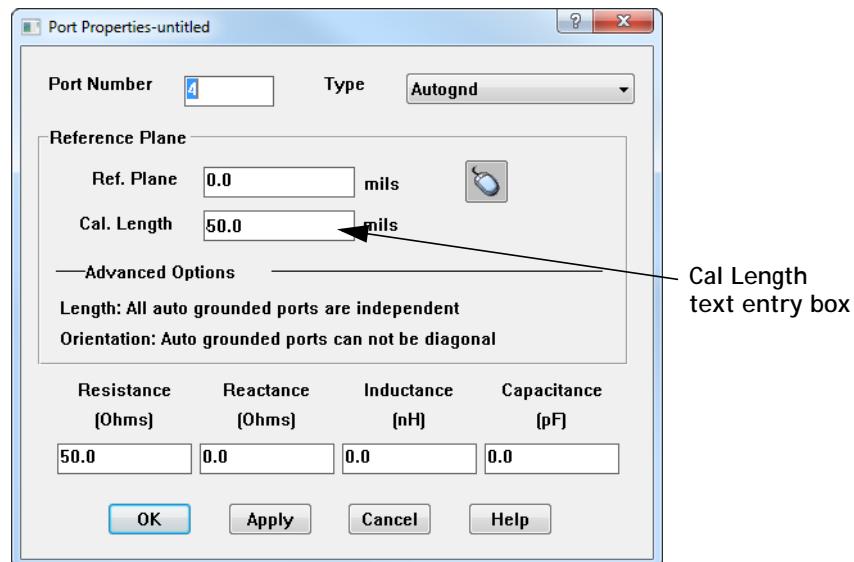


In these cases, you may wish to override the automatically determined length. The table below shows for which port types you may set the calibration length manually.

Port Type	Default Cal Length	Manual Setting of Cal Length?
Standard Boxwall with reference plane	Reference plane length	Yes: Change reference plane
Standard Boxwall without reference plane	Automatic	Yes: Enter Cal Length in the Port Properties dialog box
Diagonal Boxwall	Reference plane length	No
Co-calibrated	Auto	Yes: Enter Cal Length in the Port Properties dialog box
Autogrounded	Auto	Yes: Enter Cal Length in the Port Properties dialog box
Internal Ungrounded	Auto	No

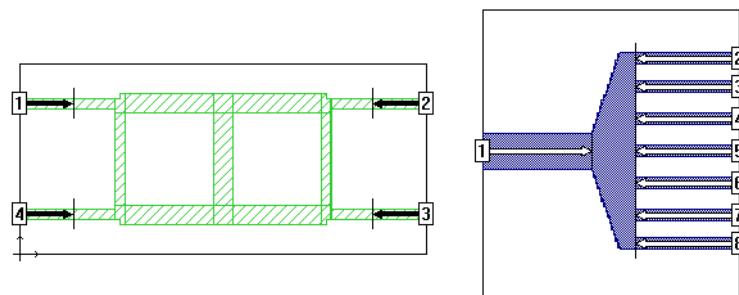
The second column shows how the default calibration length is determined. The third column shows whether a manual setting is possible and if so, how to input the desired length. For example, for standard boxwall ports which have a reference plane, changing the reference plane length changes the calibration standard

length. For ports which allow you to change the calibration length, double-click on the port and enter the desired value in the Cal Length text entry box as shown below.



## Coupled Transmission Lines

Sometimes, your circuit may contain multiple parallel feed lines as shown in the circuits below:



There are two issues in the above examples:

- 1 The ports are close to each other. When ports are close to each other, their fringing fields can couple to each other. If the ports are not calibrated at the same time, this would result in errors. Fortunately, Sonnet uses coupled lines for the calibration standards (by default), so all cou-

pling between the fringing fields of the ports is removed in the de-embedding process (see the figure on [page 122](#).)

- 2** The feed lines are close to each other. When feed lines are close to each other, they can couple to each other. If you are using reference planes, this coupling should be removed. Since calibration standards are coupled lines, this coupling is removed by default.

Both of these issues are resolved by using “shared” reference planes. When the reference planes are set to “shared,” coupled lines are used for calibration standards, thus removing the coupling between the ports and the feed lines. The alternative to shared reference planes is “independent” reference planes. The table below summarizes which types of reference planes, shared or independent, are used for which type of ports:

Port Type	Reference Plane Type
Boxwall (orthogonal) with reference plane	Shared or Independent (Determined by user)
Boxwall (diagonal)	Independent only
Co-calibrated (orthogonal using Sonnet Box Ground)	Shared only
Co-calibrated (diagonal using Sonnet Box Ground)	Independent only
Co-calibrated (Floating Ground)	None
Auto-grounded	Independent only
Internal Ungrounded	None
Via Port	None

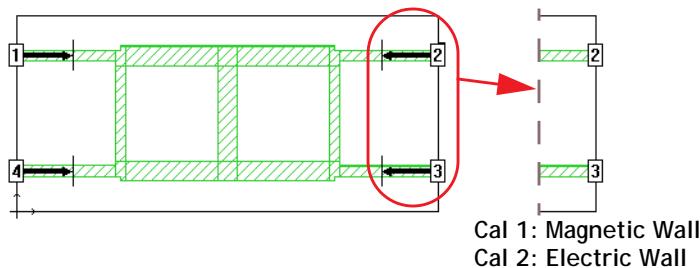
The sections below discuss the implications of using shared or independent reference planes.

## Shared reference planes

When ports have shared reference planes:

- The length of each reference plane is the same.
- Calibration standards are composed of multiple coupled lines as shown below.

- Coupling between the fringing fields of the ports is removed to provide a more accurate result.
- Coupling between feed lines is removed over the entire length of the reference plane.
- Sonnet does not calculate a  $Z_0$  or  $E_{eff}$ . This is because the lines are treated as coupled, and  $Z_0$  and  $E_{eff}$  are only valid for a single transmission line. Two coupled lines, for example, have two modes, and thus two  $Z_0$  and  $E_{eff}$  values. If you try to plot the  $Z_0$  or  $E_{eff}$  value of a shared port, you will receive an error message. If you are looking at the text output results, you will see "mp" de-embedding codes in place of the  $Z_0$  and  $E_{eff}$  values.



Calibration Standard for shared reference planes

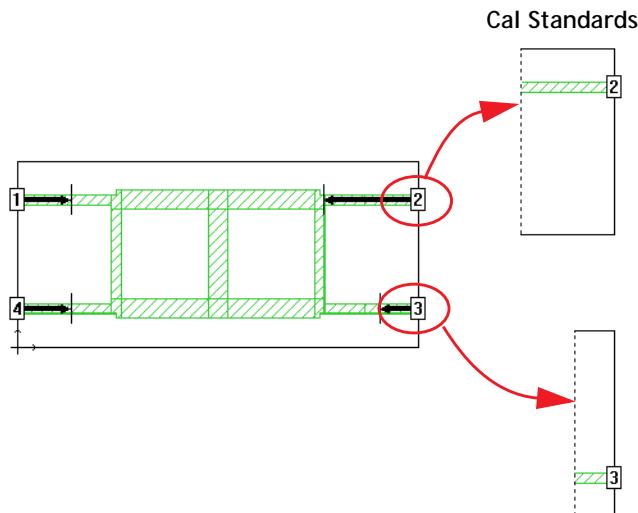
The only ports which can have shared reference planes are orthogonal boxwall ports and orthogonal co-calibrated ports with a Sonnet box ground node connection. For orthogonal boxwall ports, only ports on the same box-wall may be shared. Orthogonal box-wall ports share reference planes by default. A user may choose to make these reference planes independent. See the next section for details on changing the reference planes from shared to independent.

## Independent reference planes

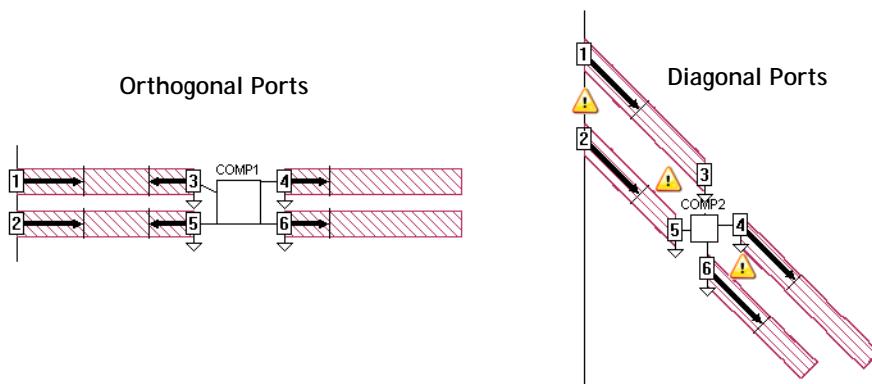
When ports have independent reference planes:

- The length of each reference plane may be set independently.
- Calibration standards are composed of single (uncoupled) lines as shown below.
- Coupling between the fringing fields of the ports is not removed. Thus, ports must be far apart to ensure an accurate result.
- Coupling between feed lines is not removed. Therefore, you should not

- use independent reference planes when the lines are close enough to couple.
- Sonnet calculates a  $Z_0$  or  $E_{eff}$  for each feed line without the influence of the nearby lines.



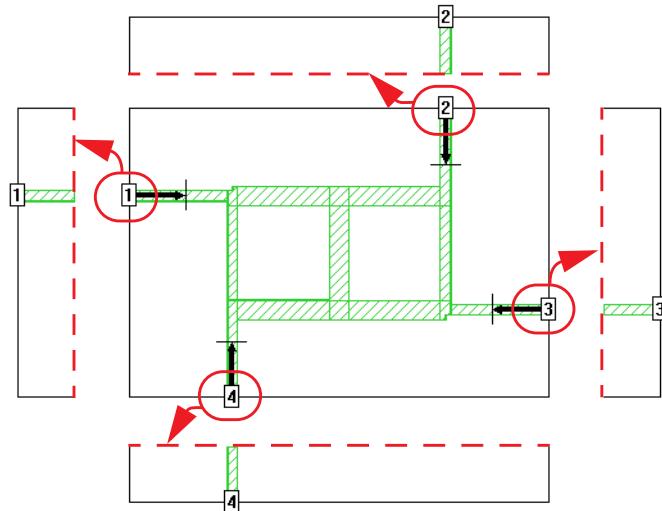
Auto-grounded ports, diagonal boxwall ports, and diagonal co-calibrated ports are always independent. Thus, they should not be used if the ports or feed lines are close to each other. This concept is shown below:



Orthogonal ports can use shared reference planes, so being in close proximity is allowed. However, Diagonal ports must use independent reference planes, so being in close proximity is not allowed.

Orthogonal boxwall ports on a given box-wall may be set to independent or shared. For instructions on how to change an orthogonal boxwall port from shared to independent, please see "[Applying the Diagonal Port Feature to a Port](#)" on page 81.

Ports on different box-walls are always treated as independent from one another as shown below.



Each port is calibrated independently because each port is on a different box-wall.

## De-embedded Output Format

The listing below shows an example of de-embedded results obtained from the analysis of a filter circuit. This example illustrates the format of the de-embedded data as output in the analysis monitor and saved as part of your project. This same format is used in any output files generated by *em* as part of the analysis or by the user when using the response viewer.

```
Run 1: Tue Jun 07 15:41:13 2011. sonnet9.aa.99999.
Em version 13.52 (32-bit Windows) on MENDOZA local.

Frequency: 8.975 GHZ
Frequency completed Tue Jun 07 15:41:16 2011.
De-embedded S-Parameters. 50.0 Ohm Port Terminations.
Magnitude/Angle. Touchstone Format. (S11 S21 S12 S22).
8.97500000 0.080414 143.59 0.996762 53.593 0.996762 53.593 0.080414 143.59
!< P1 F=8.975 Eeff=(6.65223738 0.0) Z0=(50.657578 0.0) R=0.0 C=0.07105378
!< P2 F=8.975 Eeff=(6.65223738 0.0) Z0=(50.657578 0.0) R=0.0 C=0.07105378

Analysis completed Tue Jun 07 15:41:23 2011.
```

Example showing format of results obtained when de-embedding is enabled in *em*.

You should notice the following about the results in above:

- The two lines which start with “De-embedded” and “Magnitude/Angle” are comment lines which describe the analysis results on the line below. In this example, the results are de-embedded 50 ohm S-parameters in Touchstone magnitude/angle format.
- The next line gives the analysis results.
- The remaining two lines give de-embedding information for each port in the circuit. The various fields are defined as follows:

<b>P#:</b>	Port number.
<b>F:</b>	Frequency in units defined earlier in the results file.
<b>Eeff:</b>	Effective dielectric constant of the transmission line connected to the port. Complex value: [Real Imaginary].
<b>Z0:</b>	Characteristic impedance of the transmission line connected to the port, in ohms. Complex value: [Real Imaginary].
<b>R:</b>	Equivalent series resistance of port discontinuity, in ohms.
<b>C:</b>	Equivalent series capacitance of port discontinuity, in pF.

---

**NOTE:** These values are not output for diagonal ports or ports with shared reference planes.

---

## De-embedding Codes

There are various conditions for which these values cannot be calculated and a de-embedding code is shown instead. For example, if you have shared reference planes (multiple ports on the same box-wall or multiple ports on the same side of a component, for example), there are multiple values for each item. In order to keep the output format concise, a de-embedding code is displayed in place of a longer explanation. Please see “De-embedding Codes” in Help for explanations of the code messages. To access Help, select *Help* ⇒ *Sonnet Help* from any Sonnet application.

## Plotting $Z_0$ and $E_{\text{eff}}$

When your circuit is de-embedded, the characteristic impedance,  $Z_0$ , and the effective dielectric constant,  $E_{\text{eff}}$ , of the transmission line are calculated for each port. These values are calculated for all port types except diagonal ports, ports which share a reference plane and via ports.

---

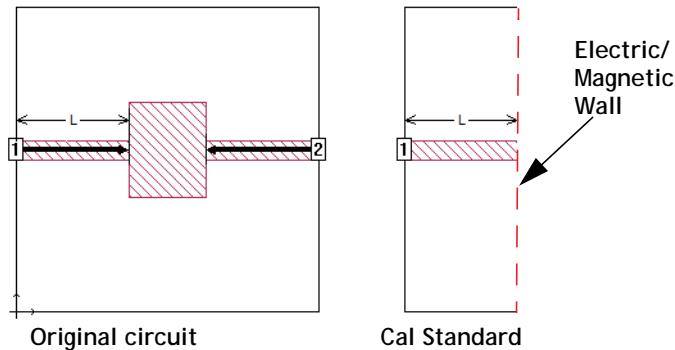
**NOTE:** For ABS sweeps,  $Z_0$  and  $E_{\text{eff}}$  are calculated only for discrete frequencies. These values are not calculated at adaptive data points.

---

## Calculating $Z_0$ and $E_{\text{eff}}$ for Boxwall Ports

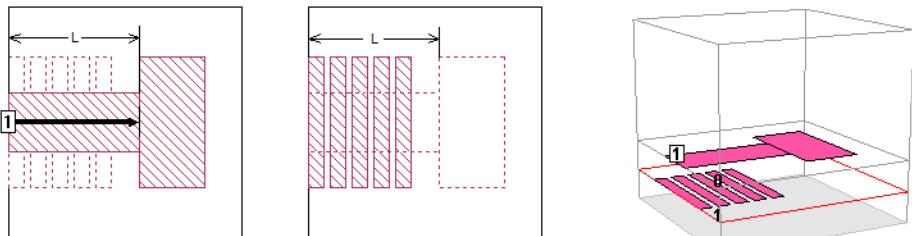
It is important to note that calculating the transmission line parameters is based on the calibration standard. For boxwall ports, the calibration standard, as discussed earlier in the chapter, is based on a cross section of any metalization touching the box-wall, on all levels, that the port under consideration is on. This cross section is extended the length of the calibration standard. Since the cross section is taken at the box-wall, the interior of the circuit does not affect the basis of the calculation.

In the first example shown below, there is a single feed line on the box-wall, with no metal on another level. Therefore, the thru line upon which the calculation is based is reasonable and the resulting values are accurate.



In the second example shown below, the circuit is more complex and the calibration standard thru line does not accurately reflect the actual circuit. Remember that the thru line used for the calculation is based on the cross section at the box-wall and is extended the length of the reference plane. In the actual circuit the metal on the level below the port only extends from the box-wall for a length much shorter than the calibration standard, but in the calibration thru line, this metal is extended

for the whole length and the other metal polygons in the interior are completely ignored. These transmission line parameters will not be as accurate as the first example.

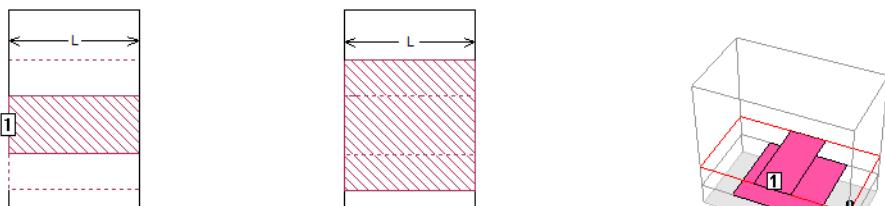


Level 0

Level 1

3D View

Actual Circuit



Level 0

Level 1

3D View

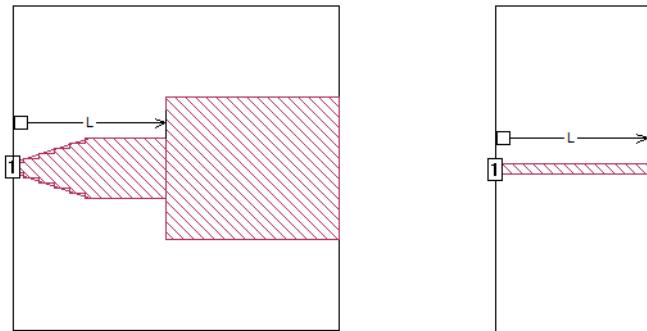
Calibration Standard Thru Line



### TIP

The RLGC parameters can be extracted from an EM simulation of a short section of any periodic structure including the feed line of the actual circuit pictured above. Please see ["N-Coupled Line Model" on page 369](#) for more information.

$Z_0$  and  $E_{\text{eff}}$  are also not as accurate for feed lines with non-uniform width. Because the cross section is extended uniformly from the wall the length of the calibration standard, a feed line with a varying width is not accurately modeled when calculating the transmission line parameters, as shown below.



Actual Circuit

Calibration Standard Thru Line

## Lengths at Multiples of a Half-Wavelength

$E_{\text{eff}}$  and  $Z_0$  cannot be calculated when the length of the calibration standard is an integral multiple of a half wavelength. At some point as the length of the calibration standard approaches a multiple of a half-wavelength, **em** is able to determine that the calculated values of  $E_{\text{eff}}$  and  $Z_0$  are becoming corrupt. When this occurs, **em** replaces the  $E_{\text{eff}}$  and  $Z_0$  data with the de-embedding code “undefined: nl” (see “De-embedding Codes” in Help). Note, however, that while **em** is unable to determine  $E_{\text{eff}}$  and  $Z_0$ , the de-embedded S-parameter results are still perfectly valid.

## Lengths Much Greater than One Wavelength

If the length of the calibration standard is more than one wavelength, incorrect  $E_{\text{eff}}$  results might be seen. However, the S-parameters are still completely valid.

**Em**’s calculation of  $E_{\text{eff}}$  is based on phase length. If the calibration standard is, say, 365 degrees long, **em** first calculates  $E_{\text{eff}}$  based on a phase length of 5 degrees. However, if a non-physical result is seen, **em** increases the calculated phase length by 360 degrees at a time until physical (i.e.,  $E_{\text{eff}} \geq 1.0$ ) results are obtained. This usually corrects the problem.

Thus, it takes a particularly long calibration standard before the  $E_{eff}$  calculation fails. When it does fail, it suddenly jumps down to a value just above 1.0.  $Z_0$  and the de-embedded S-parameter data still have full validity. This failure mode is rarely seen.

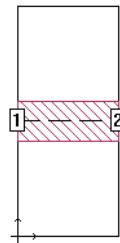
## Single Transmission Line

When you wish to determine the transmission line parameters,  $Z_0$  and  $E_{eff}$ , for a single transmission line, you should use a standard box-wall port for the best results. It is not recommended that you use an ABS sweep, since the  $Z_0$  and  $E_{eff}$  data is only calculated for the discrete data points, but not for any adaptive data; therefore an analysis will yield very few data points. If there are any problems in calculating  $Z_0$  and  $E_{eff}$ , then *em* outputs a de-embedding code in place of a value. The meaning of these codes may be found in Help under “De-embedding Codes.”

### Creating the Plot

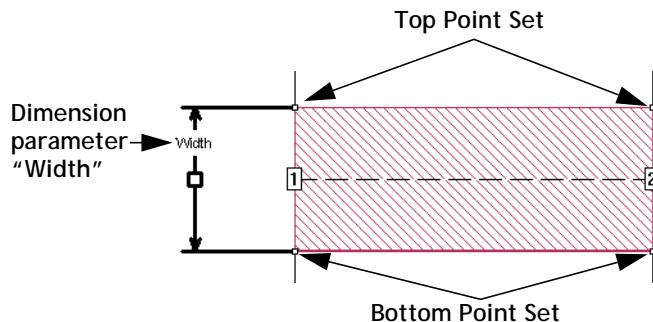
It is possible to plot  $Z_0$  or  $E_{eff}$  as a function of line width for a single transmission line by using a dimension parameter for the width of the feedline. You then analyze the circuit using a parameter sweep varying the width of the line over the desired range of values. Once you have the analysis results, you can use the response viewer to create a plot of your data. A detailed example is provided below.

This example uses a simple stripline circuit shown below.



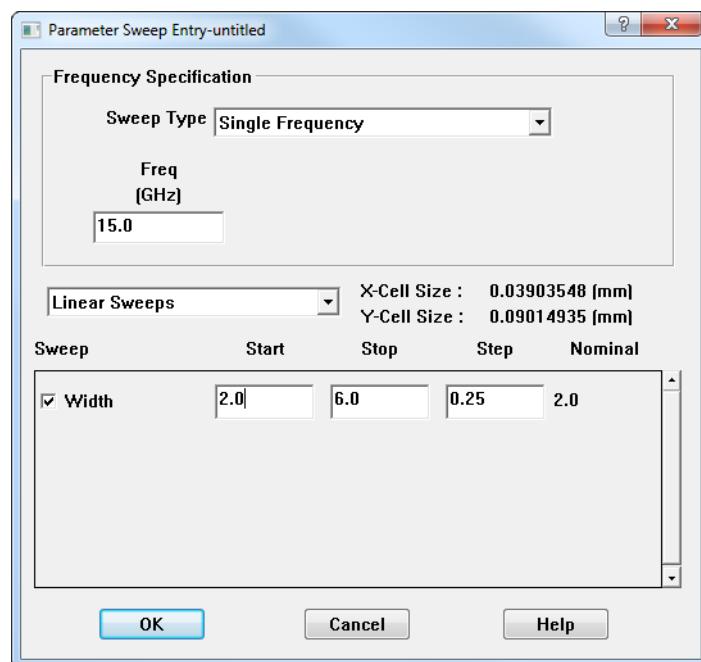
### 1 Add a dimension parameter to vary the line width.

A symmetric dimension parameter would work best with this circuit since it uses symmetry. The circuit is shown below with a symmetric dimension parameter "Width" added with its point sets highlighted. For more information about symmetric parameters and how to add them to your circuit, please see ["Symmetrical Dimension Parameters" on page 180](#).



### 2 Set up a parameter sweep using the variable "Width."

For this example, the circuit will be analyzed using a parameter sweep, varying the width of the line from 2 mm to 6 mm in 0.25 mm increments at a single frequency of 15 GHz. The Parameter Sweep Entry dialog box for the analysis is shown below. For more information on setting up a parameter sweep, please see ["Parameter Sweep" on page 189](#).



**3 Run the analysis, then open the response viewer.**



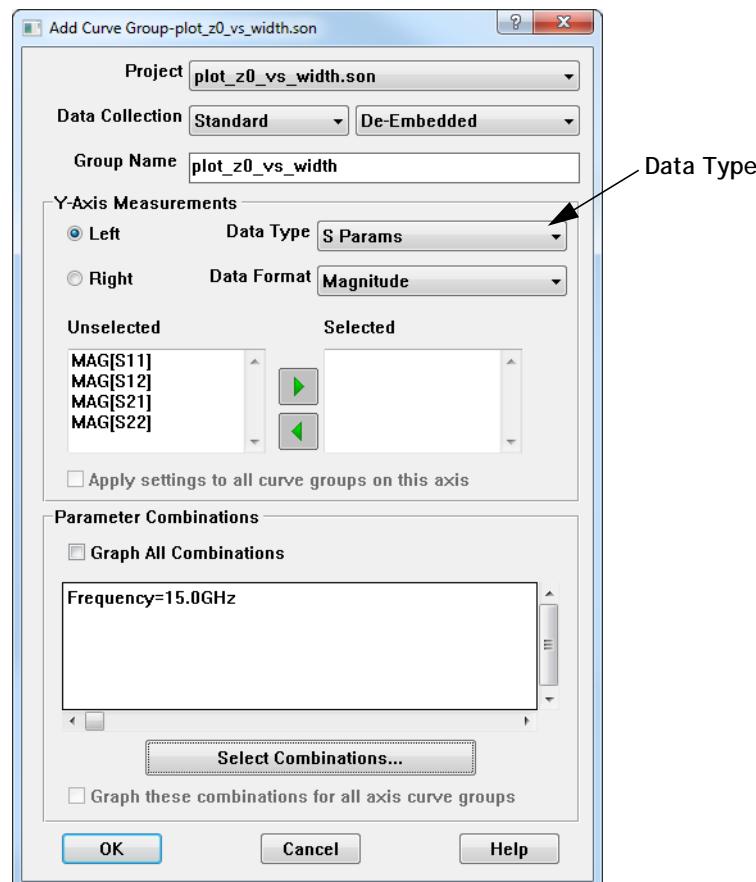
The response viewer may be opened by clicking on the Response Viewer icon in the analysis monitor (shown to the left) or selecting *Project*  $\Rightarrow$  *View Response*  $\Rightarrow$  *New Graph* from the main menu of the project editor or analysis monitor once the analysis has completed.

**4 Select *Graph*  $\Rightarrow$  *Plot Over*  $\Rightarrow$  *Parameter* from the main menu of the Response Viewer.**

This will plot the selected data as a function of the value Width. If there were more than one parameter in your circuit, you would have had to choose which to use but since this circuit contains only one parameter, Width is automatically selected.

**5 Select *Curve*  $\Rightarrow$  *Add Curve Group* from the main menu of the response viewer.**

The Add Curve Group dialog box appears on your display as shown below.



**6 Select “Port Z0” from the Data Type drop list.**

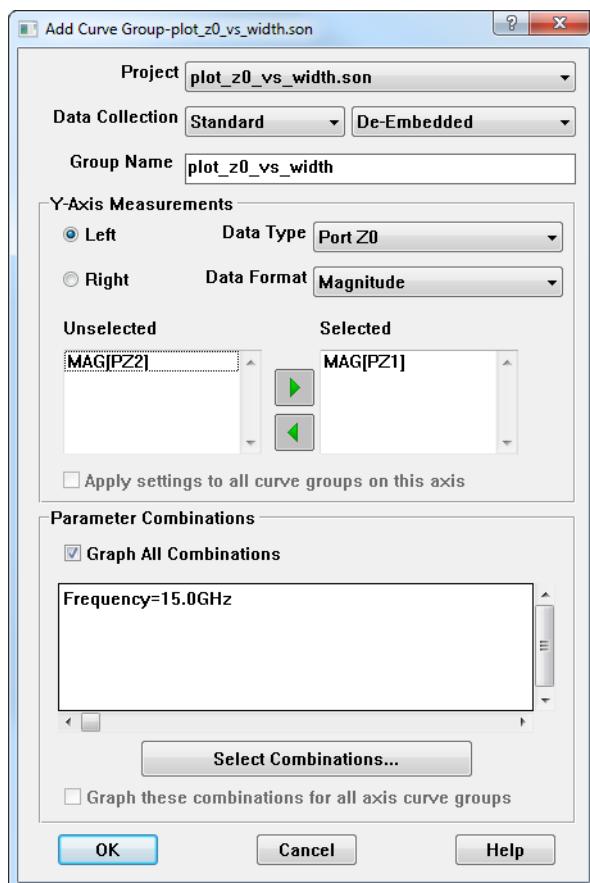
This will plot the characteristic impedance of the feedline connected to the selected port.

**7 Double click on the desired port to move it to the Selected List.**

For this example, we will plot Z0 for port 1, so MAG[PZ1] is moved to the selected list.

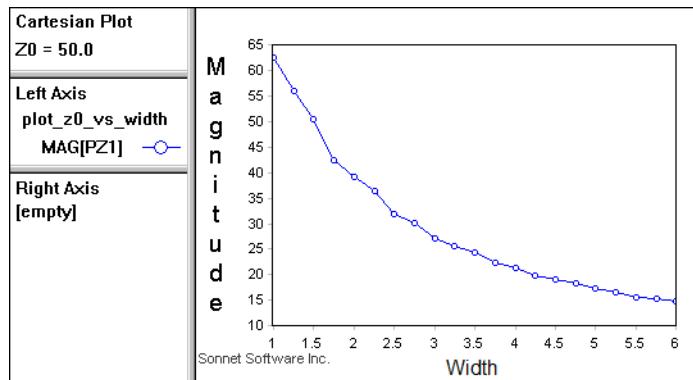
**8 Click on the Graph All Combinations checkbox in the Parameter Combinations section of the dialog box.**

There is only one frequency available, so selecting this checkbox will display that frequency. If more than one frequency were available, you would plot all available frequencies if the Graph All Combinations checkbox was selected. If you wished to plot a particular frequency or set of frequencies, you would clear this checkbox, then click on Select Combinations to choose the desired frequency(ies) to display. The dialog box should appear similar to that shown below.



**9 Click on the OK button to display the curve.**

The plot shows the magnitude of the characteristic impedance of Port 1 as a function of line width. The characteristic impedance goes down as the line gets wider. If you wished to plot the  $E_{eff}$ , you would have chosen "Port  $E_{eff}$ " from the Data Type drop list.

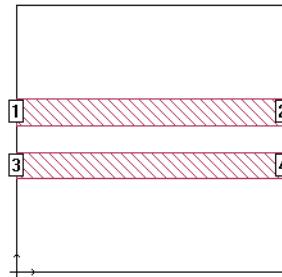


## Coupled Transmission Lines

If you have more than one port on a box-wall,  $Z_0$  and  $E_{eff}$  cannot be determined as there is more than one mode for the transmission line. In this case, your output data will use the "mp" de-embedding code indicating that there are multiple ports. Even though  $Z_0$  and  $E_{eff}$  can not be determined, the S-parameter data is still valid.

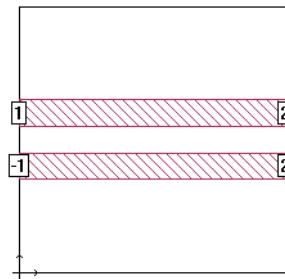
However, Sonnet can be used to compute the common and differential mode  $Z_0$  and  $E_{eff}$  values for two symmetrical coupled lines. The common and differential mode are very similar to the even mode and odd mode used by popular transmission line calculators (see "Interpreting  $Z_0$  and  $E_{eff}$ ." on page 135 for details).

To create a project which can be used to show the even and odd mode, start with a four-port coupled line circuit like the one shown below:



Notice that the lines are placed precisely in the center of the box. This is not necessary in many cases, but we suggest you do this to force the symmetry between the two lines.

Next, renumber your ports as shown below. The left two ports (+1/-1) represent the differential, or odd mode. The right two ports (+2/+2) represent the common, or even mode:



To renumber a port, right-click on the port and select Port Properties. This opens the Port Properties dialog box where you can change the port number.

### Interpreting $Z_0$ and $E_{\text{eff}}$ .

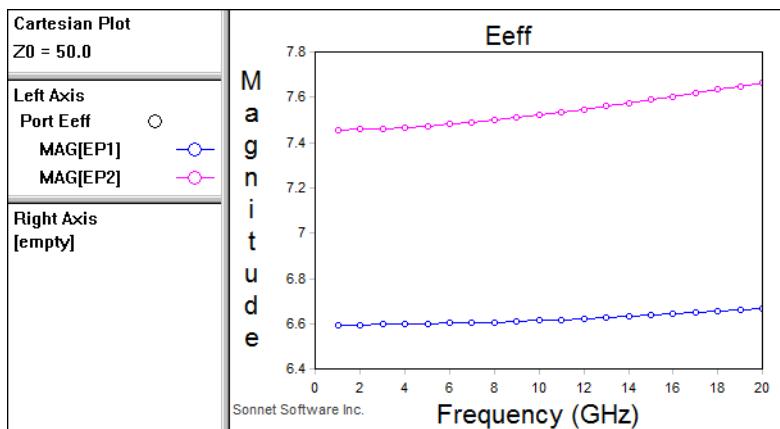
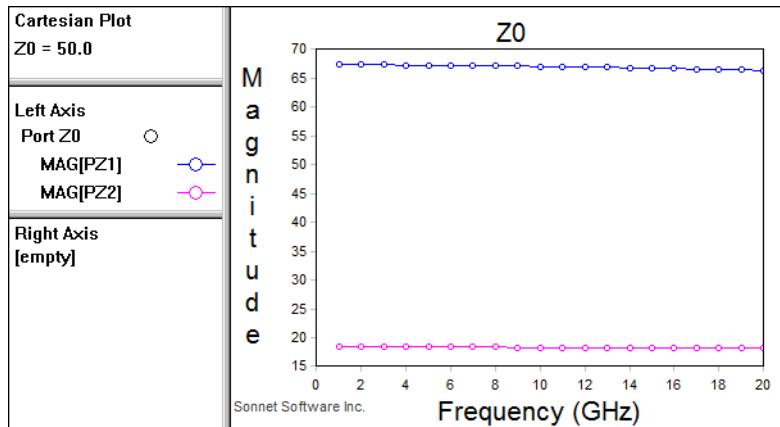
There is a subtle difference between the even/odd mode and common/differential mode  $Z_0$ <sup>[1]</sup>. Sonnet computes the common and differential mode  $Z_0$  values, but you can easily calculate the even and odd mode  $Z_0$  values by either multiplying or dividing the sonnet results by two as shown in the equations below:

$$Z_{\text{even}} = 2 \times Z_{\text{Common}}$$

$$Z_{\text{odd}} = Z_{\text{diff}} / 2$$

[1]Characterization of differential interconnects from time domain reflectometry measurements by A. Tolescu and P. Svasta

The  $E_{eff}$  values do not need to be modified. Shown below are plots of the differential mode (port 1) and common mode (port 2)  $Z_0$  and  $E_{eff}$ . Since the differential mode  $Z_0$  is about 67 ohms, the odd mode  $Z_0$  is about 33.5 ohms. And since the common mode  $Z_0$  is about 18 ohms, the even mode  $Z_0$  is about 36 ohms. As can be seen by the plots, the  $Z_0$  and  $E_{eff}$  values are all a function of frequency.



# Chapter 7 Components

---

## Introduction

The Component feature is built upon the high accuracy de-embedding technique<sup>1</sup> used for Sonnet co-calibrated internal ports technology. Using this technique, the user can insert an ideal element, measurement of a physical component, a data file created by another Sonnet project or another Sonnet project. In addition, using the Component feature in conjunction with the [Keysight ADS Interface](#), [NI AWR Microwave Office Interface - 32-Bit](#), or the [Cadence Virtuoso Interface](#) provides a powerful tool to model complex circuits.

The Components features allows for a high level of flexibility by using five different Component types:

- **Data File type:** If you would like to insert an S-parameter data file of your component into your Sonnet model.
- **Ideal type:** If you would like to insert an ideal component (R, L, or C) into your Sonnet model.
- **Ports Only type:** If you would like to use a separate circuit simulation tool for the final, combined simulation, you may insert only ports within the Sonnet model.

<sup>1</sup>"Deembedding the Effect of a Local Ground Plane in Electromagnetic Analysis," by James C. Rautio, president and founder of Sonnet Software, Inc. The article is available in PDF format in the Support section of our web site.

- **Sonnet Project:** If you would like to insert another Sonnet project into your Sonnet model.
- **User Model:** If you would like to insert a model from another source into your Sonnet model.

In all cases except where the Ports Only component types are used, the Sonnet solver uses a circuit simulation technique to produce the combined results. Since this is a post EM analysis step, the user can change the ideal component value, or associated S-parameter data file, Sonnet project or Modelithics model without requiring a new EM analysis; only the circuit simulation part of the analysis is performed by the analysis engine.

Please note that the coupling from the inside of the component to the rest of the circuit is not considered in the Sonnet analysis. Only the coupling from the component's terminals is considered.

When connecting external parts to your EM structure, we highly recommend that you use either the co-calibrated ports or the Component feature. This novel approach provides greatly enhanced accuracy for this application by perfectly de-embedding the ports, thus completely removing all coupling between them. Auto-grounded Ports have the limitation of not removing any port to port coupling.

Before proceeding to use the Component feature, it would be helpful to become familiar with the related co-calibrated internal ports discussed in [“Co-calibrated Internal Ports,” page 89](#) in the Ports chapter.

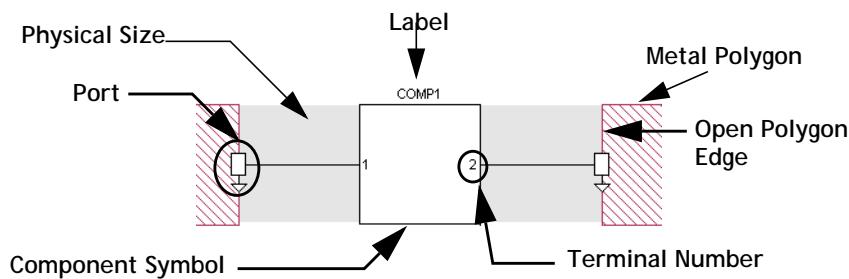
## Component Assistant

When you select any of the Add Component commands in the project editor, the Component Assistant appears on your display. Whenever you select a control in the Component Properties dialog box, the assistant provides a description of the field, and often, an illustration of the principle, so that you may select the correct setting and model your Component more accurately.

If the Component Assistant does not appear, you should select the “Use Component/Calibration Group Assistant” checkbox on the Hints tab of the Preferences dialog box in the project editor (*File ⇒ Preferences*).

# Anatomy of a Component

The Component is represented in your circuit by a component symbol. The label of the Component appears above the component symbol and the terminal numbers are identified there. Ports, indicating where the terminals of the Component are connected to the metal in your circuit, are represented by a small rectangle. Component ports are only numbered when the Component model type is Ports Only. An example of a Component as it appears in the project editor is shown below.



**Component Symbol** This is the symbol which represents your Component in your project.

**Port** The Component port defines the point at which the Component is connected to the circuit metal and must be attached to an open polygon edge. This point also serves as a reference plane for de-embedding the Component unless a reference plane is added for the port. There is one Component port for each terminal on the Component. Component ports are modeled using co-calibrated internal ports. For more information on co-calibrated ports, please see “Co-calibrated Internal Ports,” page 89. If the component type is a Sonnet project, the terminals correspond to the ports in the Sonnet project.

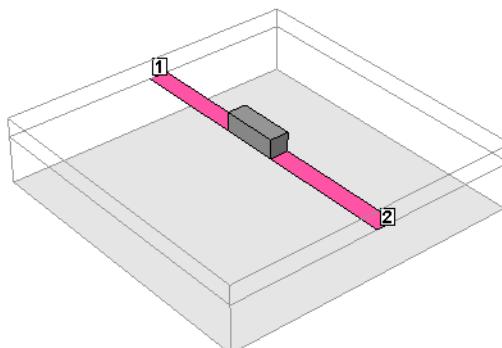
**Terminal Numbers** Terminal numbers identify the physical pin on your Component connected to the corresponding Component port.

Components may have an unlimited number of terminals, with the exception of those Components which use the Ideal Component model, since the available ideal components are limited to two terminals. Terminals and/or ports are numbered in the order in which they are added to your circuit and may be modified after the Component is added to your circuit.

**Label** The label is user-defined text which identifies the Component in your circuit. Each Component label in a project must be unique.

**Open Polygon Edge** Component ports may be placed only on open polygon edges. You may use a reference plane to control how much, if any, of your circuit metal is also de-embedded in addition to the Component. For more information, see “Reference Planes,” page 148.

**Physical Size** You may also optionally enter a physical package size for your Component. These measurements (height, width, and length) are not used in the *em* simulation but are there to provide a graphic in both the 2D and 3D view which represent the actual size of the Component. This is especially useful for design presentations and reviews. Shown below is a 3D view of the example Component pictured above:



## Component Types

There are five types of Component model: Data File, Ideal, Ports Only, Sonnet Project, and User Model. All five models are described below.

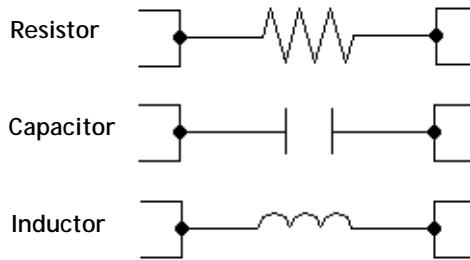
### Data File

The Data File Component allows you to add a Component to your geometry that is modeled with a user specified S, Y, or Z-Parameter file (Touchstone format). The data file used for your Component can be the result of another simulation or measured data from an actual component. There is no limit to the number of ports you may use for a data file Component.

You add a data file type Component by selecting the command *Tools* ⇒ *Add Component* ⇒ *Data File*. This command opens the Components Properties dialog box as well as the Component Assistant.

## Ideal Component

The Ideal component allows you to add a single 2-port ideal component. There are three types of ideal components available: resistor, capacitor, or inductor. All ideal components use a series element with two ports as shown below:



You add an Ideal component by selecting the command *Tools*  $\Rightarrow$  *Add Component*  $\Rightarrow$  *Ideal*. This command opens the Components Properties dialog box as well as the Component Assistant.

## Ports Only

The Ports Only Component allows you to insert internal ports in your circuit which may be used later in a circuit design program. All of the ports associated with this Component have a common ground and are simultaneously de-embedded during the electromagnetic analysis. There is no limit to the number of ports your Component may have. This Component type is functionally equivalent to using co-calibrated internal ports. For a detailed discussion of co-calibrated ports, see “Co-calibrated Internal Ports,” page 89.

You add a Ports Only type Component by selecting the command *Tools*  $\Rightarrow$  *Add Component*  $\Rightarrow$  *Ports Only*. This command opens the Components Properties dialog box as well as the Component Assistant.

### Sonnet Project

The Sonnet Project Component allows you to add a Sonnet geometry or netlist project to your geometry. If the Sonnet project used for the component does not include analysis data, then **em** runs an analysis on the Component project as part of the larger analysis. There is no limit to the number of ports you may use for a Sonnet project Component.

If the Sonnet Project contains parameters, you may open the Model Parameters dialog box by clicking on the Parameters button in the Component Properties dialog box. This dialog box allows you to define the values or link the parameters in the Component project to parameters in the main project. You may also create a new variable and assign it to a parameter value. Any change in the variables in the main project also changes the value of the any parameters in the Component project that are defined by that variable.

You add a Sonnet Project Component by selecting the command *Tools* ⇒ *Add Component* ⇒ *Sonnet Project*. This command opens the Components Properties dialog box as well as the Component Assistant.

### User Model

The User Model Component allows you to add a model from a library that has been purchased, or developed in-house, to your geometry. There is no limit to the number of ports you may use for a User Model Component.

If the model contains parameters, you may open the Model Parameters dialog box by clicking on the Parameters button in the Component Properties dialog box. This dialog box allows you to define the values of the parameters in the User Model Component. You may enter a fixed value or use a variable defined in the main project as the value of the parameter(s). You may also create a new variable and assign it to a parameter value. When a variable is assigned as a value, any change in the variable in the main project also changes the value of the parameter in the Component project.

You add a User Model Component by selecting the command *Tools* ⇒ *Add Component* ⇒ *User Model*. This command opens the Components Properties dialog box as well as the Component Assistant.

Modelithics models are presently the only supported vendor, although it is possible to create custom models. If you are interested in doing so, please contact your Sonnet Software representative for details.

---

**NOTE:** Use of the Modelithics models requires a Modelithics license, which must be purchased separately from Modelithics, Inc., in addition to your Sonnet license.

---

## Component Properties

An important part of modeling a Component in Sonnet is to consider the conditions under which the measured data or model for your Component was obtained. These conditions should be used to determine:

- The type of ground node connection
- The terminal width
- If reference planes are used for the Component ports and if so, of what length

The ***em*** environment should be set up to use the Component in the same manner that the component was measured. The correct setting of the Component properties is discussed in detail below.

### Ground Node Connection

The ground node connection defines how the ground of your Component is connected to your circuit. There are three types of ground node connections: Sonnet Box, Floating, and Polygon Edge(s).

---

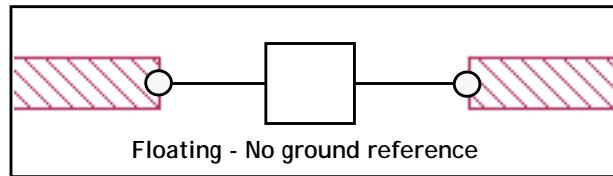
**NOTE:** The Ideal Component does not use a ground node connection by definition. The ground node connection needs to be specified for all other Component types.

---

Sonnet uses a common ground for all the Component ports associated with a given Component. This common ground should model as closely as possible how your component was measured or modeled. Vendors who supply components often have measured S-parameters or a model which may be used to create S-parameters. In either case, information about how these values were obtained should also be available. Use this information to guide your choice of ground node connection. The three types of ground node connection are discussed below.

### Floating

When your ground node connection is set to Floating, all the Component ports reference a common local ground as pictured below. This option should be used if your Component does not have a ground reference, or if there are no shunt elements in your component model or measured data. Any shunt admittance is removed by *em*.

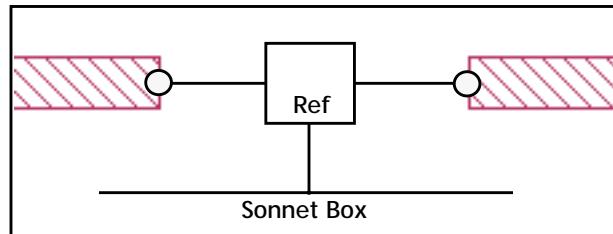


Examples of this are:

- A series RL equivalent circuit
- S-Parameter data that was measured without pads

### Sonnet Box

When your ground node connection is set to Sonnet box, all the Component ports are globally grounded to the Sonnet box. To accomplish this, a via is automatically created which connects to the top or bottom of the Sonnet box. This option should be used if your component model, or measured data, includes shunt elements and you want the Component's ground reference to be connected to the Sonnet box.



Examples of this are:

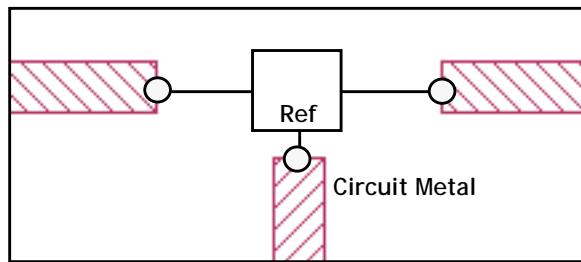
- S-Parameter data that includes pads or other coupling to ground
- Transistor data that includes a via to ground in the model/measurement

The analysis engine determines the most efficient direction the ground via extends taking into consideration both the distance and the loss of the box top or bottom. When using this type of ground, you must make sure that there is a clear path with no metal on other levels interfering with the path to either the box top or bottom.

If you are using a Component whose ground node connection is to the Sonnet box, either the box top or bottom must have loss less than 50 ohms/sq. For example, you should not use “Free Space” as your box top and bottom definition. If the loss is too high on both the box top and bottom for a ground via from the Component to be attached, the analysis engine will issue an error message.

### Polygon Edge(s)

When your ground node connection is set to Polygon Edge(s), the ground reference of your Component is connected to a polygon edge(s) selected by you when adding the Component. You may specify as many ground terminals as you need.



Examples of this are:

- Transistor data without parasitics to ground, but with a ground path included in the Sonnet structure
- A multi-pin module with one or more ground pins

---

**NOTE:**

The Polygon Edge(s) ground node connection is only available for Data File and Sonnet Project type Components.

---

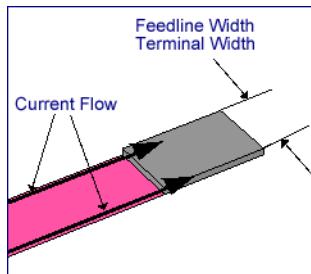
### Terminal Width

The terminal width is the electrical contact width of the Component. Entering a terminal width allows you to accurately model the current flow from the circuit geometry into the Component.

There are three types of terminal width: feedline, one cell and user defined. Each type of terminal width is illustrated and explained below.

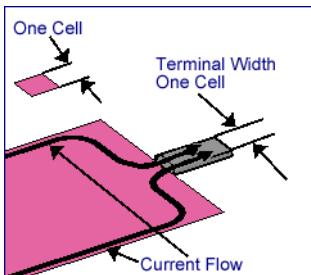
### Feedline Width

Choosing Feedline Width sets your terminal width to match the length of the polygon edge to which the Component is attached. This option should be used when the polygon edge is about the same size as the width of the Component. An example is pictured below.



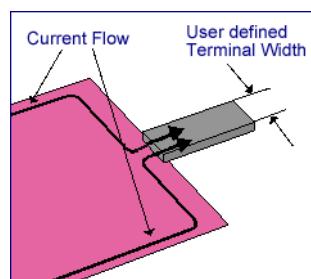
### One Cell

One Cell sets your terminal width to the smallest possible size of one cell wide as pictured below.



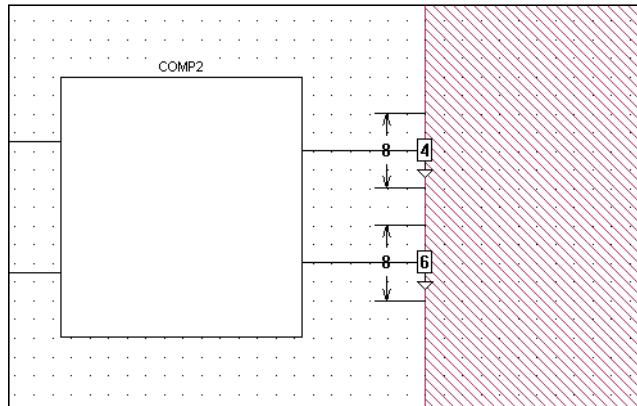
### User Defined

This option allows you to enter a known electrical contact width. An illustration of a Component with User Defined terminal widths is shown below.

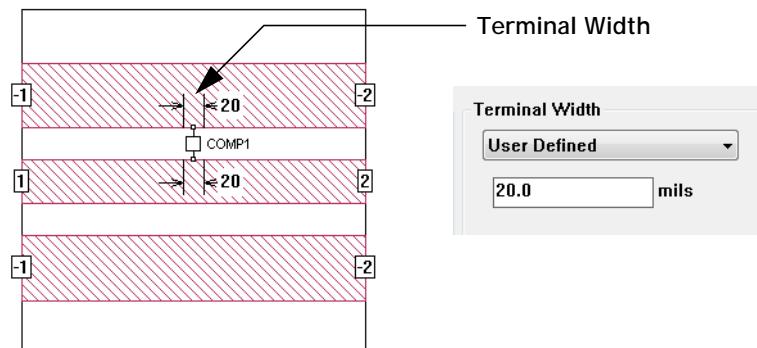


When you select user defined, your terminal width is defined based on the location of the Component port. The point at which the port is placed on the polygon edge becomes the center of the terminal width extending an equal distance on either side.

This ability to limit the terminal width size is important in cases where more than one Component port needs to connect to the same polygon edge or the polygon edge is extremely large. The next two images demonstrate these concepts.



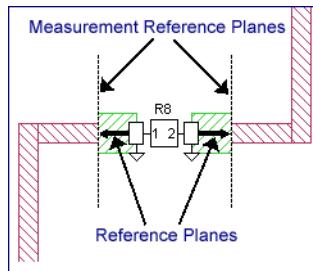
The component has two terminals connected to a single polygon edge. The cell size is 2 mils and the user-defined terminal width is 8 mils as shown.



A coplanar waveguide that uses a two port component. To the right is shown the entry in the Component Properties dialog box.

## Reference Planes

Components require their ports to be on open polygon edges. Reference Planes can be used to effectively move your port position away from the polygon edge. To accomplish this, *em* uses circuit theory to cascade a negative length of the line with the analysis results. If no reference plane is specified, then de-embedding the Component removes none of the feedline metal but the port(s) are still de-embedded. The use of reference planes is illustrated below.



---

**NOTE:** Reference planes may only be used when the ground node connection is defined as the Sonnet box.

---

All the Component ports on a side of a Component use the same reference plane. For a detailed discussion of reference planes and de-embedding, please see **Chapter 6, “De-embedding”** on page 103.

The reference planes for Component ports are set in the Component Port Properties dialog box. This dialog box may be opened by double-clicking on a component port or by selecting the component, then selecting the command *Modify* ⇒ *Port Properties* from the project editor main menu. For details on using this dialog box, please click on the Help button.

As with all Sonnet reference planes, the Component reference planes are snapped to the grid. It is important that you are working with a fine enough cell size to ensure the accurate placement of your reference planes. If the length is set to less than one half the cell size, then the reference plane will not be displayed in the project editor window.

## Calibration Lengths

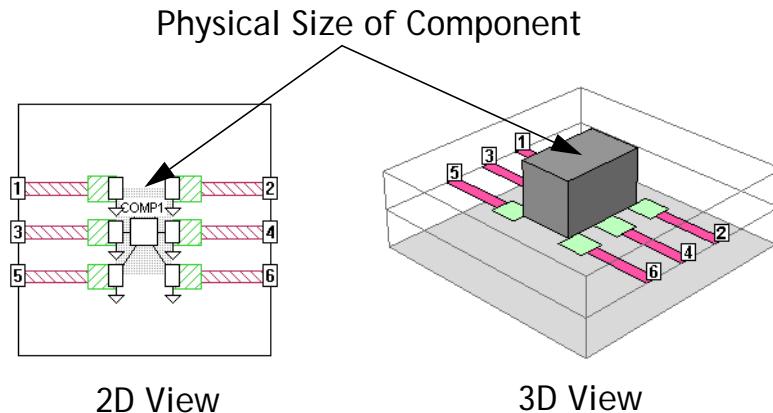
The analysis engine, *em*, will automatically determine appropriate lengths for the calibration standards used in the de-embedding algorithm. Normally, the Auto setting (default) produces efficient and highly accurate simulation results. In rare cas-

es, you may wish to manually override the automatic lengths. Before manually overriding these settings, please be sure to read **Chapter 6, “De-embedding”** on page 103.

The calibration lengths for Component ports are set in the Component Port Properties dialog box. This dialog box may be opened by double-clicking on a component port or by selecting the component, then selecting the command *Modify* ⇒ *Port Properties* from the project editor main menu. For details on using this dialog box, please click on the Help button.

## Physical Size

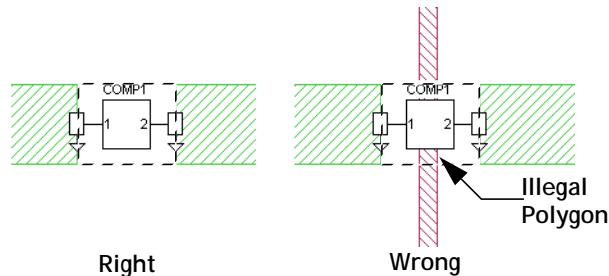
You may enter a physical size for your Component for display purposes. The physical dimensions length, width, and height, are not used in the simulation but do affect how your Component is displayed in the project editor. You may enter the precise dimensions or choose Auto to have the software choose approximate dimensions based on your Component’s port placements. An example is shown below.



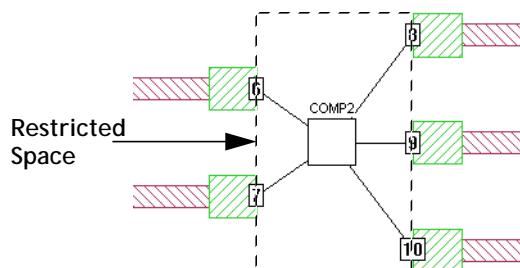
## Rules for Using Components

The following restrictions apply to the placement of Components in your circuit:

- No objects may be placed within the “restricted space” in the interior of the Component: Metal polygons, vias, or dielectric bricks may not be present in the rectangular area defined by the port locations and the terminal width, as illustrated below. The bottom circuit shows the restricted area in a multiport Component.



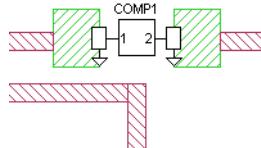
The dashed boxes identify the “restricted space” for each Component. On the circuit labeled “Wrong” a metal polygon passes through the middle of the restricted space.



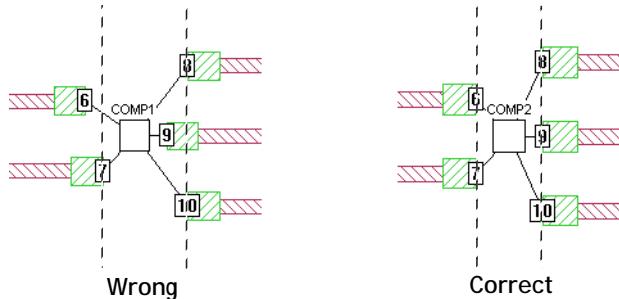
- Components whose ground node connections are defined as the Sonnet box require direct access to the Sonnet top or bottom cover. The analysis engine determines the most efficient direction the ground via extends taking into consideration both the distance and the loss of the box top or bottom. When using this type of ground, you must make sure that there is a clear path with no metal on other levels interfering with the path to either the box top or box bottom.

In addition, the box top or bottom should not have loss greater than 50 ohms/sq. If the loss is too high on both the box top or bottom for a ground via from the Component to be attached, the analysis engine issues an error message.

3. **Nearby objects should be placed so that coupling between the Component and the object does not occur:** Metal polygons, vias, or dielectric bricks which couple to the Component ports may decrease the accuracy of the analysis as shown below.



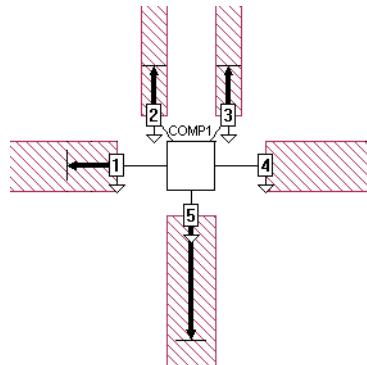
4. **Multiple ports on the same side of a Component should be aligned:** In the illustration below, the ports on Comp1 are placed incorrectly and will produce an error during an analysis. The ports on comp2 are placed correctly.



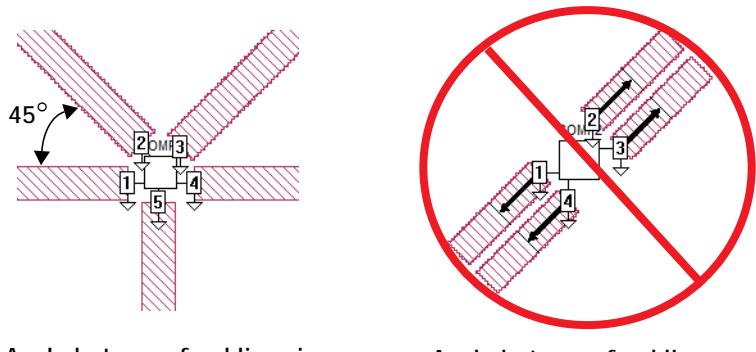
The dashed lines represent the alignment plane.

5. **Reference planes must be the same length for each side of a Component:** Note that the reference planes for ports 2 and 3 in the illustration below are the same length. It is important that these ports share a reference plane since the feedlines are close enough that coupling between them becomes a factor and needs to be removed during de-embedding. However, reference planes can be different lengths for each side of the

“component rectangle” as shown by ports 1, 4 and 5 below. Since these ports are on different sides of the component, their reference planes are independent from ports on other sides.



- The angle between feed lines of diagonal components should be at least 45 degrees: This means you should never have parallel diagonal feed lines on a single side of a component.



## Analysis of a Component

### Data File Frequencies

When using a data file Component type, frequencies in your data file do not need to precisely match the Sonnet analysis frequencies. The analysis engine will interpolate between data file frequencies if necessary, but it will not extrapolate outside the frequency range of the data file.

## Rerunning an Analysis

When the analysis engine analyzes your circuit with a Component, it first performs an electromagnetic analysis of the geometry, then uses circuit theory to connect the Component to the geometry. If you change the data file used for a Component or the value and/or type of an ideal component, in subsequent analyses **em** only needs to perform the circuit theory part of the analysis, significantly reducing processing time. Please note that any graphs of the response are not automatically updated. Instead, you need to select *Graph* ⇒ *Freshen Files* to update your graph in the response viewer.

The analysis engine, **em** runs an analysis on the Component project as part of the analysis of the main project and stores analysis results for the component project. The data resulting from the analysis of the Component project is then used in the circuit theory part of the main analysis. This data will be reused as long as no changes are made to the Component project.



## Chapter 8

## Adaptive Band Synthesis (ABS)

---

The Adaptive Band Synthesis (ABS) technique provides a fine resolution response for a frequency band requiring only a small number of analysis points. **Em** performs a full analysis at a few points and uses the resulting internal, or cache, data to synthesize a fine resolution band.



### TIP

---

This technique, in most cases, provides a considerable reduction in processing time.

---

Using the input frequency band, **em** first performs a full analysis of the circuit at the beginning and end frequencies. **Em** continues solving at discrete points, storing the full analysis data for each point. This process continues until enough internal, or cache, data is generated to synthesize a fine resolution response.

Once the frequency band response is synthesized, **em** outputs approximately 300 data points for the frequency band. These data points are a combination of the discrete analysis points and synthesized points. This combined data is referred to as adaptive data.

**Em** dedicates the bulk of the analysis time for an ABS analysis in calculating the response data at the discrete data points. Once the adaptive band synthesis is complete, calculating the adaptive data for the entire band uses a relatively small percentage of the processing time.

## DC Point Extrapolation

If you enter a value of zero as the starting frequency for an ABS sweep, the data for a DC Point is automatically calculated. Once the ABS sweep is complete and has produced the adaptive data, **em** extrapolates the adaptive data to DC. If you only need a DC Point and do not want to run an ABS sweep, you should use the DC Frequency Sweep. For details, please see “DC Point” in the index of Help.

## ABS Resolution

The ABS resolution is the value in frequency units between adaptive data points in your response output from an adaptive sweep. Normally, the resolution in an adaptive sweep is provided by **em** such that around 300 data points are output for a frequency band. It is possible for you to override this setting and use a coarser or finer resolution for your frequency band.

Entering a manual value to be used for ABS is done in the Advanced Options dialog box which is opened when you click on the Advanced button in the Analysis Setup dialog box. The Analysis Setup dialog box is opened when you select *Analysis* ⇒ *Setup* from the project editor menu. You enter the resolution by clicking on the Manual radio button in the ABS Frequency Resolution Per Sweep section of the Advanced Options dialog box and entering the desired resolution in the adjacent text entry box. For details on these dialog boxes, please refer to Help for the project editor.

There are several things to be aware of when using the manual setting for the ABS resolution. Coarse resolution does not speed things up. Once a rational polynomial is found to “fit” the solution, calculating the adaptive data uses very little processing time. A really coarse resolution could produce bad results by not allowing the ABS algorithm to analyze at the needed discrete frequencies. Fine resolution does not slow down the analysis unless the number of frequency points in the band is above approximately 1000 - 3000 points. A step size resulting in at least 50 points and less than 2000 points is recommended.

## Q-Factor Accuracy

There is a Q-Factor analysis run option available in the Advanced Options dialog box in the project editor (Select *Analysis*  $\Rightarrow$  *Setup*, then click on the Advanced button in the Analysis Setup dialog box). Selecting this option forces a higher accuracy for ABS convergence by including the Q-factor of your analysis as a criterion for convergence. This is done to insure high accuracy in the Q-Factor result when ABS is used. The Q-Factor is defined as follows:

$$|(\text{imag } Y_{nn}) / (\text{real } Y_{nn})|$$

The result is higher accuracy from the ABS sweep, but this accuracy comes at the cost of requiring more discrete frequencies to be analyzed before convergence is reached.

## Enhanced Resonance Detection

There is an Enhanced Resonance Detection analysis run option available in the Advanced Options dialog box in the project editor (Select *Analysis*  $\Rightarrow$  *Setup*, then click on the Advanced button in the Analysis Setup dialog box). This option is useful when the circuit response includes one or more extremely narrowband resonances, a common occurrence with superconductor applications. When this option is enabled, ABS performs a full-wave discrete simulation at each detected resonance to obtain the highest level of accuracy possible in terms of magnitude and frequency. Be aware that this increased accuracy comes at the cost of increased processing time since it requires analysis at more discrete frequency points.

## Running an Adaptive Sweep

To run an analysis using the Adaptive Band Synthesis technique, you do the following:

- 1 **Open your project in the project editor.**

**2 Select *Analysis*  $\Rightarrow$  *Setup* from the main menu of the project editor.**

The Analysis Setup dialog box appears on your display with an adaptive sweep already selected since Adaptive Sweep is the default for analysis control.



**3 Select Adaptive Sweep (ABS) from the Analysis Control drop list if necessary.**

This selects the ABS technique for the analysis.

**4 Enter the desired frequency band in the Start and Stop text entry boxes.**

This defines the frequency band on which you wish to perform the ABS analysis. The step size is automatically set by *em* during the analysis. See page 156 for a description of how the ABS resolution is determined.

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

**6 Save the project by selecting *File*  $\Rightarrow$  *Save* from the menu or by clicking on the Save button on the tool bar.**

You need to save the file before analyzing it.

**7 Select *Project*  $\Rightarrow$  *Analyze* from the menu or click on the Analyze button on the tool bar.**

*Em* performs an adaptive sweep on your project. The analysis monitor appears on your display, and indicates the progress of the adaptive sweep.

The Adaptive Sweep is also available within the Frequency Sweep Combinations analysis controls. This allows you to mix adaptive sweeps with other types of sweeps. For more details, see the “Frequency Sweep Combinations” topic in Help in the project editor.

## ABS Caching Level

There are three levels of ABS caching available: None, Stop/Restart, and Multi-Sweep plus Stop/Restart. The options for ABS caching level are found in the Advanced Options dialog box. To access the Advanced Options dialog box, select *Analysis* ⇒ *Setup* from the project editor main menu, then click on the Advanced button in the Analysis Setup dialog box which appears. The default is Stop/Restart.

Stop/Restart retains the cache data while the analysis is proceeding. Once the adaptive data has been calculated, the cache data is deleted from the project. This setting provides for the circumstance in which the analysis is stopped or interrupted before the adaptive data is synthesized; you will not lose the internal data produced so far.

Multi-Sweep with Stop/Restart retains all calculated cache data in your project for every analysis job run. In addition, cache data is calculated and saved for even non-ABS types of sweeps. This option can reduce processing time on subsequent ABS analyses of your project but increases project size on non-ABS sweeps. In order for the cache data to remain useful there are also subsectioning issues of which you must be aware. For a detailed discussion of the Multi-Sweep cache option, please refer to “Multiple ABS Sweeps and Subsectioning,” page 159.

The third setting for ABS caching level is None. In this setting cache data is not retained. This option should only be selected if you have constraints on disk space.



### WARNING

---

If you select None for the ABS caching level, and an ABS sweep is stopped before the adaptive data has been calculated, you will have to start the analysis over from the beginning. Any processing time invested in the analysis is lost.

---

## Multiple ABS Sweeps and Subsectioning

If you will need to run multiple ABS sweeps on a project, it is important to set your ABS caching to Multi-sweep to avoid having to re-calculate your caching data each time you analyze your circuit. But be aware that in order to maintain the validity of the caching data, the subsectioning of the circuit must remain the same. To control the subsectioning you must use the Advanced Subsectioning Controls which you open by selecting *Analysis* ⇒ *Advanced Subsectioning* from the main menu of the project editor.



### TIP

---

The most efficient way to obtain response data for your circuit is to run a single ABS sweep over the entire desired frequency band.

---

The analysis engine, *em*, uses the subsectioning frequency to calculate the wavelength which is used in setting the Maximum subsection size. The default setting used to determine the subsectioning frequency is to use the highest frequency from the present analysis job. If you perform multiple sweeps over different frequency bands then the cache data from one run will be invalid for the next, since the subsectioning frequency would be different. In order to avoid this you should select the **Previous Analysis Only** option which will use the highest frequency from all previous analysis jobs run on the project. In this case, you should analyze the frequency band with the highest upper limit first and take care to ensure that the subsectioning frequency being used provides accurate subsectioning for your circuit. For details on subsectioning, see Chapter 3 "Subsectioning" on page 27.

Another way to keep the subsectioning frequency consistent is to select the Fixed Frequency option for the subsectioning frequency and enter the desired frequency. This ensures that all analysis runs on the project will use the same subsectioning frequency. Again, care should be taken that the subsectioning frequency entered provides the desired accuracy.

## Multi-Sweep Caching Scenarios

The analysis engine always attempts to use any existing data in the project which is consistent with the present analysis. Described below are some common scenarios describing ABS analyses when the ABS caching level is set to Multi-Sweep with Stop/Restart and how data consistency is maintained.

**Higher or Lower Resolution over the Same Frequency Band:** You are running an ABS analysis over the same band as a previous ABS analysis but with higher or lower resolution; an example is shown below. In order for the caching data to be valid for the second analysis, your Advanced Subsectioning controls must be set such that the subsectioning frequency is the same for both runs. If the subsec-

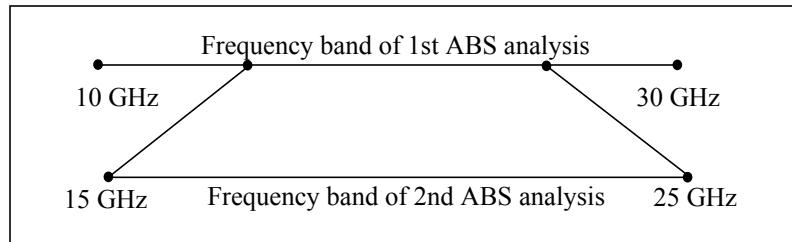
tioning frequency remains the same, the second analysis will usually not require any re-analysis and the results should be provided very quickly. The only exception would be if the difference between the resolutions is unusually high.

Frequency Band 10 - 40 GHz

1st ABS analysis: 10, 10.1, 10.2, 10.3 ... 39.8, 39.9, 40

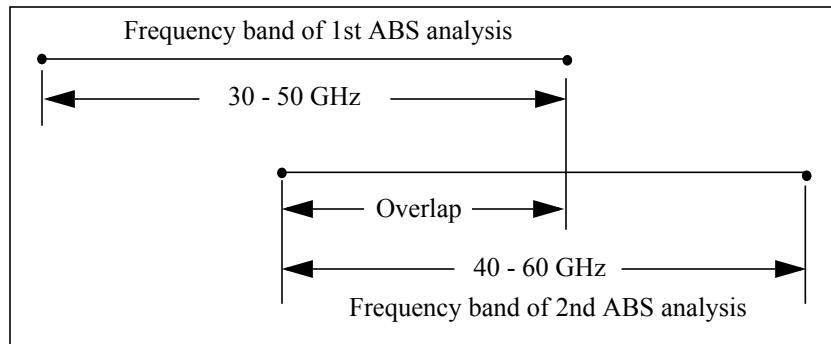
2nd ABS analysis: 10, 10.05, 10.1, 10.15, 10.2 ... 39.85, 39.9, 39.95, 40

**Zoom In:** You are running an ABS analysis over a narrower band than the previous ABS analysis of the project, as shown in the diagram below. This provides higher resolution over the narrower band since the ABS analysis defaults to approximately 300 data points. In order for the caching data to be valid for the second analysis, your Advanced Subsectioning controls must be set such that the subsectioning frequency is the same for both runs. If the subsectioning frequency remains the same, the second analysis will not require any re-analysis and the results should be provided very quickly.



**Extending the Band:** You are running an ABS analysis which overlaps a previous ABS analysis of the project, pictured in the diagram below. The caching data for the overlap between the two analyses will be reused although some calculation may need to be done in the extension of the frequency band where it does not over-

lap with a previous analysis. In order for the caching data to be valid for the second analysis, your Advanced Subsectioning controls must be set such that the subsectioning frequency is the same for both runs.



**Accuracy Assurance:** If you wish to check a particular data point in an ABS analysis and wish to ensure that a full calculation is done at a particular frequency point, you should select a Linear Sweep. This analysis will calculate caching data if Multi-sweep is selected for ABS caching data, but will not use the caching data in producing analysis results.

## Find Minimum and Find Maximum

Find Minimum determines the frequency where the circuit response reaches a minimum. Find Maximum determines the frequency where the circuit response reaches a maximum. You enter a starting and ending frequency in the Start and Stop text entry boxes, respectively and select the parameter for which you wish to determine the minimum or maximum value. **Em** performs an ABS analysis for the frequency band, then uses the adaptive data to determine the frequency where the response reaches a minimum or a maximum.

The Find Minimum and Find Maximum commands are both available in the Frequency Sweep Combinations analysis controls. For more details, see the “Frequency Sweep Combinations” topic in Help in the project editor.

## Parameter Sweep

You may choose either a linear sweep or an adaptive sweep for a parameter sweep. Selecting an adaptive sweep for a parameter sweep is done in the Parameter Sweep Entry dialog box. For more information about parameter sweeps, please see **Chapter 9, “Parameterizing your Project”** on page 171.

The following example assumes that you have already defined the parameter “Width” in your circuit. For more information on inputting a parameter, please refer to “Variables,” page 172.

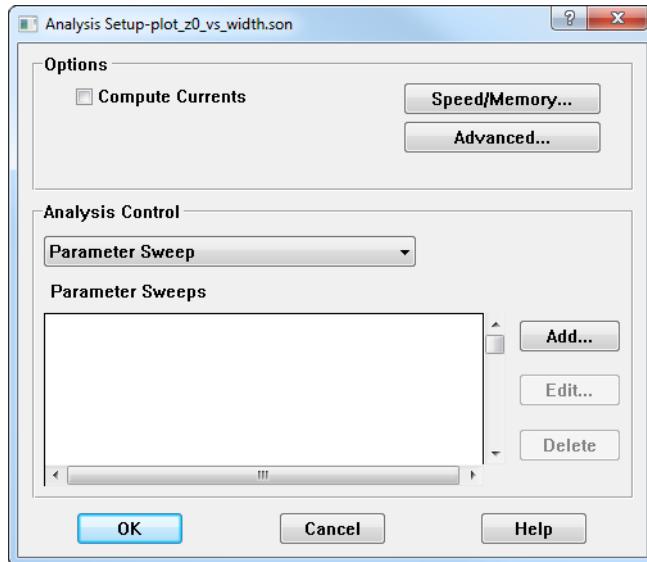
To select ABS for a parameter sweep, do the following:

- 1 **Select Analysis  $\Rightarrow$  Setup from the project editor main menu.**

The Analysis Setup dialog box appears on your display.

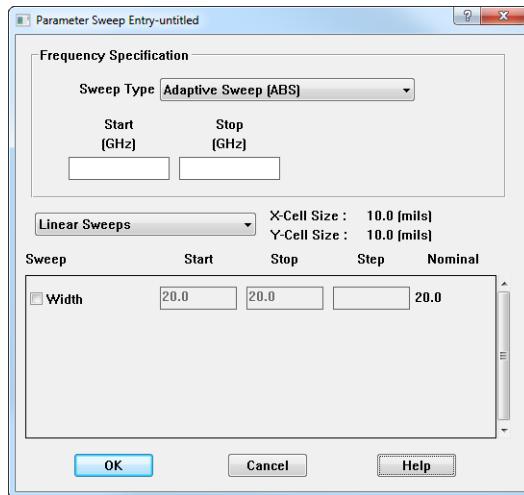
- 2 **Select Parameter Sweep from the Analysis Control drop list.**

The dialog box is updated to allow you to specify the parameter sweep.



**3 Click on the Add button to the right of the Parameter Sweep list box.**

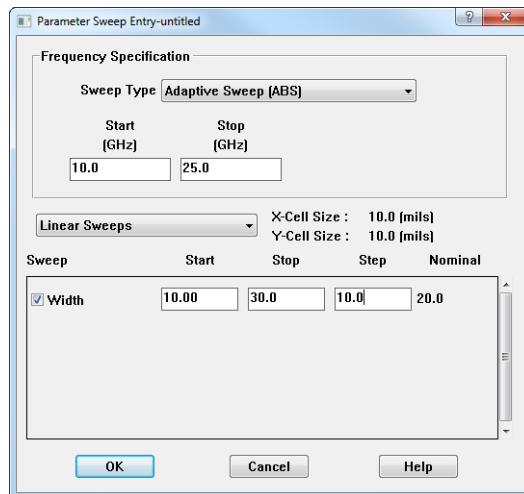
The Parameter Sweep Entry dialog box appears on your display.



**4 Select Adaptive Sweep (ABS) from the Sweep Type drop list to select an adaptive frequency sweep.**

The dialog box changes so that there are only Start and Stop text entry boxes.

**5 Enter the frequency band for the ABS in the Start and Stop text entry boxes.**



This completes setting up an ABS frequency sweep for the parameter sweep. You would also need to select the parameters which you want to use in the parameter sweep and enter their data ranges before closing this dialog box.

# Analysis Issues

There are several issues you should be aware of before using the ABS technique; these are covered below.

## Multiple Box Resonances

You should be aware that circuits with multiple box resonances make it difficult for an ABS analysis to converge. The frequency band for an adaptive sweep should not contain multiple box resonances. If multiple box resonances are present, the number of discrete full analysis points goes up dramatically and synthesis of the data becomes very difficult. If you do not know how to identify box resonances, see Chapter 21, “Box Resonances” for a detailed discussion of box resonances.

## De-embedding

Adaptive data, resulting from an ABS analysis, is either de-embedded or non-de-embedded. With other analysis types, when the de-embedding option is enabled (default), then both de-embedded and non-de-embedded response data is calculated and available for display and output. This is not true for an adaptive sweep.

In an adaptive sweep, if you run with de-embedding enabled, de-embedded data is available for the whole band. Non-de-embedded data is available only for the discrete data points at which full analyses were performed while synthesizing the response.

If you wish to have non-de-embedded data for the whole frequency band, you must perform an adaptive sweep with the de-embed option disabled. Select *Analysis* ⇒ *Setup* from the main menu to open the Analysis Setup dialog box, then click on the Advanced button to open the Advanced Options dialog box. Click on the De-embed checkbox to disable de-embedding. For details on these dialog boxes, please refer to Help for the project editor.

For more information about de-embedding, see Chapter 6 "De-embedding" on page 103.

## Transmission Line Parameters

As part of the de-embedding process, **em** also calculates the transmission line parameters,  $Z_0$  and  $E_{eff}$ . You should be aware that when running an ABS analysis these parameters are only calculated for the discrete data points at which a full analysis is run. If you need the transmission line parameters at more data points, analyze the circuit using a non-ABS analysis.

## Current Density Data

Current density data is calculated for your circuit when the Compute Current Density option is enabled in the Analysis Setup dialog box. For non-ABS sweeps, current density data is calculated for all the response data. For an adaptive sweep, the current density data is only calculated for the discrete data points, therefore, your plot in the current density viewer shows a coarse resolution of your frequency band.

If you wish to calculate the current density data at more points in your band, run a non-ABS sweep for the points in question with the Compute Current Density option enabled.

For more information about the Compute Current Density option, see the help topic “*Analysis*  $\Rightarrow$  *Setup*” in Help for the project editor.

## Ripple in ABS S-Parameters

Please note that when the value of the S-parameters is close to 1 (0 dB) over the entire band you may have small ripples or oscillations in the S-parameter values. This is due to the rational fitting model having too many degrees of freedom when trying to fit a straight line. If this is a problem, it is recommended that you analyze the frequency band in which this occurs with another type of sweep.

## Output Files

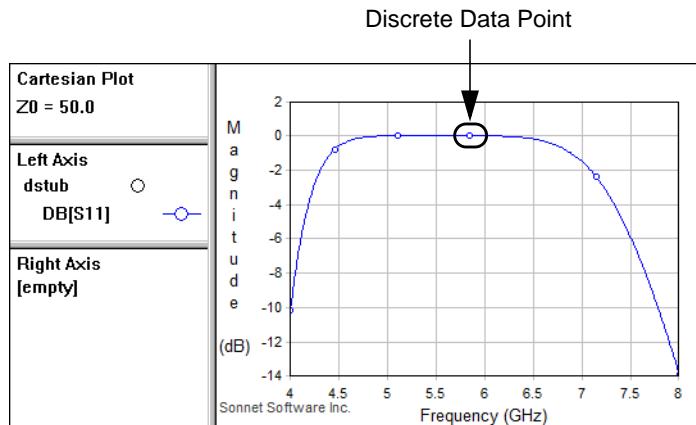
You specify additional output files in the Output Files dialog box which appears on your display when you select *Analysis*  $\Rightarrow$  *Output Files* from the project editor menu. You click on the appropriate button to open the corresponding file entry dialog box. Each entry dialog box has an option pertinent to ABS.

## Response File

When you specify an optional output file for your project, you may select which type of data to output from an adaptive sweep. The data selection is controlled by the Include Adaptive Data checkbox in the File Entry dialog box. If this checkbox is selected, which is the default, then all the adaptive data from an ABS analysis is included in the output file. If this checkbox is cleared, then only the data for the discrete data points is included in the output file. We recommend leaving this box checked.

## Viewing the Adaptive Response

When viewing an adaptive response (ABS) in the response viewer, there are several things of which you need to be aware. The adaptive data is plotted as a line. A symbol indicating a data point only appears at the discrete frequencies at which a full analysis was executed as shown in the picture below.



When exporting data, you may choose to output only the discrete frequencies or the complete response data for the ABS analysis. To output only the discrete frequency data, unselect the Include Adaptive Data checkbox in the Export Data dialog box in the response viewer. For details, see Help in the response viewer.

## ABS Preview

When running an adaptive sweep, discrete frequencies are computed until a rational polynomial is found to “fit” the solution. When the fit meets the convergence criteria, then the adaptive data is calculated and the analysis concludes. A majority of circuits usually require around five to ten frequencies to converge. In some cases, convergence may be more difficult to reach and more discrete frequencies may be required while in some instances convergence cannot be reached. 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.

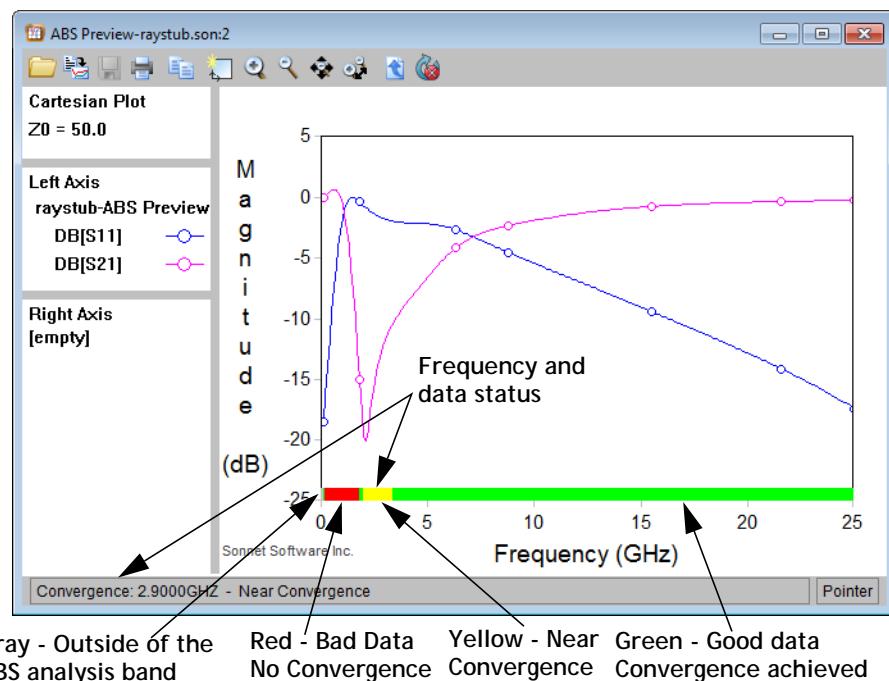
---

**NOTE:** The ABS Caching Level must be set to "Stop/Restart" in the Advanced Options dialog box in order to use this command. See ["ABS Caching Level" on page 159](#).

---

If you stop an analysis running an adaptive sweep before it completes, any ABS data which has been calculated is saved and may be examined in the response viewer using the command *View*  $\Rightarrow$  *ABS Preview*. Note that the an ABS preview does not support the use of equations but all other data curves are available.

When you stop an analysis and open a plot, initially it will have a “jagged” appearance since only the discrete frequencies that have been thus far calculated are displayed. When you select *View*  $\Rightarrow$  *ABS Preview*, a new plot is created with ABS data calculated before the analysis was stopped. A color bar at the bottom of the window indicates the type of data as shown below. If you click in the color band, the frequency you selected and its data status is displayed in the Status Bar at the bottom of the response viewer window. In the case of the graph below, the converged data starts at 3.4 GHz, so running the simulation and using 3.4 GHz as the starting frequency will provide a completed ABS analysis.





# Chapter 9

# Parameterizing your Project

---

Sonnet allows you to assign variables and equations to many properties in order to parameterize your project. Below are some of the reasons you might wish to parameterize your project:

- Avoid creating multiple projects
- Parameter Sweep of a project - allows you to vary a circuit property during an analysis
- Optimization of a project
- Easier construction and maintenance of a project
- Using equations to establish relationships between circuit properties

Variables may be used for the following properties in your project:

- Geometry dimensions
- Metal Properties
- Dielectric Properties
- Ideal Component Values
- Layer Thicknesses

The value of a variable may be changed either by you or by the analysis engine during a parameter sweep or optimization. During a parameter sweep, **em** sweeps the variable values through a user defined range, therefore changing the properties of your circuit. In an optimization, the analysis engine controls the variable value,

within a user defined range, in an attempt to reach a user defined goal. Both parameter sweeps and optimization of a project may be performed over a range of analysis frequencies.

For a tutorial which details how to add variables to your circuit and perform an optimization, please refer to **Chapter 10, “Parameter Sweep and Optimization Tutorial”** on page 197.

For most circuit properties, you simply define a variable and enter that variable in a property field. However, if you wish to vary the size of your geometry--for example, changing the width of a feed line or the length of a polygon--you define a dimension parameter which identifies the dimension you wish to change. Once the dimension parameter is defined, you assign a variable to the dimension parameter.

The first step in performing a parameter sweep or optimization is defining the variables and dimension parameters in the project editor.

## Variables

Variables are user defined circuit attributes that allow the analysis engine, *em*, to modify the circuit in order to perform parameter sweeps and optimization. Variables also provide a quick way for the user to change dimensions in the project editor or multiple elements in a circuit. For example, the length of a transmission line can be assigned the variable “L.” To change the length of the transmission line, you edit the value for “L.” Another example would be a circuit which contains 10 resistors, all of which have the same value. Entering a variable “R” for the resistance of these ideal components allows you to change the value of all 10 resistors by changing the value in only one place.

---

**NOTE:**

Just because it is possible to use a variable in any given property field, does NOT make it necessary to do so. You may enter a nominal value for the field instead of a variable if you do not wish to change that particular property during analysis.

---

A variable’s value may be defined using:

- A constant or nominal value
- Another variable
- An equation

Any of these definitions may be directly entered into a property field. For a more detailed discussion of equations, please see "Equations" on page 175.

## How to Create a Variable

You may use two basic approaches to creating variables in your project. The first approach is to define all the desired variables and then enter these variables as property values. The second approach is to enter a variable in the desired property field as needed. If the variable has not been previously defined, you are prompted to enter a definition. Using the second approach allows you to define variables as needed rather than defining them all ahead of time.

To define a variable do the following:

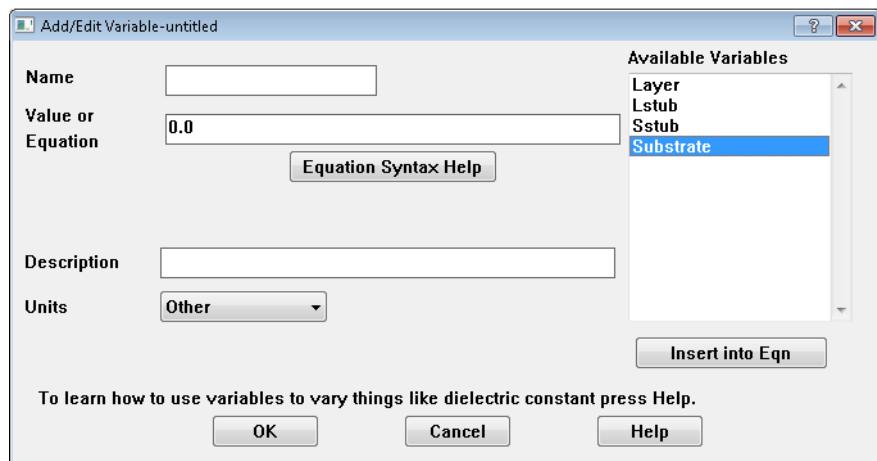
**1 Open the Add/Edit Variable dialog box.**

This dialog box may be opened one of four ways:

- By selecting *Circuit*  $\Rightarrow$  *Add Variable* from the project editor main menu.
- Select *Circuit*  $\Rightarrow$  *Variable List* from the project editor main menu, then click on the **Add** or **Edit** button in the Variable List window which appears.
- By entering an undefined variable in a property field in another dialog box. When the dialog box is closed the Add/Edit Variable dialog box is opened so you can define the variable.
- By selecting <Add Variable> from a drop list of a property field in another dialog box.



When you select <Add Variable> the Add/Edit dialog box is opened so you can define the variable.



**2 Enter the desired variable name in the Name text entry box.**

If you opened this dialog box by entering an undefined variable in another dialog box, this field will already contain that name.

**3 Enter a constant value, another variable or an equation that defines the value of the variable in the Value or Equation text entry box.**

For information about the functions available and equation syntax, click on the **Equation Syntax Help** button. If you wish to use another variable in the equation, you may select it from the list of Available Variables and click on the **Insert into Eqn** button. See the next section for more information on Equations. The variable name will be entered at the present location of the cursor in the Value text entry box.

**4 Enter a brief description of the variable, if desired, in the Description text entry box.**

This description appears in the Variable list dialog box and allows you to identify the purpose of the variable or which property it is being used to define. If the variable is created when editing a dialog box (Dielectric Layers, Metal Types, etc.) a default description is provided by the software. This may be changed by entering a different description here. This field is not required.

**5 Select the desired units for the variable from the Units drop list.**

Available units are the length, frequency, resistance, capacitance and inductance units presently set in your project. If none of these are suitable, then select Other. It is important to select the units for your variable so that if you change the units used in your project, the value of the variable is converted correctly.

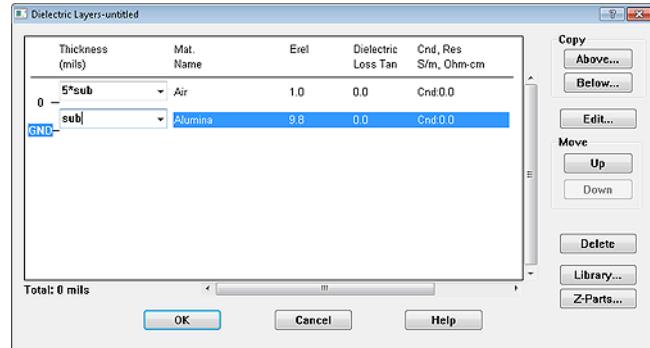
## Equations

As mentioned earlier, a variable may be defined by a constant, another variable or an equation. An equation is composed of constants and/or variables. Below are some examples of valid equations.

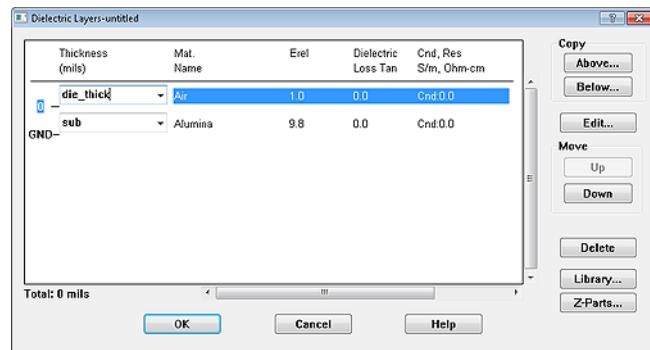
- $3*3$
- $3*pi$
- $3*H$
- $\sin(H)$
- $\sqrt{H}$

The last three equations are ones in which one variable is used to define another. This allows you to relate properties in your circuit such that changing one effects the other, maintaining a set relationship between them. For example, you wish to define a dielectric layer which is always five times the thickness of your substrate. To do so, you define a variable “sub” which you would enter as the thickness of your substrate. Then you would enter “ $5*sub$ ” as the value for the thickness of the dielectric layer. There are two methods you may use to do this. The first is to define another variable “die\_thick” which you define as “ $5*sub$ ” and the second is to simply enter the equation “ $5*sub$ ” as the thickness of the dielectric layer. You may enter an equation in any field in which you may enter a variable.

In this case, only one variable “sub” is defined and the equation “ $5*sub$ ” is entered directly in the Dielectric Layers dialog box as the thickness of the dielectric layer. The only variable which appears in the Variable List (opened using the command *Circuit*  $\Rightarrow$  *Variable List*) is “sub.”



In this case, two variables are defined: “sub” and “die\_thick.” The value entered for the variable “die\_thick” is the equation “ $5*sub$ .” Both variables appear in the Variable List.



Available functions for equations include:

- Operators such as addition, subtraction, multiplication, etc.
- Logarithmic functions
- Complex Math functions
- Trigonometric functions
- General Mathematical functions such as maximum or minimum
- Conversion functions for converting units
- Table function to use for data lookup

There are also constants available for equations including project constants such as the cell size and box size and the frequency which is discussed more below.

For complete details on all the functions and constants available for equations and their syntax, please refer to the equation syntax help topics in Help. You may access these topics by clicking on the **Equation Syntax Help** button in the Add/Edit Variable dialog box.

If you enter an equation which uses a variable as the definition of another variable, then the variable defined by the equation is dependent on the variable used in the equation. This is discussed later in the chapter, in "Dependent Variables" on page 176.

### Frequency Dependency

There is a frequency constant, FREQ, available for use in Sonnet equations. This allows you to model properties whose characteristics are frequency dependent such as a dielectric. It is important to be aware that if a variable uses the FREQ constant in its definition that the value of the variable changes during the analysis even if the variable is NOT selected for the parameter sweep or optimization.

### Dependent Variables

One variable is dependent upon another if the value of a variable is defined by an equation that uses another variable. As the value of the variable in the equation is changed, so is the dependent variable. If a variable is dependent, you may not directly edit its nominal value; instead, you change its value by changing the value of the variable on which it is dependent. You may not delete a dependent variable. If you wish to delete it, you must first remove the dependency by changing the definition of the dependent variable.

Variables which do not depend on another variable for their value are independent variables. Only independent variables may be selected for a parameter sweep. To vary a dependent variable in a parameter sweep, you must select the variable on which it depends. For more details on parameter sweeps, see "Parameter Sweep" on page 189.

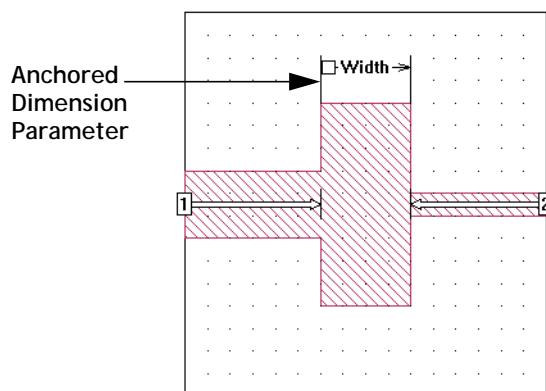
### Circular Dependencies in Variables

Care should be taken when adding dependent variables to your circuit that they do not form a circular dependency. A circular dependency is formed when two variables are dependent on each other. This can happen for two variables or multiple variables. In the case of multiple variables, the dependency extends from the first variable through all the variables until the first variable is dependent upon the last.

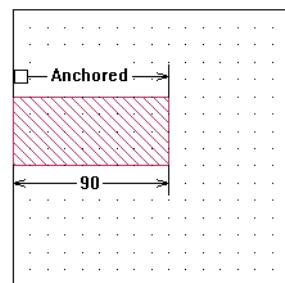
An example of a circuit dependency would be the two equations  $A=2*B$  and  $B=\sin(A)$ . For variables, if the project editor detects a circular dependency, an error message appears and you are forced to correct the problem before proceeding. You need to redefine one of the variables such that it is no longer dependent on the other. A correction of the example would be  $A=2*B$  and  $B=\sin(C)$ .

## Dimension Parameters

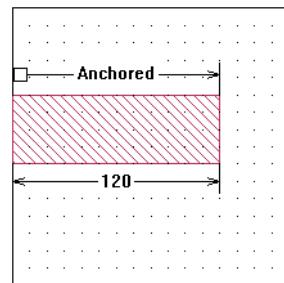
A dimension parameter allows you to identify dimensions in a geometry project and assign a variable to the dimension, which allows you to vary those dimensions within an analysis. The initial value of the dimension parameter is the length that appears in your circuit. This is the nominal value of the variable assigned to the dimension parameter. If you change the nominal value of the variable, then the circuit is redrawn with that length.



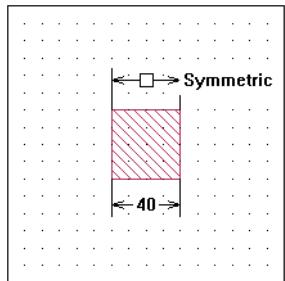
There are three types of dimension parameters: anchored, symmetric, and radial. For brevity, we refer to anchored dimension parameters as anchored parameters, symmetric dimension parameters as symmetric parameters, and radial dimension parameters as radial parameters for the remainder of this discussion. An anchored parameter allows you to fix one end of a parameter then vary its length extending from that point. A symmetric parameter allows you to fix the center point of a dimension parameter and vary the distance it extends on each side. A radial parameter allows you to fix one end of a parameter then radiate out from that fixed point; the direction is not restricted to the x or y direction, but may extend at an angle. The three dimension parameter types are described in detail in the following sections.



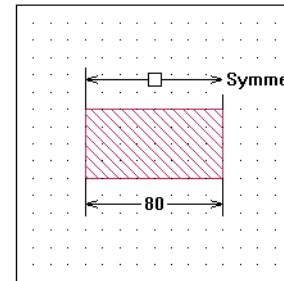
**Nominal Value = 90 mils**



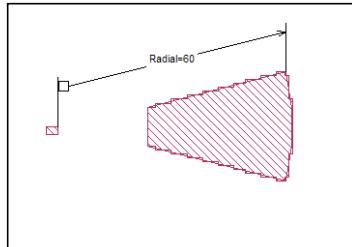
**Nominal Value = 120 mils**



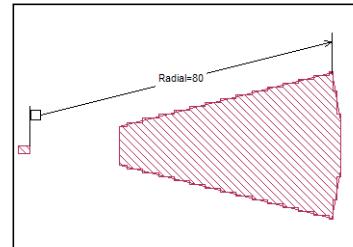
**Nominal Value = 40 mils**



**Nominal Value = 80 mils**



**Nominal Value = 60 mils**



**Nominal Value = 80 mils**

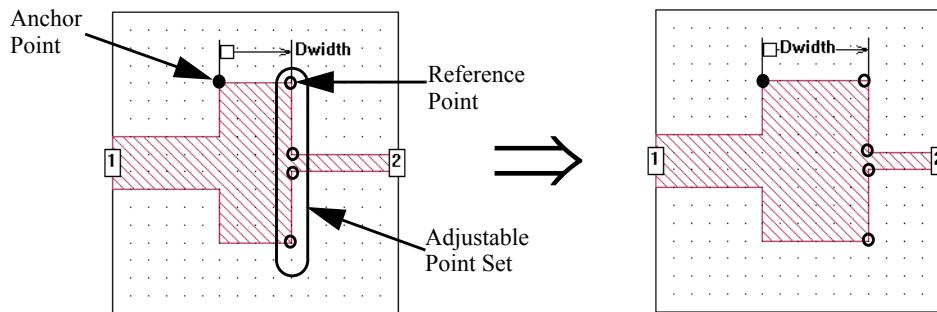
## Anchored Dimension Parameters

An anchored parameter defines a dimension using an anchor point, a reference point and an adjustable point set. The nominal value of the parameter is defined by the distance between the anchor point and the reference point. When the dimension is varied, each point moves relative to the anchor point. When defining your dimension parameter, you perform the following steps:

- You select the anchor first. This is the fixed starting point for the parameter.
- You select the reference point. The reference is the first point in the adjustable point set, which is the set of points moved relative to the anchor point when the value of the dimension parameter is changed. The distance from the anchor point to the reference point is the value of the dimension parameter. When the value of the dimension parameter is changed, the anchor retains the same position, but the reference point, moves to a new position.
- Third, you select any additional points in your circuit you wish to move when the reference point moves; this is the adjustable point set. As the value of the dimension parameter varies and the reference point is moved, the positions of the points in the adjustable point set also move. There is a setting associated with each dimension parameter that determines how the adjustable point set is moved. With the default, and simplest, option each point in the adjustable point set retains its relative distance from the reference point. For a discussion of all the options controlling moving the adjustable point set, see "Moving Adjustable Point Sets" on page 183.

Note that the anchored parameter is always defined as the distance between the anchor and the reference point in either the X direction or the Y direction, never as a diagonal distance between them.

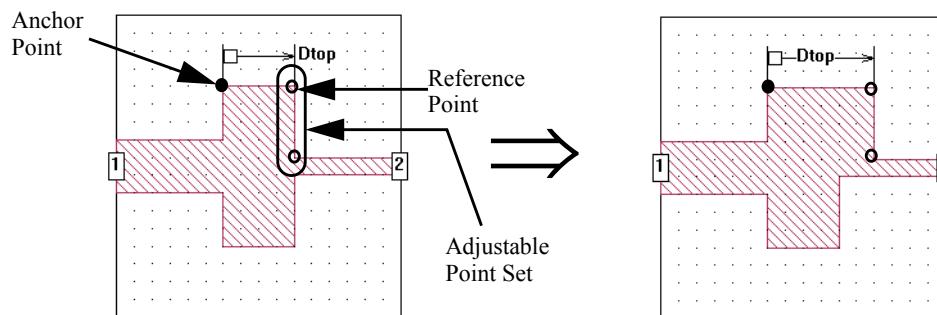
Two examples of anchored parameters, each at two different nominal values, are illustrated below. This example uses the default setting for how the adjustable point set moves.



Dwidth, nominal value = 40 mils

Dwidth, nominal value = 60 mils

Notice that although the top and bottom examples have identical anchor and reference points and starting and ending nominal values, that the resulting polygon on the top differs from that on the bottom due to a different adjustable point set (the point set is highlighted by the oval).



Dtop, nominal value = 40 mils

Dtop, nominal value = 60 mils



### TIP

---

Once you have finished adding a dimension parameter, double-clicking on the variable name and entering a new nominal value allows you to check whether the dimension parameter was defined correctly.

---

## Symmetrical Dimension Parameters

A symmetric parameter defines a dimension using two reference points and their respective adjustable point sets. The nominal value of the parameter is the distance between the two reference points. The anchor is defined as the midpoint between

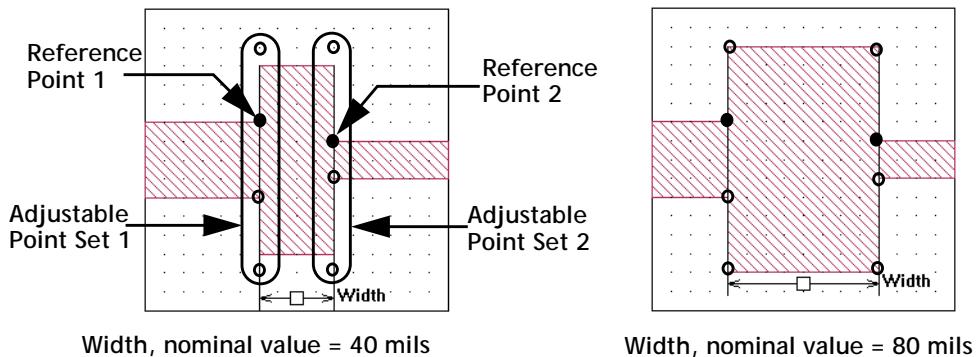
the two reference points; the user does not define an anchor point for a symmetric parameter. When the dimension is varied, each point moves relative to the anchor point. When defining your dimension parameter, you perform the following steps:

- Select the first reference point.
- Select the adjustable point set that moves with the first reference point.
- Select the second reference point. The value of the parameter is the distance between the two reference points.
- Select the adjustable point set that moves with the second reference point.

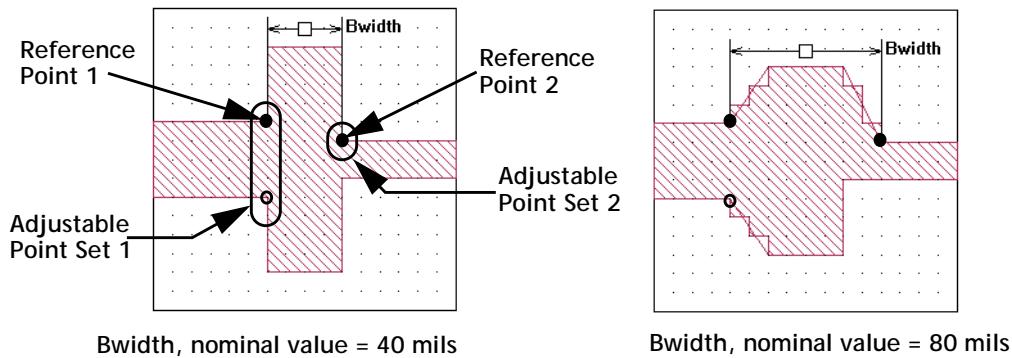
There is a setting associated with each dimension parameter that determines how the adjustable point sets are moved. With the default, and simplest, option each point set moves one half the distance of the difference between the present value and the previous value out from the middle point, maintaining its relative positive to the reference point with which it is associated. For a discussion of all the options controlling moving adjustable point sets, see "Moving Adjustable Point Sets" on page 183.

Note that the symmetric parameter is always defined as the distance between the reference points in either the X direction or the Y direction, never as a diagonal distance between them.

Two examples of symmetric parameters, each at two different nominal values, are illustrated below. This example uses the default setting for how the adjustable point set moves.

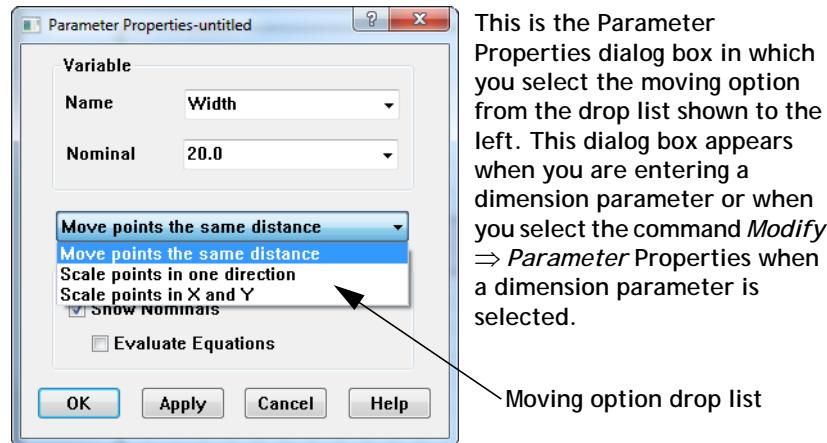


Notice that although the top and bottom examples have identical reference points and starting and ending nominal values, that the resulting polygon on the top differs from that on the bottom due to a different adjustable point set (the point set is highlighted by the oval).



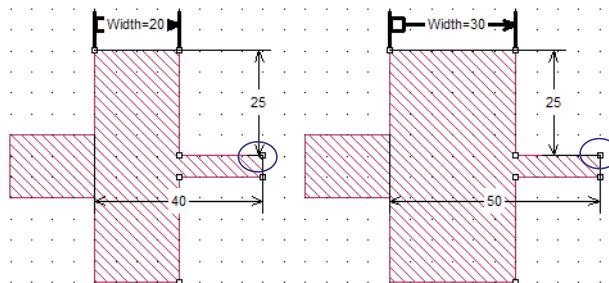
## Moving Adjustable Point Sets

Anchored and symmetric parameters have a setting which controls how the adjustable point sets are moved. There are three settings: Move points the same distance, Scale points in one direction, and Scale points in X and Y. Each type is explained below.



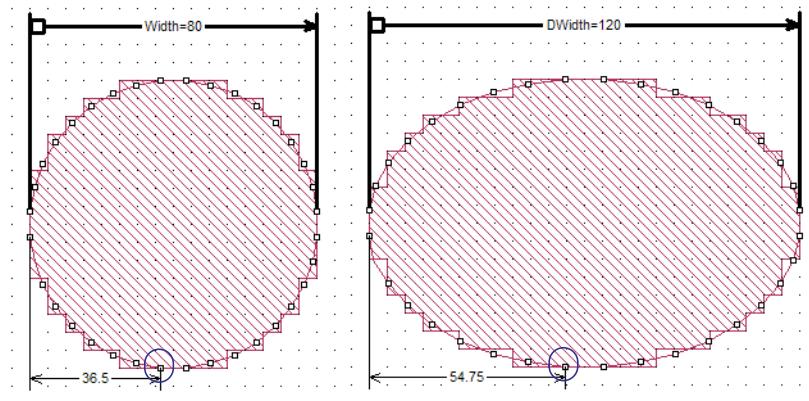
**Move Points the Same Distance:** This setting moves the adjustable point set the same distance as the reference point is moved and all the points in the adjustable point set maintain their relative position. This is the default setting. An example using an anchored dimension parameter is shown below. Note that the adjustable point sets are highlighted.

The anchored dimension parameter is increased by 10 mils, so each point in the adjustable point set is moved away from the anchor point by 10 mils. As shown in the illustration, the original distance from the anchor to the circled point was 40 mils; when the value of the parameter is increased, the point was moved by 10 mils so that it is now 50 mils away from the anchor point.



**Scale Points in one direction:** When using this setting, the geometry controlled by the parameter is scaled or stretched along either the x or y axis depending on the orientation of the parameter. Each point in the adjustable point set is moved by an amount based on its relative distance from the anchor. Points closer to the anchor are moved a smaller distance than points further away from the anchor. For symmetric parameters, the anchor is the center point between the two reference points.

The scaling factor used is the ratio of the new nominal value to the present nominal value. Each point is moved by a delta calculated by multiplying its present distance from the anchor point by the scaling factor.

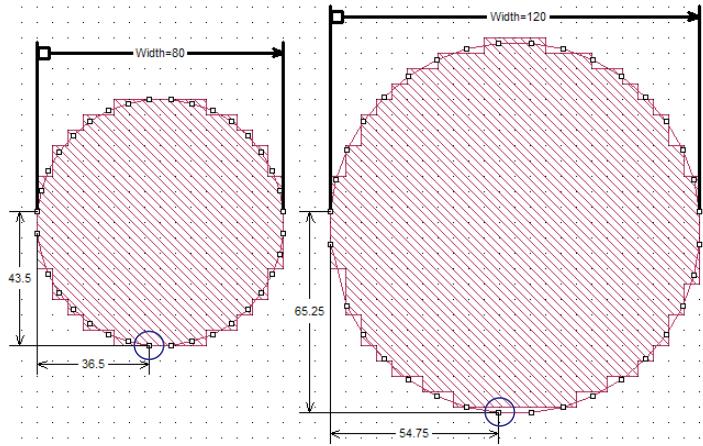


**Scale Points in x and y:** When using this setting, the geometry controlled by the parameter is scaled or stretched along both the x and y axis keeping the proportions of the geometry the same. Each point in the adjustable point set is moved by an amount based on its relative distance from the anchor. Points closer to the anchor are moved a smaller distance than points further away from the anchor. For symmetric parameters, the anchor is the center point between the two reference points.

The scaling factor used is the ratio of the new nominal value to the present nominal value. Each point is moved by a delta calculated by multiplying its present distance from the anchor point by the scaling factor.

An anchored dimension parameter being scaled in both directions is shown below. The scaling factor is 1.5 since the new nominal value is 120 mils and the present value is 80 mils. As shown in the illustration, the original distance from the anchor of the circled point was 36.5 mils along the x axis and 43.5 mils along the y axis;

when the value of the parameter is increased the point was moved to 54.75 mils away from the anchor point along the x axis and 65.25 mils along the y axis; the original distances multiplied by the scaling factor of 1.5.

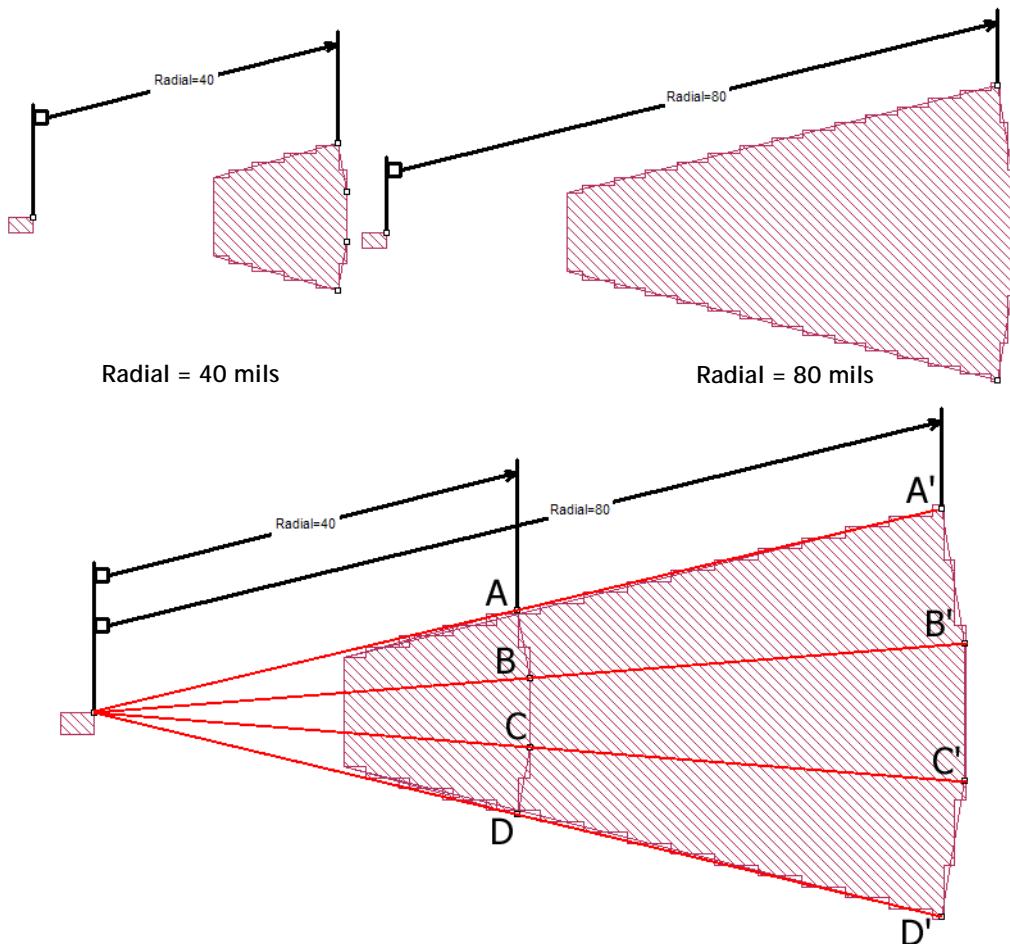


## Radial Dimension Parameters

A radial parameter defines a dimension using an anchor point, a reference point and an adjustable point set. The nominal value of the parameter is defined by the distance between the anchor point and the reference point. When the dimension is varied, each point moves along a line extending from the anchor point through its original position. Setting up a radial parameter is identical to setting up an anchored parameter. When defining your radial parameter, you perform the following steps:

- You select the anchor first. This is the fixed starting point for the parameter.
- You select the reference point. The reference is the first point in the adjustable point set, which is the set of points moved relative to the anchor point when the value of the radial parameter is changed. The diagonal distance from the anchor point to the reference point is the value of the dimension parameter. When the value of the dimension parameter is changed, the anchor retains the same position, but the reference point, along with the adjustable point set, moves to a new position.
- Third, you select any additional points in your circuit you wish to move when the reference point moves; this is the adjustable point set. As the value of the dimension parameter is varied, the reference point, as well as the rest of the adjustable point set, is moved accordingly. Each of the

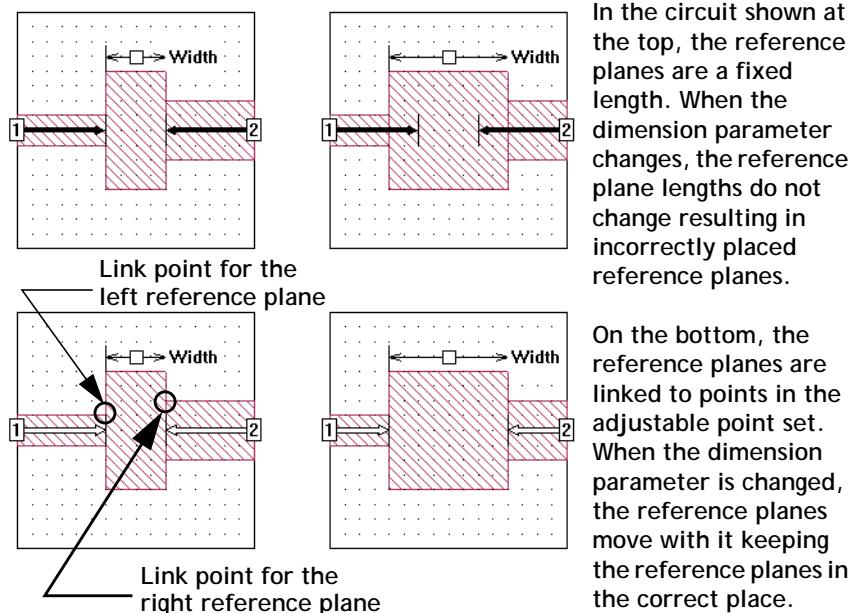
points in the point set move the same distance along their radius from the anchor point. An example is shown below.



The view of Radial=40 mils is superimposed over the view of Radial=80 mils. As can be seen, each adjustable point moves the same distance along its radius from the anchor point to its new position. Therefore line segments AA', BB', CC' and DD' are all equal to 40 mils, the difference between the two values of the dimension parameter Radial.

## Reference Planes

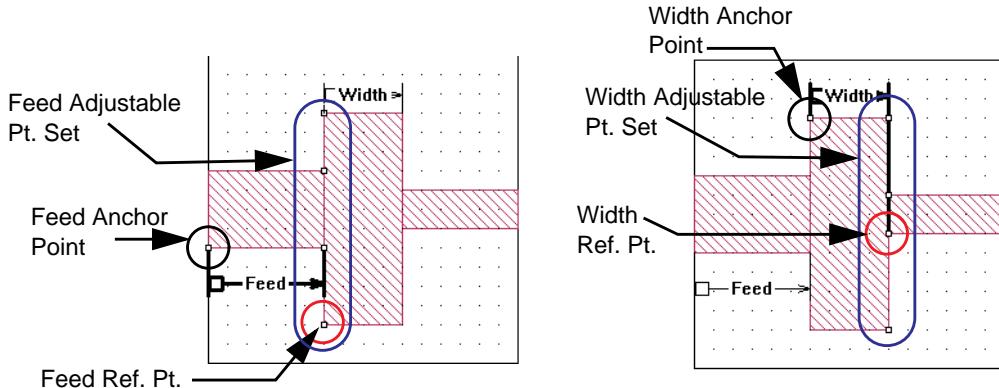
Linking your reference plane to a dimension parameter moves the reference plane in response to a change in the dimension parameter helping to ensure correct placement of your reference plane. In order to do this, link your reference plane to a point in the adjustable point set for the desired dimension parameter as illustrated below.



## Dependent Dimension Parameters

One dimension parameter is dependent upon another if the anchor point and/or the reference point(s) for the second parameter are part of an adjustable point set for the first parameter. This is allowed as long as a circular dependency is not formed; see the following section. You need to be aware of dependent dimension parameters so that you can take into consideration the complete impact on your circuit when the value of the primary dimension parameter is changed. When the primary dimension parameter is changed, a dependent parameter is adjusted, i.e., the anchor point or reference point is moved, along with the primary dimension parameter.

eter on which they depend. A picture of a dependent dimension parameter is shown below with the Anchor and Reference points highlighted as well as the point sets.



The dimension parameter "Width" is dependent on the dimension parameter "Feed". Note that the anchor point for the dimension parameter "Width", shown on the right, is part of the adjustable point set for the dimension parameter "Feed," shown on the left.

You may use the Select Dependents command in the project editor to determine if there are any dependent dimension parameters. With the project open in the project editor, do the following:

- 1 Right-click on a dimension parameter in your circuit.**

The variable name and the dimension parameter point set is highlighted and a pop-up menu appears on your display.

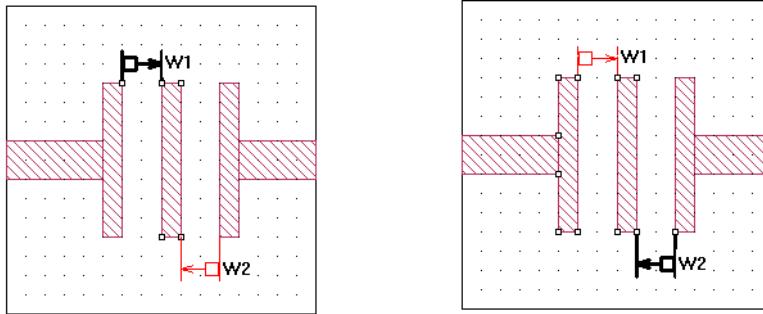
- 2 Select "Select Dependents" from the pop-up menu.**

If any dependent dimension parameters are present in your geometry, the dependent dimension parameter is highlighted.

## Circular Dependencies in Parameters

Care should be taken when adding dimension parameters to your circuit that they do not form a circular dependency. A circular dependency is formed when two dimension parameters are dependent on each other. This can happen for two dimension parameters or multiple dimension parameters. In the case of multiple dimension parameters, the dependency extends from the first dimension parameter through all the dimension parameters until the first dimension parameter is dependent upon the last.

If the project editor detects a circular dependency of dimension parameters, all the involved dimension parameters are shown in red. An example of two dimension parameters in a circular dependency is shown below.



A circular dependency is an error condition and must be corrected before you can analyze your circuit. You need to redefine one of the dimension parameters such that it is no longer dependent on the other.

## Parameter Sweep

Once you have defined variables in your circuit, you may use the variables to perform a parameter sweep. A parameter sweep allows you to run through a set of analyses, with different variable values, all in one step. This allows you to see how changes in your circuit affect the response of your circuit.

You set up the parameter sweep in the Analysis Setup dialog box in the project editor. To access this dialog box, select *Analysis* ⇒ *Setup* from the main menu of the project editor. In the dialog box which appears, select “Parameter Sweep” from the Analysis Control drop list. For details on using this dialog box, please refer to Help.

All frequency sweep types are available for a parameter sweep. You may allow one or multiple variables to change when running a parameter sweep.

Sweep modes allow you to control what variable combinations are analyzed for a parameter sweep. Sweep modes for a parameter sweep include Linear Sweep, Corner Sweep, Sensitivity Sweep and Mixed Sweep Combinations.

- A linear sweep analyzes your circuit at every available combination of variable values.
- A corner sweep allows you to analyze the “extremes” of your variables by analyzing at all the combinations of the minimum and maximum

values of the variables as well as at the nominal values. This allows you to see how your circuit varies over the data range of your variable. For a corner sweep, the number of combinations which are analyzed is

$$N_c = 2^{N_v} + 1$$

where  $N_c$  is the number of combinations and  $N_v$  is the number of independent variables selected for the parameter sweep.

- A Sensitivity Sweep allows you to see how “sensitive” your circuit is to changes in any given variable. The nominal value of each variable is combined with the maximum and minimum values of the other variables as well as the nominal values. For a sensitivity sweep, the number of combinations which are analyzed is

$$N_c = 2 \bullet N_v + 1$$

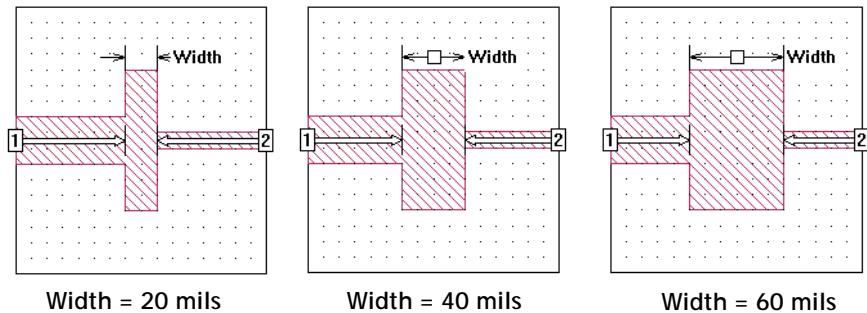
where  $N_c$  is the number of combinations and  $N_v$  is the number of independent variables selected for the parameter sweep.

- Mixed Sweep Combinations allows you to define a range of values for each variable, then perform an analysis at each possible combination of variable values. There are four types of sweeps available: Fixed, Linear, Linear (# steps), and Exponential.

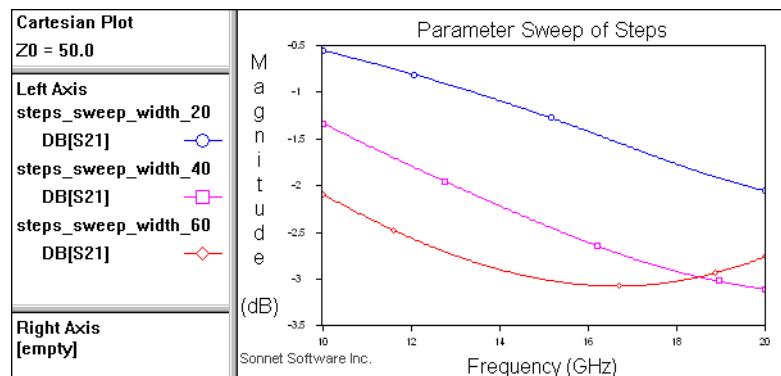
### Parameter Sweep Example

For example, the graphics below illustrate a linear parameter sweep of the circuit “steps” with a single dimension parameter whose value is defined as the variable “Width.” The parameter sweep starts with a value of 20 mils for the variable and

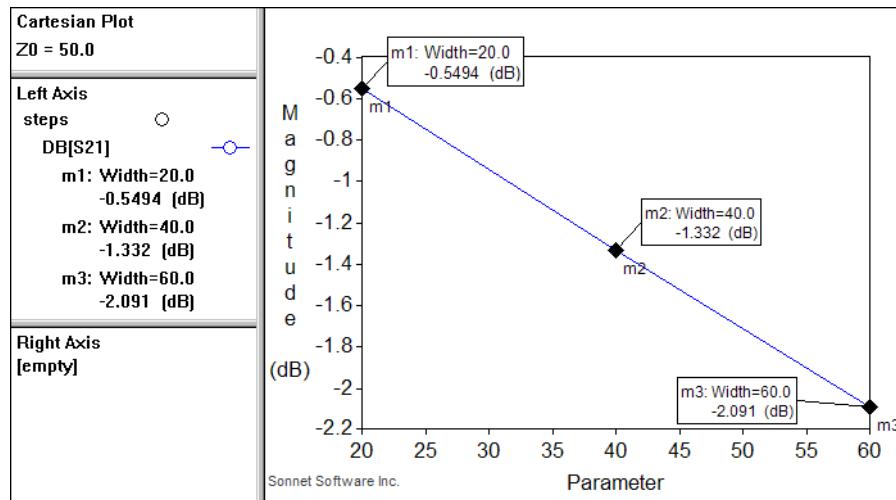
increases in steps of 20 until the variable's value is 60. **Em** automatically performs an analysis at each specified frequency for each circuit shown below when the parameter sweep is executed.



In the case of this sweep, an ABS analysis from 10 - 20 GHz was performed. The response data for the parameterization is shown in the graph below.



You may also plot your response data against the parameter values by selecting the *Graph* ⇒ *Plot Over* ⇒ *Parameter* command in the response viewer. Shown below is the plot of the magnitude in dB of S21 versus the variable Width at 10 Hz. Data markers have been added to the plot for clarity (*Graph* ⇒ *Marker* ⇒ *Add Data Marker*).



Since an analysis of the circuit at each combination of variable values is executed for each specified analysis frequency, care should be taken when choosing data ranges. The higher the number of analysis frequencies and variable values, the higher the number of analyses that must be computed by *em*. The number of combinations specified is displayed in the project editor.

## Optimization

You may also use variables to perform an optimization on your circuit. An optimization allows you to specify goals - the desired response of your circuit - and the data range for the variables(s) over which you seek the response. The software, using a conjugate gradient method, iterates through multiple variable values, searching for the best set which meets your desired goals.

The conjugate gradient optimizer begins by analyzing the circuit at the nominal variable values. It then perturbs each variable individually, while holding the others fixed at their nominal values, to determine the gradient of the error function for that variable. Once it has perturbed each variable, it then performs a line search in the direction of decreasing error function for all variables. After some iterations on the line search, the optimizer again calculates the gradients for all variables by

perturbing them from their present “best” values. Following this, a new line search is performed. This continues until one of three conditions are met: 1) the error goes to zero 2) the error after the present line search is no better than the error from the previous line search 3) the maximum number of iterations is reached. When one of these three conditions is met, the optimizer halts.

The equations used to determine the optimization goal error are as follows:

$$EqualOperatorError = \frac{\sum(Simulation(f) - Target(f))}{N}$$

where:

*Target(f)*: Target value of measurement at frequency f.

*Simulation(f)*: Simulated value of measurement at frequency f.

*N*: Number of frequencies in optimization goal.

$$GreaterThanOperatorError = \frac{\sum(Target(f)) - (MIN[Simulation(f), Target(f)]))}{N}$$

where:

*Target(f)*: Target value of measurement at frequency f.

*Simulation(f)*: Simulated value of measurement at frequency f.

*MIN[]*: Determines minimum value of the specified arguments.

*N*: Number of frequencies in optimization goal.

$$LessThanOperatorError = \frac{\sum(Simulation(f)) - (MIN[Simulation(f), Target(f)])}{N}$$

where:

*Target(f)*: Target value of measurement at frequency f.

*Simulation(f)*: Simulated value of measurement at frequency f.

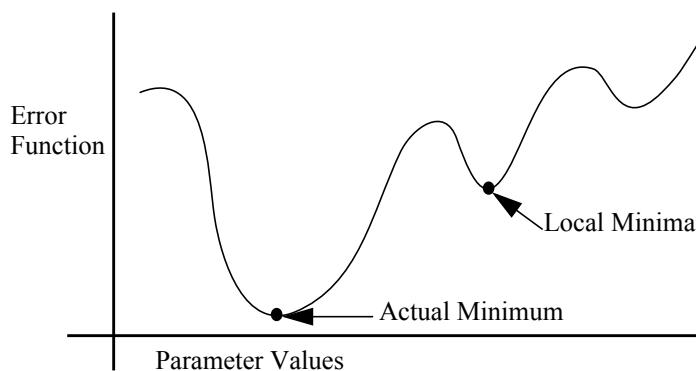
*MIN[]*: Determines minimum value of the specified arguments.

*N*: Number of frequencies in optimization goal.

Setting up an optimization consists of five parts:

- Specifying optimization frequencies
- Specifying your goals
- Choosing which variables to vary
- Specifying data ranges for the chosen variables
- Specifying the maximum number of iterations

Care should be taken when setting the nominal values for the variables to be optimized. The optimizer starts at the nominal values and converges to the minima which is closest to those nominal values. Thus, it is highly recommended that you perform some pre-analysis prior to doing the optimization to ensure that the nominal values are in the right value range when the optimizer is started. Otherwise, the optimizer may converge to a local minima for which the error is not the minimum achievable value, as pictured below.



You specify a goal by identifying a particular measurement and what value you desire it to be. For example  $S_{11} < -20$  dB. Keep in mind that the goals you specify may not be possible to satisfy. **Em** finds the solution with the least error.

You may also specify a goal by equating a measurement in one network to a measurement in another network or file. For example, you may set  $S_{11}$  for network “Model” equal to  $S_{11}$  for network “Measured.” Likewise, you may equate  $S_{11}$  for network “Model” to  $S_{11}$  for data file “meas.s2p.”

You may select one or multiple variables to optimize. For each variable that you select you must specify minimum and maximum bounds. The analysis limits the variables to values within the specified bounds.

Variables that are used in an optimization have a granularity value assigned to them; the granularity defines the finest resolution, the smallest interval between values, of a variable for which **em** will do a full electromagnetic simulation during optimization. For values which occur between those set by this resolution, **em** performs an interpolation to produce the analysis data. By default, the software determines the granularity, but you may enter a value manually.

You specify the number of iterations. For each iteration, **em** selects a value for each of the variables included in the optimization, then analyzes the circuit at each frequency specified in the goals. Depending on the complexity of the circuit, the number of analysis frequencies and the number of variable combinations, an optimization may take a significant amount of processing time. The number of iter-

ations provides a measure of control over the process. Note that the number of iterations is a maximum. An optimization can stop after fewer iterations if the optimization goal is achieved or it finds a minima (finds no improvement in the error in further iterations).

Once the optimization is complete, the user has a choice of accepting the optimal values for the variables resulting from the **em** analysis. Note that for dimension parameters, if the results of the optimization are accepted, the actual metalization in the project editor is the closest approximation which fits the present grid settings. As a matter of fact, **em** analyzes “snapped” circuits and interpolates to produce responses for circuits which do not exactly fit the grid. For more information about the grid, see Chapter 3 "Subsectioning" on page 27.



# Chapter 10

# Parameter Sweep and Optimization Tutorial

---

This tutorial shows you how to set up variables and dimension parameters in a circuit, set up and execute a parameter sweep, set up and execute an optimization and view the results of both a parameter sweep and an optimization. For a detailed discussion of parameterization and optimization, please refer to Chapter 9.

This tutorial presumes that you are familiar with Sonnet Software, especially the project editor and the analysis monitor. If you are new to Sonnet, please review the tutorials in Chapter 4 and Chapter 5 of the **Getting Started** manual before performing this tutorial.

This tutorial uses the Sonnet example Par\_dstub. The example may be obtained using the Sonnet Example browser. You may access the Sonnet Example browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window. Save the example to your working directory.

This is an example of a microstrip interdigital bandstop filter. This circuit is used to perform a parameter sweep and optimization. Most parameter sweeps and optimizations will present more of a challenge, but we have deliberately chosen a simple example to more clearly demonstrate Sonnet's methodology.

Our goal is to design the bandstop filter such that a stopband exists from 5 - 6 GHz and the passbands are from 1 - 4 GHZ and 7-10 GHz.

**1 Open the copy you made of the par\_dstub example in the project editor.**

Par\_stub is displayed in the project editor.

## Setting Up Dimension Parameters

Before executing either a parameter sweep or optimization, it is first necessary to add your variables and dimension parameters. Variables should be used for the properties most critical to the circuit's response. Dimension parameters should be added to those dimensions you deem critical to the circuit's response. These properties and dimensions are the most likely to change as the design progresses.

For this example, you will enter three dimension parameters: two anchored dimension parameters which are linked and one symmetric dimension parameter. Anchored dimension parameters are referred to as anchored parameters and symmetric dimension parameters are referred to as symmetric parameters for the rest of the tutorial.

An anchored parameter is one in which one end of the dimension parameter is a fixed point with a point set which moves in reference to that fixed, or anchored point. A symmetric parameter is one with two reference points and two point sets which move relative to the center point between the reference points.

Dimension parameters which appear in more than one place in a circuit but are of the same length and assigned the same variable are linked. Changing the value of the variable changes the length of both.

For a detailed discussion of variables and their definitions, please refer to "Variables" on page 172.

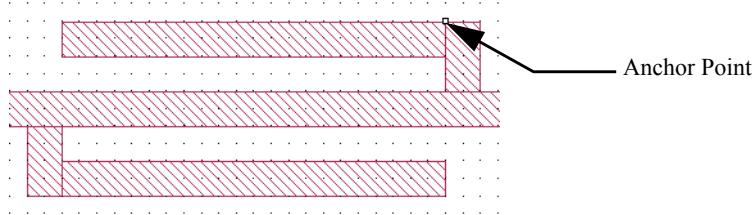
### Anchored Parameters

The linked anchored parameters are input first, followed by the symmetric.

**2 Select *Tools*  $\Rightarrow$  *Add Dimension Parameter*  $\Rightarrow$  *Add Anchored* from the project editor's main menu.**

This places the project editor in Add an Anchored Parameter mode indicated by the change in cursor. Note that the message "Click Mouse to Specify the Anchor Point" appears in the status bar at the bottom of the project editor window. As you add a parameter, directions for each step appear in the status bar.

- 3 To specify the Anchor point for the parameter, click the mouse on the corner of the upper stub, as shown in the picture below.



The anchor point is indicated by a small square which appears at the point you clicked. The next step is to select the reference point.



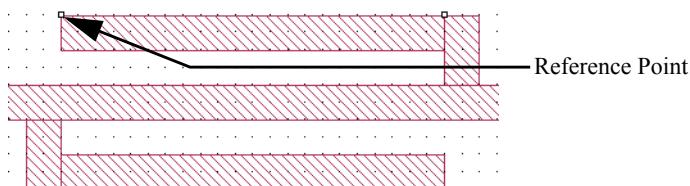
### TIP

---

If you select the wrong point for either the anchor or reference point, press the Escape key to exit without adding a parameter. You may then start over.

---

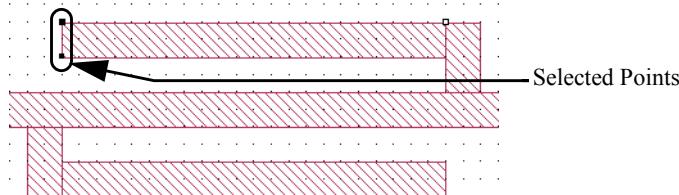
- 4 Click on the top left end of the top stub to add the reference point.



The reference point is indicated by a small square which appears at the point you clicked.

In the next step, you select the rest of the adjustable point set. Points may be selected by clicking on individual points or by lassoing a set of points with your mouse. You do not need to select the reference point since it is automatically included in the adjustable point set.

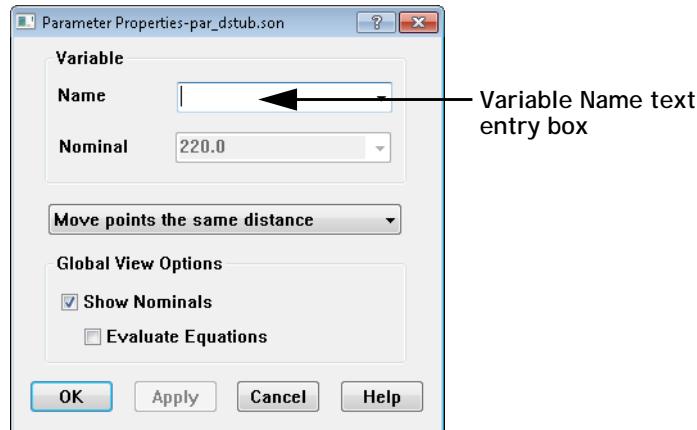
**5 Drag the mouse until both points on the end of the stub are selected.**



These points will be added to the adjustable point set. When the reference point moves in response to a change in the parameter value, these points move relative to the reference point.

**6 Once all the desired points are selected, press Enter.**

This completes the dimension parameter creation. The Parameter Properties dialog box appears on your display.

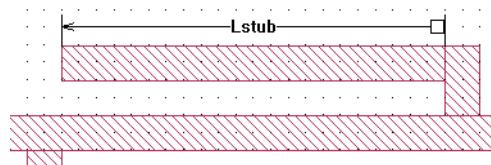


**7 Enter the variable “Lstub” in the Variable Name text entry box in the Parameter Properties dialog box and click on the OK button.**

This assigns the variable “Lstub” to the dimension parameter. When you click on the OK button an arrow indicating the length and the name appear on your display.

**8 Move the mouse until the name is positioned above the stub. When the name is in the desired position, click on the mouse.**

The dimension parameter should now appear as shown below.

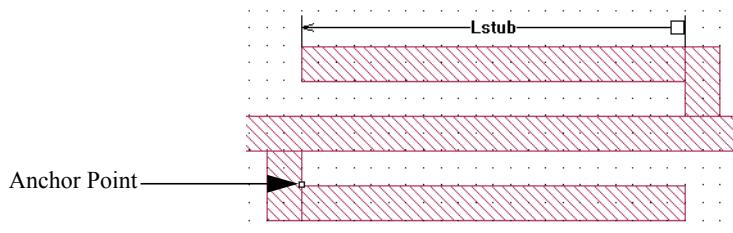


You will now enter the linked dimension parameter. A linked dimension parameter is another dimension parameter identified in the circuit that has the same nominal value and assigned variable.

- 9 Select *Tools*  $\Rightarrow$  *Add Dimension Parameter*  $\Rightarrow$  *Add Anchored* from the project editor's main menu.**

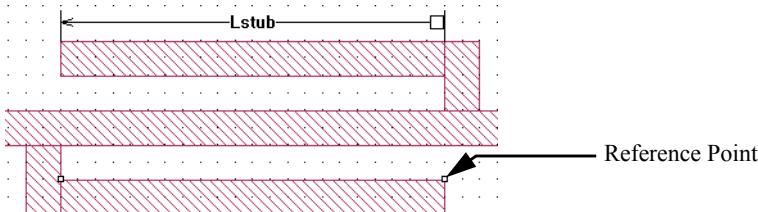
This places the project editor in Add an Anchored Parameter mode indicated by the change in cursor.

- 10 To specify the Anchor point for the parameter, click the mouse on the corner of the lower stub, as shown in the picture below.**



The anchor point is indicated by a small square which appears at the point you clicked. The next step is to select the reference point.

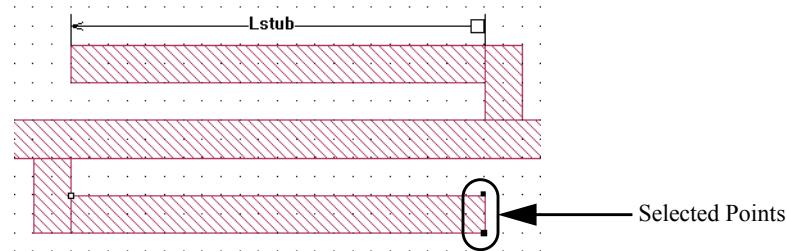
- 11 Click on the top right end of the bottom stub to add the reference point.**



The reference point is indicated by a small square which appears at the point you clicked.

In the next step, you select the rest of the adjustable point set. Points may be selected using the any of the edit commands available in the project editor.

**12 Drag the mouse until both points on the end of the stub are selected.**

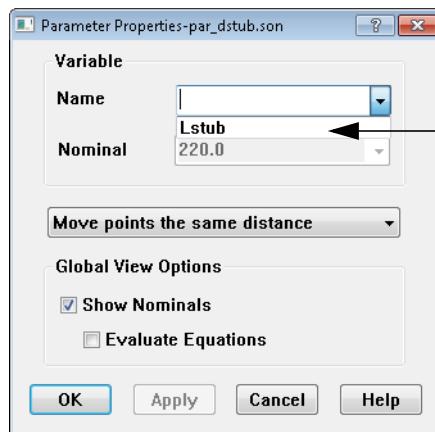


These points will be added to the adjustable point set. When the reference point moves, these points move relative to the reference point.

**13 Once all the desired points are selected, press Enter.**

This completes the dimension parameter creation. The Parameter Properties dialog box appears on your display.

**14 Select the variable name “Lstub” from the drop list in the Variable Name text entry box in the Properties dialog box and click on the OK button.**



Since you have already created the variable “Lstub” and this dimension parameter is the same length as the dimension parameter to which “Lstub” is already assigned, the variable name is available in the drop list.

This assigns the variable “Lstub” to the dimension parameter. When you click on the OK button an arrow indicating the length and the variable name appear on your display. Use your mouse to move the label to the desired location, then click.

Since this dimension parameter’s nominal value is the same as the first parameter you defined, the project editor allows you to assign the same variable to both dimension parameters. These dimension parameters are now linked. A change in value of the variable changes both dimension parameters. If the dimension parameters had been of a different length, the variable name “Lstub” would not have appeared in the list. Entering the variable name manually would invoke an error message.

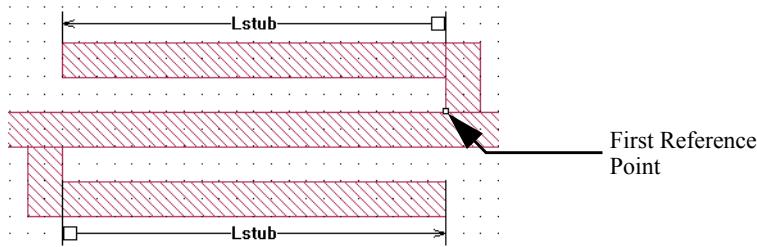
Next, you define the last dimension parameter which is symmetric.

### Symmetric Parameters

- 15 Select Tools  $\Rightarrow$  Add Dimension Parameter  $\Rightarrow$  Add Symmetric from the project editor's main menu.**

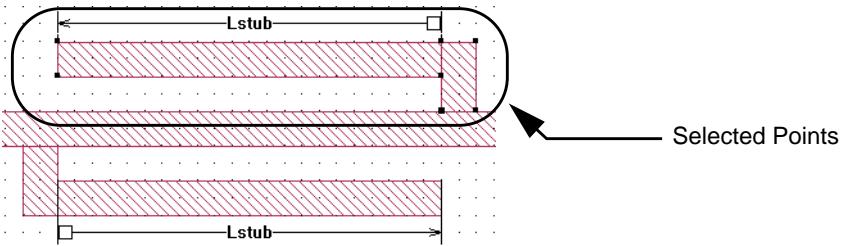
This places the project editor in Add a Symmetric Parameter mode indicated by the change in cursor. Note that the message “Click mouse to specify first reference point” appears in the status bar at the bottom of the project editor window. As you add a dimension parameter, directions for each step appear in the status bar and as tool tips on your screen.

- 16 To specify the first reference point for the dimension parameter, click the mouse on the intersection of the inside of the top stub to the transmission line, as shown in the picture below.**



The first reference point is indicated by a small square which appears at the point you clicked. The next step is to select the point set you want attached to the first reference point.

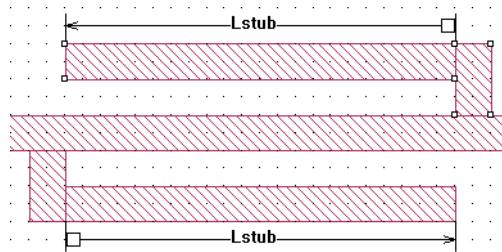
- 17 Drag the mouse until all points on the upper stub are selected.**



These points will be added to the first adjustable point set. When the first reference point moves, these points move in the same direction and distance as the reference point.

**18 Once all the desired points are selected, press Enter.**

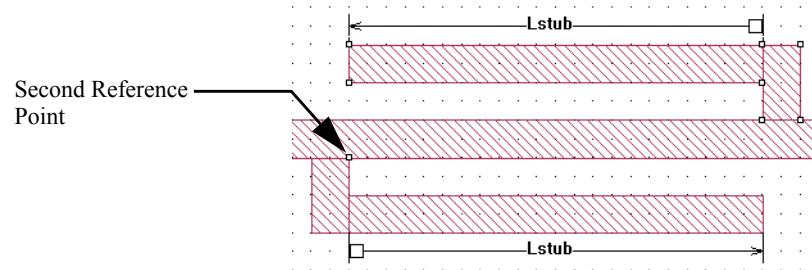
This completes the first point set. Your circuit should look similar to this:



The first point set is indicated by small open squares on all the points in the set.

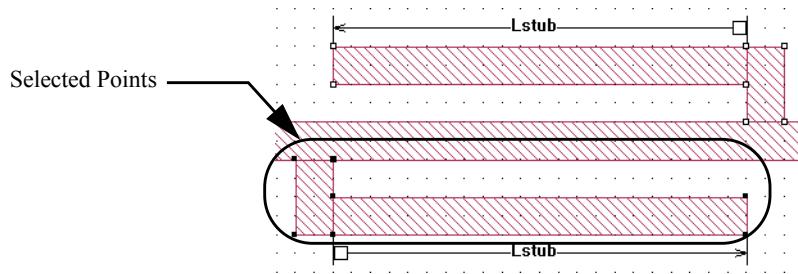
The message “Click Mouse to Specify Second Reference Point” appears on your display. Next, you will specify the second reference point and its point set.

**19 To specify the second reference point for the parameter, click the mouse on the intersection of the inside of the bottom stub to the transmission line, as shown in the picture below.**



The second reference point is indicated by a small square which appears at the point you clicked. Note that the first point set continues to be identified by small squares on all its points. The next step is to select the point set you want attached to the second reference point.

**20 Drag the mouse until all points on the lower stub are selected.**



These points will be added to the second adjustable point set. When the second reference point moves, these points move in the same direction and distance as the reference point.

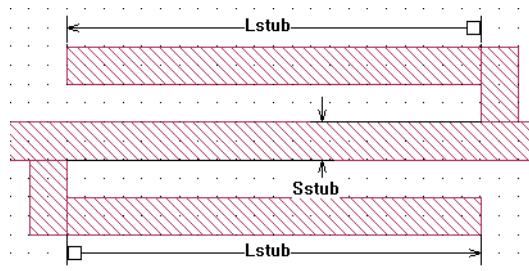
**21 Once all the desired points are selected, press Enter.**

This completes the second point set and the symmetrical parameter. The Parameter Properties dialog box appears on your display.

**22 Enter the name “Sstub” in the Variable Name text entry box in the Parameter Properties dialog box and click on the OK button.**

This assigns the variable “Sstub” to the dimension parameter. When you click on the OK button an arrow indicating the length and the variable name appear on your display. Note that since there is a difference between the reference points in both the x (horizontal) and y (vertical) direction, you may move the parameter name so that the parameter is defined in either the x or y direction.

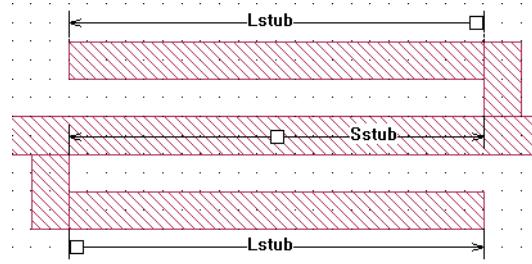
If you were to choose the y direction, moving the mouse to the left or right of both reference points to define your parameter, it would appear like this:



However, for this example, you define the parameter in the x direction, moving your mouse up or down, above or below both reference points.

- 23 Move the mouse until the name is positioned in the middle of the thru line. When the name is in the desired position, click on the mouse.**

This sets the dimension parameter in the x direction. The dimension parameter should now appear as shown below.



This completes entering the dimension parameters. Note that Lstub is affected by Sstub. As Sstub increases, although it does not directly affect the value of Lstub, the two stubs do get further apart. The dimension parameter to which Lstub is assigned is dependent on the dimension parameter to which Sstub is assigned. However, the variable Lstub is NOT dependent on the variable Sstub so that both are available to be selected for the parameter sweep and optimization covered later in this tutorial. Only independent variables may be selected for parameter sweeps and optimization.

- 24 Select *File*  $\Rightarrow$  *Save* for the project editor main menu.**

This saves the changes you have made to the circuit so that you can analyze it.

The next section of the tutorial teaches you how to setup and run a parameter sweep on the circuit.

## Parameter Sweep

The parameter sweep uses only the Lstub variable. You analyze the circuit at two different lengths for Lstub over a frequency band of 2.0 GHz to 10.0 GHz. When the sweep is complete, you view the response curves in the response viewer.

For a detailed discussion of a parameter sweep, please refer to “Parameter Sweep,” page 189.

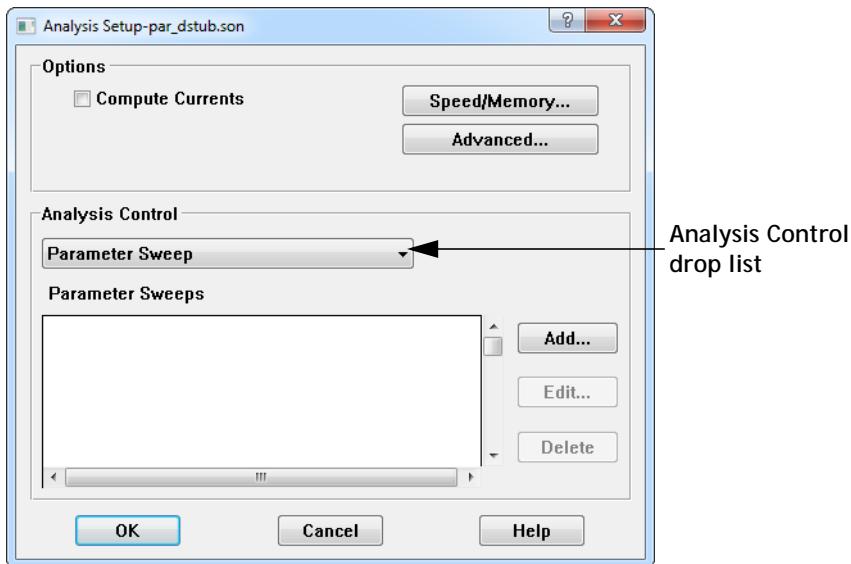
## Setting Up a Parameter Sweep

- 1 Select *Analysis*  $\Rightarrow$  *Setup* from the project editor's main menu.

The Analysis Setup dialog box appears on your display.

- 2 Select **Parameter Sweep** from the **Analysis Control** drop list.

This selects a parameter sweep as the type of analysis. The dialog box's appearance changes to accommodate the input needed for a parameter sweep.



- 3 Click on the Add Button in the Analysis Control section of the dialog box.

The Parameter Sweep Entry dialog box appears. This dialog box allows you to add a parameter sweep.

The first step is to specify the analysis frequencies you wish to use for the parameter sweep. For this example, you wish to perform an ABS sweep from 2.0 GHZ to 10.0 GHz. Since Adaptive Sweep (ABS) is the default sweep type, you do not need to take any action to select it.

- 4 Enter 2.0 in the Start text entry box in the Frequency Specification section of the Parameter Sweep Entry dialog box.

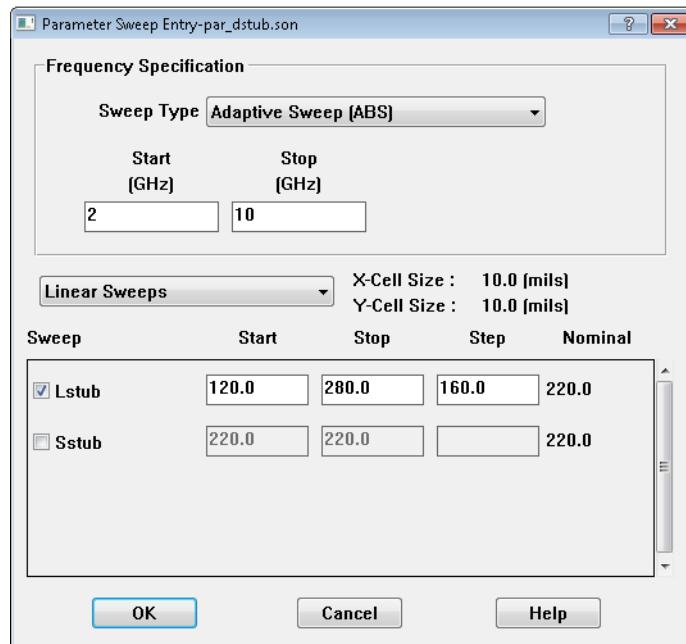
This is the starting frequency.

- 5 Enter 10.0 in the Stop text entry box.

This is the highest frequency. This defines a band of 2 GHz to 10 GHz for the adaptive sweep.

Next, you will select the variables you wish to sweep.

- 6 Click on the checkbox in the Sweep column of the entry for the Lstub variable.



It is possible to select multiple variables for a parameter sweep; however, for this example, only one variable is used. If you wished to deselect the variable, you would simply click on the checkbox again. Unchecked variables are simulated at their nominal value, so Sstub is a constant, fixed at 220 mils for the parameter sweep.

The nominal value, that is the present value of the variable in the circuit, appears in the Nominal column of the variable entry. In this case, the nominal value of Lstub is 220 mils, so the project editor shows the length as 220 mils, even though the start value for the variable is different.

- 7 Enter 120 in the Start text entry box in the Lstub row.

This sets 120 mils as the first variable value used for Lstub.

- 8 Enter 280 in the Stop text entry box in the Lstub row.

This sets 280 mils as the last variable value used for Lstub.

- 9 Enter 160 in the Step text entry box in the Lstub row.

The interval between variables is 160 mils; therefore, this parameter sweep analyzes at two variable values, 120 and 280 mils.

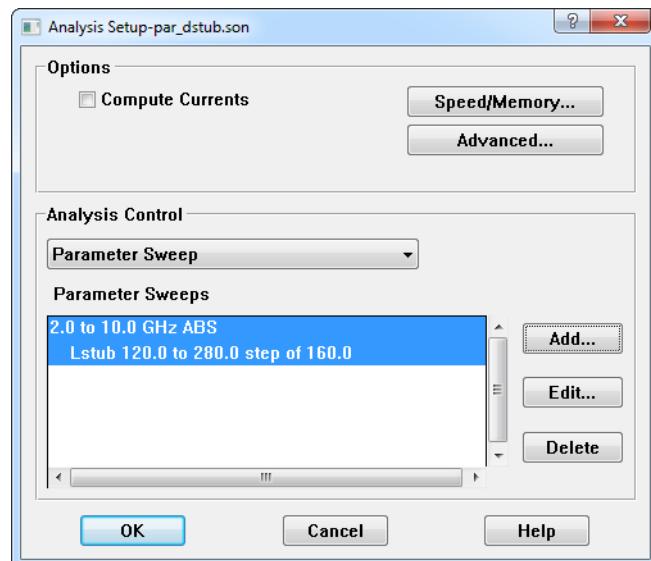
There is a drop list for the Sweep mode just above the entries for the variables. The default mode of Linear Sweep is used for this example so there is no need to take any action. For details on sweep modes, please refer to Help by clicking on the Help button in this dialog box.

This completes the setup for the parameter sweep entry.

**10 Click on the OK button to close the dialog box.**

When the dialog box is closed, the Analysis Setup dialog box is updated with an entry for the parameter sweep that you just defined. In this case, since there are two values for a single variable, there are two parameter combinations. Each combination is analyzed at each analysis frequency.

For example, you have two variables, the first has seven values and the second has eleven values. In that case, there would be 77 variable combinations for the analysis.



**11 Click on the OK button of the Analysis Setup dialog box.**

This completes the entry of the parameter sweep.

Next, you run the analysis and use the analysis monitor to observe the progress.

## Executing the Parameter Sweep

- 12 Select *Project*  $\Rightarrow$  *Analyze* from the project editor's main menu to invoke the analysis engine, *em*, and start the analysis.**

If you are prompted, save the file. The output window of the analysis monitor appears on your display.

- 13 Click on the Response Data button in the analysis monitor output window.**

This allows you to observe the analysis as it progresses. There is a progress bar at the top of the window which shows what percentage of the total analysis is complete with the number of frequencies analyzed appearing above it. The response data is output in the bottom of the window.

The analysis could take a few minutes to run depending on your computer.

Once the analysis is complete, you open the response viewer to look at your results.

## Observing the Parameter Sweep Data

You want to see the data for the  $S_{21}$  response at  $L_{stub} = 120$  mils and  $L_{stub} = 280$  mils.

- 14 Select *Project*  $\Rightarrow$  *View Response*  $\Rightarrow$  *New Graph* from the main menu of the analysis monitor output window.**

The response viewer window appears on your display with  $S_{11}$  displayed.

- 15 Right click on *par\_dstub* in the Curve Group legend.**

A pop-up menu appears on your display.

- 16 Select *Edit Curve Group* from the pop-up menu.**

The Edit Curve Group dialog box appears on your display.

- 17 Double click on *DB[S11]* in the Selected list.**

This moves the  $S_{11}$  response to the Unselected list. It will no longer appear in your plot.

- 18 Double click on *DB[S21]* in the Unselected list.**

This moves the  $S_{21}$  response to the Selected list so that it appears in your plot.

**19 Click on the Select Combinations button in the Edit Curve Group dialog box.**

The Select Parameters dialog box appears on your display. Note that by default only variables which are varied during the parameter sweep are displayed, so only Lstub appears. If you wish to also display variables which were not varied, in this case Sstub, click on the Configure Columns button and select the “Static” checkbox in the Configure Columns dialog box.

**20 Click on the Select All button above the Selected Parameter Combinations list.**

There should be only one entry in this list; the Select All button was used to demonstrate its function. This selects all the parameters in the Selected List.

**21 Click on the Up Arrow button to move all the parameter combinations to the Unselected list.**

**22 Double-click on Lstub = 120 mils in the Unselected List.**

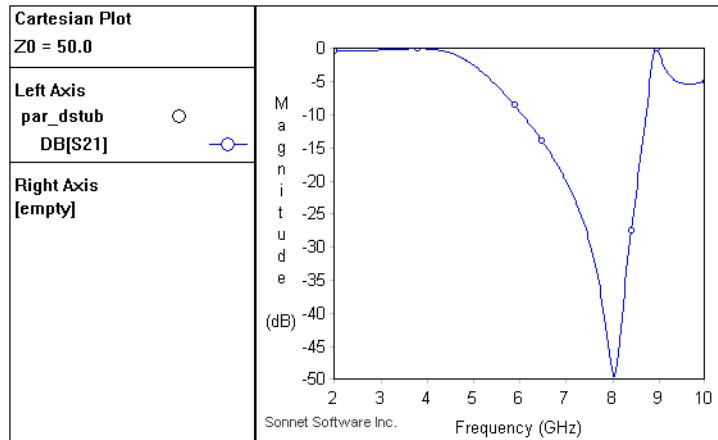
This moves this parameter combination to the selected list so that the data for this combination appears in your plot.

**23 Click on the OK button in the Select Parameters dialog box to apply the changes and close the dialog box.**

The entry “Lstub = 120.0 Sstub = 220.0” appears in the Parameter Combinations section of the Edit Curve Group dialog box.

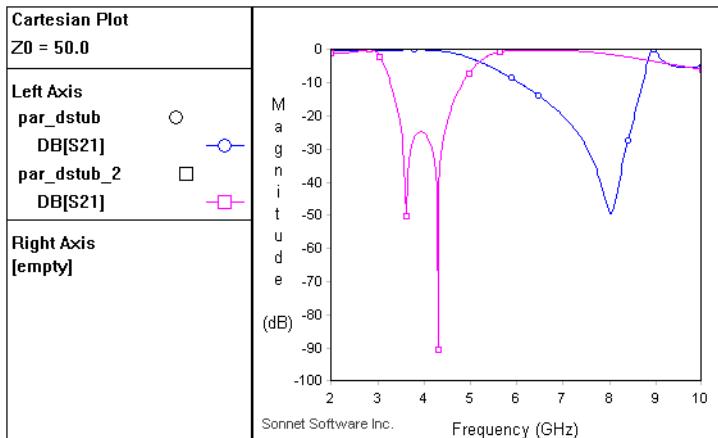
**24 Click on the OK button in the Edit Curve Group dialog box to apply the changes and close the dialog box.**

The plot is updated with the S21 response for Sstub = 220 mils, Lstub = 120 mils. The entry for the curve group, par\_dstub appears in the Curve Group Legend. The graph should be similar to the one shown below.



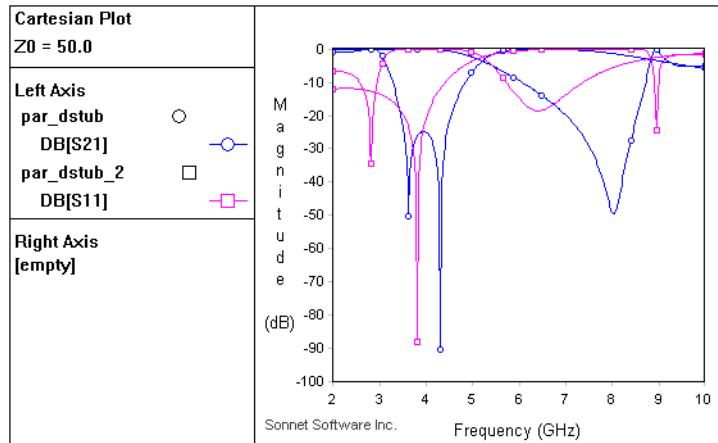
**25 To add the response at Lstub = 280 mils, select *Curve* ⇒ *Add Curve Group* from the response viewer main menu.**

The Add Curve Group dialog box appears. This curve group uses the default name of par\_dstub\_2. Following the same steps you used for par\_dstub above, set up this curve group to display the S<sub>21</sub> response for Lstub = 280 mils, Sstub = 220 mils. Your plot should now look like the one below.



You could also have right-clicked the curve group par\_dstub in the Left Axis pane of the legend and selected “Edit Curve Group” from the pop-up menu. Using the Edit Curve group dialog box, you could have added this parameter combination to

this curve group. This would result in one curve group with one symbol representing both parameter combinations. This is useful if you want multiple measurements ( $S_{21}$  and  $S_{11}$  for example). Each measurement would use a different symbol, but each parameter combination with a measurement would use the same symbol. An example is pictured below.



In the beginning, the goal of the filter design was stated as a stopband between 5.0 and 6.0 GHz. By looking at the graph of  $L_{stub}=120$  as compared to  $L_{stub}=280$ , you can see that a filter with the required stopband would fall approximately in the middle of the two curves. So a value of 220 mils is chosen for the nominal value for  $L_{stub}$  for the optimization. A nominal value of 200 mils is chosen for  $S_{stub}$ .

## Optimization

This next section of the tutorial shows how to set up and execute an optimization.

For a detailed discussion of optimization, please refer to "Optimization" on page 192.

If `par_dstub.son` is not still open in the project editor, open the file in the project editor.

## Entering New Nominal Values

Usually for this type of circuit, you would optimize using both of the defined parameters, Lstub and Sstub, but for the sake of processing time, the optimization only uses one parameter, Lstub.

The nominal value used for Sstub will be 200 mils. This was arrived at by actually executing the optimization using both parameters and using the closest value on the grid of the optimized parameter.

- 26 Double-click on the parameter Sstub in the circuit.**

The Parameter Properties dialog box for Sstub appears on your display.

- 27 Change the nominal value in the Nominal text entry box to 200.**

- 28 Click on the OK button to close the dialog box and apply the new nominal value.**

The circuit is redrawn using the new nominal value for Sstub.



### **WARNING**

**It is recommended that you change the nominal value of dimension parameters by using the properties dialog box, since changing your nominal value this way does not affect any previous response data in your project file. If you change the nominal value of the dimension parameter by changing the circuit through editing commands, such as a reshape, all previous response data is deleted from the project file when you save.**

---

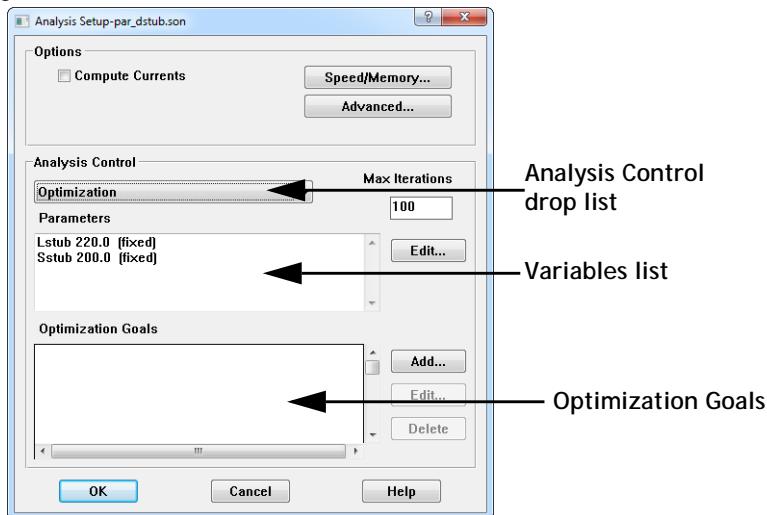
## Setting Up an Optimization

- 29 Select *Analysis* ⇒ *Setup* from the main menu of the project editor.**

The Analysis Setup dialog box appears on your display.

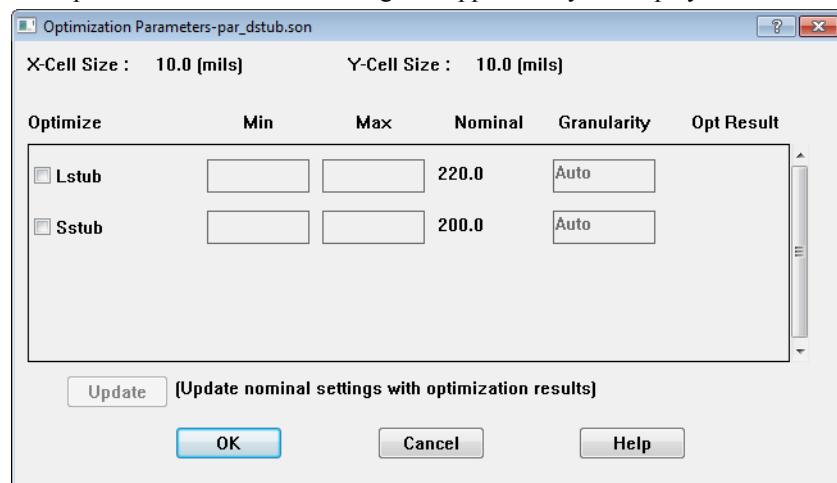
**30 Select Optimization from the Analysis Control drop list.**

This selects a optimization as the type of analysis. The dialog box's appearance changes to accommodate the input needed for an optimization. Note that initially the nominal values are listed for the variables, since no range has yet been specified.



**31 Click on the Edit button on the right side of the Variables list.**

The Optimization Parameters dialog box appears on your display.



Only one variable, Lstub, is used for this optimization. The range for Lstub is 100 mils to 300 mils.

**32 Click on the Optimize checkbox next to Lstub.**

This selects the Lstub variable to be used in the optimization. Note that the nominal value appears in the Nominal text entry box. If you wish to change the nominal value, you may do so by entering a new value. The circuit will be redrawn using the new nominal value. Since this is the first optimization run on this project file the Min and Max entries are blank. If a previous optimization had been run, the last entered values would remain.



**WARNING**

---

**Editing parameter settings or optimization goals causes any pre-existing optimization iterations to be deleted from the project file.**

---

**33 Enter 100 in the Min text entry box in the Lstub row.**

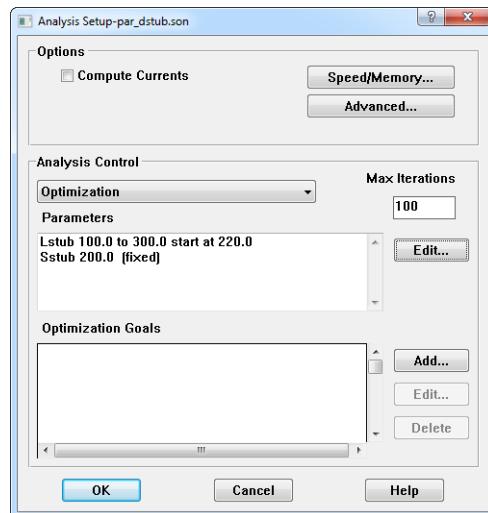
This sets the minimum value for the Lstub variable to 100 mils for the optimization.

**34 Enter 300 in the Max text entry box in the Lstub row.**

This sets the maximum value for Lstub to 300 mils for the optimization.

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

When the dialog box is closed, the Analysis Setup dialog box is updated with an entry for the optimization variables that you just defined.

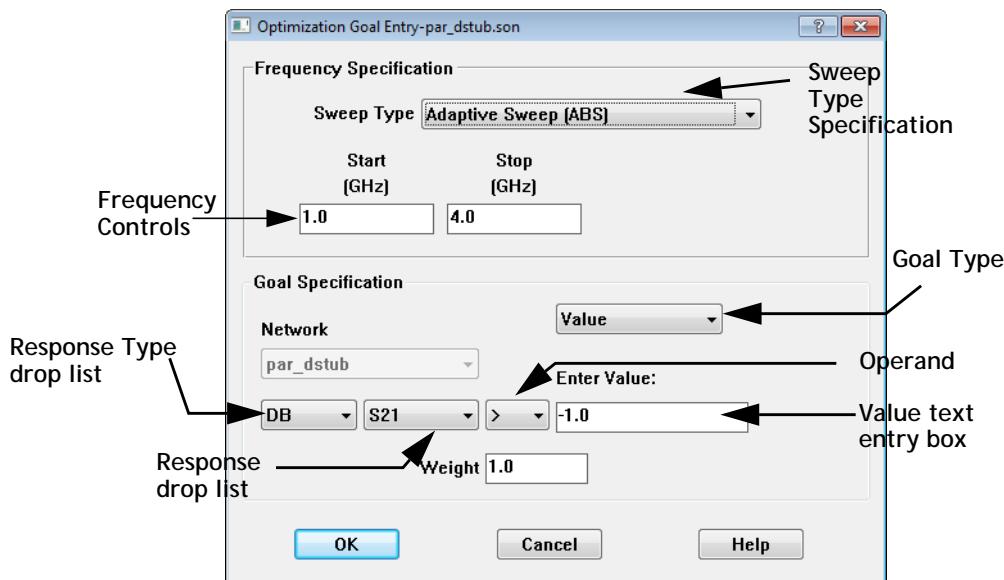


Now that you have identified which variable to vary and its range, you must specify the optimization goals. Since this is the first optimization for this project, there are no previously defined optimization goals and the list is empty. Having no goals present disables the Edit and Delete buttons. The Edit button allows you to modify an existing goal, and the delete button removes the goal from the list.

As mentioned earlier in the example, our goal for the filter is to have passbands at 1-4 GHz and 7-10 GHz with a stopband at 5-6 GHz. The optimization goals are set up accordingly. Since all three goals in this case are equally important, each uses the default Weight of 1.0. In cases where one goal is more essential, assigning it a higher weight than other goals tells *em* to concentrate more on reaching that particular goal.

**36 Click on the Add button to the right of the Optimization Goals list.**

The Optimization Goal Entry dialog box appears on your display.



The first goal corresponds to the first passband; therefore, you need to set the ABS frequency range to 1.0 GHz to 4.0 GHz. There are several different types of frequency sweeps available; you will use the default ABS sweep for this optimization.

**37 Enter 1.0 in the Start text entry box in the Frequency Specification section of the dialog box.**

This sets the beginning of the frequency band for this optimization goal at 1.0 GHz.

**38 Enter 4.0 in the Stop text entry box in the Frequency Specification section of the dialog box.**

This sets the end of the frequency band for this optimization goal at 4.0 GHz. This completes the specification of the frequency sweep for this optimization goal.

Since this is the first passband, your goal is to have DB[S21] be greater than -1.0 dB.

**39 Select DB from the Response Type drop list on the left side of the equation.**

This is the default, so DB may already be selected.

**40 Select S21 from the Response drop list.**

The display is updated with “S21.”

**41 Select “>” from the Operand drop list.**

The display is updated with “>.”

**42 Select Value from the Goal Type drop list.**

This choice allows you to put in a specific value. This is the default; you may also specify another project file or another network in your project (if the project is a Netlist project). In those cases, you may select a response for that circuit to which you wish to match your selected response.

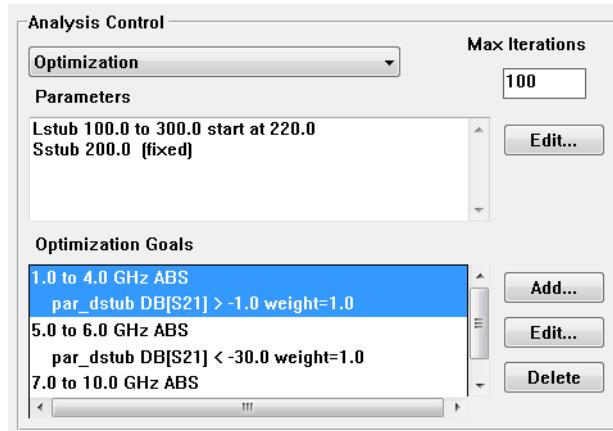
**43 Enter -1.0 in the Value text entry box.**

This sets your goal of DB[S21] > -1.0 dB.

**44 Click on OK to apply the changes and close the dialog box.**

The Analysis Setup dialog box is updated. An entry for this optimization goal now appears in the Optimization Goals list.

The other two goals should be entered in a similar manner. The second goal is an adaptive sweep from 5.0 GHz to 6.0 GHz with a desired response of DB[S21] < -30.0 dB. This is the stopband. The third goal is an adaptive sweep from 7.0 GHz to 10.0 GHz with a desired response of DB[S21] > -1.0. When you have completed entering these goals, the Optimization Goals list should appear as shown below.



This completes the setup for the optimization.

## Running an Optimization

This optimization took approximately 6.5 minutes on a 2 GHz Pentium 4.

- 45 Select *Project*  $\Rightarrow$  *Analyze* from the main menu of the project editor.**

The output window of the analysis monitor appears on your display.



### TIP

---

You can also click on the Analyze button on the project editor tool bar.

---

- 46 Click on the Response Data button in the analysis monitor output window if it is not already displaying the response data.**

This allows you to observe the optimization as it progresses. The error for the first, present and best iteration are displayed and updated as the optimization progresses. The response data is output in the bottom of the window.

You may notice that some iterations complete more quickly than others. This is because **em** can reuse portions of the response data calculated for previous iterations.

Once the analysis is complete, you open the response viewer to look at your results. Be aware that since this optimization took a number of iterations to conclude, there may be small delays in opening the response viewer and the Edit Curve Group dialog box.



### TIP

---

You may use the response viewer to observe data produced by the optimization while it is still running. The response viewer can chart any data which has been generated. This allows you to stop the optimization and start it over using new variable ranges or new goals, if the results are not desirable. If you wish to continue viewing data as the optimization runs, use the Freshen Files command in the Response Viewer.

---

## Observing your Optimization Data

Plotting the best iteration will allow you to judge whether or not to use the optimized values of the parameters.

- 47 **Select *Project*  $\Rightarrow$  *View Response*  $\Rightarrow$  *New Graph* from the main menu of the analysis monitor.**

The response viewer menu appears on your display with the DB[S11] response for the nominal values parameter combination displayed. There may be some delay while the project loads into the response viewer due to the amount of response data now included.

The plot will display the last plot for par\_dstu.son that you had open the last time the response viewer was invoked for this project.

- 48 **Click on a curve group in the legend to select it and select *Curve*  $\Rightarrow$  *Edit Curve Group* from the main menu of the response viewer.**

The Edit Curve Group dialog box appears on your display.

- 49 **Select Optimized from the Data Collection drop list in the Edit Curve Group dialog box.**

Notice that this drop list is now enabled since the project file now contains both parameter sweep data and optimized data.

**50 Move DB[S21] to the Selected list and all other responses to the Unselected list.**

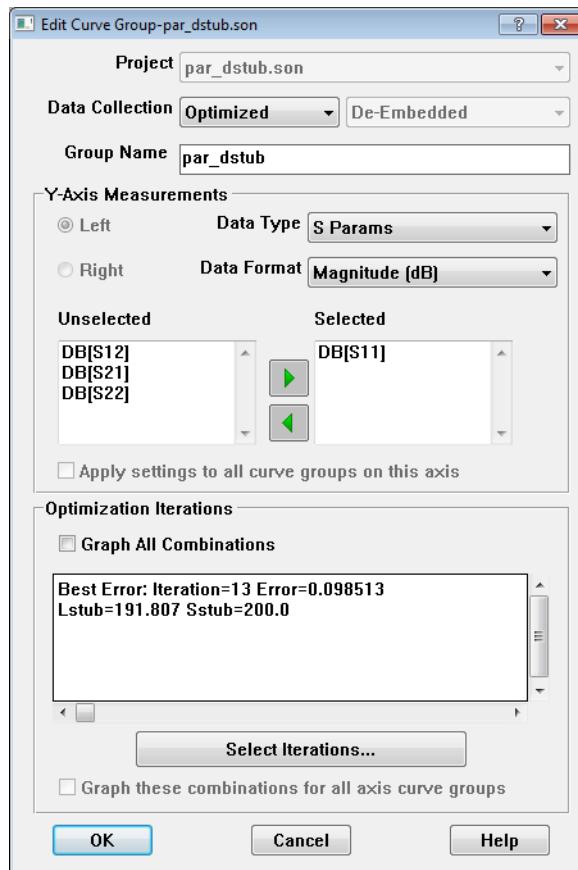
Since your optimization goals were set in reference to the DB[S21] response, you want to plot this response.

Notice that when you selected Optimized data that the parameter combination was updated to read:

```
Best Error: Iteration=14 Error=0.086658
Lstub=191.6134 Sstub=200.0
```

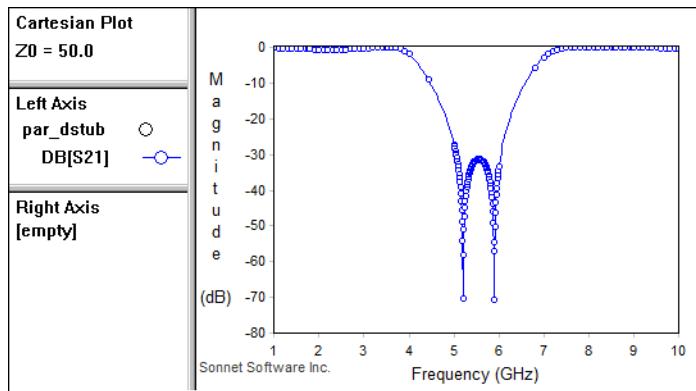
The best iteration is plotted by default when you select optimized data. If the optimization was still running, this provides a useful way of always plotting the best iteration calculated thus far. Pressing the Freshen Files button on the tool bar of the response viewer will always show the best iteration.

If you were to click on the Select Iterations button, the dialog box would appear with all the variable values used in all 25 iterations available to plot. Since you wish to see the response at the best iteration, you do not need to change the parameter value for this curve group.



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

The plot is updated showing DB[S21] for the best iteration. It should appear similar to the plot pictured below.



As you can see, the optimized circuit is producing the desired response of a stop-band from 5 - 6 GHz and the passbands from 1 - 4 GHZ and 7-10 GHz.

## Accepting the Optimized Values

Since the desired responses have been achieved by the optimization, you return to the project editor to update the nominal value of your parameters with the optimized values.

If the project par\_dstub is still open in the project editor, continue at step 54.

- 52 Click on the curve group, par\_dstub in the Curve Group legend in the response viewer.

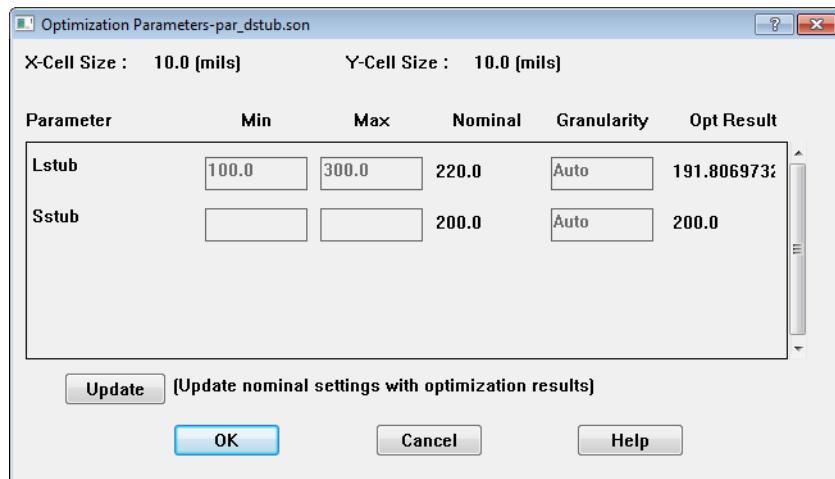
This selects a project file and thereby enables the project menu.

- 53 Select **Project ⇒ Edit** from the main menu of the response viewer.

The project editor appears on your display with par\_dstub open.

**54 Select Analysis  $\Rightarrow$  Optimization Results from project editor main menu.**

The Optimization Parameters dialog box appears on your display.



Notice that the nominal value of both variables is still the value input at the beginning of the optimization. But the optimization results for both variables are displayed in the last column. Since Sstub was not used in the optimization, its optimization result is the same as the nominal value.

**55 Click on the Update button to replace the present nominal values with the optimization results.**

Note that the entries in the Nominal text entry boxes are updated with the optimized values.

**56 Click on OK to close the dialog box and apply the changes.**

Notice that the circuit has been redrawn with the new nominal values for the dimension parameters. Since the parameter lengths are not integer multiples of the cell size, the polygons are no longer exactly on the grid. You can see this by pressing Ctrl-M to turn off the cell fill and looking at the actual polygons. The cell fill represents the actual metal **em** analyzes.

The actual metalization analyzed by **em** is not the same as the optimized values. **Em** actually interpolated from data created from analyzing “on grid” versions of the circuit. If your optimization goals did not include a full frequency sweep, it is a good idea to perform a full sweep across your frequency band to ensure that your entire band shows reasonable results. Running a full frequency sweep is detailed below.

In order for ***em*** to use previously calculated response data, you should edit your variable value(s) such that they are the closest “on grid” value to the optimization result. For example, in this case the optimized value for Lstub is 191.6134. You should edit the nominal value of Lstub to change it to 190 which, since the cell size is 10, is on the grid.

- 57 Select *Analysis*  $\Rightarrow$  *Setup* from the project editor main menu.**

The Analysis Setup dialog box appears on your display.

- 58 Select Adaptive Sweep (ABS) from the Analysis Control drop list.**

The appearance of the dialog box is changed to conform with an adaptive sweep. Enter 1.0 GHz in the Start text entry box and 10 GHz in the Stop text entry box to define a frequency band of 1 - 10 GHz for this analysis.

- 59 Click on OK to close the dialog box and apply the changes.**

- 60 Click on the Analyze button to start the *em* analysis.**

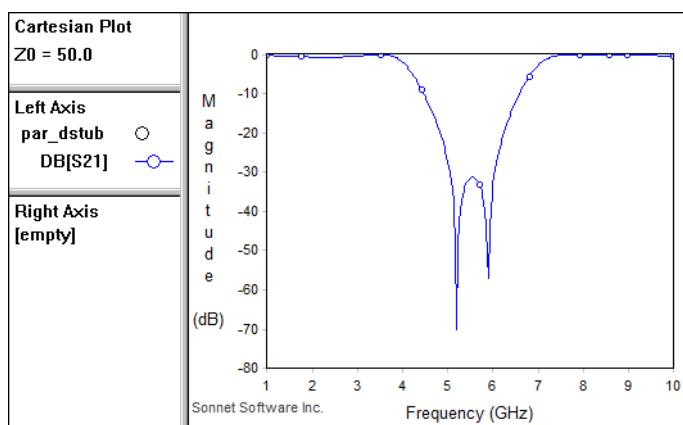
If prompted, save the circuit before analyzing.

- 61 When the analysis is complete, click on the View Response button on the analysis monitor’s tool bar.**

The response viewer appears on your display with the curve group, par\_dstub, from the last plot.

- 62 Right-click on the curve group, par\_dstub, and select Edit Curve Group from the pop-up menu which appears.**

Select DB[S21] for display, the Nominal parameter combination, and close the dialog box. Your plot should appear similar to that shown below.



The results conform closely enough to the design criteria that this optimization is considered a success. As was stressed at the beginning of this tutorial, a simple example was chosen in order to clearly demonstrate the optimization process.

You should be aware, however, that most optimization problems are much more complicated and less straightforward. The designer needs to make decisions about parameters, the parameter sweeps and optimization goals based on knowledge of the circuit and design experience. Often, you must observe an optimization while in progress, judge its viability and, as necessary, stop the optimization and start a new one with new nominal values and data ranges.

# Chapter 11

# Conformal Mesh

---

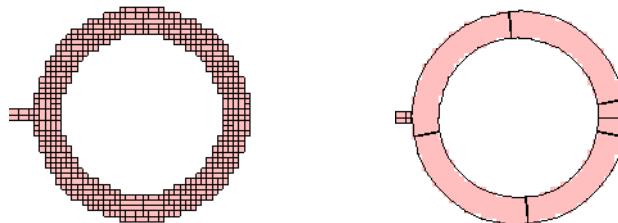
## Introduction

Analyzing circuits which have non-rectangular polygons can require extensive memory and processing time since the number of subsections needed to model the non-rectangular shapes is significantly higher than the number of subsections required for a rectangular polygon. Conformal meshing is a technique which can dramatically reduce the memory and time required for analysis of a circuit with diagonal or curved polygon edges.

This technique groups together strings of cells following diagonal and curved metal contours. Whereas staircase fill results in numerous small X- and Y-directed subsections, conformal mesh results in a few long conformal subsections. Fewer subsections yields faster processing times with lower memory requirements for your analysis.

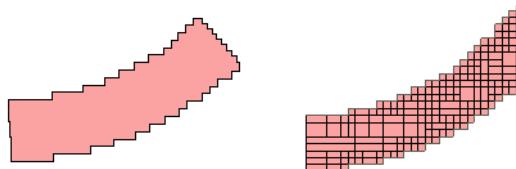
In older meshing techniques, large non-rectangular subsections did not include the high concentration of current on the edge of the lines required by Maxwell's equations. The results could significantly under-estimate loss and inductance. In contrast, the Sonnet conformal meshing automatically includes the high edge current

in each conformal section. In conformal meshing, Sonnet can achieve the speed of using large subsections, and at the same time enjoy the accuracy of using small cells. This patented<sup>1</sup> Sonnet capability is unique.



Pictured above is a typical circuit which would be appropriate for Conformal Meshing. The left picture shows the rectangular subsections created by using staircase fill. This results in approximately 800 subsections (unknowns). The right picture shows the conformal sections created by using conformal fill, resulting in only about 130 subsections. Note that each conformal section shown represents multiple subsections.

Conformal sections, like standard subsections, are comprised of cells, so that the actual metalization still shows a “jagged” edge when the polygon has a smooth edge, as pictured below. However, the sections can be much larger due to conformal meshing. You may now make the underlying grid sufficiently small to accurately resolve challenging circuit dimensions without incurring excessive memory and analysis time requirements.



Conformal meshing should be used in places where it will reduce subsection count. For rectangular polygons with no diagonal or curved edges, it is more efficient to use rectangular subsections (default). However, if a polygon contains a curved edge, conformal meshing provides a quicker analysis.

For a discussion on subsectioning when using Conformal Mesh, see “Conformal Mesh Subsectioning,” page 43.

<sup>1</sup> U.S. Patent No. 6,163,762 issued December 19, 2000.

## Use Conformal Meshing for Feed Lines, Not Patches

Conformal meshing assumes most of the current is flowing parallel to the edge of the conformal subsection. This works well for feed lines. However, this is usually not accurate for geometries like patch antennas. For large areas of metal in both x and y directions, high current can flow parallel to the X axis edges, and parallel to the Y-axis edges at the same time. Conformal meshing can include only one of these currents. Thus, conformal meshing should only be used for transmission line geometries, which have a “line width” that is small compared to wavelength.

## Applying Conformal Meshing

Conformal Meshing is applied as a property of a metal polygon or a property of a Metal Technology Layer. The following example shows how to set conformal meshing in a project which does not have any Metal Technology Layers defined. The process is very similar for a Technology Layer; you would edit the meshing properties of the Technology Layer.

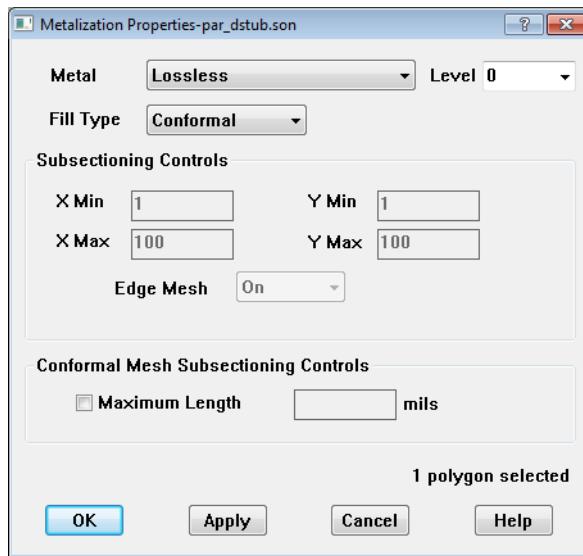
To use conformal meshing for a polygon, do the following:

- 1 **Select the desired polygon(s) by clicking or lassoing.**

The selected polygons are highlighted.

**2 Select *Modify*  $\Rightarrow$  *Metal Properties* from the main menu.**

The Metal Properties dialog box appears on your display.

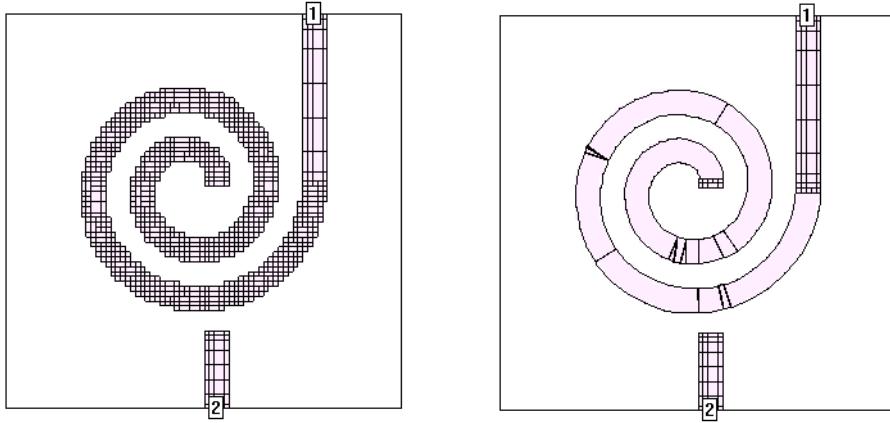


**3 Select "Conformal" in the Fill Type drop list.**

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

The polygon does not appear any different in the circuit. To see the difference, you need to use the *Analysis*  $\Rightarrow$  *Estimate Memory* command. When the Estimate Memory dialog box appears, click on the View Subsections button. Shown below is the subsectioning for the same spiral inductor. The circuit on the left uses rectangular subsections and the one on the right uses conformal subsections. Note

that the rectangular subsectioning uses a much higher number of subsections for the spiral inductor than does the conformal meshing. Rectangular subsectioning was used for the feed lines in both cases.



Spiral inductor with rectangular subsections. (Default)

Spiral inductor with conformal subsections.

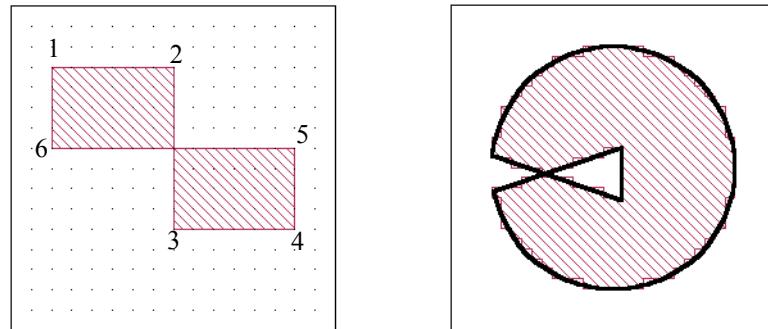
If you chose Conformal meshing, then the subsectioning controls in the Metalization Properties dialog box - Xmin, Ymin, XMax, YMax and Edge Mesh - are disabled and ignored.

## Conformal Meshing Rules

Not all conditions which may affect accuracy or processing time are automatically identified in the project editor. Below are some basic rules for using conformal meshing you may follow to prevent causing an error in the analysis engine, *em*.

## Rule 1: Figure Eight Polygons

A conformal mesh polygon should not wrap back around itself; in other words, its vertices should not form a figure eight. This will result in an error message being issued and **em** will stop the analysis job. Two examples of this type of polygon are shown below. In the polygon shown on the left, the vertices have been labelled in the order in which they were added.



## Rule 2: Adjacent Polygons Should Have No Gap

Any polygon which is adjacent to another polygon using conformal meshing should have its edges exactly touching with no gap existing between the two polygons. Extremely tiny gaps are automatically removed, but should be avoided. Tiny gaps can be easily avoided by using any of the following methods:

- Using a snap grid while drawing your circuit. (*Tools* ⇒ *Snap Setup*)
- Creating a larger polygon, then dividing the polygon (*Edit* ⇒ *Divide Polygons*) and applying conformal meshing to one of the resulting polygons. The resultant polygons are adjacent with no space in between.
- Adding a small polygon which bridges the gap and overlaps the two polygons on either side of the gap.
- Snapping the existing polygons to the grid. (*Modify* ⇒ *Snap To* command using the Preserve shape and spacing option).



### WARNING

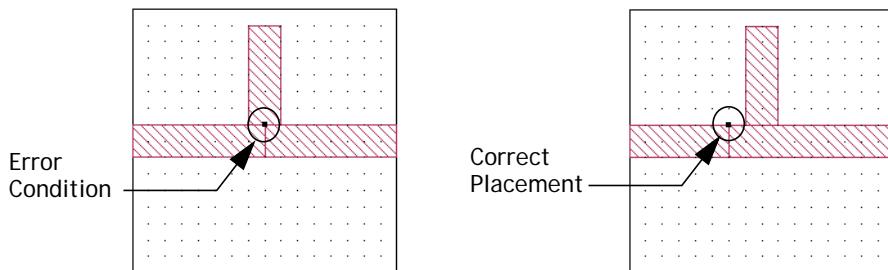
---

If you are snapping a circuit with curved edges, use the Preserve shape and spacing option in the Snap Objects dialog box. Otherwise, curved edges can become distorted and are difficult to restore in the project editor.

---

### Rule 3: Adjacent Polygons Should Not Have an Interior Vertex

When three polygons are adjacent, a vertex where two polygons meet should not occur between two vertices of the third polygon. See the illustration below.



### Single Precision Option

It is recommended that single precision not be enabled when your circuit has polygons using conformal mesh fill. This is because conformal mesh subsections are sensitive to precision error. Since using the single precision option involves reducing the required memory at the expense of increasing precision error, its use may lead to noisy S-parameter results for circuits with conformal mesh fill.

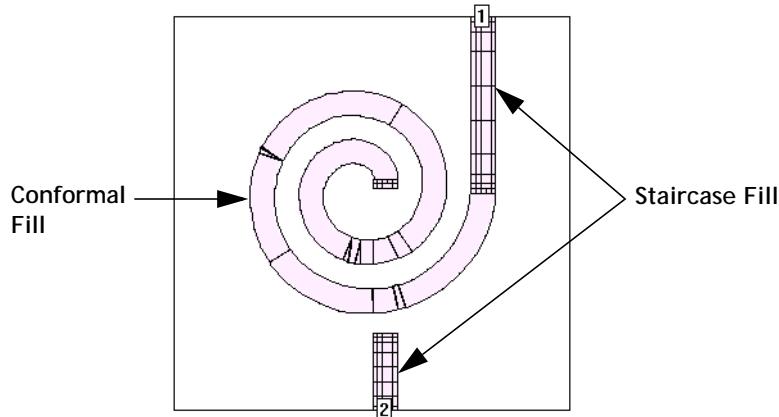
## Using Conformal Meshing Effectively

This section discusses some guidelines to use in order to get the most improvement in processing time and memory use and the most accurate results when using conformal meshing. Following these guidelines will help you to use conformal meshing in the most efficient manner.

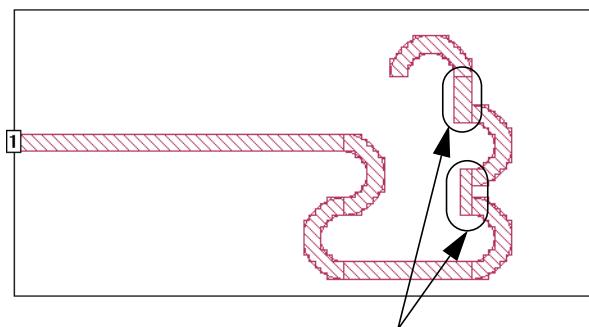
### Use Conformal Meshing for Non-Manhattan Polygons

Conformal meshing should be used for non-Manhattan polygons. Manhattan polygons are polygons which only have vertical and horizontal edges, no diagonals or curves. For these types of polygons, rectangular subsections are more efficient.

You should look at your geometry and, if necessary, divide it up into Manhattan and non-Manhattan polygons using the Edit ⇒ Divide Polygon. Then set the Manhattan polygons to staircase fill and the non-Manhattan polygons to Conformal fill. For example, the spiral conductor shown below contains Manhattan sections in the feed lines and non-Manhattan sections in the circular spiral. It should be divided up such that the feedlines are represented by polygons set to staircase fill, and the circular spiral is another polygon set to Conformal fill.



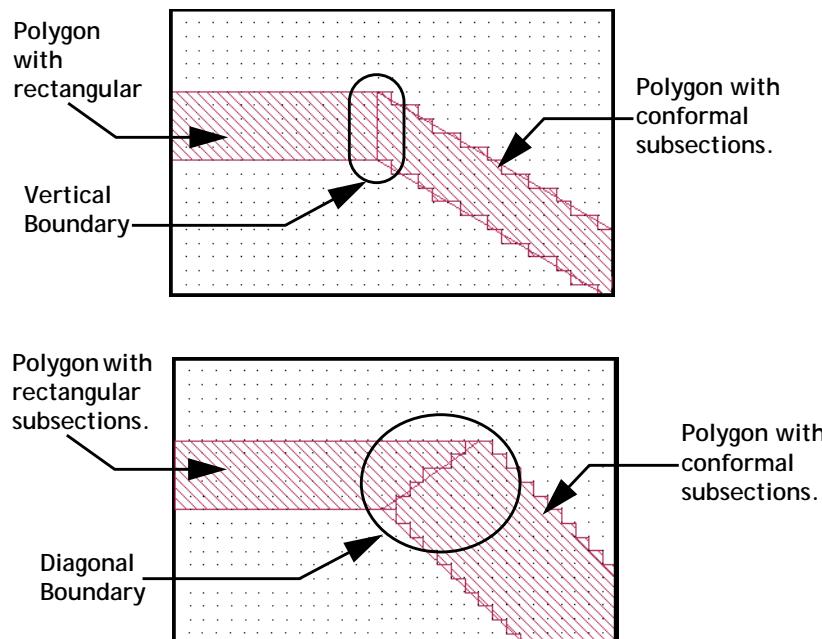
The exception to this rule is when relatively small Manhattan polygons are between conformal mesh polygons. In that case, the inefficiency of switching so frequently between staircase and conformal mesh outweighs the gain of using Manhattan polygons. In that case, conformal mesh should be applied to all the polygons. An example is shown below.



Normally, these polygons would use staircase fill, but because they are relatively small areas and in between polygons on which you would use conformal mesh, it is more efficient to apply conformal meshing to these Manhattan polygons.

## Boundaries Should Be Vertical or Horizontal

For the most efficient results, the boundaries between polygons using conformal meshing and rectangular subsectioning should be vertical or horizontal as shown in the first picture below. Diagonal boundaries, as shown in the second picture, make the analysis less efficient.



## Cell Size and Processing Time

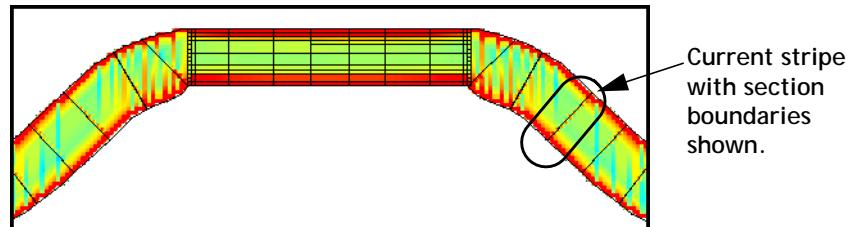
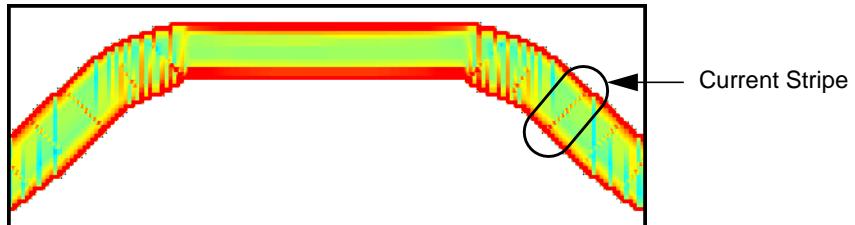
Care should be taken when choosing your cell size when using conformal mesh. Many users, especially experienced Sonnet users, will estimate processing time based on the amount of memory required to analyze a circuit. The amount of memory used for conformal mesh can be deceptive. Using a smaller cell size in a circuit which uses conformal mesh may not increase the required memory but will have a noticeable effect on processing time. The significant factor in determining processing time with conformal meshing is the number of metalized cells needed to construct a conformal section. The number of conformal mesh cells displayed as the result of the Estimate Memory command may be more reliably used as a guideline.

## Current Density Viewing

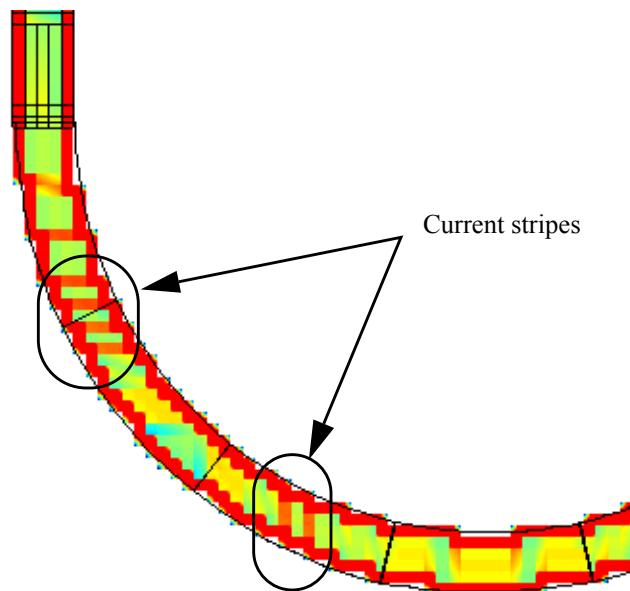
You may view the current of circuits using conformal mesh just like any other circuit. However, the current density of conformal mesh polygons might show unusual “striping”. These stripes do not represent real current, but are a by product of the conformal meshing algorithm.

There are two types of current striping:

- 1 A single stripe of current can appear on the junction between two conformal sections as shown below.



- 2 Horizontal or vertical stripes may appear within a curved conformal section producing a “ripple” effect as shown below.



For a tutorial on using conformal meshing please see the “Conformal Mesh” topic under Tips and App notes in Help. You may access Help by selecting *Help* ⇒ *Sonnet Help* from the menu of any Sonnet application or by clicking on the Help button in any dialog box.



# Chapter 12

# Netlist Project Analysis

---

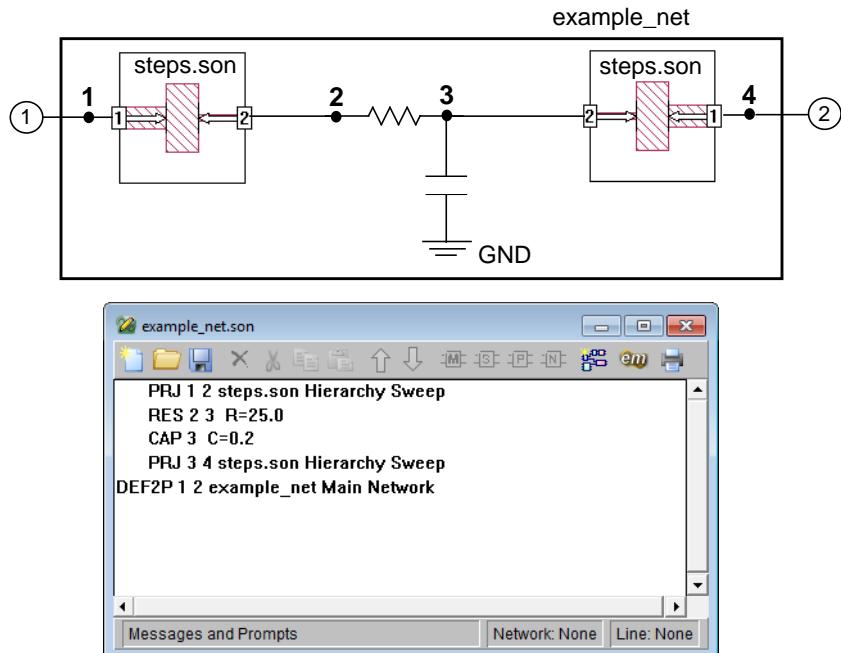
Netlist projects provide you with a powerful circuit analysis tool. Examples of ways in which the netlist may be used include:

- **Cascading Sonnet projects:** You can analyze and combine multiple projects using previously existing data for the subprojects if it is available. This is particularly useful when analyzing large, complex circuits which require circuit subdivision for an em analysis. When analyzing a netlist project, em will automatically interpolate between frequencies if there are differences between the frequency sweeps used in the subprojects. It is also possible to impose the same frequency sweep on all the subprojects in a netlist. For more information about circuit subdivision, see Chapter 13, "Circuit Subdivision".
- **Cascading S-, Y- and Z- parameter data files:** You can read and combine multiple sets of S-, Y- and Z-parameter data files. This is particularly useful if you wish to combine results from another vendor's software for use in an analysis by em. When analyzing a netlist project, em will automatically interpolate between frequencies if there are differences in the frequencies between the data files.

- **Inserting modeled elements into a circuit.** Modeled elements, such as resistors, capacitors, inductors and transmission lines, can be combined with geometry subprojects and S-, Y- and Z-parameter data files.

## Networks

A netlist project contains a netlist which consists of one or more networks with elements connected together. The netlist provides a map in which the ports of individual elements in the netlist are connected to the ports of other elements by the use of nodes. Nodes represent a connection between netlist elements. In other words, you use ports in your circuits to define connections, but you use nodes in a netlist. In order to connect a circuit in a netlist, you must assign its ports to nodes in the netlist. This is done when you define the circuit as an element in a netlist.



The picture above shows the network represented by the netlist shown in the project editor below it. The nodes are represented by the numbered black dots. The geometry project, steps.son, is connected between nodes 1 and 2 with node 1 corresponding to Port 1 in the geometry project and node 2 corresponding to Port 2. A resistor is connected between nodes 2 and 3. A capacitor is connected between node 3 and ground. The project steps.son is also connected between nodes

3 and 4, with Port 1 corresponding to node 4 and Port 2 corresponding to node 3. Port 1 of the network example\_net corresponds to node 1 and port 2 of the network corresponds to node 4.

A netlist project is simply a list of these elements, as you can see in the netlist pictured above. Notice that the first number after the name of the element is the network node which corresponds to port 1 of the element, the second number is the network node which corresponds to port 2 of the element, and so on, for all the ports in an element.

The sequence of steps for a netlist project analysis may be summarized as follows:

- 1 You input the netlist using the project editor in netlist mode. The project editor allows you to create and edit networks, network elements, project elements, modeled elements, and data file elements in your netlist. You also input the analysis controls which may include defining parameters in the netlist.
- 2 **Em** reads the netlist project which contains circuit and analysis control information. This includes S-, Y- and Z-parameter data files, modeled elements, geometry subprojects (project elements) and network elements.
- 3 **Em** uses the analysis controls input as part of the netlist project to run each electromagnetic analysis invoked by the network file. It is possible to configure the analysis controls in such a way that geometry subprojects are analyzed using their own analysis controls. Netlist subprojects always inherit their analysis controls from the present netlist.
- 4 Once the analysis of geometry subprojects is complete, **em** performs the circuit analysis specified in the netlist.
- 5 **Em** combines the electromagnetic results with the circuit results to obtain the desired output results.

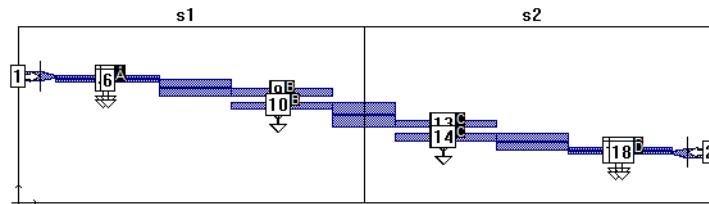
Note that the above sequence of steps is generalized for analyses which include both electromagnetic and circuit analysis. In cases where the overall analysis is restricted to either electromagnetic analysis or circuit analysis, some of the steps are omitted.

### Example of Node Assignments in a Complex Circuit

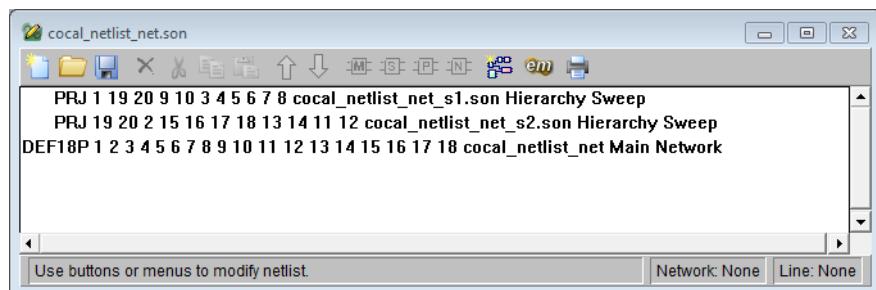
As discussed above, a node is used in a netlist to connect two elements. If an element is a geometry project with multiple internal ports, as is the case with co-calibrated ports, nodes are assigned in the netlist to each port. In larger circuits, with multiple co-calibrated ports, the netlist can be quite complex, so an example is

shown below of performing circuit subdivision of a large circuit with co-calibrated ports. The resulting netlist and subprojects will be examined to illustrate how the ports in the geometry subprojects are assigned to nodes in the netlist which is created when the circuit subdivision is performed. For a complete explanation of Circuit Subdivision, please refer to Chapter 13, “Circuit Subdivision” on page 255.

The full circuit, cocal\_netlist.son, which contains 18 ports, is pictured below with a subdivider. There are 16 co-calibrated ports, each group consisting of four ports in the interior of the circuit. The subdivider is set so that the full circuit will be subdivided into two geometry projects, each containing half of the full circuit. When the circuit is subdivided, a netlist is created which connects the two geometry projects together into a circuit equivalent to the full geometry project. When the netlist is analyzed, each geometry subproject is analyzed and then those simulation results are used to simulate the response of the netlist circuit.

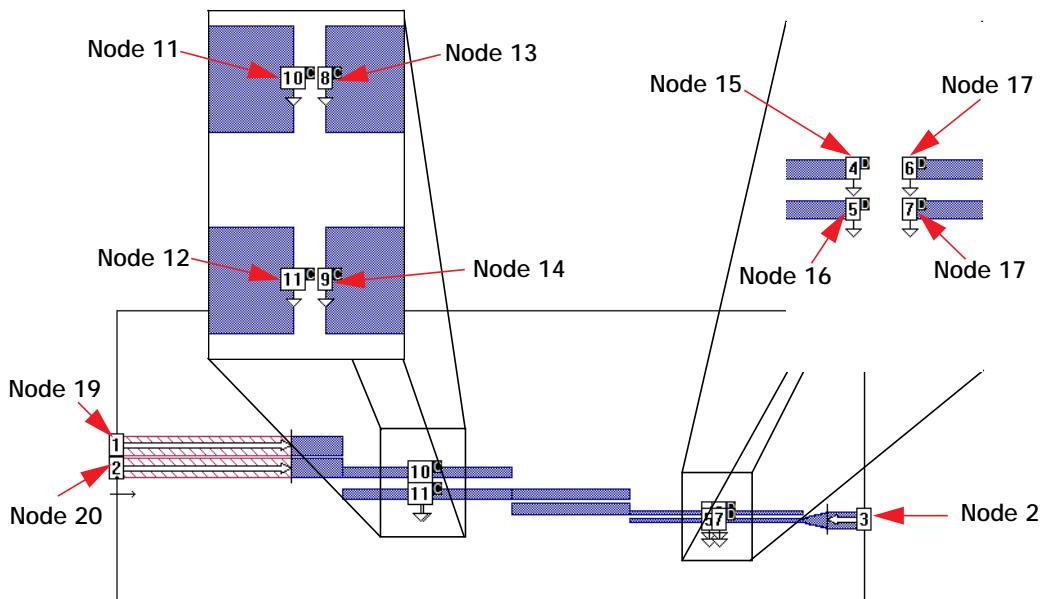
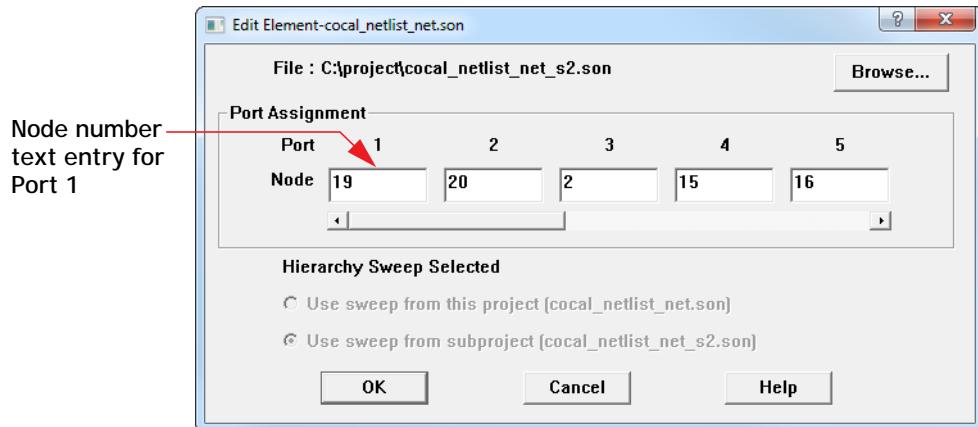


Once the subdivision is performed, three new projects are created. The netlist, cocal\_netlist\_net.son, the S1 subproject, cocal\_netlist\_s1.son, and the S2 subproject, cocal\_netlist\_s2.son. The netlist is shown below. Note that the main network has 18 ports, the same number as the original project before splitting it. For the main network, the node number and port number are identical.



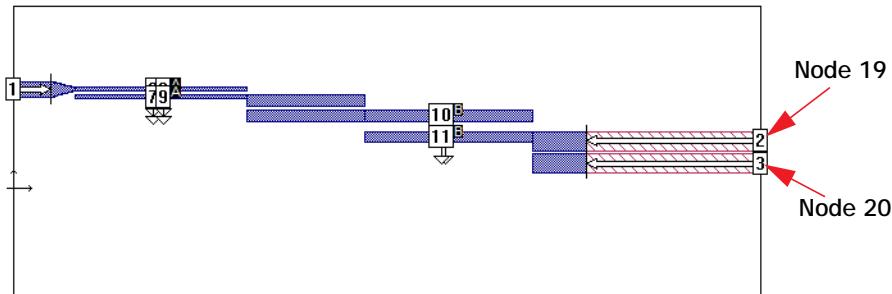
The second line of the netlist shows the Project Element which defines the inclusion of the geometry subproject cocal\_netlist\_s2.son. Show below is the Edit Element dialog box for the subproject displaying the correspondence of netlist nodes to circuit ports. Also shown is a view of the subproject with both the corresponding ports and nodes labeled. When you add an element to a netlist, each port defined in the element is assigned to a node in the netlist. The Edit Element dialog

box will contain a text entry box for each port in the element. You enter the desired node number for the port in that port's text entry box. In the case of a circuit subdivision, this assignment of nodes is done automatically by the software.



Note that there are two nodes, 19 and 20, which appear in the project elements, but are not part of the main network which uses nodes 1 through 18. Nodes 19 and 20 are used to connect ports 2 and 3 of cocal\_netlist\_s1.son to ports 1 and 2 of

cocal\_netlist\_s2.son. These were the ports added to the circuit when it was subdivided. Connecting these ports by using the same netlist nodes makes the netlist circuit equivalent to the original circuit before subdivision.



cocal\_netlist\_s1.son

## Creating a Netlist

You create a netlist using the project editor. To create a new netlist, select *File ⇒ New Netlist* from the main menu of the project editor. The project editor tool bar and menus change for the netlist editor, to allow you to add elements and networks to your netlist.

The initial netlist file contains a default two-port network named Net. The last network in the netlist is the main network. The main network is the network whose solution you are solving for in this netlist. When you analyze the netlist, the response data produced in the analysis is for the main network.

You can edit the name and attributes of this network including the number of ports by double-clicking on the entry. This is true of all entries made in the netlist; you must double-click on them to open the dialog box which allows you to edit the entry or select the item, then select the *Tools ⇒ Modify* command from the main menu.

## Netlist Example Files

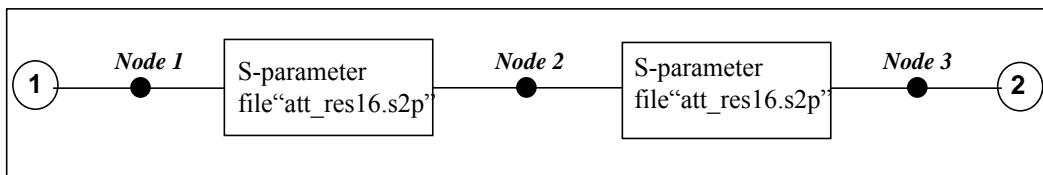
All of the example files used in this chapter are available in the “Att” example in the Sonnet examples. You should save the entire folder into your working directory if you wish to execute the examples. You may access the Sonnet Example

Browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window.

## Cascading S-, Y- and Z-Parameter Data Files

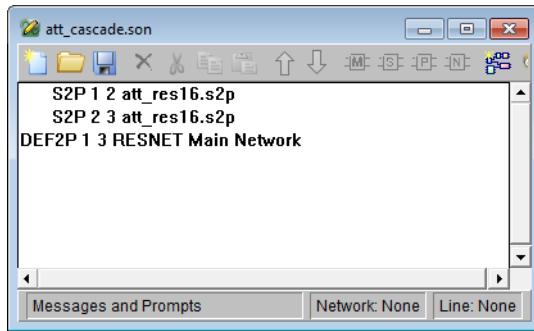
A particularly useful feature provided by a netlist project is the ability to cascade multiple S-, Y- and Z-parameter data files. There are no restrictions on the file formats which may be cascaded. For example, you can cascade **em** Z-parameter data in Touchstone format with measured S-parameter data in Super-Compact format. In addition, **em** can analyze at frequencies which are not included in the data files. **Em** automatically interpolates if there are any differences between the requested frequency points and those in the data files.

A good example of a cascading operation is the project att\_cascade.son, which is included in the Att example for this chapter. A schematic representation of the two-port circuit is shown below. This circuit consists of two identical thin film resistors connected in series. The S-parameters from the geometry project analysis on the thin-film resistor are used as a data file element in the netlist. The desired output network is the series combination of resistors. The S-Parameter data file, att\_res16.s2p, as well as the geometry project, att\_res16.son, used to generate the data file, are included in the examples for this chapter.



The two-port S-parameters contained in file "att\_res16.s2p" are cascaded to obtain an overall set of two-port S-parameters.

The netlist, att\_cascade.son, for the circuit is pictured below.



The main network, Resnet, has two ports indicated by the “2” in DEF2P. Port 1 corresponds to node 1 in the network. Port 2 corresponds to node 3. There are two data file elements in the network. The first entry is the response file att\_res16.s2p with 2 ports. Port 1 corresponds to node 1 of the network, which, as mentioned above, is port 1 of the whole circuit. Port 2 for the data file corresponds to node 2 of the network. The other entry is also for the data file att\_res16.s2p except that port 1 of the data file goes to node 2 of the network which means that port 1 of the second data file is connected to port 2 of the first data file. Port 2 of the second data file corresponds to node 3 of the network. Node 3 of the network is Port 2 of the network “RESNET”.

The S-Parameters for an analysis of the netlist are shown below.

```
Frequency: 200 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
200.000000 0.250782 -5.309 0.748778 -6.263 0.748778 -6.263 0.250782 -5.309

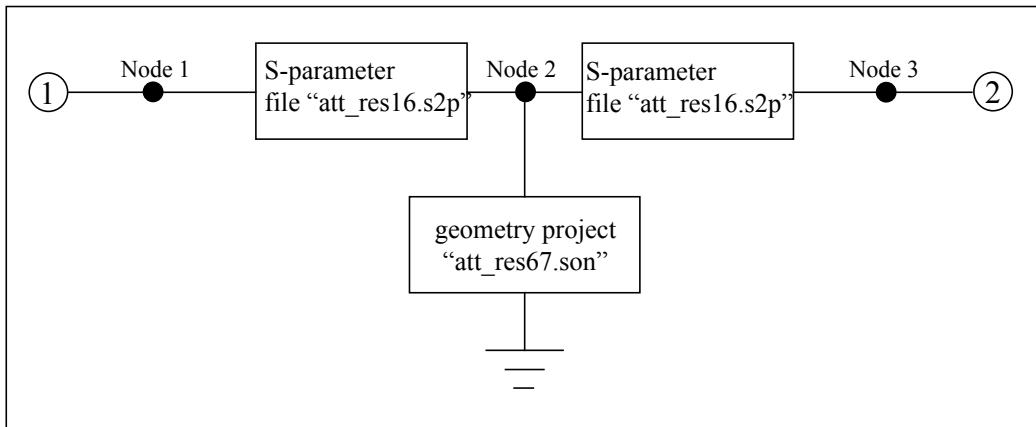
Frequency: 300 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
300.000000 0.250310 -7.963 0.748702 -9.395 0.748702 -9.395 0.250310 -7.963

Frequency: 400 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
400.000000 0.249650 -10.62 0.748595 -12.53 0.748595 -12.53 0.249650 -10.62
```

## A Network File with Geometry Project

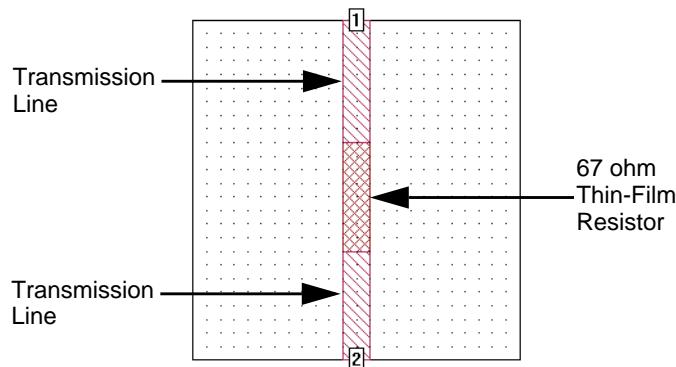
The next example demonstrates a netlist project analysis which invokes a geometry project analysis in conjunction with using previously generated data.

To demonstrate a netlist with a geometry project, the two-port T-attenuator shown below will be analyzed.

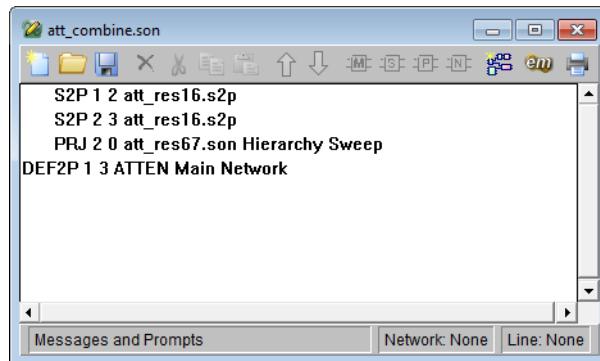


The two-port T-attenuator will be analyzed with *em* to demonstrate a combined electromagnetic/circuit analysis.

Pictured below is the geometry project “att\_res67.son”, which is a 67 ohm thin-film resistor. This project is read by **em** and analyzed during the netlist analysis. The results of the project analysis are used to compute the results for the netlist.



The netlist project, att\_combine.son is shown below. The project att\_combine.son is available as part of the Att example for this chapter.



The primary distinction between the netlist shown above and the previous netlist is that this netlist contains an instruction to perform a project analysis. The PRJ keyword instructs **em** to run an electromagnetic analysis on the project “att\_res67.son” using the analysis controls from the netlist. The analysis control use is indicated by “Hierarchy Sweep” in the PRJ statement. When control is set to “Hierarchy Sweep”, **em** automatically analyzes the subproject at the same frequency sweep and run options as the netlist.

During the analysis, **em** performs the following steps:

- 1     Reads S-parameter data from the file “att\_res16.s2p”.
- 2     Performs an electromagnetic analysis of the geometry project “att\_res67.son”, a 67 ohm thin-film resistor.
- 3     Combines the S-parameter results from the electromagnetic analysis with the S-parameter results from “att\_res16.s2p” to obtain an overall set of S-parameters for the T-attenuator.



### TIP

---

Before executing a PRJ statement, **em** checks for the existence of data at the specified control frequencies. If the data already exists, and the project has not changed since the data was generated, **em** does not execute an electromagnetic analysis, but uses the available data.

---

The listing below shows the output of the netlist analysis, as it appears in the analysis monitor, which contains the overall set of S-parameters for the T-attenuator.

```

Frequency: 200 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
200.000000 0.008924 67.700 0.500516 -5.758 0.500516 -5.758 0.008924 67.700

Frequency: 300 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
300.000000 0.013072 68.918 0.501160 -8.647 0.501160 -8.647 0.013072 68.918

Frequency: 400 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
400.000000 0.017177 67.763 0.502055 -11.54 0.502055 -11.54 0.017177 67.763

```

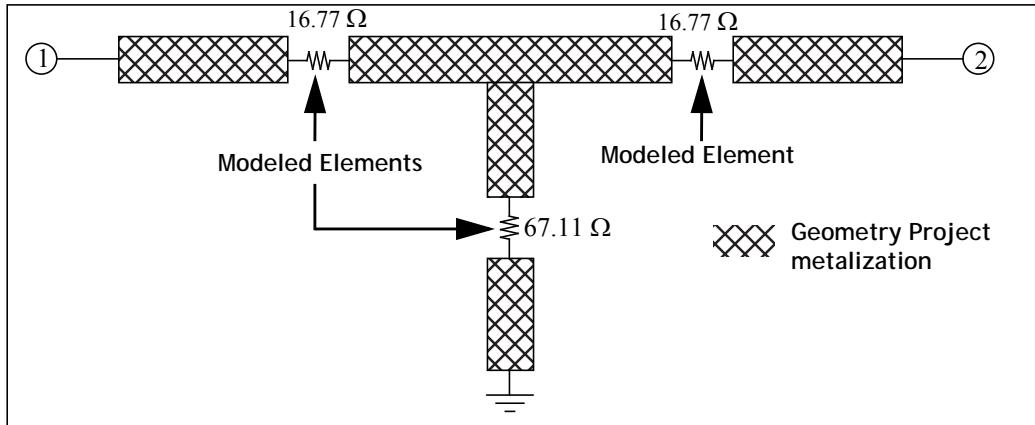
## Inserting Modeled Elements into a Geometry

Another very useful feature of the netlist project is the ability to insert modeled elements into a geometry project after an electromagnetic analysis has been performed on that circuit. A modeled element is an ideal element such as a resistor, inductor, capacitor or transmission line, which has a closed-form solution. No electromagnetic analyses are performed on modeled elements.

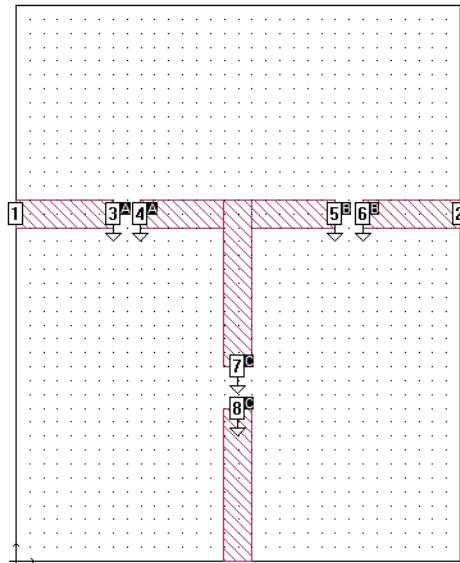
To demonstrate the use of modeled elements, we will again analyze the T attenuator. However, instead of the attenuator being the result of connecting the results of electromagnetic analyses as shown previously in the chapter, in this case, the geometry project, att\_lgeo.son, has the full attenuator with cutouts where the modeled elements need to be inserted. The three resistors will not be analyzed as part of the geometry project, but will be inserted as modeled elements in the netlist. The figure below shows the circuit layout with the modeled resistor elements. A geometry project for the transmission line structures are created first. A netlist project will then be used to insert the three resistors and calculate two-port S-parameters for the overall circuit.

To accomplish this task, it is necessary to create a geometry project with the transmission line structure and three “holes” where modeled elements will eventually be inserted. The figure on page 250 shows such a geometry project. Here, pairs of co-calibrated ports have been placed on the edges of each modeled element “hole”. Co-calibrated internal ports are identified as a calibration group with a common ground node connection and a defined terminal width. When **em** performs the electromagnetic analysis, the co-calibrated ports within the group are simultaneously de-embedded. When the modeled elements are inserted later on,

each is connected across the corresponding pair of co-calibrated ports. Note that under certain conditions, ungrounded-internal ports can be used instead of co-calibrated ports. See “Using Ungrounded-Internal Ports,” page 252, for details.

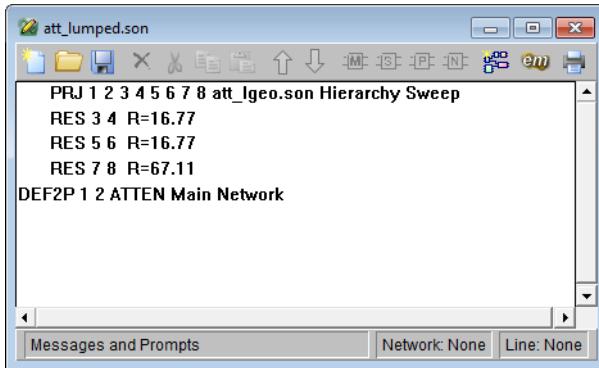


The two-port T attenuator will be re-analyzed to demonstrate the use of modeled elements.



The geometry file “att\_lgeo.son” contains three sets of co-calibrated ports placed at locations where modeled elements will eventually be inserted. This file is available as part of the Att example used for this chapter.

Below is the netlist, att\_lumped.son, that will be used for this example.



The screenshot shows a software interface with a title bar 'att\_Lumped.son'. Below the title bar is a toolbar with various icons. The main area contains the following text:

```
PRJ 1 2 3 4 5 6 7 8 att_lgeo.son Hierarchy Sweep
RES 3 4 R=16.77
RES 5 6 R=16.77
RES 7 8 R=67.11
DEF2P 1 2 ATTEN Main Network
```

At the bottom of the window are three tabs: 'Messages and Prompts', 'Network: None', and 'Line: None'.

The netlist above instructs **em** to perform the following steps:

- 1 Perform an electromagnetic analysis on the geometry file “att\_lgeo.son” using the Frequency sweep and run options defined for this netlist. Note that according to the PRJ line, Ports 1-8 correspond to nodes 1-8 respectively in the main network, atten. The node numbers are listed after the PRJ keyword in the order of ports in the circuit.
- 2 Insert a 16.77 ohm resistor between nodes 3 and 4 which is the equivalent of inserting the resistor between autogrounded ports 3 and 4 in the geometry project.
- 3 Insert a 16.77 ohm resistor between nodes 5 and 6.
- 4 Insert a 67.11 ohm resistor between nodes 7 and 8.
- 5 Calculate an overall set of S-parameters for the T attenuator.

The two projects, att\_lgeo.son and att\_lumped.son are available in the Att example for this chapter.

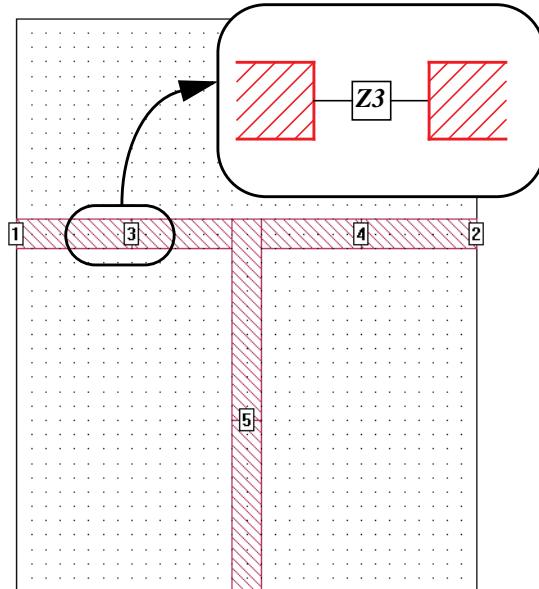
The listing below is the analysis output as it appears in the analysis monitor. Note that these results are similar to the results given above for distributed elements.

```
Frequency: 200 MHz      Frequency completed Tue May 03 14:28:44 2011.  
S-Parameters. 50.0 Ohm Port Terminations.  
Magnitude/Angle. Touchstone Format. (S11 S21 S12 S22).  
200.000000 0.008537 67.966 0.500406 -5.009 0.500406 -5.009 0.008537 67.966  
  
Frequency: 300 MHz      Frequency completed Tue May 03 14:28:44 2011.  
S-Parameters. 50.0 Ohm Port Terminations.  
Magnitude/Angle. Touchstone Format. (S11 S21 S12 S22).  
300.000000 0.012487 70.314 0.500823 -7.517 0.500823 -7.517 0.012487 70.314  
  
Frequency: 400 MHz      Frequency completed Tue May 03 14:28:45 2011.  
S-Parameters. 50.0 Ohm Port Terminations.  
Magnitude/Angle. Touchstone Format. (S11 S21 S12 S22).  
400.000000 0.016447 70.118 0.501406 -10.03 0.501406 -10.03 0.016447 70.118
```

## Using Ungrounded-Internal Ports

In the example presented above, a pair of co-calibrated ports was placed at each location in the *em* circuit layout where a modeled element would eventually be inserted. It is also possible to perform the same analysis using ungrounded-internal ports, because each resistor in this example is a series modeled element without access to ground. Any time access to ground is not required for a modeled element, you can replace the pair of auto-grounded ports with a single ungrounded-internal port.

The figure below shows a geometry project for the T attenuator with ungrounded-internal ports at each modeled element location. Note that the gaps between polygons at these locations have been removed. This is because you must attach ungrounded-internal ports between two abutted polygons. This slightly impacts the overall performance of the attenuator.



The geometry project “att\_Igeo2.son” uses ungrounded-internal ports at locations where modeled elements will eventually be inserted.

The network file shown below connects the desired resistors across the ungrounded-internal ports of the network shown on page 253. Since ungrounded-internal ports do not have access to ground, only a single node is specified when connecting an element across them.



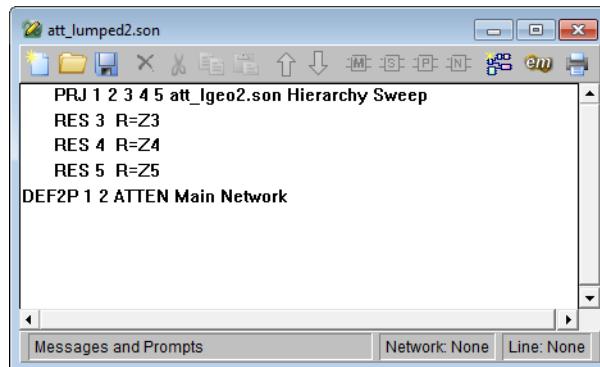
## WARNING

---

Ungrounded-internal ports have one terminal connected to an edge of a polygon and the second terminal connected to an abutted edge of a second polygon. Ungrounded-internal ports do not have access to ground. Therefore, only 1-port elements or 1-port networks may be connected across ungrounded-internal ports. Resistors, capacitors, and inductors are technically one-port elements and therefore, may be inserted in place of an ungrounded-internal port in a netlist.

---

The netlist for this circuit, att\_lumped2.son, is shown below. Both the geometry project, att\_lgeo2.son, and this netlist are available in the Att example provided for this chapter.



```
PRJ 1 2 3 4 5 att_lgeo2.son Hierarchy Sweep
RES 3 R=Z3
RES 4 R=Z4
RES 5 R=Z5
DEF2P 1 2 ATTEN Main Network
```

An important feature to notice in this netlist is the use of parameters. Three parameters, Z3, Z4 and Z5 have been defined in the netlist project and their values used for the three resistor modeled elements. Parameters are defined in a netlist by selecting *Circuit*  $\Rightarrow$  *Parameters* from the main menu, then entering the parameter name and nominal value in the Parameters dialog box which appears. Z3 and Z4 are equal to 16.77 and Z5 is equal to 67.11.

The listing below shows the S-parameter results obtained from the analysis with ungrounded-internal ports. These results are very similar, but not identical, to the results for co-calibrated ports. The differences are primarily due to the change in the gap size between polygons at the points where lumped elements are inserted.

```
Frequency: 200 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
200.000000 0.009217 68.496 0.500482 -5.785 0.500482 -5.785 0.009217 68.496

Frequency: 300 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
300.000000 0.013510 70.114 0.500994 -8.683 0.500994 -8.683 0.013510 70.114

Frequency: 400 MHz
50-Ohm S-Params. Mag/Ang. Touchstone Format. (S11 S21 S12 S22).
400.000000 0.017788 69.364 0.501707 -11.59 0.501707 -11.59 0.017788 69.364
```

# Chapter 13

# Circuit Subdivision

---

## Introduction

Sonnet provides the capability to take a large circuit and split it into any number of smaller projects, then connect the results in a netlist project to produce a response for the whole circuit. This method can significantly reduce the required processing time and memory necessary to analyze the circuit while still obtaining an accurate answer.

The number of subsections in a circuit is one of the most important factors in determining processing time since the matrix solve time is proportional to  $N^3$ . To illustrate how circuit subdivision reduces processing time, consider two subprojects each with half as many subsections as the source project. The total matrix solve time is now four times faster:

$$2(N/2)^3 = N^3/4$$

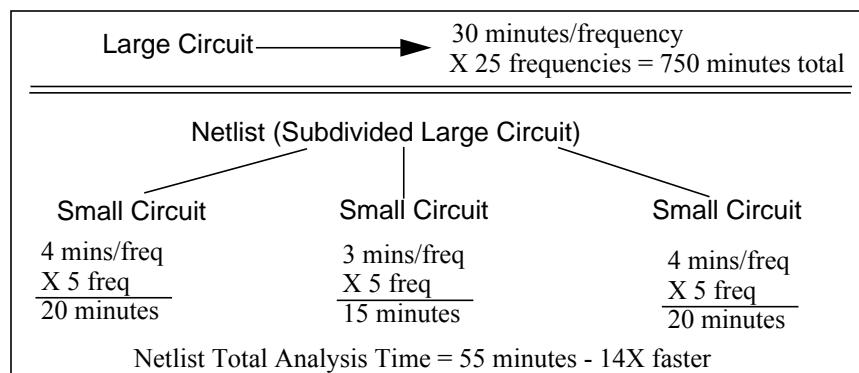
Circuit subdivision allows you to take advantage of this technique by breaking your circuit into smaller parts with fewer subsections, hence, requiring less processing time and memory to analyze. The trade off is that you introduce some error into the analysis. However, by subdividing the circuit appropriately you can minimize the error while still obtaining the reduction in processing time.

The circuit should be split where there is no coupling across the subdivision line. Areas where significant coupling occurs must be contained within a subproject. In this way, all the significant coupling in the circuit is accounted for. If care is taken when subdividing the circuit, the accuracy of the results is very high.

Circuit subdivision is not appropriate for every design, but in the cases of large circuits (5-10 minutes processing time per frequency) where it is applicable, you can obtain marked increases in processing efficiency.

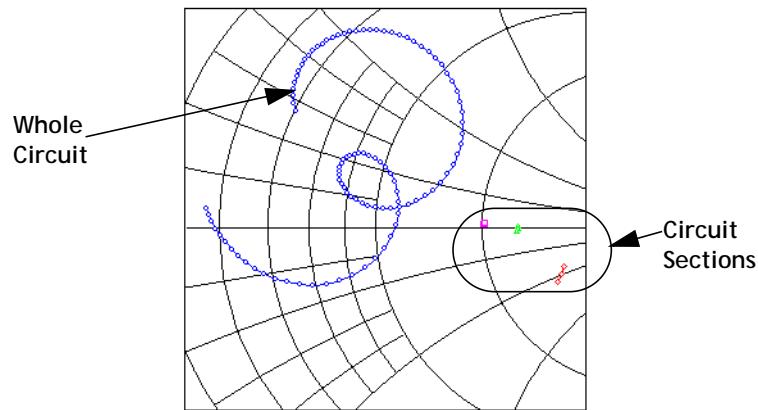
Another advantage of circuit subdivision is the use of frequency interpolation in the master netlist analysis. A netlist is used to connect the response data of the subprojects of the circuit to simulate the full circuit. If the subprojects are chosen in such a way that their response data does not vary significantly over the frequency band, very few frequency points need to be calculated for the subproject. So not only do the smaller files require less time and memory because of their smaller size, but you can also analyze these smaller circuits at fewer frequency points. Interpolating in the netlist file requires much less processing than calculating data for a frequency point in a geometry project.

Shown below is an example showing the typical advantages of using this approach.



When the netlist analysis is performed, *em* will interpolate to provide simulation data at frequencies not specified in the subprojects. Each subproject should be analyzed at the same minimum and maximum frequency as the overall analysis and at enough points in between to provide for reasonable interpolation of data at frequencies which fall between these values. As you can see from the Smith chart below, while you need many frequency points to obtain reasonable response data for

the whole circuit, you need far fewer frequency points to obtain accurate data for the smaller pieces of the whole circuit, whose response data does not vary appreciably.

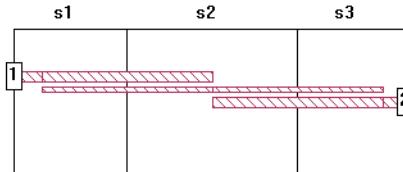


Be aware, however, that in some cases, you may need the added precision of analyzing all the pieces at the same resolution of the frequency band. Interpolation is best used when the response of a subproject varies little over the frequency band and the analysis time of the subproject is appreciable.

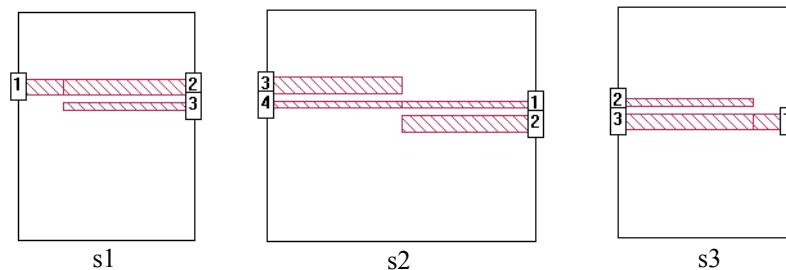
## Circuit Subdivision in Sonnet

Circuit subdivision in Sonnet allows you to insert subdivision lines in your geometry in the project editor. These subdivision lines create the sections from which the subdivide command makes geometry subprojects. When you select the subdi-

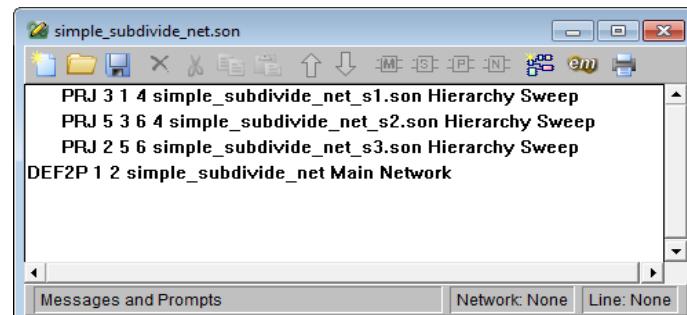
vide command, the software creates a main netlist file and the geometry subprojects. The main netlist connects the subprojects so that the response data for the netlist may be substituted for the response data of the source project.



Source Circuit with Subdivision Lines Added



Generated Subprojects



Generated Main Netlist

You should also be aware that if your main circuit contains any parameters or dimensions, they are removed during the subdivision process. After the subprojects are created, you may enter parameters in any of the geometries. In fact, it is possible to run optimizations on the main netlist project using a parameter in one of the subprojects.

Performing circuit subdivision as a method of analysis should, in general, be done as follows:

- 1** You should input as many of the circuit properties as possible before subdividing. Dielectric layers, dielectric brick and metal types, grid size, top cover height, etc. are inherited by the created geometry subprojects.

- 2 Decide where to subdivide your circuit. This step often requires expertise and experience to avoid splitting the circuit at a junction where there is coupling across the subdivision line.
- 3 Create the subdivision lines in the project editor. These lines are used to create the subprojects. A geometry project is created for each segment of your circuit. These geometry projects contain significantly smaller geometries that may be analyzed faster using less memory.
- 4 If you plan to take advantage of the netlist interpolation feature, set up the analysis frequency controls as a coarse resolution of the entire desired frequency band in the project before subdividing. Both the netlist and geometry subprojects all inherit these frequency specifications. Entering these frequency controls now in the Analysis Setup dialog box saves having to enter them in each individual subproject.
- 5 Subdivide the circuit in the project editor to create the subprojects and netlist project which connects the individual subprojects in a network equivalent to the circuit as a whole. Ports and reference planes are added to the subprojects as needed to connect to the larger circuit.
- 6 Edit the subprojects to fine tune the geometries, if needed. Possible adjustments would include the use of a binary box, adjusting the grid size, setting z-partitions for bricks, changing the frequency sweep specification, and adding parameters.



### TIP

---

If you add parameters to a subproject in a netlist, the parameters are not automatically displayed in the netlist. You must save the main netlist and re-open it to display the parameters and make them available for editing.

---

- 7 Set up the analysis controls in the netlist to use the complete set of desired analysis frequencies if the subprojects are already set to analyze the coarse frequency sweep. When the analysis is performed on the master netlist project, **em** interpolates between the frequency points in the subprojects, saving processing time.  
You may also use a Hierarchy Sweep in which the frequency band set up in the master netlist is imposed on the analyses of all the subprojects. This is useful when additional accuracy is needed in the data and you do not wish to use interpolation. This is accomplished by setting the Hierarchy Sweep option in the Analysis Setup dialog box in the project editor.
- 8 Analyze the netlist project. The data response for the netlist project provides analysis results that may be used for the whole circuit.

- 9 It is often a good idea after analyses are complete on the resultant subprojects to check the response data to verify that data was calculated for enough frequency points to provide accurate interpolated data.

Since it is possible for a netlist project to include a netlist subproject, it is possible to use “double” subdivision. After subdividing your initial circuit, you then may use subdivision on one of the resulting geometry subprojects. In this case, you would need to change the name on the appropriate PRJ line from the old geometry subproject to the new netlist subproject.

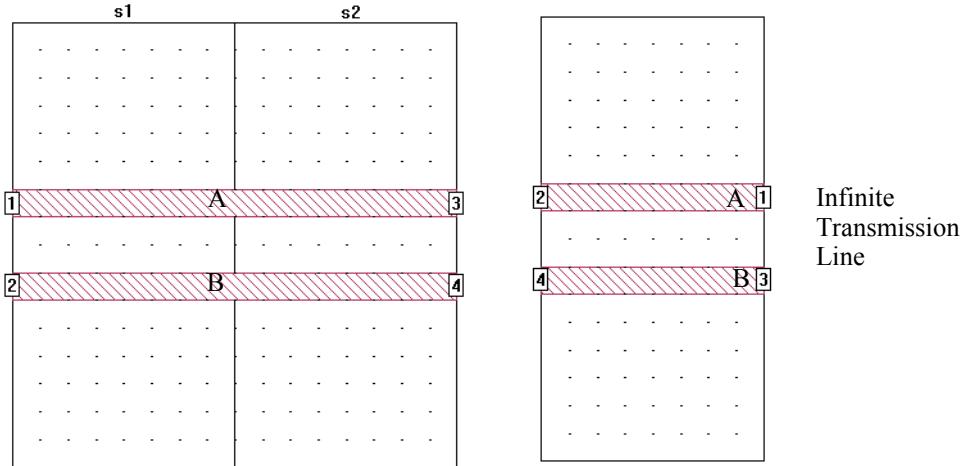
## Choosing Subdivision Line Placement

As mentioned above, the difficult part in using circuit subdivision is to decide where to place your subdivision lines to split the circuit. The subdivision lines should be placed between polygons which have negligible coupling. Places on the circuit where a high degree of coupling or rapidly varying currents are present should be kept within an individual subproject.

The de-embedding of the port discontinuity in Sonnet is done by essentially modeling infinitely long transmission lines at the port. This allows transmission lines to be subdivided with very little loss of accuracy. This includes microstrip lines, stripline, and coupled lines including coplanar. This point is illustrated below.

The circuit shown below, on the left, consists of a coupled transmission line. This is too simple a circuit to require subdivision but is very useful in demonstrating the principle. When subdivided, the circuit is split into two subprojects both of

which would resemble the circuit shown on the right. Since the port discontinuity is modeled as an infinite transmission line when the port is de-embedded the coupling between points A and B is accounted for.

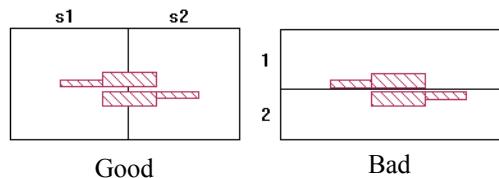


It is important to avoid areas where there is coupling across the subdivision line. Subdivision lines should not split any diagonal polygon edges. Illustrated below are good placements and bad placements of subdivision lines.

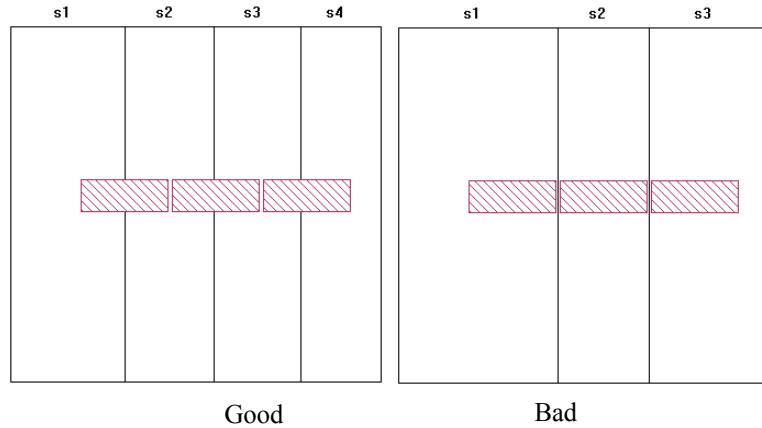
## Good and Bad Placements of Subdivision Lines

This section contains a series of illustrations each showing the good placement of a subdivision line in a circuit and its counterpart showing a bad (and in some cases illegal) placement of a subdivision line. Setting a subdivision line perpendicular to one or more transmission lines provides a good general guideline for line placement.

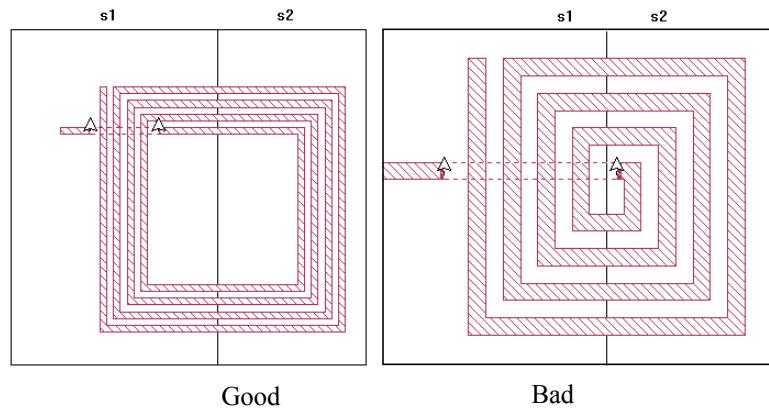
The first example is a pair of coupled lines. As explained above, when you split coupled lines as on the left, very little loss of accuracy results. However, on the right, you have split the coupled pair along the axis where significant interaction takes place. The subprojects have no way to account for this coupling and will produce bad data.



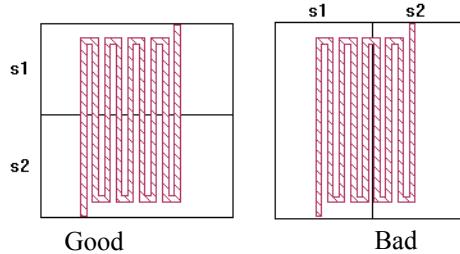
The second example shows how to split a series of resonators. In this type of structure, there is strong coupling at the gaps between adjacent resonators. The example on the left is good since the subdivision lines do not prevent this inter-resonator coupling. The example on the right is incorrect since the resulting sub-projects do not contain the inter-resonator coupling.



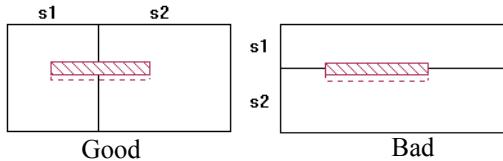
The third example shows a square spiral. The example on the left is a good placement since the location where the spiral is divided is essentially a group of coupled transmission lines and the subdivision line is perpendicular to those lines. Here, the left side of the spiral is sufficiently far from the right side so that coupling is negligible. The example on the right is bad because the lines on the left side of the spiral do couple strongly with the lines on the right side.



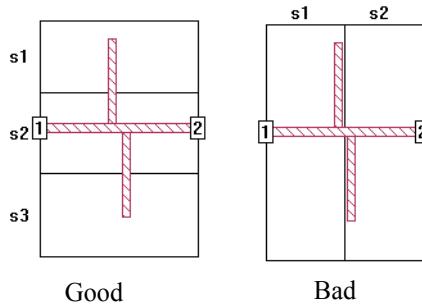
The meander line on the left is split in such a way that Sonnet provides an accurate answer since the bends on the top are far enough away from the bends on the bottom that coupling between them is negligible. The example on the right provides an inaccurate result because the coupling between two close transmission lines is eliminated by the subdivision.



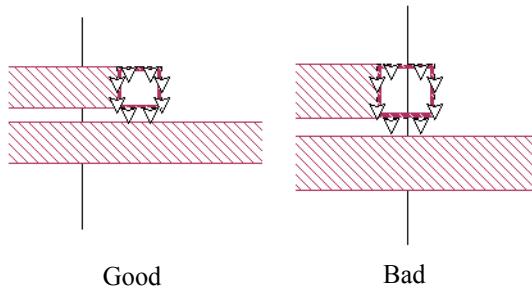
The circuit shown below has coupled transmission lines on two different layers. Once again, it is correct to place a subdivision line perpendicular to the transmission lines, but not parallel to them. Subdivision is valid for multi-layer structures as long as the coupling across the subdivider is negligible.



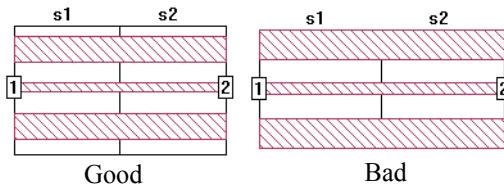
In the double stub circuit shown on the left, the subdivision lines split the polygon perpendicular to the direction of current flow and far from any discontinuities. The circuit on the right however, shows the subdivision line splitting the bases of the two stubs which may be coupled.



The subdivision line shown in the circuit on the right is wrong since the circuit is split in the middle of a via between layers. In general, subdivision lines should never be placed on top of discontinuities, such as vias. The subdivision line on the left is the correct placement.

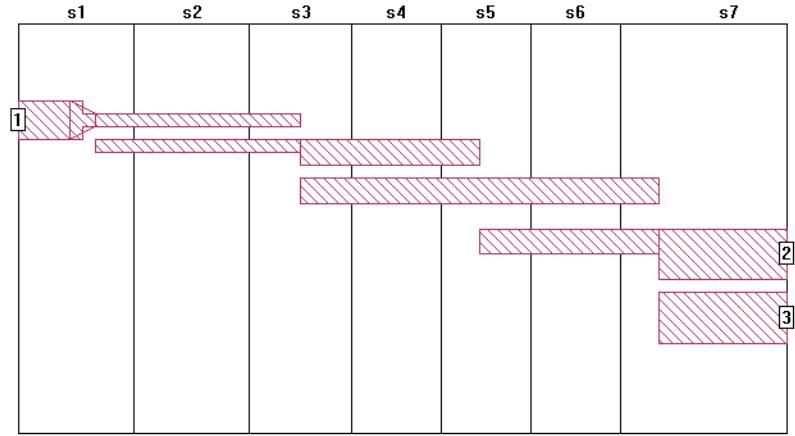


The subdivision line shown in the circuit on the right splits a polygon at the box-wall which is an illegal placement for a subdivision line. It is illegal to subdivide polygons grounded to the box-walls since such polygons do not behave like transmission lines. Also, the new ports added during the subdivide would be shorted to the box-wall. The circuit on the left is correct since there is no contact between the top and bottom polygons with the top and bottom box-wall.

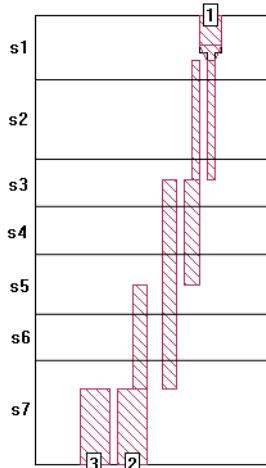


## Subdivision Line Orientation

Subdividers may split the circuit on a horizontal axis or a vertical axis, but you may not mix orientation. Choosing the direction in which you split your circuit is dependent upon the structure of your circuit. Shown below is a typical circuit in which you would use the vertical orientation and another example in which you would use the horizontal orientation.



Example of Vertical Subdividers



Example of Horizontal Subdividers

You may use both orientations by using double subdivision mentioned earlier. The first time you subdivide your main circuit you choose an orientation for your subdivision lines. Then use circuit subdivision on the resulting geometry subprojects, this time using the opposite orientation for your subdivision lines.

Before adding subdividers to your geometry project, you should ensure that specification of your circuit is complete. Subprojects created when you execute the subdivision inherit their properties from the source project. Such properties as cell size, metal types, properties of the dielectric layers, dielectric bricks, metal levels, etc. are all used in the resultant subprojects.

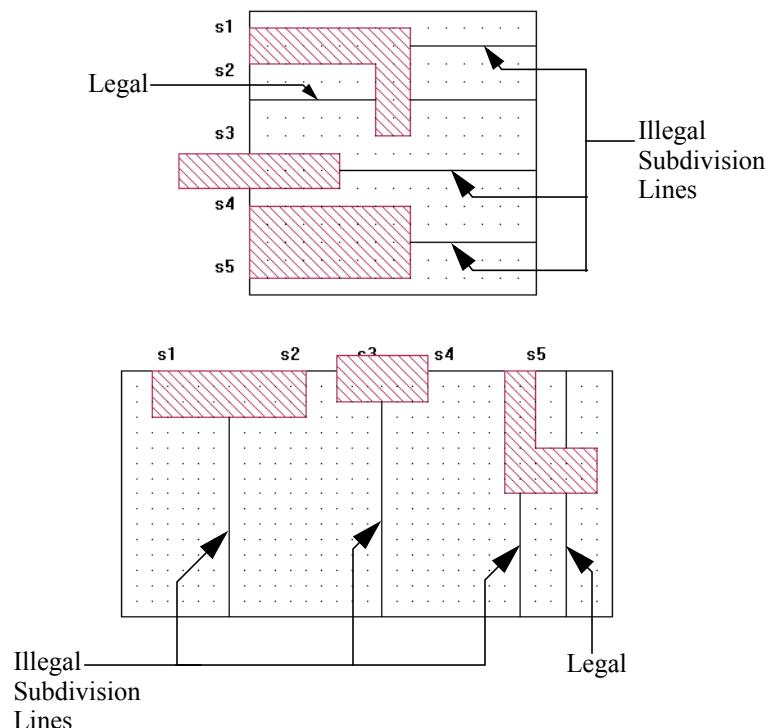
When you place a subdivider in your circuit, a line representing the subdivider appears in the horizontal or vertical plane running through the point at which you clicked. The resultant sections of the circuit are automatically labeled.

Subdivision sections are labeled from left to right, or top to bottom, depending upon orientation. These labels are always sequential and are non-editable.

Once a subdivider has been added to your circuit, you may edit the subdivider as you would any other object in your geometry. You may click on the subdivider and move it. You may also control the display and selection of the subdivider lines and labels in the Object Visibility dialog box and the Selection Filter dialog box.

The following are illegal conditions for subdivision lines:

- May not be off grid.
- Should not be placed where there is coupling across the subdivision line.
- May not be colinear with polygon edges.
- May not split a diagonal polygon edge.
- May not split a port.
- May not be below the line of symmetry.
- May not split a polygon at a box-wall. See the picture below.



Once you have completed adding all the desired subdividers to your circuit, you must save the project before performing the subdivision.

## Setting Up Circuit Properties

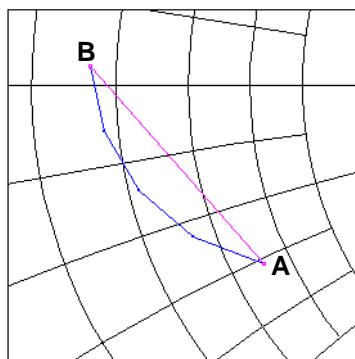
Since the geometry subprojects created by the subdivide inherit their properties from the source project, you should complete entering all the desired attributes for your circuit before performing the subdivide. This includes such things as defining the dielectric layers (which includes the height of the box top), top and bottom box metals, metal and dielectric brick materials, cell size and box size. This saves the effort of having to enter these values in each of the subprojects.

## Setting Up the Coarse Step Size Frequency Sweep

If you plan to use interpolation to obtain response data when analyzing the master netlist project, you should input the coarse frequency sweep at which you wish to analyze the geometry subprojects before subdividing the circuit. These frequencies should cover the same frequency range as the analysis frequencies for the whole circuit but use a coarser step size. The subprojects should be analyzed at the same minimum and maximum frequency as the overall analysis and at enough points in between to provide for reasonable interpolation of the response.

By inputting the coarse frequency sweep prior to subdivision, the master netlist and geometry subprojects created by the subdivide command will all inherit the analysis setup. After subdividing, you will need to enter the desired finer frequency step size in the master netlist project before analyzing it. In addition, you will need to turn off Hierarchy Sweep.

The figure below shows a Smith chart with a circuit analyzed at five frequency points, next to the same circuit analyzed at only two points. As you can see, using only two data points would result in more interpolating error than using five data points. Whether or not two data points is acceptable depends upon the proximity of points A and B. If A and B are very close, then two data points are sufficient. If A and B are far away, then five or more data points should be used.



It is always a good idea to check the Smith chart for the response data of your subprojects to ensure that you have chosen enough frequency points at which to calculate data so that any interpolated data is reasonably accurate.

## Subdividing Your Circuit

The actual subdividing of your circuit into separate geometry subprojects and a master netlist project is performed by the software. You enter the desired names for the master netlist and geometry subprojects. You may also automatically add feedlines of lossless metal to any ports generated in the subprojects.

Feedlines should be added when discontinuities contained in sections of your source circuit need to be moved away from the box-wall to prevent interaction between the box-walls and the discontinuity. The use of feedlines are optional; if you choose to add a feedline, you may use the suggested length calculated by the software or input your own value. By default, the software creates feedlines using the suggested length.

When the subdivide is executed, Sonnet creates a geometry subproject for each section of the circuit in which you placed the subdividers. It also creates a master netlist that connects the geometry subprojects together to produce an equivalent circuit for the original geometry project that you subdivided.

Each of the geometry subprojects uses the properties of the original circuit: cell size, dielectric layers, dielectric and metal materials, analysis setup, etc. Therefore, all the geometry subprojects contain the same analysis setup with the same analysis frequencies specified.

## Analyzing Your Subdivided Circuit

To obtain the desired response data, edit the analysis setup for the master netlist so that all of the desired analysis frequencies are specified. Each of the geometry subprojects are set up with the coarser resolution of analysis frequencies. When the netlist is analyzed, **em** runs the geometry project analyses first to produce response data for each part of the network. Then the analysis of the whole network is executed. Em interpolates to produce data for frequency points in between those available from the analysis of the geometry subprojects.

If properly subdivided, the results of the netlist analysis should provide an accurate solution for your difficult to handle circuit using fewer resources. The use of circuit subdivision is demonstrated in Chapter 14 "Circuit Subdivision Tutorial" on page 271.



# Chapter 14

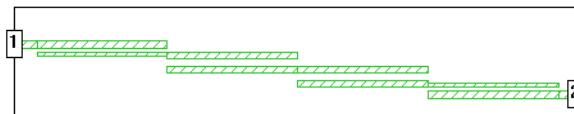
# Circuit Subdivision

# Tutorial

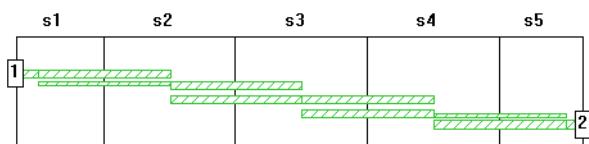
---

This tutorial walks you through how to add subdivision lines, subdivide your circuit, and analyze the final netlist. The results of this subdivision are compared to the analysis of the complete circuit in order to demonstrate the accuracy of the results of the subdivision and the savings in memory. For a detailed discussion of circuit subdivision and the use of subdividers, please refer to Chapter 13.

The circuit, an edge-coupled microstrip bandpass filter, is a fairly simple example of a circuit which you might decide to subdivide. In addition, it is not a very good filter design. This circuit was chosen for the purposes of clarity in explaining circuit subdivision.



You will use four vertical subdivision lines to split the circuit into five sections as shown below.

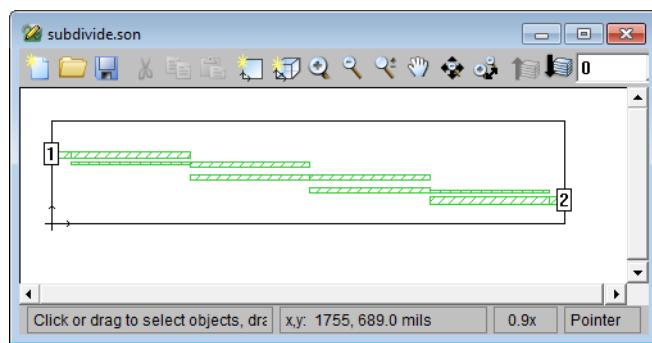


## Obtaining the Example File

You use the example subdivision for this tutorial. The example may be obtained using the Sonnet Example browser. You may access the Sonnet Example browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window. Save the example to your working directory; there are multiple files.

### 1 Open the project **subdivide.son** in the project editor.

The circuit appears as shown below.



## Adding the Subdivision Lines

The first step in subdividing a circuit, as discussed in "Choosing Subdivision Line Placement" on page 260, is to place the subdivision lines that indicate where you wish to split your circuit. Subdivision lines should be placed in locations where there is negligible coupling across the lines. The best place to put subdivision lines in the example used here is at points in the circuit on the coupled lines as far from the discontinuities as possible. Therefore, a vertical subdivision line will be placed in the middle of each coupled pair of polygons.

Each coupled pair of polygons is 595 mils in the x direction. Subdivision lines must be placed on the grid. The closest value to halfway which still remains on the grid is 295 mils. For the first subdivider, you must take into account the feedline polygon which is 100 mils in length. Therefore, the first subdivision line should be placed at 100 mils + 295 mils = 395 mils from the left box-wall.

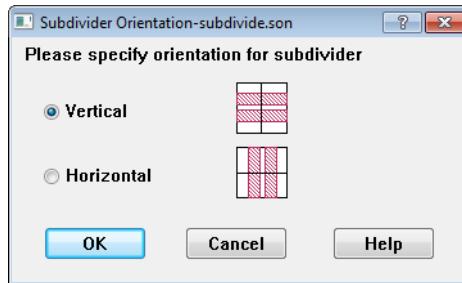
- 2 Select **Tools**  $\Rightarrow$  **Add Subdivider** from the project editor menu while holding down the shift key.



### TIP

Holding down the shift key allows you to enter multiple subdivision lines without having to select the command multiple times.

Since there were no subdivision lines in the circuit when you selected the Add Subdivider command, the Subdivider Orientation dialog box appears on your display.



All subdividers in your circuit must have either a vertical (up-down) orientation or a horizontal (left-right) orientation on the substrate.

- 3 Click on the vertical radio button to select the vertical orientation for your subdividers.

This sets the orientation for all subdividers subsequently added to your circuit.

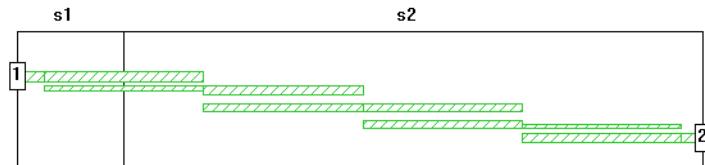
This dialog box does not appear again if you select **Tools**  $\Rightarrow$  **Add Subdivider**. The new subdivider assumes the same orientation. If all the subdividers are deleted from a circuit, then when the Add Subdivider command is used again, this dialog box appears.

- 4 Click on the OK button to apply your selections and close the dialog box.

The cursor changes to indicate that you are adding subdivision lines and a line appears which moves with your cursor.

**5 Move your cursor until the X coordinate of the cursor position in the status bar is 395.0 and click.**

A line representing the subdivider appears in the vertical plane running through the point at which you clicked. The sections of the circuit are now labeled “s1” and “s2”. Subdivision sections are labeled from left to right, or top to bottom, depending upon orientation. These labels are always sequential and are non-editable.

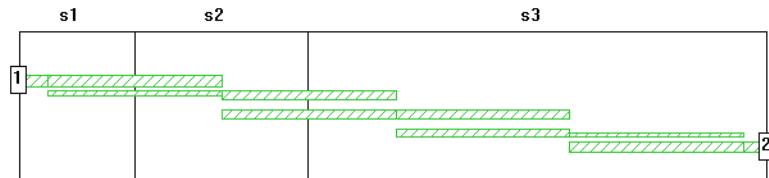


Subdivision lines are always snapped to the grid and may not be placed on top of each other. Once a subdivider has been added to your circuit, you may edit the subdivider as you would any other object in your geometry. You may click on the subdivider and move it. You may also control the display of the subdivider lines and labels in the Object Visibility dialog box, invoked by selecting *View* ⇒ *Object Visibility* from the project editor's main menu.

Since each of the coupled line segments are 595 mils long and you wish to place the subdivision lines at the halfway point, each subsequent subdivision line should be placed 595 mils further to the right in the circuit. So the second subdivision line should be placed at 990 mils from the left box-wall.

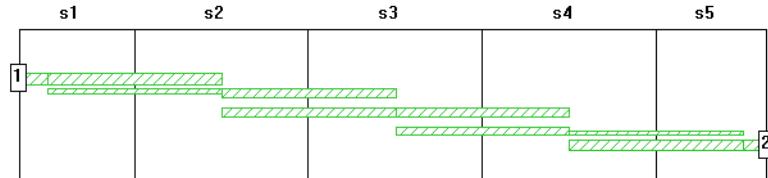
**6 Move the cursor until the X coordinate is 990.0 in the status bar and click to place the second subdivision line.**

The subdivision line appears on your circuit and the sections are relabeled as shown below.



**7 Add subdividers at 1585 mils and 2180 mils from the left box-wall.**

Once you have completed adding all the subdivision lines, press the Escape key to return to pointer mode. Your circuit should now appear like this:



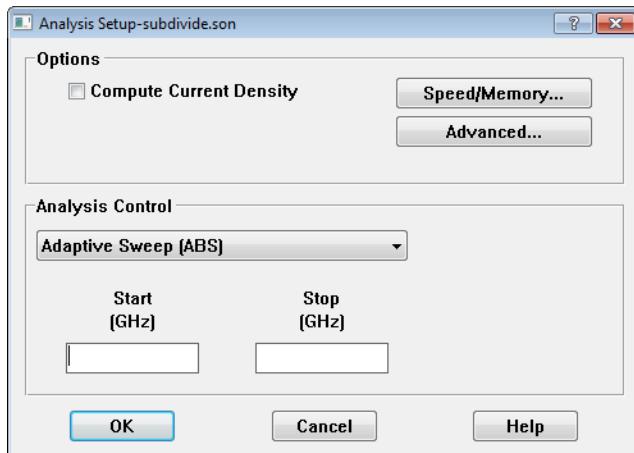
## Setting Up Circuit Properties

For this example, the circuit properties such as box size, dielectric layers, metal materials, etc. have already been input in the example circuit. It is important to have the circuit properties input before performing the subdivide since these are the properties used for all the subprojects created as the result of the subdivide. If you do not enter all the desired properties, you will need to enter them individually in each subproject or modify the original source project and execute the subdivide again.

For this example, you will analyze the netlist using an adaptive sweep (ABS) with Hierarchy Sweep turned on. When the Hierarchy Sweep option is used, the analysis control settings for the netlist are used to analyze all the subprojects in the netlist. The desired frequency band for the circuit is 2.3 GHz to 2.5 GHz. An adaptive sweep provides approximately 300 data points. For more information on the Adaptive Band Synthesis technique, see [Chapter 9](#).

**8 Select *Analysis* ⇒ *Setup* from the main menu.**

The Analysis Setup dialog box appears on your display.

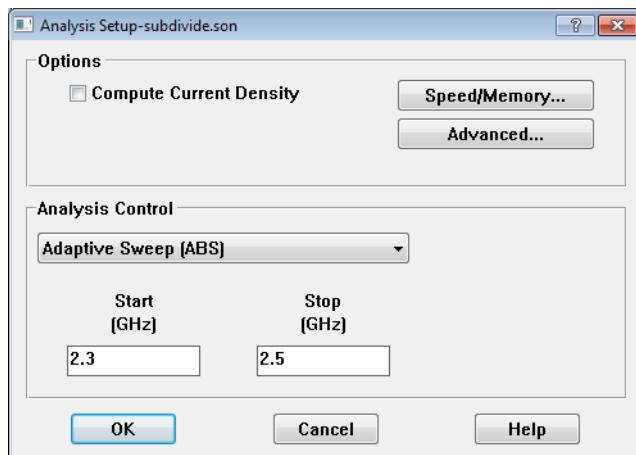


**9 Select “Adaptive Sweep (ABS)” from the Analysis Control drop list if it is not already selected.**

This selects the Adaptive Sweep as your type of analysis. The adaptive sweep provides a fine resolution of response data over the given frequency band. Note that the text entry boxes are updated to reflect your choice of analysis.

**10 Enter 2.3 in the Start box and 2.5 in the Stop box.**

This sets up the analysis frequency band. This analysis setup is duplicated in all of the geometry subprojects when the subdivide is executed, as well as in the main netlist. The Analysis Setup dialog box should appear as shown below.



- 11 Click on the OK button to save the analysis setup and close the dialog box.**
- 12 Select *File*  $\Rightarrow$  *Save* from the main menu.**

The file must be saved before executing the subdivide. The position of the subdivision lines are saved as part of your source project.

## Subdividing Your Circuit

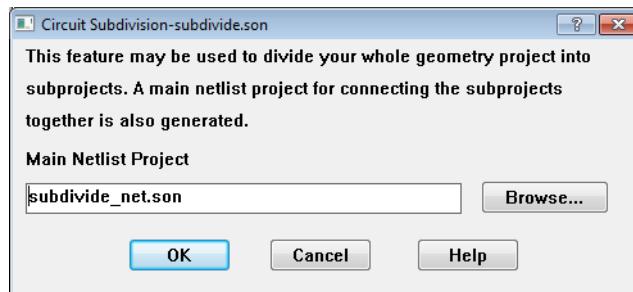
The actual subdivision of the project is executed by the software but you must enter names for the resulting main netlist file and subproject files produced as well as, optionally, defining a feedline length to be added to the subprojects.

Feedlines should be added to the subprojects if you feel it necessary to move discontinuities in the various sections of the circuit further away from the box-walls to prevent any interaction between the discontinuities and box-walls. This can provide a more accurate analysis result for each section of the circuit. Any added feedlines are of lossless metal, regardless of the metal type to which they are attached.

Sonnet software provides a default recommended value for the feedline or you may enter your own value.

- 13 Select *Tools*  $\Rightarrow$  *Subdivide Circuit* from the project editor main menu.**

The Circuit Subdivision dialog box appears on your display.



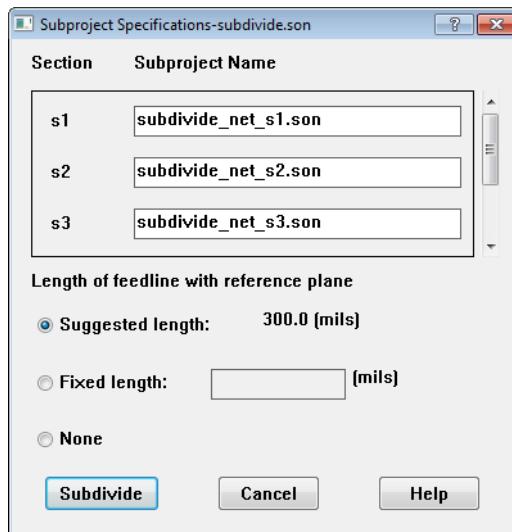
- 14 The name “subdivide\_net.son” is provided by default in the Main Netlist Project text entry box.**

This name is used for the main netlist which connects the geometry projects resulting from the subdivide. The default name is the basename of the source project with a “\_net” added on. You may use any project name you wish but it must be different than the project name of the source file.

If you wish to change the directory in which the resulting files are created, click on the Browse button to open a browse window. If you select an existing project file, you are prompted if you wish to overwrite the existing file.

**15 Click on OK to set the name and close the dialog box.**

The Subproject Specifications dialog box appears on your display as shown below. This dialog box allows you to enter names for each of the geometry subprojects that result from performing the subdivide. Default names, consisting of the main netlist project name with the section number added, are provided but may be edited. For this example, use the default names.



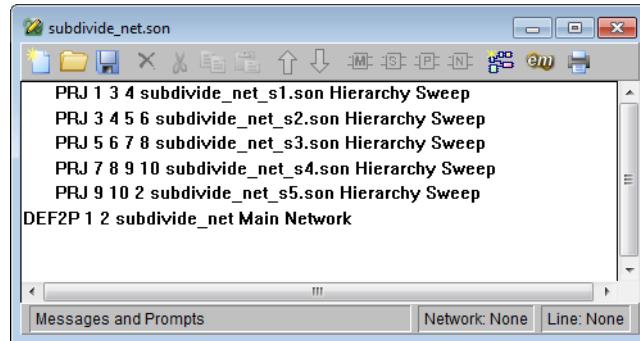
The names for the subprojects must be unique and must be different from the source project name and main netlist name.

The suggested length option is already selected for the feedline length. This feedline of lossless metal is added to ports generated when the subdivide is executed.

To enter your own feedline length, you would select the fixed length radio button and enter the value in the corresponding text entry box. Select the None radio button if you do not wish to add a feedline.

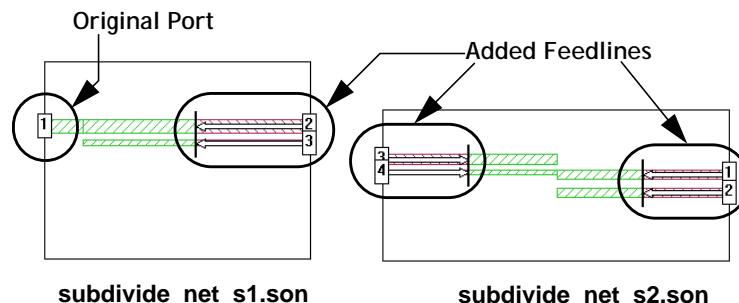
**16 Click on the Subdivide button to execute the subdivide.**

The main netlist and subprojects are created using the names input by you. The main netlist project is opened in the project editor.



The main network is defined as subdivide\_net and has two ports. This corresponds to the source circuit. There is a project (PRJ) entry line for each of the subprojects. The project line includes the setting for the source of the analysis frequencies. A Hierarchy sweep, in which the netlist frequency sweep is imposed on all the project elements, is on by default. If you turn this off, the project default setting of using its own sweep is displayed.

Pictured below are the geometries for the first two sections, subdivide\_net\_s1.son and subdivide\_net\_s2.son. Note that in subdivide\_net\_s1.son, feedlines with a reference plane have only been added to ports 2 and 3, the ports created in the subdivide, but not port 1 which is contained in the source project. All the ports in subdivide\_net\_s2 have feedlines since all were created in the subdivide. Note that the feedlines are all of lossless metal.



## Analysis of the Network File

The last step to complete the analysis of the filter is to analyze the netlist project created by the subdivide. The analysis controls you entered in the original project are the ones you wish to use to analyze the netlist, so the analysis setup is already complete. An adaptive sweep from 2.3 GHz to 2.5 GHz will be performed on the netlist.

- 17 Click on the project editor window containing the netlist to make this the active file.**

This is indicated by the title bar on the netlist being highlighted.



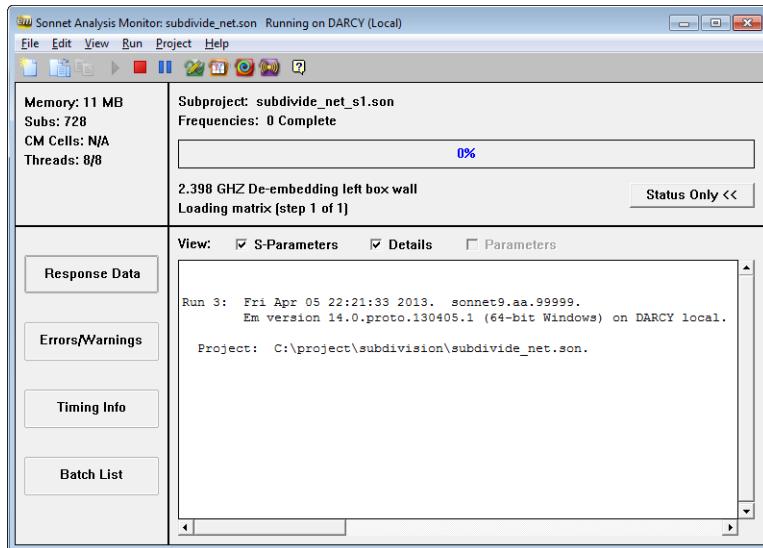
### TIP

You can switch the active file in the project editor by clicking on the title bar of the project window or by selecting the project from the Windows menu on the main menu.



- 18 Click on the Analyze button to launch the netlist analysis.**

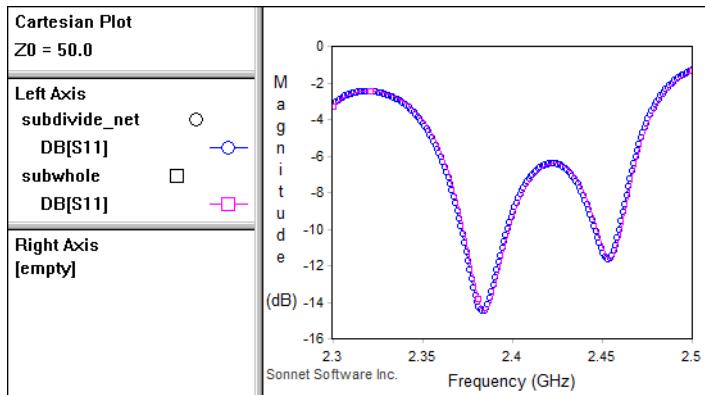
The analysis monitor appears on your display.



The project legend indicates that `subdivide_net_s1.son` is being analyzed. **Em** will perform an adaptive sweep on each of the five subprojects and then use the resulting data to analyze the network. Status messages are output under the progress bar.

There are two results that are significant to observe. A comparison of the netlist analysis data with the analysis data from the source circuit, and a comparison of the amount of time and memory each analysis used. We have provided the source project file including analysis data, subwhole.son, under the example subdivision available in the Sonnet Examples.

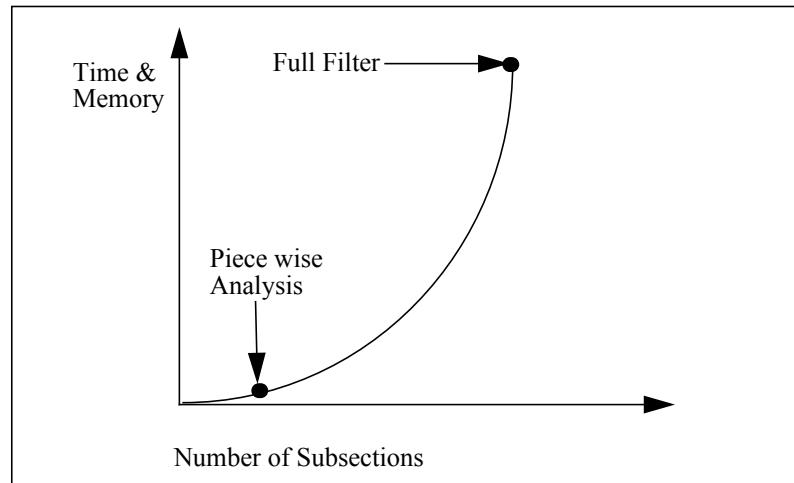
The graph below shows the results of the netlist analysis versus the results of a full analysis of the source project.



As you can see there is very good agreement between the two analysis results. Both files were analyzed on the same computer. The time required for the netlist was actually longer than the time required to analyze the circuit as a whole because this was a simple example chosen for clarity, and the benefits of circuit subdivision are only seen for larger circuits.

Using circuit subdivision reduces your memory requirements for analysis of a large circuit. Each of the subprojects requires less subsections to analyze than the complete circuit. This improvement comes as a result of reducing the number of subsections for any given analysis since both computation time and memory requirements rise sharply as the subsections go up, as shown on the chart below. For

in this example, the entire filter circuit used 2006 subsections while the largest individual piece only required 1400 subsections and the smallest only required 854 subsections.



On many larger circuits the use of the automatic circuit subdivision features in Sonnet can greatly improve the efficiency of your **em** usage.

## Additional Improvements

There are two other ways this circuit could have been made even more efficient. You could have refrained from adding the automatic feedlines and you could have taken advantage of the fact that some of the subprojects were virtually identical.

For the purpose of illustration, this tutorial added feedlines to all ports generated in the subdivide using the recommended length. Feedlines are added to a circuit to move the discontinuities in the subprojects far enough from the box-walls to prevent interaction. In the case of this example, either discontinuities were not present or they were already far enough from the box-wall that additional feedlines were unnecessary.

If you leave the feedlines out by selecting None in the Subprojects Specifications dialog box, the netlist analysis runs 1.5X faster than previously.

The last method that would allow you to decrease the processing time would be to use fewer subprojects in the netlist to create the circuit. Observation of the circuit geometry and response data shows that subdivide\_net\_s1.son and subdivide\_net\_s5.son are virtually identical. The same is true for

subdivide\_net\_s2.son and subdivide\_net\_s4.son. You could edit the main netlist, subdivide\_net.son so that you only use three files: subdivide\_s1.son, subdivide\_net\_s2.son and subdivide\_net\_s3.son to create the whole circuit. This eliminates the need to calculate data for two out of five subprojects. This analysis is 2X faster than the analysis using feedlines and all five subprojects.



# Chapter 15

# Vias

---

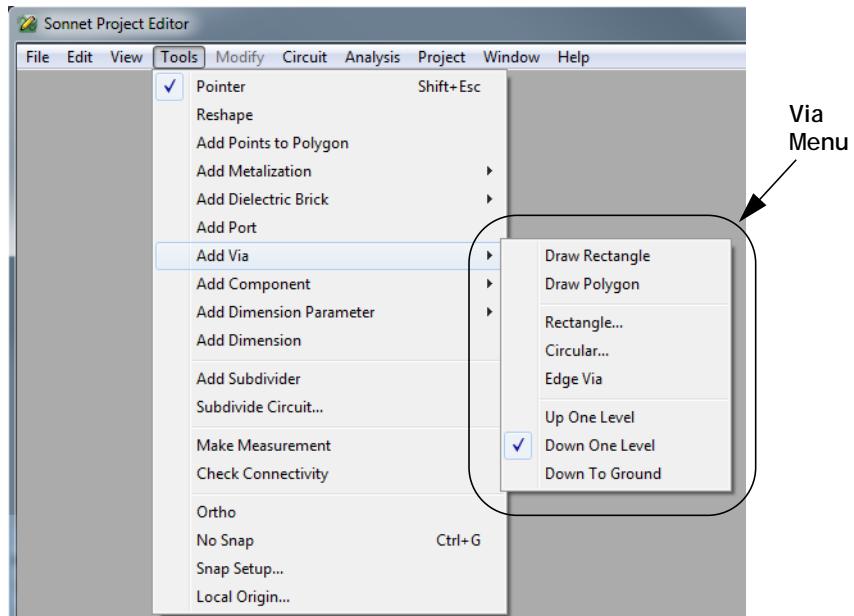
## Introduction

Sonnet vias can be used to connect metal on any level in a circuit to metal on any other level. They are commonly used to model vias to ground, connections to airbridges, and can even be used to approximate wirebonds.

Sonnet vias use a uniform distribution of current along their height and thus are not intended to be used to model electrically long vertical structures. Therefore, the height of the via should be a small fraction of a wavelength. If you need to make a connection through an electrically thick dielectric, you may need to divide the dielectric into multiple layers to achieve an accurate analysis.

## Creating Vias

Vias may be added to your circuit in a number of ways. Both the direction and type of via is chosen in the *Tools* ⇒ *Add Via* menu. You may also add vias to your



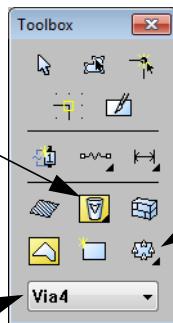
circuit by creating Via Technology Layers, then assigning via polygons to the Technology Layer. For more information about Technology Layers, please see “Technology Layers” in the index of Help. (*Help* ⇒ *Sonnet Help*)

**TIP**

You may also use buttons in the Project Editor Tool Box, shown below, to add vias and control the direction in which the via extends.

For vias not associated with a Technology Layer, this button controls the direction of the via: up one level, down one level or down to ground. Right-click on the button to open a popup menu of all the options.

If you add a via to a Via Technology Layer, it inherits its properties from the Via Technology Layer.

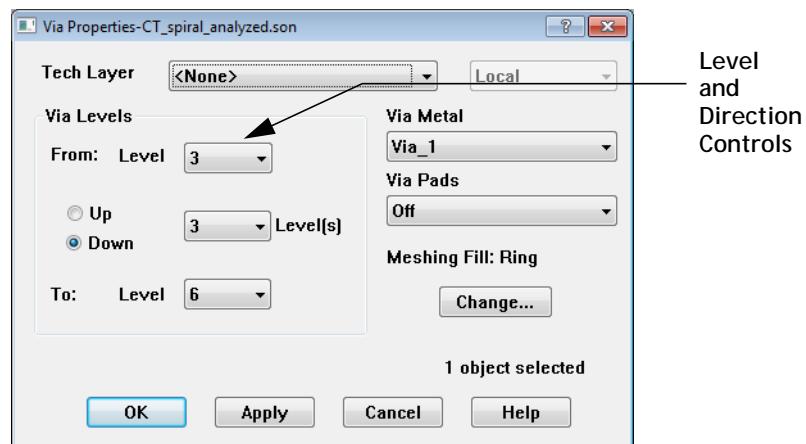


This button controls the type of via polygon: Rectangular, circular or edge via. Right-click on the button to open a popup menu of all the options.

**Via Direction**

Vias connect metal from one level to metal on another level and can extend through multiple layers; therefore, the connected metal levels do not need to be adjacent. The initial direction of the via and the layers through which it extends are based on your default settings when the via is added to your circuit or the settings in the Via Technology Layer to which the via polygon is being added. However, the layers between which a via extends can be easily changed at any time. The default may be set to go up one level, down one level, or down to ground, through multiple layers if necessary. A via which goes down to ground extends from the level of metalization to which it is added through all intervening levels until it reaches the ground (bottom) of the box.

To change the direction of a via, or which metal levels it connects, you use the *Via Properties* dialog box, shown below. You can use the *Modify* ⇒ *Via Properties* command, enabled when at least one via in your circuit is selected, to open this dialog box. Double-clicking on a via also opens this dialog box.



## Via Polygons

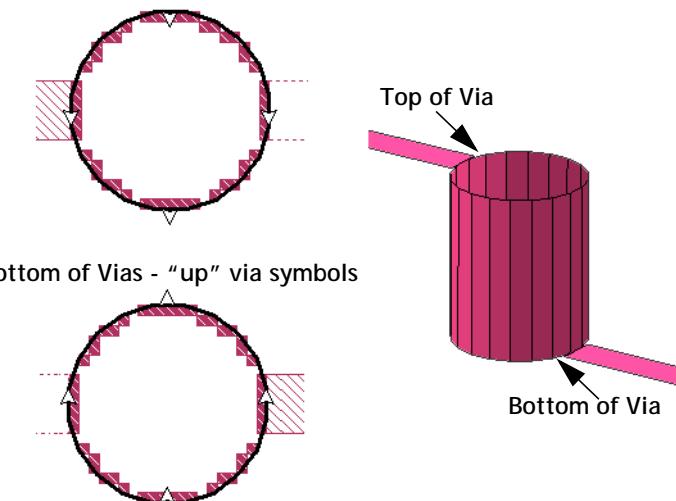
There are two types of vias: via polygons and edge vias. Via polygons provide the most flexibility and are most commonly used by the majority of designers. Via polygons are also the type of via created when you import a circuit using any of Sonnet's translators. Edge vias provide a quick, convenient way to add a one cell wide via. The rest of this section applies to via polygons. Edge vias are discussed on [page 300](#).

Via polygons can be rectangles, circles or any arbitrary shape and are added to your circuit in much the same way that metal polygons are added to your circuit. You may add rectangular or circular vias or draw an arbitrary polygon by placing each vertex in its desired location.

An example of a via polygon is shown below. The shape drawn by the user appears in black and the subsections used by *em* are shown with the fill pattern assigned to the via metal. A 3D view of the via is shown on the right. The via is modeled using the Ring meshing fill so that the middle of the via is hollow. (Note that for clarity, these vias are shown with Via Pads turned off.) For more information on via meshing fill, see "Meshering Fill for Vias" on page 290. The "up" via symbol indicates that the via connects this level to the next level above. The vias

on the upper level are shown with a “down” via symbol which is a “down” triangle. Via symbols were automatically created on the destination level when the via was added to the source level.

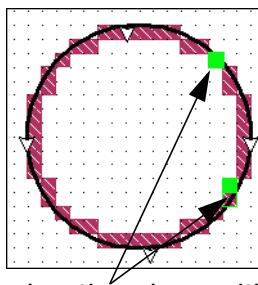
Top of Vias - “down” via symbols



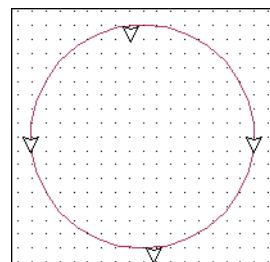
Bottom of Vias - “up” via symbols

## Via Subsections

With the metalization turned on (default setting), by setting *View*  $\Rightarrow$  *Cell Fill* to “On”, the via subsections are also displayed using the pattern assigned to the via metal type being used for the via.



Via subsections shown with the “lossless” pattern. Two of the via subsections are highlighted in green.



The via is indicated by only triangles and an outline when cell fill is off.

Each via subsection is a rectangular cylinder of current, extending between the present level to the next level above or below (depending on the direction of the via). A via subsection has a horizontal cross-sectional area equal to one cell and a

height equal to the thickness of the dielectric layer. If you change the cell size, then the via is resubsectioned into via subsections using the new cell size. The placement of via subsections is determined by the meshing fill chosen for the via.

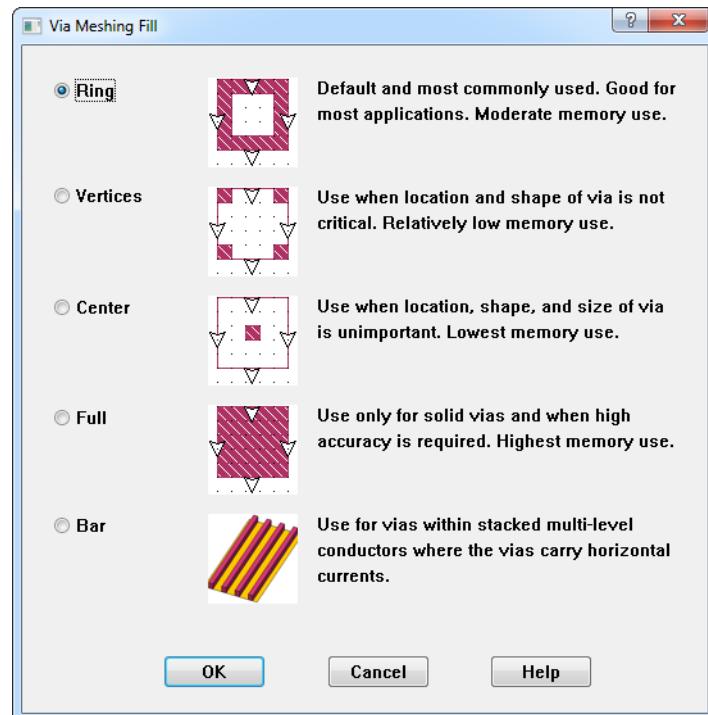
### Meshing Fill for Vias

There are five types of meshing fill for via polygons: Ring, Vertices, Center, Full and Bar. The meshing fill provides the user with a way to control the trade off between accuracy and processing time.

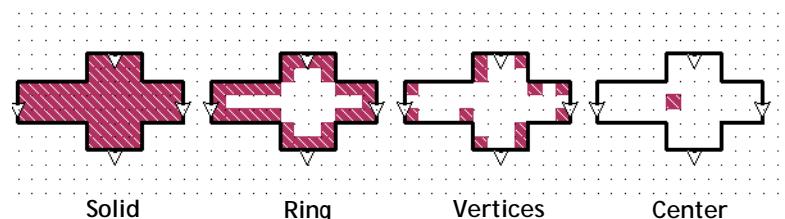
You set the Meshing Fill for a via in the Via Meshing Fill dialog box, opened by pressing the Change button in the Via Properties dialog box (*Modify*  $\Rightarrow$  *Via Properties*) or clicking on the Edit button in the Meshing Properties section of the Technology Layer dialog box, opened by double-clicking on a Via Technology Layer in the Stackup Manager.

**NOTE:**

Note that for each mesh setting, the analysis engine adjusts the loss of the individual via subsections to give the best possible overall DC and RF loss corresponding to the loss model assigned to your via. However, keep in mind that using less via subsections can change the RF path of the current, thus changing the performance at RF.



Examples of via polygons using each type of meshing fill are shown below. Bar meshing fill is not shown, since after the analysis engine simplifies bar via groups, the resulting via is modeled using the Vertices fill. For more information, see [“Simplifying Bar Vias in the Analysis Engine,” page 292](#). The shape drawn by the user appears in black. The actual via subsections are shown by the fill pattern which was assigned to the via metal being used for the via.



**Ring:** This is the default meshing fill for new vias and is the recommended meshing fill type for modeling both solid and hollow vias. The via is modeled as a one cell wide wall of via subsections and is hollow in the middle, containing no metal. This model provides a good compromise between speed and accuracy for both solid and hollow vias. This meshing fill adequately models solid vias since at RF most of the current travels along the perimeter of the via. Note that when used in conjunction with the Volume loss model, both the DC and RF affects are accurately taken into account. For more information about the Volume loss model, please see ["Volume Loss Model" on page 59](#).

**Vertices:** This meshing fill models the via by placing a single cell wide via subsection at each vertex of the polygon. Notice that if you use this model, the current may need to travel a longer path (i.e., longer than the real circuit) before it reaches the via. Therefore, you should use this setting if you are not concerned about the accuracy of the via but want a reasonable approximate model.

**Center:** A single cell wide via subsection is placed in the center of the via polygon. Notice that if you use this model, the current may need to travel a longer path (i.e., longer than the real circuit) before it reaches the via. Therefore, you should use this option when you are not concerned about accuracy of the via but just want to provide a connection from one level to another.

**Full:** This meshing fill models the via as solid metal. All subsections in the interior of the via are filled with metal extending the length of the via. This type of meshing fill is the most costly in terms of processing time and memory. This model should only be used if the via you are modeling is actually solid and you require a high level of accuracy.

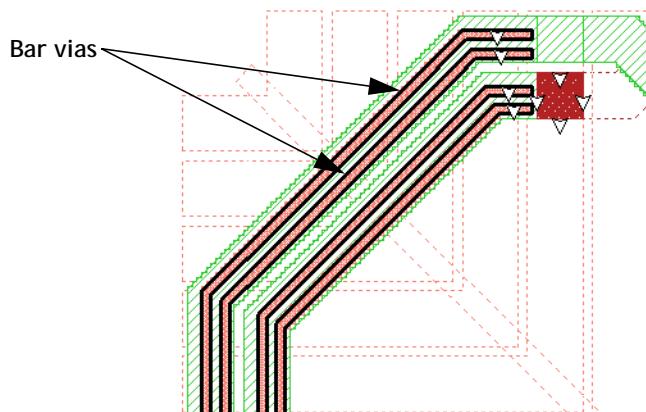
**Bar:** This meshing fill identifies the via as being a Bar via. Use this fill model for vias within stacked multi-level conductors where the vias carry horizontal currents. During the analysis, multiple adjacent bar vias are merged into one wider via to save memory and processing time. When the bar vias are grouped and simplified by the analysis engine, they are meshed using a Vertices fill.

### Simplifying Bar Vias in the Analysis Engine

Bar vias are vias whose length is significantly longer than their width. Bar vias are identified in the project editor by assigning the Bar meshing fill to a via polygon which uses the Volume loss model. This assignment may also be done automatically, based on a length to width ratio, when using a translator or interface to Sonnet. For more information, please refer to the [Via Simplification](#) chapter in any translator or interface manual.

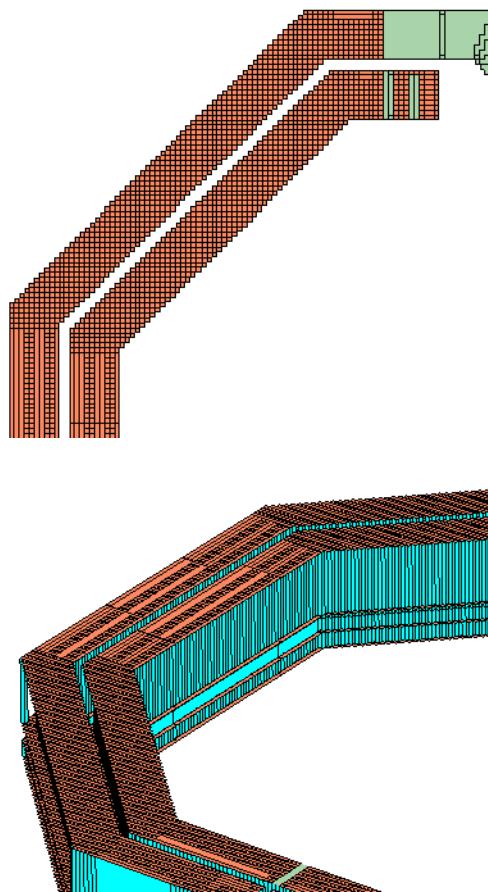
Since bar vias are only identified in the project editor, the via polygon using the Bar mesh fill is displayed as drawn by the user. The actual metal analyzed by *em* is calculated during the analysis. Bar via groups are identified and combined, then modeled using the Vertices mesh fill. An example is shown below.

The illustration below shows a section of an inductor which uses bar vias. The bar via polygons are selected and highlighted with a black outline. Note that only part of the via polygons is shown for clarity.

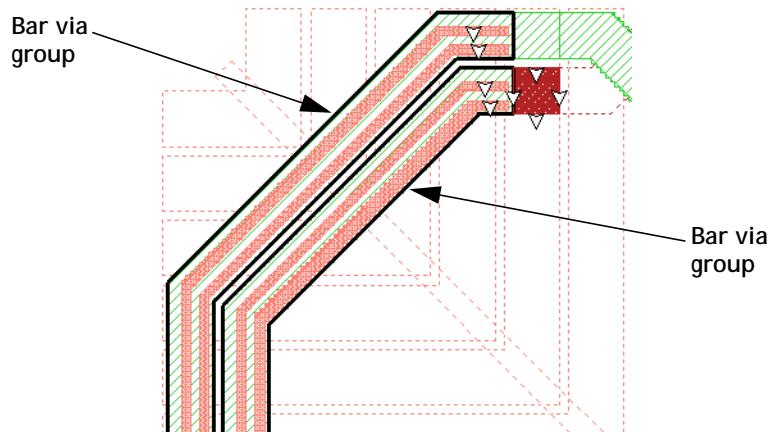


The Sonnet subsectioning automatically places more subsections in critical areas where current density is changing rapidly, but allows larger subsections in less critical areas, where current density is smooth or changing slowly. Therefore, if these vias, placed on the interior of the traces, are not identified as bar vias, the subsectioning requires a much finer resolution, which in turn dramatically increases the memory and time needed for the analysis. The 2D view and 3D view of the subsectioning of the inductor with the vias on the traces, which are not identified as bar vias, is shown below. Planar metal using staircase fill is in orange, planar metal using conformal mesh is green and via metal is blue. Note that even though

conformal mesh is applied to the traces, the placement of the vias on the traces forces a staircase fill which is much less efficient for this circuit. The estimated memory to analyze this circuit when not identifying bar vias is 55,994 MB.

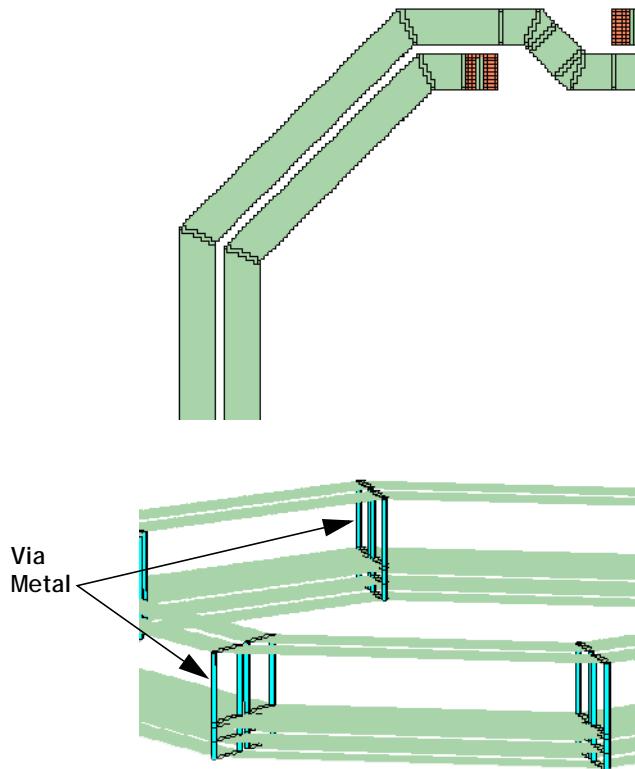


Now we will look at the effect identifying the bar vias has on subsectioning and memory requirements. When bar vias are identified, the analysis engine, *em*, simplifies bar vias to reduce the required memory and processing time. Bar via groups are first identified. In this example, the sets of two bar vias on the same trace make up a bar via group.



Bar via groups are combined into one simplified via whose edges match the edge of the trace, rather than being on the interior of the trace. This simplified via uses the Vertices meshing fill. The subsectioning of the metal now matches that of traces with no vias as pictured earlier. The 2D and 3D views of the subsectioning of the circuit once the vias have been simplified is shown below; note that via metal only appears at the vertices. In the 3D view, planar metal using staircase fill is depicted in orange, planar metal using conformal mesh is depicted in green and via metal appears in blue. As you can see, the traces can now be modeled using con-

formal mesh in addition to the savings in metal from simplifying the bar vias. The estimated memory when bar vias are identified is 794 MB, making this a much more manageable analysis.



### Adding a Rectangular Via to Ground

A via to ground can be added from any metalization level. This example shows a rectangular via; the same steps would apply to a circular or polygon via; you would just use the appropriate command for that shape of via. To add a via to ground, go to the level from which you wish the via to extend downwards and perform the following:

- 1 **Select Tools  $\Rightarrow$  Add Via  $\Rightarrow$  Down to Ground** from the project editor menu.

Any vias subsequently added to your circuit will extend from the level to which they are added down through all intervening levels to ground.

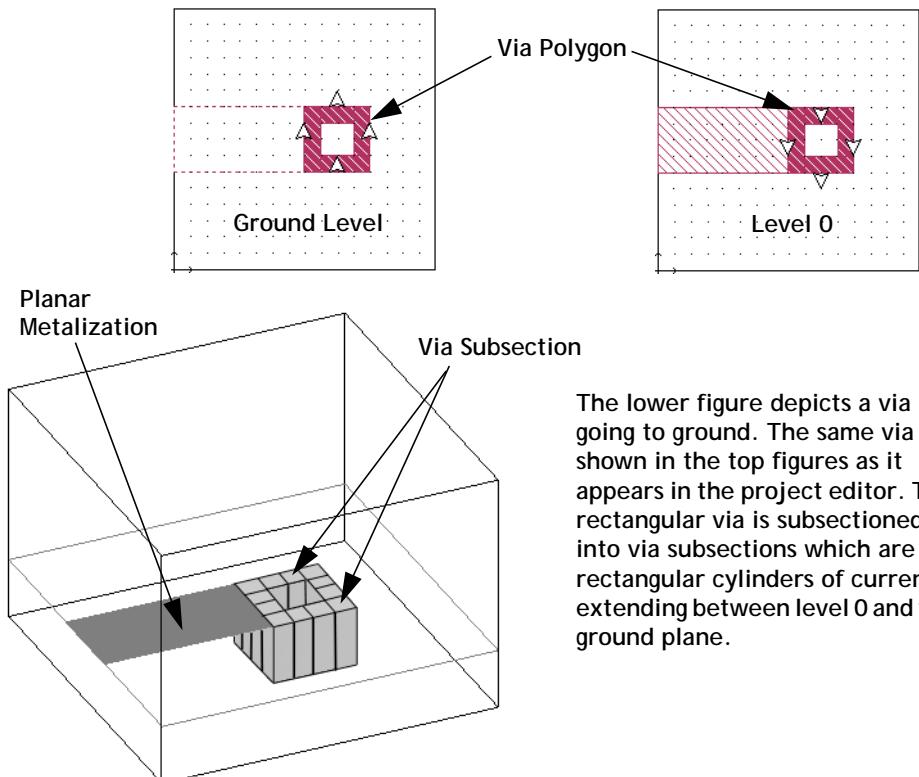
**2 Select Tools  $\Rightarrow$  Add Via  $\Rightarrow$  Draw Rectangle to add the via.**

This puts the project editor in Add via mode indicated by a change in the cursor.

**3 Click and drag to draw the desired rectangle.**

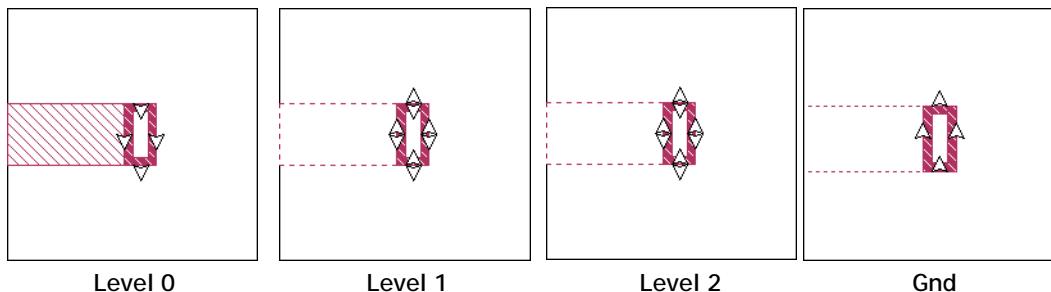
The via you input is drawn on the present level all the way down to the ground level. An example of a via polygon going from level 0 to ground.

The via polygon metalization is shown on Level 0. Note that the arrows are pointing down, indicating the direction of the via. Since this via uses the Ring meshing fill, the center of the via does not contain metalization but is filled with the dielectric of the dielectric layer. The ground level is completely metallized; however, the subsectioning of the via is displayed on the ground level to indicate that there is a via metalization in the layer above.



The lower figure depicts a via going to ground. The same via is shown in the top figures as it appears in the project editor. The rectangular via is subsectioned into via subsections which are rectangular cylinders of current, extending between level 0 and the ground plane.

If the via to ground is added when there are multiple intervening metal levels between the present level and ground, the via polygon can be seen on each level. The intervening levels have via arrows pointing in both directions to indicate that the via extends both upward and downward. Below is shown a rectangular via polygon extending from metal level 0 to ground in a three level circuit.



The via shown above extends from level 0, the highest metal level in the circuit down to the ground level. The via arrows on the rectangular via on level 0 point only in the downward direction. The via polygon appears on levels 1 and 2 in the same position but with via arrows pointing in both the upward and downward position indicating that the via extends in both directions from these levels. The via polygon is drawn on the ground plane to indicate its position with the via arrows pointing upward indicating that the via extends upward. Since the complete ground plane is metalization, the via polygon is drawn simply as a reference for the user.

## Via Loss

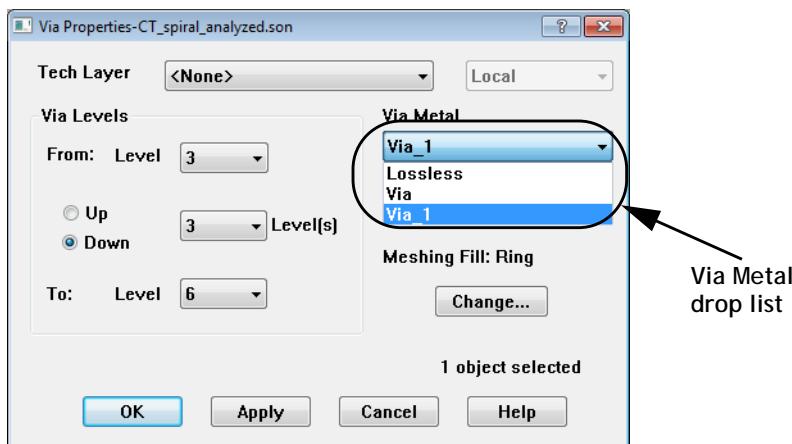
The loss of a via polygon is determined by the metal type it uses. When you draw a via polygon, its metal type is set to the default metal used for new vias or the metal type of a Via Technology Layer if it is being added to a Technology Layer. The default metal type is controlled in the Metal Types dialog box accessed by selecting *Circuit ⇒ Metal Types* from the project editor's menu. This metal type is used for new independent vias or for new Via Technology Layers.

You may also change the metal type of the via polygon or Via Technology Layer after it has been added to the circuit. You may use any via metal type defined in your circuit for a via polygon. For a complete discussion of how loss is modeled in via metal, see ["Via Metalization Loss" on page 58](#).

- 1 Right-click on the via polygon and select Properties from the pop-up menu which appears on your display.

The Via Properties dialog box appears on your display.

- 2 Select the desired metal type from the Via Metal drop list.



## Via Pads

Vias have an option to add via pads. This feature is off by default for new vias but may be turned on in the Via Properties dialog box. When this option is on, a via pad is added to the top and bottom of the via which is the same size and shape as the via polygon. The pad is modeled using a planar metal type with loss equivalent to the loss of the via metal being used for the via polygon.

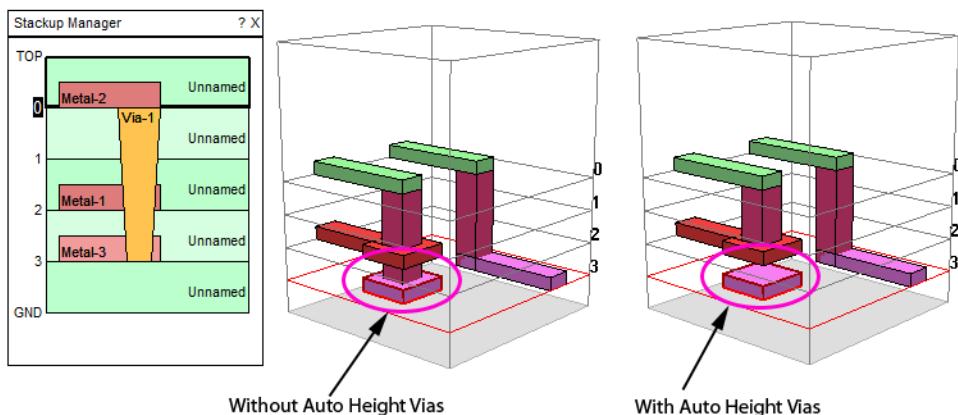
## Via Ports

For a detailed discussion of via ports, please see "Via Ports" on page 98.

## Auto Height Vias

Normally vias extend from the specified starting metal level to the specified ending metal level. If the auto height vias option is selected, then a via will terminate if its path, starting from the upper metal level and extending downward, contacts planar metal. For example, if you define a via from metal level 3 to metal level 0

and there is a planar polygon in its path on level 2, then the via is terminated on level 2 and therefore extends from level 0 to level 2. Note that the via is depicted in both the 2D and 3D view as the actual metal which will be analyzed. However, the associated Via Technology Layer still appears unchanged in the stackup since other vias associated with the Technology Layer may still extend the fully defined length as shown below.



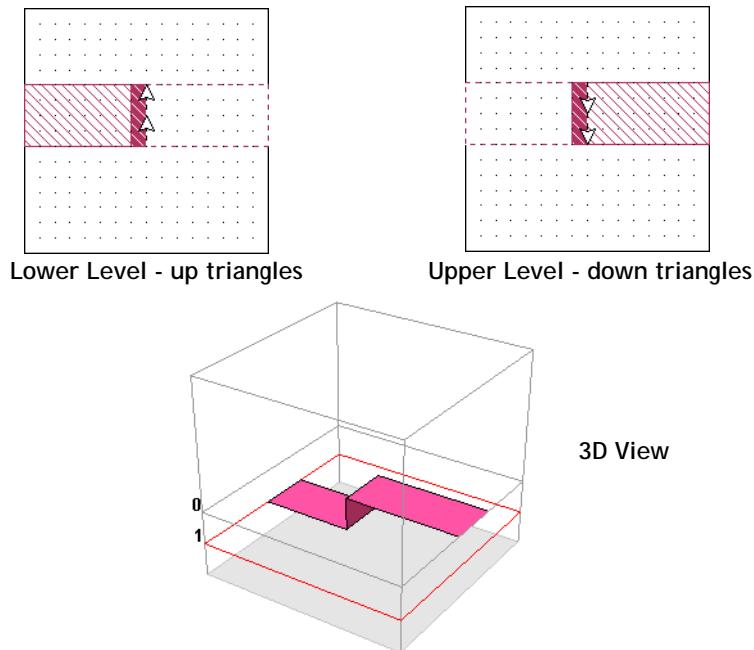
Only polygons which completely cover the cross sectional area of the via are guaranteed to affect the height of the via. Note that planar metal that is on an intervening metal level but does not intersect the path of the via does not affect the height of the via.

This option may be set in the project editor in the Advanced Subsectioning Controls dialog box (*Analysis*  $\Rightarrow$  *Advanced Subsectioning*).

## Edge Vias

Edge vias are vias that are attached to the edge of a metal polygon. As mentioned earlier in the chapter, edge vias provide a quick, convenient way to add a one cell wide via which extends for the length of the polygon edge. Edge vias are of the same metal type as the polygon to which they are attached. The actual via metal

is shown by the fill pattern which is the reverse video of the planar metal pattern. The example shown below uses an edge via to connect two polygons on adjacent levels.



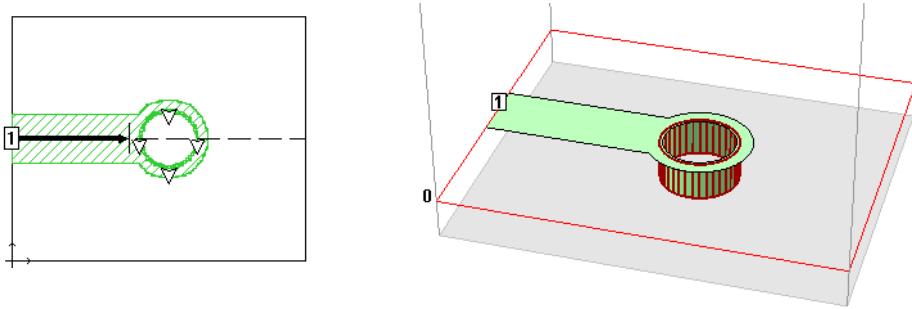
Enough via subsections are placed to cover the entire length of the polygon edge. If you change the cell size, then the via is resubsectioned into via subsections with the new cell size.

## Edge Via Loss

Loss in an edge via is modeled using the loss model of the planar metal type of the polygon to which the edge via is attached. Edge via subsections are always one cell wide, but be aware that the total DC loss of the edge via is independent of the subsection size. For example, if your edge via is modeled using one micron thick Gold, but your cell size is five microns, then the loss is based on the via being one micron thick not five microns.

## Simple Via Example

A simple via is stored in the example named “Via” and is shown below. To obtain a copy of this example, open the Example Browser by selecting *Help* ⇒ *Browse Examples* from the menu of any Sonnet application. In the Search section of the Example browser, select the “Name” radio button. Enter “Via” in the Search text entry box, then click on the Search button. The via example is displayed.



A simple via to ground. On the left, as it would appear in the project editor. On the right, a view in perspective.

# Chapter 16      Thick Metal

---

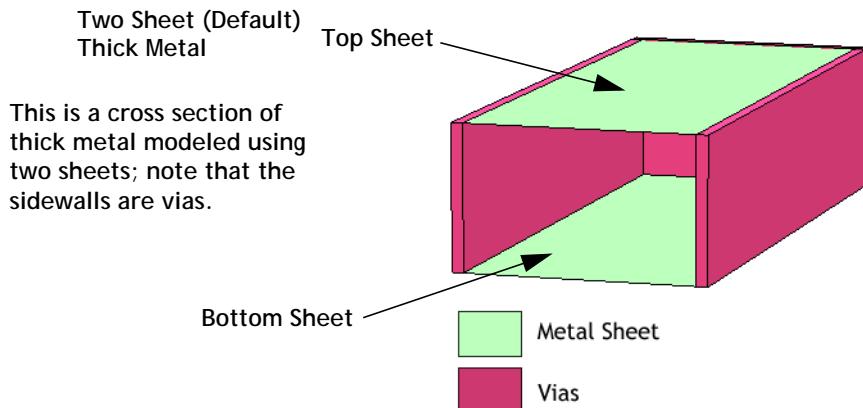
## Thick Metal Type

The Thick Metal metal type allows you to model physically thick metal. All other metal types are modeled as having zero thickness where the entered thickness value only affects the metal loss calculations. The Thick Metal type allows you to more accurately model the true 3D characteristics of thick conductors. Using a thick metal model not only allows for proper modeling of loss, but also includes the EM effects of physically thick metal, such as coupling between closely spaced conductors.

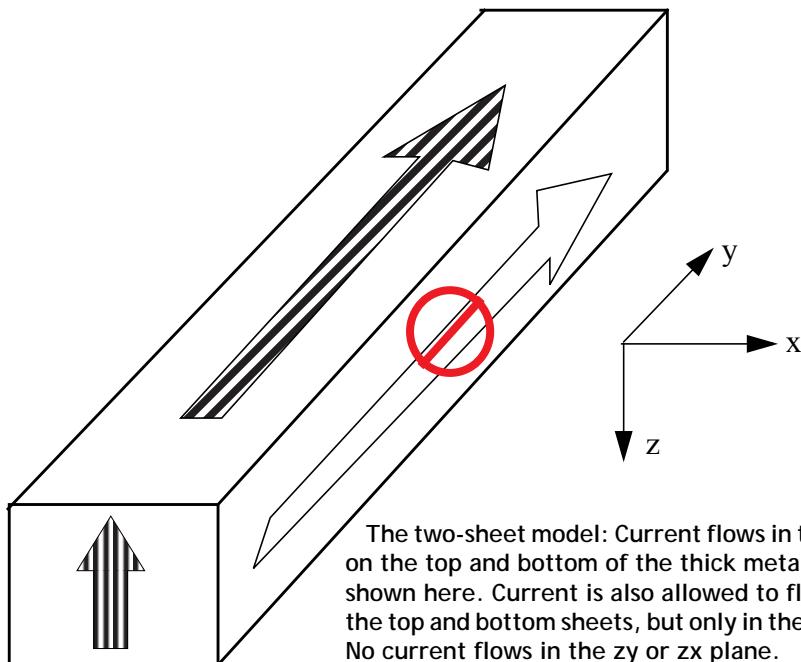
Since thick metal increases both your processing time and memory requirements, it should only be used when necessary. Metal is considered to be thick metal when its thickness is comparable to other dimensions in the circuit such as the width of a conductor or gap between conductors.

When using thick metal, the structure is approximated by two or more infinitely thin sheets of metal. The Number Of Sheets setting defines how many sheets of horizontal metal are in the vertical stack that models the thick metal. For the typical two sheet model, one sheet represents the top surface of the structure and a

second sheet represents the bottom surface of the structure. Vias are placed automatically around the perimeter to allow current to flow between the sheets. An example using two sheets is illustrated below.

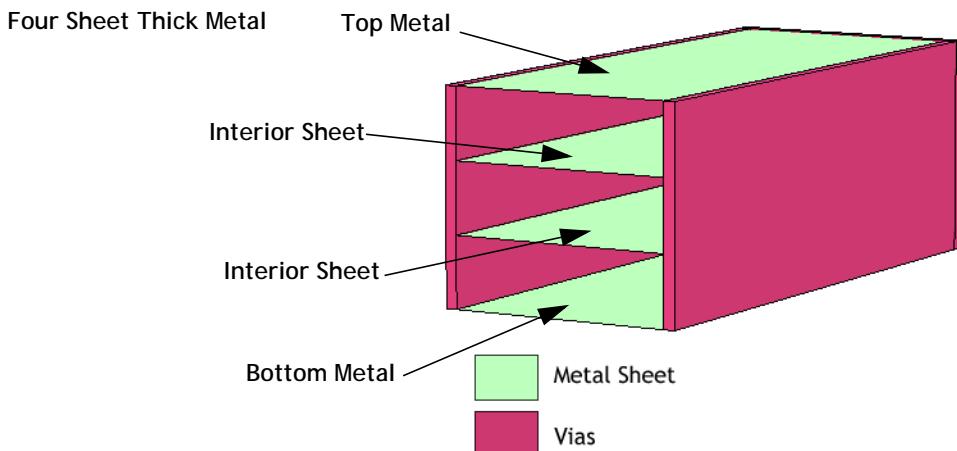


For Number Of Sheets set to two sheets, the current travels on only the top and bottom surface of the thick metal. Current on the sides of the thick conductor can be approximated by using three or more sheets.



For most cases, using the default of two sheets provides a high accuracy solution. However for very tightly coupled lines, where the gap between the lines is much less than the metal thickness, the coupling between them may be underestimated. In these cases, you may need to increase the number of sheets. However, increasing the number of sheets increases the memory requirements and processing time.

Increasing the number of sheets adds more layers of infinitely thin metal between the top and bottom metal sheet. A cross section of a four sheet model is shown below.



This is a cross section of thick metal modeled with four sheets; note that the sidewalls are vias.

## Creating a Thick Metal Polygon

To create a thick metal polygon in your circuit, you must first define a metal type using the Thick Metal model, then apply that metal type to the polygon in your circuit. This creates a thick metal polygon which extends upwards (toward the box top) from the metal level on which it is created. To do this, perform the following:

- 1 In the project editor, select *Circuit* ⇒ *Metal Types* from the main menu.**

The Metal Types dialog box appears on your display.

- 2 Click on the Add Planar button in the Metal Types dialog box.**

The Planar Metal Editor dialog box appears on your display.

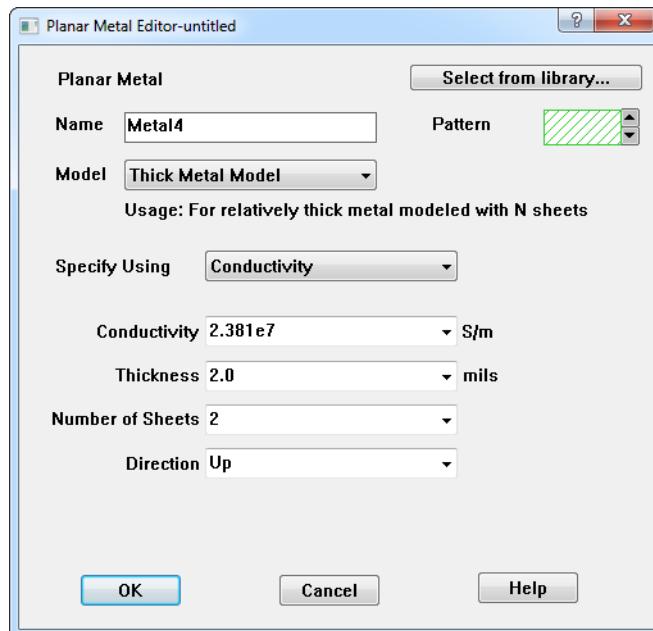
**3 Select Thick Metal Model from the type drop list in the Metal Editor dialog box.**

This updates the dialog box with the text entry boxes for the four parameters needed for the thick metal model: Thickness, Number of Sheets, Direction and a fourth parameter which is set by selecting either Conductivity, Resistivity, or Sheet Resistance at DC from the Specify Using drop list. The default is Conductivity. The example below uses the Sheet Resistance at DC. Note that this is the overall Sheet Resistance of the metal, and is not related to the number of sheets being used. For more information on Conductivity, Resistivity, and Sheet Resistance at DC, please see “Conductivity, Resistivity, or Sheet Resistance at DC,” page 51.

The direction parameter defines whether the metal extends upward into the dielectric layer above (the default setting) or extends downward into the dielectric layer below.

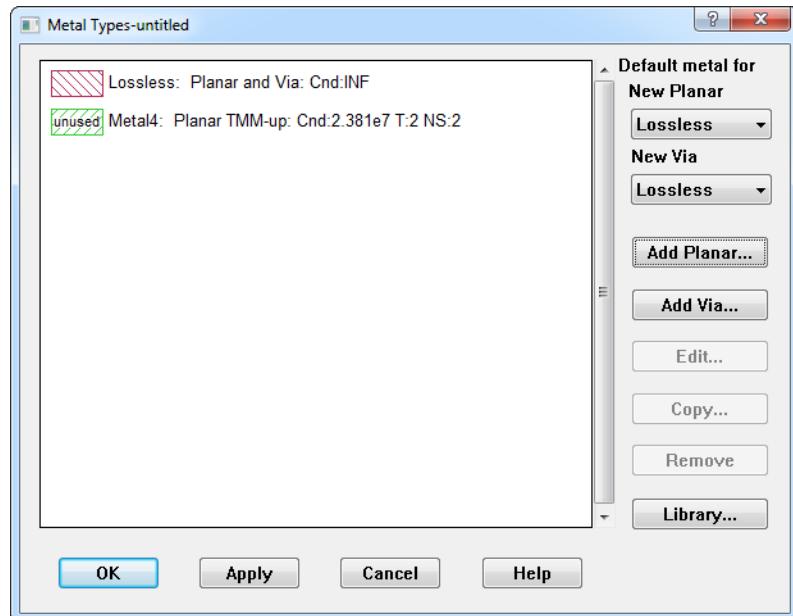
- 4 Enter the four parameters in the appropriate text entry boxes.**
- 5 If you do not wish to use the default metal name, enter the desired name for the metal type in the Name text entry box.**

The Metal Editor dialog box should appear similar to the picture below.



- Click on the OK button to close the Planar Metal Editor dialog box and apply the changes.

The Metal Types dialog box is updated with the new metal type.



- Click on the OK button to close the Metal Types dialog box.

The thick metal is now available to use in your project.

- Enter the desired polygon, then double-click on the polygon to open the Metalization Properties dialog box.
- Select the thick metal model metal type from the Metal drop list in the Metalization Properties dialog box.

This will apply the metal type which uses thick metal to the selected polygon. In this case, since the Direction selected was “Up,” the thick metal extends upwards from the level on which the polygon was drawn. If you had selected “Down” for the direction, the metal would have extended downward from the metal level.

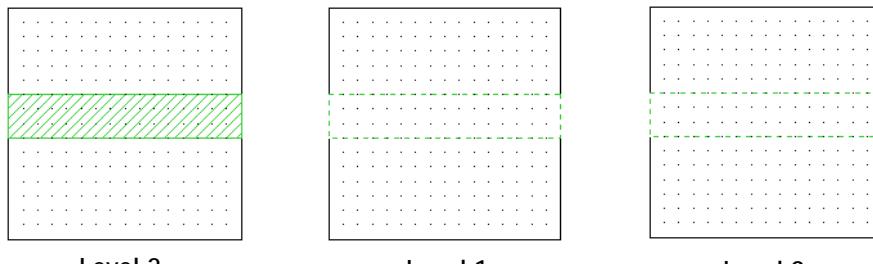
- Click on the OK button to close the Metalization Properties dialog box and apply the changes.

The fill pattern of the polygon changes to the fill pattern used by the thick metal. If the thick metal polygon is thicker than the dielectric layer(s) above it, the polygon also appears on metal levels above.

You may also assign a thick metal model metal type to a Metal Technology Layer.

## Viewing Thick Metal in the Project Editor

The thick metal can extend upward through the dielectric layer from the level on which the polygon is drawn or downward into the dielectric layer below. If the thick metal is not as thick as the dielectric layer, then the polygon only appears on the level where it was drawn. If the thick metal is the same thickness or thicker than the dielectric layer it appears on both the metal level where it was drawn and on the next metal level. Note that the top of the thick metal may not appear in the project editor if it is embedded in a dielectric layer. You will be able to view the top sheet of metal in the current density viewer which is discussed later in the chapter. Examples of both instances are shown below.

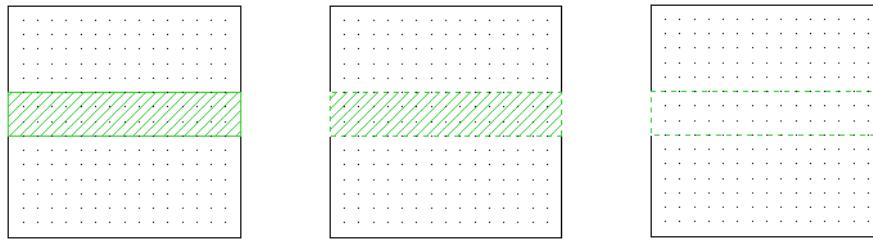


Level 2

Level 1

Level 0

A 3 mil thick metal polygon is drawn on level 2 below a 5 mil thick dielectric layer. The polygon is visible on level 2 where it was drawn, but only the outline is visible on level 1 above since the thick metal does not pierce the dielectric.



Level 2

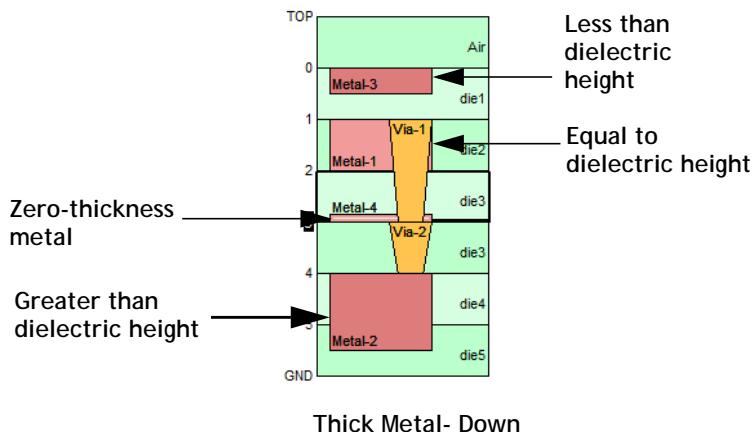
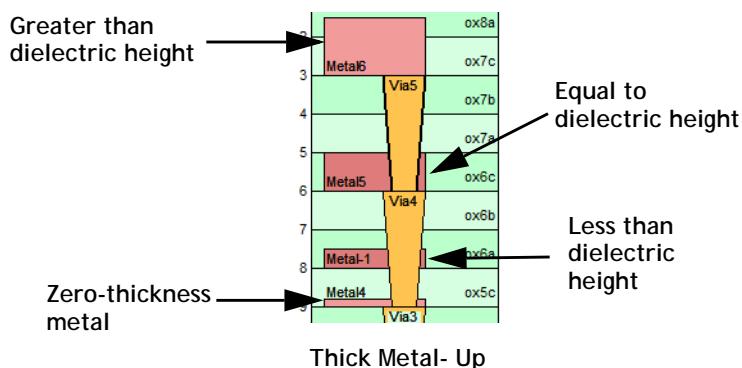
Level 1

Level 0

A 5 mil thick metal polygon is drawn on level 2 below a 5 mil thick dielectric layer. The polygon is visible on level 2 where it was drawn, and also on level 1 above since it is the same thickness as the dielectric layer. Note that on level 1, the border of the polygon is drawn with a dashed line to indicate that the origin of this polygon is not on this level.

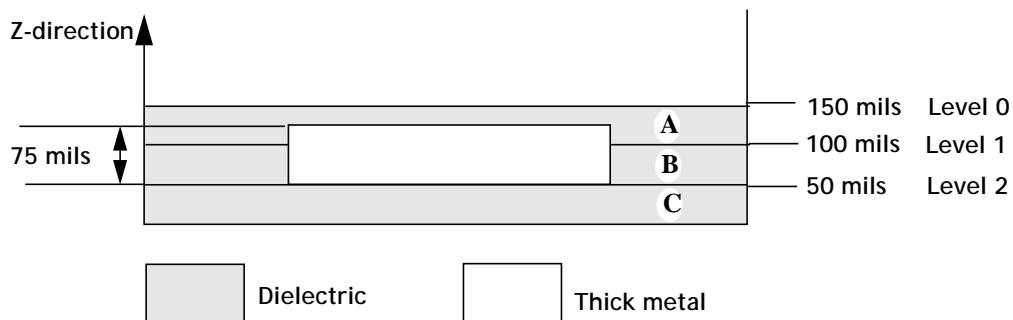
## Stackup Manager

If a Metal Technology Layer is assigned a thick metal, then how it appears in the stackup manager is based on its relationship to the dielectric layers. The thick metal height is not drawn to scale in the stackup manager, but if its thickness is less than that of the dielectric into which it extends, then the symbol covers half of the thickness of the dielectric layer. If it is the same thickness as the dielectric layer then the symbol is drawn the same height as the dielectric. If it is thicker than the dielectric then it also extends half of the height of the next dielectric layer. Examples of thick metal which extends upward and downward are shown below. Note that all dielectric layers are represented as the same height in the stackup manager regardless of their actual physical thickness.



## Modeling

A side view of a circuit with three 50 mil dielectric layers (A, B, and C) and a 75 mil thick metal polygon on level 2 is shown below. Note that the top of the thick metal only extends halfway through dielectric layer. The top of this thick metal is not visible in the project editor.



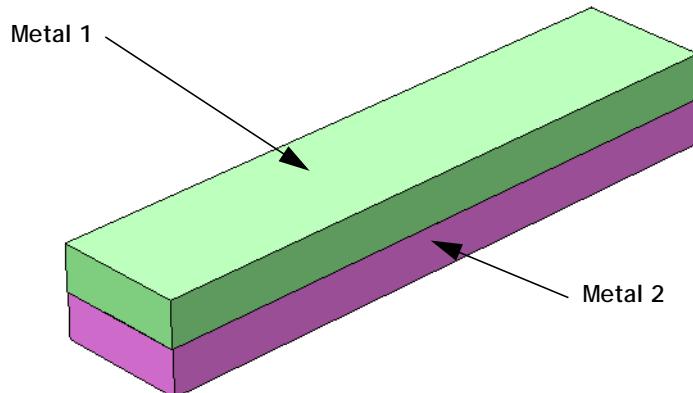
Another important thing to note about the modeling of a thick metal which ends in the interior of a dielectric layer is that even though you model the thick metal with 2 sheets, the software actually uses three sheets. The bottom sheet is on level 2 where the polygon originates. The top sheet is in the interior of dielectric layer A. The third sheet is on metal level 1. Whenever a thick metal polygon traverses a metal level, a sheet is added on that level. This adds additional computational time and should be kept in mind when using thick metals which encompass more than one dielectric layer.

If Single Level select is enabled, then you may only select the thick metal polygon on the level where it was drawn.

Thick metal polygons are connected to thin metal polygons by drawing the thin metal polygon on the same level on which the thick metal was drawn and placing the thin metal polygon adjacent to or overlapping the thick metal. This connects the two structures electrically.

## Modeling Plated Metal

Plated metal can be modeled in Sonnet by stacking two or more thick metal conductors directly on top of each other as illustrated below with a stack of two different metals.



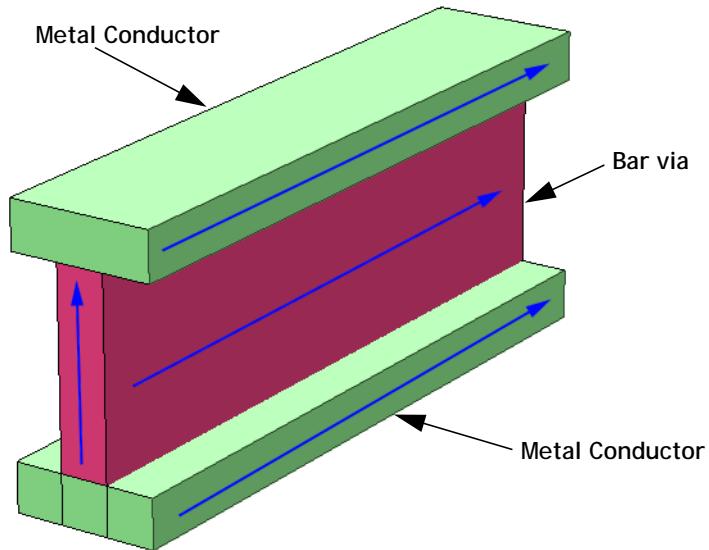
Independent properties can be defined for each individual layer of metal in the stackup. The analysis engine, *em*, will precisely account for the unique properties and thickness of each metal layer as it computes the loss of the combined stackup of metals.

## Modeling Conductor-Via-Conductor Stackups

Metal with conductor-via-conductor cross-sections, as is typical in RFIC applications, can be modeled in Sonnet by stacking thick metal conductors with vias connected between them. There are two general configurations for these stackups, characterized by the geometry of the vias: the bar via configuration and the via array configuration.

## Bar Via Configuration

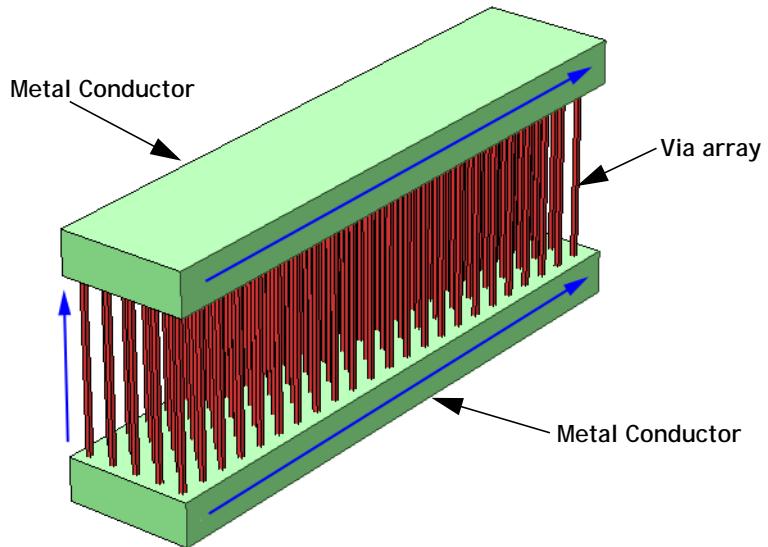
With the bar via configuration, solid vias connect the thick metal conductors as illustrated below. Because the vias are solid, these vias carry horizontal sidewall currents in parallel to the currents carried by the thick metal conductors. These sidewall currents have the effect of decreasing the loss of the overall metal stack-up, and are accurately accounted for by the Sonnet analysis engine, *em*.



For more information on how bar vias are modeled in Sonnet, please see ["Simplifying Bar Vias in the Analysis Engine" on page 292](#).

## Via Array Configuration

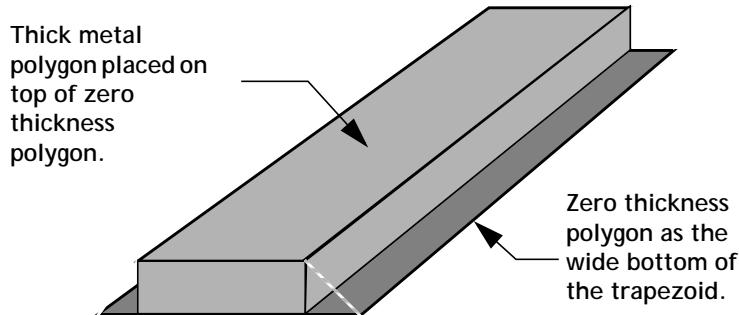
In the case of the via array configuration, there is physical separation between each of the individual vias making up the array. Thus, there is no horizontal side-wall current on these vias. Only the thick metal conductors carry horizontal current. This condition is accurately accounted for by the Sonnet analysis engine, **em**, as it computes the loss of stackups with this type of cross-section.



For more information on how via arrays are modeled in Sonnet, please see [Appendix I “Via Simplification”](#) in any translator or interface manual.

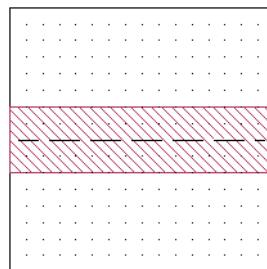
## Modeling an Arbitrary Cross-Section

In this section, we use a combination of thick metal and Normal (zero thickness) metal to approximate thick metal lines where the vertical cross-section has an arbitrary geometry. To demonstrate this capability, we use a simple trapezoidal geometry, the cross section shown in the figure below.



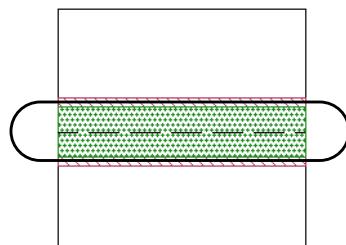
A trapezoidal cross-section transmission line viewed in perspective. If the line has no current going around the edge, it can be modeled, as shown, as two infinitely thin sheets of current, one at the top and the other at the bottom of the actual metal.

To create the thick metal trapezoid, set up the dielectrics so that there is one layer of dielectric with the same thickness as the thick metal. Then, place a polygon representing the wider bottom side of the thick metal on the bottom side of that dielectric layer. This polygon should use the Normal model for the metal type, which is modeled as a zero-thickness metal.

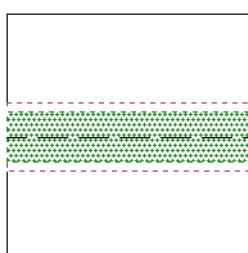


The wide bottom of the trapezoidal line is made up of a polygon using the Normal model for the metal type. This is a zero-thickness metal.

Then place a polygon representing the top side of the thick metal on the bottom side of that dielectric layer using the Thick Metal metal type. Make this polygon as thick as the dielectric layer.



Thick metal polygon on level 1 where it is drawn and placed on top of the wider zero-thickness polygon



The same polygon shown on level 0. Since the thick polygon is the same thickness as the dielectric layer, the metal also appears on this level. Only the outline of the zero-thickness metal is shown on this level.

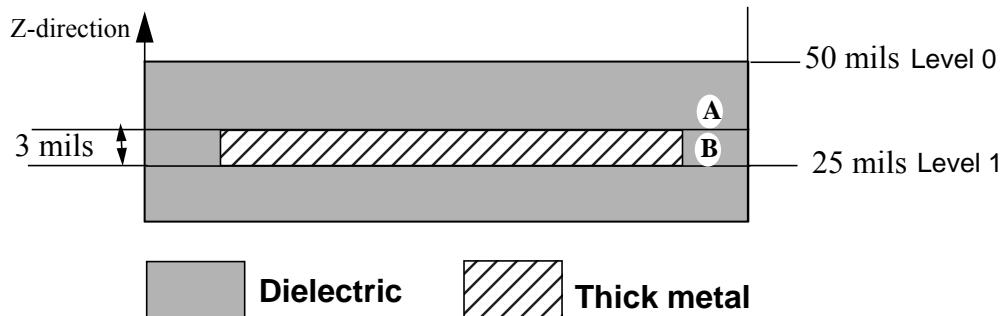
Next, place any desired ports on the thick metal polygon, not on the thin metal polygon. Since the thick metal polygon is placed on top of the zero-thickness polygon, the two are connected electrically and the port is across both polygons.

## Thick Metal in the Current Density Viewer

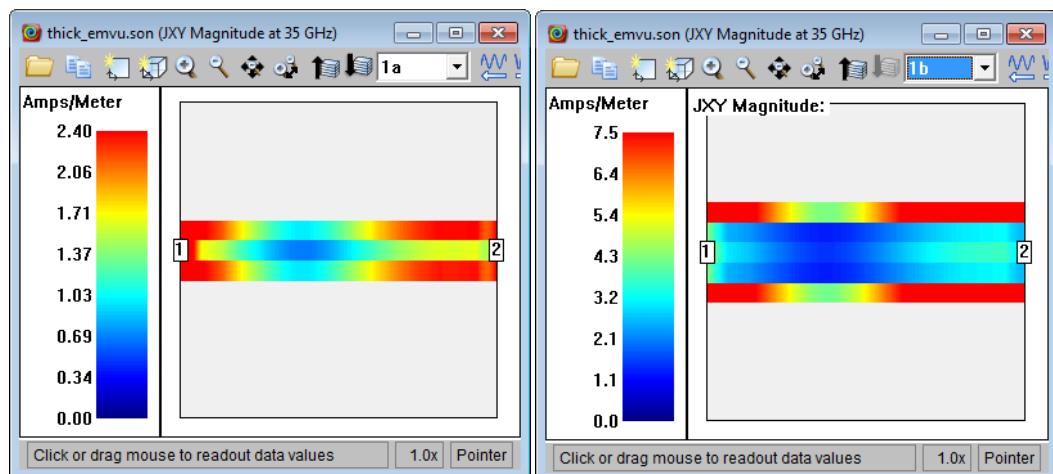
The current density viewer allows you to view the current density distribution in your circuit. When you select the Compute Current Density option in the Analysis Setup dialog box in the project editor, **em** calculates current density data for all the metal levels in your circuit. When you have thick metal in your circuit which ends in the interior of a dielectric layer, then the current density viewer creates “sublevels” of metal in order to display all the current density data.

For instance, you have a circuit with 3 mil thick metal using the default 2 sheets placed on metal level #1 below a 25 mil dielectric layer as pictured below. The top of the thick metal structure is placed in the interior of the dielectric layer. The cur-

rent density viewer displays levels 1b and 1a, where 1b is the metal level on which the thick metal was drawn and 1a is the top of the thick metal structure embedded in the dielectric layer.



Below are shown the views of level 1a and 1b in the current density viewer. Note that 1a is the top of the thick metal structure and is not visible in the project editor. 1b is the bottom where the polygon was drawn and is visible in the project editor.



The current density viewer creates as many “sublevels” as are needed. A thick metal which is defined as having 4 sheets placed on level 2 would appear in the current density viewer as 2a, 2b, 2c and 2d with 2a being the top of the thick metal structure and 2d being the bottom drawn on level 2.

# Chapter 17      Dielectric Bricks

---

Although **em** is primarily a planar electromagnetic simulator, it also has the capability to add “dielectric brick” material anywhere in your circuit. A dielectric brick is a solid volume of dielectric material embedded within a circuit layer. See the illustration below. Dielectric bricks can be made from any dielectric material (including air) and can be placed in circuit layers made from any other dielectric material (including air). For example, dielectric bricks can be used to simulate structures such as an embedded capacitor in an “air” circuit layer, or an “air hole” in a dielectric substrate circuit layer.

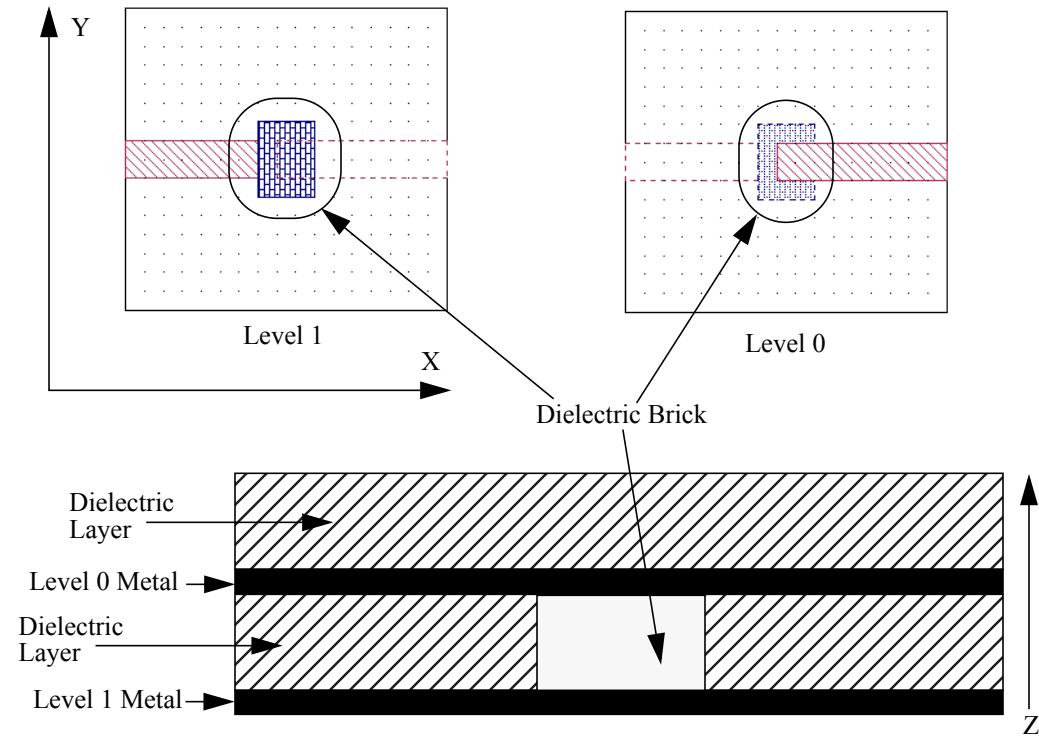


## WARNING

---

Misuse of dielectric bricks can lead to significantly inaccurate results. It is highly recommended that you read this entire chapter before attempting to use dielectric bricks.

---



Side View of Circuit shown above.

All realizable values for the dielectric constant, loss tangent and bulk conductivity or resistivity can be used. Furthermore, it is possible to set these parameters independently in each dimension to create anisotropic dielectric bricks.

**Em** is appropriate for simple structures using very localized dielectric bricks; however, when your design requires large areas of brick material, you may need a full 3-D electromagnetic analysis tool.

You should also be aware that the use of dielectric bricks can dramatically increase the memory requirements, and thus the simulation time, for your circuit. Bricks should only be used where strictly necessary for the accuracy of your simulation.

Care should be taken when using dielectric bricks, since improper modeling of your dielectric brick can yield highly inaccurate data. We recommend that you run a convergence test by doubling (or halving) the number of your Z-partitions, re-

analyzing your circuit and comparing the two results to ensure that you are using a sufficient number of Z-partitions. For more information on Z-partitioning, see “Z-Partitioning,” page 325.

## Applications of Dielectric Bricks

The use of dielectric bricks is appropriate for applications where the effects of dielectric discontinuities or anisotropic dielectric materials are important. Examples of such applications include dielectric resonators, dielectric overlays, airbridges, microstrip-to-stripline transitions, dielectric bridges and crossovers, microslab transmission lines, capacitors and module walls.

## Guidelines for Using Dielectric Bricks

### Subsectioning Dielectric Bricks

A dielectric brick simulates a volume of dielectric material. Because a brick simulates a volume, it must be subsectioned in the X, Y and Z dimensions. The more subsections (finer resolution) used in each dimension, the more accurate the analysis.

X/Y subsectioning of dielectric bricks is identical to X/Y subsectioning of metal polygons. You can control the X/Y subsectioning of both through your choice of grid size, XMIN, YMIN, XMAX, YMAX and subsections-per-wavelength. See Chapter 3, “Subsectioning,” for details.

Z subsectioning of dielectric bricks is controlled by the “number of Z-partitions” parameter. This parameter specifies the number of Z subsections for all dielectric bricks on a particular dielectric layer. See the “Z Partitions dialog box” topic in the project editor’s Help for information on setting this parameter.

### Using Vias Inside a Dielectric Brick

Vias through dielectric bricks are treated the same as vias through the standard dielectric layers. Note that via ports inside dielectric bricks are not allowed.

## Air Dielectric Bricks

Dielectric bricks can be made of any dielectric material and can be placed in any circuit layer. This allows, for instance, “alumina” bricks to be created in an “air” circuit layer. However, it is also possible to reverse this scenario. Dielectric bricks made of “air” can also be created in alumina circuit layers. This is an important consideration to remember. Depending upon the circuit geometry for a given application, this ability to reverse the dielectric characteristics may simplify the circuit and make it faster to analyze.

## Limitations of Dielectric Bricks

### Diagonal Fill

Diagonal fill is not allowed for dielectric bricks. All dielectric bricks must use “staircase fill”. Thus, dielectric bricks with curved or rounded edges must be stair step approximated. Note that the error caused by such an approximation decreases as the X and Y cell sizes are decreased. Thus, it is possible to make this error arbitrarily small by choosing sufficiently small X and Y cell sizes.

### Antennas and Radiation

The far field viewer does not support dielectric bricks. Circuits containing dielectric bricks can be analyzed with the far field viewer, but the radiation effects of the dielectric bricks are not accounted for in the analysis.

### Interfaces

The Keysight ADS Interface, Cadence Virtuoso Interface and NI AWR MWOffice Interface do not create dielectric bricks.

# Dielectric Brick Concepts

## Creating a Dielectric Brick

To create a dielectric brick in the project editor, do the following:

- 1 Move to the circuit level where the base of the dielectric brick is to be located.**

The dielectric brick that is created will rest on this circuit level, and will extend upward to the next level. Dielectric bricks can be placed on any level, including the ground plane. If a brick is placed on the highest circuit level (level 0), it will extend up to the top cover of the metal box.

- 2 Create a base polygon which defines the cross-section of the brick.**

This is done by selecting either *Tools*  $\Rightarrow$  *Add Dielectric Brick*  $\Rightarrow$  *Draw Rectangle* or *Tools*  $\Rightarrow$  *Add Dielectric Brick*  $\Rightarrow$  *Draw Polygon* from the project editor's main menu. The first option allows the vertices of arbitrarily shaped base polygons to be entered on a point by point basis. This option is used to create dielectric bricks with any cross-sectional shape. However, if the cross-section is rectangular in shape, it is often quicker to create dielectric bricks using the second option.

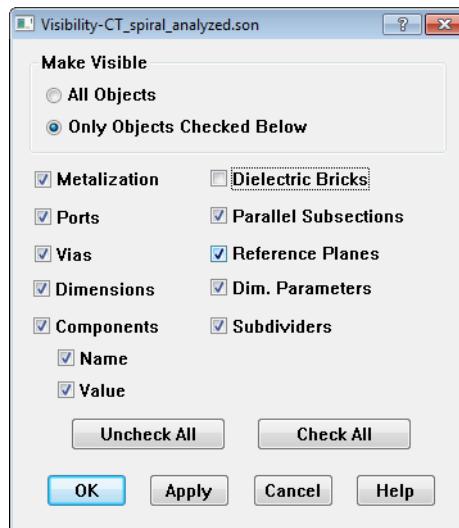
## Viewing Dielectric Bricks

Once a dielectric brick has been created in the project editor, it is possible to "see" the brick from both the circuit layer where the base of the brick is located and the circuit layer where the top of the brick is located. On both levels, you will see a polygon which defines the cross-sectional shape of the dielectric brick. The brightness of the polygon, however, will vary. When you are on the top level, you will see a "dim" polygon; on the base level you will see a "bright" polygon.

Only the outline of the dielectric brick is visible from levels other than the origination and termination of the dielectric brick.

Note that while it is possible to "see" a brick from two different circuit levels, "selecting" a brick, for cutting, copying, moving, changing attributes, etc., can only be done from the circuit level where the base of the brick is located if you are in Single layer edit mode. The polygon can be selected on either level if you are in multilayer select.

Finally, it is possible to turn the display of dielectric bricks “on” or “off” in the project editor. You select *View*  $\Rightarrow$  *Object Visibility* from the main menu of the project editor which opens the Object Visibility dialog box shown below.



Click on the *Only Objects Checked Below* radio button to enable the object choices, then click on the Dielectric Bricks checkbox to turn off the display of the bricks.

This will make any bricks present in the circuit invisible and unselectable, but does not remove them from the circuit. The dielectric bricks can be turned back “on” by once again selecting *View*  $\Rightarrow$  *Object Visibility* and clicking the *Dielectric Bricks* checkbox or the *All Objects* radio button.

Occasionally, when a circuit contains many layers, with overlapping metal polygons and dielectric bricks, it may be somewhat difficult to distinguish the metal polygons and dielectric bricks from one another. The ability to turn dielectric bricks “off” usually makes it easier to view such circuits.

## Defining Dielectric Brick Materials

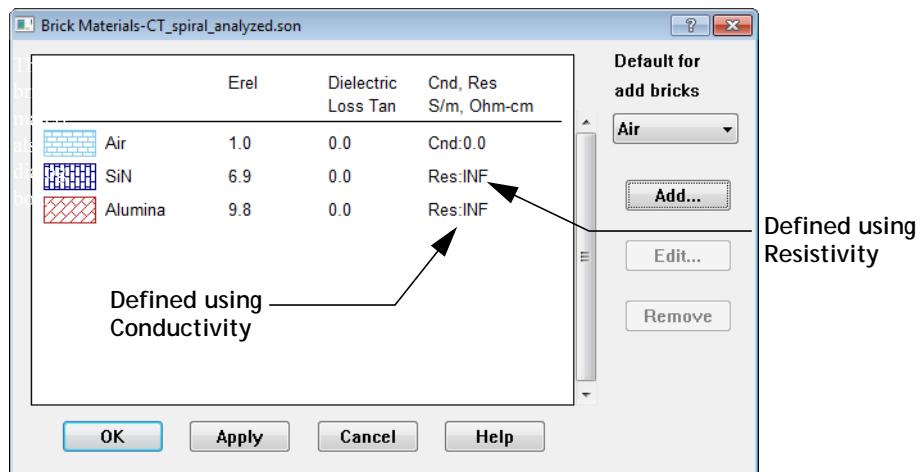
Just as it is possible to define a variety of metal types, each with different properties, it is also possible to define a variety of dielectric brick materials. Dielectric brick materials are defined using the parameters below:

- **E<sub>rel</sub>:** The relative dielectric constant ( $\epsilon_r$ ). The ratio ( $\epsilon/\epsilon_0$ ), where  $\epsilon$  is the real part of the permittivity of the dielectric layer material, and  $\epsilon_0$  is the permittivity of free space. The ratio is dimensionless.

- **Dielectric Loss Tan:** The dielectric loss tangent. The ratio ( $\epsilon''/\epsilon'$ ), where  $\epsilon = \epsilon' - j\epsilon''$ , and  $\epsilon$  is the complex permittivity of the dielectric layer material. The ratio is dimensionless.
- **Conductivity:** Dielectric conductivity,  $\sigma$ , where  $\sigma$  is the conductivity of the dielectric. The default unit is S/m. You may define your dielectric loss using conductivity or resistivity. To use conductivity, you should select “Conductivity” from the Specify Using drop list.
- **Resistivity:** Resistivity = 1.0 / conductivity. Resistivity has a default unit of Ohm-cm. You may define your dielectric loss using conductivity or resistivity. To use resistivity, you should select “Resistivity” from the Specify Using drop list.

To define a new dielectric brick material, or to modify the characteristics of an existing material, you use the Brick Materials dialog box, which is opened by selecting *Circuit*  $\Rightarrow$  *Brick Materials* from the main menu of the project editor.

The Brick Materials dialog box, shown on page 323, shows all the dielectric brick materials previously defined, the color/fill pattern assigned to each brick material, and whether the material is isotropic or anisotropic. To modify the settings for a particular dielectric brick material, edit that material's text entry boxes.



If the brick type is isotropic only one set of parameters, X, will be set. Conversely, if the brick material is set to anisotropic, each parameter is defined separately for the X, Y, and Z dimensions. If you wish to make a brick material anisotropic, click on the *Ani* checkbox.

Note that Electrical Loss may be defined using Conductivity or Resistivity. Which parameter is used is indicated by the prefix appearing before the value Conductivity/Resistivity column as pictured above.

The “default” material used when new dielectric bricks are created can also be set in the Brick Materials dialog box. Select a brick type from the *Default for add bricks* drop list. Once the default material has been set, all bricks created thereafter will be made of that material.

## Changing Brick Materials

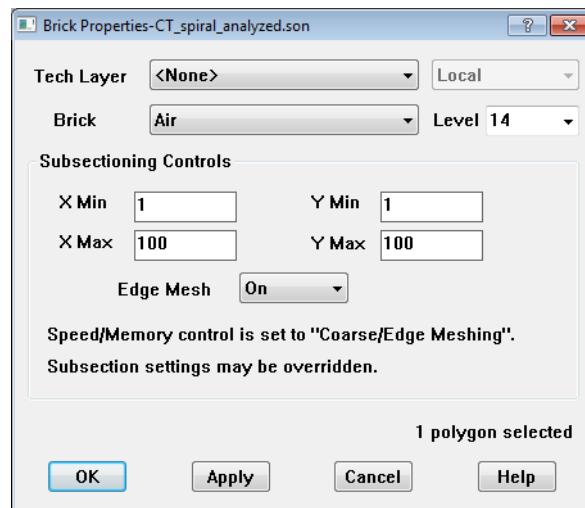
The material type for bricks that already exist in a circuit can be changed by following the procedure given below:

- 1 **Select the brick(s) by clicking on it or lassoing it.**

The brick is highlighted.

- 2 **Select *Modify* ⇒ *Brick Materials* from the main menu of the project editor.**

This will open the Dielectric Brick attributes dialog box, shown below.



- 3 **Select the brick material you desire from the drop list labeled Brick.**

This drop list contains all the types defined for dielectric bricks including the default type, air.

- 4 Click on the OK button to apply your selection and close the dialog box.

## Z-Partitioning

A dielectric brick simulates a volume of dielectric material. Because a brick simulates a volume, it must be subsectioned in the X, Y, and Z dimensions. The more subsections (finer resolution) used in each dimension, the more accurate the analysis.

X/Y subsectioning of dielectric bricks is identical to X/Y subsectioning of metal polygons. You can control the X/Y subsectioning of both through your choice of grid size, XMIN, YMIN, XMAX, YMAX, and subsections-per-lambda.

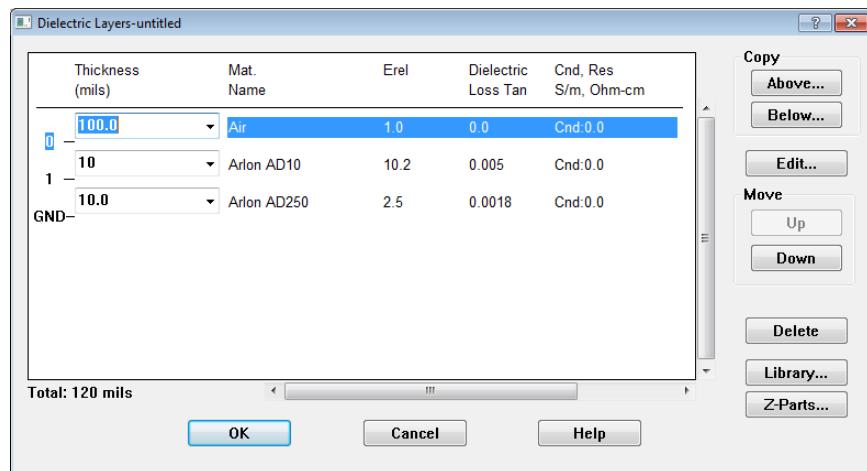
Z subsectioning of dielectric bricks is controlled by the Z Partitions dialog box which is opened when you click on the Z Parts button in the Dielectric Layers dialog box (*Circuit*  $\Rightarrow$  *Dielectric Layers*). You may enter a Z Parts value for each dielectric layer in your circuit. This parameter specifies the number of Z partitions for all dielectric bricks on a particular circuit layer.

The default for this parameter is zero so that you are forced to enter a value for this field. If you use a dielectric brick in a layer, and do not set the z-partitions, *em* reports an error and exits the analysis. You must enter a non-zero integer value for this parameter in order to run an analysis. The value of this parameter is highly dependent on your circuit design; therefore, Sonnet cannot determine a “reasonable” value. This is the reason we suggest you run a convergence test, discussed earlier in the chapter, on your circuit to determine the best value for the Z-partitioning.

To set this parameter in the project editor, do the following:

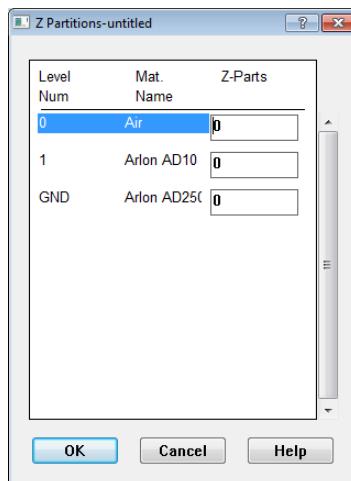
**1 Select *Circuit*  $\Rightarrow$  *Dielectric Layers* from the project editor main menu.**

The Dielectric Layers dialog box, shown below, is displayed.



**2 Click on the Z-Parts button in the Dielectric Layers dialog box.**

The Z Partitions dialog box appears on your display.



**3 Enter the number of z partitions to be used for each dielectric layer in the appropriate Z Parts text entry box.**

Note that changing this value for a particular layer will have absolutely no affect on the analysis if there are no bricks on the layer. If there are multiple bricks on the layer, the Z subsectioning for all of those bricks will be identical. It is not possible to apply different Z partitions to brick polygons which appear on the same layer.

# Chapter 18

# Antennas and Radiation

---

To this point, this manual has been focused on using Sonnet for the analysis of high frequency circuits and transmission structures. However, there is a large class of radiating structures for which Sonnet has proven very useful. This chapter describes how to use Sonnet to analyze 3-D planar radiating structures, such as microstrip patch arrays and microstrip discontinuities, using the “Open Waveguide Simulator” technique. The underlying assumptions of this technique are described in detail. Common modeling mistakes are also pointed out. Examples are provided to illustrate the correct use of the modeling technique.

This chapter also discusses the far field viewer, an analysis and viewing tool which calculates far field antenna patterns for arbitrary 3-D planar geometries. The far field viewer uses the current distribution data in the project as input, and creates a pattern. The pattern may be viewed as a cartesian, polar or surface plot.

---

**NOTE:**

The far field viewer is only available if you have purchased a far field viewer license from Sonnet. A far field viewer license is an optional add-on to Sonnet Suites and must be purchased separately. It is only available in the Sonnet Suites package and is not available in the Blink package.

---

## Background

Since **em** is an analysis of 3-D planar circuits in a completely enclosing, shielding, rectangular box, the analysis of radiating structures is not an application which immediately comes to mind.

However, **em** can be used to simulate infinite arrays using a waveguide simulator. In this technique, as shown in on page 329, a portion of the array is placed within a waveguide. The waveguide tube is vertical, connecting the radiating patches to the termination, which is a matched load. The images formed by the waveguide walls properly model the entire infinite array scanned to a specific angle.

The waveguide simulator inspired what we now call the Open Waveguide Simulator Technique described in the next section.

## Modeling Infinite Arrays

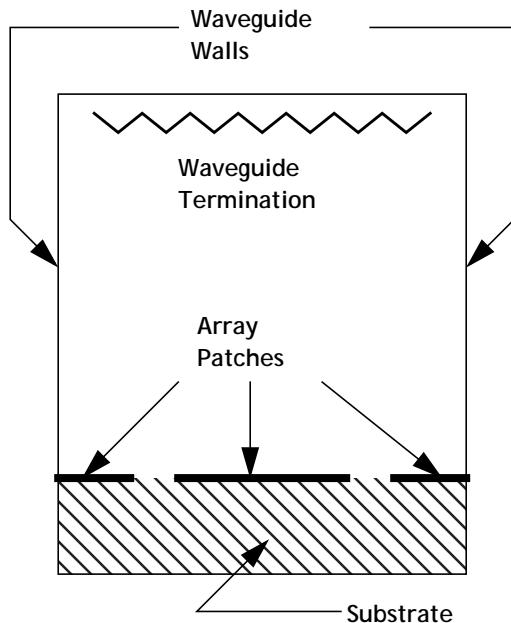
The sidewalls of the shielding box in the **em** analysis easily represent the sidewalls of the waveguide in the infinite array waveguide simulator. A side view is shown in the figure on page 329.

Providing a termination for the end of the waveguide requires a little more thought. Any waveguide mode can be perfectly terminated by making the top cover resistivity in **em** equal to the waveguide mode impedance. This can be done in the project editor automatically at all frequencies and all modes by selecting “WGLOAD” from the metals in the Top Metal drop list in the Box Settings dialog box.

$$Z_{TE} = \frac{\eta}{\sqrt{1 - \left(\frac{f_c}{f}\right)^2}} \quad f > f_c$$

$$Z_{TM} = \eta \sqrt{1 - \left(\frac{f_c}{f}\right)^2} \quad f > f_c$$

$$f_c = \frac{v_c}{2\pi} \sqrt{\left(\frac{m\pi}{A}\right)^2 + \left(\frac{n\pi}{B}\right)^2}$$



The waveguide simulator for infinite arrays inspired the technique described here. In this side view, the waveguide walls form images of the array of microstrip patches, simulating an infinite array.  $v_c$  is the velocity of light in the medium filling the waveguide.

In a phased array with the array scanned to a specific direction, a single waveguide mode is generated. The **em** software can model the waveguide simulator of that infinite array just by setting the top cover impedance to the impedance of the excited waveguide mode.

## Modeling an Open Environment

If we can use a closed (i.e., terminated) waveguide to model an infinite array, we can also model radiation from a finite array; although, it must be done under certain conditions. It is important to keep in mind that, unless the analysis is carefully prepared, these conditions are easily violated, yielding incorrect results. When the conditions are met, useful results can be obtained, as shall be demonstrated.

**First Condition:** *Make both of the lateral substrate dimensions greater than one or two wavelengths.*

When using the Open Waveguide Simulator, we view the sidewalls of the shielding box as forming a waveguide whose tube extends in the vertical direction, propagating energy from the antenna toward the “Termination” as shown on page 329. Radiation is then approximated as a sum of many waveguide modes. If the tube is too small, there are few, if any, propagating modes, violating the First Condition.

There is an easily made mistake when modeling radiation from small discontinuities. Discontinuities are usually small with respect to wavelength. For a discontinuity analysis, the sidewalls are usually placed one or two substrate thicknesses from the discontinuity. In this case, the substrate dimensions are unlikely to meet the First Condition. If the sidewalls form below a cut-off waveguide, there is no radiation.

**Second Condition:** *Make sure the sidewalls are far enough from the radiating structure that the sidewalls have no affect.*

Another way to look at this condition is to consider the image of the structure (discontinuity or antenna) created by the sidewall. Position the sidewall so that the image it forms has no significant coupling with the desired structure.

Usually two to three wavelengths from the sidewall is sufficient for discontinuities. For single patch antennas, one to three wavelengths is suggested. Requirements for specific structures can easily be greater than these guidelines. If the First Condition requires a larger substrate dimension than the Second Condition, it is very important that the larger dimension is used.

If you are using the far field viewer, the larger the box the better. The far field viewer assumes that S-parameters from **em** are from a perfect open environment. If some of the power is reflected due to a box that is too small, the input power calculated by the far field viewer will be slightly incorrect. The far field viewer then calculates antenna efficiencies greater than 100%. If this occurs, the box size should be increased.

**Third Condition:** *Place the top cover outside the fringing fields (i.e., near field) of the radiating structure, preferably a half wavelength.*

If this condition is violated, the resistive top cover becomes involved in the reactive fringing fields which form the near field of the radiator. This changes what would have been reactive input impedance into resistive input impedance, overestimating the radiation loss.

Do not place the top cover thousands of wavelengths away from the radiator. Extreme aspect ratios of the box should be avoided. Empirical data for patch antennas has shown that a distance of about 1/2 wavelength works best.

**Fourth Condition:** *Set the top cover to Free Space.*

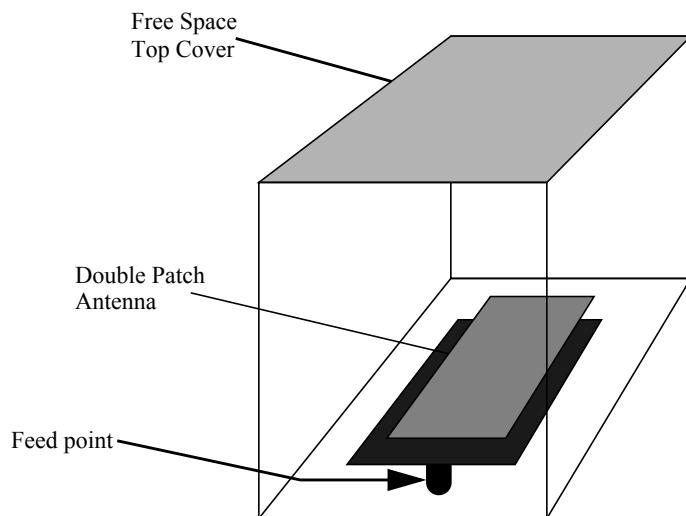
This value is a compromise. As shown by the equations on the previous page, all TE modes have a characteristic impedance larger than 377 ohms ( $\Omega$ ), while all TM modes are lower. Thus, while a 377 Ohms/square top cover does not perfectly terminate any mode, it forms an excellent compromise termination for many modes. This approximates removing the top cover of the box. If the box is large, it, in turn, approximates radiation, as shall be demonstrated.

**Fifth Condition:** *The radiating structure can not generate a significant surface wave.*

If there is a significant, compared to required accuracy, surface wave, it is reflected by the sidewalls of the box. Unless this is the actual situation, such antennas are inappropriate for this technique. Actually, the Fifth Condition is a special case of the Second Condition, since if there is significant surface wave, the Second Condition cannot be met. This condition is stated explicitly because of its importance.

In general, any surface wave is both reflected and refracted when it encounters the edge of the substrate. This boundary condition is different from either the conducting wall of Sonnet or the infinite substrate provided by a true open space analysis.

A dual patch antenna is illustrated conceptually below.

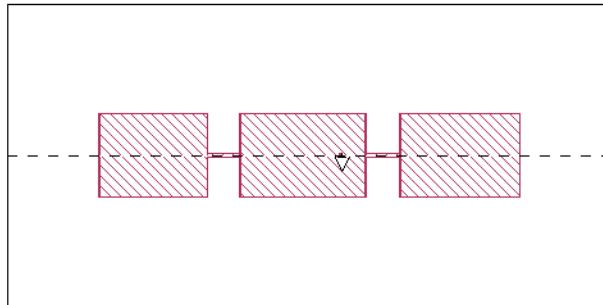


Radiation can be simulated by including a lossy top cover, a lossy dielectric layer (optional) and by placing the sidewalls far from the radiator (drawing not to scale). Place the top cover one half wavelengths from the radiator.

The feed point is created in the project editor by creating a via to ground at the feed point. Then the ground end of that via is specified as a port, just as one would specify a more typical port on the edge of the substrate at a box sidewall. A file showing an antenna similar to this one is named “dual\_patch.son” and is available in the Sonnet examples.

## Validation Example

For validation, we offer work performed by E. Ongareau of Matra Defense, Antennas & Stealthness Dept., France, as presented at the 1993 EEs of User's Group meeting at HYPER in Paris. (Reprinted with permission.) The antenna is a triple patch structure, with a top view shown below. The antenna is a test realization intended only for validation. It is not designed for optimum VSWR.



Top view of a triple patch antenna (courtesy of Matra Defense). The central patch is fed with a coaxial probe (indicated by a down pointing triangle). Each patch is resonant at a different frequency to increase the overall antenna bandwidth.

Good results are also regularly obtained on single microstrip patch antennas. We cite this example as one of the more sophisticated antennas analyzed using the Open Waveguide Simulator technique.

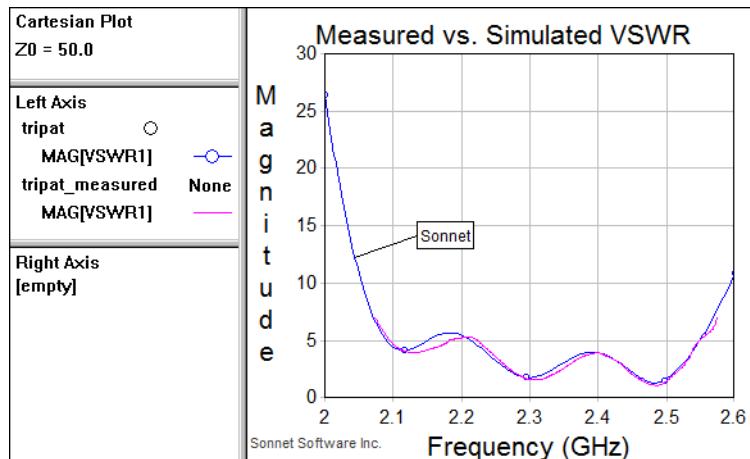
In this antenna, each patch has a slightly different resonant frequency, resulting in an increased bandwidth. The antenna is fed from below with a coax probe attached to the central patch. The feed point is indicated with a triangle.

The substrate is 3.04 mm thick with a dielectric constant of 2.94. The drawing is to scale with substrate dimensions of 200 mm x 100 mm. The top cover is 200 mm above the substrate surface. Cell size is 0.78125 mm square. A loss tangent of 0.001 is used in both air and substrate. The small air loss helps terminate the propagating modes.

The antenna project, Tripat, is available in the Sonnet examples and may be obtained using the Sonnet Example browser. You may access the Sonnet Example browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window.

The chart below shows the result. We see that the low VSWR points of each patch have differences between measured and calculated of about 1%. This is typical of most analyses of patch antennas using this technique. The differences in resonant

frequency (i.e., the reflection zeros) then determine the differences in the rest of the plot. The degree to which these differences are due to analysis error, fabrication error and measurement error cannot be determined from this data.



The measured and calculated data for the triple patch antenna were obtained completely separately, so there was no chance to "tweak" the model for agreement.

If the typical differences between measured and calculated data shown above are acceptable, given the specific requirements for a particular project, then the Open Waveguide Simulator technique can provide useful results.

## Far Field Viewer

The purpose of the far field viewer is to calculate the far field pattern of an antenna for a given excitation and set of directions (for example, phi and theta ranges). The far field viewer starts by reading the current density data generated by *em* for the antenna at the desired frequencies. The far field viewer uses the current distribution information in the project and generates the desired far field antenna pattern information. This pattern information is viewed in one of three ways: Cartesian, polar, or surface plot. A default set of values for directions, port excitations and terminations are used to calculate plots for the first frequency upon start-up of the far field viewer. Thereafter, the user specifies the frequencies, directions for the radiation pattern and the desired port excitations and terminations.

Since the far field viewer uses the current density data generated by **em**, it can analyze the same types of circuits as **em**. These include microstrip, coplanar structures, patch antennas, arrays of patches, and any other multi-layer circuit. As with **em**, the far field viewer can analyze any number of ports, metal types, and frequencies. The far field viewer cannot analyze circuits which radiate sideways, structures with radiation due to vertical components, coaxial structures, wire antennas, surface wave antennas, ferrite components, or structures that require multiple dielectric constants on a single layer.

Be aware that although the current data is calculated in **em** with a metal box, the metal box is removed in the far field viewer calculations. The modeling considerations discussed earlier in the chapter are important, however, for the accuracy of the far field viewer data relies on the accuracy of the **em** simulation.

By default, the far field viewer analyzes the first frequency in the current density data stored in the project at a default set of angles and port excitations when the file is opened.

To obtain the antenna pattern for other than the first frequency, you must select *Graph* ⇒ *Calculate* from the far field viewer main menu. The Calculate dialog box allows you to set up all the parameters for the data you desire to calculate. The far field viewer calculates the fields radiated by the current that is stored in the project. The analysis is performed in an open environment with a substrate of infinite extent. For details on the Calculate dialog box, please refer to the Help in the far field viewer program.

Please refer to “Far Field Viewer Tutorial,” page 341 for a tutorial on using the far field viewer.

## Analysis Limitations

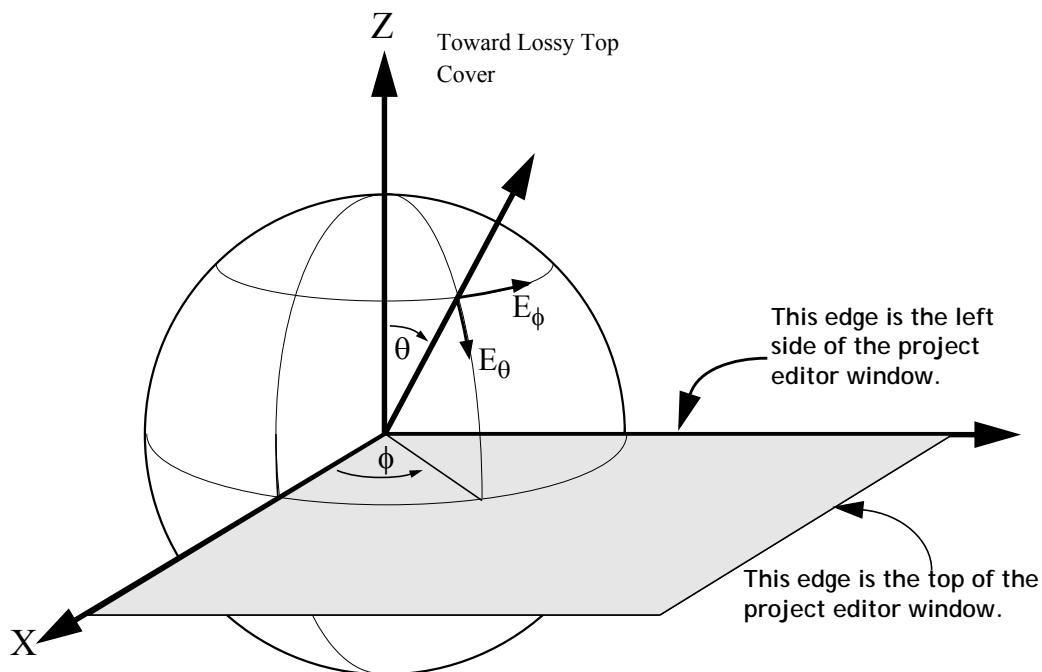
The analysis of the far field viewer has the following limitations:

- The plotted antenna patterns do not represent de-embedded data. Therefore, the effect of the port discontinuity is still included even if you specify de-embedding when running **em**.
- Radiation from triangular subsections (i.e., diagonal fill) is not included.
- The far field viewer patterns are for a substrate which extends to infinity in the lateral dimensions.

## Spherical Coordinate System

You view your antenna plot using the spherical coordinate system, which is described below.

To view an antenna plot, the far field viewer uses the spherical coordinate system shown below. The X, Y, and Z coordinates are those used in the analysis engine, *em*, and the project editor. The XY plane is the plane of your project editor window, with the Z-axis pointing toward the top of the box. The spherical coordinate system uses theta ( $\theta$ ) and phi ( $\phi$ ) as shown in the figure below.



The far field viewer allows values for theta from 0 to 180 degrees. However, values of theta greater than 90 degrees are below the horizon and are only useful for antennas without infinite ground planes. To view just the top hemisphere, sweep theta from 0 to 90 degrees and sweep phi from -180 to + 180 degrees.

The X and Y axes in the figure above correspond to the X and Y axes in the project editor. The origin is always in the lower left corner of the project editor window.

To look at an E-plane cut or an H-plane cut, set phi ( $\phi$ ) to 0 or 90 degrees, and sweep theta ( $\theta$ ) from 0 to 90 degrees. To view an azimuthal plot, set  $\theta$  and sweep  $\phi$ .

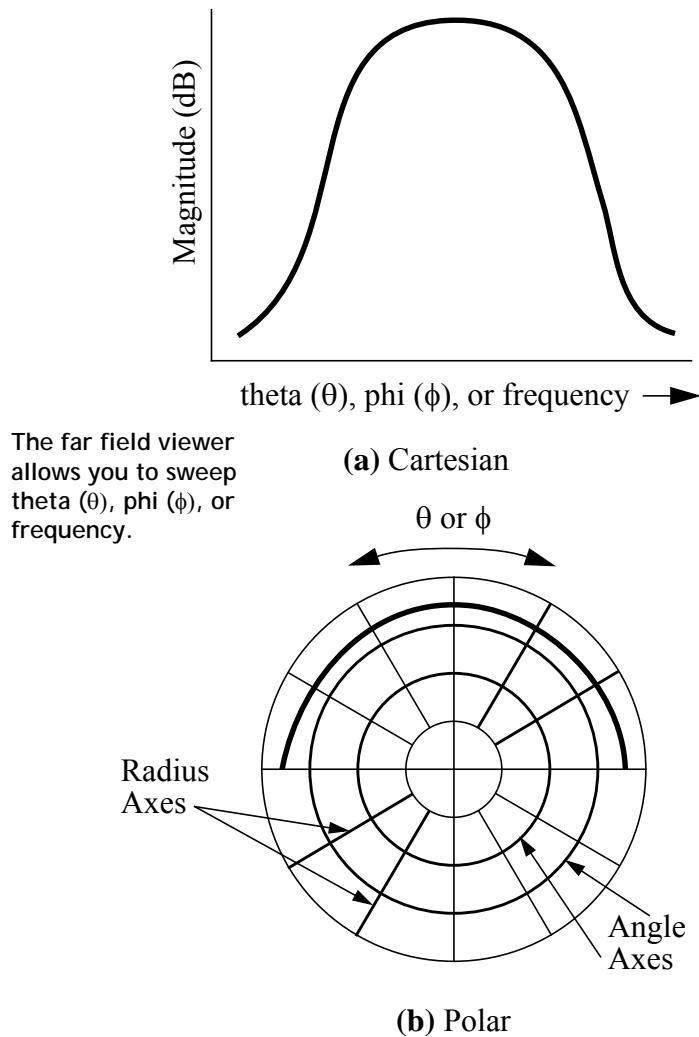
---

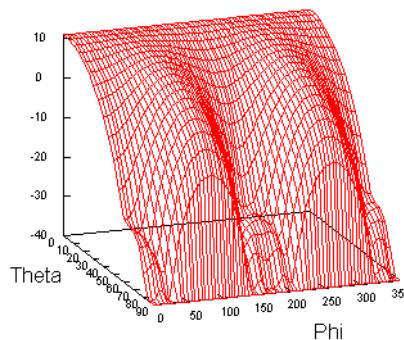
**NOTE:**

The far field viewer will allow the user to analyze the same space twice with the user determining the appropriate angle ranges for each analysis. For details, see "Graph - Select" in the far field viewer's help.

---

The far field viewer displays three plot types; cartesian, polar, and surface. All three types of plots are shown below. The cartesian plot allows the magnitude (in dB) to be plotted on a rectangular graph with your choice of theta ( $\theta$ ), phi ( $\phi$ ), or frequency for the X-axis as shown in the figure below. The polar plot allows you to select either theta ( $\theta$ ) or phi ( $\phi$ ) for the angle axis. The surface plot shows all the calculated values of theta and phi plotted against the gain for a single frequency.





(c) Surface Plot

## Normalization

There are three types of normalization to choose from in the far field viewer. By default, the far field viewer displays the power gain. The far field viewer can also provide directive gain and absolute values. The three types of normalization are discussed below.

The power gain is defined as the radiation intensity divided by the uniform radiation intensity that would exist if the total power supplied to the antenna were radiated isotropically<sup>[1]</sup>.

Directive gain is defined as the radiation intensity from an antenna in a given direction divided by the uniform radiation intensity for an isotropic radiator with the same total radiation power.<sup>[2]</sup>

The Gain and Directive gain may be displayed relative to the isotropic antenna (i.e. 0 dB), the maximum value of E for the antenna, or an arbitrary value.

Selecting Absolute for the normalization displays the radiated power in Watts/stereadian at a given angle.

You may change the normalization in the far field viewer using the Select Normalization dialog box which is opened by selecting *Graph*  $\Rightarrow$  *Normalization* from the far field viewer main menu.

## Polarization

The far field viewer displays the magnitude of the electric field vector for a given direction. The magnitude may be represented as the vector sum of two polarization components, E-theta ( $E_\theta$ ) and E-phi ( $E_\phi$ ) as shown in the figure describing spherical coordinates on page 336.

The far field viewer allows you to see the total magnitude or either component of the magnitude. Other polarizations are also available in the far field viewer and are discussed in “Graph - Polarization” in the far field viewer help.

## References

- [1] Simon Ramo, John R. Whinnery and Theodore Van Duzer, Fields and Waves in Communication Electronics, John Wiley & Sons, Inc. 1994, pg. 601.
- [2] Ibid, pg. 600.

## Chapter 19

## Far Field Viewer Tutorial

---

This tutorial describes an example of using the far field viewer to display plots. The far field viewer displays far field radiation patterns using the current density data created during an *em* analysis. In this example, we analyze an infinitesimal dipole antenna above a ground plane, shown below, and compare the results to the exact theoretical antenna pattern shown on page 358, as provided by reference 2.

---

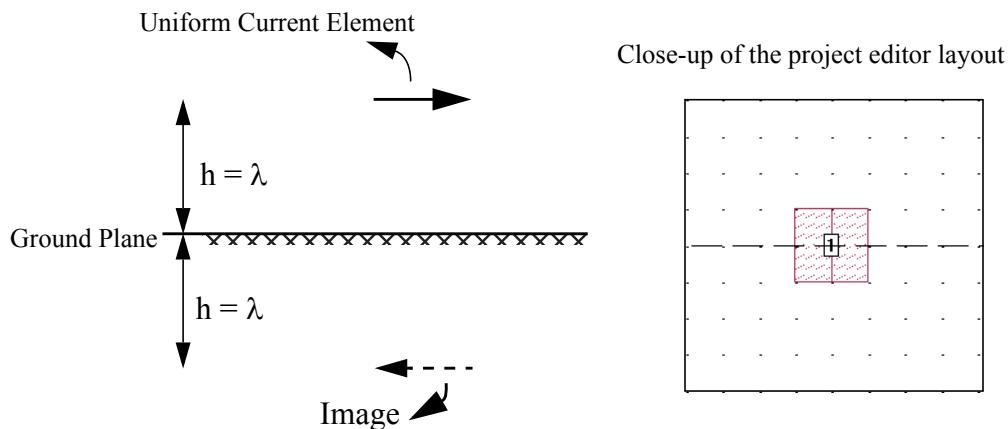
**NOTE:**

The far field viewer is only available if you have purchased a far field viewer license from Sonnet. A far field viewer license is an optional add-on to Sonnet Suites and must be purchased separately. Please see your system administrator if you are unsure about the availability of this program.

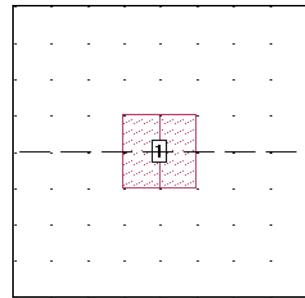
---

For more information about modeling antennas and using the far field viewer, please refer to Chapter 18.

Although this example is not very practical, it is a good example to use for validation because of its simplicity. The infinitesimal electric dipole is placed one wavelength (300 mm at 1 GHz) above the ground plane (an electric field reflection boundary).



Close-up of the project editor layout



## Creating an Antenna Pattern File

This tutorial uses an infinitesimal dipole one wavelength above the ground plane. The project, Infpole, may be obtained using the Sonnet Example browser. You may access the Sonnet Example browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window. Save the file to your working directory.

### **1 Save a copy of “infpole.son” to your working directory.**

The file “infpole.son” is the circuit geometry project file for the dipole antenna which was created using the project editor. The dipole geometry can be viewed by using the project editor.

It is important to remember that in order to produce data for input into the far field viewer, the Compute Current Density option must be selected in the Analysis Setup dialog box in the project editor.

Infpole was analyzed at a linear frequency sweep from 0.8 GHz to 1.2 GHz in intervals of 0.1 GHz.

## Running the Far Field Viewer



- 2 Click on the View Far Field button on the Sonnet task bar to invoke the far field viewer.**

A pop-up menu appears on your display.

- 3 Select “Browse for Project” from the pop-up menu.**

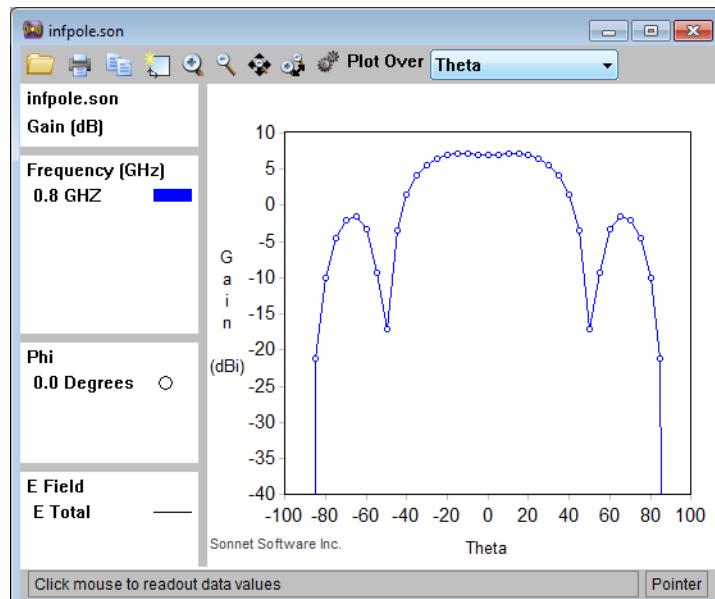
A browse window appears on your display.

- 4 Using the browse window, select your saved copy of “inpole.son”.**

The far field viewer window opens on the project file, “inpole.son”.

After the initial calculation is complete, a plot appears on your display as shown below. When a new file is opened, the far field viewer performs an analysis on the first frequency based on a default set of values for directions, port excitations and terminations and displays the Gain (dB) versus theta for the first value of phi. The calculation defaults are as follows:

- There are two values of phi: 0° and 90°
- Theta ranges from -90° to +90° in 5° intervals.
- Port 1 is set to a 1.0 V source magnitude with a 50.0 Ω load.



The far field viewer display defaults to a cartesian plot with theta selected on the X-axis. The polarization defaults to *Theta/Phi*. The Y-axis is set to display the Gain (in dB) of the pattern response and is normalized to power gain of the ideal isotropic antenna.

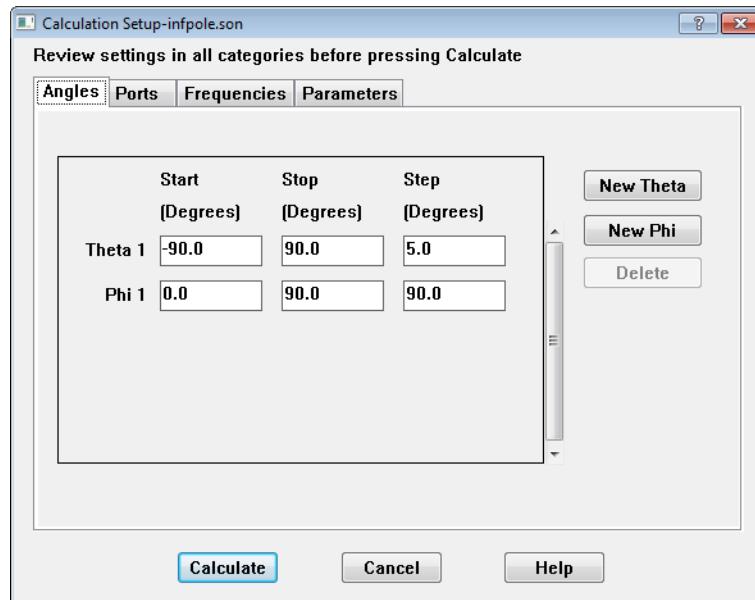
To change the calculation and display defaults, see File - Preferences in the far field viewer's help.

## Calculating the Response

As mentioned above, when the far field viewer is invoked, the response data is calculated for only the first frequency in the current response file. To calculate data for the other frequencies at additional angles, perform the following:

- 5 **Select *Graph*  $\Rightarrow$  *Calculate* from the far field viewer main menu.**

The Calculation Setup dialog box appears on your display with the Angles tab selected as shown below.



## Selecting Phi Values

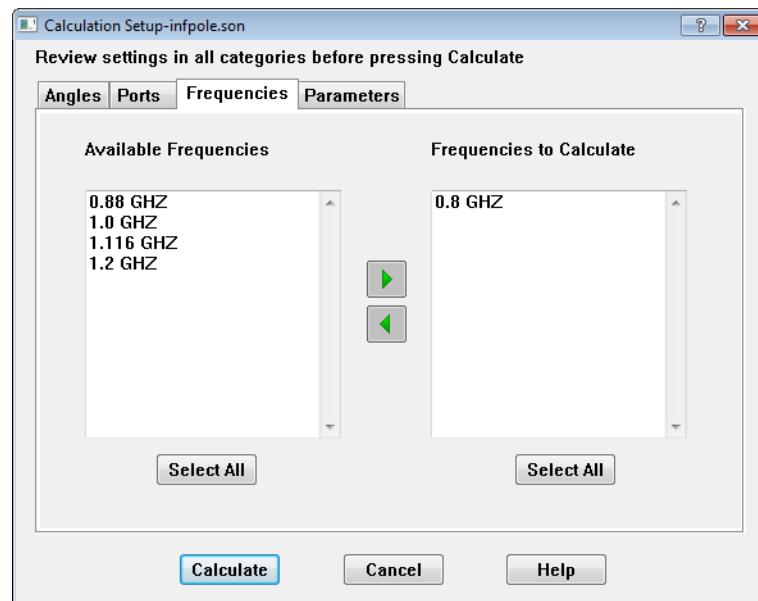
- 6 Enter 0 in Start text entry box, 90 in the Stop text entry box and 5 in the Step text entry box of the Phi line.

This analyzes data points from  $\phi = 0^\circ$  to  $90^\circ$  in intervals of  $5^\circ$ .

## Selecting Frequencies

- 7 Click on the Frequencies tab in the Calculation Setup dialog box.

The Frequencies tab is now displayed, as shown below.



- 8 Click on the Select All command button under the Available Frequencies.

All of the frequencies are highlighted.

- 9 Click on the Right Arrow button.

This moves all the selected frequencies to the Calculated Frequencies column.

**10 Click on the Calculate command button.**

There is a delay while the far field viewer calculates the requested data. The calculations for each frequency are performed using the defaults cited above for phi, theta, port excitation and exclusions, since none of these items were changed before selecting Calculate.

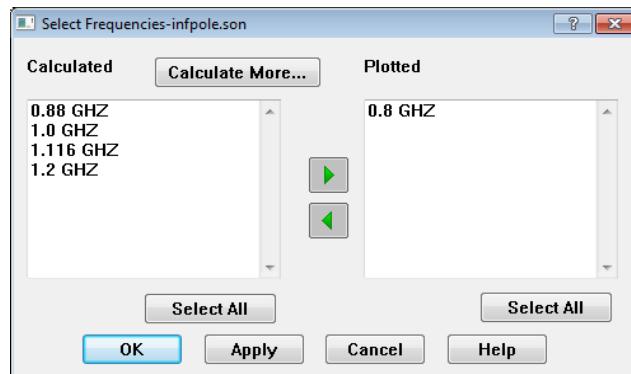
A status box appears on your display to provide updates on calculation progress. The display is updated when the calculation is completed. Be aware that for larger, more complicated circuits, this delay might be a considerable one.

## Selecting the Response

The far field viewer allows you to select which data items you wish displayed at any given time. In the next session, you display 1 GHz data at different phi's.

**11 Select *Graph*  $\Rightarrow$  *Select*  $\Rightarrow$  *Frequencies* from the far field viewer main menu.**

The Select Frequencies dialog box appears on your display.



The Calculated column displays the frequencies for which data has been calculated, but is not presently displayed. The Plotted Column shows those frequencies which are presently displayed. In this case, 0.8 GHz.

**TIP**

---

You may also open the Select Frequencies dialog box by right-clicking in the Frequency area of the legend and selecting *Select* from the menu which appears on your display.

---

**12 Double-click on 0.8 in the Plotted column.**

This moves 0.8 to the calculated column, i.e., this frequency is not displayed.

**13 Double-click on 1.0 in the Calculated column.**

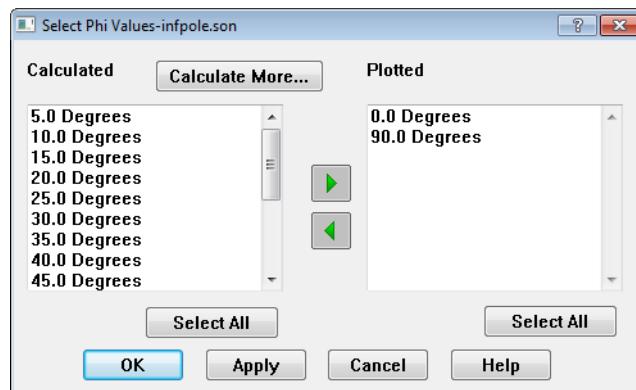
This moves the value 1.0 to the Plotted column.

**14 Click on the OK command button.**

This closes the dialog box and updates the display with the data for 1.0 GHz at Phi = 0°.

**15 Select *Graph* ⇒ *Select* ⇒ *Phi* from the far field viewer main menu.**

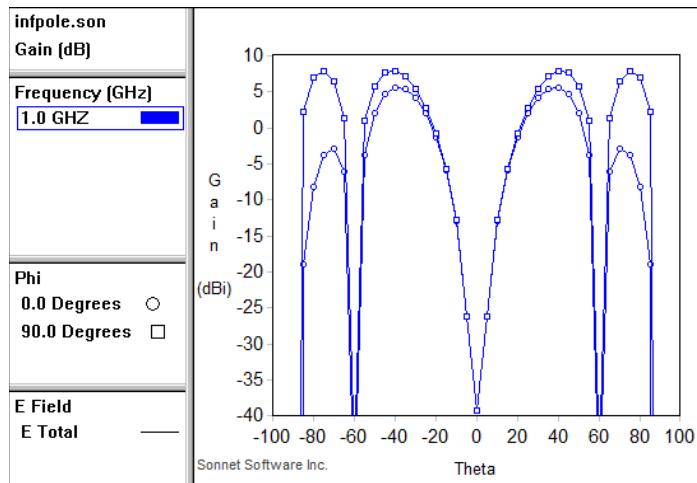
The Select Phi's dialog box appears on your display.

**16 Use the scroll bar on the Calculated Column to move down the list until 90.0 Degrees is displayed.****17 Double-click on 90.0 Degrees to move this value to the Plotted column.**

Values in this column are displayed in the far field viewer.

**18 Click on the OK command button.**

The dialog box disappears and the far field viewer display is updated. It should appear similar to the figure below.



The plot is drawn showing two curves:  $E_{\text{total}}$  at  $\phi = 0$  and  $90$  degrees. The upper curve is the radiation pattern at  $\phi = 90$  degrees. The lower curve is the radiation pattern at  $\phi = 0$  degrees.

The far field viewer automatically selects an appropriate scale for the plot.

**19 Select *Graph*  $\Rightarrow$  *Select*  $\Rightarrow$  *Phi* from the far field viewer main menu.**

The Select Phi's dialog box appears on your display.

**TIP**

---

You may also access the Select Phi's dialog box by right-clicking in the Phi box in the legend and selecting *Select* from the pop-up menu which appears.

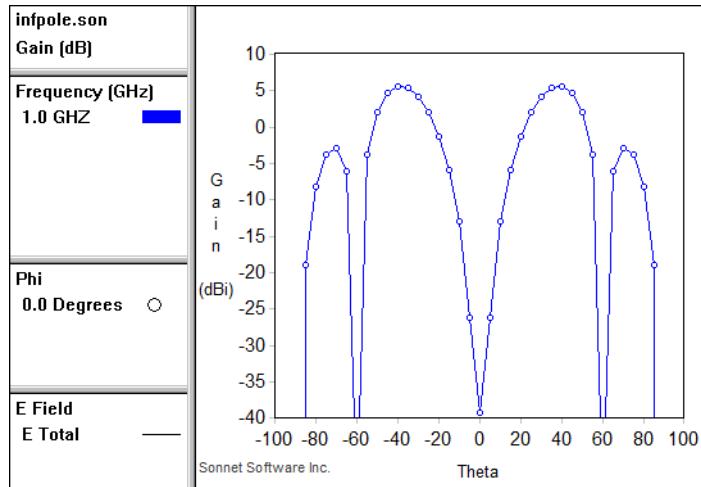
---

**20 Double-click on 90.0 degrees in the Plotted column to move it from the Plotted column to the Calculated column.**

This removes 90.0 degrees from the plot.

**21 Click on the OK command button.**

The dialog box disappears and the far field viewer plot is updated. It should appear as below.



## Zooming

The zoom button, located on the tool bar, may be used to magnify a specific area in the plot.



**22 Click on the Zoom In button on the Tool bar.**

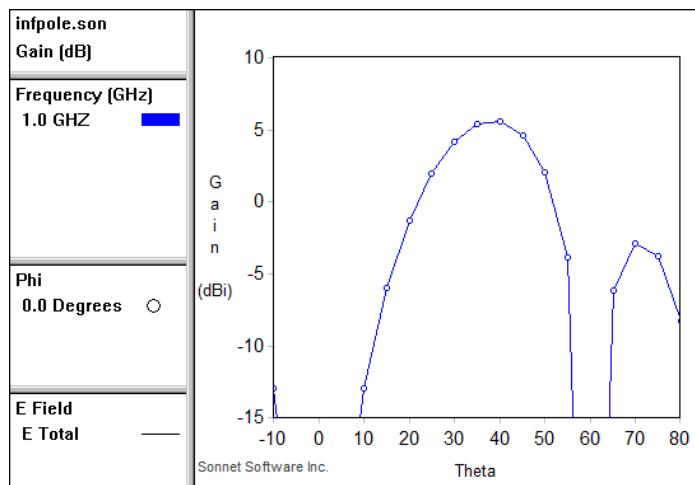
For this example, zoom in on the area from 0 to 10 dB in Gain, where Theta ranges from  $20^\circ$  to  $50^\circ$ .

You may also use *View*  $\Rightarrow$  *Zoom In* or the Space Bar for the zoom function.

**23 Click on the point in the plot corresponding to 0 dB Gain and Theta =  $20^\circ$ , then drag the mouse to the point in the plot corresponding to 10 dB Gain and Theta =  $50^\circ$**

A rubber band surrounding the area to be magnified follows the mouse.

When the mouse is released, the plot is updated with a magnified view of the selected area, as shown below.



### 24 Click on the Full View button on the Tool bar.

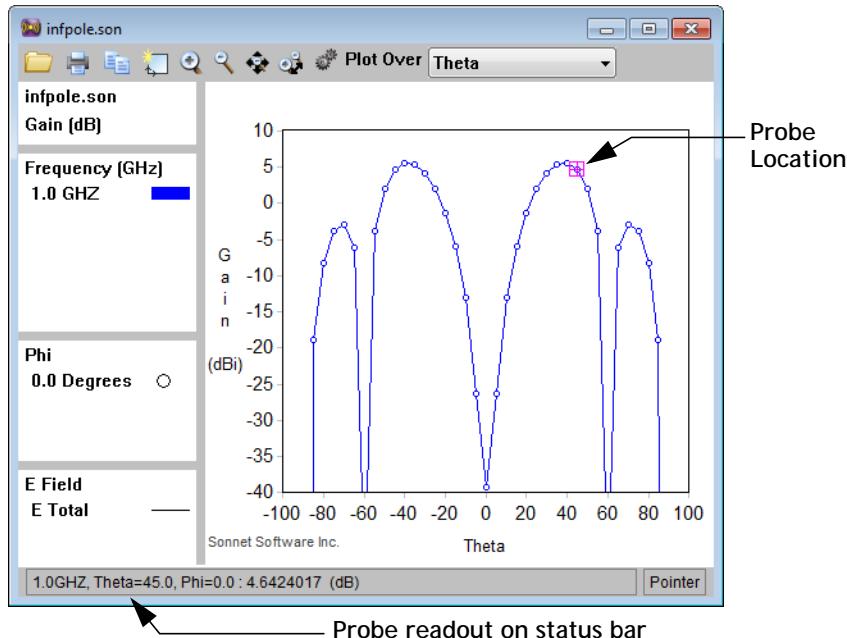
The full plot once again appears on your display.

## Probing the Plot

To evaluate the pattern response at any location in your plot you simply click at the desired location.

**25 Click on the theta = 45° point on the plot.**

A square appears around the point, as shown below. The readout for the point including the frequency, value of theta and phi, and the gain appear in the status bar at the bottom of the far field viewer window.



**26 Press the left arrow key,  $\leftarrow$ , to move to the theta = 40° point on the plot, or alternately, click on that point.**

The probe box now appears at that point and the data is updated in the status bar.

Note that if there is more than one data curve displayed, the up and down arrow keys,  $\uparrow$  and  $\downarrow$ , would move the data probe between curves, while the left and right arrow keys,  $\leftarrow$  and  $\rightarrow$ , move between data points on any given curve.

## Re-Normalizing the Plot

By default, the far field viewer displays the power gain. The power gain is defined as the radiation intensity divided by the uniform radiation intensity that would exist if the total power supplied to the antenna were radiated isotropically<sup>[1]</sup>.

We shall now normalize the plot to the maximum value.

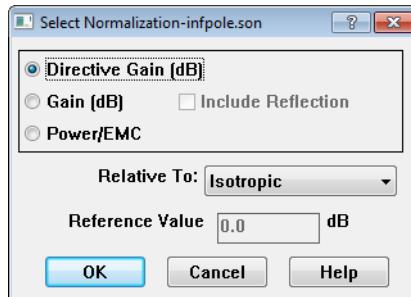
**27 Select Graph  $\Rightarrow$  Normalization to change the normalization.**

The Select Normalization dialog box appears on your display.

**28 If it is not already selected, click on the Directive Gain (dB) radio button.**

The directive gain is defined at the radiation intensity from an antenna in a given direction divided by the uniform radiation intensity for an isotropic radiator with the same total radiation power.

The far field viewer calculates the directive gain based on the total power radiated by your circuit. The far field viewer calculates the total power by using all the theta and phi points to integrate over the entire surface. Therefore, the more theta and phi points calculated, the more accurate the values provided for directive gain. A Figure of Merit (F.O.M.) appears in the Gain section of the legend when directive gain is selected to provide you with an idea of how accurate is the data (100% is perfect). If this figure is too low, try recalculating using more theta and phi data points. If this figure is too high, that is an indication that the problem is over calculated. i.e., angles are being analyzed twice. In this case, check for duplicate angles such as theta = 180 and theta = -180.

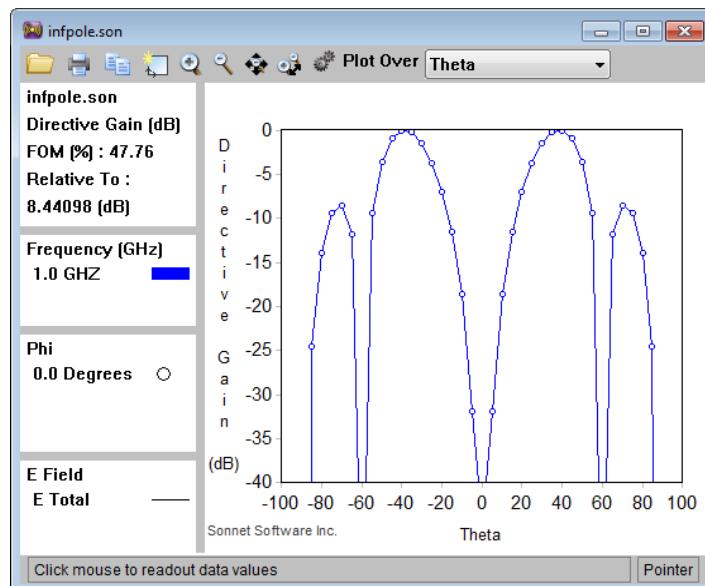


**29 Select Max from the Relative To drop list.**

This selects the maximum value of radiation for the plot to be the 0 dB point of the plot.

**30 Click on the OK command button.**

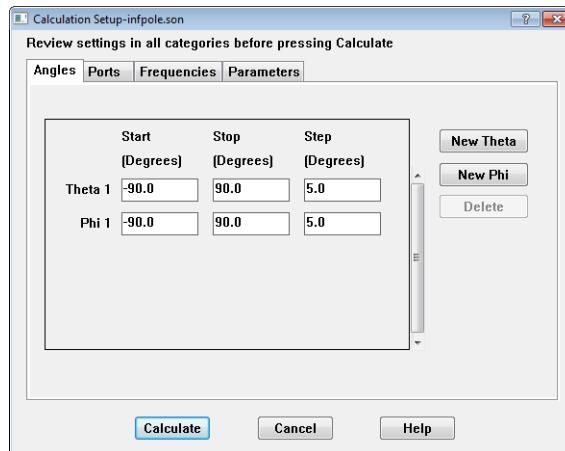
The dialog box is closed and the display is updated with the data normalized to the maximum value, which in this case is 8.44098 dB. Note that the FOM is only 47.76. As mentioned above, to increase this figure, you increase this by recalculating for more theta and phi angles.

**31 Select *Graph*  $\Rightarrow$  *Calculate* from the main menu.**

The Calculation Setup dialog box appears on your display.

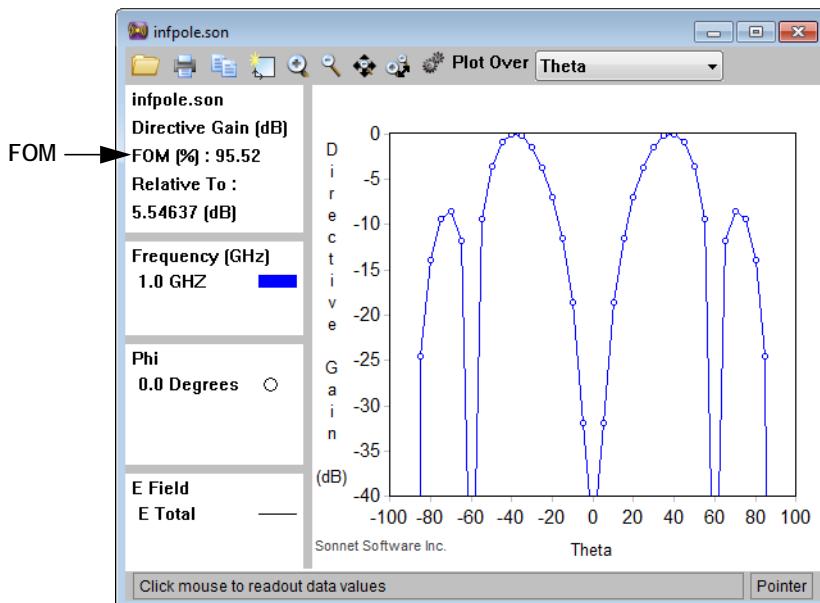
- 32 If it is not already selected, click on the “Angles” tab.
- 33 Enter “-90.0” in the Phi 1 Start text entry box.

This analyzes data points from phi = -90° to 90° in intervals of 5° which doubles the number of analysis points for phi.



- 34 Click on the Calculate button to close the dialog box and calculate new values.

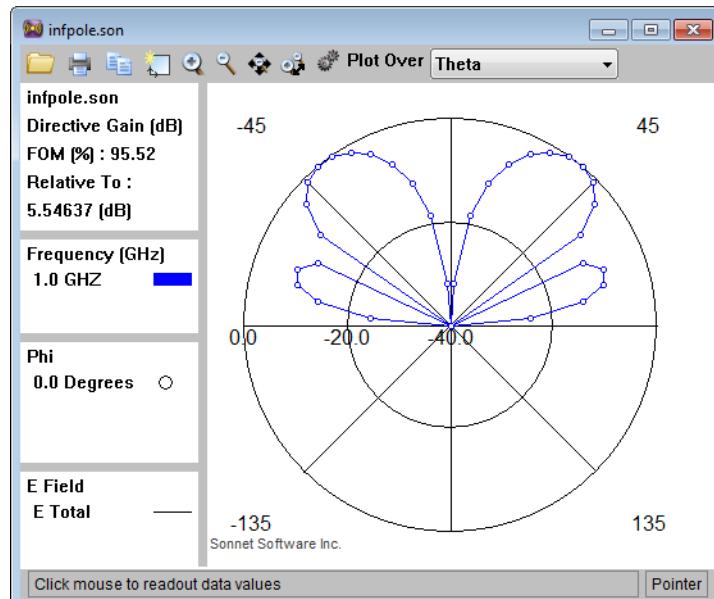
The plot is updated and now shows more data points. Note that the FOM has gone up to 95.52%.



## Changing to a Polar Plot

35 Select **Graph**  $\Rightarrow$  **Type**  $\Rightarrow$  **Polar** to select a polar plot for the display.

A polar plot is chosen since the theoretical data for an infinitesimal dipole is shown in a polar plot. The display is updated using the polar coordinate system. Phi is held constant and theta is swept.



### TIP

---

You may select another type of plot by right-clicking in the plot title area of the far field viewer display and selecting *Type* from the pop-up menu which appears.

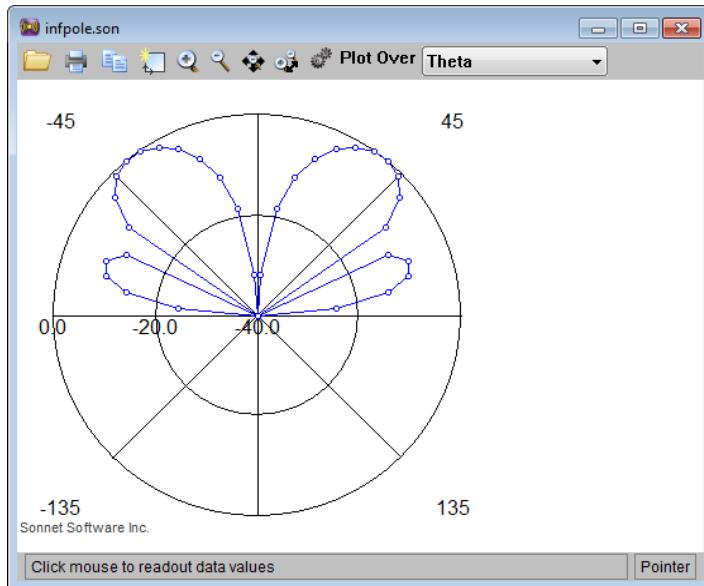
---

## Turning Off the Legend

Since the legends take up a lot of space on the display, you may turn them off, allowing the plot to fill the extra space.

**36 To turn off the legend, select *View* ⇒ *Legend*.**

This turns “off” the legend and the far field viewer redraws the plot without the legends. The menu item toggles the display state of the legend, so that selecting *View* ⇒ *Legend* again displays the legend.

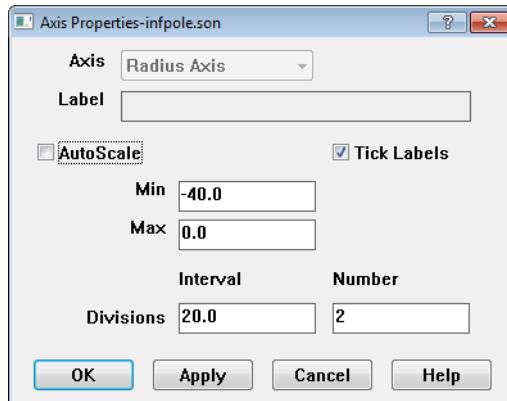


## Changing the Radius Axis

You can change the radius axis limits of the plot to another value. For this example, you will change the intervals from 20 dB to 10 dB.

- 37 Select **Graph**  $\Rightarrow$  **Axes** from the main the far field viewer menu.

The Axes Properties dialog box appears on your display.



- 38 Click on the **AutoScale** checkbox to turn it “off.”

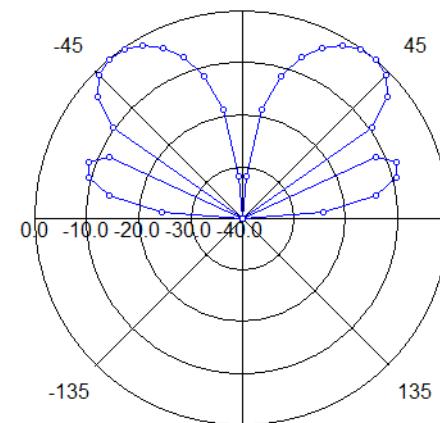
This enables the Min and Max text entry boxes under Autoscale and the Interval and Number text entry boxes under Tick Labels.

- 39 Enter “10” in the Interval text entry box.

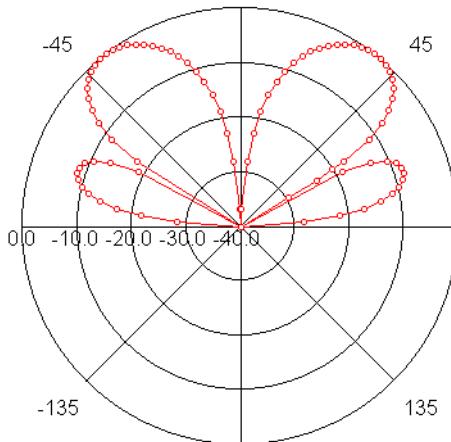
This sets the intervals on the plot grid to 10 dB.

- 40 Click on the **OK** command button.

The dialog box disappears and the far field viewer display is updated with the new interval value for the axes. Your display should be similar to the one shown below.



Shown above is the far field viewer calculated far field antenna pattern for the very short dipole in the file infpole.son. The result should be compared with theoretical result in the next figure.



Exact far field antenna pattern from reference [2] of an infinitesimal dipole one wavelength above a ground plane.

## Selecting a Frequency Plot

To see how the antenna pattern changes with frequency, you use a frequency plot. Before you can select a frequency plot, you must return to a cartesian plot.

- 41 **Select *Graph*  $\Rightarrow$  *Type*  $\Rightarrow$  *Cartesian* from the far field viewer main menu.**

Your display is updated with a cartesian plot. Note that the autoscale is automatically turned back on when you switch plot types.

- 42 **Select *View*  $\Rightarrow$  *Legend* from the far field viewer main menu.**

The legend once again appears in your display.

- 43 Select Frequency from the Plot Over drop list on the far field viewer tool bar.



- 44 Select **Graph**  $\Rightarrow$  **Select**  $\Rightarrow$  **Theta** from the far field viewer main menu.

The Select Theta's dialog box appears on your display.

- 45 Double-click on -90.0 degrees in the Plotted list.

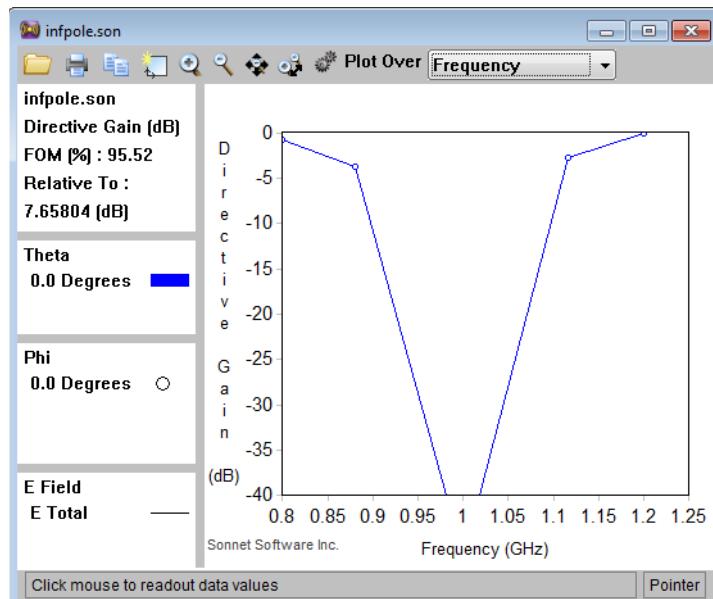
This value moves to the Calculated list which removes it from the display.

- 46 Double-click on 0.0 degrees in the Calculated list.

This value moves to the Plotted list which adds it to the display.

- 47 Click on the OK command button.

The display is updated with a frequency plot as shown below. Note that the gain is now calculated relative to 7.65804 dB, since the Normalization is relative to Max and this value is the maximum value of radiation for this plot.





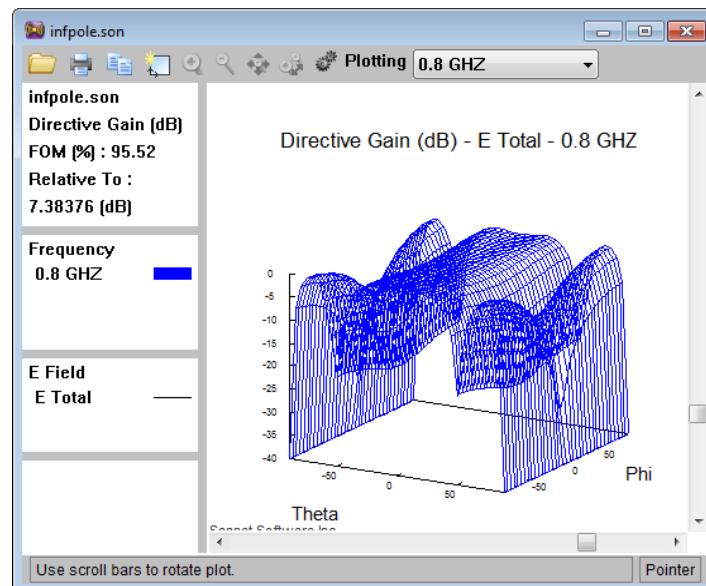
Notice that E-Total at Theta = 90 is shown in the legends, but does not appear on the graph. This occurs because the magnitude is too small to show on the plot.

## Viewing a Surface Plot

The surface plot shows all the calculated values of theta and phi plotted against the gain for a single frequency.

- 55 Select **Graph**  $\Rightarrow$  **Type**  $\Rightarrow$  **Surface** from the main menu.

Your display is updated with a surface plot with the first frequency, 0.8 GHz, selected for display.



## Saving the Far Field Viewer File

- 56 Select **File**  $\Rightarrow$  **Save As** from the far field viewer main menu.

The file is saved to the same filename with a “.pat” extension, i.e., “infpole.pat”. This saves any data calculated during this the far field viewer session.

## Exiting the Far Field Viewer Program

**57 To stop the program, select *File* ⇒ *Exit*.**

The far field viewer window disappears from your display. This completes the tutorial.

## References

- [1] Simon Ramo, John R. Whinnery and Theodore Van Duzer, Fields and Waves in Communication Electronics, John Wiley & Sons, Inc. 1994, pg. 601.
- [2] Constantine A. Balanis, **Antenna Theory Analysis and Design**; New York; Harper & Row, 1982, section 4.7.3

# Chapter 20 SPICE Model Synthesis

---

Sonnet's analysis engine, *em*, provides a frequency domain solution in the form of S- Y- and Z-parameters. Many time domain simulators, such as traditional SPICE engines, do not have the capability to import frequency domain data, or have problems with efficiency, stability, or accuracy when using frequency domain data.

To address these problems, Sonnet provides the capability to output your frequency domain data into a SPICE-compatible file. You may choose from four different models, depending on your needs. A brief description of each is provided below, and each model is discussed in detail in the sections following.

- **PI Model:** Appropriate for circuits that are electrically small (less than 1/20 wavelengths) and can be represented by a simple PI topology.
- **N-Coupled Line Model:** Appropriate for single or coupled transmission lines. Generates an RLGC matrix which can be used in the Cadence Virtuoso mtlne element.
- **Broadband Spice Model:** Appropriate for most circuits. Generates a lumped model based on a rational polynomial fit. Requires Broadband Spice Extractor license.
- **Inductor Model:** Appropriate for spiral inductors. Generates a simple lumped element model which is specific to spiral inductors. Requires Broadband Spice Extractor license.

## PI Model

The PI Model may be used for circuits that are electrically small and can represented by a simple PI topology (see "Topology Used for PI Model Output" on page 368). The results are generally valid only over a narrow frequency band. The model may be saved in PSpice or Cadence Spectre format.

The PI model Spice generation capabilities are intended for any circuit which is small with respect to the wavelength at the highest frequency of excitation. Typically, 1/20th wavelength is an appropriate limit. (If a circuit is too large, you can often split it into two or more circuits and analyze each separately.) This limitation is due to the circuit theory limitations of modeling a circuit with just a few lumped elements. The Sonnet electromagnetic analysis is not intrinsically limited in this fashion.

The model generated by the analysis includes any lumped elements (including mutual inductors) between any ports of the circuit layout. Lumped elements from any port to ground are also included. The synthesis capability does not allow internal nodes (nodes which are not connected to a port in the layout) with the single exception of the internal node required to specify a resistor in series with an inductor.

Any circuit which requires internal nodes for an accurate model should be split into several parts so that the required points become nodes. Internal ports without ground reference give incorrect results. Any internal ports should be carefully specified and checked for reasonable results.

### Using The PI Model Spice Option

The PI Model synthesis needs electromagnetic results for at least two frequencies to accomplish its work. It is not possible to create a PI model if the circuit is analyzed at only one frequency. A PI model is created for pairs of frequencies. The second frequency is determined by taking the first frequency and adding a percentage specified by the user. The second frequency then becomes the first frequency for the next pair of frequencies for which a SPICE model is generated. The synthesis continues in this way until all the frequencies have been used.

The default value for the separation percentage is 10%. In this case, a SPICE model is generated using the first frequency and the next highest frequency which provides a 10% gap. This continues until there are no more frequencies. For ex-

ample, for a frequency sweep from 10 - 40 GHz with a 0.1 GHz resolution using a separation of 10%, the first few frequencies used would be 10, 11, 12.1 and 13.3 GHz.

After completing the analysis, always do a “reality check” for reasonable values. If you have bad data, the frequency may be too high or too low. If the frequency is too low, the solution may have unity S-parameters, causing a strange SPICE model. To be absolutely sure your results are good, select a different frequency band and re-analyze the circuit. You should obtain similar results between the two analyses.

You may obtain PI Model Spice data in two different ways. The first is to specify an optional output file before executing your analysis. The second is to generate a PI model from the response viewer. The second method has the advantage of allowing you to perform the data check mentioned above before creating the SPICE data file.

To specify a PI Model Spice output file from the response viewer, perform the following:

**1 Analyze the circuit at the desired frequencies.**

The analysis monitor appears on your display to show the progress of the analysis. It is important to note that if your results contain more than two analysis frequencies, then multiple Spice models, one for each pair of frequencies, will be created in one file.



**2 When the analysis is complete, click on the View Response button on the analysis monitor's tool bar.**

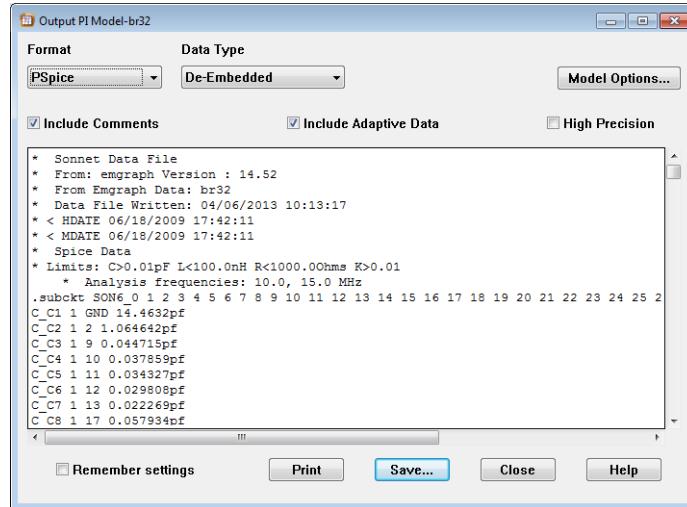
The response viewer is invoked with a plot of your response data.

**3 Click on the project name in the response viewer legend to select it.**

An outline appears around the project name to indicate that it is presently selected. If you have multiple projects open in the response viewer and have not selected a project before using the extraction command, then a window appears which allows you to select the desired project.

## 4 Select **Output $\Rightarrow$ PI Model File** from the response viewer main menu.

The Output PI Model dialog box appears on your display. The contents of the output window in the Output PI Model dialog box displays the Spice data for the PI Model in the PSpice format which is the default.

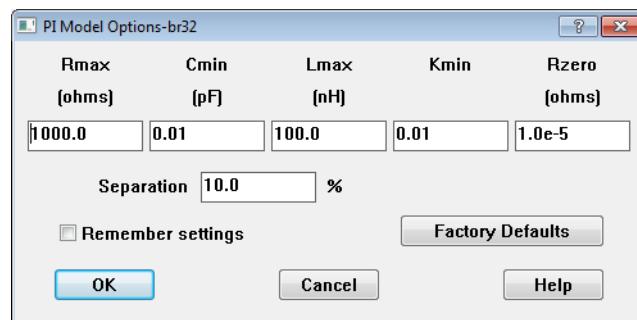


## 5 Select the desired file format from the Format drop list.

If you select Spectre as the Format (PSpice is the default), the contents of the output window is updated.

## 6 To change the parameters for the SPICE data, click on the Model Options button.

The PI Model Options dialog box appears on your display. To reduce the number of lumped elements in the model, open circuit limits can be specified here as shown below.



The values are defined as follows:

**RMAX:** Maximum allowed resistance (ohms). The default value is 1000.0 ohms.

**CMIN:** Minimum allowed capacitance (pF). The default value is 0.01 pF.

**LMAX:** Maximum allowed inductance (nH). The default value is 100.0 nH.

**KMIN:** Minimum allowed mutual inductance (dimensionless ratio). The default value is 0.01.

**RZERO:** Resistor to go in series with all lossless inductors (resistance in ohms). Needed for some versions of SPICE. The default value is 0.0.

**Separation:** This is the calculation interval between the two frequencies used to generate the SPICE model specified as a percentage. The second frequency is obtained by adding the specified percentage of the first frequency to the second frequency.

All calculated component values which fall outside the allowed range specified by the user in the model options are excluded from the resulting lumped model. The RZERO entry is provided for those versions of SPICE which need inductors to have some small loss to avoid numerical difficulties. The default value of 0.0 disables this capability.

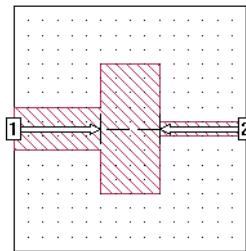
Enter the desired values for the parameters in the PI Model Options dialog box.

- 7 **Click on the OK button in the PI Model Options dialog box to apply the changes and close the dialog box.**
- 8 **Click on the Save button in the Output PI Model dialog box.**

A browse window appears which allows you to save the data displayed in the output window. The file extension depends on which type of SPICE format you have selected.

## A Simple Microwave Example

Shown below is a simple step discontinuity followed by the PI model produced when you set up an optional PI Model Spice output file.



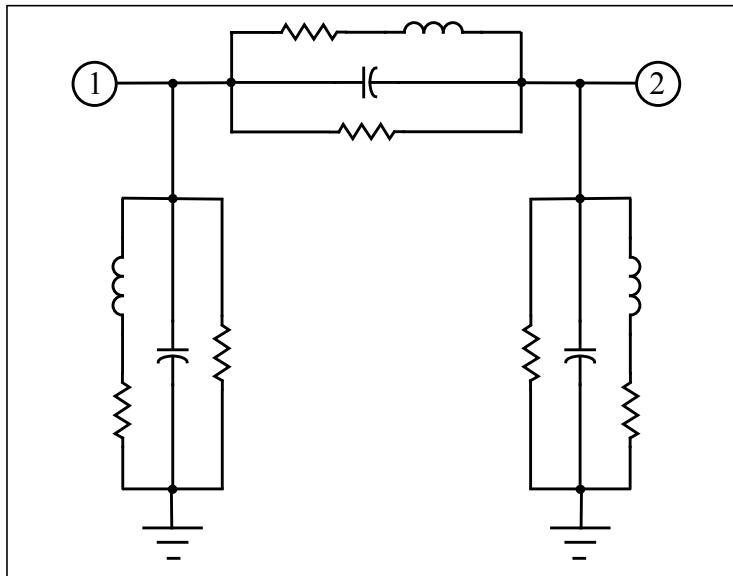
```
* Limits: C>0.01pF L<100.0nH R<1000.0Ohms K>0.01
* Analysis frequencies: 200.0, 220.0 MHz
.subckt spiral_0 1 2 GND
C_C1 1 GND 1.123431pf
C_C2 2 GND 1.316483pf
L_L1 1 3 30.66429nh
R_RL1 3 2 1.57292
.ends spiral_0
```

There are two capacitors to ground (node GND), one inductor connected between node 1 and node 3, and one resistor connected between node 3 and node 2 in the lumped element model.

## Topology Used for PI Model Output

The topology of the lumped element model generated by **em** depends on the circuit being analyzed. In general, the model contains an inductor (in series with a resistor if using loss), a capacitor and a resistor (when using loss) connected in parallel from each port to ground. A similar parallel RLC network is also connected between each port. Therefore, a four-port circuit can contain more elements than a two-port circuit. Each inductor may also have a mutual inductance to any

other inductor in the network. The figure below shows the most complex equivalent circuit possible for a two-port (mutual inductances not shown). Any values that are outside of the open circuit limits are not included.



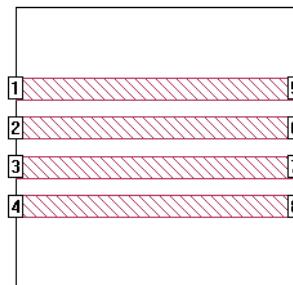
Equivalent circuit of a two-port structure using the PI Model. Mutual inductances also exist between all inductors, but are not shown. Any component whose value is outside of the open circuit limits are not printed in the SPICE output file.

## N-Coupled Line Model

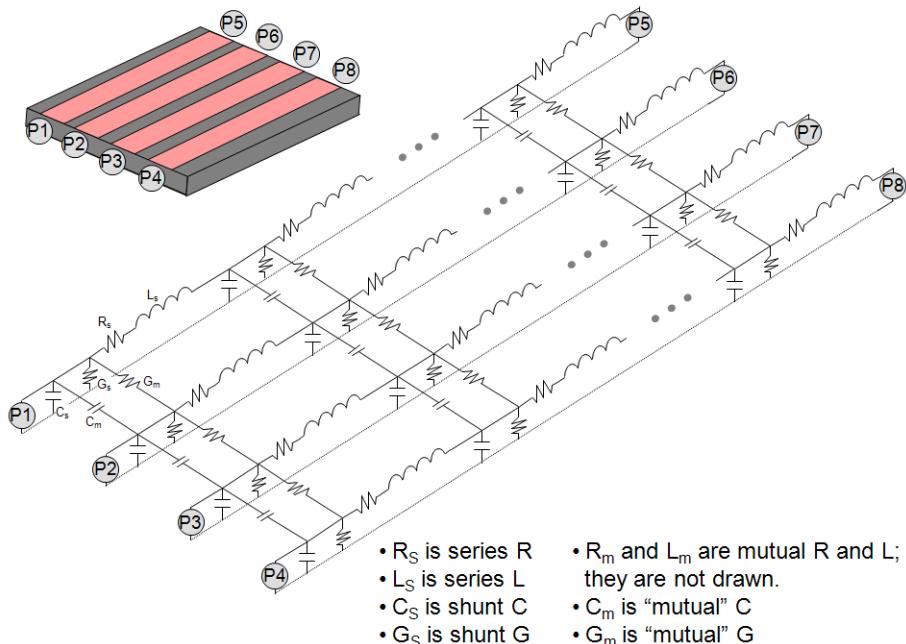
Most circuit design programs provide models for single and multiple-coupled transmission lines. However, it is often desirable to use EM simulated data in circuit design programs. These programs often provide transmission line models which utilize RLGC parameters. R, L, G, and C are the resistance, inductance, conductance and capacitance per meter of a transmission line. The RLGC parameters can be extracted from an EM simulation of a short section of the transmission line. They can then be used to model any length of line having the same cross-section.

Sonnet can export the RLGC parameters in a format compatible with the `mtline` component in Cadence Virtuoso Spectre.

Shown below is an example of a project composed of four transmission lines.



As in the circuit shown above, the input ports of your project must be numbered 1 through N and the output ports N+1 through 2N. The input of line M should be port M and its output should be port M+N. The software does not check for this condition, but issues a warning message if the number of ports is not an even number. The equivalent circuit is shown below for the four transmission lines shown above.



You may generate RLGC parameters automatically by setting up an output file in the project editor. To setup an output file in the project editor, select *Analysis*  $\Rightarrow$  *Output Files* from the project editor main menu, then click on the N-coupled Line Model button in the Output Files dialog box which appears. You may also gener-

ate an output file of RLGC parameters from previously generated response data in the response viewer. To do so, open your project in the response viewer and select *Output*  $\Rightarrow$  *N-coupled Line Model* from the response viewer's main menu. RLGC parameters are generated for each analysis frequency.

Shown below are the RLGC parameters in Spectre format for a two-line project:

```

FORMAT Freq: L1:1 L2:1 L2:2
      R1:1 R2:1 R2:2
      C1:1 C2:1 C2:2
      G1:1 G2:1 G2:2

;
; Matrix values extracted from structure of length 1.25e-4 meters.
;
; Z0 Matrix Eigenvectors:
; 0.70709973181 1.01204653e-7 0.70710975084 0.000000000000
; 0.70711383049 0.000000000000 -0.7071038115 -4.2771946e-8
;
;
; Z0 Matrix Eigenvalues:
; 72.7927707171 -0.1079821489 | 16.5623742613 -0.1588181627
; Z0e (Even Mode) Z0o (Odd Mode)
;
; Gamma Matrix Eigenvectors:
; 0.70709735571 2.43871317e-7 0.70712912656 0.000000000000
; 0.70711620654 0.000000000000 -0.7070844351 -5.7766439e-7
;
;
; Gamma Matrix Eigenvalues:
; 0.28101510488 189.350366291 | 0.97887215574 102.080263771
; γe (Even Mode) γo (Odd Mode)
;
2000000000.0 : 6.15674302e-7 4.81156085e-7 6.15695041e-7
                 36.6622803867 4.23884026037 36.6646940705
                 3.4873268e-10 -1.417338e-10 3.4873269e-10
                 1.33833238e-6 -4.3751531e-7 1.33785617e-6

```

After the Sonnet header, the format of the RLGC data is provided. This is followed by RLGC data and comments for each analysis frequency. Each comment line begins with the “;” character. Though the comment lines can be ignored, they contain useful modal information. Each value in the comment section has both a real and imaginary part. Characteristic impedance ( $Z_0$ ), propagation constant ( $\gamma$ ), and the modal excitation vectors are provided for each mode.

## Broadband SPICE Model

---

**NOTE:** The Broadband Spice Model is only available 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 feature.

---

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.

To create a Broadband Spice file, you open your project in the response viewer and select *Output*  $\Rightarrow$  *Broadband Model File* from the main menu. (You may also create a Broadband Model by using the *Analysis*  $\Rightarrow$  *Output Files* command in the project editor. See Help for Details) This opens the Output Broadband Model File window which allows you to calculate a broadband SPICE file based on the analysis data for your project. You need a minimum of 50 frequency points in order to generate a Broadband Spice file; therefore, we highly recommend that you use an ABS sweep when analyzing your circuit to ensure the correct number of analysis frequencies. If your circuit contains parameterization or optimization data and you select more than one parameter or iteration combination, then you may choose to create one file which contains all the Broadband Spice Models or multiple files, one for each specified combination.

## Class of Problems

Be aware that there are several types of responses for which an accurate Broadband Spice Model may not be produced:

- If your response data contains a data point which sharply deviates from the data curve, such as you would see for a box resonance, or a narrow band spike, the Broadband Spice model may not accurately model that response.
- The Broadband Spice model is generally not accurate for response data below -60 dB.
- A gentle curve may sometimes get fitted with a straight line.
- Broadband Spice Extractor has only been tested for passive circuits.
- Broadband Spice Extractor has only been tested using S-parameters produced by *em*. However, you should be able to use S-parameters produced by other sources such as other simulators or measured data to create a Broadband Spice model.

If you are concerned with the accuracy of the model, you should visually inspect the predicted S-Parameter data, produced by the same rational polynomial which was used to create the Broadband model, to determine the usefulness of the Broadband Spice model.

---

NOTE:	Be aware that the processing time needed to create your models can be significant. The processing time is proportional to the number of analysis frequencies times the square of the number of ports in your circuit.
-------	---

---

## Creating a Broadband Spice Model

You use the response data created as the result of an *em* analysis to create a Broadband Spice model. For the best results use an Adaptive sweep (ABS) to analyze your circuit and produce response data spread evenly over the frequency band. The following procedure demonstrates the method to be used in the response viewer. For detailed instructions for setting up a Broadband Model file in the project editor, please refer to Help.

Once you have completed the *em* analysis of your circuit, do the following to create a Broadband Spice model:

- 1 Open your project in the response viewer.**
- 2 Select *Output*  $\Rightarrow$  *Broadband Model File* from the main menu of the response viewer.**

If you have more than one project open in the response viewer, and a project is not presently selected, a window appears which allows you to select the desired project. If there is only one project or one is presently selected, then the Output Broadband Model File dialog box is opened immediately.

- 3 Select the desired project from the project drop list.**

You may only create a Broadband Spice Model for one project at a time.

- 4 Click on the OK button to select the project and close the dialog box.**

The Output Broadband Model File dialog box appears on your display. You may access Help by clicking on the Help button or use context sensitive help for an explanation of a particular control or entry.

- 5 Select the file format of the Broadband Model: PSpice or Spectre from the Format drop list.**

PSpice and Spectre are the two formats of Spice file supported. A PSpice file uses the extension ".lib" and a Spectre file uses the extension ".scs." The PSpice file is compatible for use with HSPICE.

**6 Enter the desired name for the Spice model file in the Model File text entry box.**

A default filename is provided which places the Broadband Spice model file in the same directory as your source project. If you have selected multiple parameter combinations, a file is produced for each combination. If you wish to use another name or save the file in another location you may edit the text entry box.

**7 Enter the desired error threshold.**

The lower the error threshold you set, the more processing time is required to calculate the model. A lower threshold error also produces a more complex model, which requires more time to process in the Spice simulator. The error threshold is the error present between the source data and the fitted curve and is defined as follows:

$$Error = \frac{\left( \sum_{f=1}^N |s(f)_{Source} - s(f)_{Fit}| \right)}{N} \cdot 100$$

where  $f$  = the number of the frequency point

$N$  = the total number of frequency points

$s(f)_{Source}$  = the value of the S-parameter at the frequency point  $f$  in the project source response

$s(f)_{Fit}$  = the value of the S-parameter at the frequency point  $f$  in the fit curve data

The calculation of the Broadband Spice model stops when this error threshold is reached or when it proves impossible to improve the error. We recommend using the default value of 1.0% initially and not setting the threshold below 0.1%.

**8 Enter the desired stability factor.**

For more information about the stability factor, please refer to "Broadband Spice Extractor Stability Factor" on page 379.

**9 Click on the Advanced button.**

The Advanced Broadband Model options dialog box appears on your display.

**10 Select the Write data file checkbox if it is not already selected.**

This option is selected by default. When this option is selected, the Predicted S-Parameter data upon which the model is based is also output to a file. The name of the file appears below the checkbox and may not be changed. The file is created

in the same directory in which your model file is created. Once the creation of the model is completed, you may use the response viewer to compare your original response data to the Predicted S-Parameter data to evaluate the accuracy of the Broadband model.

By default, the predicted S-parameter data output from the model consists of the original analysis band, plus a band from the highest frequency to three times the highest frequency. This data is also used to check the stability of the model. You may override this default by selecting the Override default additional frequencies checkbox and entering the desired band.

**11 Click on the OK button to close the Advanced Broadband Model Options dialog box and apply the changes.**

Please note that the predicted data band is set by default and the output file is created by default, it is usually not necessary to open the Advanced Broadband Model Options dialog box.

**12 Click on the Extract button to create the Broadband Spice model.**

A progress window appears on your display. Be aware that the processing time needed to create your models can be significant. The processing time is proportional to the number of analysis frequencies times the square of the number of ports in your circuit. If you wish to stop the process before it is complete, click on the Cancel button in the progress window.

**13 Once the calculation is complete, the Broadband Spice details dialog box appears on your display.**

A log of the creation process appears in this dialog box. The log contains information on the error for each parameter, which parameter had the greatest error, and the filename of the predicted S-Parameter data. It will also indicate whether the model achieved the error threshold. Use this information to determine which parameters to examine in the response viewer. You should look at the S-parameter with the greatest error as well as any critical S-parameters whose error was greater than 0.1%.

For a detailed explanation of the log, see “Broadband Spice Extractor Log,” page 377.

**14 Click on the Plot button to open a plot of your response data versus the predicted S-Parameter data.**

The Predicted S-Parameter data is opened in the response viewer along with the original response data. You should view the S-parameter with the greatest error and any other critical response data whose error was greater than 0.1% to judge if the "fit" or accuracy of the model is sufficient for your needs.

If the model is not accurate enough, see “Improving the Accuracy of the Broadband Spice Model,” page 378 for suggestions on improving your Broadband Spice model.

## Checking the Accuracy of the Broadband Spice Model

It is possible to save the predicted S-parameter data created while calculating your Broadband Spice model so that you may visually check the accuracy of your model when it is complete. To save the predicted S-parameter data, you should select the **Write data file** checkbox in the Advanced Broadband Model Options dialog box when you create your model. Note that this checkbox is selected by default. The predicted S-parameter data file is created in the same directory as your Broadband Spice model file.

Upon the completion of creating your Broadband model, you should open the original project in the response viewer, then add the predicted S-parameter data file to your graph and compare the two responses. If you are creating your Broadband Model in the response viewer, you may do this automatically by clicking on the Plot button in the Broadband Spice Details dialog box which appears upon completion of your model. Use the log information in the Details window, which is detailed in the next section, to determine which parameter had the highest error and any critical parameters whose error was greater than 0.1%. Check these parameters to see how much the curve fit data varies from your circuit response.

If your Broadband Spice Model is to be used for a transient analysis, be aware that the frequency response of the model up to  $1/T$  where  $T$  is the minimum time step of the transient analysis is important. You should use the Advanced Broadband Model options dialog box to specify additional predicted data up to  $1/T$ . You access this dialog box by clicking on the Advanced button in the Broadband Model File entry dialog box in the project editor or in the Output Broadband Model dialog box in the response viewer. This allows you to view the frequency response of your model at data points not included in your **em** analysis. You should look for anomalies in the response that indicate a problem with the model, such as S-parameters greater than one or unexpected sharp resonances.

You may also use the stability factor in order to ensure a stable transient analysis. Please see "Broadband Spice Extractor Stability Factor" on page 379.

If the model is not accurate enough, see “Improving the Accuracy of the Broadband Spice Model,” page 378 for suggestions on improving your Broadband Spice model.

## Broadband Spice Extractor Log

The Broadband Spice Extractor log, displayed in the Broadband Spice details window, contains detailed information about the creation of your Spice model file. You may view a summary of the log or the complete log. To view the summary of the log, click on the Summary button at the bottom of the window. To return to the full log, click on the Complete button. The information in the log file is also included in the beginning of the output file.

You use the log to determine which parameters to examine in order to determine if the Spice model is accurate enough for your use. Two log files are shown below, the first log is for a model which achieved the error threshold and the second log is one in which the error threshold was not achieved on all the S-Parameters. A warning message is issued for all S-parameters whose error is greater than the error threshold.

```

Generating files cocross.lib and cocross_predict.snp.
--Model Log for cocross--
Data set has 397 points and 4 ports

--Model Options--
Error threshold(%) = 1           ← Error Threshold
Stability Factor: 0.001          ← Stability Factor
Output predicted file: C:\project\cocross_predict.snp
S Parameter fitting enabled.
Max order target: 40           ← Maximum Order
Using default predicted frequencies

--Model Results--
S11 Order = 2      Error(%) = 0.02663896
S12 Order = 2      Error(%) = 0.01073755
S13 Order = 2      Error(%) = 0.01566697
S14 Order = 2      Error(%) = 0.004835758
S22 Order = 2      Error(%) = 0.06025392
S23 Order = 2      Error(%) = 0.01543952
S24 Order = 2      Error(%) = 0.02405850
S33 Order = 2      Error(%) = 0.04248355
S34 Order = 2      Error(%) = 0.004751939
S44 Order = 2      Error(%) = 0.01848211

--Model Summary for cocross--
Maximum error was for S22, Error(%) = 0.0602539
Total model time: 0.158 seconds
Model cocross successful

```

Curve Fit Data File

Error Threshold

Stability Factor

Maximum Order

Error values for S-parameters

S-parameter with greatest error

Indicates that the Error Threshold was achieved for all S-parameters

```

Generating files matchnet.lib and matchnet_predict.snp.
--Model Log for matchnet--
Data set has 1246 points and 2 ports

--Model Options--
Error threshold(%) = 0.5 ← Error Threshold      Curve Fit
Stability Factor: 0.001                           Data File
Output predicted file: C:\Program Files\sonnet\project\matchnet_predict.snp
Max order target: 200                            → Warning Messages

--Model Results--
S11 Order = 203   Error(%) = 1.387911   WARNING: Error threshold
of 0.5 (%) not achieved
S12 Order = 208   Error(%) = 2.984112   WARNING: Error threshold
of 0.5 (%) not achieved
WARNING: Poor figure of merit on S12 parameter. Visual inspection
of predicted S12 recommended.
S22 Order = 214   Error(%) = 1.833756   WARNING: Error threshold
of 0.5 (%) not achieved
WARNING: Model prediction is not passive at 112 frequency points.
Error threshold may need to be decreased or input data may be non
passive.

--Model Summary for matchnet--
Maximum error was for S12, Error(%) = 2.98411
Total model time: 25 minutes 43 seconds
Model matchnet finished with no errors, 5 warnings

```

→ Indicates that a "fit" was found for all S-parameters but for some the error exceeded the error threshold.

## Improving the Accuracy of the Broadband Spice Model

If you need to increase the accuracy of your Broadband Spice model, there are several strategies you may use.

- If the Broadband model met your error threshold criteria but is still not acceptable, you may decrease the error threshold to increase the accuracy of the model. Be aware, however, that the processing time may be significantly increased by lowering the error threshold. Typically, values below 0.1% result in unacceptably long analysis times.
- If there are more than 200 frequency points in your response data, try decreasing the number of frequencies in your response data. To do so, use the *Analysis* ⇒ *Clean Data* command in the project editor to remove the response data, then run another Adaptive sweep (ABS)

using a coarser resolution to produce less data points but still more than 50 data points. You may change the resolution of an adaptive sweep in the Advanced Options dialog box in the project editor. Select *Analysis* ⇒ *Setup*, then click on the Advanced button in the Analysis Setup dialog box which appears.

- Increase the number of data points in the critical frequency band in which you are concerned and decrease the number of data points in frequency ranges which are not as important.
- You may be able to increase the accuracy of your model by using the Stability factor. Please see "Broadband Spice Extractor Stability Factor" in the next section.

## Broadband Spice Extractor Stability Factor

Although the extracted model may be a good fit to the S-parameters, a transient analysis which uses the model may be unstable. To help with this problem, Sonnet provides a control which pushes the poles away from the unstable region. However, pushing the poles too far could result in a less accurate fit usually by causing a decrease in the Q of the circuit. Therefore, you should only use this control if you are having stability problems with the model.

The Broadband Spice stability factor allows you to control the amount by which the model fitting forces the poles of your model away from the unstable region. The factor represents a magnitude ratio for which the real part of a pole cannot be less than the magnitude of the pole.

The higher the stability factor the greater the effect on the model. Reasonable values for the stability factor are between 1.0e-5 to 0.5; the default value is 1.0e-3. If you are having stability problems, raising this value may result in a stable model. The forcing done using the stability factor could result in a decrease in Q such that strongly resonant structures need a lower value, but unstable structures need a higher value to maintain their stability.

You input the stability factor in the Broadband Model File Entry dialog box, opened when you set up a Broadband Spice output file in the project editor (*Analysis*  $\Rightarrow$  *Output Files*) or response viewer (*Output*  $\Rightarrow$  *Broadband Model File*). You enter the stability factor in the Stability Factor text entry box as shown below.



## Inductor Model Extractor

---

### NOTE:

The Inductor Model Extractor (IME) feature is only available 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 feature.

---

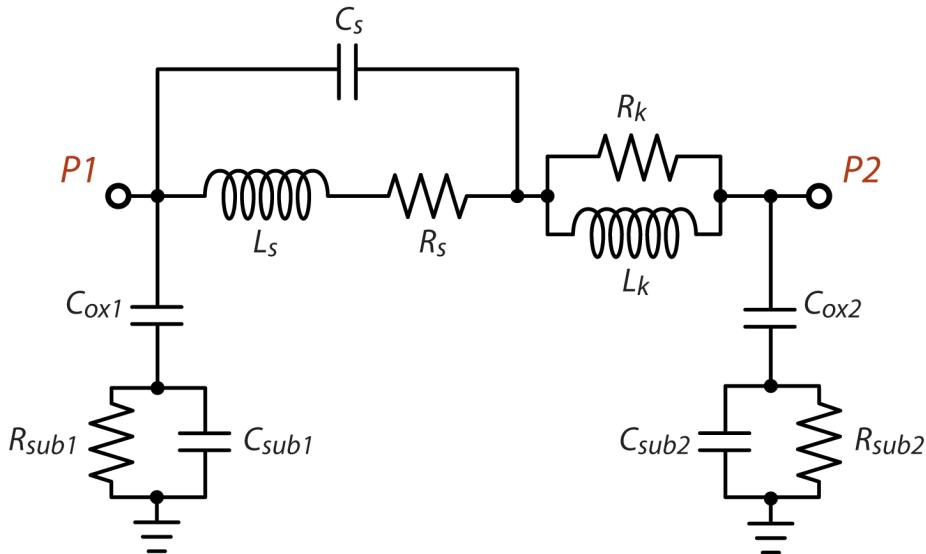
The Inductor Model Extractor (IME) feature allows you to extract an intuitive equivalent circuit model from the EM analysis of a spiral inductor.

### Model Topologies

There are two topologies available for the Inductor model: the “Untapped” model and the “Center Tapped” model. Both topologies generate a simple, physical, fixed-topology circuit model.

## Untapped Topology

Shown below is the schematic of the “Untapped” inductor model topology.

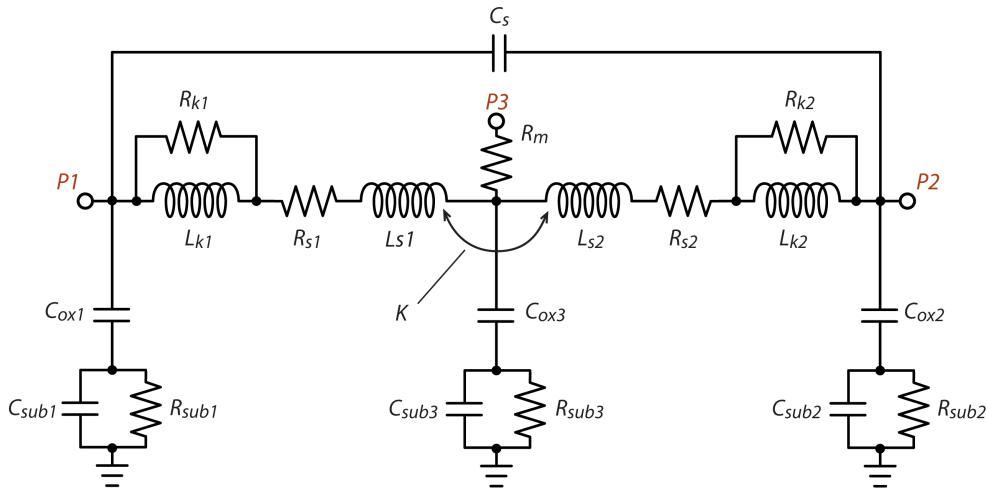


The Untapped model is appropriate under the following conditions:

- The inductor must have at least two ports. Ports 1 and 2 must be the input/output ports of the inductor. If you include any additional ports, the algorithm assumes that you will be connecting them to ground in your circuit simulator. Thus, you may connect ports to a patterned ground shield or ground ring if you wish.
- The analysis frequency band should extend from the frequency where Q exceeds unity to approximately twice the frequency at which the maximum Q-factor is found.
- This model extraction technique has proven most effective with spiral inductors on a conductive substrate, such as those usually found in RFIC processes involving lossy silicon substrates.

## Center Tapped Topology

Shown below is the schematic of the “Center Tapped” inductor model topology.



The Center Tapped model is appropriate under the following conditions:

- The inductor must have at least three ports. Ports 1 and 2 must be the input/output ports, and port 3 must be the center-tap port. If you include any additional ports, the algorithm assumes that you will be connecting them to ground in your circuit simulator. Thus, you may connect ports to a patterned ground shield or ground ring if you wish.
- The analysis frequency band must include the frequency where Q is maximum. In addition, it is recommended to do a wideband sweep to improve the model performance. Since this topology is not meant to be used above the self-resonance frequency, any data above the self-resonance frequency is ignored by the extraction algorithm.
- This model extraction technique has proven most effective with spiral inductors on a conductive substrate, such as those usually found in RFIC processes involving lossy silicon substrates

## Model Output

When the model is extracted, a netlist with the specified topology is written. The format of the netlist can be Spectre, PSpice, or Sonnet Netlist, and is specified when setting up your output file. You also have the option of requesting a Touchstone file with the s-parameter response of the extracted model. For more details, please see the following sections.

## Using the Inductor Model Extractor (IME)

You have two options of when to create this model. You may specify an output file in the project editor before your analysis, so that the inductor model output file is created automatically at the end of your analysis. It is also possible to create the output file after the analysis is complete when looking at your analysis response data in the response viewer. Both methods are explained below.

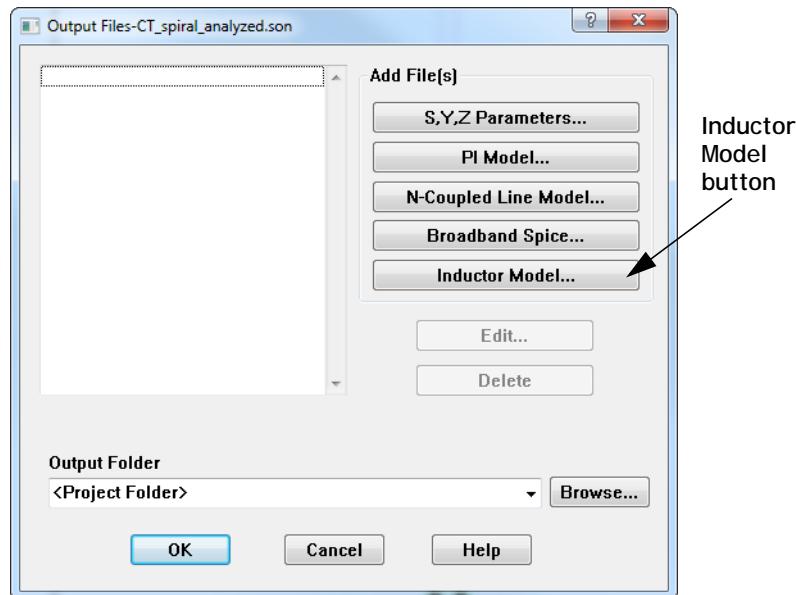
### Using the IME in the Project Editor

To setup your output file so that it is automatically generated as part of your EM analysis, do the following:

- 1 Open the desired project in the project editor.**

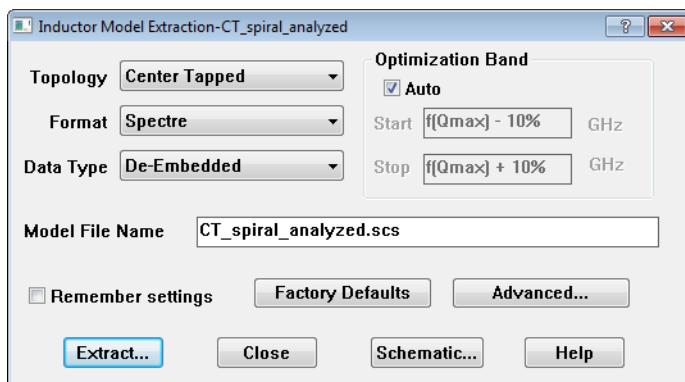
**2 Select Analysis  $\Rightarrow$  Output Files from the project editor main menu.**

The Output Files dialog box appears on your display, as shown below.



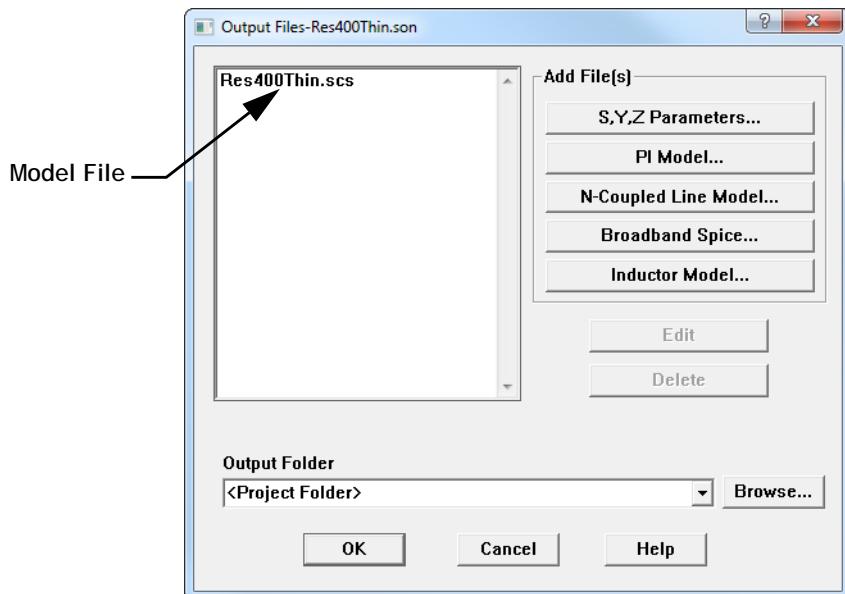
**3 Click on the Inductor Model button.**

The Inductor Model File entry dialog box appears on your display. This dialog box allows you to specify the desired topology and format of your output model file. You may also choose to output the S-parameter data file for the inductor model. This is useful if you wish to see how well the model output fits your original data. For information about the controls in this dialog box, please click on the Help button in the dialog box.



- 4 Once you have finished selecting your format choices, click on the OK button.

This closes the Inductor Model file dialog box and applies the changes. The Output Files dialog box is updated and the file you just specified now appears in the file list. An example is pictured below.



- 5 Click on the OK button in the Output Files dialog box.

This closes the dialog box and completes the specification of the model file. When you run your analysis, this output file is created and written to the location specified.

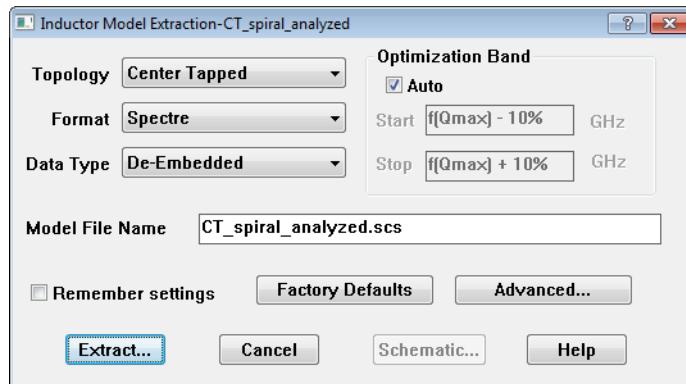
### Using the IME in the Response Viewer

To extract an inductor model from your analysis results after the analysis is complete, you may create the file using the response viewer. To create the file, do the following:

- 1 Open the desired project in the response viewer.

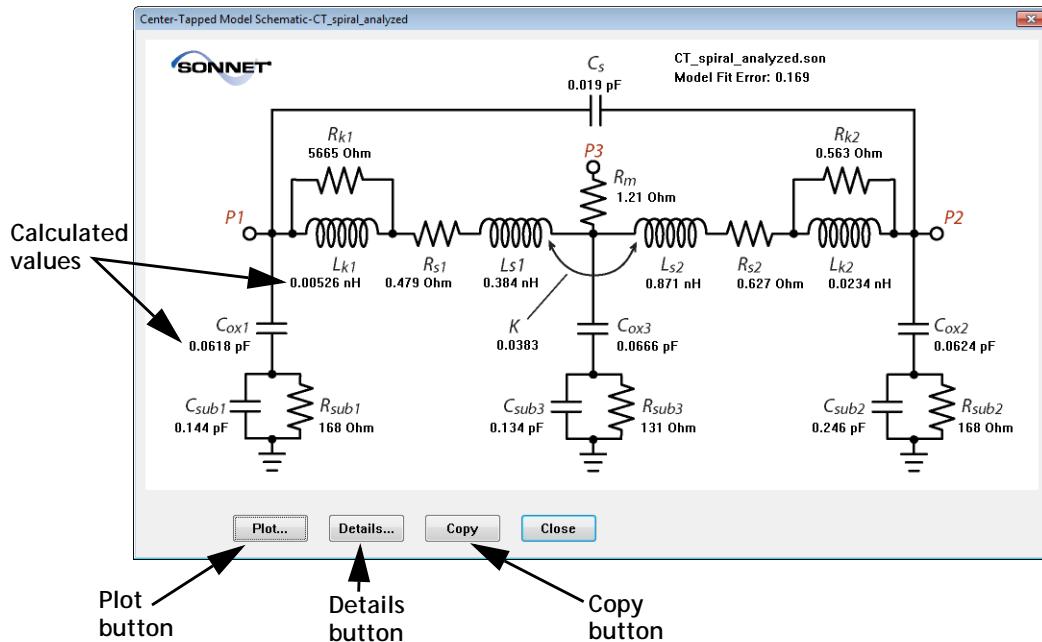
**2 Select *Output*  $\Rightarrow$  *Inductor Model File* from the response viewer main menu.**

The Inductor Model File dialog box appears on your display. This dialog box allows you to specify the format of your output model file. You may also choose to output the S-parameter data file for the inductor model. This is useful if you wish to see how well the model output fits your original data. For information about the controls in this dialog box, please click on the Help button in the dialog box.

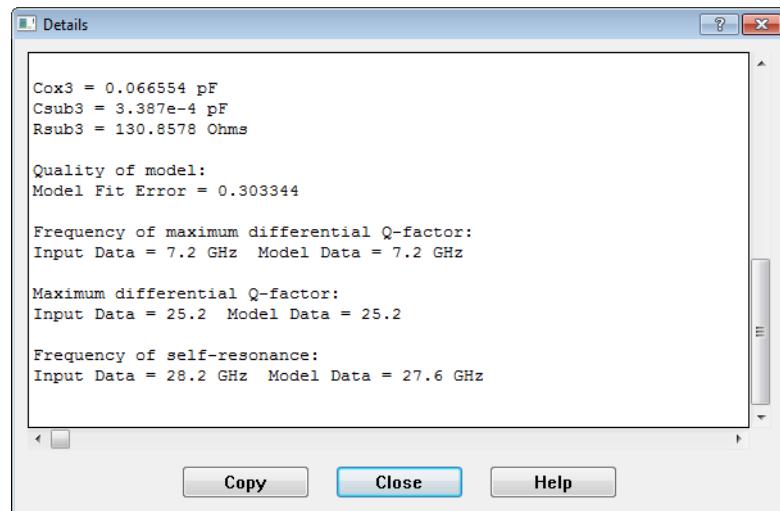


**3 Once you have finished selecting your format choices, click on the Extract button.**

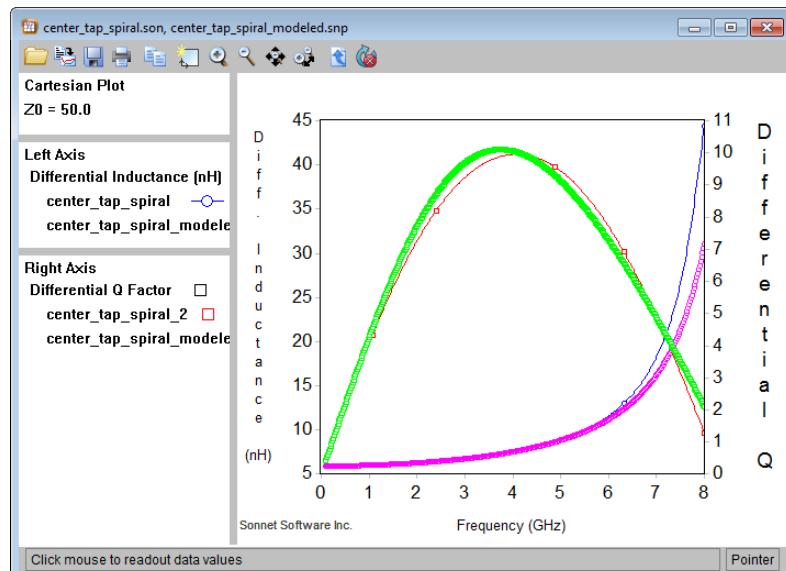
This closes the Inductor Model file dialog box and creates the output model file. The Schematic window appears on your display with the schematic for your topology displayed including the calculated values for your model. An example is shown below. You may copy the schematic to use in presentations by clicking on the Copy button (Windows only).



You may open the Details window by clicking on the Details button in the Schematic window. The Details window displays a report on the model created. You will need to use the scroll bar in the window to view the whole report. For details on the information displayed and options available in this window, please click on the Help button in the Details dialog box.



Clicking on the Plot button in the schematic window opens up a plot that shows the inductance and Q factor of both the original response data and the data generated using the extracted model so that you may compare them.



# Chapter 21

# Box Resonances

---

## Introduction

The Sonnet analysis engine, ***em***, is not limited to a purely TEM or Quasi TEM analysis. For a given structure, if a higher order mode (TE or TM) can propagate or an evanescent mode can exist, it will be included in the Sonnet results. These modes are sometimes called waveguide modes, cavity modes, or box modes. If the cavity dimensions are large enough, these modes can resonate, causing spikes (abrupt changes) in your measurements. Since ***em*** is a full-wave analysis engine, these resonances will also be present in your Sonnet results. Thus, Sonnet can be used to predict resonances inside rectangular packages and module housings.

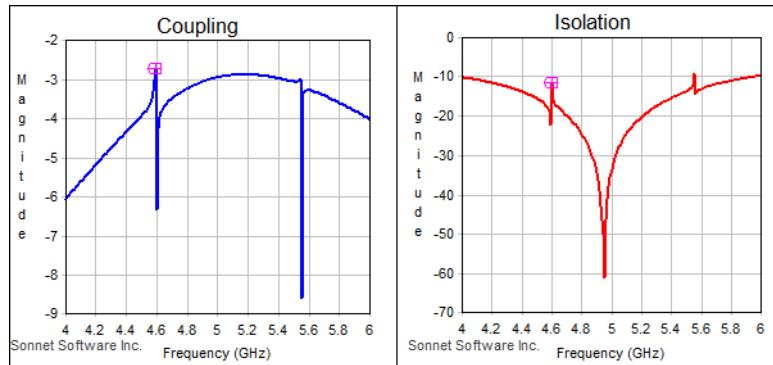
In this chapter, we will outline several ways to detect and remove box resonances within the Sonnet simulation. As you will see, this allows you to evaluate a package design early in the design cycle.

We will also discuss how to handle situations where your real circuit is not in a metal box, but the Sonnet simulation shows resonances because of the ***em*** analysis box which is not present in the real circuit. In other words, the simulation environment is introducing resonances which are not present in the real circuit.

# Initial Indicators of a Box Resonance

There are several typical indicators of a box resonance:

- Suspicious "spikes":** A box resonance usually shows up as a spike (i.e., a rapid change) in your response. Shown below are S-parameter plots of a coupler with box resonances, but you may also notice spikes when plotting magnitudes or phases of S, Y, Z-parameters, or equations, such as inductance, capacitance and/or Q-factor.

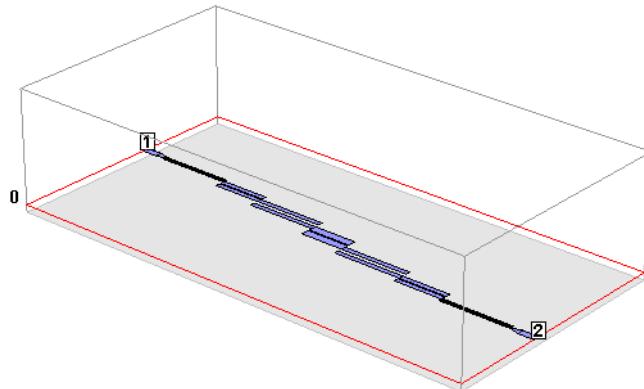


- Large number of discrete frequencies required for an ABS sweep:** Adaptive Band Synthesis (ABS) works best when your circuit does not have box resonances. When your circuit contains box resonances, ABS will still give accurate results, but may require a large number of points around the resonance. If you find your analysis is requiring an unusually large number of discrete points, then box resonances could be the cause.
- Gain in your response:** As mentioned in the de-embedding chapter of this manual, de-embedding requires the analysis of calibration standards. If one of these calibration standards has a box resonance, then the de-embedding results could be inaccurate at the resonant frequencies. Often, this shows up as gain in the results, even though you are analyzing a passive circuit.
- Two or more large box dimensions:** A good rule of thumb for determining the lowest resonant frequency for a rectangular box is to calculate the frequencies where the smallest two dimensions of your box are a half wavelength. The lower of these two frequencies is often fairly close to the lowest box resonance frequency. So, if your box has two dimensions larger than a half wavelength, and you have not done anything to prevent box resonances, box resonances will probably occur.

## Bandpass Filter Example with Resonances

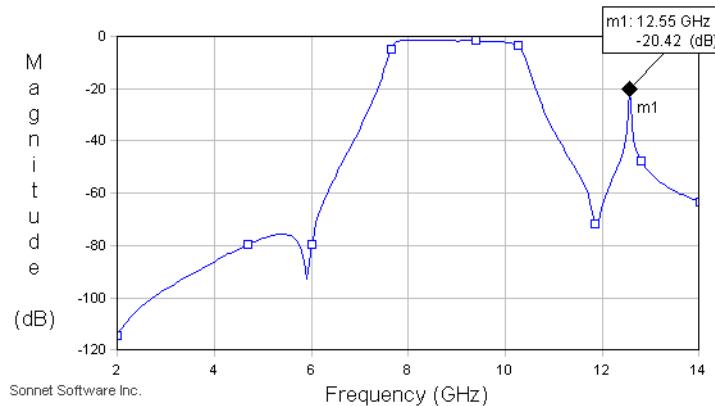
This chapter uses a bandpass filter as an example circuit which is suspected to have box resonances, and shows how to determine if these resonances are caused by the box. Various techniques are discussed to remove unwanted box resonances. The bandpass filter and the variations mentioned in this chapter can be found in the Sonnet example files under the name “box\_resonance.” You may access the Sonnet Example Brower by selecting *Help □ Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Brower, please click on the Help button in the Example Brower window. Be aware that this example contains multiple files.

The filter, “Filter\_original.son,” is shown below. It is a microstrip edge coupled filter which has been placed in a box which is large enough to support at least one box resonance in the band of interest.



## Is It a Box Resonance?

Below is an S21 plot for the example project “Filter\_Original”, included in the box resonance example set. Notice that there is a sharp change, or “spike”, in the data at about 12.55 GHz.

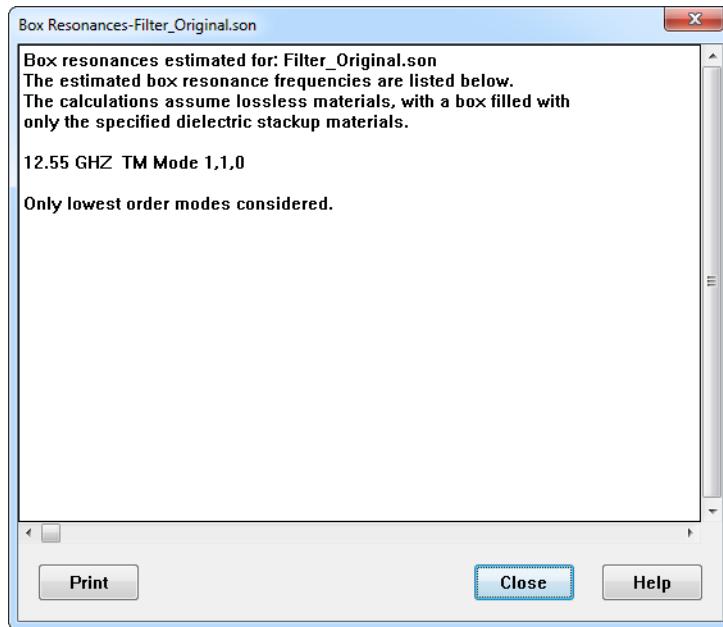


But not every resonance in your response is necessarily a box resonance. For example, it is possible that there could be resonances as part of the filter response. The next section shows the simplest way to check your circuit for box resonances.

## The Box Resonance Estimator

The quickest way to understand the box resonance situation within your circuit is to use the Box Resonance Estimator tool before running an analysis. Routinely using this tool before running a simulation can prevent a lot of wasted simulation time. It is an extremely useful tool because it allows you to make modifications to the box size and/or dielectric properties of your circuit and instantly gauge its impact on box resonances. The Estimator can also be used after a simulation is complete to help determine which characteristics of a complex data curve are related to box resonances. The results of the tool are theoretical values based on lossless materials, with the circuit metal removed. For most circuits, this is a good assumption.

To access the Box Resonance Estimator, select *Analysis □ Estimate Box Resonances* from the project editor main menu. The Box Resonances dialog box appears on your display. The result for our example circuit is shown below.



Notice the estimated frequency of the first box resonance is close to the frequency of the resonance in the S-parameter response. Therefore, we can conclude that the cause of the resonance is a strong box resonance which has induced coupling between the input and output ports at the resonant frequency.

The Box Resonance Estimator displays not only the resonant frequencies contained in the simulation frequency range, but also tells the user the particular mode type. It is sophisticated enough to realize that when Symmetry is enabled for a circuit that Even Y modes will not exist. For a complete description of propagation modes, please refer to any classic field theory textbook.

The Box Resonance Estimator is nearly instantaneous because it ignores your circuit metal and only considers the dielectric stackup. In addition, it assumes the following:

- **Lossless materials:** All six walls of the analysis box are considered lossless. All dielectrics are lossless.

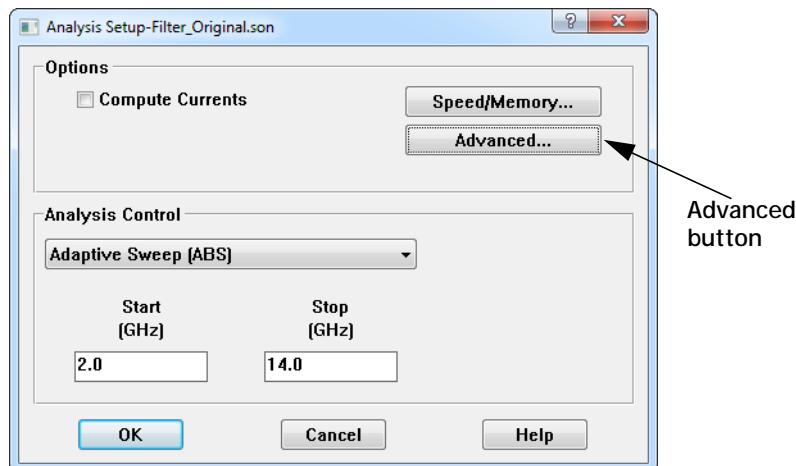
- **Metal that creates a smaller box is ignored:** As mentioned above, the Estimator ignores circuit metal; therefore, even if you have a plane of metal that divides the box into two sections, it will be ignored. This could occur in a Coplanar Waveguide (CPW) or slotline circuit where a large portion of the layer is metalized.
- **Vias ignored:** Sometimes vias can be used to eliminate box resonances. However, since circuit metal is ignored by the Estimator, the vias have no affect on the estimated box resonances.
- **De-embedding ignored:** The Box Resonance Estimator only checks the primary structure, not the de-embedding calibration standards. To determine if a box resonance is caused by a de-embedding calibration standard, see the section "Runtime Warning Messages" below.

## Runtime Warning Messages

You may use the Box Resonance Info run option to estimate box resonances and output warning messages in the analysis monitor during the simulation. To use this option, do the following:

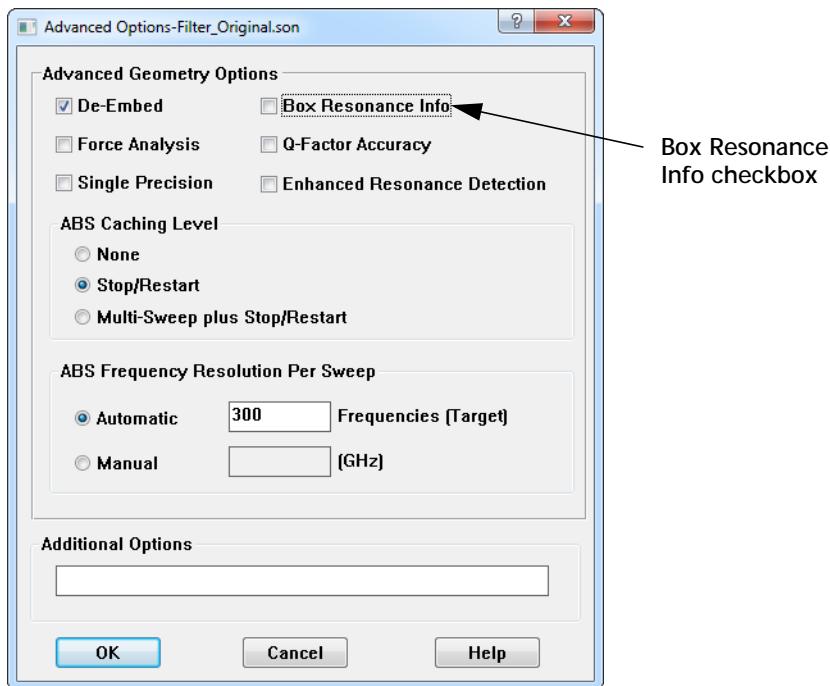
- 1 Select *Analysis □ Setup* from the project editor main menu.**

The Analysis Setup dialog box appears on your display.



**2 Click on the Advanced button in the Analysis Setup dialog box.**

The Advanced Options dialog box appears on your display



- 3 Click on the Box Resonance Info check box in the Advanced Options dialog box**
- 4 Close both the Advanced Options dialog box and the Analysis Setup dialog box by clicking on the OK button to close the dialog box and apply the changes.**



The warning symbol, shown to the left, appears in the analysis monitor when a box resonance is detected. If you click on the Errors/Warning button in the analysis monitor, you can view all of the warning messages associated with that particular analysis.

Below is an example of the first type of box resonance warning message from an analysis of the project. When this message appears there is a box resonance detected in the primary structure.

```
Sonnet Warning EG2680:  
    Circuit has potential box resonances.  
    Circuit: primary structure  
    The estimated box resonance frequencies are listed below.  
    The calculations assume lossless materials, with a box filled with  
    only the specified dielectric stackup materials.  
    12.55 GHZ  TM Mode 1,1,0  
    Only lowest order modes considered.
```

Note that the warning message is describing box resonances which appear in the primary structure. The term “primary” refers to the actual structure being analyzed (as opposed to a de-embedding calibration standard). As with the Box Resonance Estimator, the specified dielectric stackup is considered, but the effect of any circuit metallization and loss parameters are not.

Keep in mind that this option invokes the same algorithm as the Box Resonance Estimator. However, it also checks for resonances in the de-embedding calibration standards. If it detects a resonance in one of the calibration standards, a similar warning message to the one above will appear except the “Circuit” line will name the calibration standard. For example, if one of the calibration standards used to de-embed ports on the left box wall is detected, the line will look something like this:

```
Circuit: left box wall SOC magnetic wall standard
```

For more information on the calibration standards used for de-embedding, please refer to ["Calibration Standards" on page 114](#).

## Removing artificial box resonances

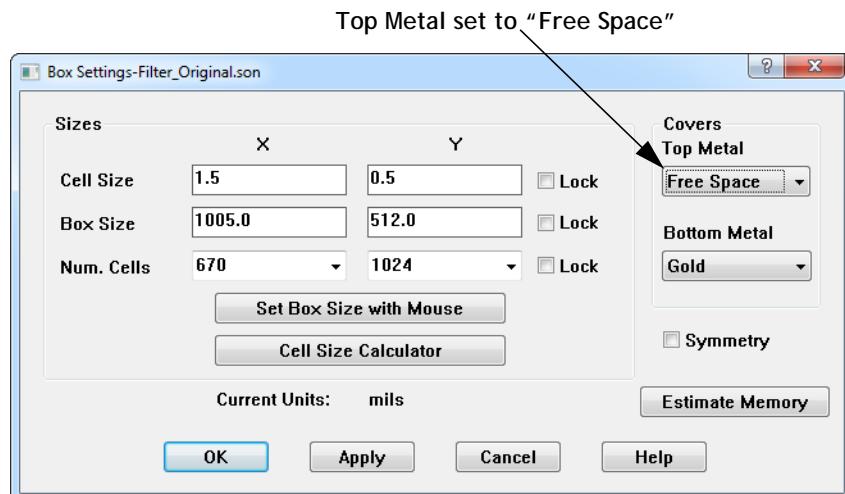
Sometimes you want to see the response of your circuit in an unshielded environment. In those cases, any resonances caused by the analysis box are considered an artifact of the simulation and do not represent what you would measure. This section shows several methods for removing, attenuating, or moving these artificial resonances out of band.

## Change box size

One way to remove a box resonance is to change the size of the box - making it either larger or smaller - to move the resonant frequency out of band. If the problem occurs in de-embedding, you may be able to change the length of the calibration standard in the project editor to move the box resonance out of the band of interest. See ["Modifying Calibration Standard Lengths" on page 118](#) for more information. The Box Resonance Estimator, discussed earlier, can be used to help you find the proper dimensions which will move the resonances out of band.

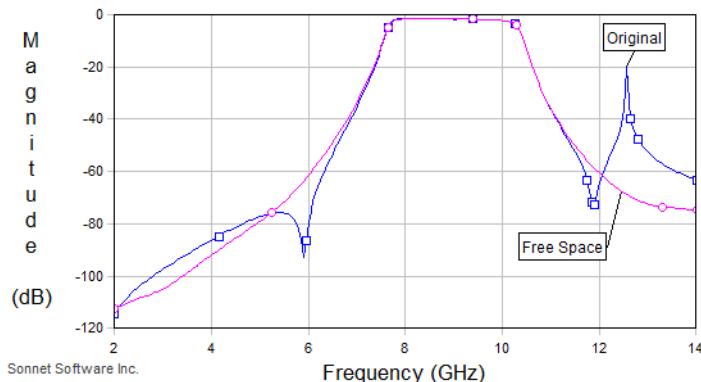
## Use a free space cover

Another simple way to remove, or at least attenuate, a box resonance is to remove the top or bottom cover. We can create an approximation of this condition by setting one of the covers to “Free Space” (a resistivity of 377 ohms/square). To do this using Sonnet’s project editor, open the Box Settings dialog box (*Circuit □ Box*) and change the top or bottom cover to “Free Space” by selecting “Free Space” from the Top Metal or Bottom Metal drop list. If you are using an interface to Sonnet, please refer to the manual for your interface for information on changing the top or bottom box metal.



This is an accurate approximation provided the cover is not so close that it interacts with the evanescent fringing fields surrounding the circuit. Typically, the cover should be separated from your circuit by a layer of air that is about a half-wavelength thick. An example of this is shown in the example project,

"Filter\_FreeSpace.son." Shown below is a plot of the original filter (blue) and the filter with a free space top cover (magenta). Notice the resonance is no longer present in the circuit with the free space top cover.



As discussed earlier, the Box Resonance Estimator calculates resonances based on lossless covers. Therefore, when your circuit has a free space cover, the Box Resonance Estimator gives a warning message that the reported resonances may be partially or entirely suppressed:

```
*****
The circuit has a free space cover.
The resonant modes may be partially or entirely suppressed.
*****
```

12.55 GHZ TM Mode 1,1,0 ← Resonances based on  
lossless covers

However, notice that below the warning message, the resonances based on lossless covers are still shown. These frequencies should typically be ignored, but are listed so you know where the resonances would have been if the top cover was not set to free space. Thus, you should not use the Estimator to determine if the cover suppressed your box resonances. Instead, you should simulate your circuit, or use the technique described in "Using a Probe Circuit for a Quick Analysis" on page 403.

# Suppressing Real Resonances

In the preceding section we mentioned some simple techniques for removing resonances that were an artifact of the simulation. But what if you need to put your circuit in a large box that supports box resonances? In this case, if you do not fix the problem, you will have unwanted resonances in your real circuit.

RF and microwave designers have developed many techniques for suppressing unwanted resonances in their package. Sonnet can be an invaluable tool by allowing you to test these techniques and evaluate the impact on box resonances. This section describes just a few of those techniques and how to implement them in Sonnet.

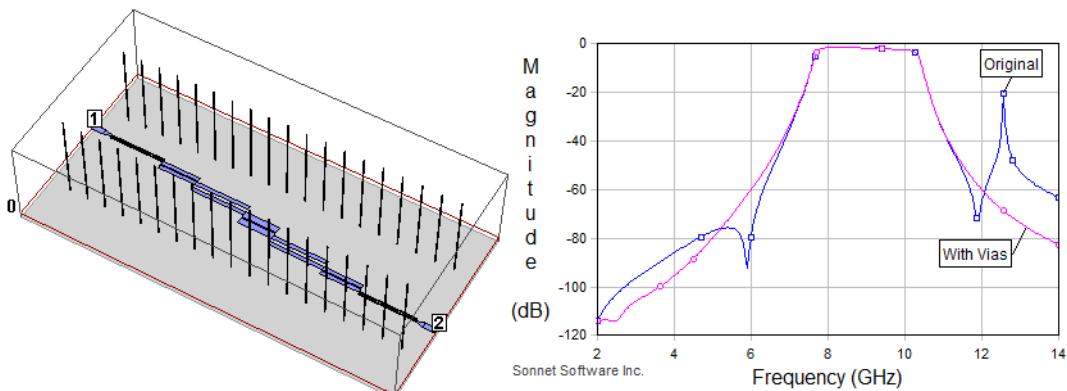
## Changing the Box Size

As noted earlier, decreasing the box size usually moves the box resonances higher in frequency and the Box Resonance Estimator may be used to determine if a simple change in one of the box dimensions will move the resonance out of band. However, moving the box walls too close to the circuit could change the circuit response. We recommend first changing the box dimensions, then checking the new dimensions with the Box Resonance Estimator. Repeat these two steps until an acceptable box size is found. Then, analyze the circuit with the new dimensions to determine your new response.

## Adding Vias

In many cases, ground vias or ground planes will help prevent box resonance modes from forming. In simple terms, they have the effect of dividing the structure into smaller compartments, thus pushing the box resonances higher in frequency. The use of a via fence or CPW type transmission line can help “channelize” the circuit, preventing unwanted coupling between traces and reducing unwanted box resonance effects. All of these features can be included in the Sonnet model.

Shown below is our example filter with vias connected from the top to the bottom of the box (see example project “Filter\_Vias.son”). The plot on the right shows the response of the original filter (blue curve) and the new response with the vias (magenta curve). Notice that the vias have eliminated the box resonance.

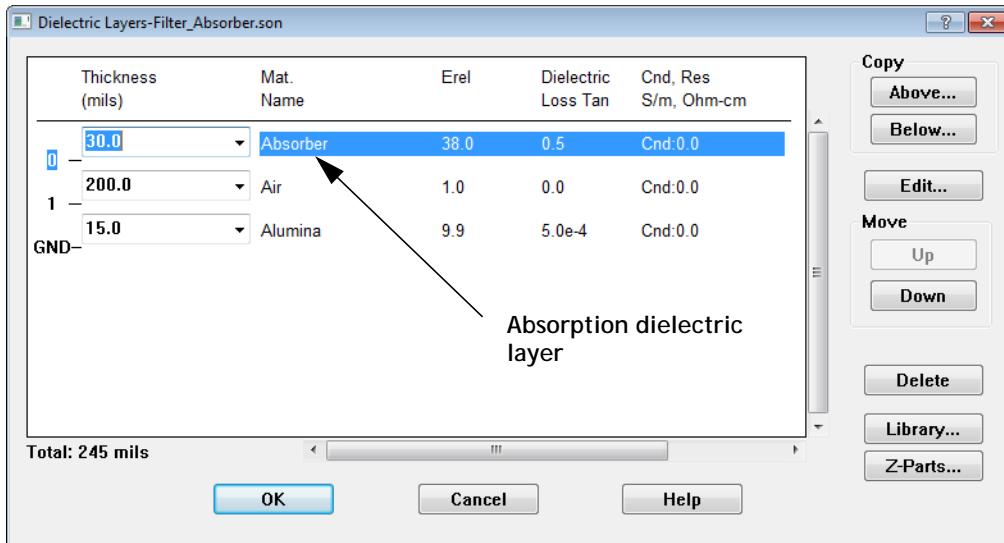


It should also be noted that the Box Resonance Estimator will still predict the box resonance even though the circuit no longer has a resonance. This is because the Estimator ignores the vias in the circuit. To determine the effect of the vias on box resonances, you will need to simulate your circuit, or use the technique described in the "Using a Probe Circuit for a Quick Analysis" on page 403.

## Adding Absorbing Material

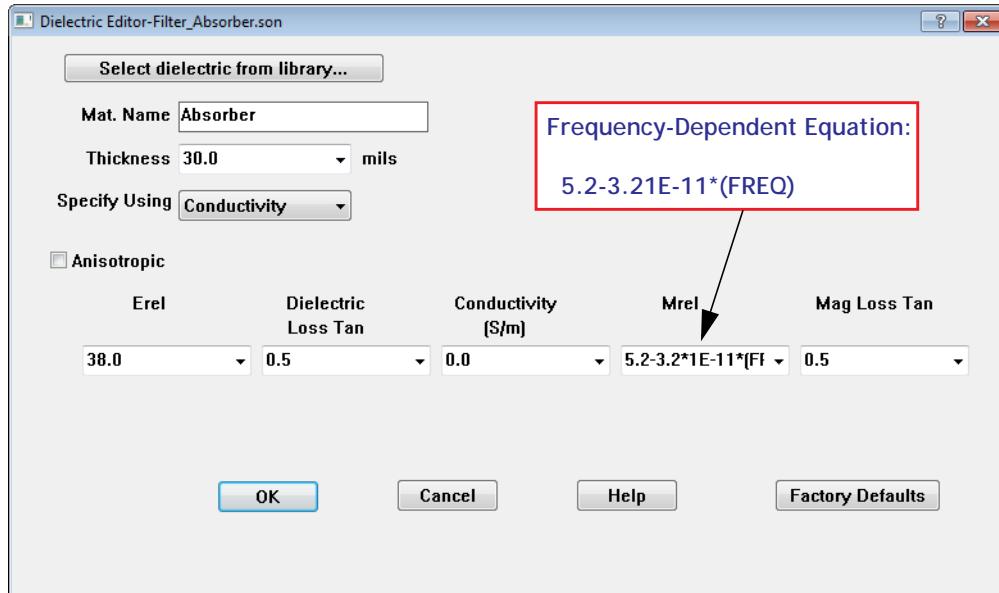
Another approach used by engineers to eliminate box resonances is to add an absorbing material in the housing cavity. The material is normally iron or carbon loaded so it can provide a fairly high magnetic or electric loss tangent. It comes in various forms such as liquid or sheet and is usually placed at a position such that it has a minimal effect on the circuit performance and a large effect on the box resonance. If the material is placed across the entire XY plane, you can easily model its effects in Sonnet by adding a dielectric layer to the stackup. For example, the

example file “Filter\_Absorber.son” has an absorbing material attached under the top lid of the box. To do this in Sonnet, simply add an extra dielectric layer which has the properties of the absorbing material.



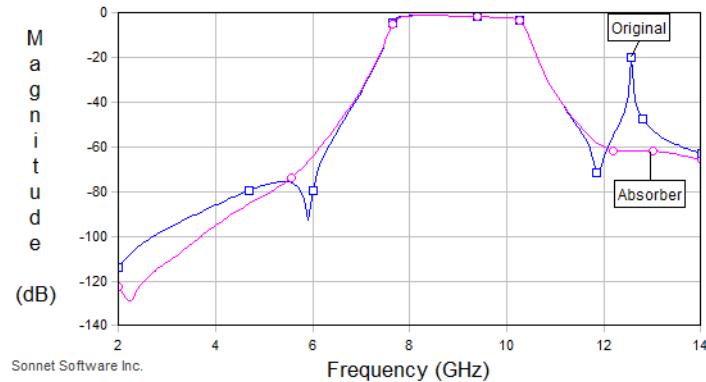
You will need to obtain the properties of the material from the supplier. If the values for the permittivity, dielectric Loss Tangent, permeability, or magnetic loss tangent are a function of frequency, then you will either need to analyze over a

narrow band where the values do not vary much, or use a frequency-dependent equation. The example project “Filter\_Absorber.son” contains a frequency-dependent equation for Mrel.



To learn more about using equations to represent dielectric properties, see the “Equation Syntax” topic in Help (enter “Syntax” in the index tab).

Shown below is our example filter with the absorber material at the top of the box. The plot shows the response of the original filter (blue curve) and the new response with the absorber (magenta curve). Notice that the absorber has eliminated the box resonance.

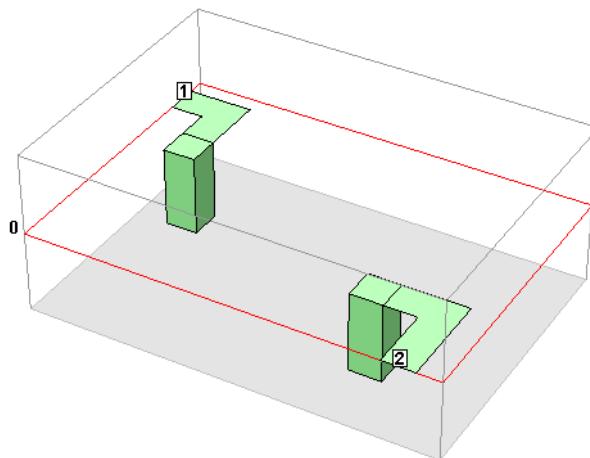


It should also be noted that the Box Resonance Estimator will still predict the box resonance. This is because the Estimator ignores losses in the dielectrics. To determine the effect of the absorber on box resonances, you will need to simulate your circuit, or use the technique described in the "Using a Probe Circuit for a Quick Analysis" section below.

## Using a Probe Circuit for a Quick Analysis

The following section describes how to run a quick analysis of your box without having to analyze the real circuit. This can be useful if you wish to quickly try out different techniques for suppressing resonances (either real or artificial). To create a project which does not require a lot of processing time, start with your original circuit, keeping the box size and dielectric properties unchanged, but delete all of the metal. Then set the cell size to be about 1/20 wavelength at your highest frequency.

Once you have deleted your original circuit, you add two small “probes” to excite and detect box resonances. The probes are just small (one or two cells long) shorted lines with a port on one end and a via on the other. The objective is to have metal which has current in the X, Y, and Z directions, so it will excite and detect all possible modes. Add a bend so you are creating both X and Y currents, and use a via to create the Z-directed current. During a resonant situation, the coupling will increase between the two probes.

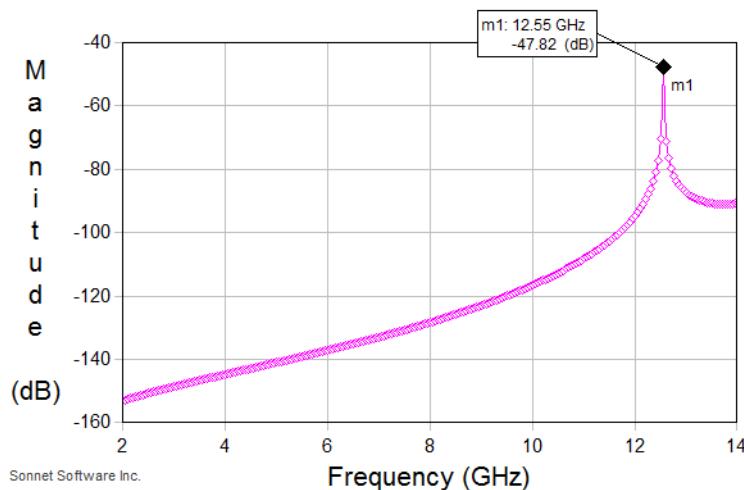


Sweep the frequency in fine steps and look for resonances. We recommend using a linear sweep because an ABS sweep could potentially miss a narrow band resonance. You should run the analysis without de-embedding since de-embedding as-

sumes there are no box resonances which can cause erroneous results. When the box resonates, there should be stronger coupling between your two probes. This can easily be seen by plotting the magnitude of S21.

Since there is no longer any circuit metal, except for the two small probes, any resonances in the results must be box resonances. Now that you have a simple circuit which shows the box resonances, you can try different techniques, such as those mentioned above, to see if any given technique will suppress the box resonance(s).

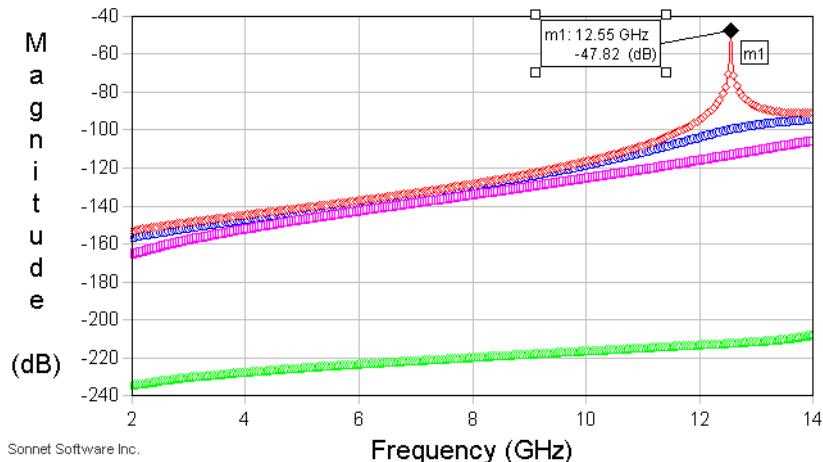
Shown below are the results of using the above technique on our filter example (see example project “Probe\_Original.son”). Notice the strong coupling at about 12.55 GHz, the frequency that was predicted to have box resonances:



There are two advantages of this approach:

- Since the circuit is so simple, you know that any resonance you see is a box resonance.
- The analysis is fast, so you can quickly try many different variations.

Shown below are the results of probe versions of all of the techniques discussed above. The original probe circuit is also included for comparison purposes. Notice that all of the techniques suppressed the box resonance.

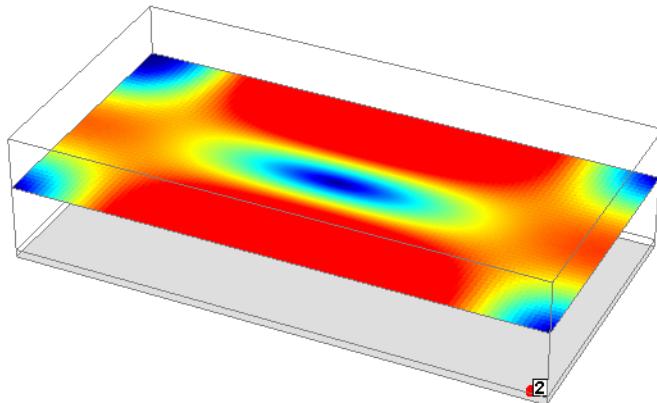


The above technique assumes that you are investigating a resonance in the primary structure. If you want to investigate a resonance in a calibration standard, you will need to change the box length to be the same length as the calibration standard length. The calibration standard length is reported in the log files and can be accessed by selecting *Project □ View Log □ Timing Info*.

## Viewing Electric Fields

Viewing the fields in the box at the resonant frequency can often give you an intuitive understanding of the resonance, helping you to find an appropriate method to suppress the box resonance. Sonnet allows you to view the tangential electric

fields at any XY plane in your box. Below is a picture of the electric fields in an XY plane which is 100 mils from the top of the box. This plot was generated by adding a sense layer to the probe project (see “Probe\_Sense.son”).



The frequency was chosen to be 12.55 GHz, which is the frequency where the Box Resonance Estimator predicted a  $\text{TM}_{110}$  mode box resonance. For further information on using a sense layer to view the tangential electric fields in your circuit, see “View Tangential Electric Fields” under Tips and App notes in Help.

## Summary

Indications that you may have a box resonance include:

- Seeing spikes in your response.
- Seeing gain in your response.
- An ABS sweep requires a large number of discrete frequencies.
- Any combination of the above items.

The techniques discussed in this chapter can be used to first determine if you have a box resonance, and then to remove the box resonance from either your simulation or your real measurement, or both.

# Chapter 22

# Accuracy

# Benchmarking

---

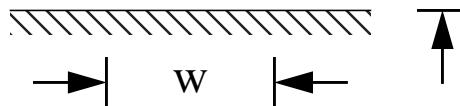
Electromagnetic analyses are often described as providing what is called “Good Agreement Between Measured And Calculated” (GABMAC). However, in the past, there has been little effort to decide just what “good” means. The more useful result is the “Difference Between Measured And Calculated” (DMAC).

There is an example of a coupled stripline benchmark available in Help under Applications.

## An Exact Benchmark

What we need to calculate DMAC is an exact benchmark. One source of an exact benchmark is stripline. The characteristic impedance of a stripline has an exact theoretical expression  $K(k)$  and is the complete elliptical integral of the first kind.

For evaluation on a computer, a polynomial for  $K(k)$  is available in **Abramowitz and Stegun, Handbook of Mathematical Functions**, pp. 590 - 592. (Be sure to note the errata,  $m_1 = 1-m_2$ , not  $1-m$ ):



$$Z_0 \sqrt{\epsilon_r} = \frac{n_0 K(k')}{4 K(k)}$$

$$k = \tanh\left(\frac{\pi w}{2 b}\right) \quad k' = \sqrt{1 - k^2}$$

$$n_0 = 376.7303136$$

The expression for  $K(k)$  cited above provides an accuracy of about  $1 \times 10^{-8}$ . When programmed on a computer, the following values are obtained for three different transmission line impedances (unity dielectric constant):

**Table 1 Stripline Benchmark Dimensions**

Z <sub>0</sub> (ohms)	w/b
25.0	3.3260319
50.0	1.4423896
100.0	0.50396767

For a length of stripline, there are two parameters of interest: characteristic impedance and propagation velocity. With the w/b given above, we know the exact answer (to within  $1 \times 10^{-8}$ ) for Z<sub>0</sub>. With a dielectric constant of 1.0, we also know the exact answer for the propagation velocity. It is the speed of light, known to about  $1 \times 10^9$ . Any difference from these values is error, or, DMAC.

Each of the above three benchmarks is available in the Sonnet example stripline\_benchmarks. The example may be obtained using the Sonnet Example browser. You may access the Sonnet Example browser by selecting *Help*  $\Rightarrow$  *Browse Examples* from the menu of any Sonnet application. For instructions on using the Example Browser, please click on the Help button in the Example Browser window. Save the file to your working directory.

The “b” dimension is exactly 1.0 mm, the “w” dimension is given by the above table and the length of each line is 4.99654097 mm with a dielectric constant of 1.0. Each of these lines is precisely 0.25 wavelengths long at 15.0 GHz. The geometry projects have the subsectioning set so the lines are 16 cells wide and 128 cells long.

To evaluate DMAC, do an analysis of the line at 15 GHz, with de-embedding enabled. For the error in characteristic impedance, take the percent difference between the calculated value and the exact value, above. For the error in propagation velocity, take the percent difference between the calculated  $S_{21}$  phase and -90 degrees. Total error, in percent, is the sum of the two errors.

Some types of analyses do not calculate characteristic impedance. A detailed error analysis shows that, to first order for a 1/4 wavelength long 50 ohm line, the value of  $|S_{11}|$  is equal to the error in characteristic impedance. For example, an  $|S_{11}| = 0.02$  means that there is about 2% error in characteristic impedance. To use this approximation for, say, a 25 ohm line, the S-parameters must be converted to 25 ohm S-parameters. This may be done by adding transformers in a circuit theory program.

## Residual Error Evaluation

We have performed a detailed analysis of the relationship between subsectioning and residual error (DMAC). The simplest way to subsection a line is to use subsections the same width as the line. In Sonnet, and in many other analyses, this results in a uniform current distribution across the width of the line. In reality, the current distribution is singular at the edges of the line.

Since the current distribution is symmetrical about the center line, using either one or two subsections across the width of the line gives the same amount of error.

We find that a one or two subsection wide line gives 5% to 6% error. If there is not much stray coupling, circuit theory can often give a better result. When the line is 16 cells wide, we see about 1% error, much more reasonable. We have found (and you can verify) that convergence is very strong: Double the number of cells per line width and the error is cut in half.

When we vary the number of cells per wavelength, along the length of the line, we see an inverse square relationship. Double the number of cells per wavelength along the length of the line and the percent error decreases by a factor of four.

An equation which expresses the error as a function of subsectioning is:

$$E_T \approx \frac{16}{N_W} + 2\left(\frac{16}{N_L}\right)^2 \quad N_W \geq 3 \quad N_L \geq 16$$

where

$N_W$  = Number of cells per line width,

$N_L$  = Number of cells per wavelength along line length,

$E_T$  = Total Error (DMAC) (%).

This equation estimates subsectioning error only. For example, any de-embedding errors are added to the above error. This error estimate should be valid for any electromagnetic analysis which uses roof-top subsectioning.

Notice that the quantities used for the error estimate are in terms of cells, not subsections. Cells are the smallest possible subsections size. In Sonnet, subsections in the corners of polygons are one cell on a side. Subsections along the edge of polygons are one cell wide and can be many cells long. Interior subsections can be many cells in both dimensions.

We have found that, for most cases, the cell size is the important parameter in determining error. Or in other words, the smallest subsection size is important. For example, the stripline benchmark geometry projects, mentioned before, are set to make the lines 16 cells wide, even though those 16 cells may be merged into only 4 or 5 subsections. It is the 16 cells which determine the level of error, not the 4 or 5 subsections.

In performing this error evaluation, we also found that the error in characteristic impedance due to  $N_W$  is always high, never low. Also, there is very little variation in the error for different impedance lines. The above equation can be very accurate in evaluating error. And, finally, for  $N_L$  above about 40 cells per wavelength, all the error is in the characteristic impedance. The error in velocity of propagation is essentially zero.

The above equation can be very accurate in evaluating error. With this precise knowledge of the error, we can now do something about it.

## Using the Error Estimates

The above error estimate can be used to estimate the error for an overall circuit. Let's say that a cell size is used that makes some high impedance transmission lines only 1 cell wide. Other, low impedance transmission lines, are, say, 30 cells wide. The 1 cell wide lines give us about 5% error. The 30 cell wide lines give about 0.5% error. In non-resonant situations, you can expect the total error to be somewhere between 5% and 0.5%. If most of the circuit is the low impedance line, the error is closer to 0.5%, etc.

However, let's say that our circuit has resonant structures. Let's say it is a low pass filter. It is easy to verify by means of circuit theory that the low pass filter is very sensitive to the high impedance lines. This means we can expect about 5% error, even though the high impedance lines only make up half the filter.

Given this information, there are several courses of action. First, if 5% error is acceptable, no further effort is needed.

More likely, we wish to analyze the filter with less error. Since we now know the error in the characteristic impedance is 5%, we can physically widen the line so that the characteristic impedance is 5% lower to compensate for the known increase in characteristic impedance due to subsectioning the line only one cell wide. Very precise analyses are possible using this compensation technique.



# Appendix I

## *Em and xgeom* Command Line for Batch

---

### ***em* Command Line**

If you wish to set up batch or script files to run your analyses overnight or at times of the day when the processing load is lighter, it is possible to use command lines to run *em* from a batch file. You should also be aware that it is possible to setup batch files with start and stop times using the analysis monitor. For directions on how to do so, please see **How do I create a batch file to run multiple analysis jobs?** in online help.

You should be aware that running from the command line does not provide all of the status information that is provided in the analysis monitor while running an analysis.

The syntax of the command line is as follows:

**em -[options] <project name> [external frequency file]**

where:

**<options>** is one or more of the run options shown in the table below. If you use multiple options they should be typed with no spaces in between after the minus sign. Note that other run options may be set in the Analysis Setup dialog box for your project and will be used during the analysis.

Option	Meaning
-Dlicense	Used for debugging <i>em</i> licensing problems. Displays all environment information relevant to licensing.
-N	Display number of subsections and estimated required memory. <i>Em</i> then exits without running a full analysis.
-test	Run <i>em</i> on a test circuit. Used to verify that <i>em</i> can get a license and run successfully.
-v	Display analysis information as the analysis is performed. The analysis information is output to the command prompt window or terminal from which the batch was executed.
-Lossless	Perform simulation as if the entire circuit is completely lossless. All metal polygons and vias, all dielectric layers, all dielectric bricks, and the Sonnet top and bottom box covers are simulated as if they are lossless.
-AbsCacheNone	Disable ABS caching (overrides setting in project file).
-AbsCacheStopRestart	Enable ABS stop-restart caching (overrides setting in project file).
-AbsCacheMultiSweep	Enable ABS multi-sweep plus stop-restart caching (overrides project file).
-AbsNoDiscrete	Used when running ABS with pre-existing cache data. Tells the analysis engine not to do any more discrete frequencies. If pre-existing cache data is sufficient to get converged ABS solution, then that solution is written to output. Otherwise, no processing is performed.

Option	Meaning
-SubFreqHz[value]	where [value] is the subsectioning frequency in Hz. Note there is no space before the value field. This option allows subsectioning frequency to be specified on the command line, thereby overriding the settings in the project file.
-ParamFile <filename>	where <filename> is the name of a file which contains the value(s) which you wish to use for parameter(s) in the circuit being analyzed. These values override the value contained in the geometry project for the analysis, but do not change the contents of the geometry project. The syntax for the parameter file is <parname>=fnum where <parname> is the name of the parameter and fnum is a floating point number which defines the value of the parameter for the analysis.

<**project name**> is the name of the project which you wish to analyze. If there is no extension, then the extension “.son” is assumed. This field is required.

[**external frequency file**] is the name of an optional external frequency control file whose extension is “.eff”. This extension must be included when specifying the control file. You may create an external frequency control file in the project editor. For details see Frequency Sweep Combinations in online help in the project editor. The frequencies in this file override the frequencies in the project.

For example, if you wish to analyze the project steps.son in a batch file using the -v option, the command line would be:

**em -v steps.son**

An example of a batch file which runs multiple analyses is shown below.

```
em -v steps.son
em -v filter.son filter.eff
em -v airbridge.son filter.eff
em -v airbridge.son
```

To execute a batch file on the PC, you should create a text file containing the command lines with a “.bat” extension. Then open a DOS prompt window and type the filename at the prompt and press return.

To execute a batch file on Linux, create a text file containing the command lines. The filename does not need any extension. Then change the permissions mode of the file to allow you to execute it. For example:

### **chmod a+x <filename>**

where <filename> is the name of the batch file you wish to execute. Then, type the name of the file at the Linux prompt and press return.

On Linux systems there are several additions to the command line which are useful to know. Placing “nice” before the command runs it at lower priority. Placing “&” at the end of the command runs it in background, so you get your cursor back. Entering “nohup” before the command line allows you to log off while the em job(s) keep running. If you are using the “&” or the “nohup”, you might want to consider redirecting the output using “> outfile.” See your system administrator for details on any of these options.

## Causal Dielectrics

If you wish to export data from a Sonnet analysis to be used in a time domain tool, you may wish to use causal dielectrics. Using a causal dielectric will vary the properties of the dielectric layer - the relative dielectric constant, the dielectric loss tangent, and the dielectric conductivity- slightly over a frequency band. The variations are modeled using Debye functions based on a reference frequency<sup>1</sup>. The reference frequency is the frequency at which the values entered for the properties are known to be accurate based on obtained measurements or given by the manufacturer. The options to use causal dielectrics are provided below.

Option	Meaning
-CausalDielectrics<freq>	where <freq> is the reference frequency in Hz at which you know the values for the dielectric properties are accurate. For example if your reference frequency is 3.0e9 Hz than the option you enter would be -CausalDielectrics3.0e9.
-CausalDielectrics	Use this option if you do not know the reference frequency. In this case, the analysis engine uses the average of the minimum and maximum analysis frequencies to obtain a reference frequency.

<sup>1</sup>For a detailed technical discussion of Causal Dielectrics and the use of Debye functions in modeling them, please see the following article:  
Morgan, Chad, “Solutions for Causal Modeling and A Technique for Measuring Causal, Broadband Dielectric Properties,” DesignCon 2008, February 4 - 7, 2008.

## xgeom Command Line

You may also run the project editor, **xgeom**, from a command line in order to perform an export of a project file in DXF, GDSII or Gerber format.

The syntax of the command line is as follows:

**xgeom <project> -Export <type> -RWOut <file> -ExpRegistry -ExpOptions <exfile>**

- <project>** The name of the project which you wish to export. If there is no extension, then the extension “.son” is assumed. This field is required.
  - <type>** This identifies the format of the output file. Choices are DXF, GDS or GERBER. This field is required.
  - <file>** For DXF and GDSII, this is the filename to which you wish to write your output. For Gerber, this is the folder to which you wish to write your output files. This field is required. Use “.” as the folder name, if you wish to use the present folder.
  - ExpRegistry** This option instructs **xgeom** to use the export options from the registry. This field is optional. If neither -Exp option is included then the default export options are used. If both -Exp options are included in the command, the export options are read first from the registry and then from the specified <exfile>. Any options set in the file will override those in the registry. Any options in the registry overrides the default setting. The export options in the registry are what was used the last time you performed an export from the project editor.
  - ExpOptions <exfile>** This option instructs **xgeom** to use the export options from the specified file <exfile>. This field is optional. If neither -Exp option is included then the default export options are used. This option is not applicable for a GDSII export. Any options set in this file override any default options or options set in the registry.
- Options which can be entered in the Export Options file <exfile> are in the following table. The <boolean> field can use any of the following entries: yes, no, true, false, on, off. Each option should be entered on a separate line. Not all options need to be specified.
- New** This option opens a new geometry project in xgeom.
  - NewNET** This option opens a net netlist project in xgeom.
  - 3D** This option opens a 3D view of the project in xgeom.

Each entry corresponds to a control in the DXF or Gerber Export Options dialog box. Each entry provides syntax and the control to which it corresponds. Please refer to Help in the project editor for details about the control. You may access Help by opening the appropriate Export Options dialog box and clicking on the Help button in the dialog box.

Entry	Definition
SepObj= <boolean>	Separate By Object Types checkbox (DXF & Gerber)
SepMat= <boolean>	Separate By Material Types checkbox (DXF & Gerber)
DivideMulti= <boolean>	Divide Multi-layer Vias checkbox (DXF & Gerber)
Circles= <boolean>	Convert Vias to Circles checkbox (DXF & Gerber)
CircleType= <type>	<type> is inscribed or manual. Inscribed corresponds to the Auto radio button. Manual corresponds to the Manual radio button. This option may only be used when the Circles option is used.
CircleSize= <float>	<float> is a floating string number. Manual text entry box. This option is only used if CircleType is set to "manual."
KeepMetals= <boolean>	Metal Polygons checkbox (DXF & Gerber)
KeepVias= <boolean>	Via Polygons checkbox (DXF & Gerber)
KeepBricks= <boolean>	Brick Polygons checkbox (DXF & Gerber)
KeepEdgeVias= <boolean>	Edge Vias convert to one cell wide vias checkbox (DXF & Gerber)
KeepParent= <boolean>	Output as Metal checkbox. This is only applicable if the KeepEdgeVias option is on.
ConvertParent= <boolean>	Convert to Vias checkbox. This is only applicable if the KeepEdgeVias option is on.
GbrUnits=<unit>	<unit> is either inch or mm (Gerber Only)
GbrWholeDigits=<num>	<num> is an integer number from 2 to 6. Whole digits drop list
GbrDecimalDigits=<num>	<num> is an integer number from 2 to 6. Decimal digits drop list
GbrFilenameType=<type>	<type> is default, custom or project. Default corresponds to the default radio button. Custom corresponds to the Custom Prefix radio button. Project corresponds to the Project name radio button. (Gerber Only)

<b>Entry</b>	<b>Definition</b>
GbrFilenamePrefix=<string>	<string> is a character string of alphanumeric characters. Custom Prefix text entry box. This is only applicable if the GbrFilenameType option is set to custom. (Gerber Only)
GbrFilenameExt=<string>	<string> is a character string of alphanumeric characters. ext. text entry box. (Gerber Only)
GbrJobFilenameType=<type>	<type> is custom or project. Custom corresponds to the Custom Prefix radio button in the Netex-G Job Filename section of the dialog box. Project corresponds to the Project Name radio button. (Gerber Only)
GbrJobPrefix=<string>	<string> is a character string of alphanumeric characters. Custom Prefix text entry box. This is only applicable if the GbrJobFilenameType option is set to custom. (Gerber Only)

## Example of *xgeom* Command Line

An example command line is shown below.

**xgeom test.son -Export DXF -RWOut outfile.dxf -ExpRegistry -ExpOptions expopt.txt**

The above command creates a DXF output file named outfile.dxf from the Sonnet project test.son using the export options provided in the Export Options file expopt.txt. this feature creates a DXF file called outfile.dxf. The programs closes once the export is complete.

An example of an Export Options file is shown below:

```

SepObj=yes
SepMat=no
DivideMulti=no
Circles=yes
CircleType=inscribed
CircleSize=5.8
KeepMetals=yes
KeepVias=true
KeepBricks=no
KeepEdgeVias=off
KeepParent=TRUE
ConvertParent=OFF

```



## Appendix II      Sonnet References

---

This appendix contains articles written by Sonnet authors or articles which directly impacted the analysis theory used by Sonnet. An extensive list of articles in which Sonnet was used as the analysis tool is available on Sonnet's web site at [www.sonnetsoftware.com](http://www.sonnetsoftware.com). Search for "References."

- [1] Allen F. Horn, John W. Reynolds, Patricia A. LaFrance, James C. Rautio, "Effect of conductor profile on the insertion loss, phase constant, and dispersion in thin high frequency transmission lines," DesignCon Best Paper Award, February 2010.
- [2] James C. Rautio, "In Search of Maxwell," Microwave Journal, Vol. 49, No. 7, July 2006, pp. 76-88.
- [3] Heng-Tung Hsu, James C. Rautio, and San-Wen Chang, "Novel Planar Wideband Omni-directional Quasi Log-Periodic Antenna," Asia-Pacific Microwave Conference 2005, Suzhou, China, December 4-7, 2005.
- [4] James C. Rautio and Vladimir I. Okhmatovski, "Unification of Double-Delay and SOC Electromagnetic Deembedding," IEEE Transactions on Microwave Theory and Techniques, Vol. 53, No. 9, September 2005, pp 2892 - 2898.
- [5] James C. Rautio, "Applied numerical electromagnetic analysis for planar high-frequency circuits," Encyclopedia of RF and Microwave Design, Wiley, New York, Vol. 1, 2005, pp. 397-413.
- [6] James C. Rautio, "Deembedding the Effect of a Local Ground Plane in Electromagnetic Analysis," IEEE Transactions on Microwave Theory and Techniques, Vol. 53, No. 2, February 2005, pp. 770 - 776.

- [7] James C. Rautio, "Comments on "On Deembedding of Port Discontinuities in Full-Wave CAD Models of Multiport Circuits"," IEEE Transactions on Microwave Theory and Techniques - Letters, Vol. 52, No. 10, October 2004, pp. 2448 - 2449.
- [8] James C. Rautio, "A Space-Mapped Model of Thick, Tightly Coupled Conductors for Planar Electromagnetic Analysis," IEEE Microwave Magazine, Vol. 5, No. 3, September 2004, pp. 62 - 72.James C. Rautio, "Accurate and Efficient Analysis of Large Spiral Inductors with Thick Metal and Narrow Gaps Using Space Mapping," IEEE MTT-S International Microwave Symposium, Workshop Notes & Short Courses - WFD-7, 6-11 June 2004.
- [9] James C. Rautio, "In Defense of Uselessness," IEEE Microwave Magazine, Vol. 5, No. 1, March 2004, pp. 100 - 102.
- [10] James C. Rautio, "A Conformal Mesh for Efficient Planar Electromagnetic Analysis," IEEE Transactions on Microwave Theory and Techniques, Vol. 52, No. 1, January 2004, pp. 257 - 264.
- [11] David I. Sanderson, James C. Rautio, Robert A. Groves, and Sanjay Raman, "Accurate Modeling of Monolithic Inductors Using Conformal Meshing for Reduced Computation," Microwave Magazine, Vol. 4, No. 4 December 2003, pp. 87 - 96.
- [12] James C. Rautio, "Testing Limits of Algorithms Associated with High Frequency Planar Electromagnetic Analysis," European Microwave Conference Digest, Munich, October 2003, pp. 463 - 466.
- [13] Rautio, James C., "Generating Spectrally Rich Data Sets Using Adaptive Band Synthesis Interpolation," Workshop WFA, IEEE MTT Symposium Digest, Philadelphia, June 8 - 13, 2003.
- [14] James C. Rautio, "Conformal Subsections for Accurate EM Analysis," Microwave Journal, Vol. 46, No. 6, June 2003, pp. 116 - 120.
- [15] James C. Rautio and Veysel Demir, "Microstrip Conductor Loss Models for Electromagnetic Analysis," IEEE Transactions on Microwave Theory and Techniques, Vol. 51, No. 3, March 2003, pp. 915 - 921.
- [16] James C. Rautio, "Electromagnetic Analysis Speeds RFID Design," Microwaves & RF, Vol. 42, No. 2, February 2003, pp. 55 - 62.
- [17] James C. Rautio, "EM Approach Sets New Speed Records," Microwaves & RF, Vol. 41, No. 5, May 2002, pp. 81 - 96.
- [18] James C. Rautio, Shawn Carpenter, et. al., "CAD/EDA virtual panel discussion," Microwave Engineering, May 2002, pp. 15 - 25.
- [19] Shawn Carpenter, "Analysis and Optimization of a Compact CPW Filter Using Planar EM Software," Microwave Product Digest, October 2001, pp. 10-14, 28, 50.

- [20] James C. Rautio, "Making Practical High Frequency Electromagnetic Simulators-- Past, Present, and Future," IEICE Transactions on Electronics, Vol. E84-c, No. 7, July 2001, pp. 855-860.
- [21] J. C. Rautio, "An Investigation of Microstrip Conductor Loss," IEEE Microwave Magazine, Volume 1, Number 4, December 2000, pp. 60-67.
- [22] Shawn Carpenter, "Break and Interpolate Technique: A Strategy for Fast EM Simulation of Planar Filters," Microwave Project Digest, October 2000, pp. 18 - 27, 56 - 58.
- [23] James C. Rautio, "The Impact on Education of Widely Available Commercial 3-D Planar Electromagnetic Software," Computer Applications in Engineering Education, Vol. 8, No. 2, September 2000, pp. 51 - 60.
- [24] G. L. Matthaei, J. C. Rautio, and B. A. Willemesen, "Concerning the influence of housing dimensions on the response and design of microstrip filters with parallel-line couplings," IEEE MTT Transactions, Vol. 48, August 2000, pp. 1361 –1368.
- [25] James C. Rautio, "Tips and Tricks for Using Sonnet Lite - Free EM software will radically change the way you do high frequency design," Microwave Product Digest, November 1999, pp. 30 - 34, 67 - 70.
- [26] James C. Rautio, "An Investigation of Microstrip Conductor Loss," IEEE MTT Magazine, December 2000, pp. 60-67. This article is available in the Support Section of the Sonnet website, [www.sonnetsoftware.com](http://www.sonnetsoftware.com).
- [27] Shawn Carpenter, "Break and Interpolate Technique: A Strategy for Fast EM Simulation of Planar Filters," Microwave Project Digest, October 2000, pp. 18 - 27, 56 - 58.
- [28] James C. Rautio, "The Impact on Education of Widely Available Commercial 3-D Planar Electromagnetic Software," Computer Applications in Engineering Education, Vol. 8, No. 2, September 2000, pp. 51 - 60.
- [29] James C. Rautio, "Free EM Software Analyzes Spiral Inductor on Silicon," Microwaves & RF, September 1999, pp. 165 - 172.
- [30] Takashi MIURA, Hideki NAKANO, Kohji KOSHIJI and Eimei SHU, "Reduction of time required for electromagnetic analysis by dividing circuit," Faculty of Science and Technology, Science University of Tokyo, Japan Institute of Electronics Packaging, pp.79-80, Mar. 1999. (Article in Japanese)
- [31] James C. Rautio, "Tips and Tricks for Using Sonnet Lite - Free EM software will radically change the way you do high frequency design," Microwave Product Digest, November 1999, pp. 30 - 34, 67 - 70.
- [32] James C. Rautio, "EM Simulation," 1999 IEEE MTT-S International Microwave Symposium, Microwave and Millimeter-Wave Design Tool Applications Workshop, Anaheim, CA, June 13, 1999.

- [33] James C. Rautio, "Application of Electromagnetic Analysis Software to 3-D Planar High Frequency Design", International Multilayer Circuits Symposium (IMCS), March 1999, pp. B2-1 - B2-17.
- [34] Shigeki Nakamura, "Top Interview: Electromagnetic Analysis is not Difficult - Big Rush to Install PC Version," Electronic Products Digest, Vol. 16, No. 1, January 1999, page 48. (Japanese Article)
- [35] James C. Rautio, "Comments on 'Revisiting Characteristic Impedance and Its Definition of Microstrip Line with a Self-Calibrated 3-D MoM Scheme,'" IEEE Transactions on Microwave Theory and Techniques, Vol. 47, No. 1, January 1999, pp. 115 - 117.
- [36] James C. Rautio and George Mattheai, "Tracking Error Sources in HTS Filter Simulations," Microwaves and RF, Vol. 37, No. 13, December 1998, pp. 119 - 130.
- [37] J.C. Rautio, "-Electromagnetic Analysis for Microwave Applications," Computational Electromagnetics and Its Applications, Vol. 5, Boston: Kluwer Academic Publishers, 1997, pp. 80-96.
- [38] Yasumasa Noguchi, Shin-ichi Nakao, Hideaki Fujimoto and Nobuo Okamoto, "Characteristics of Shielded Coplanar Waveguides on Multilayer Substrates," Electronic Information and Communications University Meeting, Electronics Society Conference, June 29, 1998. (Japanese Article)
- [39] Erik H. Lenzing and James C. Rautio, "A Model for Discretization Error in Electromagnetic Analysis of Capacitors," IEEE Transactions on Microwave Theory and Techniques, Vol. 46, No. 2, February 1998, pp. 162-166.
- [40] J. C. Rautio, "Retracing Key Moments In the Life of Maxwell," Microwaves & RF," Vol. 36, No. 11, November 1997, pp. 35-51.
- [41] J. C. Rautio, "Electromagnetic Analysis for Microwave Applications," NASA CEM (Computational Electromagnetics) Workshop, Newport News, VA, May 1996.
- [42] J. C. Rautio, "Seven Years Later," Applied Microwave and Wireless, November/ December 1996, pp. 99-100.
- [43] J. C. Rautio, "Questionable Reviews," The Institute (IEEE newspaper), Jan. 1996, pg. 11.
- [44] J. C. Rautio, "An Investigation of an Error Cancellation Mechanism with Respect to Subsectional Electromagnetic Analysis Validation," International Journal of Microwave and Millimeter-Wave Computer-Aided Engineering, Vol. 6, No. 6, November 1996, pp. 430-435.
- [45] J. C. Rautio, "The Microwave Point of View on Software Validation," IEEE Antennas and Propagation Magazine, Vol. 38, No. 2, April 1996, pp. 68-71.

- [46] J. C. Rautio and Hiroaki Kogure, "EMI Applications Of The Electromagnetic Analysis By The Method Of Moments-Electromagnetic Analysis Applied To Analog And Digital PCB Design," JPCA Show 96 Text: Today and Tomorrow of EMI Design, pp. 11-19.
- [47] J. C. Rautio, "EM-Analysis Error Impacts Microwave Designs," Microwaves and RF, September 1996, pp. 134-144.
- [48] James R. Willhite, "Turning Clean Theory into Reality," Wireless Design and Development, March 1996, Vol. 4, No. 3, pp. 19-20.
- [49] J. C. Rautio, "Response #2. Comments on Zeland's Standard Stripline Benchmark Results - MIC Simulation Column," International Journal of Microwave and Millimeter- Wave Computer-Aided Engineering, Vol. 5, No. 6, November 1995, pp. 415-417.
- [50] J. C. Rautio, "EMI Analysis from a Wireless Telecommunication and RF Perspective," Proceedings of the 1995 Nepcon West Conference, Anaheim, CA, USA, pp. 749-755.
- [51] J. C. Rautio and Hiroaki Kogure, "An Overview of the Sonnet Electromagnetic Analysis," Proceedings of the 1994 IEICE Fall Conference, Tokyo, pp. 325-326.
- [52] J. C. Rautio, "An Ultra-High Precision Benchmark For Validation Of Planar Electromagnetic Analyses," IEEE Tran. Microwave Theory Tech., Vol. 42, No. 11, Nov. 1994, pp. 2046-2050.
- [53] J. C. Rautio, "A Precise Benchmark for Numerical Validation," IEEE International Microwave Symposium, Workshop WSMK Digest, Atlanta, June 1993.
- [54] "Comparison of Strategies for Analysis of Diagonal Structures," Sonnet Application Note 51-02.
- [55] J. C. Rautio, "MIC Simulation Column - A Standard Stripline Benchmark," International Journal of Microwave & Millimeter-Wave Computer-Aided Engineering, Vol. 4, No. 2, April 1994, pp. 209-212.
- [56] J. C. Rautio, "Response #3. Standard Stripline Benchmark - MIC Simulation Column," International Journal of Microwave and Millimeter-Wave Computer-Aided Engineering, Vol. 5, No. 5, September 1995, pp. 365-367.
- [57] J. C. Rautio, "Some Comments on Approximating Radiation," International Journal of Microwave and Millimeter-Wave Computer-Aided Engineering, Vol. 4, No. 2, 1994, pp. 454-457.
- [58] J. C. Rautio, "Synthesis of Lumped Models from N-Port Scattering Parameter Data," IEEE Tran. Microwave Theory Tech., Vol. 42, No. 3, March 1994, pp. 535-537.
- [59] J. C. Rautio, "Educational Use of a Microwave Electromagnetic Analysis of 3-D Planar Structures," Computer Applications in Engineering Education, Vol. 1, No. 3, 1993, pp. 243-254.

- [60] J. C. Rautio, "Characterization of Electromagnetic Software," 42nd ARFTG Conference Digest, San Jose, CA, Dec. 1993, pp. 81-86.
- [61] J. C. Rautio, "Some Comments on Electromagnetic De-Embedding and Microstrip Characteristic Impedance" International Journal of Microwave & Millimeter-Wave Computer-Aided Engineering, Vol. 3, No. 2, April 1993, pp. 151-153.
- [62] J. C. Rautio, "Some Comments on Electromagnetic Dimensionality," IEEE MTT-S Newsletter, Winter 1992, pg. 23.
- [63] J. C. Rautio, "Sonnet Software Reveals Tangential Fields," EEs of Wavelengths, Vol. 9, No. 1, March 1993, pg. 12.
- [64] J. C. Rautio, "Sonnet Introduces Antenna Pattern Visualization in New Release," EEs of Wavelengths, Vol. 9, No. 2, June 1993, pg. 21.
- [65] J. C. Rautio, "EEs of Joins Forces With Sonnet Software," EEs of Wavelengths, Vol. 8, No. 3, Sept. 1992, pg. 14.
- [66] J. C. Rautio, "Electromagnetic Design of Passive Structures - Emerging Technology in Microwave CAD," IEEE MTT-S Newsletter, Fall 1990, pp. 21-22.
- [67] J. C. Rautio, "Electromagnetic Microwave Design," RF/Microwave Applications Conference, Santa Clara, CA, March 1992, pp. 105-109.
- [68] J. C. Rautio, "Experimental Validation of Microwave Software," IEEE International Microwave Symposium, Panel Session PSB Digest, Albuquerque, June 1992.
- [69] J. C. Rautio, "Current Developments in 3-D Planar Microwave Electromagnetics," Microwave Hybrid Circuits Conference, Oct. 1991, Arizona.
- [70] J. C. Rautio, "Current Developments in 3-D Planar Microwave Electromagnetics," Microwave Hybrid Circuits Conference, Oct. 1992, Arizona.
- [71] J. C. Rautio, "Current Developments in 3-D Planar Microwave Electromagnetics," Microwave Hybrid Circuits Conference, Oct. 1993, Arizona.
- [72] J. C. Rautio, "Current Developments in 3-D Planar Microwave Electromagnetics," Microwave Hybrid Circuits Conference, Oct. 1994, Arizona.
- [73] J. C. Rautio, "Experimental Validation of Electromagnetic Software," International Journal of Microwave & Millimeter-Wave Computer-Aided Engineering, Vol. 1, No. 4, Oct. 1991, pp. 379-385.
- [74] J. C. Rautio, "Electromagnetic Microwave Analysis," IEEE International Microwave Symposium, Workshop WSA Digest, Albuquerque, June 1992.
- [75] J. C. Rautio, "EM Visualization Assists Designers," Microwaves and RF, Nov. 1991, pp. 102-106.
- [76] J. C. Rautio, "Reviewing Available EM Simulation Tools," Microwaves & RF, June 1991, pp. 16A-20A.

- [77] "Generating Spice Files Using the *em* Electromagnetic Analysis," Sonnet Application Note 104a, Dec. 1998.
- [78] J. C. Rautio, "A New Definition of Characteristic Impedance," MTT International Symposium Digest, June 1991, Boston, pp. 761-764.
- [79] J. C. Rautio, "A De-Embedding Algorithm for Electromagnetics," International Journal of Microwave & Millimeter-Wave Computer-Aided Engineering, Vol.1, No. 3, July 1991, pp. 282-287.
- [80] J. C. Rautio, "Triangle Cells in an Electromagnetic Analysis of Arbitrary Microstrip Circuits," MTT International Microwave Symposium Digest, Dallas, June 1990, pp. 701-704.
- [81] J. C. Rautio, "Experimental Validation of Microwave Software," 35th ARFTG Conference Digest, Dallas, May 1990, pp. 58-68. (Voted best paper at the conference.)
- [82] J. C. Rautio, "Preliminary Results of a Time-Harmonic Electromagnetic Analysis of Shielded Microstrip Circuits," 27th ARFTG Conference Digest, Dallas, Dec. 1986. (Voted best paper at the conference.)
- [83] J. C. Rautio, "An Experimental Investigation of the Microstrip Step Discontinuity," IEEE Tran. Microwave Theory Tech., Vol. MTT-37, Nov. 1989, pp. 1816-1818.
- [84] J. C. Rautio, "A Possible Source of Error in On-Wafer Calibration," 34th ARFTG Conference, Ft. Lauderdale, FL, Dec. 1989, pp. 118-126.
- [85] J. C. Rautio, "Microstrip Program Improves Accuracy of Circuit Models," Microwaves & RF, Vol. 27, No. 12, pp. 89-96, Nov. 1988.
- [86] J. C. Rautio, "Reflection Coefficient Analysis of the Effect of Ground on Antenna Patterns," IEEE Antennas and Propagation Society Newsletter, Feb. 87, pp. 5-11.
- [87] J. C. Rautio and R. F. Harrington, "An Electromagnetic Time-Harmonic Analysis of Shielded Microstrip Circuits," IEEE Trans. Microwave Theory Tech., Vol. MTT-35, pp. 726-730, Aug. 1987.
- [88] J. C. Rautio and R. F. Harrington, "An Efficient Electromagnetic Analysis of Arbitrary Microstrip Circuits," MTT International Microwave Symposium Digest, Las Vegas, June 1987, pp. 295-298.
- [89] J. C. Rautio and R. F. Harrington, "Results and Experimental Verification of an Electromagnetic Analysis of Microstrip Circuits," Trans. of The Society for Computer Simulation, Vol. 4, No. 2, pp. 125-156, Apr. 1987.
- [90] J. C. Rautio, "A Time-Harmonic Electromagnetic Analysis of Shielded Microstrip Circuits," Ph. D. Dissertation, Syracuse University, Syracuse, NY, 1986.

- [91] J. C. Rautio, "Preliminary Results of a Time-Harmonic Electromagnetic Analysis of Shielded Microstrip Circuits," ARFTG Conference Digest, Baltimore, pp. 121-134, June 1986. (Voted best paper at the conference.)
- [92] J. C. Rautio, "Techniques for Correcting Scattering Parameter Data of an Imperfectly Terminated Multiport When Measured with a Two-Port Network Analyzer," IEEE Trans. Microwave Theory Tech., Vol. MTT-31, May 1983, pp. 407-412.
- [93] R. Horton, B. Easter, A. Gopinath, "Variation of Microstrip Losses with Thickness of Strip", Electronics Letters, 26th August 1971, Vol. 7, No.17, pp.490-481.
- [94] R. F. Harrington, **Time-Harmonic Electromagnetic Fields**, New York: McGraw-Hill, 1961, section 8-11, pg. 8.

---

# Index

---

**A**

ABS caching level 159  
  subsectioning 159  
ABS see *adaptive band synthesis*  
absolute power 339  
adaptive band synthesis 155-167  
  ABS caching level 159  
  analysis issues 165-167  
    current density data 166  
    de-embedding 165  
    multiple box resonances 165  
    optional files 166  
    ripple in S-Parameters 166  
    transmission line parameters 166  
  find maximum 162  
  find minimum 162  
  parameter sweep 163  
  resolution 156  
  running an adaptive sweep 157  
  viewing the response 167  
adaptive sweep 157  
add  
  ports  
    auto-grounded 101  
    standard 88  
    via ports 99  
Add Subdivider 273  
adjacent polygons 232, 233  
adjustable point set 180, 181, 183  
ADS interface 17  
analysis  
  limitations 335  
  starting 210

analysis frequencies 207  
analysis monitor 210  
Analysis Setup dialog box 159, 207, 214  
analyze 210  
anchored dimension parameters 178, 179  
  adding 198  
  anchor point 179, 185  
  reference point 179, 185  
Angle Axis 337  
anisotropic dielectric bricks 318, 323  
antennas 327-334  
  microstrip 333  
  pattern 342  
  surface wave 335  
  wire 335  
array loss model 63  
array model  
  horizontal loss 63, 65  
att 244, 254  
att\_cascade.son 245  
attenuator 250  
auto height vias 299  
auto-grounded ports 99, 100  
autogrouping  
  viewing 89  
autoscale 357  
axes 357  
axis  
  Z 336  
Axis Properties dialog box 357

**B**

balanced ports 76  
bandpass filter 197  
bar via  
  horizontal loss 62

- bar vias
  - simplifying in analysis engine 292
- batch file 413
- benchmark 407-411
  - residual error 409
- best iteration 221
- boundaries 235
- box resonance 165
- box-wall ports 87
- bricks see *dielectric bricks*
- broadband spice 371
  - checking accuracy of model 376
  - class of problems 372
  - creating a model 373
  - improving accuracy of model 378
  - stability factor 379
- C**
  - Cadence Virtuoso Interface 17
  - CAE software 22
  - calculate 335, 344
    - calculating the response 344
  - Calculation Setup dialog box 344
  - calculations
    - defaults 343
    - status of 346
  - Calculator
    - Cell Size 30
  - calibration group properties 89
    - ground node connection 90
    - terminal width 92
  - calibration lengths
    - for co-calibrated internal ports 97
    - for Components 148
  - capacitor 141
  - capacitors 240, 319
  - cartesian plot 327, 334, 337, 344, 358
  - cascading data files 239, 245
  - causal dielectrics 416
  - cell size 320
    - determining error 410
    - processing time 235
    - selecting 29-31
    - subsectioning 28, 38
    - wavelength 29
- Cell Size Calculator 30
- center tapped topology 382
- circuit analyses 239, 241
- circuit properties 267
- circuit subdivision 255-269
  - additional improvements 281
  - analysis 269
  - coarse frequency sweep 268
  - comparison of results 281
  - executing 269, 277
  - feedlines 278
  - illegal conditions for subdivision lines 266
  - main netlist 258
  - placement of subdivision lines 260-264
  - procedure 258
  - reference planes 278
  - subdivision line orientation 265
  - subproject naming conventions 278
  - tutorial 271-283
  - using parameters 259
- Circuit Subdivision dialog box 277
- circular dependencies 177, 188
- CKT 248
  - coarse frequency sweep 268
  - coaxial structures 335
  - co-calibrated internal ports 89-97
    - adding 93
    - calibration group properties 89
    - calibration lengths 97
    - de-embedding coupling 89
    - GLG metal 91
    - ground node connection 90
    - placement rules 94
    - reference planes 97
    - terminal width 92
    - use in Components 138, 139, 141
    - viewing autogrouping 89
  - combined circuit analyses 241
  - combining data files 239
  - command line 413
  - components 137
    - analysis 152
    - anatomy of 139
  - calibration lengths 148

- Component Assistant 138  
component symbol 139  
Data File type 140  
defining physical size 149  
definition 139  
ground node connection 143  
Ideal component 141  
label 139  
physical size 140  
ports 139  
Ports Only type 141  
properties of 143  
reference planes 96, 148, 151  
restricted space 94, 150  
restrictions 149  
settings 143  
terminal number 139  
terminal width 145  
types of 140  
use rules 149
- conductivity 51  
conformal mesh 227-237  
    adjacent polygons 232  
    adjacent polygons with interior vertex 233  
    applying 229  
    cell size 235  
    current density viewer 236  
    current striping 236  
    figure-eight polygons 232  
    Manhattan polygons 233  
    polygon boundaries 235  
    processing time 235  
    rules of use 231  
    single precision 233  
    transmission lines 229  
    using effectively 233
- conjugate gradient optimizer 192  
convergence test  
    dielectric bricks 318  
coordinate system  
    spherical 336
- coplanar 76, 335  
corner sweep 189  
coupling 256, 261  
current density data 166  
current density file 335  
current density viewer 16, 41  
current striping 236
- D**
- data blocks  
    CKT 248  
Data File 140  
data files 239  
    cascading 245  
DC point 156  
de-embedding 103-136, 335  
    box-wall ports 105, 107  
    components 89, 137  
    coupled transmission lines 109  
    enabling 105  
    port discontinuities 105  
    reference planes 106, 109
- de-embedding codes 126  
    nl 129
- dependent variable 176
- design suite 15
- diagonal fill 320
- diagonal ports 80
- dialog box  
    Parameter Properties 200
- dialog boxes  
    Analysis Setup 159, 207, 214  
    Axis Properties 357  
    Calculation Setup 344  
    Circuit Subdivision 277  
    Dielectric Layers 66  
    Metal Editor 306  
    Metal Properties 230  
    Optimization Goal Entry 217  
    Optimization Parameters 215  
    Optimization Results 224  
    Parameter Sweep Entry 207  
    Port Impedance 75, 79  
    Select Frequencies 346

Select Normalization 352  
Select Phi's 347, 360  
Subdivider Orientation 273  
Subproject Specifications 278  
dielectric 335  
  causal 416  
  multiple constants 335  
dielectric bricks 317-326  
  air 320  
  anisotropic 323  
  applications 319  
  convergence test 318  
  far field viewer 320  
  limitations 320  
  materials 322, 324  
  subsectioning 319  
  vias 319  
  visibility 322  
  Z-partitions 325  
dielectric layers  
  creating new 69  
  loss 47, 66  
  loss parameters 66  
Dielectric Layers dialog box 66  
dielectric library 69  
dimension parameter  
  adjusting point set 183  
  anchored 178, 179  
  radial 178, 185  
  symmetric 178, 180  
dipole  
  infinitesimal 341, 342, 358  
directions 334  
directive gain 339  
dual\_patch 332  
DUT 103  
DXF Translator 17  
*dxfgeo* 17

## E

E total 348  
*ebridge* 17  
edge mesh option 32  
editing a circuit 16  
Eeff 129, 166  
electrically thick conductors 53  
*em*  
  batch file 413  
  command line 413  
  description 20  
  theory 21-22  
*emgraph* 16  
*emstatus* 16  
*envu* 16  
E-plane 336  
equivalent circuit 91  
error function 192  
error, residual 409  
E-Total 361  
example files  
  att 254  
  att\_cascade.son 245  
  dual\_patch 332  
  infpole 358  
  par\_dstub 197  
  steps 240  
  sub\_whole 281  
  Tripat 333  
  via example 302  
examples  
  att 244  
excitations 334  
  default values 343

exit 362  
exiting 362  
extrapolation of a DC Point 156

**F**

far field 327, 334, 358  
Far Field Viewer  
    exiting 362  
    saving a file 361  
    tool bar 359  
    tutorial 341  
far field viewer 334-340  
    dielectric bricks 320  
    normalization 339  
    polarization 340  
    spherical coordinate system 336  
feedlines 278  
ferrite components 335  
figure eight polygons 232  
file  
    exit 362  
    save 361  
filter 197  
Find Maximum 162  
find maximum 162  
Find Minimum 162  
find minimum 162  
FINDMAX 162  
FINDMIN 162  
first frequency 335  
first reference point 181  
floating ground 144  
frequencies 338  
    default value 335  
    selecting to calculate 345  
    selecting to plot 346, 359  
frequency  
    maximum 30  
frequency interpolation 256  
frequency plot 358  
frequency specification 207  
frequency sweep  
    coarse 268

full view button 350

**G**

GABMAC 407  
gain  
    directive 339  
    power 339  
    relative to 339  
    max 352  
*gds* 17  
GDSII translator 16, 17  
general metal type 54  
Generalized Local Ground 91  
geometry project 246  
geometry subprojects 257  
Gerber Translator 17  
GLG metal 91  
global ground 144  
goal 194  
goals 217  
granularity 194  
graph  
    axes 357  
    calculate 335, 344  
    normalization 339, 352  
    select  
        frequencies 346  
        phi 347, 348, 360  
        theta 359, 360  
    type  
        cartesian 358  
        polar 355  
        surface 361  
ground node connection 90, 143  
    floating 91, 144  
    polygon edge(s) 145  
Sonnet box 90, 144

ground plane 341, 358  
ground reference 144  
ground terminals 145

## H

highest frequency 30  
horizontal loss  
    array model for via 63, 65  
    bar via 62  
    volume model for via 62  
horizontal orientation 265  
H-plane 336

## I

ideal component 141  
ideal element 137  
impedance 74, 75, 78, 79  
    surface 56  
importing  
    measurement data 137  
independent reference planes 80  
independent variable 177  
inductor 141  
inductor model extractor 380-388  
    center tapped topology 382  
    creating the model 383, 385  
    model topology 380  
    untapped topology 381  
inductors 240  
infinite array 328  
infinitesimal dipole 341, 342, 358  
infpole 358  
interdigital 197  
interface basics 23  
interpolation 256  
isotropic 339  
iteration 221  
iterations 194  
    selecting for display 221

## K

Keysight ADS Interface 17  
kinetic inductance 54

## L

lateral dimensions 335  
legend  
    selecting frequencies 346  
    selecting phi's 348, 360  
    turning off 355  
    view 356, 358  
libraries  
    metal 65  
library  
    dielectric 69  
    metal 65  
limitations 335  
linear sweep 189  
local ground de-embedding 89, 137  
loss 22, 47-69  
    creating metal types 50, 59  
    dielectric layer 66  
    dielectric layer library 69  
    dielectric layer parameters 66  
    general metal type 54  
    low frequency 52, 62  
    metal libraries 65  
    microstrip 50, 52  
    normal metal type 51  
    planar metal types 48  
    problems in determining 49  
    properties of commonly used metals  
        54  
    Rdc/Rrf metal type 52  
    related to frequency 52, 63  
    resistor metal type 52  
    sense metal type 58  
    Sonnet model 48  
    thick metal type 52  
    vias 298  
loss calculations  
    surface loss model 62  
    volume loss model 61

- 
- lumped elements 249
    - capacitors 240
    - inductors 240
    - inserting 240
    - inserting in a geometry 249
    - resistors 240
    - transmission lines 240
  - M**
    - magnetic permeability 53, 63, 67
    - main netlist 258
      - default name 277
    - Max 352
    - maximum frequency 30
    - maximum response 162
    - maximum subsection size 41
    - measurement data
      - importing 137
    - meshing fill
      - vias 290
    - metal
      - plated 311
      - stacked 311
    - Metal Editor dialog box 306
    - metal libraries 65
    - metal library 65
    - metal loss
      - normal loss model 51
      - Sonnet model 48
    - Metal Properties dialog box 230
    - metal type
      - creating 50, 59
      - general 54
      - instructions for creating. 58, 69
      - libraries 65
      - normal 51
      - properties of commonly used metals 54
    - Rdc/Rrf 52
    - resistor 52
    - sense 58
    - thick metal 52, 303
    - metalization
      - commonly used metals 54
    - loss 22, 47
    - metallization loss 48, 58
    - microstrip 197, 335
      - patch antennas 333
    - microstrip loss 50, 52
    - Microwave Office Interface 17
    - minima 195
    - minimum response 162
    - mixed sweep combinations 189
    - modeled elements 240
      - inserting 249
      - inserting in a geometry 249
    - modeling assumptions 335
    - modify
      - attributes
      - ports 74, 101
    - moving adjustable point set 183
    - mtline element 363
    - multi-sweep plus Stop/Restart caching 159
  - N**
    - netlist
      - creating 244
      - invoking an analysis of a subproject 246
    - netlist editor 239
    - netlist project 239
      - analysis 241
    - network file
      - analysis 280
    - networks 240
    - newlink pimodel 369
    - NI AWR interface 17
    - nl de-embedding code 129
    - nominal value 177, 178
      - changing 214
    - nominal values
      - update with results 224
    - non-Manhattan polygons 233
    - normal loss model 51
    - normal metal type 51
    - normalization 339, 351
      - changing 339

default 339, 344  
directive gain 339  
power gain 339  
number of iterations 194

## O

open environment 335  
opening a graph 210  
optimization 171-195  
    best iteration 221  
    conjugate gradient method 192  
    data 220  
    data range for parameters 216  
    example 197, 213  
    executing 219  
    methodology 192  
    number of iterations 194  
    process 193  
    results 195, 223  
    selecting parameters for 216  
    setting up 214  
    specifying goals 194, 217  
    tutorial 197, 213  
Optimization Goal Entry dialog box 217  
Optimization Parameters dialog box 215  
optimization results  
    update 224  
Optimization Results dialog box 224  
optimized values 223  
optional files 166  
orientation of subdivision lines 265  
origin 336  
output files  
    lumped.rsp 252

## P

package resonances 389-??  
par\_dstub 197  
Parameter Properties dialog box 200  
parameter sweep 163, 189, 197  
    analysis frequencies 207  
    data 220  
    setting up 207  
    viewing the response 210

Parameter Sweep Entry dialog box 207  
parameterization 171-195  
    tutorial 197  
parameters  
    adding 198, 203  
    adjustable point set 179, 180, 181, 185  
    adjusting point set 183  
    anchor point 179, 185  
    anchored 178, 179  
    circular dependencies 177, 188  
    data range for optimization 216  
    example 197  
    first reference point 203  
    in a subproject 259  
    nominal value 177, 178, 214  
    ports 75, 78, 79  
    radial 178, 185  
    reference planes 187  
    reference point 179, 185  
    second reference point 204  
    select dependents 188  
    selecting for optimization 216  
    symmetric 178, 180  
patch antenna 327  
patch antennas 335  
pattern 334, 342  
pattern response 350  
*patvu* 17  
permeability 53, 63, 67  
phased arrays 327  
phi 336, 337  
    default values 343  
    selecting values to plot 347, 348, 360  
    specifying range for calculation 337, 345  
physical size of Component 140  
pin number 139  
planar metal types  
    general loss model 54  
    loss 48  
    normal loss model 51  
    Rdc/Rrf loss model 52  
    resistor loss model 52  
    sense loss model 58

- thick metal loss model 52  
plated metal 311  
plot  
    cartesian 327, 334, 337, 344, 358  
    frequency 358  
    polar 327, 334, 337, 355  
    probing 350  
    selecting type 355  
    surface 327, 334, 337, 361  
    title area 355  
    types 337  
Plot Over drop list 359  
polar 337, 355  
polar plot 327, 334  
polarization 340  
    default 344  
polygon edges 261  
Port Impedance dialog box 75, 79  
ports 71-102  
    auto-grounded 99, 100, 101  
    balanced 76  
    box-wall 87, 105, 107  
    co-calibrated internal 89  
    component 139  
    diagonal 80  
    discontinuities 105  
    excitations 334, 343  
    impedance 74, 75, 78, 79  
    modify attributes 101  
    numbering 78, 88  
    push-pull 76  
    renumbering 75  
    standard 88  
    terminations 334, 343  
    ungrounded-internal 99, 250, 252  
    using with symmetry 79  
via 99  
via ports 99, 299  
Ports Only Component 141  
power gain 339  
probe readout 351  
probing the plot 350  
processing time 235  
Product Editor origin 336  
Project Editor 342  
project editor 16  
    netlist editor 239  
project file  
    circuit geometry 342  
properties  
    circuit 267  
push-pull ports 76  
**R**  
radial dimension parameters 178, 185  
radiation 327, 327-334  
radius axis  
    changing 356  
Rdc/Rrf metal type 52  
reactive surface impedance 49  
reference planes 106, 109  
    de-embedding without 110  
    for co-calibrated internal ports 97  
    for components 148  
    independent 80  
    parameters 187  
    short length 110  
reference point 179, 185, 203, 204  
    first in symmetrical 181  
    second in symmetrical 181  
references 340, 358, 362  
reflection boundary 341  
residual error 409  
resistance 66  
resistance per via 61  
resistivity 51  
resistor 141  
resistor metal type 52  
resistors 240  
    thin film 48, 245  
response  
    maximum 162

minimum 162  
response data  
  calculating 344  
  selecting 346  
response viewer  
  invoking 210  
results of optimization  
  accepting 223  
right-click 188  
right-clicking 346, 355, 360  
ripple in S-parameters 166  
Rrf 52, 53, 63  
rules for co-calibrated ports 94  
run options  
  edge mesh 32

**S**

saving a file 361  
scroll bar 347  
second reference point 181  
sections 257  
select  
  phi 348, 360  
  theta 359, 360  
select dependents 188  
Select Frequencies dialog box 346  
Select Normalization dialog box 352  
Select Phi's dialog box 347, 360  
selecting the response 346  
sense metal type 58  
sensitivity sweep 189  
series RL equivalent circuit 91  
sheet resistance at DC 51  
shunt elements 91, 144  
simplifying vias  
  bar vias 292  
single precision  
  using with conformal mesh 233  
skin effect coefficient 52, 53, 63  
SMD 137  
Sonnet  
  suite 15  
space bar 349  
S-parameter data  
  importing 137

S-Parameters  
  ripple in 166  
Spectre 17  
spherical coordinate system 336  
stability factor 379  
stacked metal 311  
steps 240  
steradian 339  
Stop/Restart caching 159  
stripline benchmark 407  
sub\_whole 281  
Subdivide Circuit command 277  
Subdivider Orientation dialog box 273  
subdividing 269  
subdivision 255-269  
  procedure 258  
subdivision lines 257  
  adding 272  
  coupling across 256  
  illegal conditions 266  
  orientation 265, 273  
  placement 256, 260-267  
  placement examples 261  
Subproject Specifications dialog box 278  
subprojects 257  
  naming conventions 278  
subs/lambda 41  
subsectioning 20  
  ABS caching level 159  
  cell size 28, 38  
  frequency 160  
  vias 290  
subsections 27, 41  
  of polygon 35  
  XMIN 39  
  YMIN 37  
superconductor 54  
  surface impedance 56  
surface impedance 56  
surface loss model 62  
  loss calculations 62  
surface plot 327, 334, 337, 361  
  selecting 361

- 
- surface resistance 66
  - surface wave antennas 335
  - sweep
    - corner 189
    - linear 189
    - mixed combinations 189
    - sensitivity 189
  - symbol
    - component 139
  - symmetric dimension parameter 178, 180
    - adding 203
    - first reference point 181
    - second reference point 181
  - symmetry 266
    - port placement 79
  - T**
  - T attenuator 247, 249
    - with ungrounded-internal ports 253
  - terminal number 139
  - terminal width 92
    - feedline width 92, 146
    - one cell 93, 146
    - user defined 93, 146
  - terminations 334
    - default values 343
  - theory 21-22
  - theta 336, 337
    - default values 343
    - selecting values to plot 359, 360
    - specifying range for calculation 337
  - thick metal 76, 303-316
    - arbitrary cross-section 314
    - creating a polygon 305
    - number of sheets 303
    - viewing in the current density viewer 315
    - viewing in the project editor 308
  - thick metal type 52, 303
  - thin film resistor 48, 245
    - two connected 245
  - tool bar 359
    - full view button 350
    - zoom in button 349
  - Tools**
    - Add Subdivider 273
    - Subdivide Circuit 277
  - top cover
    - resistive 407
  - top hemisphere 336
  - translator 16
    - GDSII 17
    - Gerber 17
  - transmission line parameters 166
  - transmission lines 240
    - structures 249
  - triangular subsections 335
  - Tripat 333
  - tutorial 341
    - optimization 197
    - parameter sweep 197
  - two-port T attenuator 247
  - type
    - cartesian 358
    - polar 355
    - surface 361
  - U**
  - ungrounded-internal ports 250, 252
  - untapped topology 381
  - update nominal values 224
  - user interface 23
  - V**
  - validation 341
  - variables 172
    - circular dependencies 177
    - dependent 176
    - independent 177
  - vertical components 335
  - vertical orientation 265
    - setting 273
  - via loss 58
    - array loss model 63
    - resistance per via 61
    - surface loss model 62
    - volume loss model 59

via metal types 58  
array loss model 63  
surface loss model 62  
volume loss model 59  
via ports 99, 299  
vias 302  
    auto height 299  
    example 302  
    inside dielectric bricks 319  
    loss 298  
    meshing fill 290  
    subsections 289  
    symbol 288  
    to ground 296  
view  
    legend 356, 358  
    zoom in 349  
viewing the response 210  
volume loss model 59  
    loss calculations 61  
volume model  
    horizontal loss 62

**W**  
warnings 253  
waveguide simulator 328  
wavelength 29  
wire antennas 335

**X**  
*xgeom* 16  
XMAX 40  
XMIN 37, 39

**Y**  
YMAX 40  
YMIN 37, 39

**Z**  
Z0 129, 130, 166  
Z-axis 336  
zoom in 349  
zoom in button 349  
zooming 349  
Z-partitioning 318, 319, 325-326