GETTING STARTED

RELEASE 11

Cover: James Clerk Maxwell (1831-1879). A professor at Cambridge University, England, Maxwell established the interdependence of electricity and magnetism. In his classic treatise of 1873, he published the first unified theory of electricity and magnetism and founded the science of electromagnetism.

GETTING STARTED

Printed: March 2007

Release 11

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

TABLE OF CONTENTS

CHAPTER 1 WELCOME

Welcome to Sonnet’s 3D Planar Electromagnetic software. Sonnet's suites of high-frequency electromagnetic (EM) Software are aimed at today's demanding design challenges involving predominantly planar (3D planar) circuits and antennas. Predominantly planar circuits include microstrip, stripline, coplanar waveguide, PCB (single and multiple layers) and combinations with vias, vertical metal sheets (z-directed strips), and any number of layers of metal traces embedded in stratified dielectric material.

The Sonnet Suites develop precise RF models (S-, Y-, Z-parameters or extracted SPICE model) for planar circuits and antennas. The software requires a physical description of your circuit (arbitrary layout and material properties for metal and dielectrics), and employs a rigorous Method-of-Moments EM analysis based on Maxwell's equations that includes all parasitic, cross-coupling, enclosure and package resonance effects.

DOCUMENTATION: Sonnet provides manuals, context sensitive help, and an online knowledge base as detailed below.

MANUALS: A set of hardcopy manuals are provided with Sonnet Level2 Basic and above. All Sonnet Suites including Sonnet Lite and Sonnet LitePlus come with a complete set of manuals in PDF Format. PDF format manuals are accessed by clicking on the Manuals button in the Sonnet task bar. The complete list of manuals is provided below:

Manual	Description	Format
Getting Started	Quick windows installation & Sonnet Basics including Start up tutorials	Hardcopy/PDF
User's Guide	Theory, features and feature tutorials	Hardcopy/PDF
Translators	Translators and Interfaces with tutorials	Hardcopy/PDF
UNIX & Linux Installation	UNIX & Linux Installation Instructions	Hardcopy/PDF
Remote em Processing	Installing remote ***em*** Processing	PDF Only
Windows Installation	Windows Installation including troubleshooting	PDF Only
emCluster Computing	Installing emCluster Computing	PDF Only
FLEXnet User's Manual	Licensing Manager Software User's Manual	PDF Only

HELP: May be accessed in any of the following ways:

- Selecting *Help* ⇒ *Contents* from any Sonnet application
- Clicking on the Help button in any dialog box
- **Windows**: Click on the ? button at the top of a dialog box and click on the desired control
- **UNIX & Linux:** Press Shift+F1, then click on the desired control

SONNET TECHNICAL SUPPORT: :

- email: support@sonnetsoftware.com
- Voice: (315)453-3096 Toll Free in US: (877) 776-6638
- Fax: (315) 451-1694
- Web site: www.sonnetsoftware.com/support

CHAPTER 2 INSTALLATION

UNIX and Linux Installation

If you are installing the software on a UNIX or Linux system, please refer to the UNIX and Linux Installation manual provided in your set of hardcopy manuals or PDF manuals in your download. If you are a new Sonnet user, once you have completed your installation return to the tutorials in this manual.

Windows Installation

Requirements

- OS: Windows XP and Windows Vista. For Windows XP, Service Pack 2 is recommended for the best performance of the analysis engine.
- Hardware: Pentium or Athlon AMD processor, 128 Mbytes RAM and 125 Mbytes of disk space.
- To perform the installation on Windows XP or Windows Vista, you must be logged in as the Administrator or as a member of the

administrator group.

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

http://www.sonnetsoftware.com/requirements

Compatibility

This release is backward compatible with all prior releases. However, you may not use a release 11 project file in an earlier release of Sonnet.

License Server

TIP

For a standard evaluation, node-locked license or university license, the local machine is the license server.

For a floating license, one machine in the network should be designated as the license server. This machine requires a hardware key attached to it. Client machines do not require a hardware key. The license manager, FLEXNet, executes on the license server.

For a discussion on types of licenses, please see “Licenses,” page 9 in the **Windows Installation Manual** available in PDF format from the Sonnet task bar.

Windows Installation Instructions

1 Uninstall any previous installations of Sonnet by selecting *Start* ⇒ *Programs* ⇒ *Sonnet* ⇒ *Uninstall*. If this is the first time you’ve installed Sonnet, continue at Step 2.

NOTE: If you want to keep an earlier version of the software be aware that the license server for a previous release may not be run simultaneously with the license server for Version 11 nor can you run an older version using a version 11 license due to an incompatibility of license file formats. The license servers must be installed on different computers or run one at a time.

2 Both new and previous Sonnet User's require a new license for Release 11.52. Go to www.sonnetsoftware.com/license and fill out the License Request Form. Once you receive your license from Sonnet, you will be able to enable your Sonnet Software.

3 Load the software from the CD-ROM on your system, following the installation instructions on your screen.

4 If you are using a hardware key, attach it to your computer, then install the Hardlock driver by selecting *Admin* ⇒ *Hardware Key* ⇒ *Install Driver* from the Sonnet task bar menu and follow the on-screen instructions. To open the Sonnet task bar, select *Start* ⇒ *Program Files* ⇒ *Sonnet* ⇒ *Sonnet* from the Windows start menu.

5 Follow the directions in the licensing information e-mail you received from Sonnet Software in response to the License Request Form.

6 Select *Project* ⇒ *New Geometry* from the main menu of the Sonnet task bar to launch the project editor, ***xgeom***, to verify that the installation is correct.

If your installation is not successful, please refer to the Windows Installation manual in PDF format for instructions on troubleshooting your installation.

For advanced feature installations and setups, please find the applicable installation instructions in PDF format, using the Manuals button on the Sonnet task bar after you have installed Sonnet. If you are viewing the PDF of this manual, click on the appropriate link below.

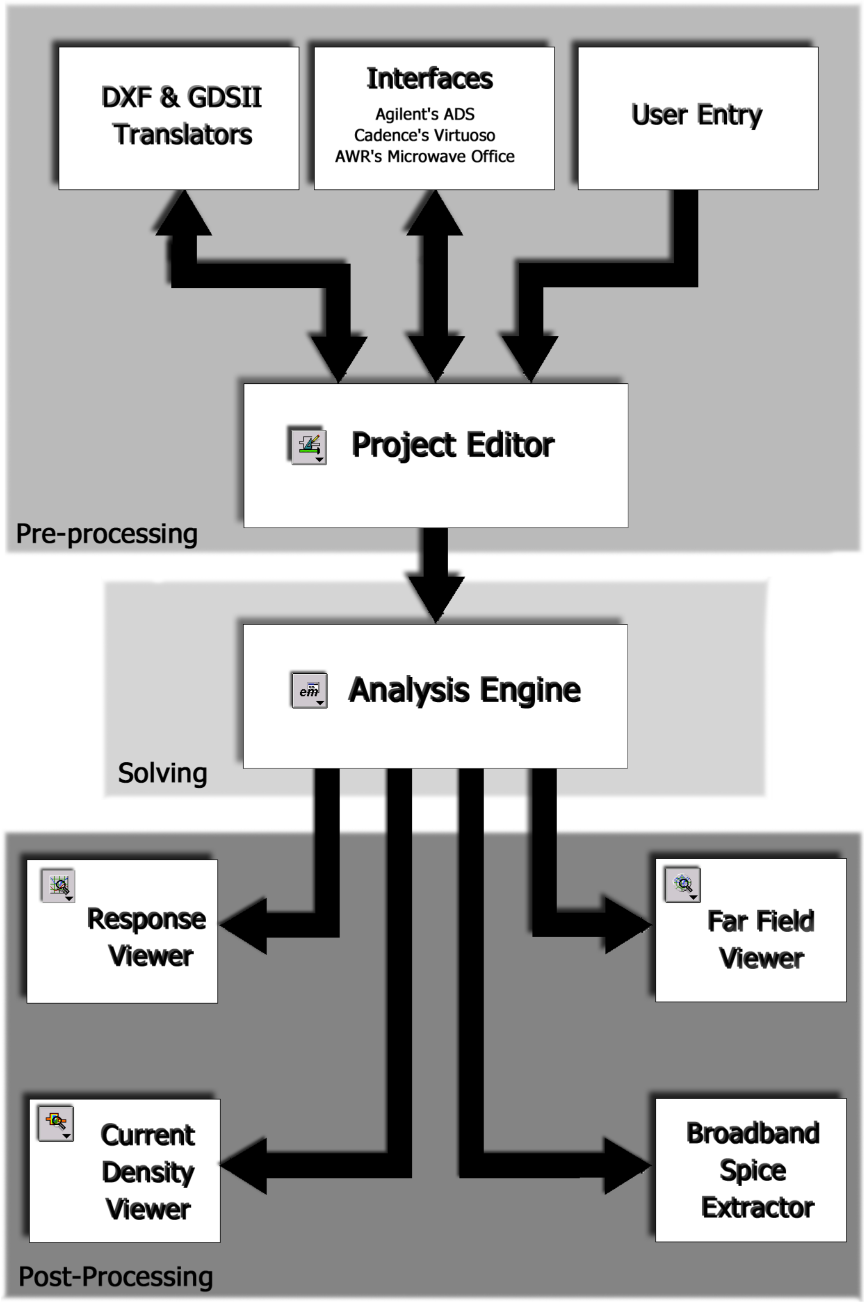
- Remote ***em*** processing
- ***emCluster*** computing

CHAPTER 3 THE BASICS

This chapter is intended to provide you with basic information about the Sonnet Design suite which will allow you to quickly and accurately model your circuit while avoiding some of the common mistakes made during EM simulation. The chapters which follow after contain tutorials on using the Sonnet suite.

The Sonnet Design Suite

The suite of Sonnet analysis tools is shown below. Using these tools, Sonnet provides an open environment to many other design and layout programs.


DXF & GDSII
Translators
Interfaces
Agilent's ADS
Cadence's Virtuoso
AWR's Microwave Office
User Entry
Project Editor
Pre-processing
Analysis Engine
Solving
Response
Viewer
Far Field
Viewer
Current
Density
Viewer
Broadband
Spice
Extractor
Post-Processing

Sonnet Applications

Sonnet's suites of high-frequency electromagnetic (EM) Software are aimed at designs involving predominantly planar (3D planar) circuits and antennas. Predominantly planar circuits include:

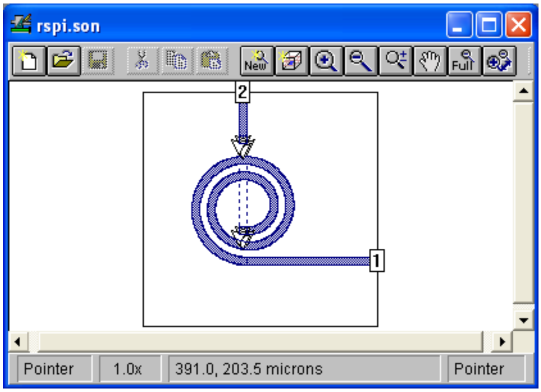
- Arbitrary microstrip & stripline circuits
- Planar spiral inductors
- RFIC & MMIC circuits
- Planar filter analysis
- High Speed Digital Interconnects
- High density interconnects
- Multi-layer circuits like LTCC and PCB technology
- Microwave circuit discontinuities
- Planar (patch) antennas
- Circuits with vias, vertical metal sheets (z-directed strips), and any number of layers of metal traces embedded in stratified dielectric material.

For more information on Sonnet’s applications, see our website at:

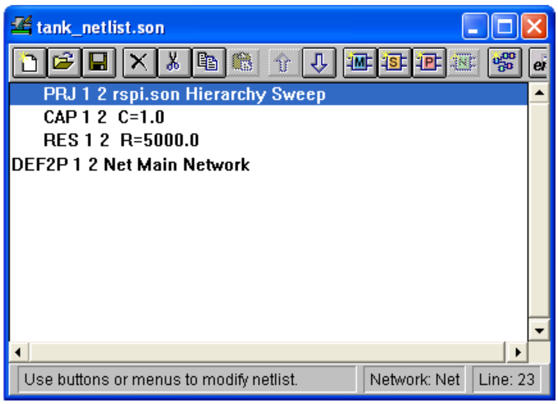
http://www.sonnetsoftware.com/products/em/applications.asp

Sonnet Projects

There are two types of projects in Sonnet: the Geometry Project and the Netlist Project. A geometry project contains a circuit whose properties are entered by the user or provided by translating a file to the Sonnet environment. A netlist project contains a circuit defined by a netlist which consists of one or more networks with elements, which can be other geometry projects, connected together. A geometry project and a netlist project, as they appear in the project editor, are shown below.



Geometry Project



Netlist Project

The most commonly used type of project is the geometry project, and a majority of the documentation concerns the input and analysis of this type of project. The geometry project contains the layout and material properties of a circuit and any internal components. When you analyze a geometry project, ***em*** performs an electromagnetic simulation of your circuit.

The Netlist project contains ideal components, data files and Sonnet geometry projects. A network is defined which connects any combination of these items together. A common use for a netlist project is to connect two or more geometry projects together. The analysis engine, ***em***, uses circuit theory to analyze the network.

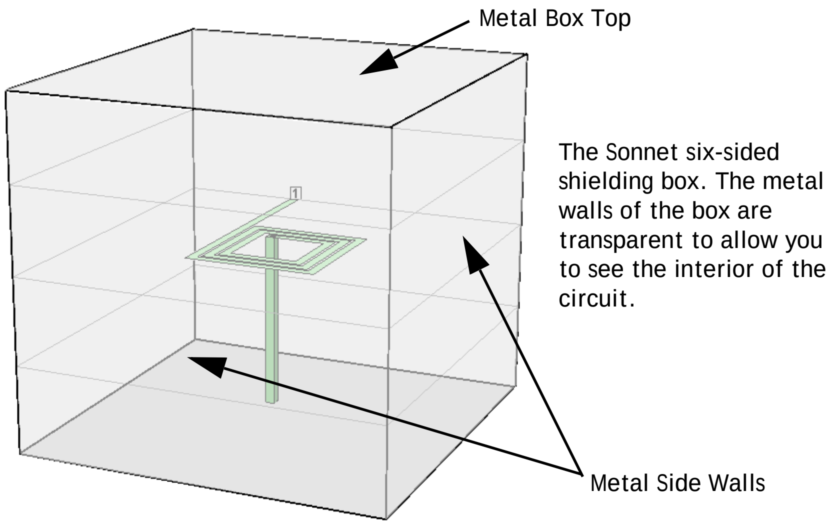
For both types of projects, the analysis controls (such as analysis frequencies, run options, etc.) and analysis data are stored in the project. You use the same Sonnet project for all the Sonnet programs.

In addition to viewing your response within Sonnet, you may also export data files for use in other tools. The table below shows which formats are available.

Table 1 Optional Output Files

Format	Default File Extension	Descriptions
Touchstone	.s2p, .s3p, etc.	Touchstone response file containing S, Y, or Z-parameter data. The file may contain any number of ports.
Databank	.s2p, .s3p, etc.	Databank format response file containing S, Y, or Z-parameter data. The file may contain any number of ports.
SCompact	.flp	SCompact format response file
Spreadsheet	.csv	Comma separated value data for a spreadsheet such as Excel.
Spectre	.scs	Cadence format Spice file - Pi model or Broadband model
PSpice	.lib	PSpice file - Pi model or Broadband model
Agilent MDF	.mdf	Agilent’s Microwave Data Interchange Format
LCT	.lct	N-Coupled Line (Transmission Line) Spice Model

The Sonnet Box



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

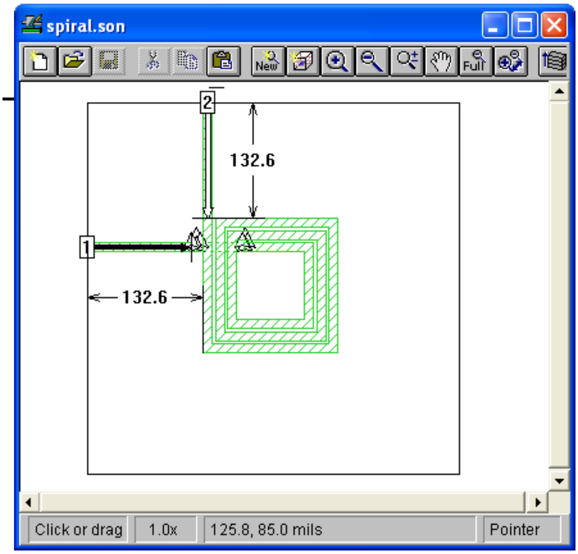
The four sidewalls of the box are lossless metal, which provide several benefits for accurate and efficient high frequency EM analysis:

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

As an example, a microstrip circuit can be modeled in Sonnet by creating two dielectric layers: one which represents your substrate, and one for the air above the substrate. The metal polygons for the microstrip would be placed on the metal level between these two dielectric layers. The bottom of the box is used as the ground plane for the microstrip circuit. The top and bottom of the box may have any loss, allowing you to model ground plane loss.

Coupling to the Box

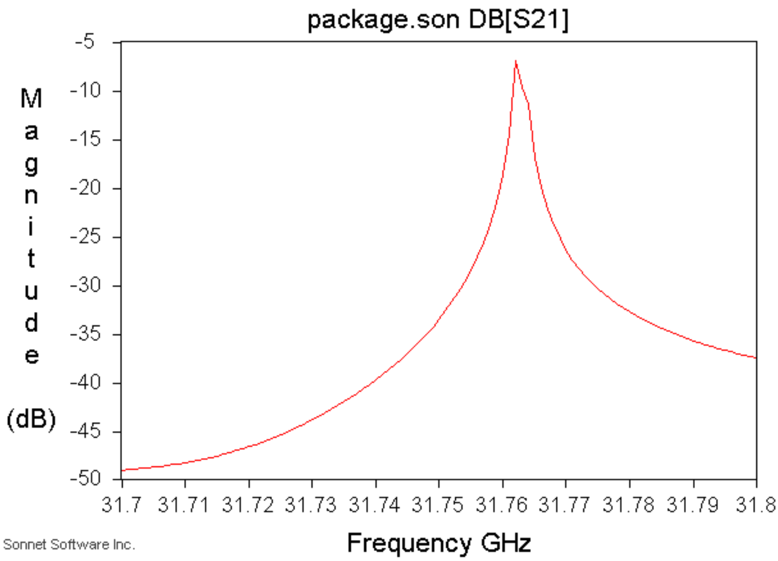
Since the four sidewalls of the box are lossless metal, any circuit metal which is close to these walls can couple to the walls - just like what would happen if you fabricated and measured a real circuit with the same box. If you do not want to model this coupling (for example, your real circuit does not have sidewalls), then you must keep the circuit metal far away from the box sidewalls. A good rule to use is at least three to five substrate thicknesses as shown below.



This circuit has a substrate of 25 mils. The spiral should be kept at least 75-125 mils from the box walls.

Box Resonances

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



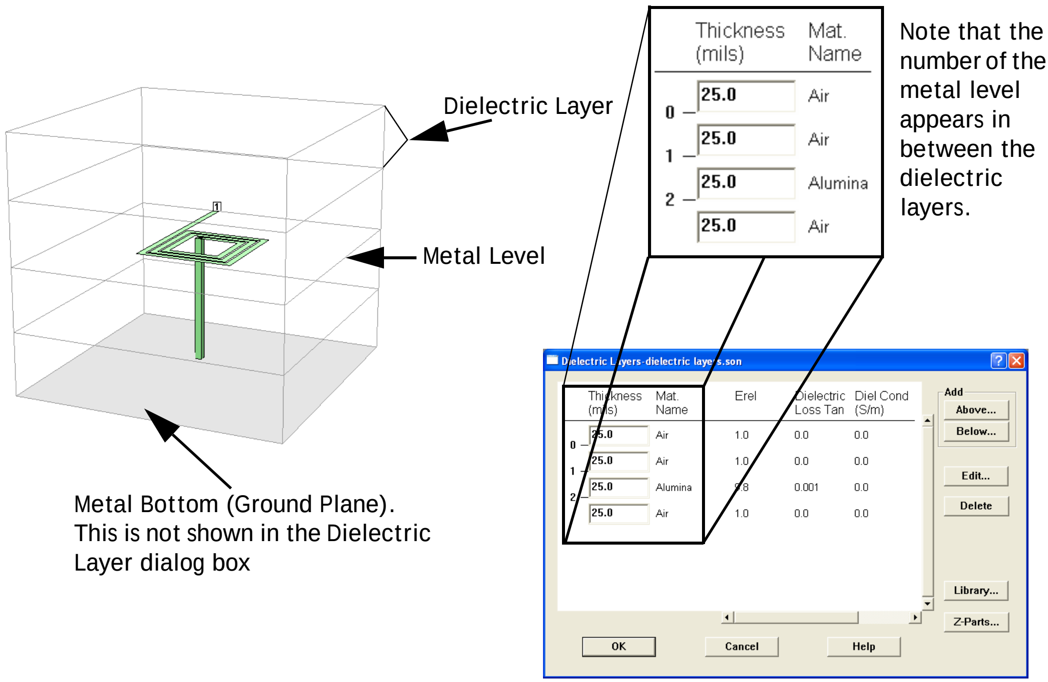
You can detect box resonances by observing your simulated results, through runtime warning messages or by using the Box Resonance Estimator (*Analysis* ⇒ *Estimate Box Resonances*). For a detailed discussion on box resonances and how to detect them, please see **Chapter 23, "Package Resonances"** on page 329 of the **Sonnet User's Guide**.

Dielectric Layers and Metal Levels

All Sonnet geometry projects are composed of two or more dielectric layers. There is no limit to the number of dielectric layers in a Sonnet geometry project, but each layer must be composed of a single dielectric material. Metal polygons are placed at the interface between any two dielectric layers and are usually modeled as zero-thickness, but can also be modeled using Sonnet's thick metal model. Vias may also be used to connect metal polygons on one level to metal on another level.

You will use the Dielectrics dialog box (*Circuit ⇒ Dielectric Layers*), as shown below, to add dielectrics to your circuit. Each time a new dielectric layer is added, a corresponding metal level is also added to the bottom of the new dielectric layer.

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



Below is a glossary of some commonly used terms in Sonnet which relate to the box model.

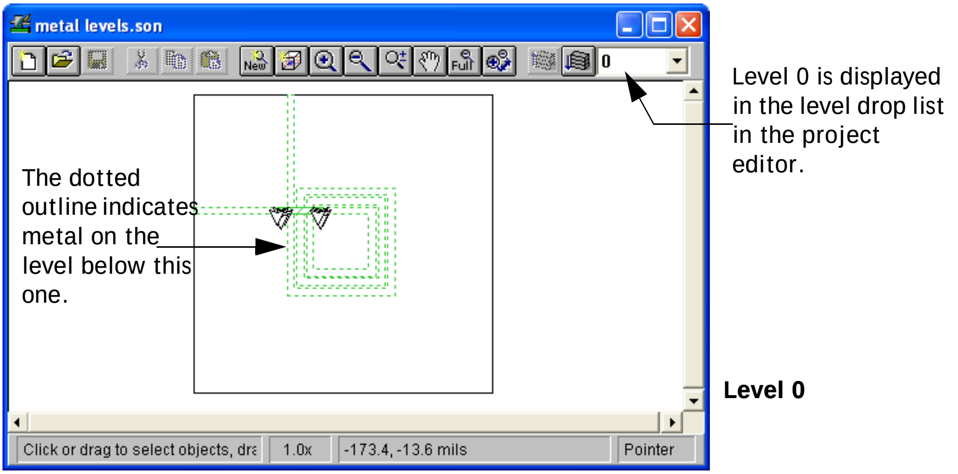
Dielectric Layer: This refers only to dielectrics, NOT metals. In the example above, there are four dielectric layers. There is an entry for each dielectric layer in the Dielectric Layers dialog box (*Circuit ⇒ Dielectric Layers*).

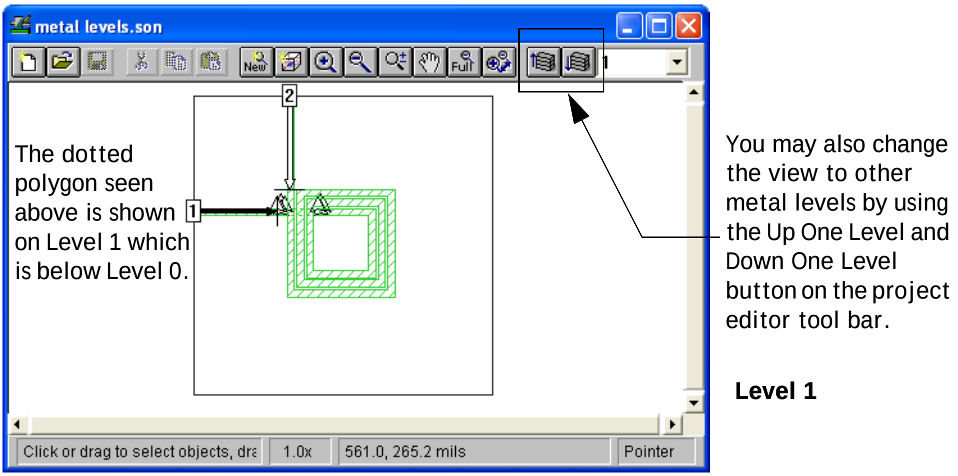
Metal Level: Metal levels are modeled as zero thickness and are attached to the dielectric layer ABOVE them. In the example above, there are three metal levels in addition to the box top and bottom. Since no metal may be placed on the top of the box, it may not be accessed by the user or viewed in the project editor. The bottom of the box is referred to as the GND level and may be accessed in the project editor. It is not labeled in the dielectric window. The top and bottom of the box are lossless metal by default, but can be changed in the Box Settings dialog box (*Circuit ⇒ Box Settings*). You can use as many different metal types as you wish on a single metal level; for instance, you may use a silver polygon and copper polygon on the same metal level.

NOTE: **A "layer" refers to a dielectric layer while "level" refers to the metal level which is sandwiched between the two dielectric layers. So, technically, there is no such thing as a "metal layer" in Sonnet.**

GND Level: You can place polygons on the GND level, but they have no effect because this level is already completely metalized. However, cases do exist in which you may want to place a polygon on the GND level in order to place a via or a dielectric brick there.

Viewing Levels: When you view your circuit in the normal 2D view in the project editor, you are always "on" a particular level, as shown by the level drop list in the Project Editor tool bar as shown below. The top level is always "0" and increases as you move downward through the box. You can switch levels by using your arrow keys, or using the level drop list. By default, all polygons on your present level are shown in solid, and all polygons on all other levels are shown as dashed outlines.





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

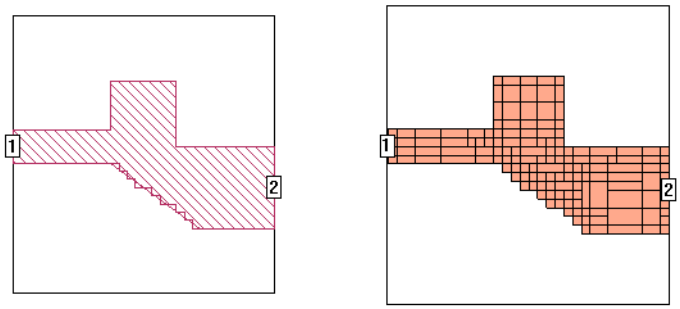
The total height of the box is determined by the sum of the thicknesses of the dielectric layers since metal is either modeled as zero-thickness or protrudes into the dielectric layer above. If you wish to model microstrip circuits, you will need to place a thick layer of air above your circuit; i.e. the topmost dielectric layer should be at least three to five times the substrate thickness.

Circuit Metal

You input your circuit by placing metal polygons on the metal levels of your geometry project. You define what type of metal a given polygon is by defining metal types in your circuit. (*Circuit ⇒ Metal Types*). There is no limit as to how many different metals are used on any given level or how many metal types you may define for a project. To model loss in your metal, you enter the metal's loss characteristics when you define the metal type. It is also possible to use a provided library of metal types or define your own metal library to use in multiple projects.

Cell Size and Meshing

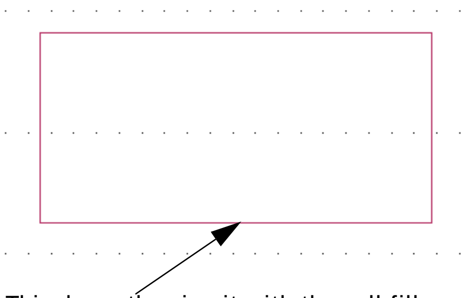
The Sonnet analysis engine, ***em***, calculates the response data for your circuit by dividing the metal into small "subsections" and analyzing the coupling between these subsections.



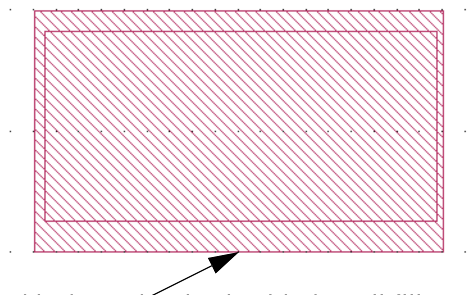
The picture on the left shows the circuit as viewed in the project editor. On the right is shown the subsectioning used in analyzing the circuit.

The Sonnet subsectioning is based on the uniform cell size which is represented in the project editor by small dots. The small dots are placed at the corners of a "cell" as shown in the illustration below. One or more cells are automatically combined together to create subsections. Cells may be square or rectangular (any aspect ratio), but must be the same over your entire circuit. Note that the cell dimensions do not have to be an integer nor do they have to be equal. For example, you may define a cell size of 1.687 by 2.453 microns.

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.



This shows the circuit with the cell fill turned off. The outline shows the polygon drawn by you. Notice that this polygon is not "on" the grid.



This shows the circuit with the cell fill on. The patterned area shows the actual metal that is analyzed by the analysis engine, *em*. Note that this metal is always "on" the grid.

The cell size is specified in the project editor in the Box Settings dialog box which is opened by selecting *Circuit* => *Box* from the project editor menu. The analysis solves for the current on each subsection. Since multiple cells are combined together into a single subsection, the number of subsections is usually considerably less than the number of cells. This is important because the analysis solves an N x N matrix where N is the number of subsections. A small reduction in the value of N results in a large reduction in analysis time and memory.

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

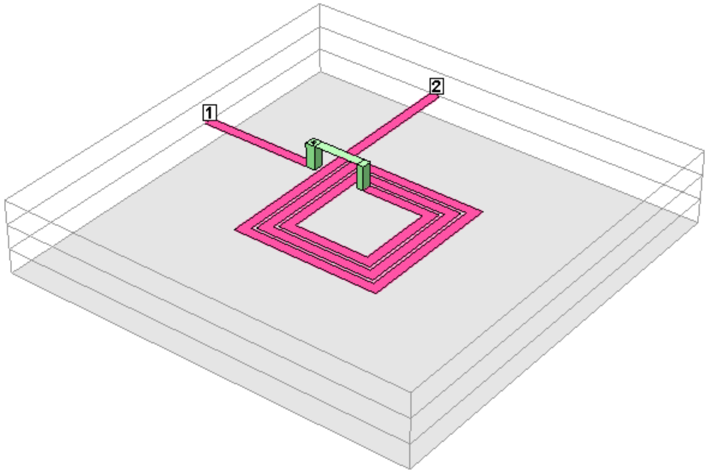
For a more detailed discussion of cell size and subsectioning, please refer to Chapter 3, **Subsectioning**, on page 31 of the **Sonnet User's Guide.**

Tips

The following items may not look very important to you now, but will be very useful further along in your use of Sonnet.

3D View

It is often helpful to view your circuit in 3D. To do so, select the *View* ⇒ *3D View* command. An example of a 3D view is shown below.



Estimate Memory

Once you are done specifying your circuit in the project editor, you may determine if it is possible to analyze your circuit with the computer resources available to you. Selecting *Analysis* => *Estimate Memory* from the project editor's main menu opens the Estimate Memory dialog box and subsections your project. A simple status popup appears to indicate that the circuit is being subsectioned. When the subsectioning is complete, the memory use estimate and the subsectioning information, listed by level and total amounts, appears in the dialog box. You may also view the subsectioning used by ***em***.

CHAPTER 4 TUTORIAL: A QUICK TOUR

This tutorial provides you with an overview of the Sonnet products and how they are used together. The second tutorial, in Chapter 5, walks you through a simple example from entering the circuit in the graphical project editor, to viewing your data in the response viewer, our plotting tool. The third tutorial covers design issues which need to be considered in the project editor.

There are tutorials on different Sonnet features in the **Sonnet User's Guide** as well as tutorials on the use of the translators in the Translator manual.

The first tutorial is designed to give you a broad overview of the Sonnet suite, while demonstrating some of the basic functions of the Sonnet products. The following topics are covered:

- Invoking Sonnet programs.
- Opening a circuit geometry project in the project editor.
- Running a simple analysis in ***em***.
- Plotting S-parameter data in the response viewer.
- Performing a simple animation in the current density viewer.

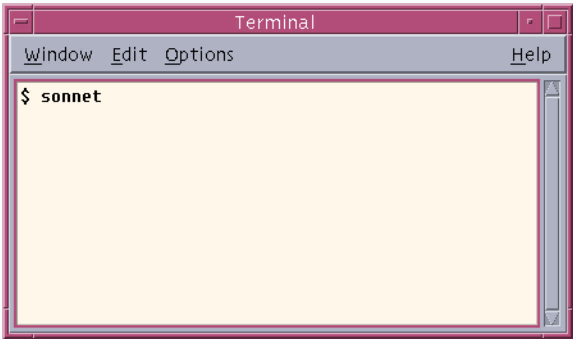
Invoking Sonnet

You use the Sonnet task bar, shown below, to access all the modules in the Sonnet Suite. Opening the Sonnet task bar, for both Windows and UNIX systems is detailed below.

UNIX

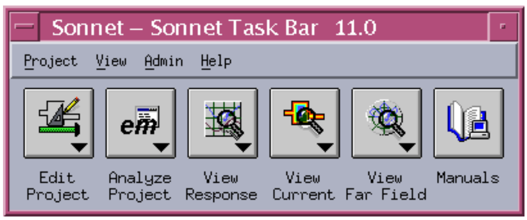
1 **Open a terminal.**

If you do not know how to do this, please see your system administrator.



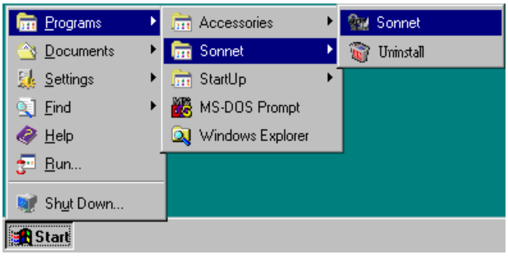
2 **Enter "sonnet" at the prompt.**

The Sonnet task bar appears on your display.

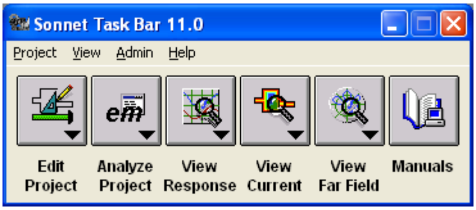


Windows

1 **Select *Start* ⇒ *Programs* ⇒ *Sonnet* ⇒ *Sonnet* from the Windows desktop Start menu.**



The Sonnet task bar appears on your display.



Obtaining the Example Projects

Example projects are supplied with your software and are available through the PDF manuals on your computer. The Application Examples manual, when accessed in PDF format on your computer, allows you to open the project in the Project Editor or, in the case of multifile examples, copy the example folder to your personal working directory.

NOTE: **You must have Adobe Acrobat Reader installed on your system to access the manuals in PDF format. If you do not have the program, it is available for installation by selecting *Admin* ⇒ *Install Acrobat* from the Sonnet Task Bar main menu.**

To copy the Filtwall project to use in this tutorial, do the following:

1 **Click on the Manuals button on the Sonnet Task Bar.**

The file sonnet_online.pdf is opened on your display. The manuals available in PDF format are identical to the hard copy manuals which came with your installation package.

2 Click on the Application Examples button in the PDF document.

3 Click on the Complete List button.

A page appears with a complete list in alphabetical order of the example files.

4 Click on Filtwall in the list.

This will take you to the Filtwall example project in the Application Examples manual.

5 Click on the Load into Project Editor button at the top of the page.

The project editor is invoked on your display with the file “filtwall.son” open.

This file is a read-only to prevent corrupting the example files.

6 Select File ⇒ Save As from the project editor main menu.

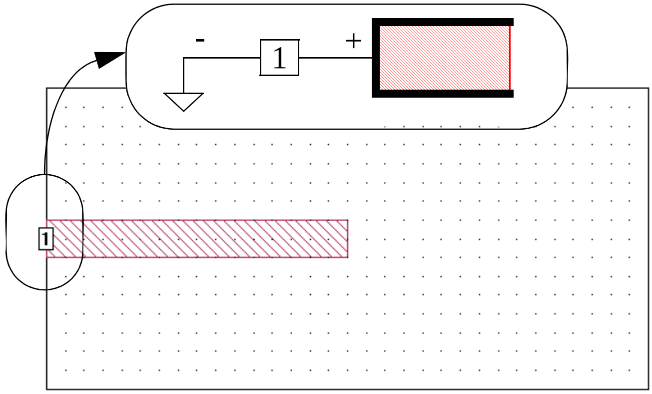
The Save As browse window appears. Save the project in your working directory. This allows you to save any changes you make to the circuit.

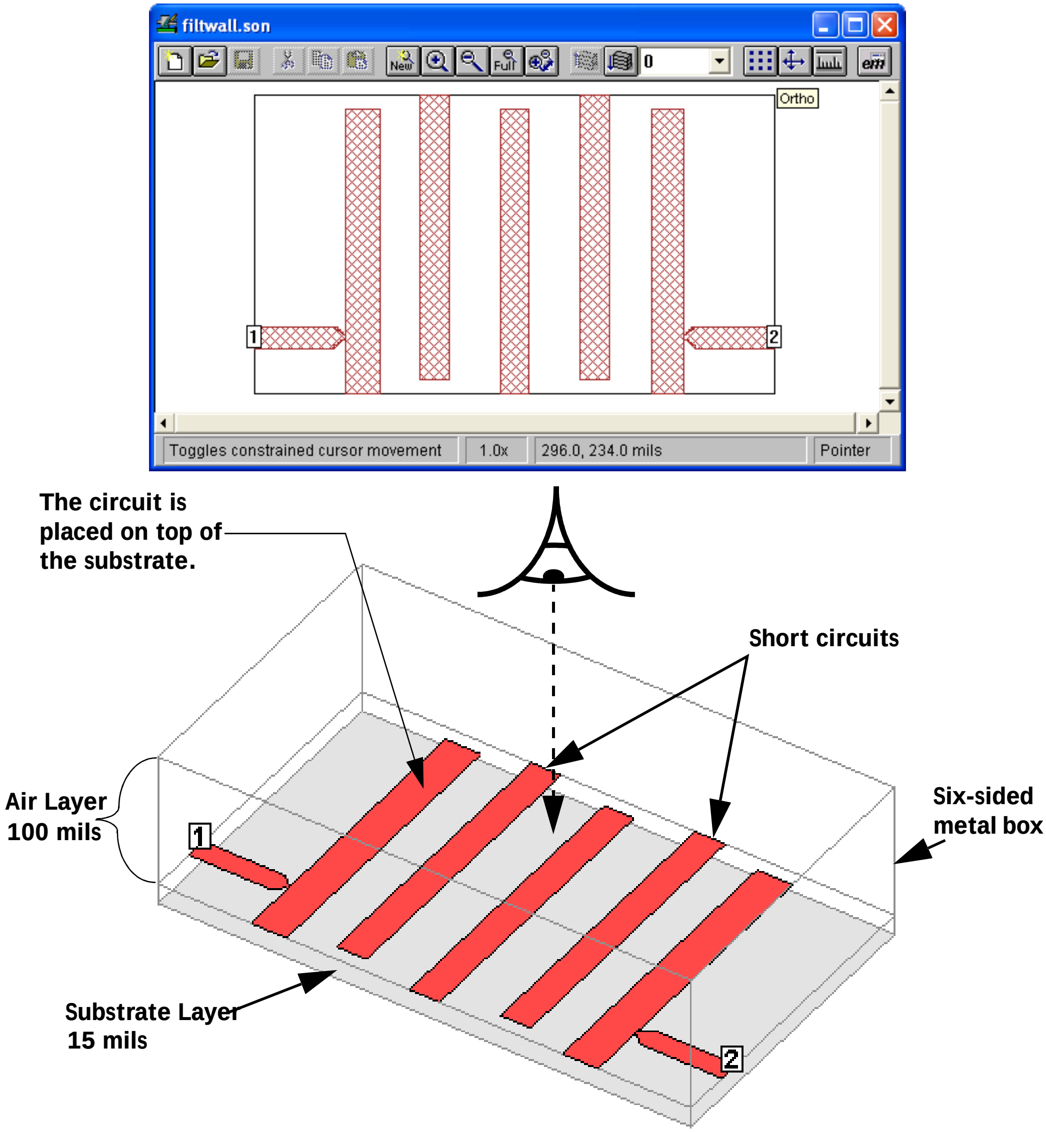
The Project Editor

The project editor, when editing a circuit geometry project, provides you with a graphical interface which allows you to specify all the necessary information concerning the circuit you wish to analyze with the electromagnetic simulator, ***em***.

The example circuit, “filtwall.son”, shown on page 34, is a two port microstrip filter with a 15 mil Alumina substrate and 100 mils of air above. Note that the resonators run to the edge of the substrate, shorting them to the box wall.

Circuits in Sonnet are modeled as being enclosed in a six-sided metal box which is ground. Any circuit metal touching the box is shorted to ground. This does not apply to the two ports shown on the circuit. 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 model for a box wall port is pictured below.





A three-dimensional view of the circuit in the six-sided metal box modeled in the project editor. The view shown in the project editor window is a two-dimensional view from the top looking directly down on the substrate. This view is from the 3D viewer which is opened by selecting *View* ⇒ *View 3D* from the project editor main menu.

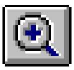
Zoom and Cell Fill

The project editor has a zoom feature which allows you to take a closer look at any part of your circuit. You will zoom in on a section of the filter, to observe an example of staircase cell fill.

The cell fill represents the actual metalization analyzed by ***em***. There are various types of cell fill, one of which is staircase. Staircase uses a “staircase” of cells to approximate any angled edges of polygons. Therefore, the actual metalization analyzed by ***em*** may differ from that input by you.

7 Select *View* ⇒ *Cell Fill* from the project editor main menu to turn off the cell fill.

This command toggles the state of cell fill. Only the outline of the polygons are displayed.

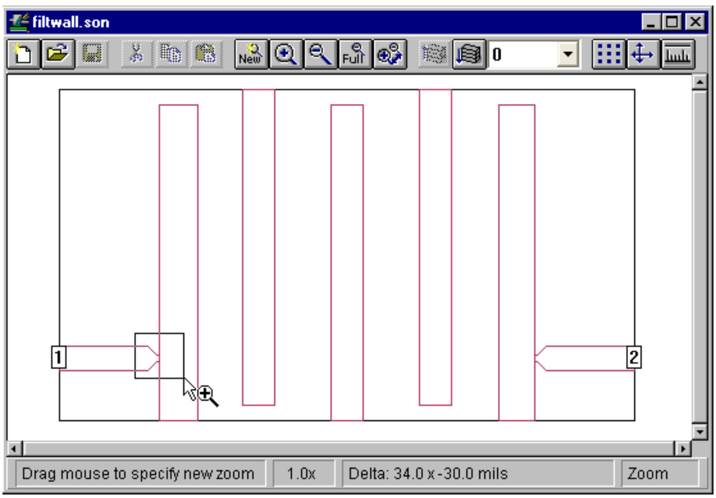


8 Click on the Zoom In button on the tool bar.

The appearance of the cursor changes. A magnifying glass with a plus sign, the icon on the Zoom In button, appears next to the cursor.

9 Select the section of the circuit you wish to zoom in on.

Move the mouse to the upper left of the junction of the feed line and the first resonator, as shown below. Click and drag the cursor down and to the right. A rubber band surrounding the area follows the cursor. When you release the mouse button, the project editor zooms in on the selected area.

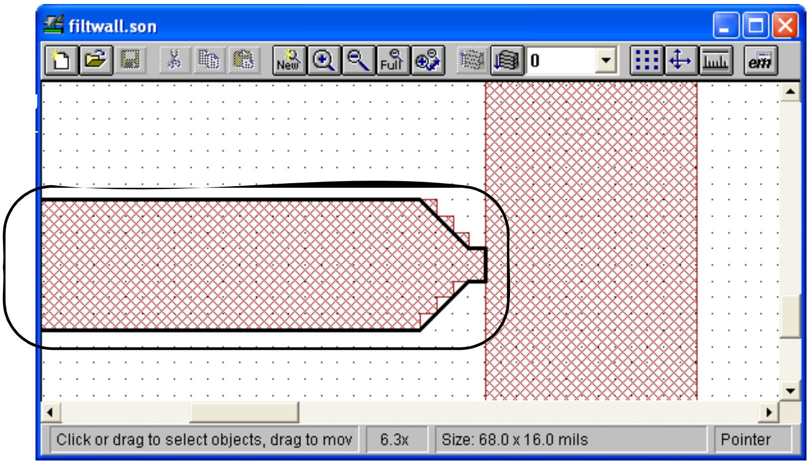


As you can see in the enlargement on page 36, one of the metal polygons has a diagonal edge.

10 Select *View* ⇒ *Cell Fill* from the project editor main menu to turn on the cell fill.

The metal fill pattern appears in the polygons on your display. Note that the outline of the actual metal has a “staircase” edge which approximates the diagonal edge drawn in the project editor. The default setting in the project editor is staircase fill.

Note that ***em*** analyzes the staircase edged metal, not the diagonal polygon.



NOTE: Diagonal fill is not available in Sonnet Lite and Sonnet LitePlus. If you are using either of these suites, skip to step 15. This will not affect simulation results since staircase fill is restored before running the analysis.

11 Click on the polygon circled above to select it.

The polygon is highlighted to indicate selection.

12 Select *Modify* ⇒ *Metal Properties* from the project editor main menu.

The Metalization Properties dialog box appears on your display as shown below.

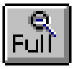
13 Select Diagonal from the Fill Type drop list.

Diagonal fill type, while more accurately modeling the input edge, requires more processing time.

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

The metal fill is updated. As you can see, the metal ***em*** analyzes is closer to your input than in the case of staircase fill. But the increased accuracy comes at the price of increased processing time.

In this case, staircase fill provides the required degree of accuracy and will be used for the analysis. The fill type will be changed back to staircase later in the tutorial.



15 Click on the Full View button.

The whole circuit appears on your display.

Metal Types

The project editor allows you to define any number of metal types to be used in your circuit. Multiple metal types may be used on any given level. A metal type specifies the metalization loss used by ***em***. Both the DC resistivity and the skin effect surface impedance are accurately modeled in ***em***.

Metal types are defined in the Metal Types dialog box where a fill pattern is assigned as part of the definition. For a detailed discussion of metal types and loss, see Chapter 4, “Metalization and Dielectric Layer Loss” in the Sonnet User’s Guide.

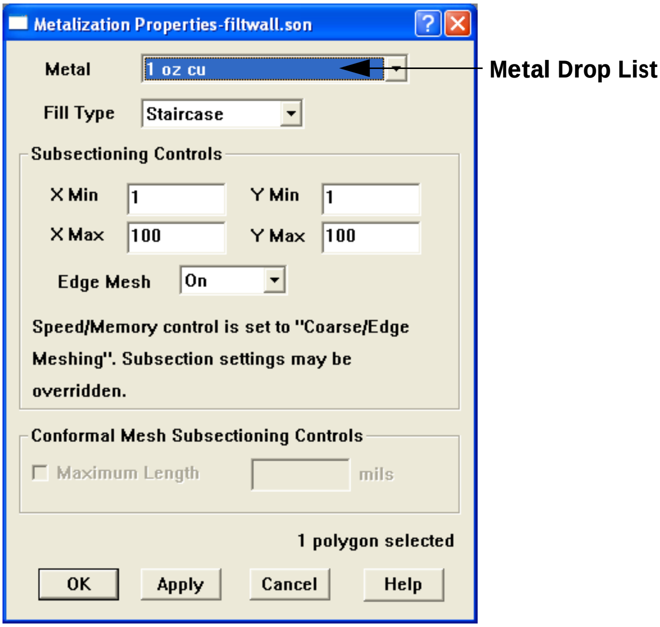
After a polygon is drawn in your circuit, you can change the metal type. In our example, all the polygons are comprised of Copper metal. An example is given below of changing the metal type of one polygon to “Lossless” metal.

16 Click on any resonator of the filter to select it.

The polygon is highlighted to indicate selection.

17 Select *Modify* ⇒ *Metal Properties* from the project editor main menu.

The Metalization Properties dialog box appears on your display as shown below.



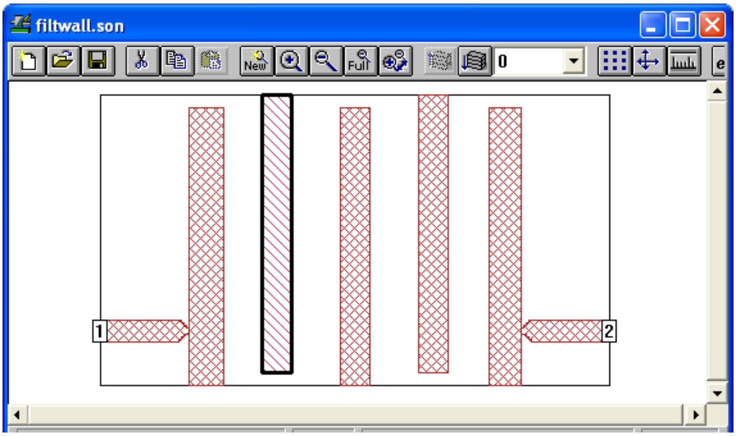
NOTE: **Note that depending which Sonnet Suite you have purchased, some controls may be disabled.**

TIP

Double-clicking on a polygon, port, via or component in your circuit opens the appropriate Properties dialog box.

18 Click on the Metal drop list and select "Lossless" from the list. Click on the OK command button to apply the change and close the dialog box.

The fill pattern changes for the selected polygon as shown below.



The metal for this polygon now has the loss associated with the default lossless metal. The project editor allows you to define any number of metal types for use in your circuit. For details on defining metal types, see the project editor's online help.

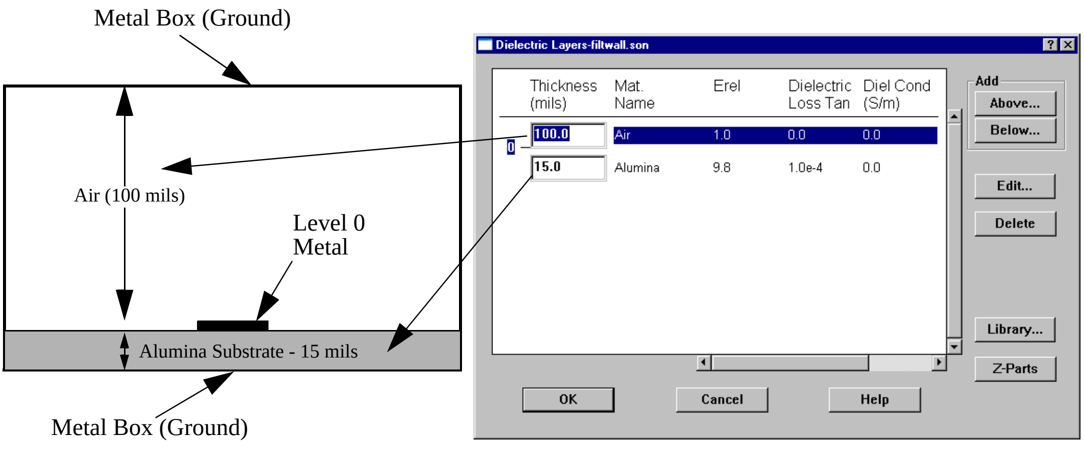
You will not be analyzing with lossless metal; the metal is switched back later in the tutorial.

Dielectric Layers

19 Select *Circuit* ⇒ *Dielectric Layers* from the project editor main menu.

The Dielectric Layers dialog box, which allows specification of the dielectric layers in the box, appears on your display, providing you with an approximate "side view" of your circuit. The project editor "level" number appears on the left. A "level" is defined as the intersection of any two dielectric layers and is where your circuit metal is placed.

.



Refer to page 34 for a three-dimensional drawing of the dielectric layers. The air layer is required to keep the metal top of the box away from the circuit metal.

20 Click on the OK command button to close the Dielectric Layers dialog box.

21 Select *File* ⇒ *Revert to Saved* from the project editor main menu.

This returns “filtwall.son” to the original form which you copied from the example directory. This step ensures that you are analyzing the original file and that you are able to successfully launch ***em*** from the project editor.

Em - The Electromagnetic Simulator

Em performs electromagnetic analysis for arbitrary 3-D planar geometries, maintaining full accuracy at all frequencies. ***Em*** is a “full-wave” analysis engine which takes into account all possible coupling mechanisms.

The project editor provides an interactive windows interface to ***em***. This interface consists of menus and dialog boxes which allow you to select run options, set up the analysis type and input parameter values and optimization goals. You may save the settings for an analysis file in a batch file using the analysis monitor.

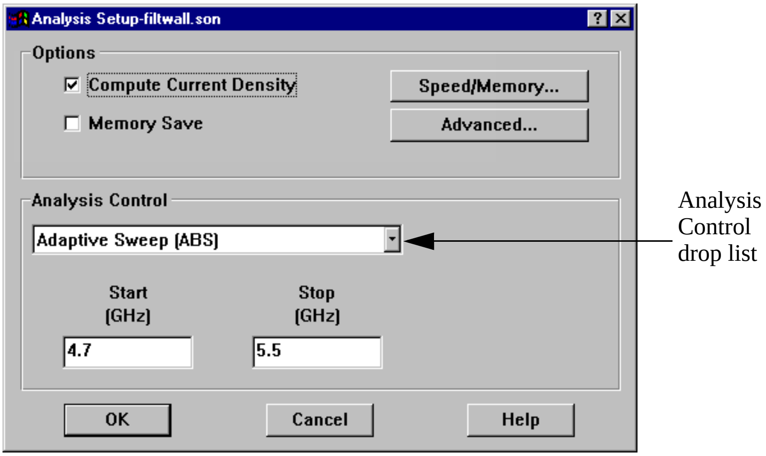
In the next part of this tutorial, you set up the analysis for and analyze the circuit “filtwall.son” which you examined in the project editor. If you have not already done so, load “filtwall.son” into the project editor.

Setting Up the Analysis

These analysis controls have already been input to this example file, but the steps are explained in order to show how the information is entered. You will be using the Adaptive Band Synthesis (ABS) technique to analyze the circuit. ABS provides a fine resolution response for a frequency band with a small number of analysis points. ***Em*** performs a full analysis at a few points and uses the resulting internal data to synthesize a fine resolution band. This technique, in most cases, provides a considerable reduction in processing time. The points at which a full analysis is performed are referred to as discrete data. The remaining data in the band is referred to as adaptive data. For a detailed discussion of ABS, see Chapter 9, “Adaptive Band Synthesis (ABS)” in the **Sonnet User’s Guide.**

22 Select *Analysis* ⇒ *Setup* from the project editor main menu.

The Analysis Setup dialog box appears on your display as shown below.



Setting Up Analysis Frequencies

23 Select Adaptive Sweep (ABS) from the Analysis Control drop list.

Since the default type of sweep is Adaptive, you will not need to take any action for this step. The adaptive sweep uses ABS to analyze the circuit. This enables the Start and Stop text entry boxes.

24 Enter the Start, and Stop values in the appropriate text entry boxes.

The frequency band you wish to analyze is from 4.7 GHz to 5.5 GHz, therefore, enter 4.7 in the Start text entry box, and 5.5 in the Stop text entry box. These entries may already be present.

If you needed to change the frequency units, you would do so using the Units dialog box, accessed by selecting *Circuit* ⇒ *Units* from the project editor main menu.

Analysis Run Options

Run options for ***em*** are available in the Analysis Setup dialog box. This example uses the De-embed option, which is set by default, and the Compute Current Density option. De-embedding is set in the Advanced Options dialog box which you access by clicking on the Advanced button in the Analysis Setup dialog box. Since it is set by default, you do not need to do this for this example.

De-embedding is the process by which the port discontinuity is removed from the analysis results. Inaccurate data may result from failing to implement this option, even when you are not using reference planes. For a detailed discussion of de-embedding refer to Chapters 7 and 8 in the **Sonnet User's Guide**.

If this option is on, de-embedded response data is output to the project file.

The Compute Current Density run option instructs ***em*** to calculate current density information for the entire circuit which can be viewed using the Current Density Viewer. Be aware that for an adaptive sweep, current density data is only calculated for discrete data points.

TIP

The Memory save option (not used in this example) uses less memory by storing matrix elements as single precision numbers rather than double precision. Its use is recommended in order to execute a simulation that otherwise might not be possible within the bounds of your memory limitations.

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

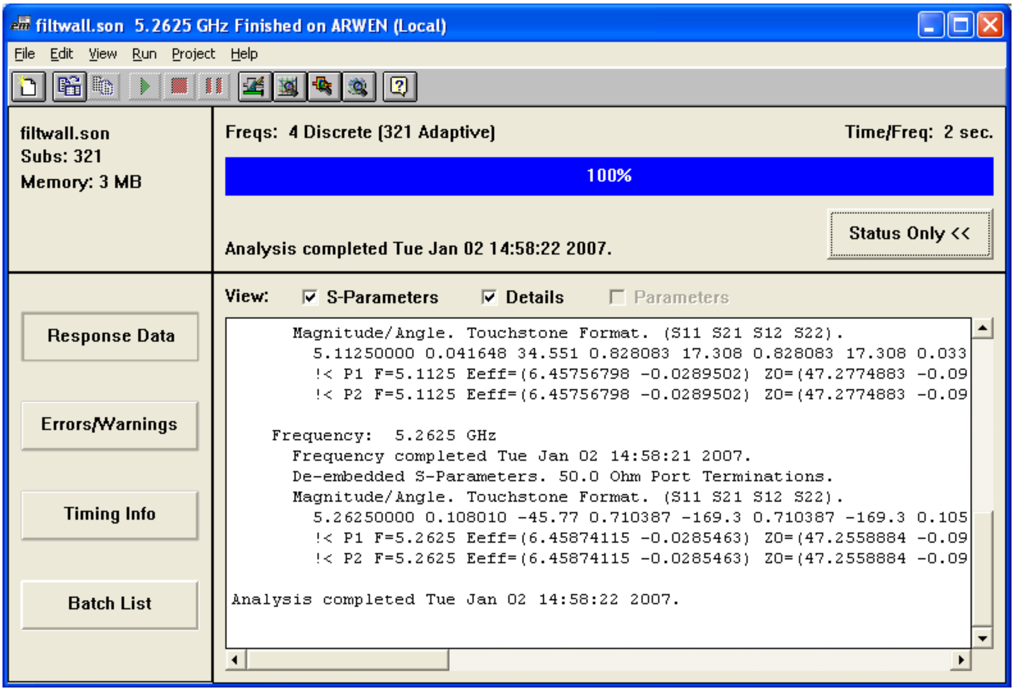
A Current Density Notice may appear on your display; if you do not wish this warning to appear again, select the Don't show me this again checkbox. Click OK to close this notice.

Executing an Analysis Run

The control frequencies and run options are all specified. The analysis time for the circuit will vary depending on the platform on which the analysis is performed.

26 Select *Project* ⇒ *Analyze* from the project editor main menu to invoke *em* and run the analysis.

The analysis monitor appears on your display with the output window displaying response data. The analysis time will vary depending on the platform on which you are running the software. This analysis needs only four discrete frequencies to create the adaptive data and complete the analysis. When the analysis is complete, the message "Analysis completed" followed by the data and time appears in the analysis monitor as shown below.



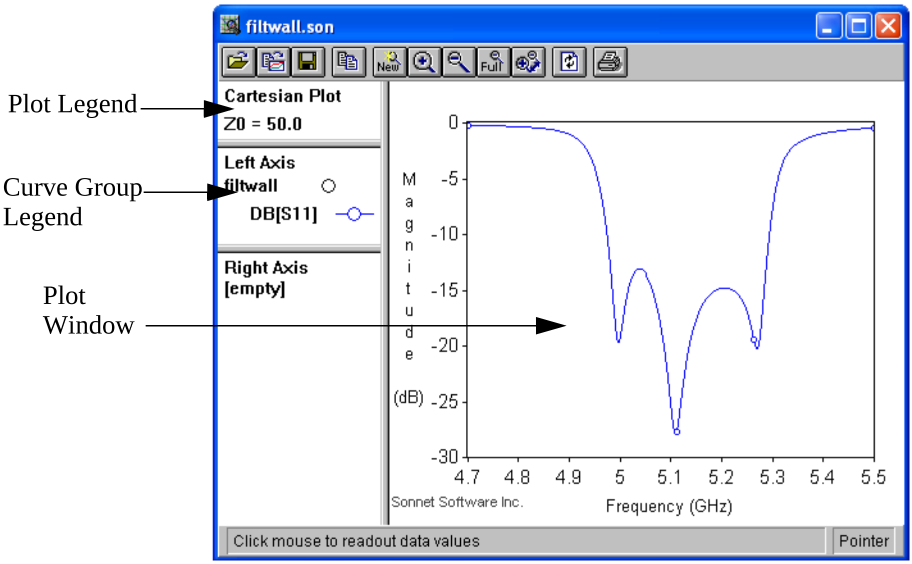
The Response Viewer- Plotting Your Data

The response viewer is used to display the results from an ***em*** analysis as either a Cartesian graph or Smith chart. S-, Y- and Z-Parameters can be displayed alone or simultaneously as well as transmission line parameters. You can also display multiple curves from multiple projects on a single plot or choose to open multiple plots at the same time. It is also possible to display parameter sweeps and optimization results in a number of advanced ways. This tutorial covers only the most basic of the response viewer's functions.

Invoking the Response Viewer

27 **Select *Project* ⇒ *View Response* ⇒ *New Graph* from the project editor main menu to invoke the response viewer.**

The response viewer window appears on your display with a Cartesian graph of the curve group filtwall with curve DB[S11] displayed. The project file contains multiple curves which are displayed in user-defined curve groups.



A data point is the measurement for a particular frequency. The data for all frequencies makes up a measurement. A measurement is a set of response data uniquely identified by parameter type, response type, port numbers and project file. An example would be:

DB[S11] in the filtwall.son file

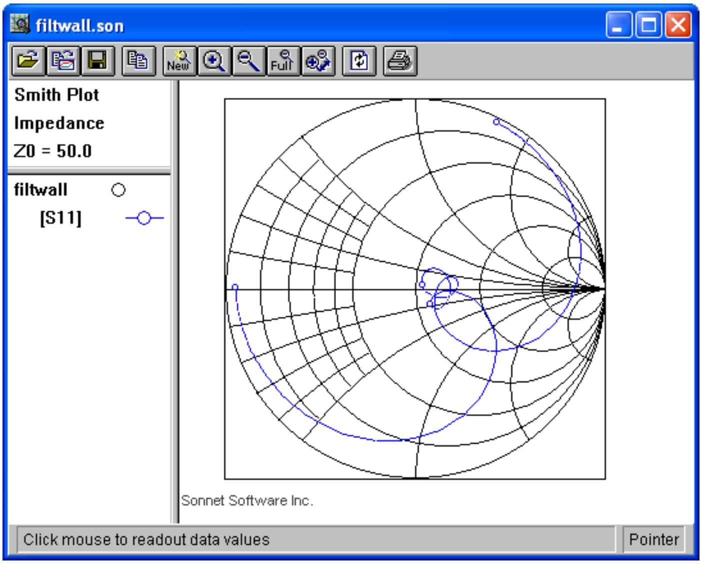
DB identifies the response type as magnitude in dB. S identifies the parameter type as an S-Parameter. 11 identifies the output port as Port 1 and the input port as Port 1. The project file from which the measurement originated is “filtwall.son.” A curve group is made up of one or more measurements.

Different curve groups may come from different project files yet be displayed simultaneously

Displaying a Smith Chart

28 Change the plot to a Smith Chart by selecting *Graph* ⇒ *Type* ⇒ *Smith* from the response viewer main menu.

The plot is changed to a Smith chart of “filtwall.son” which appears on your display. Again, the curve group filtwall with the DB[S11] curve is displayed. This curve group is the default displayed upon startup.



TIP

You can right-click on most parts of the response viewer window to invoke pop-up menus. For example, right clicking in the plot window and selecting *Type* ⇒ *Smith* from the pop-up window also displays a Smith Chart.

Selecting Another Curve for Display

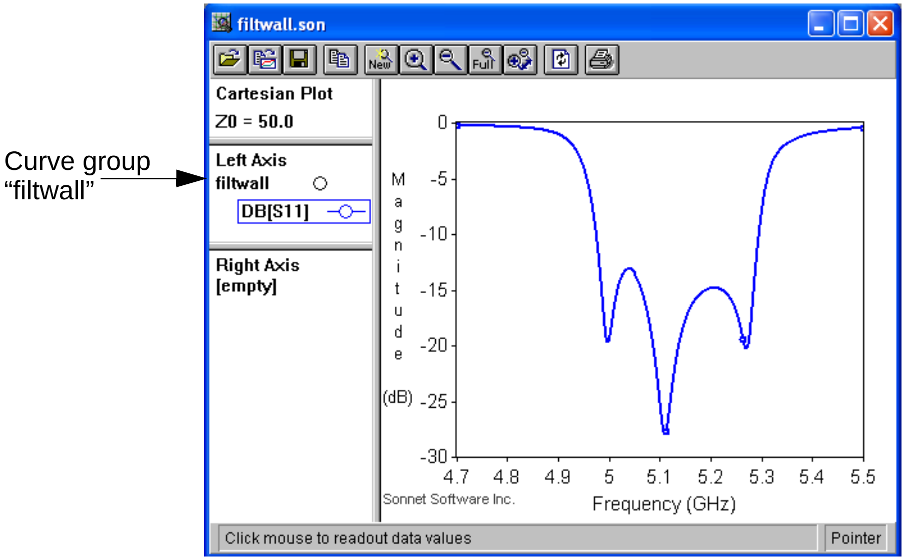
You may display multiple curves from multiple files simultaneously. Curve groups contain one or more measurements. To add a different measurement for display in a curve group, perform the following.

29 Select *Graph* ⇒ *Type* ⇒ *Cartesian* from the main menu.

This changes the plot back to a Cartesian graph.

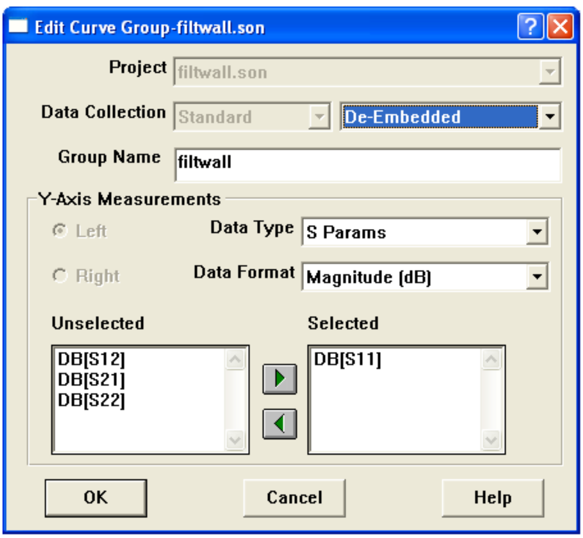
30 Click on the curve group, filtwall, in the curve group legend to select it.

A box appears around the curve group name to indicate its selection.



31 Select *Curve* ⇒ *Edit Curve Group* from the response viewer main menu.

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



TIP

Double-clicking on the Curve Group in the Curve Group legend also opens the Edit Curve Group dialog box.

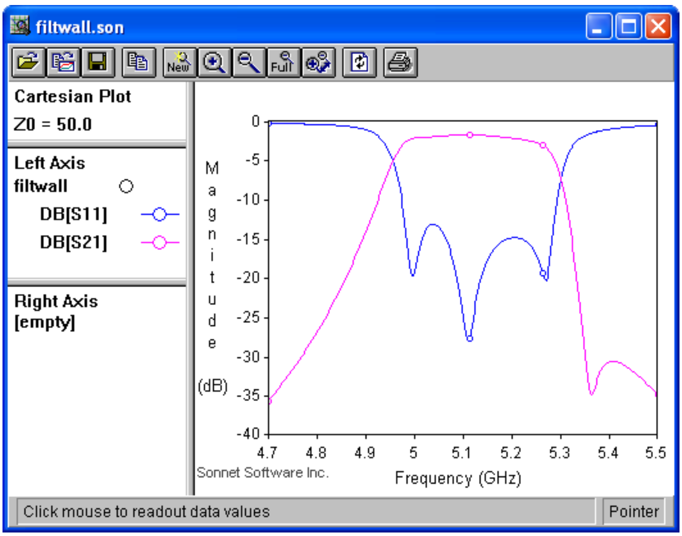
If this project file contained parameter combinations or optimization data, the dialog box would include a section that allowed you to select parameter combinations to display.

32 Double-click on DB[S21] in the Unselected list to move it to the Selected list.

DB[S21] moves to the Plotted list.

33 Click on the OK command button to apply these changes.

The Edit Curve group dialog box is closed and the Cartesian graph is updated to display the curve group filtwall, which now contains the two curves, DB[S11] and DB[S21]. Since the axes are set to autoscale, the values on the axes are automatically adjusted. The graph should appear similar to that pictured below.



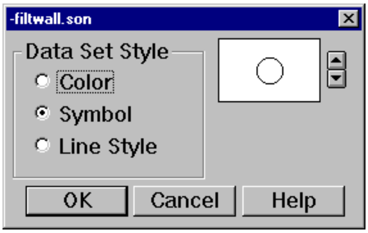
Note that very few of the symbols (O), indicating data points, appear on the curves. When you are displaying adaptive data from an ABS analysis, the only points highlighted with a symbol in the response viewer are those from the discrete points where a full analysis was performed.

34 Right-click on the curve group name, filtwall, which appears in the Curve Group legend and select Properties from the pop-up menu which appears on your display.

The Data Set Style dialog box appears on your display.

35 Click on the Symbol radio button in the dialog box.

The dialog box should appear similar to that shown below.

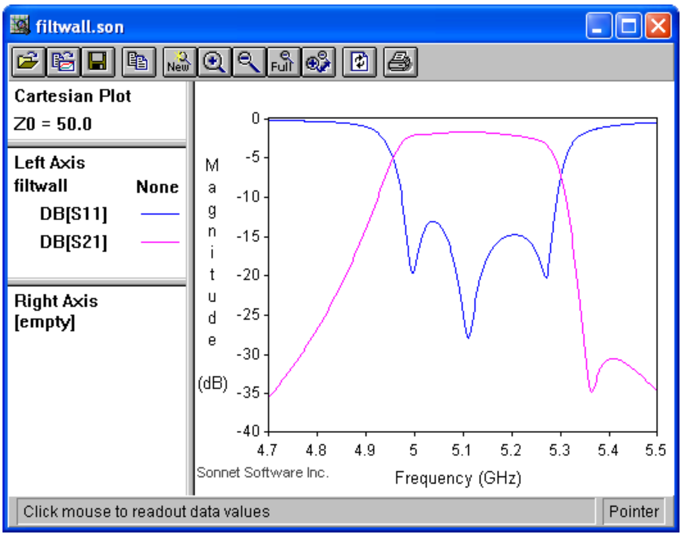


36 Click on an arrow button next to the Symbol display until the word "None" appears.

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

The symbols for the discrete data points are no longer displayed. Only the line represents all the adaptive data.

The graph should appear similar to that pictured below.



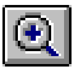
Data Readouts - Smith Chart

The response viewer can provide a readout on any given data point on your plot in two different ways. The information provided is dependent upon the type of data point selected. In the next section, you see how to obtain data on the Smith chart.

38 Right-click in the plot window, then select *Type* ⇒ *Smith* from the pop up menu which appears on your display.

The plot is changed to a Smith Chart. It is important to note that the curve group filtwall only contains one curve: DB[S11]. The curve groups defined for Cartesian graphs are independent of those defined for Smith charts.

Zooming in on a section of the Smith chart makes it easier to distinguish individual points.



39 Click on the Zoom In button on the tool bar.

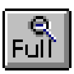
A change in the cursor indicates that you are in zoom in mode.

TIP

If you have a three-button mouse, clicking on the center mouse button when the cursor is in the plot window, will put you in zoom in mode.

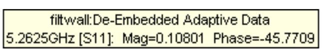
40 Click and drag your mouse in the Smith chart until the rubber band surrounds the area you wish to magnify. Then release the mouse button.

The display is updated with a magnified picture of the selected area. To return to the full graph, click on the Full View button on the tool bar. Zooming operates in the same manner on a Cartesian graph. For our example, select one of the end points of the graph to enlarge.



41 Move the cursor over a discrete data point in the Smith Chart.

A discrete data point is indicated by the O symbol. A popup appears, similar to that shown below, on your display with data about that point.

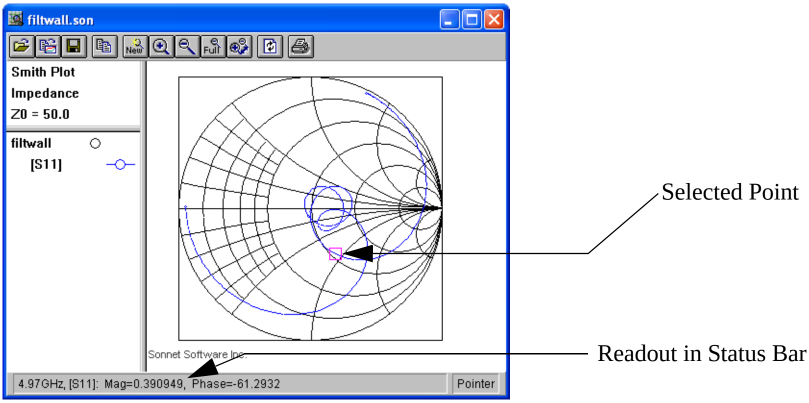


42 Click on the Full View button on the tool bar.

The full view of the Smith chart appears in the response viewer window.

43 Click on a data point in the Smith chart.

The readout for that data point appears in the Status Bar at the bottom of the response viewer window.



To move between points on a curve use the → and ← keys. To move between curve groups on the display use the ↑ and ↓ keys.

Adding a Data Marker

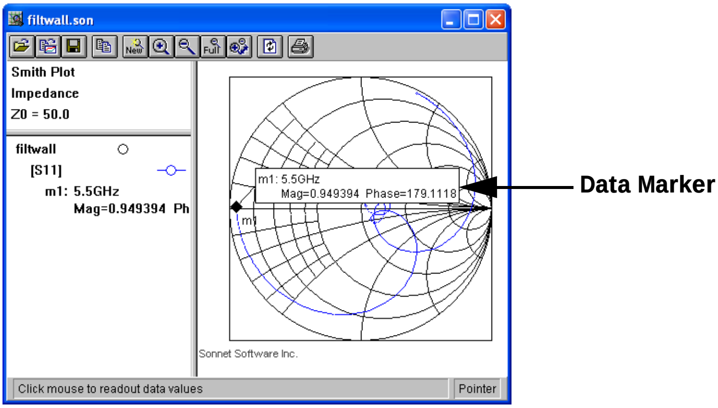
There are markers available in the response view that allow you to annotate your results in order to better interpret your data or make presenting your data simpler. There are six types of markers available; this section teaches you to add a data marker. For more information on markers, please refer to online help.

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

The appearance of the cursor changes.

45 Click on the data point to which you wish to add a marker.

A label appears on your plot as shown below. Double clicking on the marker allows you to control the data displayed in the marker.



Closing the Response Viewer

To close the program, perform the following:

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

The response viewer display is closed.

Current Density Viewer

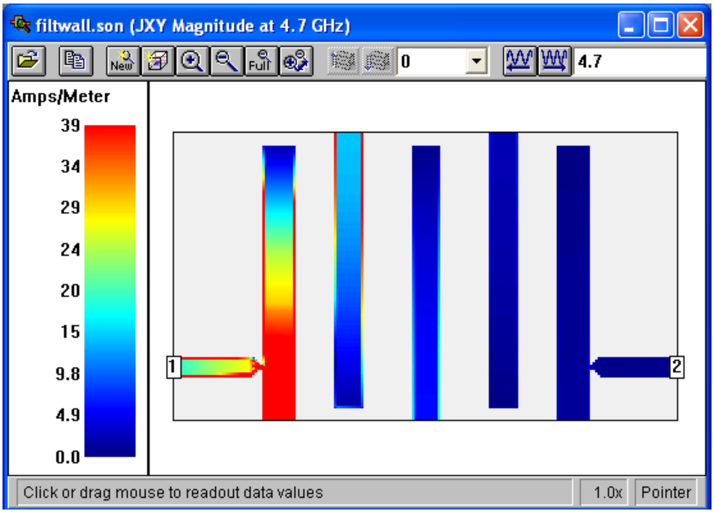
The current density viewer is a visualization tool used to view the results of circuits previously analyzed with ***em***. ***Em*** saves the resulting current density information in the project file ready for input to the current density viewer.

To produce the current density data, you must select the Compute Current Density option in the Analysis Setup dialog box in the project editor. This was done for you in the example file.

Invoking the Current Density Viewer

47 Select *Project* ⇒ *View Current* from the main menu of the project editor or the main menu of the response viewer after clicking on the curve group Filtwall in the legend.

The current density viewer window appears on your display with the first frequency in “filtwall.son” displayed. The color plot should appear similar to the one shown below. You are viewing the magnitude of the current density with a 1 Volt, 50 ohm source and Port 2 with a 50 ohm load.



Red areas represent high current density, and blue areas represent low current density. A scale is shown on the left of the window which defines the values of each color.

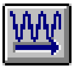
Current Density Values

48 Click on any point on your circuit to see a current density value.

The current density value (in amps/meter) at the point that you clicked is shown in the status bar at the bottom of your window along with the coordinates of the location.

Frequency Controls

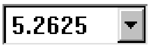
A different frame is displayed for each analysis frequency.



49 Click on the Next Frequency button on the tool bar.

The lowest frequency appears on your display as the default, which in this case is at 4.7 GHz. When you click on the Next Frequency button, the display is updated with the current density at 5.1125 GHz.

NOTE: The scale to the left of the display is also updated.



50 Click on the Frequency Drop list, and select 5.2625.

The drop list allows you to go directly to any of the analysis frequencies.

Animation

The current density viewer provides two types of animation: frequency and time. To do an animation in the frequency domain, the current density viewer takes a picture of your circuit at each frequency point and links the pictures together to form a “movie”. In the time domain, the current density viewer takes a picture of your circuit at instantaneous points in time at a given frequency by changing the excitation phase of the input port(s). Animation allows you to see how your response changes with frequency or time, providing insight into the properties of your circuit.

The animation menu and controls are the same whether animating as a function of frequency or time. The current density viewer accomplishes this by translating the data into frames. The animation menu then allows you to step one frame at a time, or “play” the frames by displaying them consecutively. How the data relates to a frame in either animation mode is discussed below.

Time Animation

For Time Animation, each frame corresponds to an input phase. The type of data plotted is determined by the *Parameters* ⇒ *Response* menu. In this case, the data is the JXY Magnitude, the total current density. This is the default setting upon opening a current density viewer window. For more details about response types, see “Parameters - Response” in the current density viewer’s help. To invoke help for the current density viewer, select *Help* ⇒ *Contents* from its main menu.

51 Select *Parameters* ⇒ *Scale* from the current density viewer main menu.

The Set Scale dialog box appears on your display.

52 Click on the User Scale checkbox and enter a value of 100 in the Max Value text entry box.

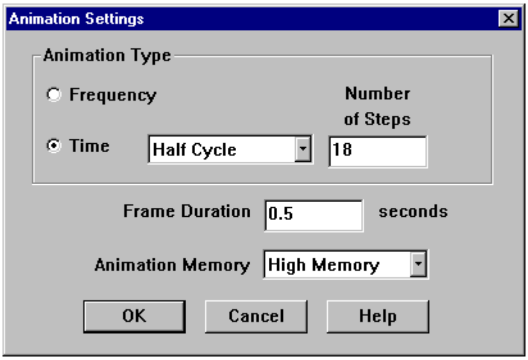
This allows you to set a fixed scale to avoid automatic updates of the scale during the animation.

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

The dialog box disappears and the scale to the left of the current density viewer window is now fixed at a maximum value of 100.

54 Select *Animation* ⇒ *Settings* from the current density viewer main menu.

The Animation Settings dialog box appears on your display as shown below.



55 Click on the Time Radio button to select Time Animation.

This enables the cycle type drop list and the Number of Steps text entry box.

56 Select Half Cycle from the cycle type drop list.

Our example uses half cycle, the default, in which the phase ranges from 0° to 180°.

For Full Cycle the phase ranges from 0° to 360°. For Quarter Cycle, the range is 0° to 90°.

57 Enter a value of 36 for the Number of Steps.

The number of steps is used to determine how many frames the animation includes. There are 180° in a half cycle; therefore, 36 steps yield a value of 5° for the phase interval.

The lowest phase, 0°, corresponds to the first frame and the highest phase, 175°, corresponds to the last frame. This phase value represents an offset from the Source Phase for any given port set in the Port Parameters dialog box. For details on how to set the Source Phase, see “Parameters – Ports” in the current density viewer’s online help. To access the current density viewer’s help, select *Help* ⇒ *Contents* from its main menu.

For this example, the Source Phase for port 1 is set to the default of 0°. Port 1 is terminated with a 50 ohm load.

58 Enter 0.1 in the Frame Duration text entry box.

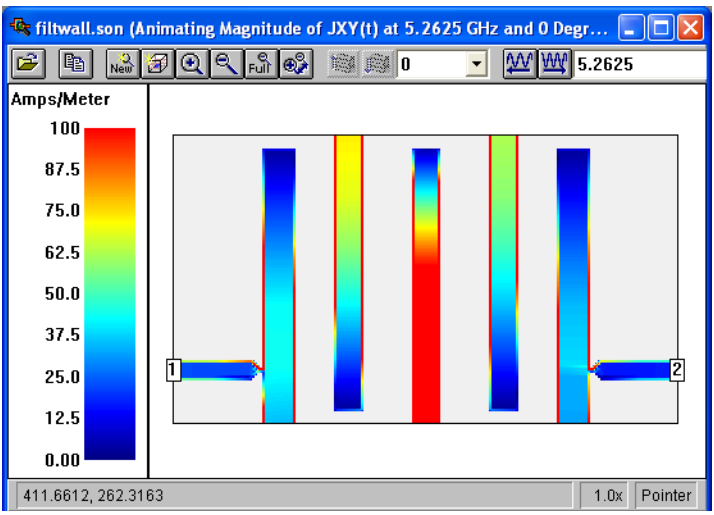
Before “playing” the frequency data by displaying consecutive frames, you may set the duration for each frame. This is the amount of time that each frame will appear on the display. The default value is 0.5 seconds. For this example, the frame duration is 0.1 seconds.

59 Click on the *OK* command button in the Settings dialog box.

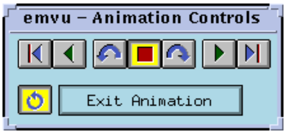
This saves the settings and closes the Animation Settings dialog box.

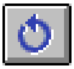
60 Select *Animation* ⇒ *Animate View* from the current density viewer main menu.

The current density viewer enters animation mode. The first frame, which displays the Magnitude of JXY(t) at 5.2625 GHz and 0 degrees, appears in the window and should resemble the picture below. The response type and frequency are determined by the settings when animation mode was launched. The present mode and frame is identified in the title bar of the window.



The Animation Controls, shown below, also appear on your display.

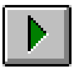




61 Click on the Continuous Play button in the Animation Controls.

This command toggles the *Play* commands in and out of continuous mode. In continuous mode, when the last frame is reached, the current density viewer starts over at the first frame and continues to cycle through the frames until a stop command is received. If the play direction is reverse, when the first frame is reached, the current density viewer starts over at the last frame and continues to display frames, in reverse order, until you press stop.

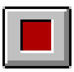
Continuous mode is indicated by a change of color of the Continuous Play mode icon.



62 Click on the Play button in the Animation Controls.

Selecting *Play* starts the animation at the present frame, 5.2625 GHz and 0 degrees, which corresponds to t = 0. Subsequent frames are displayed consecutively in 5° intervals. Each frame is displayed for 0.1 seconds, the delay set earlier. When the last frame is reached, the display starts over with the first frame and repeats until you press the Stop button in the Animation Controls.

Be aware that the first time through, the current density viewer is calculating the displays for each frame. Subsequently, the displays update at a slower rate. After the initial loading cycle of all the frames is complete, the display updates at the input rate of 0.1 seconds.



63 Click on the Stop button in the Animation Controls.

This stops the animation, although the window remains in animation mode.

64 Click on the Exit Animation button in the Animation Controls.

This exits the animation mode. The Animation Controls disappear from the display.

Frequency Animation

For Frequency Animation, each frequency will have its own frame. The lowest frequency, 4.7 GHz, corresponds to the first animation frame and the highest frequency, 5.5 GHz, corresponds to the last frame.

65 Select *Animation* ⇒ *Settings* from the current density viewer main menu.

The Animation Settings dialog box appears on your display.

66 Click on the Frequency radio button to select Frequency Animation.

If the radio buttons, Time and Frequency, in this dialog box are disabled, you stopped the animation but did not exit the animation mode. You must click on the Exit Animation button in the Animation controls to exit the animation mode and allow you to modify the animation settings. Only the Frame Duration may be changed while running an animation.

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

68 Select *Animation* ⇒ *Animate View* from the current density viewer main menu.

The current density viewer enters animation mode. The first frame, which displays the JXY Magnitude response for 4.7 GHz, appears in the window. The response type and frequency are determined by the settings when animation mode was launched.

Since the ABS band was defined from 4.7 GHz to 5.5 GHz, the first discrete frequency at which the circuit was analyzed is 4.7 GHz. Remember that current density data for an ABS sweep is only calculated for discrete data points not for all the adaptive data.

The Animation Controls also appear on your display. Note that Continuous Play mode is still "on" from the previous example.

69 Click on the Play button in the Animation Controls.

Selecting *Play* starts the animation at the present frame, 4.7 GHz. Subsequent frames, consecutively by frequency, are displayed, each for 0.1 seconds, the delay set previously. When the last frame is reached, the display starts over with the first frame and repeats until you press the Stop button in the Animation Controls.

Note that a description of each frame's contents appears in the title bar of the window.

70 Select *File* ⇒ *Exit* from the current density viewer main menu.

This command exits the current density viewer.

71 Select *File* ⇒ *Exit* from the project editor main menu.

This command exits the project editor program.

This completes the first Sonnet tutorial. The next tutorial concentrates on entering a circuit in the project editor.

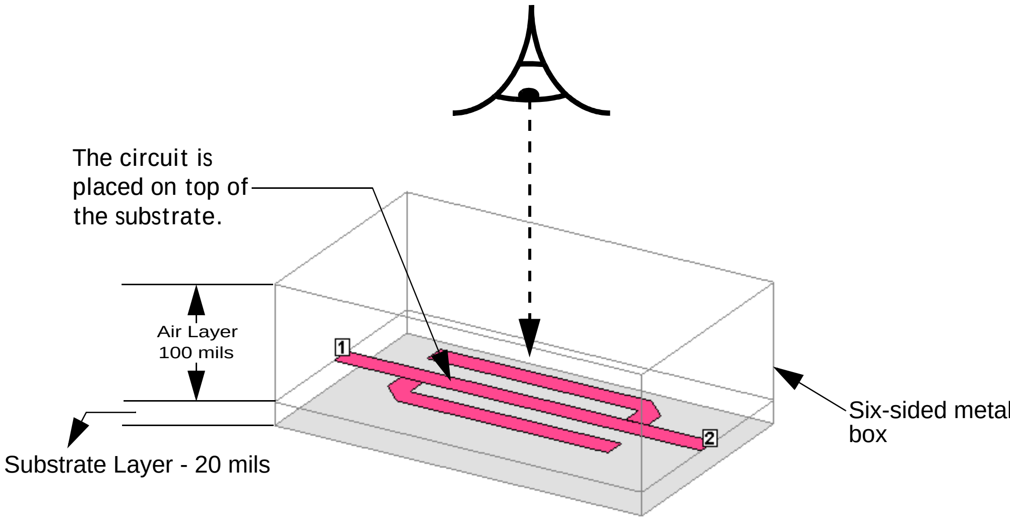
CHAPTER 5 CREATING A CIRCUIT: DSTUB

The second tutorial walks you through the process of entering a circuit in the project editor, analyzing the circuit using the analysis engine, ***em***, and observing your analysis data in the response viewer. Some of the concepts covered in this tutorial are:

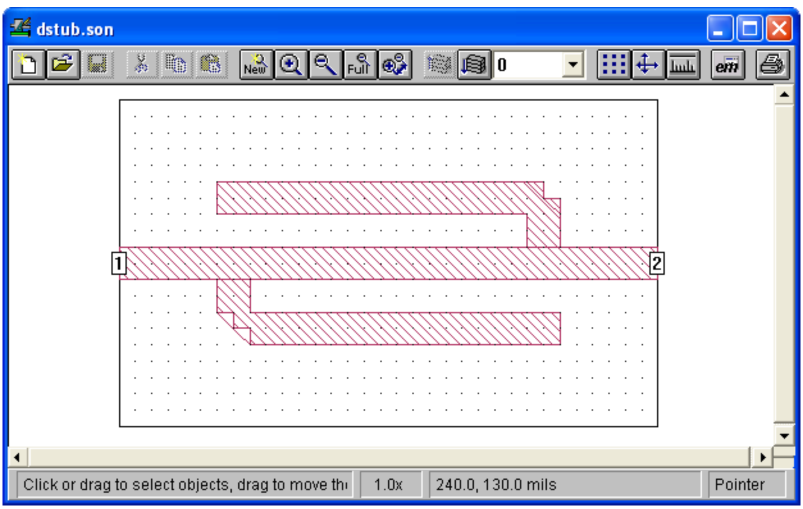
- Setting up box settings
- Determining cell sizes
- Setting up dielectric layers
- Adding metal rectangles to a circuit
- Adding metal polygons to a circuit
- Copying, flipping and moving polygons in a circuit
- Performing a simple analysis
- Plotting response data

The example for this tutorial is a circuit geometry project “dstub.son.” The circuit, as it appears upon completion, is shown following, along with a view showing the orientation when looking at the project editor display.

Even if you plan on using a translator to import your circuits, this tutorial provides you with the basics of the project editor environment which you need to accurately translate your circuit.



A three-dimensional view of the circuit in the six-sided metal box modeled in the project editor. The view shown in the project editor window is a two-dimensional view from the top looking directly down on the substrate.



The Substrate, Subsectioning, and Cell Size

To input a circuit geometry in the project editor, you must set up the box and substrate parameters as well as drawing the circuit. Below is a discussion of some of the factors to consider when setting up the box and cell size.

Specifying the parameters of the enclosing box defines the dimensions of the substrate, since the substrate covers the bottom area of the box. There are three interrelated box size parameters: Cell Size, Box Size, and Number of Cells. Their relationship is as follows:

(Cell Size) X (Num. Cells) = Box Size

Therefore, changing one parameter may automatically cause another parameter to change. Any one of the parameters can be kept constant when changing sizes by clicking on the corresponding Lock checkbox.

The electromagnetic analysis starts by automatically subdividing your circuit into small rectangular subsections. ***Em*** uses variable size subsections. Small subsections are used where needed and larger subsections are used where the analysis does not need small subsections.

A “cell” is the basic building block of all subsections, and each subsection is “built” from one or more cells. Thus a subsection may be as small as one cell or may be multiple cells long or wide. Thus the “Dimensions” of a cell determine the minimum subsection size.

The smaller the subsections are made, the more accurate the result is and the longer it takes to get the result. Therefore, there is a trade off between accuracy and processing time to be considered when choosing the cell size. The project editor gives you a measure of control over the subsection size. For a detailed discussion of subsectioning, please see Chapter 3, “Subsectioning” in the **Sonnet User’s Guide**.

Note that the dimensions of a cell do not need to be a “round” number. For example, if you want to analyze a line that is 9.5 mils wide, you need not set your cell dimension to 0.5 mils. You may want to set it to 4.75 mils (9.5 mils divided by 2) or 3.16667 mils (9.5 mils divided by 3). This will speed up the analysis because fewer subsections will be used. Also, the width of the cell does not need to be the same as the length.

When viewing your substrate, notice that the project editor displays a grid made up of regularly spaced dots. The distance between each grid point is the cell size.

The cell size may be entered directly or you may enter the number of cells. This sets the number of cells per substrate side, thus implicitly specifying the cell size in relationship to the box size. Note that the number of cells must be an integer number.

TIP

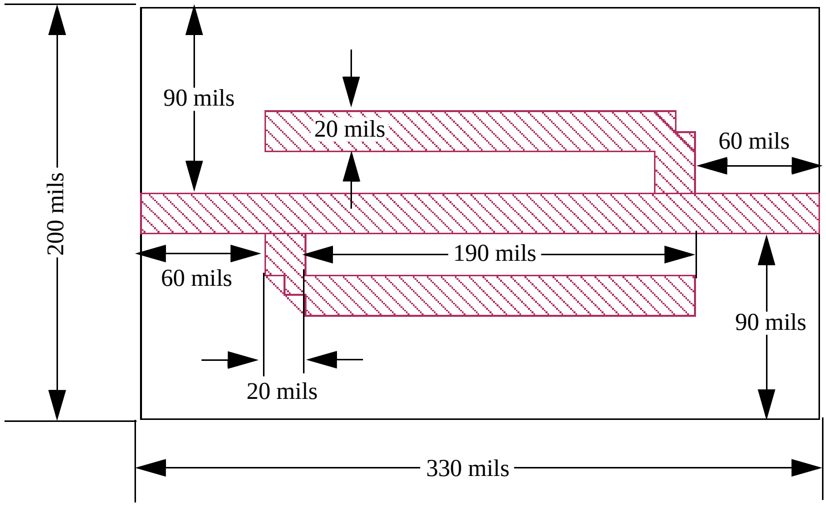
When the number of cells per side is large, selection of a power of two value provides ***em*** with a significant speed increase. For example, 256 cells per side is more efficient than 250 cells per side.

Metalization Levels and Dielectric Layers

The analysis engine and the project editor can handle any number of metalization levels. The default in a new project file is two dielectric layers with one metalization level in between. Typically metalization is referred to as “levels” and dielectric as “layers”. Each metalization level is sandwiched between two dielectric layers. It may be helpful to think of a level as being attached to the bottom of the dielectric layer above.You add another metalization level by adding dielectric layers through the Dielectric layers dialog box. When you are in the main window, you are viewing a metalization level of your circuit.

Entering your Circuit

In this section of the tutorial, you input the circuit “dstub.son.” The complete circuit, with dimensions, is shown below.



The dimensions of the example file, dstub.son.

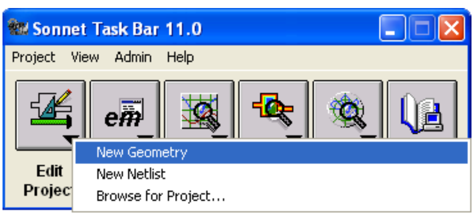
Invoking the Project Editor

1 Open the Sonnet task bar.

If you do not yet know how to do this, please refer to “Invoking Sonnet,” page 30.

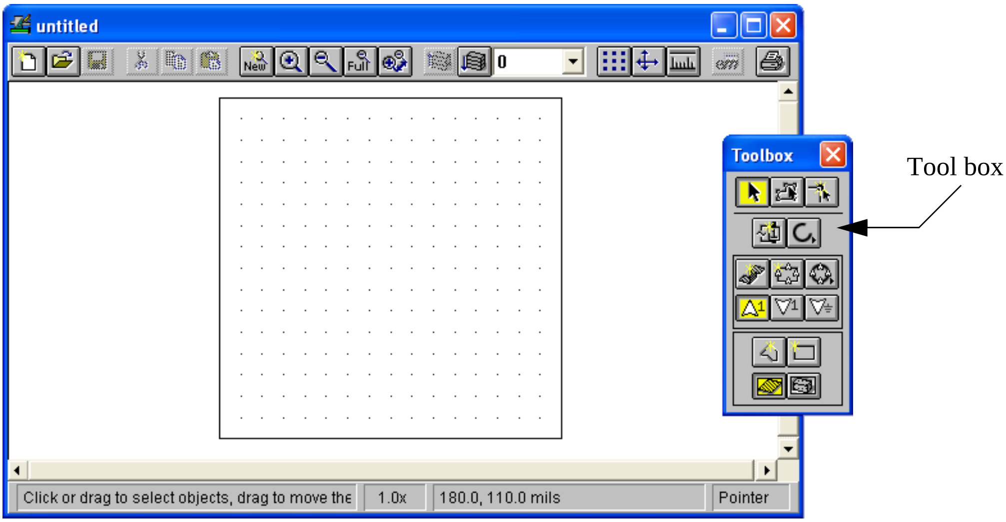
2 Click on the Edit Project button in the Sonnet task bar.

A pop-up menu appears on the task bar.



3 Select *New Geometry* from the pop-up menu.

A project editor window, with an empty substrate, appears on your display as shown below. The view shown in the project editor window is a two-dimensional view from the top looking directly down on the substrate. The tool box, which allows easy access to commonly used functions, also appears on your display.



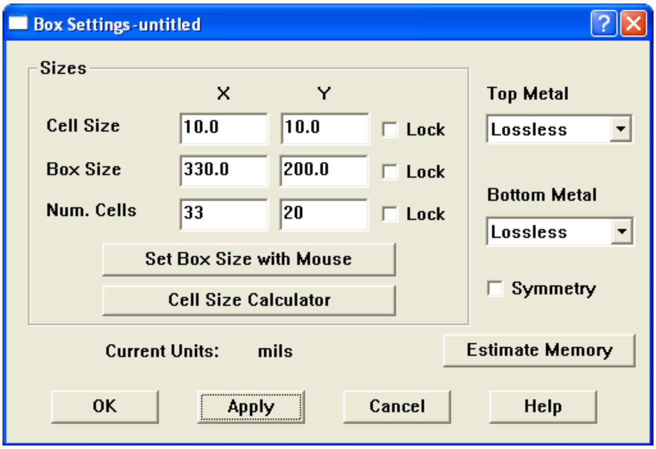
Specifying Box Settings

Before drawing the circuit, you must specify the parameters of the enclosing box, which includes the dimensions of the substrate and the cell size.

4 Select *Circuit* ⇒ *Box* from the project editor main menu.

The Box Settings dialog box appears on your display.

This dialog box is used to set the box size dimensions for your circuit. The box size for our example is 200 mils by 330 mils.



5 **Enter 330 in the X text entry box in the Box Size row of the Box Settings dialog box.**

This sets the x dimension of the box, and thereby, the substrate, to 330 mils.

6 **Enter 200 in the Y text entry box in the Box Size row of the Box Settings dialog box.**

This sets the Y dimension of the box, and thereby, the substrate, to 200 mils.

Note that when these values are entered, the number of cells value is updated to correspond to the new box size.

Setting Cell Size

In observing the dimensions of the circuit, shown on page 63, it can be seen that common factors of the dimensions, in both the x and y directions, are 2, 5, and 10 mils, in order for all the metalization to fall on the cell grid. Off-grid metalization is possible; however, that is addressed elsewhere.

A cell size of 10 mils by 10 mils provides enough accuracy for this circuit while needing the least processing time. These are the default values, so that steps 7 and 8 may not be necessary.

7 **Enter 10 in the X text entry box in the Cell Size row of the Box Settings dialog box.**

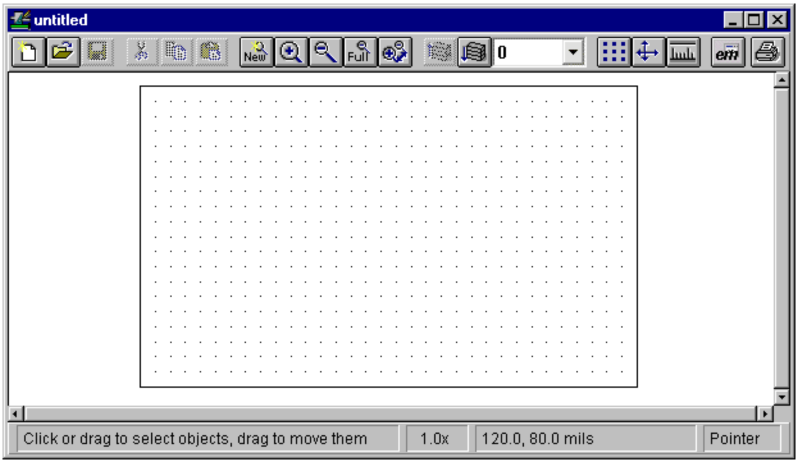
This sets the x dimension of the cell to 10 mils.

8 **Enter 10 in the Y text entry box in the Cell Size row of the Box Settings dialog box.**

This sets the y dimension of the cell to 10 mils.

9 **Click on the OK command button in the Box Settings dialog box.**

The dialog box disappears from your display and the substrate is updated to show the new size.



Setting the Dielectric Layers

Next, you need to specify the dielectric layers.

10 **Select *Circuit* ⇒ *Dielectric Layers* from the project editor main menu.**

The Dielectric Layers dialog box, which allows specification of the dielectric layers in the box, appears on your display. A new geometry project always has two default layers whose material is Air. The project editor “level” number appears on the left providing you with an approximate “side view” of your circuit. A “level” is defined as the intersection of any two dielectric layers and is where your circuit metal is placed.

11 **Enter a value of “100” in the Thickness text entry box in the first line of the Dielectric Layers dialog box.**

This dielectric layer is the air above the actual microstrip. The layer thickness has absolutely no impact on execution time or accuracy. Remember, the analysis is done inside of a six-sided metal box, so there is a metal top cover above the 100 mils of air. Specifying a small number for this thickness moves the top cover closer to the circuit metalization, providing stronger coupling between the top cover and the circuit metalization. If you do not know the actual dimensions of the box, or if you do not have a top cover, a good rule of thumb for microstrip is to set the air thickness to several (3-5) substrate thicknesses.

The rest of the parameters are the default values for Air and do not need to be changed.

12 Click on the second entry line in the Dielectric Layers dialog box to select it, then click on the Edit button.

The Dielectric Editor dialog box is opened. You did not need the Dielectric Editor for the first layer since you only wanted to change its thickness and this can be done directly in the Dielectric Layers dialog box. However, to change any other parameter of the dielectric layer you must use the Dielectric Editor dialog box.

TIP

Double-clicking on an entry in the Dielectric Layers dialog box, opens the Dielectric Editor dialog box on that entry.

13 Enter “Alumina” in the Mat. Name text entry box.

The material you wish to use for this dielectric layer is Alumina.

14 Enter a value of “20” in the Thickness text entry box.

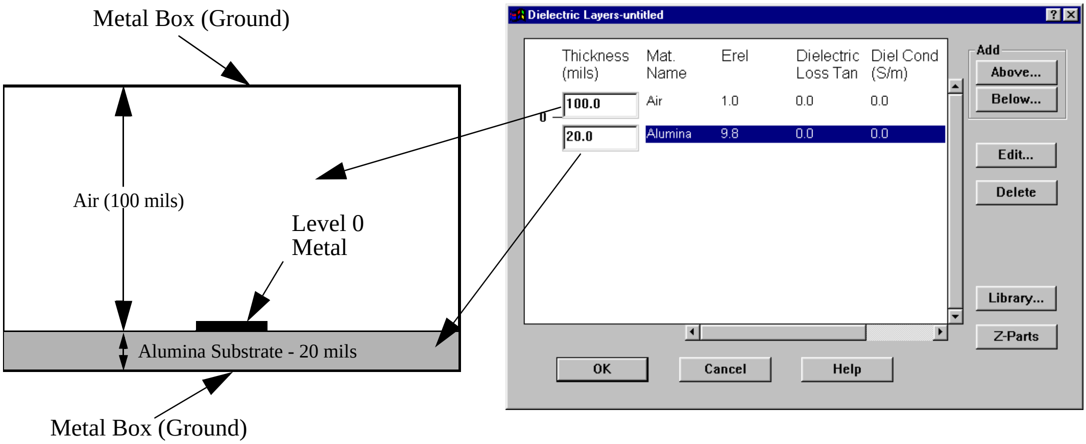
This specifies a 20 mil thick substrate.

15 Enter a value of “9.8” in the Erel text entry box.

The dielectric constant for alumina is 9.8.

16 Click on the OK button to close the Dielectric Editor and apply the changes.

The Dielectric Layers dialog box is updated and should appear similar to that shown below.



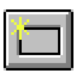
Note that the thickness of each layer must be specified. If not, the default value, 0.0, causes ***em*** to issue an error message and stop execution.

The setup of the box size, and dielectric layers is complete. The only metal type used for this circuit is lossless, the default metal type available in any new project editor file. In the next section, you input the metal polygons which make up the circuit.

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

Adding Metalization

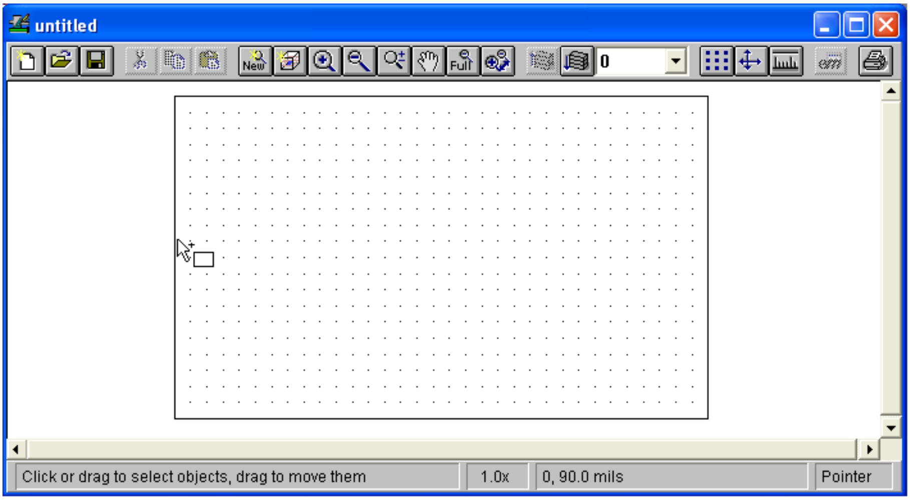
First, you add the center rectangle of the circuit.



18 Click on the Add a Rectangle button in the tool box.

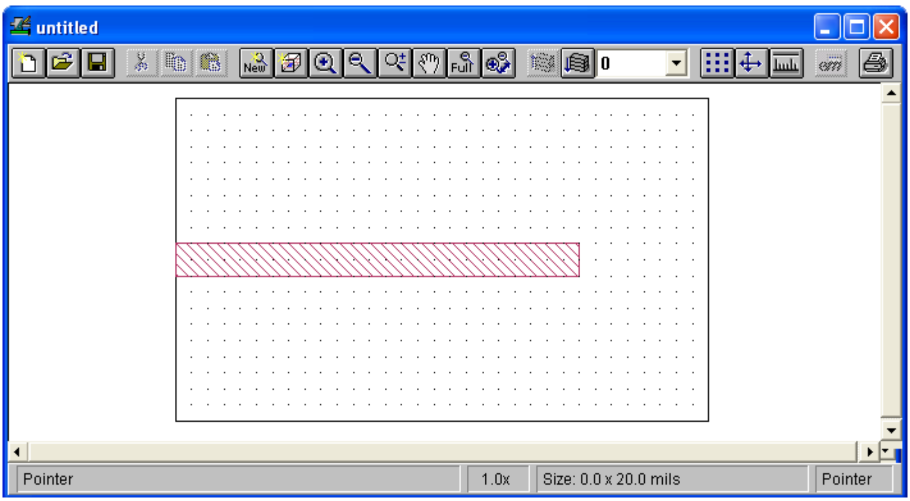
The appearance of the cursor changes.

19 Move the cursor until the readout in the pointer area of the status bar reads "0.0, 90.0 mils" then click the mouse.

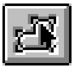


20 Move the cursor up and to the right until you go up two cell grids and towards the right edge of the box. Click on a point a few cells in from the box wall.

A rubber band stretches from the first point you clicked to the location of the cursor. When you click a second time, a metal rectangle appears in your circuit as shown below. We have deliberately made the polygon too short in order to demonstrate the Reshape tool. It should appear similar to the picture below.



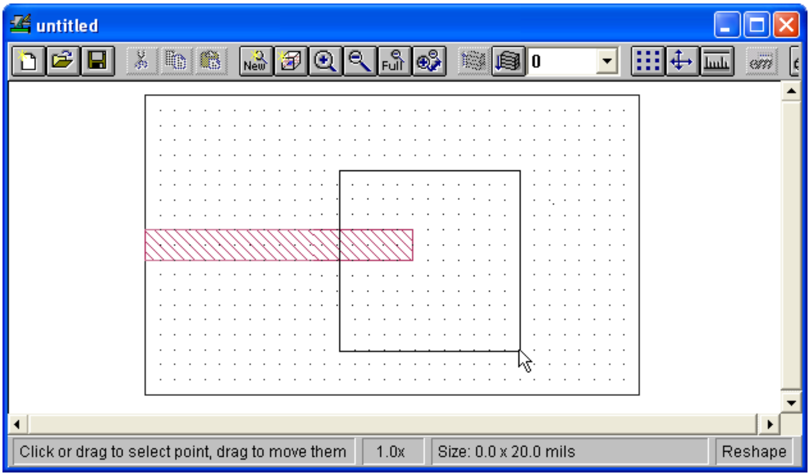
Next, we'll use the reshape tool to extend the polygon to the box wall.



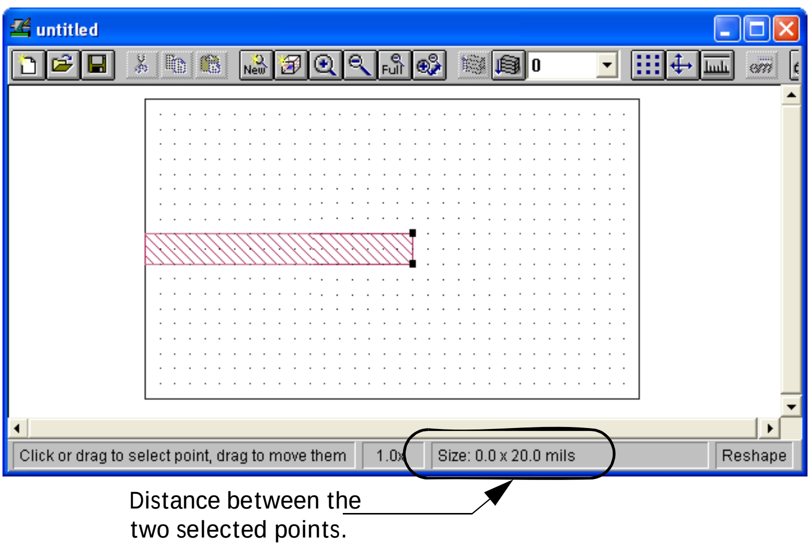
21 Click on the Reshape button in the Tool Box.

The appearance of the cursor changes to indicate that you are in reshape mode.

22 Drag the mouse so that the two end points of the polygon are selected.



The two selected end points are highlighted as shown below.

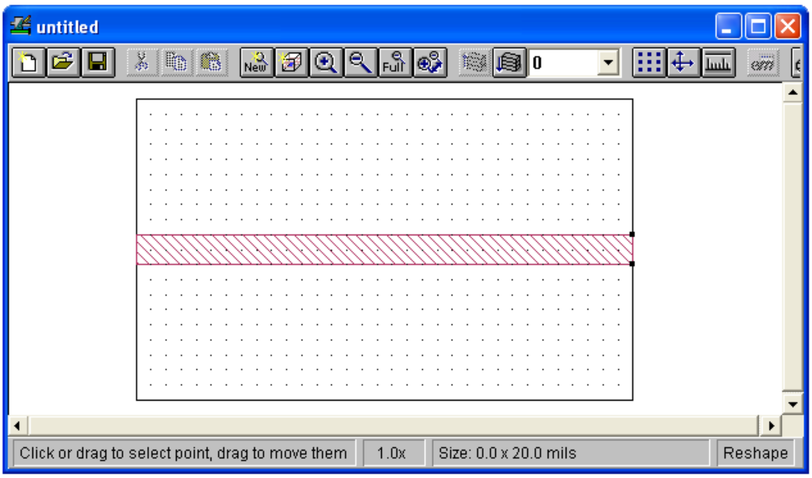


TIP

When two vertices of a polygon are selected, the distance between them is displayed in the status bar at the bottom of the window.

23 Click on one of the selected points and drag it to the box wall.

The circuit should appear as pictured below. You may select anywhere from one vertex to all the vertices when in reshape mode. When more than one vertex is selected, moving one vertex causes all other selected vertices to move and retain their relative position to the first vertex as happened in this example. It is also possible to add additional vertices to a polygon by using the Add Points button in the tool box.



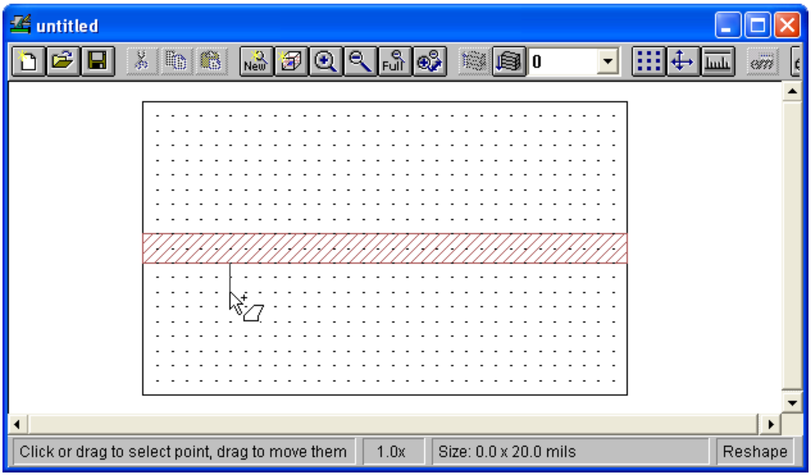
Next, you'll add the polygons.

24 Click on the Add a Polygon button in the Tool Box.

The appearance of the cursor changes.

25 **Type “60,90” followed by Enter on the keyboard.**

This enters the first point on the polygon. A rubber band stretches from this point to the location of the cursor.



26 **Type “@20<270” followed by Enter on the keyboard.**

The sign “@” indicates a relative distance from the last point. The “<270” provides the angle at which the segment should emanate. Therefore, a segment 20 mils in length going straight down is added to the polygon and a line connecting the two points appears.

27 **Move the cursor down 20 mils and to the right 20 mils until the status bar reads “20.0 X -20.0 mils” and then click.**

Another segment is added to the polygon. As before, the rubber band now stretches from the last point to be added.

TIP

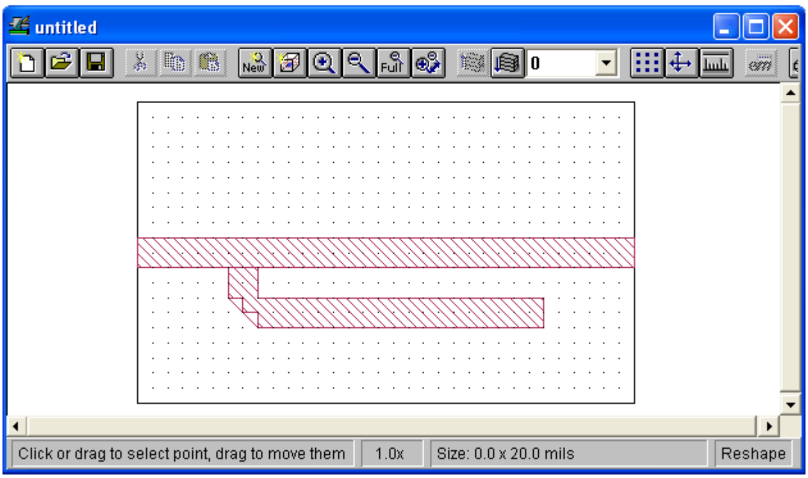
If you add a point in the wrong location, hitting the backspace, or delete key, deletes the last entered point of the polygon.

28 Enter “270,50” followed by Enter to enter the next point.

Continue adding points in this manner until the polygon is complete. The table below lists all the points in the polygon.

Point	Coordinates
1	60,90
2	60,70
3	80,50
4	270,50
5	270,70
6	80,70
7	80,90
8	60,90

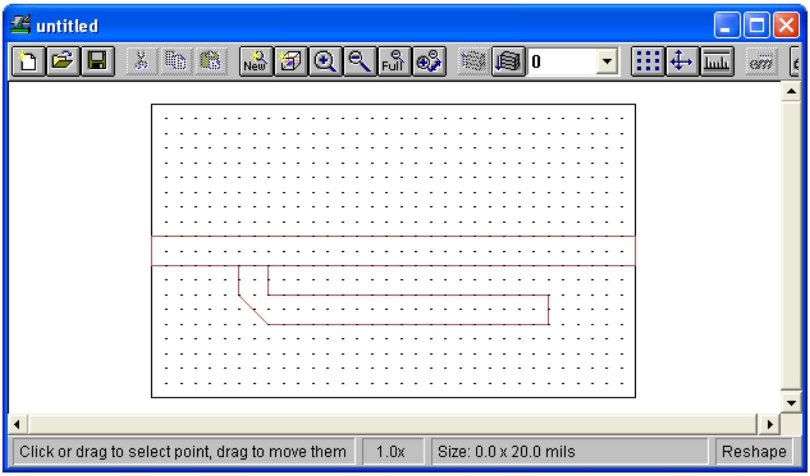
Note that you can complete the polygon by double-clicking on point 7 or by clicking on point 8 after adding point 7. Your circuit should now appear as shown below. Note that the metal polygons whose edges are touching are connected electrically.



When a polygon is complete, a fill pattern which defines the actual metalization analyzed by ***em*** appears on your display. In the picture above, notice the staircase approximation for the mitered bend. ***Em*** does not analyze the bend as you entered it; the staircase approximation is analyzed instead.

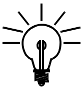
29 Enter ^M (Control-M) at the keyboard to turn off the metal fill.

The circuit, shown below, appears only with the outline of the polygons you have entered.



30 Enter ^M (Control-M) at the keyboard to turn the metal fill back on.

The display is updated with the metal fill on. The metalization, as mentioned above, represents the actual metal used in the simulation by ***em***.



TIP

Selecting Diagonal fill type for this polygon would have provided a much closer approximation of metalization to the entered polygon, at the cost of increased processing time. That degree of accuracy was not required for this circuit. For more details on Fill Types, see “Modify - Metal Properties” in the project editor’s help. To access the project editor’s help, select *Help* ⇒ *Contents* from its main menu.

Duplicating a Polygon

You now create the other stub by duplicating the one just entered.

31 Click on the completed polygon to select it.

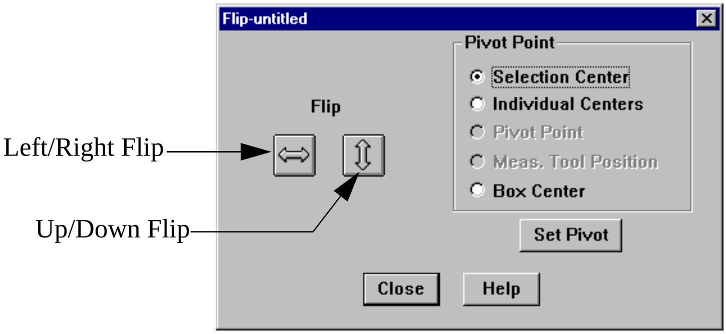
The polygon appears highlighted.

32 **Select *Edit* ⇒ *Duplicate* from the project editor main menu.**

A copy of the polygon appears on your circuit. Note that the duplicate polygon is highlighted, indicating its selection for edit commands.

33 **Select *Modify* ⇒ *Flip* from the project editor main menu.**

The Flip dialog box appears on your display.



34 **Click on the Left/Right Flip button in the Flip dialog box.**

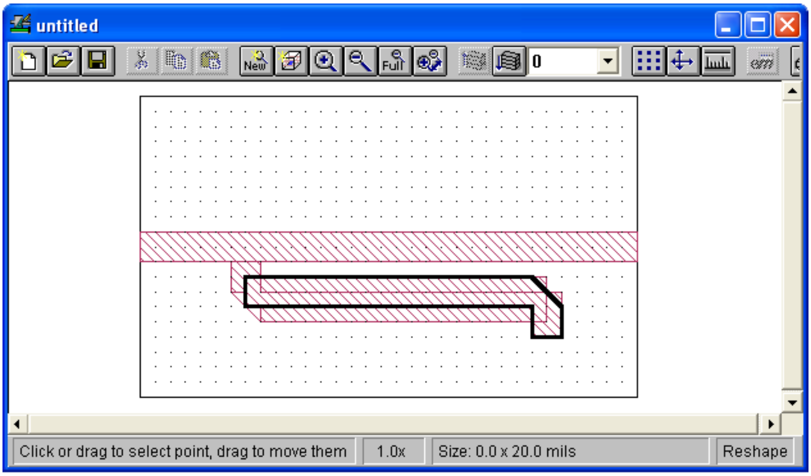
The polygon flips about its center in left/right direction.

35 **Click on the Up/Down Flip button in the Flip dialog box.**

The polygon flips about is center in the up/down direction.

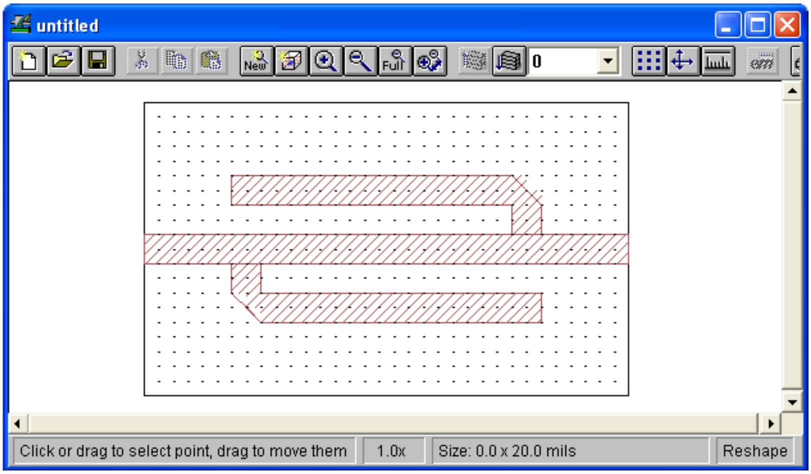
36 **Click on the Close button in the Flip dialog box.**

The Flip dialog box disappears from your display. Your circuit should now appear as shown.



37 Move the polygon, by dragging it with the mouse, until it is opposite the other polygon on the other side of the strip.

The circuit should now look like this:



Adding Metal Loss

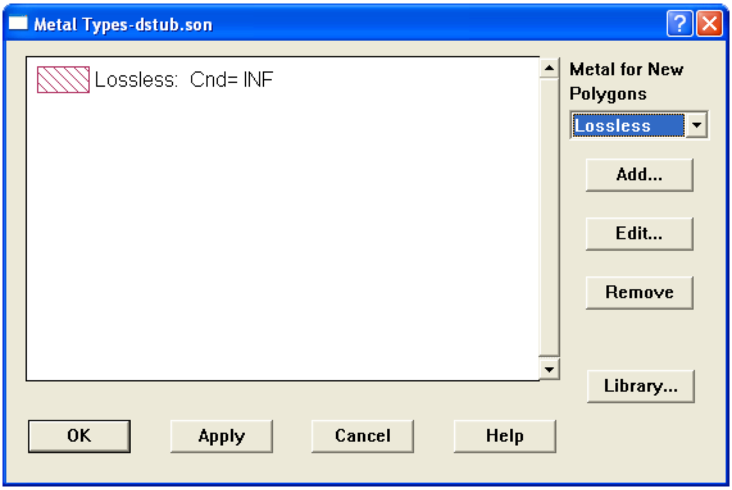
In the next section of the tutorial, you will define metal loss for all the metalization in your circuit. In order to model metal loss, you must first define a metal type by inputting its loss parameters. Once you have created the metal type, it needs to be applied to the desired polygons in order for that metalization to use the correct loss.

This change is done here for the purpose of demonstration; the polygons will be changed back to lossless metal before analyzing the circuit.

Defining a Metal Type

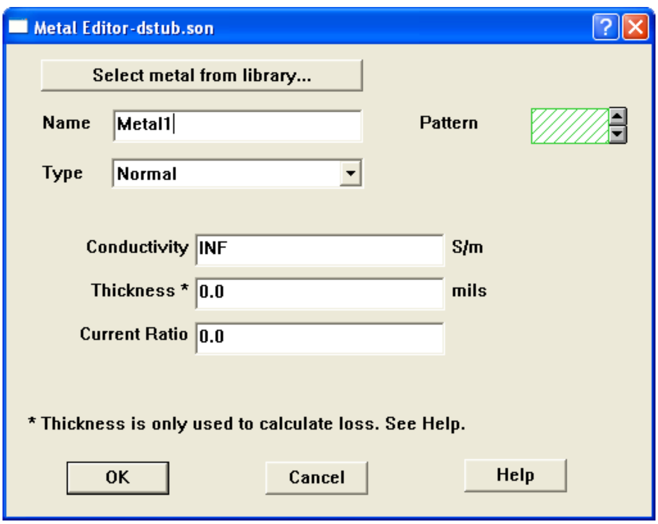
38 Select Circuit ⇒ Metal Types from the project editor main menu.

The Metal Types dialog box appears on your display. The only metal presently defined for this circuit is the default Lossless metal whose entry is displayed in the Metal Types dialog box as shown below.



39 Click on the Add button in the Metal Types dialog box.

The Metal Editor dialog box appears on your display. Note that a default name, "Metal1," is provided for the metal type and the default parameters define a lossless metal.



40 Enter the name "Half Oz Copper" in the Name text entry box.

This will be the name of the metal type and will appear in any metal type menus. You may enter any string of characters for the metal Name.

You will use the default type of "Normal" which is used for most circuit metal. The metal loss is based on the bulk conductivity, thickness of the conductor and the current ratio.

TIP

The metal type you are defining will only be available for use in this circuit unless you add the metal type to your Global or Local metal library. Refer to "Metal Libraries" on page 58 of the **Sonnet User's Guide** for details on using metal libraries.

41 Enter "5.8e7" in the Conductivity text entry box.

This is the bulk conductivity of Copper in S/M.

42 Enter "0.7" in the Thickness text entry box.

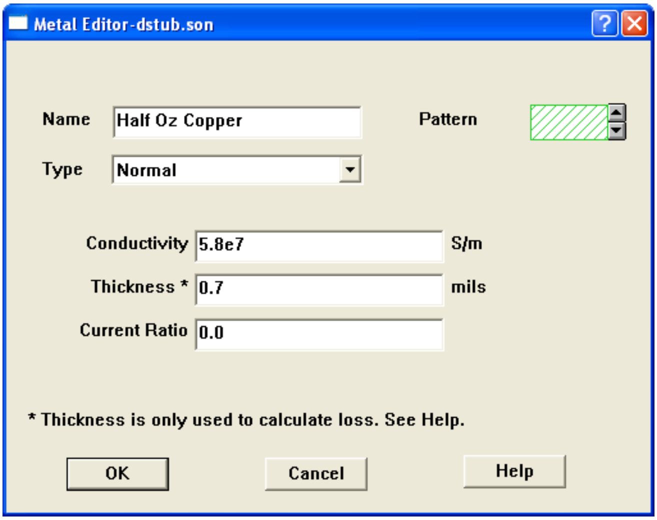
This will use a thickness of 0.7 mils to calculate loss. Note that the metal thickness is used only in calculating loss; it does not change the physical thickness of metalization in your circuit. The metalization in your circuit is still modeled as zero-thickness. To model physically thick metal, you would use the Thick Metal type.

The default value of zero for the current ratio does not need to be changed for this metal. For most microstrip, zero is usually a good value. Time does not allow for an explanation of current ratio here, but for a detailed explanation you may click on the Help button in the dialog box.

TIP

Online help is available by clicking the Help button in any dialog box or by selecting Help ⇒ Contents from the menu of any Sonnet application. Clicking on the Help button in a dialog box takes you directly to a topic which describes the functions and purpose of the controls in that dialog box.

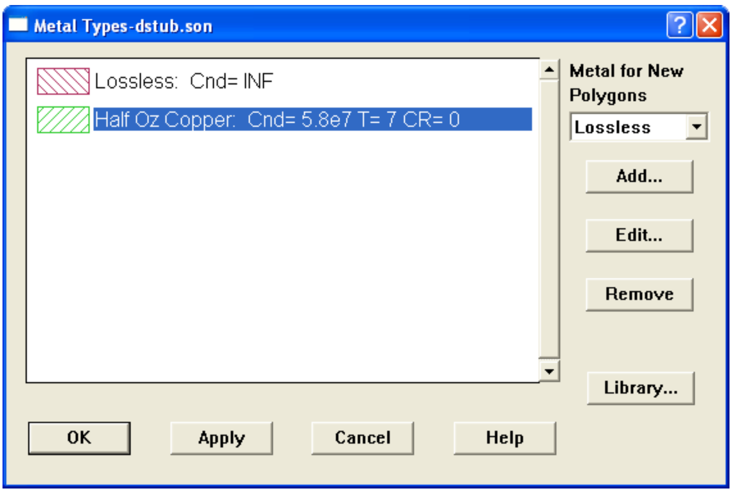
The dialog box should now appear similar to the illustration below.



Note the drop list for the pattern which is used to identify the metal where it is used in your circuit. We are using the default. If you wish to change the Fill Pattern click on the arrows to cycle through your choices.

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

The Metal Types dialog box now displays two metal types in the list: lossless and Half Oz Copper.



44 **Click on the OK button in the Metal Types dialog box to apply the changes and close the dialog box.**

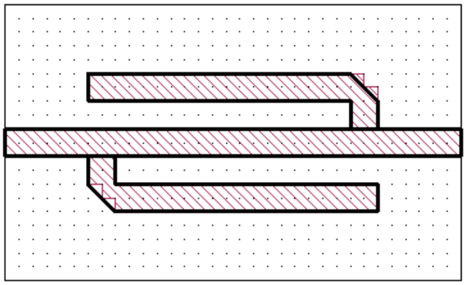
Now that the Half Oz Copper metal type is defined for this circuit, you need to apply the metal type to the desired polygons to use the loss model in your circuit.

Any combination of these two metals, lossless and Half Oz Copper, may be used anywhere in your circuit. You may define as many metal types as you wish. Any combination of these metal types may be used on any metal level and on any type of polygon. You may use multiple metal types on the same metal level.

Applying a Metal Type

45 Drag your mouse or press Control-A to select all the polygons in the circuit.

All the polygons are highlighted to indicate that they are selected.



46 Select Modify ⇒ Metal Properties from the project editor main menu.

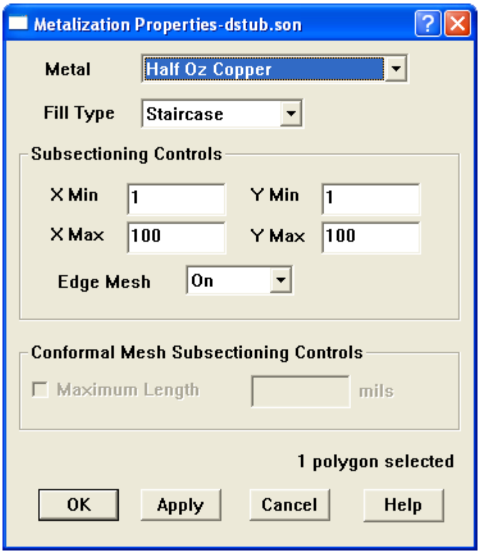
The Metalization Properties dialog box appears on your display. You may use this dialog box to change the properties of multiple polygons. If two or more of the selected polygons have different attributes, the field will display “Mixed” to indicate different values. Any changes you make in the dialog box will be applied to all selected polygons when it is closed. If you do not change a “Mixed” field than no change is made to the polygons.

TIP

You may also access the Metalization Properties dialog box by double-clicking on any polygon or right-clicking on a polygon and selecting “Metal Properties” from the pop-up menu which appears.

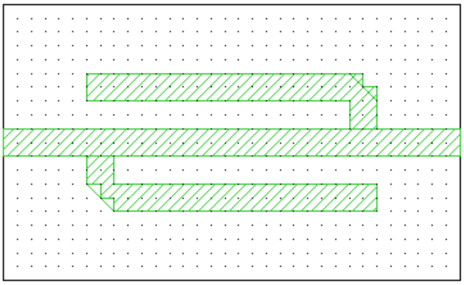
47 Select “Half Oz Copper” from the Metal drop list in the Metalization Properties dialog box.

This applies the Half Oz Copper metal type to all the selected polygons. The loss for this metalization will be calculated using the definition you entered for Half Oz Copper.



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

Note that the appearance of the circuit has changed, with the fill pattern for Half Oz Copper now appearing on all the polygons.



49 In the same manner, change the metal type for all the polygons back to “Lossless.”

This applies the lossless metal type once again to your whole circuit.

Adding Ports

To complete the circuit, you need to add the two ports. The simplest type of port, used in this example, is the boxwall port. A boxwall port is connected between a polygon and the box wall.

NOTE: **The polygon must be touching the box wall.**

50 Click on the Add Port button in the tool box while holding down the shift key.

The shape of the cursor changes to indicate Add Port mode. Holding down the shift key allows you stay in Add Port mode after adding the first port. This allows you to add multiple ports without having to select the Add Port button each time.

TIP

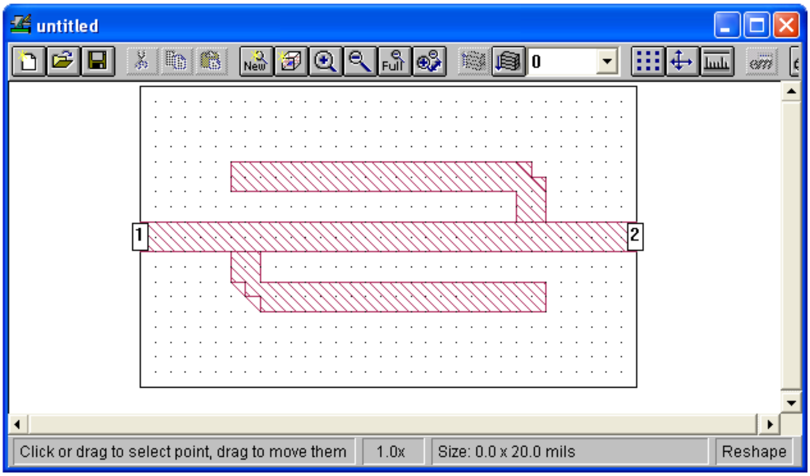
Holding down the shift key when selecting any Add command allows you to add multiple instances of that geometry element. When you are done, press the Escape key to return to the pointer mode. Also, double clicking on an icon in the tool box acts the same as pushing the shift key.

51 Click on the left edge of the circuit on the left box wall.

Port 1 is added to the circuit. Ports are numbered automatically in the order that they are added. It is possible to change port numbers and types after they have been added to the circuit. For more details, see Chapter 5, “Ports” in the **Sonnet User’s Guide**.

52 Click on the right edge of the circuit on the right box wall.

Port 2 is added to the circuit which should appear as shown below.



53 Push the ESC key.

This exits Add Port mode and returns the project editor to pointer mode. Note that the ports are actually attached to the polygon; therefore, moving or copying the polygon also moves or copies the ports.

Saving the Circuit

You have completed defining the circuit in the project editor, but before you analyze the circuit using ***em***, you need to save the circuit.

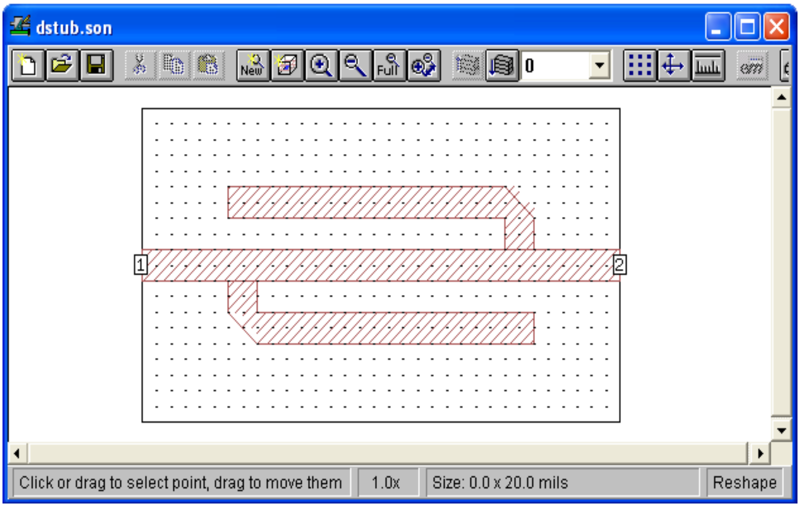
54 Select *File* ⇒ *Save* from the project editor main menu.

The Save File dialog box appears on your display.

55 If necessary, browse to the directory in which you wish to save the file.

56 Enter "dstub.son" in the File Name text entry box and click on the Save command button.

The file is saved as a circuit geometry project with the name "dstub.son." Notice that this name now appears in the title bar. This completes the circuit editing section of this tutorial.



Em - The Analysis Engine

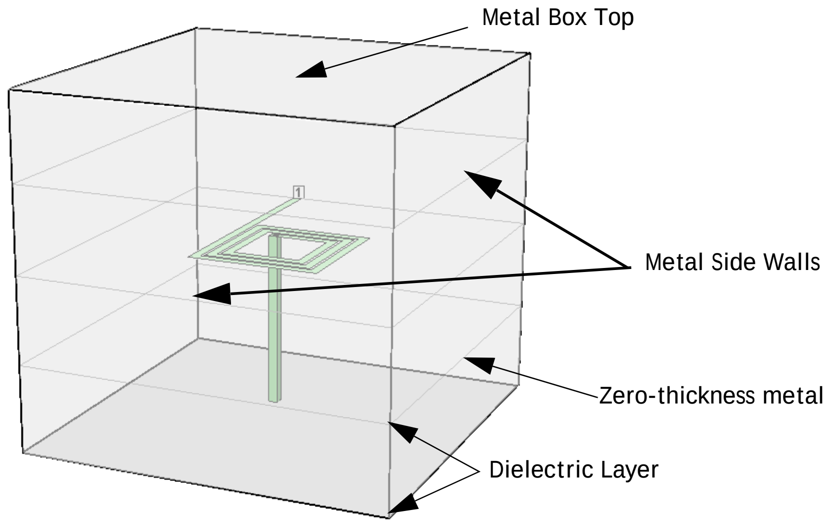
In the next section of the tutorial, the circuit you entered is analyzed using the electromagnetic simulator engine, ***em***. The following discussion provides background on ***em*** and explains the theory behind the analysis engine. Instructions continue at step 57 on page 87.

The analysis engine**, *em*,** performs electromagnetic analyses for arbitrary 3-D planar geometries, maintaining full accuracy at all frequencies. ***Em*** is a "full-wave" analysis engine which takes into account all possible coupling mechanisms.

Analyses are set up in the project editor. The analysis monitor provides an interface that allows you to monitor analyses and manage batch files for running analyses. Your analysis control settings are saved as part of your project file.

A Simple Outline of the Theory

Em performs an electromagnetic analysis of a microstrip, stripline, coplanar waveguide, or any other 3-D planar circuit, such as the one shown below, by solving for the current distribution on the circuit metalization using the Method of Moments.

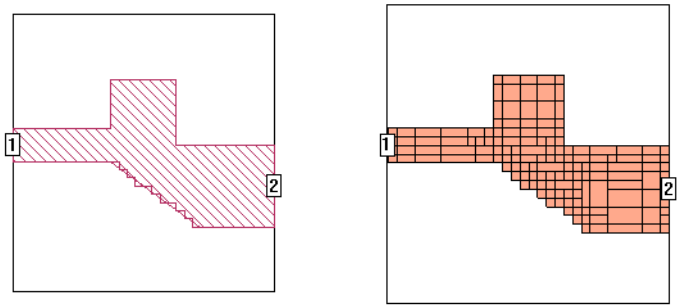


Em analyzes planar structures inside a shielding box.
Port connections are usually made at the box sidewalls.

Subsectioning the Circuit

The analysis starts by subdividing the circuit metalization into small rectangular subsections. In an actual subsectioning, shown below, small subsections are used only where needed. Otherwise, larger subsections are used since the analysis time is directly related to the number of subsections. Triangular subsections can be used to fill in the diagonal “staircase” at the user’s discretion. These subsections are based on the cells specified in the geometry file.

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.

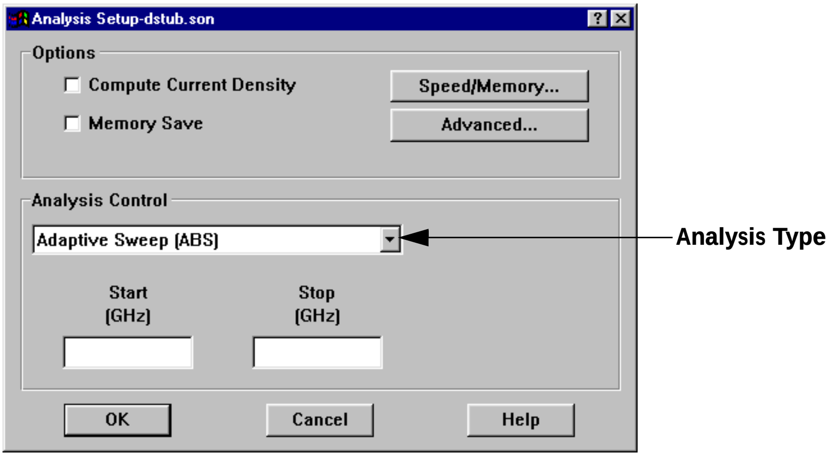
Frequency Sweep

There are three types of analyses that may be done in ***em***: frequency sweep, parameter sweep and optimization. This section of the tutorial describes how to set up a linear frequency sweep, and invoke ***em***.

Linear Frequency Sweep

The analysis control section of the Analysis Setup dialog box in the project editor allows you to select the type of analysis and the frequencies used in analyzing your circuit.

A linear frequency sweep is used to execute an analysis using one or multiple frequencies evenly spaced in an ascending order.



Selecting Analysis Options

There are two options available in the Analysis Setup dialog box, which may be turned on when you wish to use the option and set to off if the option is not desired. In addition there are other run options in the Advanced Options dialog box which are invoked by selecting the Advanced Options command button in the Analysis Setup dialog box. The De-embed option, found in the Advanced Options dialog box, is set to on by default for a new analysis. This example uses the De-embed option.

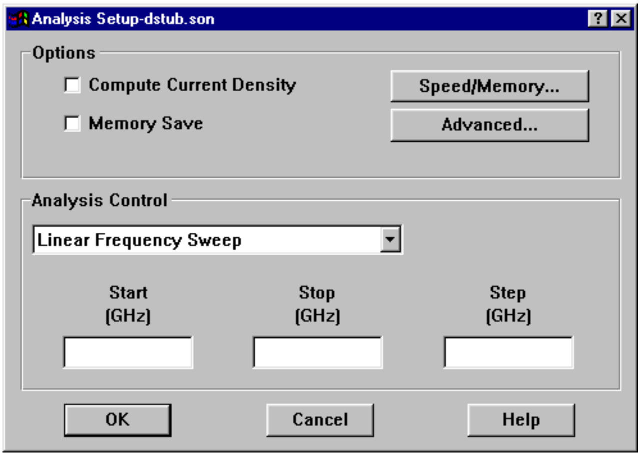
If the project editor is not still open from the previous part of the tutorial, invoke the project editor and open the file "dstub.son."

57 **Select *Analysis* ⇒ *Setup* from the project editor main menu.**

The Analysis Setup dialog box appears on your display. The default type of analysis is the Adaptive Sweep (ABS) already displayed in the Analysis Control drop list. The other types of analysis are Linear Frequency Sweep, Frequency Sweep Combinations, Parameter Sweep, Optimization and External Frequency File.

58 Select "Linear Frequency Sweep" from the Analysis Control drop list.

The appearance of the dialog box is updated for the linear frequency sweep to include the Start, Stop and Step text entry boxes as shown below.



Input the Linear Sweep

Frequency Control for this analysis is provided by direct input into text entry boxes. The circuit is analyzed from 4.0 GHz to 8.0 GHz in 0.25 GHz intervals.

59 Enter "4.0" in the Start text entry box and enter "8.0" in the Stop text entry box.

The analysis starts at 4.0 GHz and ends at 8.0 GHz.

60 Enter "0.25" in the Step text entry box.

The constant intervals between analysis frequencies is 0.25 GHz.

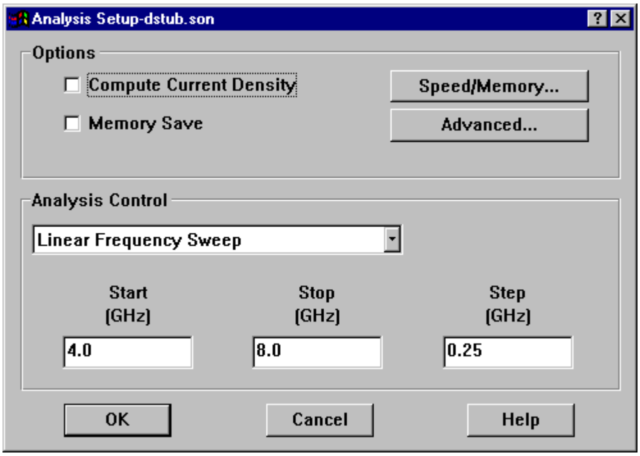
Run Options

This analysis uses the De-embed run option. The De-embed option is on by default.

De-embed Option: De-embedding is the process by which the port discontinuity and transmission line effects are removed from the analysis results. Inaccurate data may result from failing to implement this option. When the De-embed option is set, the circuit is automatically de-embedded to the specified reference planes, or the box edge if no reference planes are specified in the circuit geometry. For a detailed discussion of de-embedding refer to Chapters 7 and 8 in the **Sonnet User's Guide**.

In our case, no reference planes are specified, so that the circuit is de-embedded to the box walls.

This completes setting up the analysis run. The Analysis Setup dialog box should appear similar to that shown here.



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

This completes the setup for your analysis.

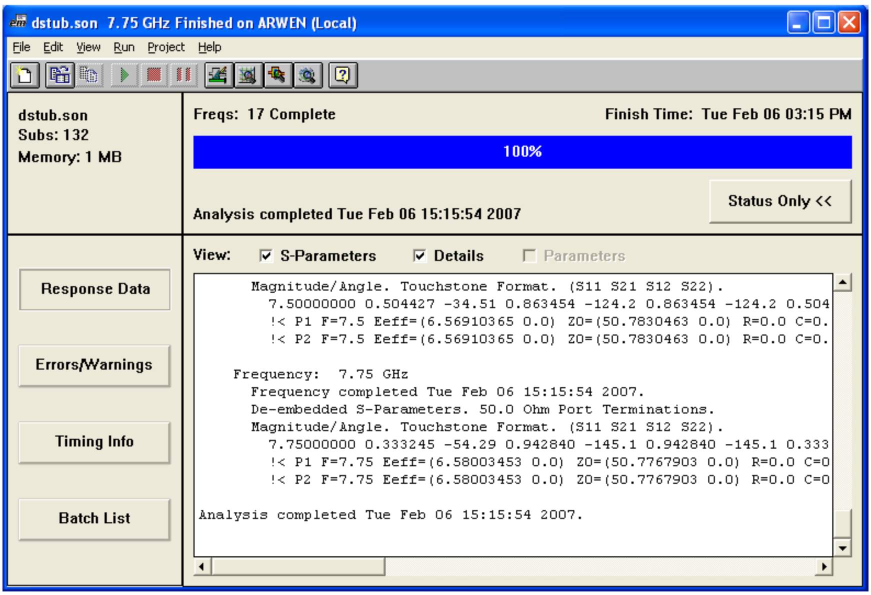
62 Select File ⇒ Save from the project editor main menu.

The file must be saved before analyzing the circuit. If you do not execute a save before analyzing, you will be prompted to do so.

Executing the Analysis

63 Select *Project* ⇒ *Analyze* to execute the analysis.

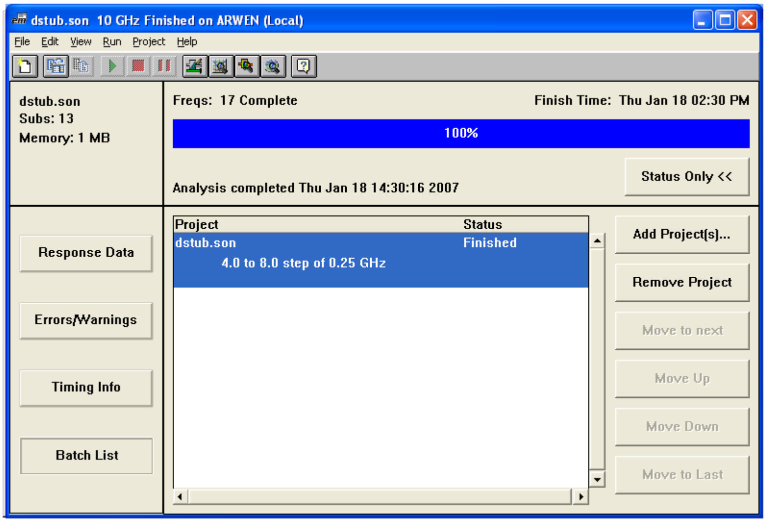
The analysis monitor appears on your display with the Response Data window displayed. As the analysis progresses, the response data is output to the window. The progress bar indicates the ongoing status of the analysis. When the job is complete, the message “Analysis successfully completed” appears just below the progress bar and the progress bar is completely filled in.



The output data from the analysis is shown in the Response Data window, as pictured above. You can use the scroll bars to the right and bottom of the window to see all your response data.

64 Click on the Batch List command button in the analysis monitor window.

The batch list appears. The batch list shows the history of your runs and allows you to set up a list of analyses to run sequentially. The data displayed in the analysis monitor applies to the presently selected batch list entry.



For more information about batch lists, see help for the analysis monitor. To access the analysis monitor's help, select *Help* ⇒ *Contents* from its main menu.

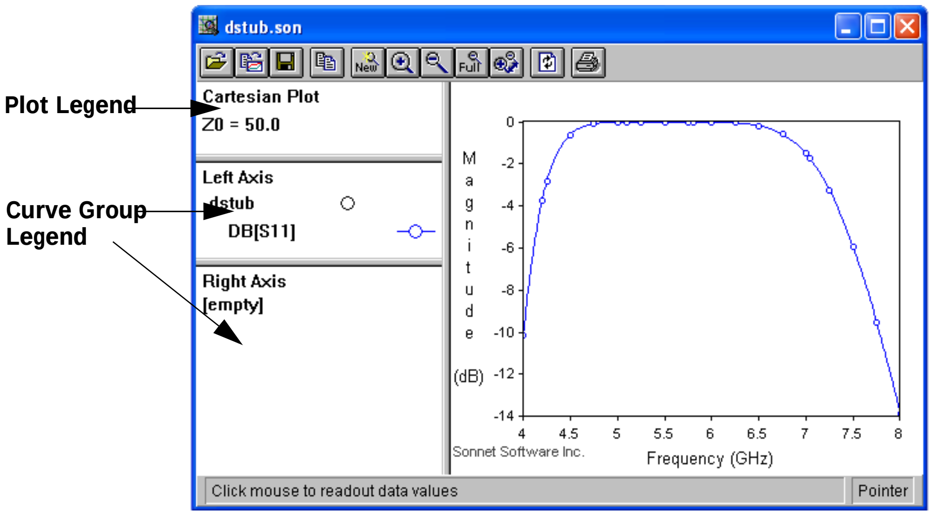
Observing the Data

For this example, you shall display a Cartesian graph of your response data.

Invoking the Response Viewer

65 Select *Project* ⇒ *View Response* ⇒ *New Graph* from the analysis monitor main menu.

The response viewer window appears on your display with a Cartesian graph displaying the curve group dstub which consists of the S11 curve from the response data.



Editing Curve Groups

The project file contains multiple curves which are displayed in user-defined curve groups. A measurement is a set of response data uniquely identified by parameter type, response type, port numbers and project file. An example would be:

```
DB[S21] in the dstub.son file
```

DB identifies the response type as magnitude in dB. S identifies the parameter type as an S-Parameter. 21 identifies the input port as Port 1 and the output port as Port 2. The project file from which the measurement originated is “dstub.son”. A curve group is made up of multiple measurements.

A data point is the measurement for a particular frequency. The data for all frequencies makes up a measurement. Curve groups consist of a set of measurements defined by the user. The measurements in a curve group must all come from the same project file and be the same response and parameter type. Different curve groups may come from different project files yet be displayed simultaneously.

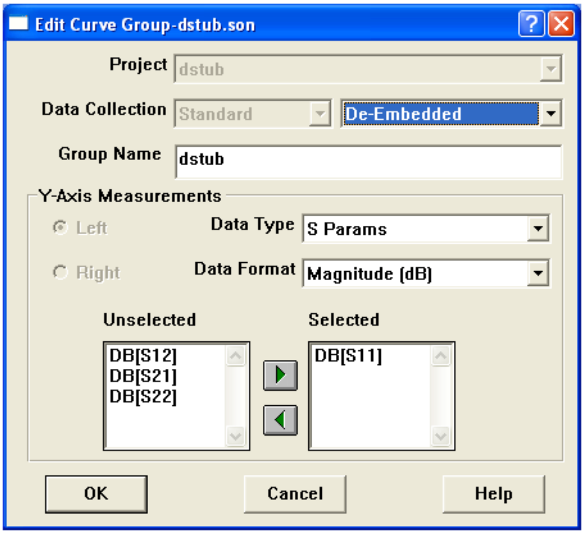
Since this analysis only included a frequency sweep, no parameter combinations are available.

66 Click on Dstub in the Curve Group Legend.

A box appears around Dstub in the Curve Group legend and the curve group is highlighted on the plot.

67 Select *Curve* ⇒ *Edit Curve Group* from the response viewer main menu.

The Edit Curve Group dialog box appears on your display.



TIP

You may also access this dialog box by right-clicking in the Curve Group Plot and selecting “Edit Curve Group” from the pop-up menu which appears.

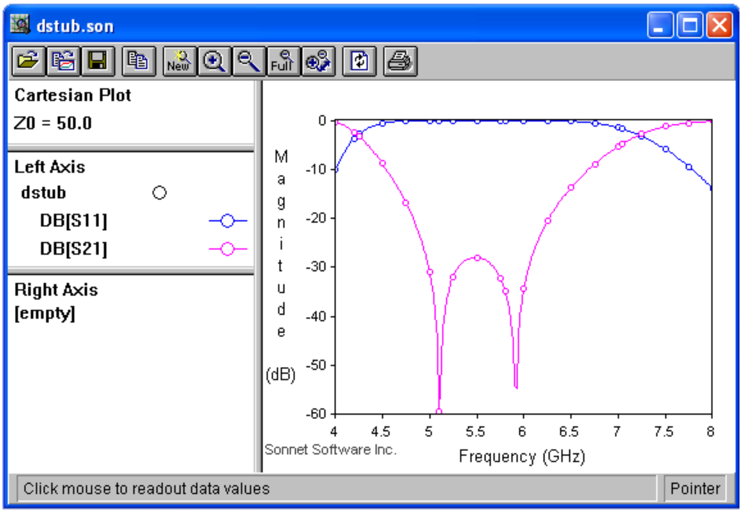
The project and data collection drop lists are disabled since you are editing an existing curve group.

68 Double click on DB[S21] to move this curve from the unselected list to the selected list.

This adds the DB[S21] measurement to the dstub curve group.

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

The dialog box disappears from your display and the plot is updated to include both data curves.



Another way to add the DB[S21] curve would have been to create an additional curve group. That is shown below.

Removing a Measurement from a Curve Group

70 Click on DB[S21] in the Curve Group legend to select it.

A box appears around the entry in the legend and the plot is highlighted.

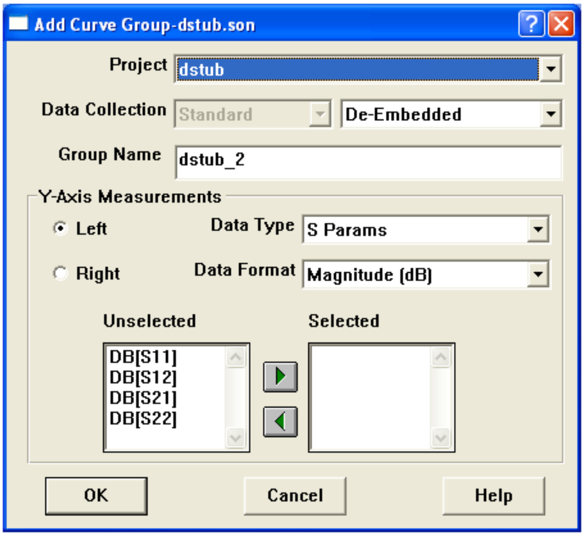
71 Right click on DB[S21] and select Delete Measurement from the pop-up menu which appears.

The curve legend and plot window are updated. Your graph should now appear as it did when the response viewer was invoked.

Adding a New Curve Group

72 Select *Curve* ⇒ *Add Curve Group* from the response viewer main menu.

The Add Curve Group dialog box appears on your display. This dialog box is the same as the Edit Curve Group dialog box, but here the project drop list, group name and axis radio buttons are enabled.



The default name of dstub_2 appears in the Group Name text entry box. The response viewer uses the basename of the project file with an incremental number attached as the default for curve group names. If you wish to change the curve group name, edit the text in the entry box.

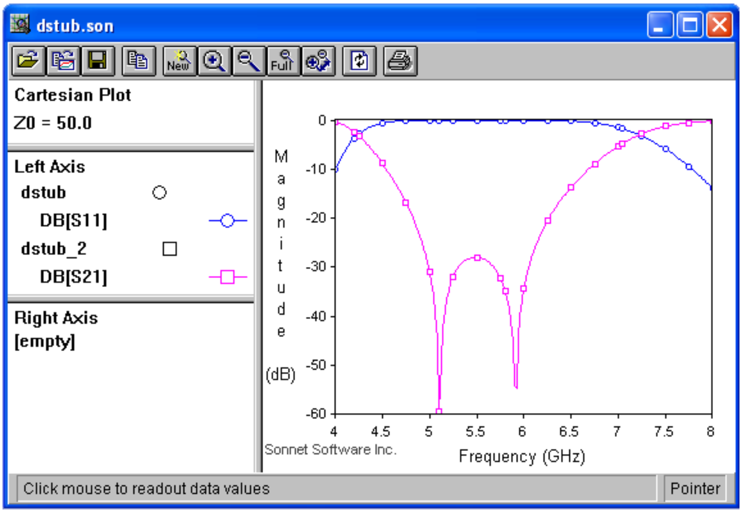
73 Double click on DB[S21] in the Unselected list.

DB[S21] is moved from the Unselected list to the Selected list.

Since a curve group defaults to the left axis with the magnitude (dB) of S-Parameters plotted, no other changes need to be made.

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

The curve group, dstub_2 is added to the plot. Note that the appearance is very similar to when both measurements were in the same curve group. There are two major differences. Different symbols are used to represent data points in this graph. In the original graph, since the two measurements were in the same curve group, the same symbol was used to represent data points. This graph is also different, in that there are now two entries, one for each curve group, in the Curve Group legend.



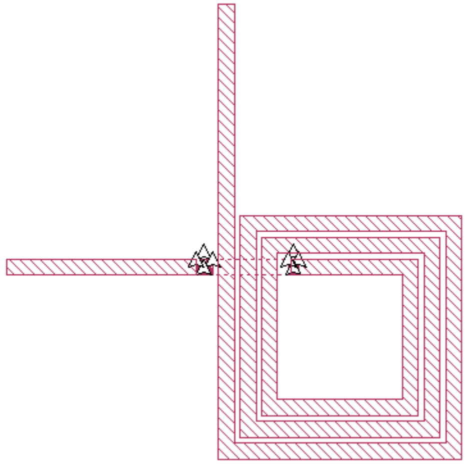
This completes the second tutorial.

CHAPTER 6 DETERMINING CELL SIZE

The third tutorial is designed to give you a demonstration in using the Cell Size Calculator to obtain the optimal cell size for your circuit as well as using the palette of standard geometries and vias. Some of the following topics are covered:

- The cell grid
- Using the Cell Size Calculator to determine cell size
- Using the Palette of Standard Geometries
- Vias

For this tutorial, you analyze a rectangular spiral with an airbridge, as shown in the figure below. First, the circuit is entered in the project editor, then an analysis is run using ***em***. You observe your output data using the response viewer, Sonnet's plotting tool. A comparison is made using a circuit with exact dimensions versus a circuit with approximations that yields a significant improvement in processing time. The goal of this tutorial is to teach you how to make wise choices in balancing the needed level of accuracy versus the processing time requirements and to teach you how vias are modeled.



The circuit is a 3 turn spiral inductor whose conductor width is 9.8 mils and spacing is 3.4 mils. The overall size is 150 mils by 150 mils. Before you enter the circuit in the project editor, you need to make some design decisions.

Calculating Cell Size for Non-Integer Dimensions

In general, you should follow these steps when determining your cell size:

- Determine critical parameters.
- Enter the critical parameters in the Cell Size Calculator to determine the optimal cell size with the minimal reduction in accuracy.

Before using the Cell Size Calculator to determine the cell size for your circuit, you must decide which parameters are the most critical. You use the most critical parameters to calculate the best cell size for your circuit in Sonnet.

In the case of the spiral inductor the most critical parameters are the conductor widths and spacings. The overall size of the spiral, 150 mils by 150 mils, is not as critical

For this example, the conductor width is 9.8 mils and the spacing is 3.4 mils. These might be the values obtained from an optimization or synthesis software. In order to model these two dimensions exactly, it would be necessary to choose a cell size of 0.2. This cell size would require an inordinately large number of subsections and hence a prohibitive amount of processing resources. So you need to calculate a cell size that provides a level of accuracy within your tolerance while using less of your memory and processing time.

For more information about cell sizes and subsectioning, please refer to Chapter 3, “Subsectioning,” in the **Sonnet User’s Guide**.

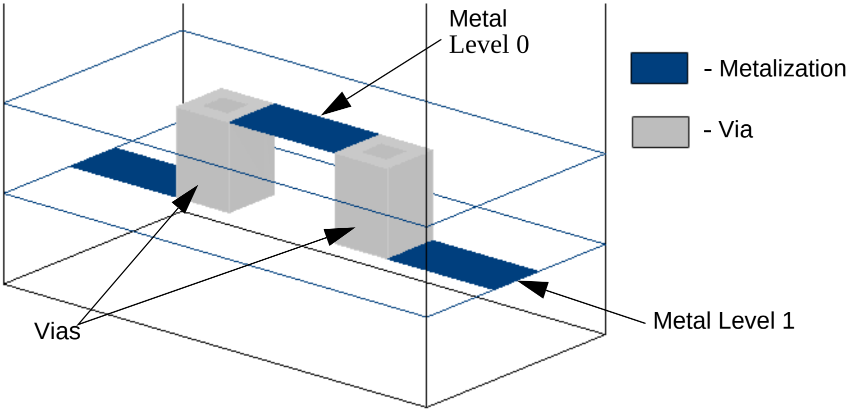
Vias

The spiral inductor uses an airbridge to connect one end of the inductor to the box-wall port. Vias, which are a special kind of subsection which allows current to flow in the z-direction between metal levels, are needed to model the airbridge.

The via features in Sonnet are quite versatile. They can be used to create the many via structures commonly found in multilayer designs. In Sonnet, vias can connect between any two metalization levels. This allows the user to create internal “level to level” vias which extend between any two metalization levels in the circuit including the two outermost levels (Top and Gnd) or the present level and ground (GND). An internal “level to level” via is used to create the vertical portion of the airbridge in this example.

Sonnet’s vias use a uniform distribution of current along their length and thus are not intended to be used to model resonant length vertical structures. Keep the via lengths small with respect to a wavelength.

To create vias, use the project editor to enter via polygons where desired. Sonnet places subsectional vias (called “via-posts”) along the entire perimeter of the via polygon. This perimeter is always one cell wide. Vias extend in the direction presently selected in the Tools menu. The length of the via is equal to exactly the thickness of the dielectric layers which it traverses. The via-posts are rectangular cylinders with a horizontal cross-sectional area equal to one cell. If you make the cell size smaller, the vias become smaller with more of them along the edge of the via polygon. Of course, the length of the via is unchanged. Current in a subsectional via is uniform throughout the body of the via and is Z directed. Via loss is determined by the metal type used for the via.

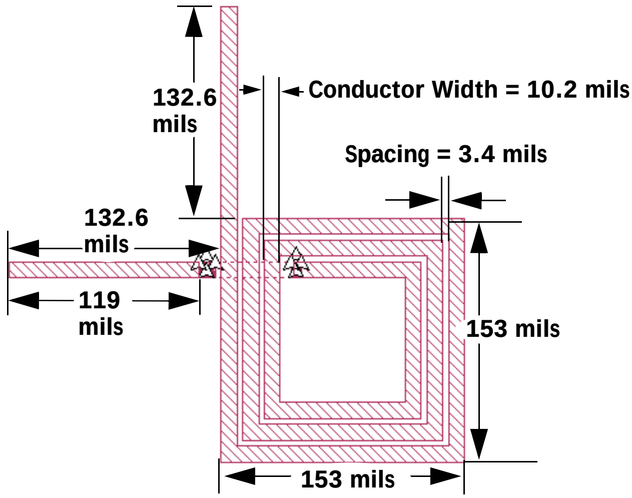


Via polygons used to construct the airbridge.

For a detailed discussion of all the types of vias and how they are modeled, please refer to Chapter 17, “Vias and 3-D Structures” in the **Sonnet User’s Guide**.

Inputting the Circuit in the Project Editor

In this section of the tutorial, you input the circuit “spiral.son.” The complete circuit, with dimensions, is shown below.



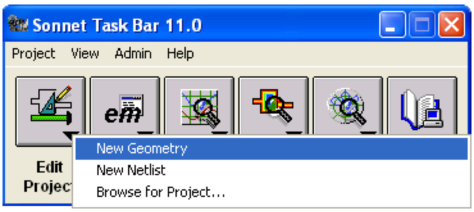
The dimensions of the example file, spiral.son.

Invoking the Project Editor

First, open the Sonnet task bar. If you do not yet know how to do this, please refer to “Invoking Sonnet,” page 30.

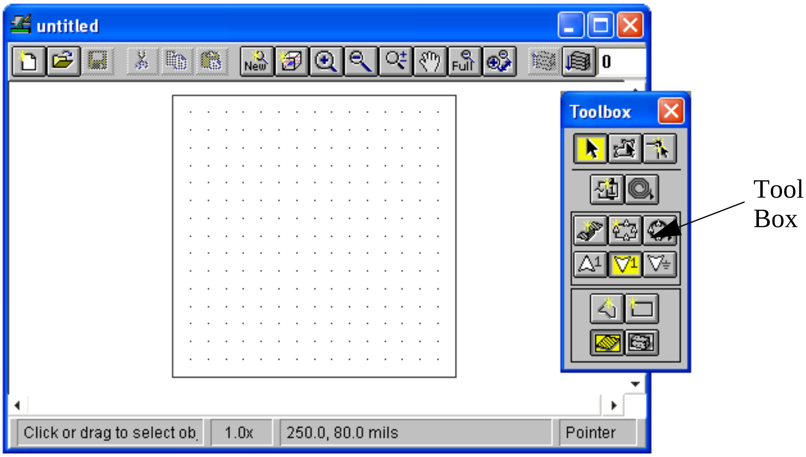
1 **Click on the Edit Project button in the Sonnet task bar.**

A pop-up menu appears on the task bar.



2 Select *New Geometry* from the pop-up menu.

The project editor window, with an empty substrate, appears on your display as shown below. The view shown in the project editor is a two-dimensional view from the top looking directly down on the substrate. The tool box, which allows easy access to commonly used functions, also appears on your display.

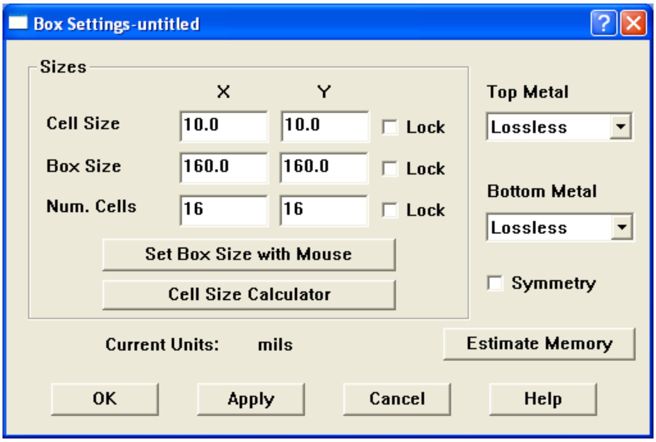


Specifying Box Settings

Before drawing the circuit, you must specify the parameters of the enclosing box, which includes the dimensions of the substrate and the cell size.

3 **Select *Circuit* ⇒ *Box* from the project editor main menu.**

The Box Settings dialog box appears on your display. This dialog box is used to set the box size and cell size dimensions for your circuit.

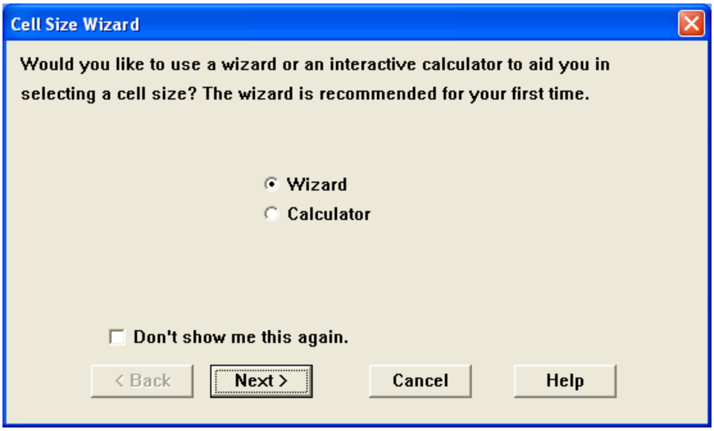


Using the Cell Size Calculator

You will enter the critical dimensions into the Cell Size Calculator to determine the optimal cell size. The automatic algorithm calculates the cell size which provides the desired accuracy while using the minimum of processing resources. This eliminates the need for you to do multiple analyses in order to find the optimal cell size.

4 Click on the Cell Size Calculator button in the Box Settings dialog box.

A dialog box appears on your display asking if you wish to go straight to the calculator or use the Wizard. For the first few times you use the Cell Size Calculator function, we suggest you use the wizard until you are familiar with how the calculator functions. The Wizard is already selected as the default choice.

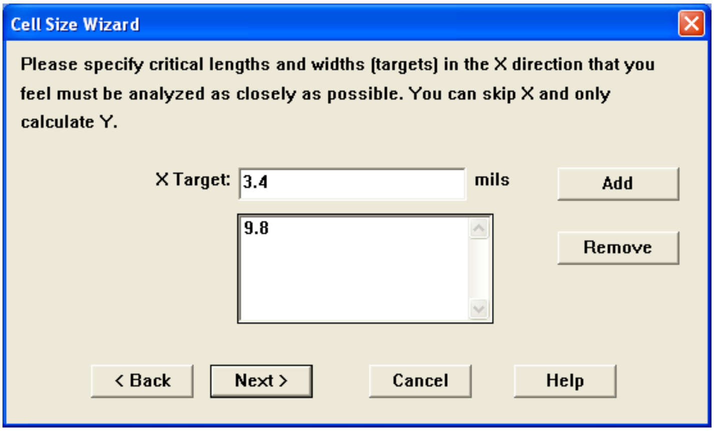


5 Click on the Next button to continue using the Wizard.

The X Direction Target Entry box appears on your display. In the introduction to this tutorial, the trace width and spacing were identified as the critical dimensions. The critical dimensions are the same in both the x and y direction for this case. It is possible to have different dimensions be critical in each direction. Remember that the cell size does not have to be a square; the x and y dimensions of a cell may be different. Hence, you enter critical dimensions in both the x and y direction when using the Cell Size Calculator.

6 **Enter 9.8 in the X Target text entry box and click on the Add button to the right.**

The trace width, 9.8 mils, is added to the list of critical dimensions in the x direction.



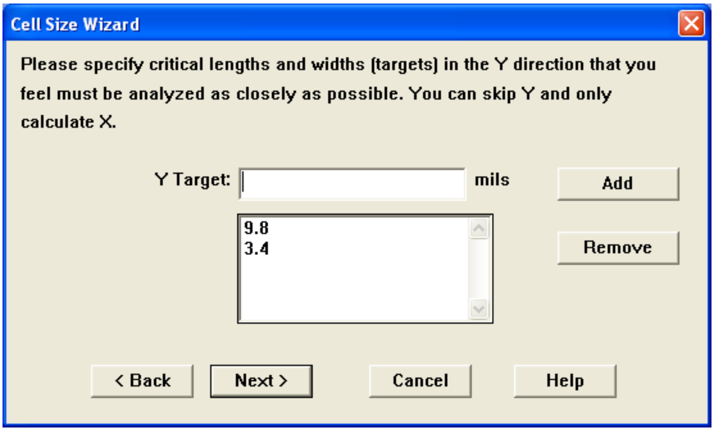
7 **Enter 3.4 in the X Target text entry box and click on the Add button to the right.**

The spacing, 3.4 mils, is added to the list of critical dimensions in the x direction. This completes the list of critical dimensions in the x direction.

8 **Click on the Next button to continue.**

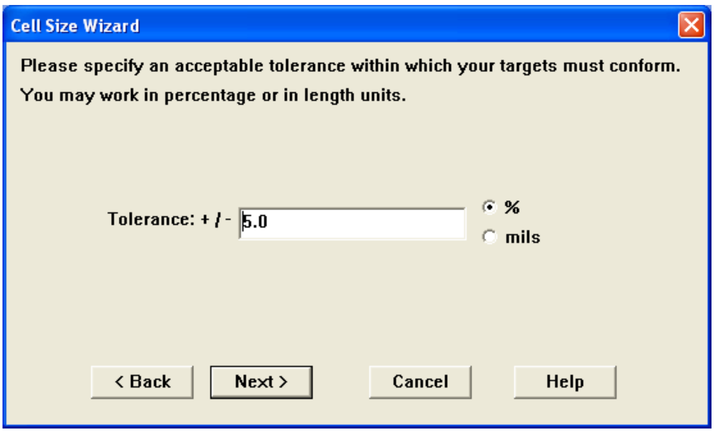
The Y Direction Target Entry box appears on your display. This is identical to the previous entry but applies to the y direction. Since the trace width and spacing are the same in both the x and y direction, the same values are entered for the y direction as were entered in the x direction.

9 Enter the values 9.8 and 3.4 in the same manner cited above for the x direction.



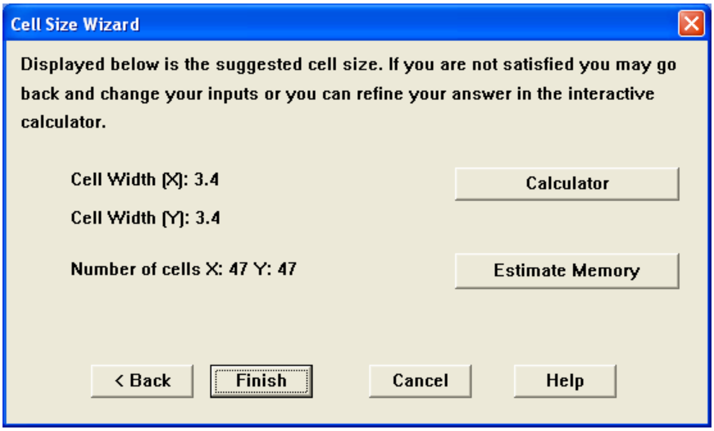
10 Click on the Next button to continue.

The Target Tolerance Entry box appears on your display. You may enter the tolerance of your dimensions as a function of percentage or length units. Your tolerance should be a non-zero value. The default tolerance is 5% which is a good value for this example. You do not need to change any settings.



11 Click on the Next button to continue.

The suggested cell dimensions are displayed, 3.4 mils by 3.4 mils for the cell size. This would make the trace width 10.2 mils.



12 Click on Finish to complete the Wizard and enter the suggested dimensions for the cell size.

The cell size dimensions now appear in the Box Settings dialog box. Note that when the new cell size is entered, the box size and Num. Cells entries are updated.

The cell size calculator used the original box size of 160 mils by 160 mils to arrive at a box size of 47 cells by 47 cells. To allow enough space for the spiral, we shall change the box size to 125 cells by 125 cells.

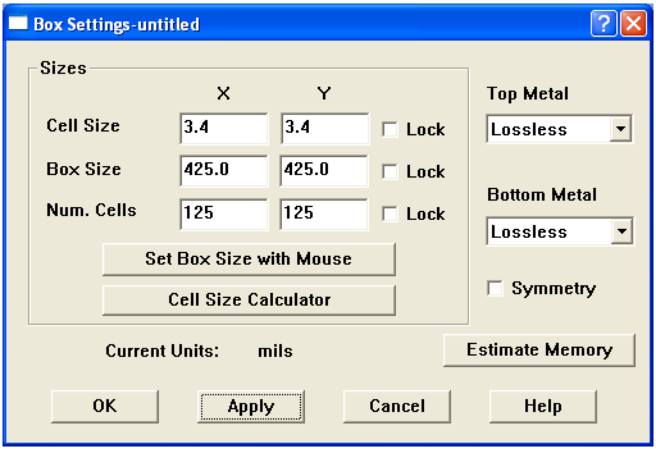
13 Enter 125 in the X text entry box in the Num. Cells row of the Box Settings dialog box.

This sets the x dimension of the box, and thereby, the substrate, to 125 cells. The box size is updated to 425 mils.

14 Enter 125 in the Y text entry box in the Num. Cells row of the Box Settings dialog box.

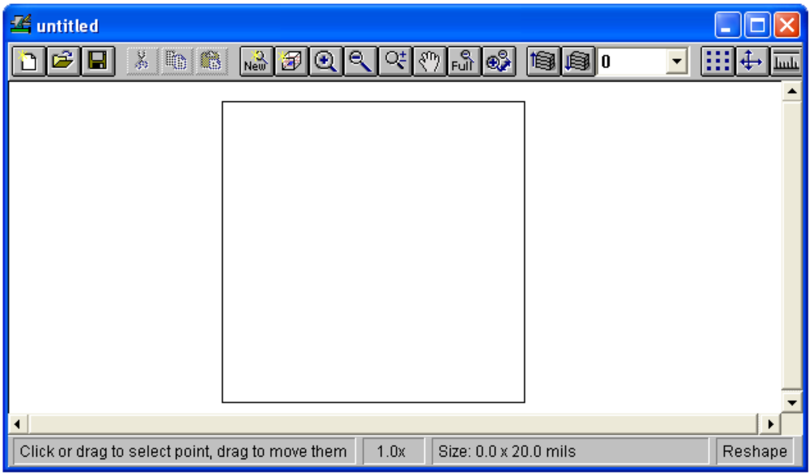
This sets the Y dimension of the box, and thereby, the substrate, to 125 cells.

Note that when these values are entered, the box size value is updated to correspond to the new number of cells.



15 Click on the OK command button in the Box Settings dialog box.

The dialog box disappears from your display and the substrate is updated to show the new size. The substrate appears blank with no cell grid visible. In fact, the cell size is simply too small for the grid to show up at a magnification of 1.0. In this particular case, zooming in to a magnification of approximately 6.0x makes the cell grid visible.The magnification level appears in the Status Bar at the bottom of the project editor window.



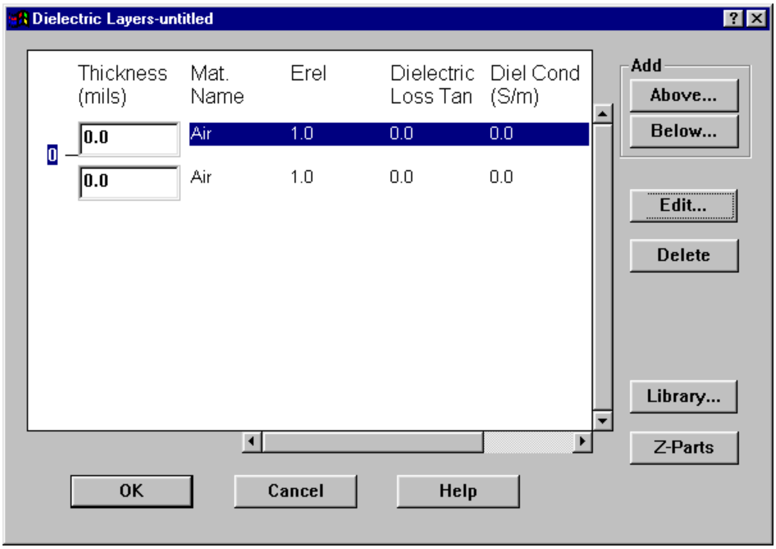
Setting the Dielectric Layers

Next, you need to specify the dielectric layers.

16 Select *Circuit* ⇒ *Dielectric Layers* from the project editor main menu.

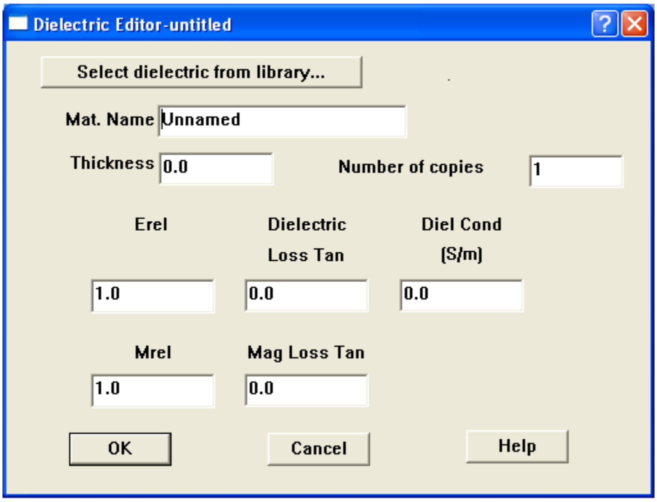
The Dielectric Layers dialog box, which allows specification of the dielectric layers of the structure, appears on your display providing you with an approximate “side view” of your circuit. The project editor “level” numbers appear on the left. A “level” is defined as the intersection of any two dielectric layers and is where your circuit metal is placed.

This circuit requires two metal levels in order to place an airbridge above the spiral. Since a level is defined as the intersection of two dielectric layers, the addition of a dielectric layer also adds another metal level.



17 Click on the Above button in the Dielectric Layers dialog box.

The Dielectric Editor dialog box appears on your display. This dialog box allows you to edit the parameters of a dielectric layer in your circuit.



18 Enter a value of "250" in the Thickness text entry box and Air in the Mat. Name text entry box.

This dielectric layer is the air above the actual microstrip. The layer thickness has absolutely no impact on execution time. Remember, the analysis is done inside of a six-sided metal box, so there is a metal top cover above the 250 mils of air. Specifying a small number for this thickness moves the top cover closer to the circuit metalization, providing stronger coupling between the top cover and the circuit metalization. This cover is set high enough to prevent coupling. Since our substrate thickness is 25 mils, this means the top cover is 10X this distance; the reason 250 mils was selected.

19 Enter a value of "1.0" in the Erel text entry box.

This is the dielectric constant for air. This value may already be present. The rest of the parameters are correct for air and do not need to be edited.

20 Click on the OK button to close the dialog box and add the dielectric layer.

The Dielectric Layers dialog box is updated to include the new entry.

21 Enter a value of "1.0" in the Thickness text entry box in the middle entry line of the Dielectric Layers dialog box, presently labeled Unnamed.

This specifies a 1 mil thick layer. The other parameters define air as the dielectric so there is no need to edit them. To edit a layer entry, double-click on the entry line to open the Dielectric Editor dialog box which allows you to change the parameters.

22 Double-click on the bottom entry line to open the Dielectric Editor dialog box.

23 Enter a value of "25.0" in the Thickness text entry box.

This specifies a 25 mil thick substrate.

24 Enter a value of "9.8" in the Erel text entry box.

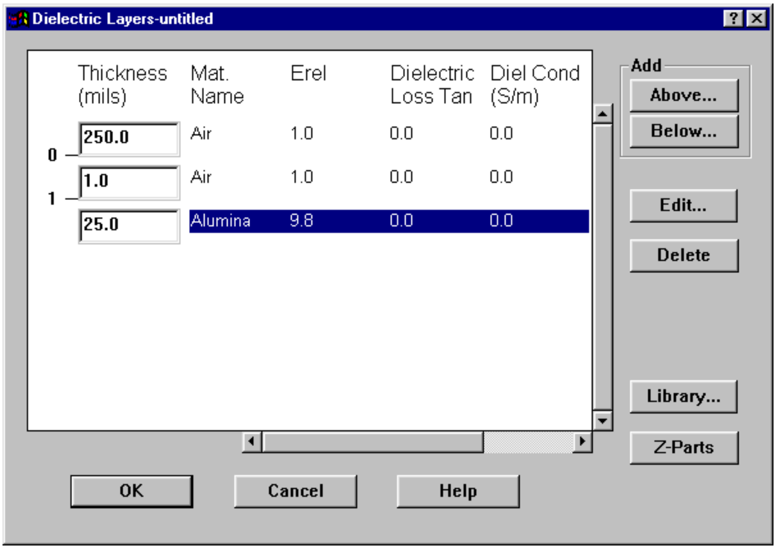
Since the dielectric constant for alumina is 9.8, this defines a 25 mil thick alumina substrate for your circuit.

25 Enter Alumina in the name text entry box.

This identifies this dielectric layer as alumina.

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

The Dielectric Layers dialog box should appear similar to that shown below.



Note that the thickness of each layer must be specified. If not, the default value, 0.0, causes ***em*** to issue an error message and stop execution.

The setup of the box size, and dielectric layers is complete. The only metal type used for this circuit is lossless, the default metal type available in any new project file. In the next section, you input the metal polygons which make up the circuit.

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

Palette of Standard Geometries

The metalization in this circuit consists of a rectangular spiral and airbridge. You use the Palette of Standard Geometries, pre-defined metalization shapes, to add the rectangular spiral to your circuit.

28 Select "1" from the Level drop list on the tool bar to go to level 1 in your circuit.

The view in the project editor is now level 1 where you wish to place the spiral.

29 Select *View* ⇒ *Measuring Tool* from the project editor main menu.

The measuring tool Readout dialog box appears on your display. You will use the Readout dialog box to position the anchor where you want to place the rectangular spiral on the substrate.

30 Click on the Anchor button in the Readout dialog box.

The Anchor Setup dialog box appears on your display.

31 Enter "132.6" in the X text entry box and "292.4" in the Y text entry box in the Anchor Setup dialog box.

These values give the position of the upper left hand corner of the rectangular spiral in your circuit. The upper left hand corner was selected because this is the reference point used in the standard palette spiral inductor.

32 Click on the OK button to close the dialog box.

The Anchor, a large +, appears on your substrate. The anchor is used to help position the spiral.

NOTE: **The precision used in placing the geometry in this example is done to ensure that the results are consistent with results displayed later in the tutorial. In many cases, the exact placement of the spiral would be unimportant.**

33 Select *Tools* ⇒ *Add Metalization* ⇒ *Rectangular Spiral* from the project editor main menu.

The Rectangular Spiral Attributes dialog box appears on your display.

34 Enter "3" in the Number of Turns text entry box.

The rectangular spiral has 3 turns.

35 Enter "10.2" in the Conductor Width text entry box.

The width of the conductor for the spiral is 10.2 mils; the value obtained in the calculations at the beginning of this tutorial.

36 Enter "3.4" in the Conductor Spacing text entry box.

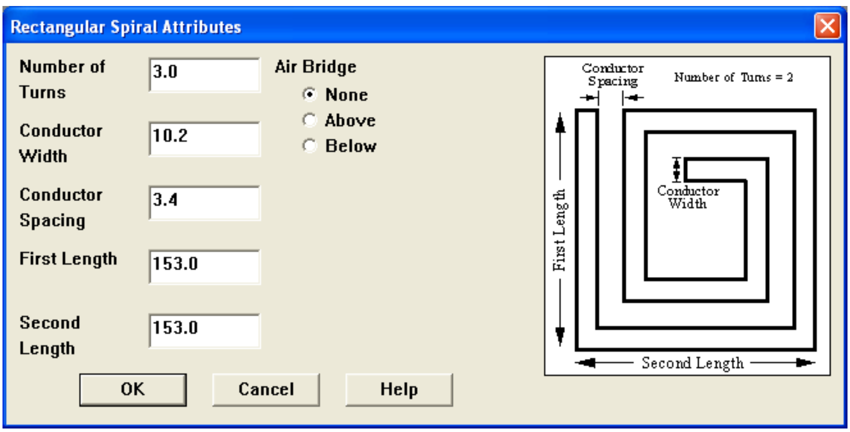
The spacing for the spiral is 3.4 mils; the value obtained in the calculations at the beginning of this tutorial.

37 Enter "153" in the First Length and Second Length text entry boxes.

This sets the overall dimensions of the rectangular spiral to 153 by 153; the values obtained in the calculations at the beginning of this tutorial.

38 Click on the "None" radio button under Air Bridge.

This setting adds no airbridge to the rectangular spiral when adding it to your circle. This setting could have been used in this case, but for the purposes of the tutorial was not used in this case. The dialog box should appears as shown below:



39 Click on the OK button to add the Spiral.

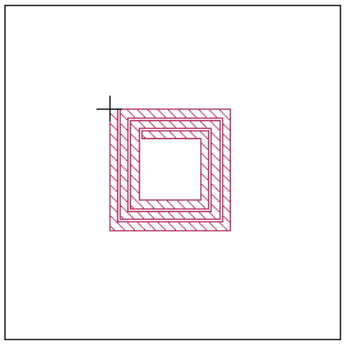
A black outline of the spiral with a cursor fixed at the upper left hand corner appears on your circuit display. As you move the mouse the position of the cursor and spiral change accordingly.

TIP

You may want to zoom in on the location of the spiral to make it easier to position the spiral before adding it to your circuit. You may zoom in using the Space Bar, middle button on your mouse, or the button on the tool bar.

40 Move the spiral until the cursor is on the Anchor and click the left mouse button.

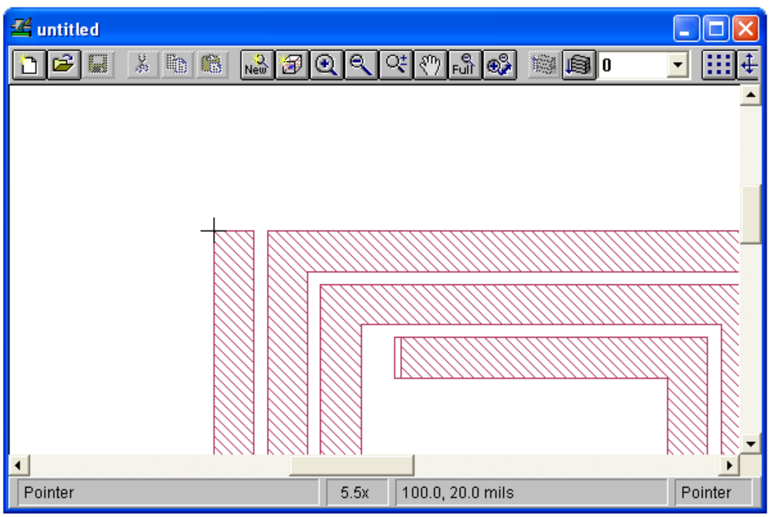
The metalization of the spiral is drawn on your circuit. This places the rectangular spiral at the desired location in your circuit.



Next, you need to extend the conductor to the box wall.

41 If you have not already done so, zoom in on the end of the conductor.

Your circuit should appear similar to that pictured below.



42 **Click on the Reshape button in the project editor tool box.**

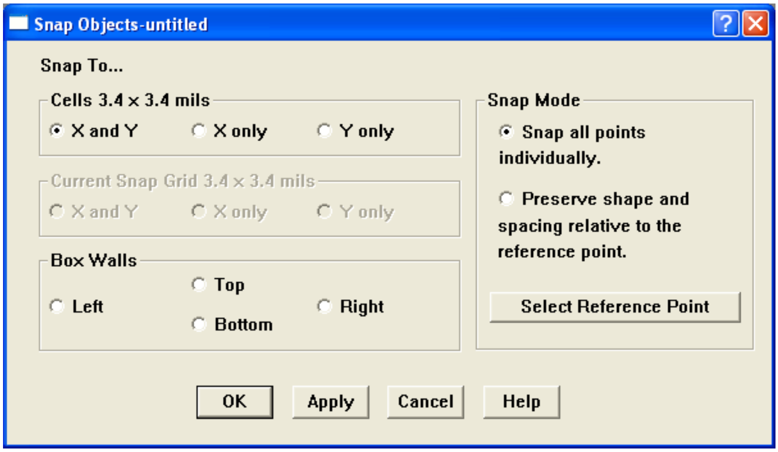
This mode, as indicated by the cursor, allows you to select points on a polygon and move them.

43 **Drag your mouse to select the end points of the conductor.**

The two points appear highlighted.

44 **Select Modify ⇒ Snap To from the main menu.**

The Snap Objects dialog box appears on your display.

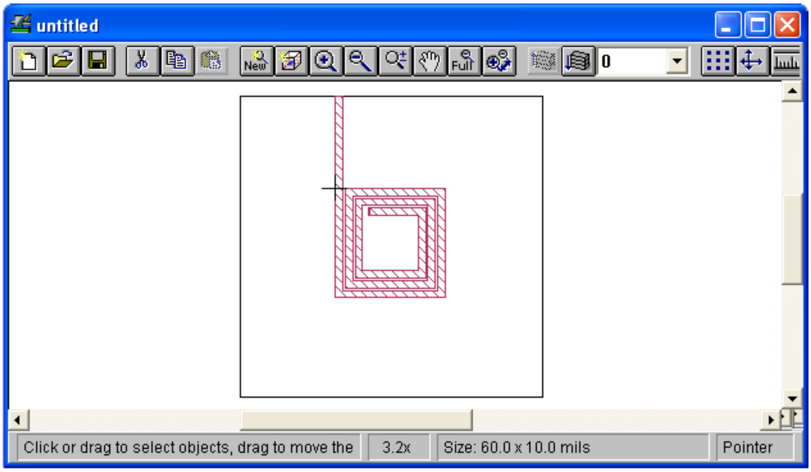


45 **Click on the Top radio button in the Box Walls section of the dialog box.**

This will extend the conductor to the box wall.

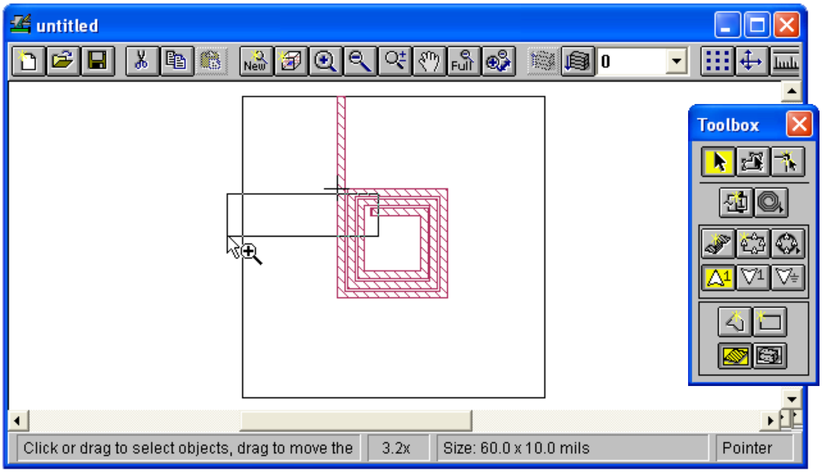
46 Click on the OK button to close the dialog box and apply the changes.

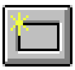
The full view of your circuit should resemble the one shown below.



Creating an Airbridge

47 Zoom in on the area of the upper left hand corner of the spiral to the left box wall.





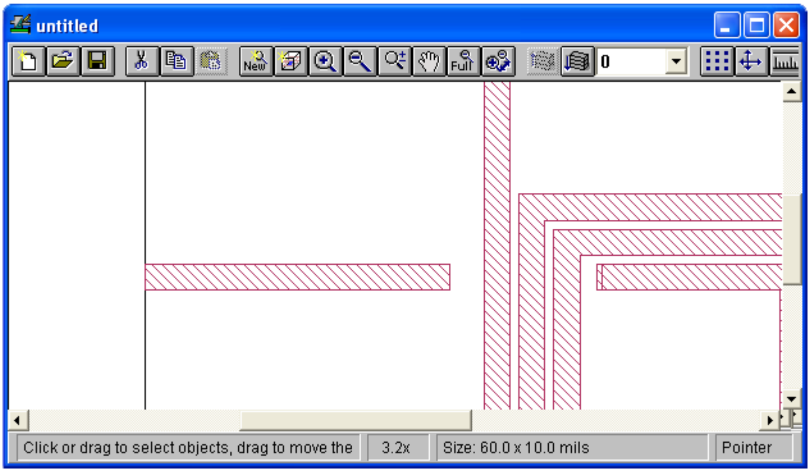
48 Click on the Add a Rectangle button in the project editor tool box.

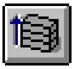
49 Drag the mouse to create a rectangle 119 mils by 10.2 mils.

As you move your mouse when adding the rectangle, the size of the polygon is shown in the status bar. Use this to get the proper size rectangle. This polygon is a feedline which connects to the spiral using an airbridge.

50 **Move the rectangle so that the left end is on the box wall and the top lines up with start of the spiral.**

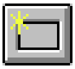
Your circuit should look like this:





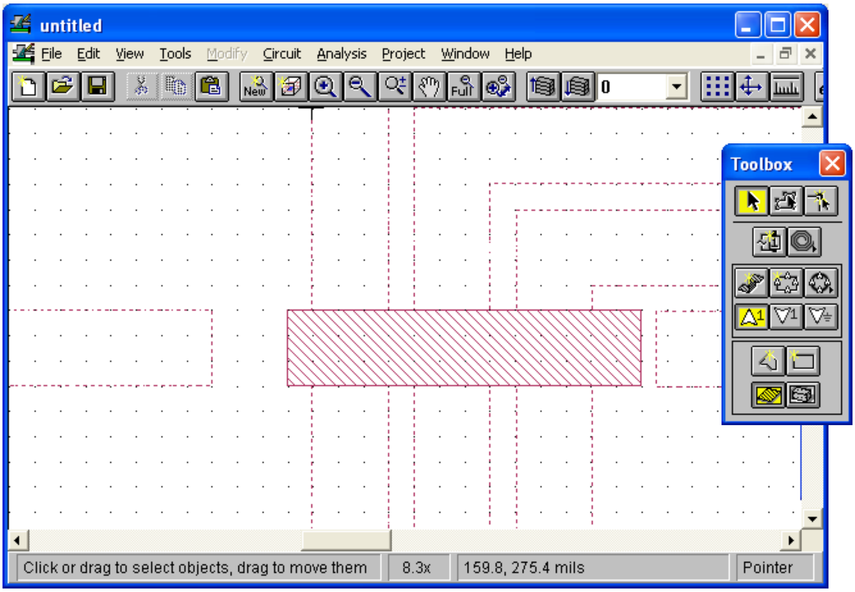
51 **Click on the Up One Level button on the tool bar.**

Level 0 of your circuit is displayed. There is no metalization on this level yet, however, the outline of the metalization placed on Level 1 appears as a dashed line.



52 **Add a rectangle which extends from three cells after the end of the feed line to a cell before the beginning of the spiral (44.2 mils by 10.2 mils).**

Zoom in if you need to. Use the dashed lines as guides for the ends of the polygon.



53 **Select Tools ⇒ Add Via ⇒ Down One Level if it is not already selected.**

Any vias added will extend down one level from the originating level.

TIP

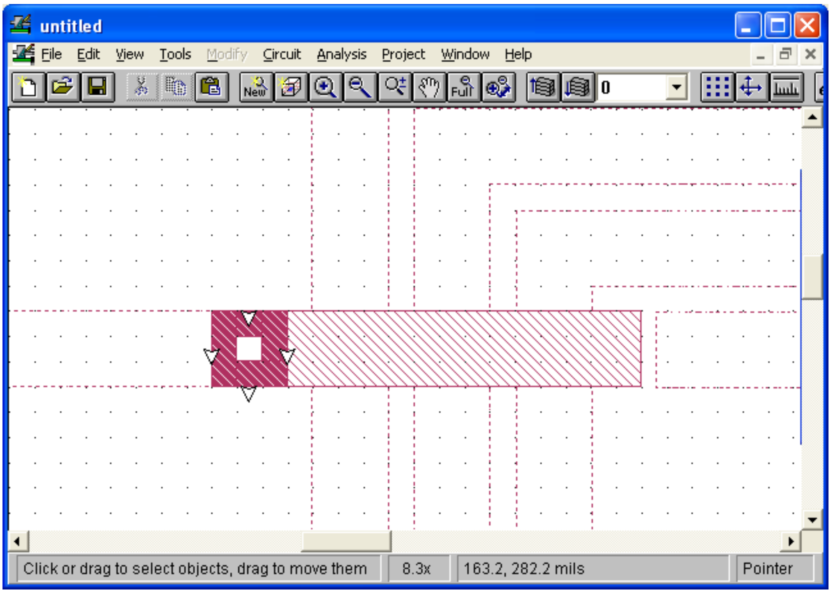
You may click on the Via Down One Level button in the tool box.

54 **Hold down the Shift key and select Tools ⇒ Add Via ⇒ Draw Rectangle to add the via polygon.**

Holding down the shift key allows you to add multiple via polygons to your circuit. If the Via Direction Notice appears, click on OK to close the message. The Add Via Rectangle mode is indicated by a change in the appearance of the cursor.

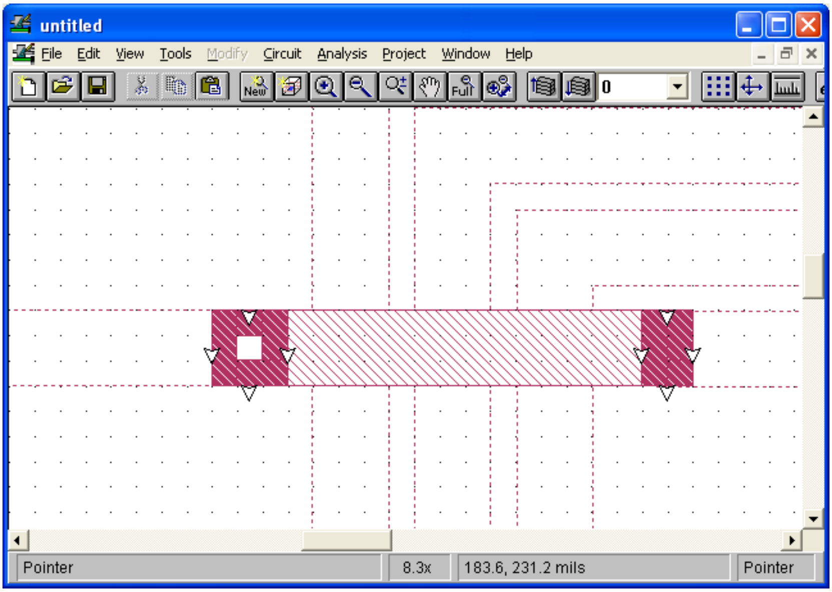
55 **Click on the upper corner of the outline of the metal below and drag your mouse to the lower corner of the metal polygon, then release the mouse.**

The via polygon is drawn in your circuit and should appear similar to that shown below. Note that the arrows on the via point downward to indicate the direction of the via polygon.

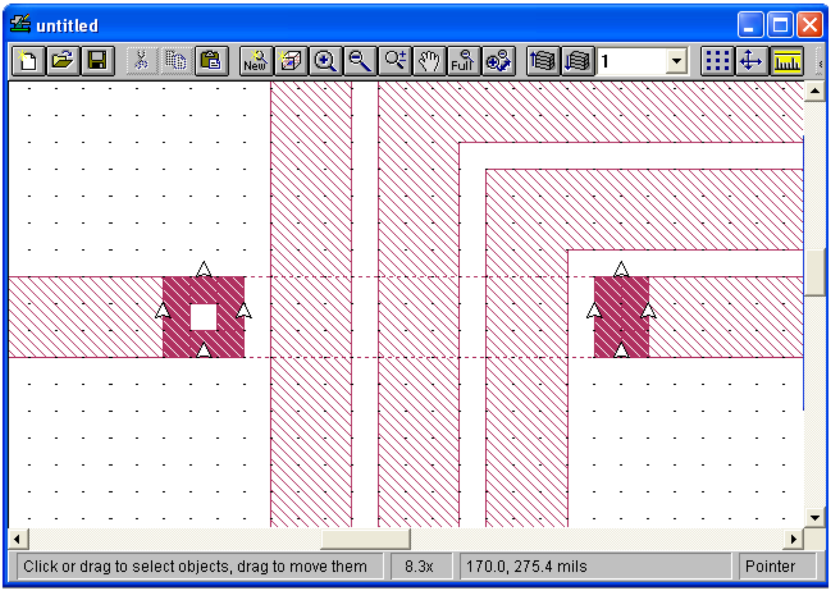


56 Click on the upper right hand corner of the metal polygon on this level and drag your mouse over two cells and down three. Release the mouse.

A via polygon connecting the airbridge to the beginning of the spiral is added to your circuit as shown below. Note that the first via polygon does not have metal in the middle but this one does. A via polygon has metalization in a one cell thick wall around the perimeter. The second via polygon is only two cells wide so that there is no space in the middle.



If you go down one level, the vias now appear in your circuit, but with the triangles pointing up, to indicate that the via originated on the level above. The via on the right overlaps the metal of the spiral; the overlap area does not appear in reverse video, but the triangle indicates where the edge of the via polygon is.



57 **Press the Escape key to return to pointer mode.**

This exits Add Via mode.

Adding Ports and Reference Planes

58 **Go to Level 1 if you are not already there.**

59 **Hold down the Shift key and click on the Add Port button in the project editor tool box.**

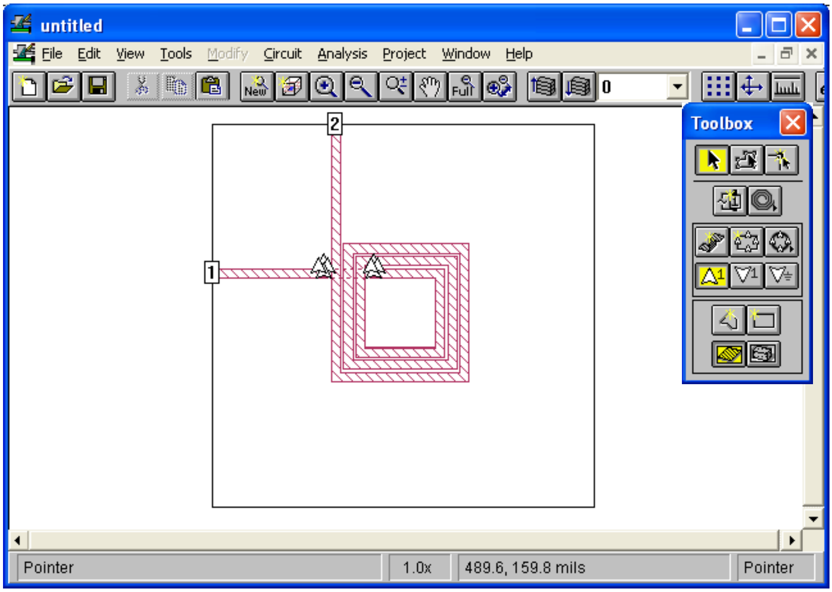
Holding down the shift key allows you to add multiple ports without returning to pointer mode.

60 **Click on the feedline at the left box wall.**

This adds Port 1 to the circuit.

61 Click on the end of the conductor at the top box wall.

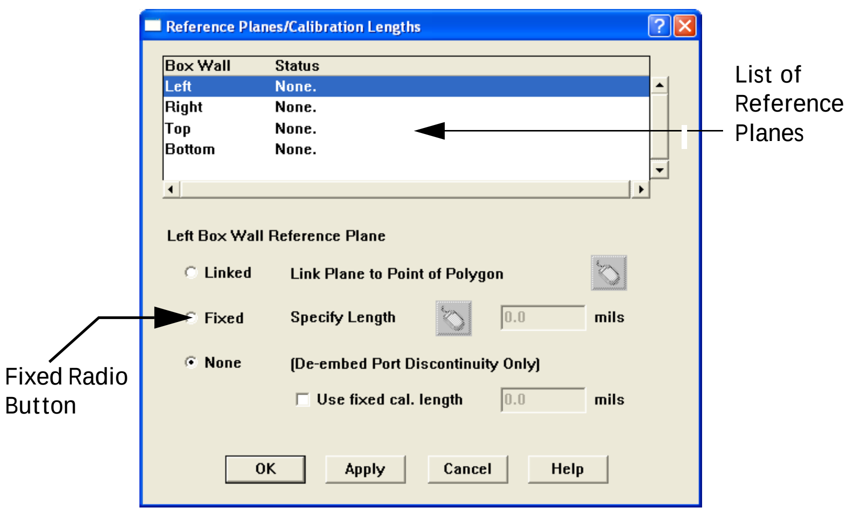
This adds Port 2 to the circuit.



62 Push the Escape key to return to pointer mode.

63 Select *Circuit* ⇒ *Ref. Planes/Cal. Length* from the project editor main menu.

The Reference Planes/Calibration Lengths dialog box appears on your display. In the list, Left, Right, Top and Bottom refer to the box wall on which the port is situated.



64 Select the "Left" entry in the list of reference planes.

The Left entry line is highlighted.

65 Click on the Fixed radio button.

This enables the text entry box for the fixed value. This choice uses a fixed value for the reference plane.

66 Enter "119" in the Length text entry box for the Fixed radio button and click on the Apply button.

The reference planes extends from the port on the left side of the box 119 mils into the circuit. The left entry line now reads "Fixed, plane length of 119 mils."

67 Select the "Top" entry in the list of reference planes.

The Top entry line is highlighted.

68 Click on the Linked radio button.

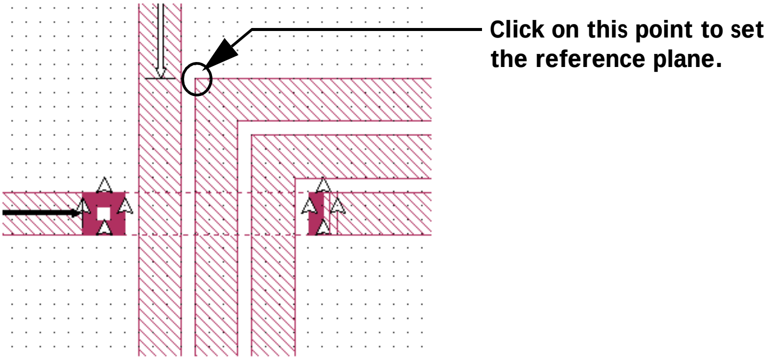
This choice allows you link the reference plane to a point on a polygon. If that point is moved for any reason, the length of the reference plane changes accordingly.

69 Click on the Mouse button next to the linked radio button to select a point.

The dialog box disappears and the cursor changes to a cross.

70 Click in the circuit on the corner of the spiral as pictured below.

You will need to zoom in on this area of the circuit.



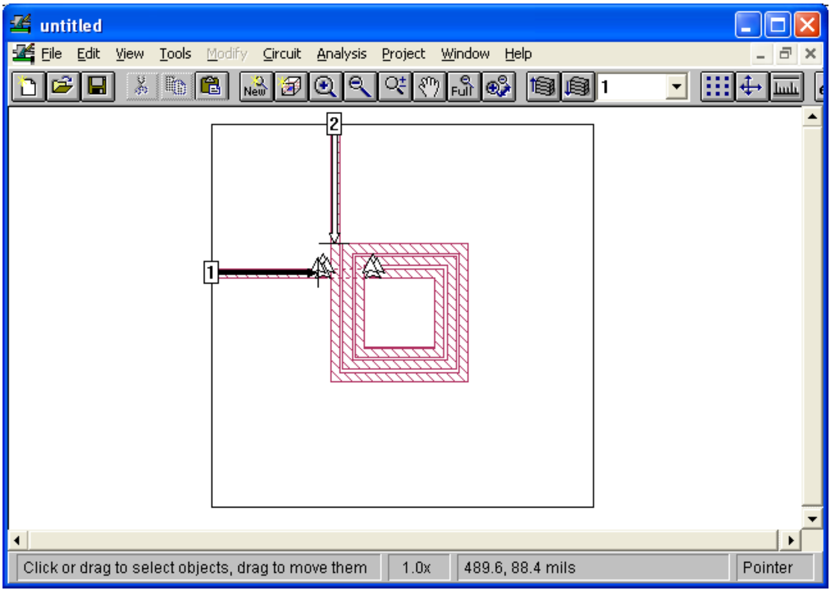
The dialog box re-appears. The top entry line now reads "Linked, distance from port: 132.6 mils."

71 Click on the OK button to apply the reference planes and close the dialog box.

The reference planes extend from the box wall for the length input in the Reference Planes dialog box. When analyzed by ***em***, the circuit is automatically de-embedded to the reference planes when the De-embed option is selected. De-embedding is the process by which the port discontinuity and any reference plane lengths are removed from the analysis results.

Note that the fixed length reference plane is represented by a solid black arrow on the circuit and the linked reference planes is shown by the outline of an arrow.

Inputting the circuit is now complete. Your circuit should appear as shown below.



72 Select *File* ⇒ *Save* from the project editor main menu and save the file under the name "spiral.son" in your working directory.

The Save dialog box appears on your display. You need to save the circuit file before analyzing with ***em***.

Em - The Electromagnetic Simulator

In the next part of this tutorial, you analyze the circuit "spiral.son" which you input in the project editor.

Setting Up the Analysis

73 Select *Analysis* ⇒ *Setup* from the project editor main menu.

The Analysis Setup dialog box appears on your display. For this circuit, you will analyze using an adaptive sweep from 0.2 GHz to 2 GHz. An adaptive sweep provides approximately 300 data points in the band. For a detailed discussion of Adaptive Band Synthesis, see Chapter 9, “Adaptive Band Synthesis (ABS),” in the **Sonnet User’s Guide**.

74 Select Adaptive Sweep (ABS) from the Analysis Control drop list if it is not already selected.

The text entry boxes for the sweep are updated for an adaptive sweep.

75 Enter “0.2” in the Start text entry box and “2” in the Stop text entry box.

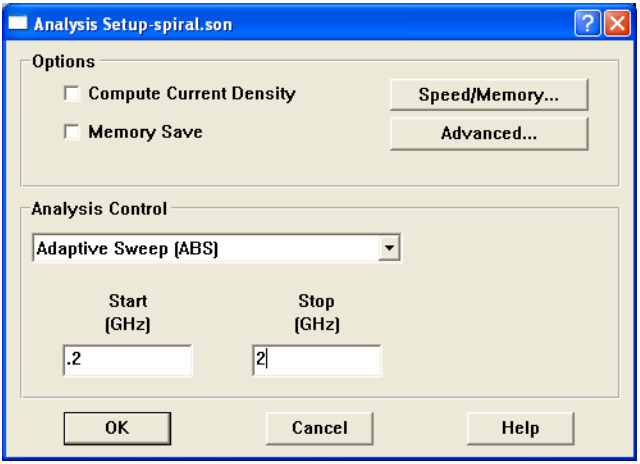
This defines the frequency band of the adaptive sweep as 0.2 GHz to 2.0 GHz.

Selecting Run Options

Run options for ***em*** are available in the Analysis Setup dialog box in the project editor. This example uses only the De-embed option, which is set by default.

De-embedding is the process by which the port discontinuity and any reference plane lengths are removed from the analysis results. Inaccurate data may result from failing to implement this option, even when you are not using reference planes. For a detailed discussion of de-embedding refer to Chapters 7 and 8 in the **Sonnet User’s Guide**.

The analysis setup is now complete. The dialog box should appear similar to that shown below.



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

77 Select *File* ⇒ *Save* from the project editor main menu.

This saves the analysis setup as part of your project file. You must perform the save before running ***em***. If the file is not saved when ***em*** is invoked, a request to save the file appears before ***em*** executes.

Executing the Analysis

78 Click on the Analyze button on the project editor tool bar.

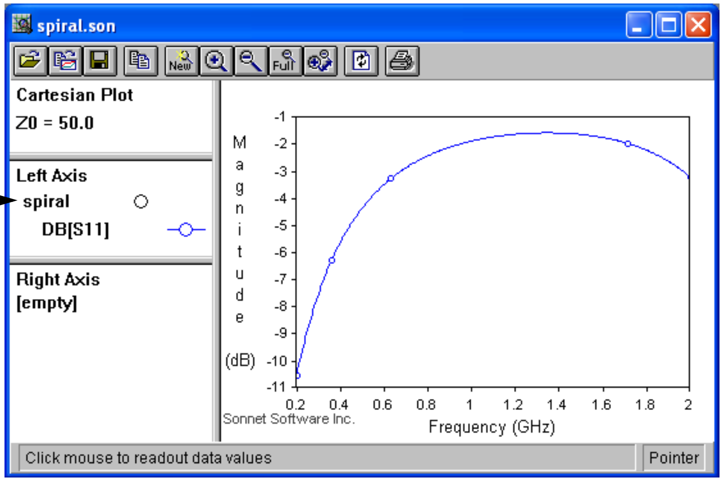
The analysis engine, ***em***, is launched and the analysis monitor appears on your display with the Response Data window shown. As the analysis progresses, the response data is output to the Analysis monitor and the progress bar is updated. When the analysis is complete, the "Analysis successfully completed." message appears in the analysis monitor window.

Viewing your Response

In the next part of this tutorial, you plot an equation in the response viewer and compare the data to the analysis results of the circuit analyzed at the exact dimensions.

79 **Select *Project* ⇒ *View Response* ⇒ *New Graph* from the analysis monitor main menu.**

The response viewer appears on your display with the curve group spiral which includes the DB[S11] measurement. Note that since the analysis is an adaptive sweep, symbols only appear on the discrete data points.



"spiral" curve group is listed in the Curve Group legend.

You will use the equation for inductance to plot the inductance of spiral.son and the inductance of the circuit which uses the exact dimensions so that you can compare the results.

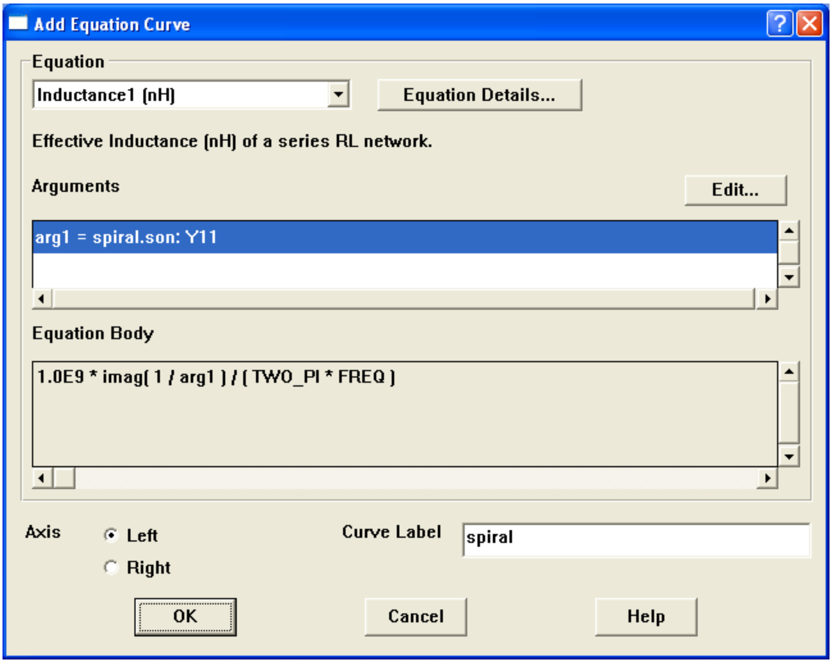
NOTE: **You can invoke the response viewer as soon as the analysis of one frequency is complete. To input subsequent information produced by *em*, select *Graph* ⇒ *Freshen Files* from the response viewer menu.**

TIP

You could also use the View Response button on the analysis monitor's tool bar to invoke the response viewer.

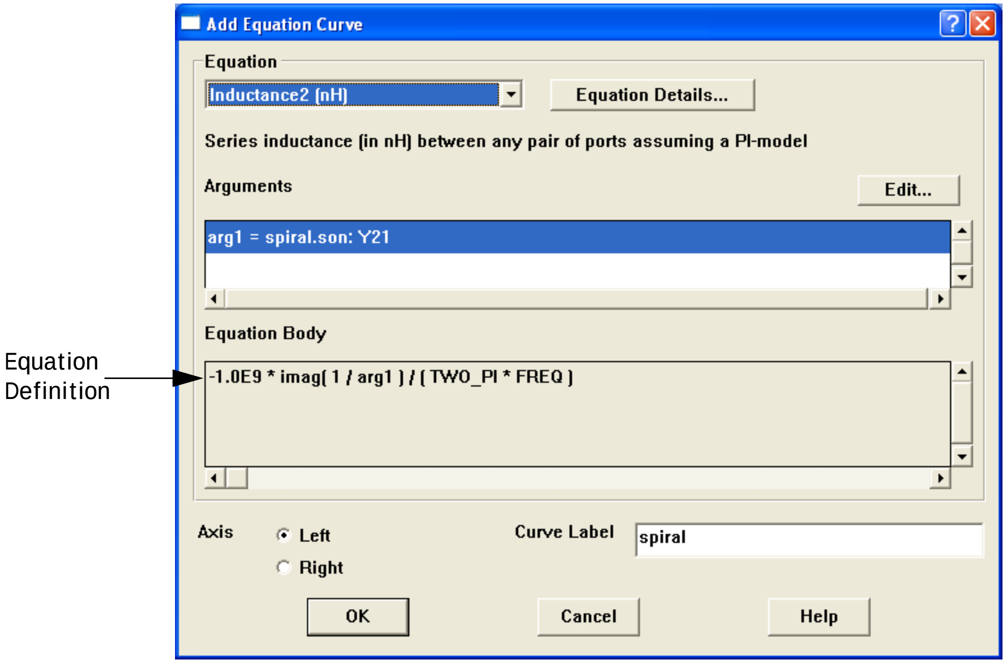
80 Right-click in the Curve Group legend and select “Add Equation Curve” from the pop-up menu which appears. Or you may select Curve ⇒ Add Curve Group from the main menu of the response viewer.

The Add Equation Curve dialog box appears on your display.



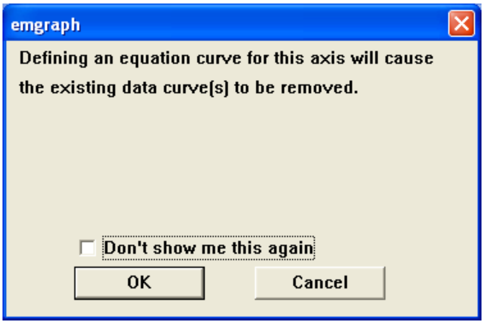
81 Select "Inductance2 (nH)" from the Equation drop list.

The equation for inductance appears in the Arguments section of the Add Equation Curve dialog box as shown below. Inductance2 is the series inductance and does not include any capacitance to ground. The definition of any given equation is displayed in the Equation Body section of this dialog box.



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

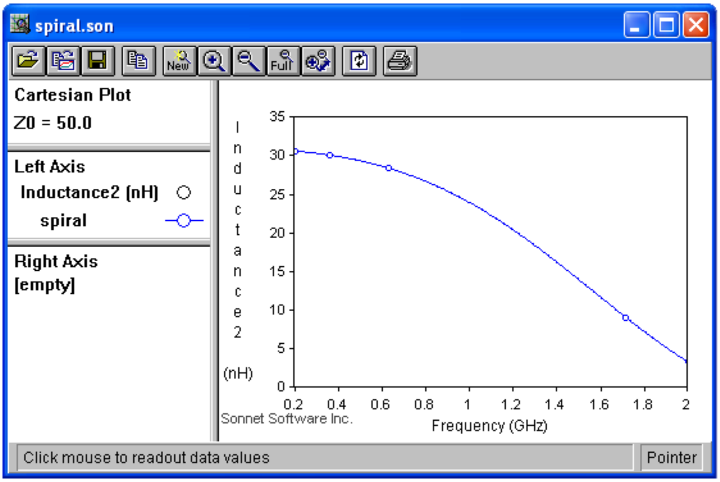
You may not plot a data curve and an equation curve on the same axis of a plot; therefore, when you add an equation curve, it is necessary to delete the existing data curve. A warning message about the deletion of the data curve appears as shown below.



83 Click OK to close the warning message box.

If you do not wish to have this warning appear, click on the "Don't show me this again" checkbox" before you click on the OK button.

The plot is updated with the inductance of your circuit as a function of frequency.



Adding a File to a Graph

For comparison, add the project file for the spiral inductor analyzed at the exact dimensions. The project file for the exact circuit, which includes the response data, has been provided.

84 Copy the example, Spi_exact, to your working directory.

You can access the example through the online manuals. If you do not know how to do this, see "Obtaining the Example Projects," page 31. Save the file to your working directory.

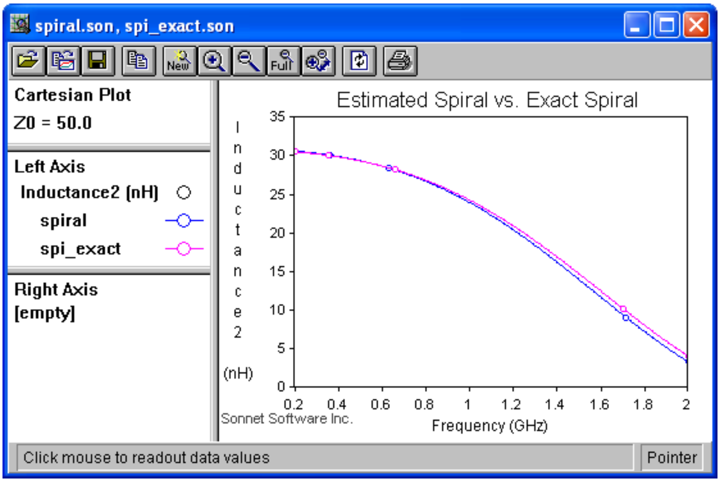
85 Click on the Add File button on the tool bar of the Response Viewer.

A Browse dialog box appears on your display. Use the browser to locate the "spi_exact.son" project file which was copied in the previous step.

86 Click on the OK button to add the new project file.

If an equation is presently displayed in your plot, adding another project file will add the same equation curve for the new project.

The plot now displays the inductance of both projects. As you can see the results achieved with the estimated circuit are very close to those of the exact circuit; however, the estimated circuit only took a fraction of the processing time due to a much bigger cell size.



The graph title was added by right clicking in the area above the graph in the plot window, then selecting Options from the pop-up menu that appears. The title is entered in the Graph Options dialog box opened by selecting this command.

This completes the third tutorial.

Index

K

L

M

N

O

P

R

S