

Sonnet® Supplemental Tutorials

Release 10



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.

Sonnet® Supplemental Tutorials

Printed: November 2004

Release 10

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

© Copyright 1989,1991,1993, 1995-2005 Sonnet Software, Inc. All Rights Reserved

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

Copyright Notice

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

Trademarks

The program names ***xgeom***, ***emstatus***, ***emvu***, ***patvu***, ***dxfgéo***, ***ebridge***, ***emgraph***, ***gds***, ***emserver***, ***emclient*** and ***sonntawr*** (lower case bold italics), Lite, LitePlus, Level2 Basic, Level2 Silver, and Level3 Gold are trademarks of Sonnet Software, Inc.

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

UNIX is a trademark of Unix Systems Labs.

Windows 95, Windows 98, Windows2000, Windows ME, Windows XP and Windows NT are trademarks of Microsoft, Inc.

X Window System is a trademark of the Massachusetts Institute of Technology.

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

SPARCsystem Open Windows, SUN, SUN-4, SunOS, Solaris, SunView, and SPARCstation are trademarks of Sun Microsystems, Inc.

HP, HP-UX, Hewlett-Packard are trademarks of Hewlett-Packard Company.

ADS, Series IV, Touchstone, and Libra are trademarks of Agilent Technologies.

GDSII is a trademark of Calma Company.

FLEXlm is a registered trademark of Globetrotter Software, Inc.

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

IBM is a registered trademark of International Business Machines Corporation.

MS-DOS and Windows are registered trademarks of Microsoft Corporation.

Table of Contents

Table of Contents	5
1 Introduction.....	9
2 Parameter Sweep and Optimization Tutorial	11
Setting Up Parameters	12
Anchored Parameters.....	13
Symmetric Parameters.....	18
Parameter Sweep	22
Setting Up a Parameter Sweep	22
Executing the Parameter Sweep	26
Observing the Parameter Sweep Data	27
Optimization.....	31
Entering New Nominal Values	32
Setting Up an Optimization.....	32
Running an Optimization.....	38
Observing your Optimization Data	39
Accepting the Optimized Values	42
3 Circuit Subdivision Tutorial	47
Obtaining the Example File	48
Adding the Subdivision Lines	49
Setting Up Circuit Properties.....	52
Subdividing Your Circuit.....	54
Analysis of the Network File.....	58
Additional Improvements	61

4	Conformal Mesh Tutorial	63
	No Polygon Overlap	65
	Invoking Conformal Meshing	67
	Viewing Conformal Meshing	68
5	Microwave Office Interface Tutorial	71
	Introduction	71
	Tutorial Topics	72
	Obtaining the Example Project	73
	Editing in Microwave Office	76
	Selecting Sonnet as your EM Analysis Engine	77
	Selecting Analysis Controls	78
	Running the Simulation	81
	Native Editor	86
	Editing your EM Structure in Sonnet	88
	Running the Simulation	93
	Working Outside Microwave Office	95
6	GDSII and DXF Translator Tutorial	101
	Obtaining the Translator Example Files	103
	Determine Level Mapping	103
	Define Dielectric Layer and Metallizations	115
	Remove Parts of the Circuit Not Being Used	116
	Decide on a Substrate Size and Cell Size	117
	Change Polygons to Have the Proper Fill	118
	Align the Circuit to Grid Points	119
	Move Points Around as Needed	121
	Add Vias	121
	Add Ports and Reference Planes	122

7	Agilent Interface Tutorial	123
	A Simple Example	124
	Adjusting the Sonnet Project Prior to Analysis	130
	Substrate Size and Cell Size	131
	Change Polygons to Have the Proper Fill	134
	Align the Circuit to Grid Points	136
	Move and add points as needed	138
	Using a drawn bounding box	143
	Adjustments that may be needed	144
8	A Two-Dimensional Far Field Viewer Tutorial.....	145
	Creating an Antenna Pattern File	146
	Running the Far Field Viewer	147
	Calculating the Response	148
	Selecting Phi Values	149
	Selecting Frequencies	150
	Selecting the Response	151
	Zooming	155
	Probing the Plot	156
	Re-Normalizing the Plot	158
	Changing to a Polar Plot	160
	Turning Off the Legend	160
	Changing the Radius Axis	161
	Selecting a Frequency Plot	164
	Viewing a Surface Plot	168
	Saving the Far Field Viewer File	168
	Exiting the Far Field Viewer Program	169
	References	169
	Index.....	171

Chapter 1 Introduction

This manual is intended to provide you with supplemental tutorials on important features in the analysis engine and on how to use other Sonnet modules. There are presently seven tutorials as listed below:

- [Parameterization and Optimization](#) - teaches you how to add parameters to a circuit and perform a parameter sweep and an optimization
- [Circuit Subdivision](#) - shows you how to use circuit subdivision to obtain accurate answers for very large circuits
- [Conformal Mesh](#) - shows a simple example using Conformal Mesh
- [Microwave Office Interface](#) - teaches you how to use the Microwave Office Interface
- [GDSII and DXF Translator](#) - teaches you how to use both the DXF and GDSII translators since the translators are very similar in operation
- [Agilent Interface](#) - teaches you how to use the Agilent Interface
- [Far Field Viewer](#) - teaches you how to use the far field viewer to observe radiation patterns

Sonnet Supplemental Tutorials

These tutorials assume you are familiar with Sonnet programs and their functions and are capable of running a simple analysis. If you are a new Sonnet user, we suggest you start with the Sonnet Tutorial manual included in your documentation. These tutorials will teach you the basics of Sonnet.

The **Sonnet User's Guide** contains detailed background discussions of these topics. Each tutorial will provide a cross reference to the pertinent topics in the User's Guide.

Chapter 2 Parameter Sweep and Optimization Tutorial

This tutorial shows you how to set up parameters in a circuit, set up and execute a parameter sweep, set up and execute an optimization and view the results of both a parameter sweep and an optimization. For a detailed discussion of parameterization and optimization, please refer to Chapter 10 "Parameterization and Optimization of a Geometry Project" on page 155 in the **Sonnet User's Guide**.

This tutorial presumes that you are familiar with Sonnet Software, especially the project editor and the analysis monitor. If you are new to Sonnet, please review the tutorials in Chapter 2 and Chapter 3 of the **Sonnet Tutorial** before performing this tutorial.

This examples uses the Sonnet example [Par_dstub](#). If you do not know how to obtain a Sonnet example, select *Help* \Rightarrow *Examples* from any program menu, then click on the **Instructions** button.



TIP

If you are using the PDF manuals to read this section, click on the blue link above to take you to the Par_dstub example.

This is an example of a microstrip interdigital bandstop filter. This circuit is used to perform a parameter sweep and optimization. Most parameter sweeps and optimizations will present more of a challenge, but we have deliberately chosen a simple example to more clearly demonstrate Sonnet's methodology.

Our goal is to design the bandstop filter such that a stopband exists from 5 - 6 GHz and the passbands are from 1 - 4 GHz and 7-10 GHz.

- 1 Open Par_dstub in the project editor.**
- 2 Select *File* ⇒ *Save As* from the project editor main menu.**

Since this file is a Sonnet example, it is a read only project. In order to be able to edit the circuit and save those changes, you must save a copy to your working directory.

Use the Save As browse window to save a copy of par_dstub.son to your working directory.

Setting Up Parameters

Before executing either a parameter sweep or optimization, it is first necessary to parameterize your circuit. Parameterization should be done to those dimensions you deem critical to the circuit's response and which are therefore more likely to change as the design progresses.

For this example, you will enter three parameters: two anchored parameters which are linked and one symmetric parameter.

An anchored parameter is one in which one end of the parameter is a fixed point with a point set which moves in reference to that fixed, or anchored point. A symmetric parameter is one with two reference points and two point sets which move relative to the center point between the reference points.

Parameters which appear in more than one place in a circuit but are of the same length and name are linked. Changing the value of one also changes the value of the other.

For a detailed discussion of parameters and their definitions, please refer to “Parameters,” on page 156 of the **Sonnet User’s Guide**.

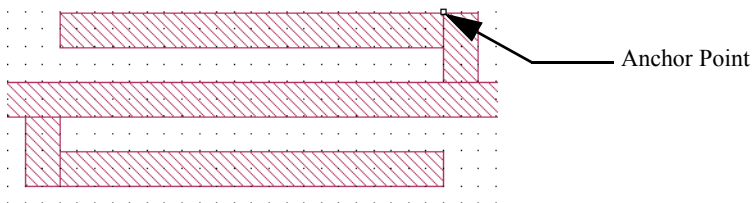
Anchored Parameters

The linked anchored parameters are input first, followed by the symmetric.

- 3 Select Tools ⇒ Add Parameter ⇒ Add Anchored from the project editor’s main menu.**

This places the project editor in Add an Anchored Parameter mode indicated by the change in cursor. Note that the message “Click Mouse to Specify the Anchor Point” appears in the status bar at the bottom of the project editor window. As you add a parameter, directions for each step appear in the status bar.

- 4 To specify the Anchor point for the parameter, click the mouse on the corner of the upper stub, as shown in the picture below.**



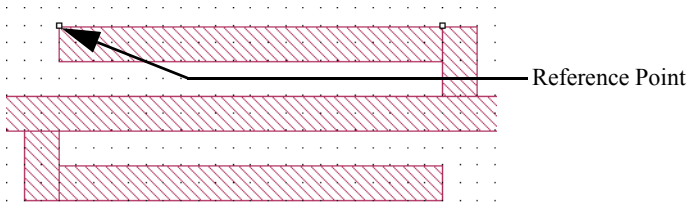
The anchor point is indicated by a small square which appears at the point you clicked. The next step is to select the reference point.



TIP

If you select the wrong point for either the anchor or reference point, press the Escape key to exit without adding a parameter. You may then start over.

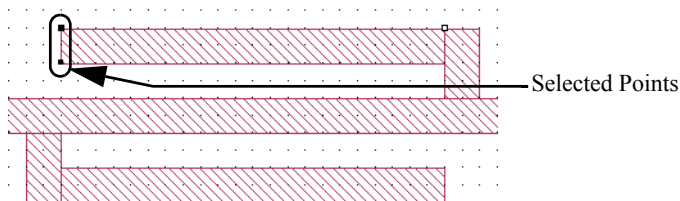
- 5 Click on the top left end of the top stub to add the reference point.**



The reference point is indicated by a small square which appears at the point you clicked.

In the next step, you select the rest of the adjustable point set. Points may be selected by clicking on individual points or by lassoing a set of points with your mouse. You do not need to select the reference point since it is automatically included in the adjustable point set.

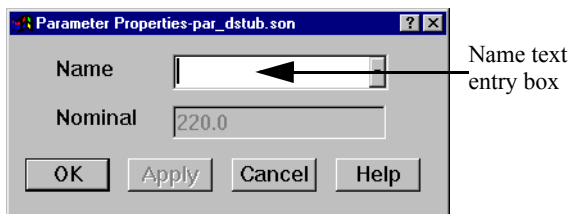
- 6 Drag the mouse until both points on the end of the stub are selected.**



These points will be added to the adjustable point set. When the reference point moves in response to a change in the parameter value, these points move relative to the reference point.

- 7 Once all the desired points are selected, press Enter.

This completes the parameter creation. The Parameter Properties dialog box appears on your display.

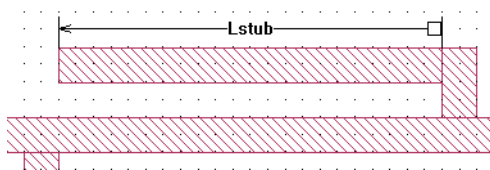


- 8 Enter the name "Lstub" in the Name text entry box in the Parameter Properties dialog box and click on the OK button.

This names the parameter. When you click on the OK button an arrow indicating the length and the name appear on your display.

- 9 Move the mouse until the name is positioned above the stub. When the name is in the desired position, click on the mouse.

The parameter should now appear as shown below.

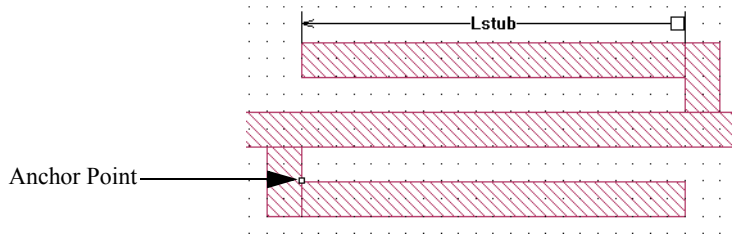


You will now enter the linked parameter. A linked parameter is another parameter identified in the circuit that has the same nominal value and name.

- 10 Select **Tools** ⇒ **Add Parameter** ⇒ **Add Anchored** from the project editor's main menu.

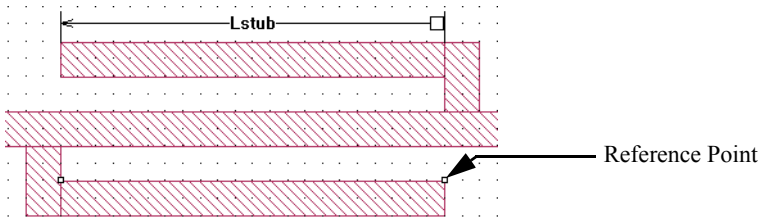
This places the project editor in Add an Anchored Parameter mode indicated by the change in cursor.

- 11 To specify the Anchor point for the parameter, click the mouse on the corner of the lower stub, as shown in the picture below.**



The anchor point is indicated by a small square which appears at the point you clicked. The next step is to select the reference point.

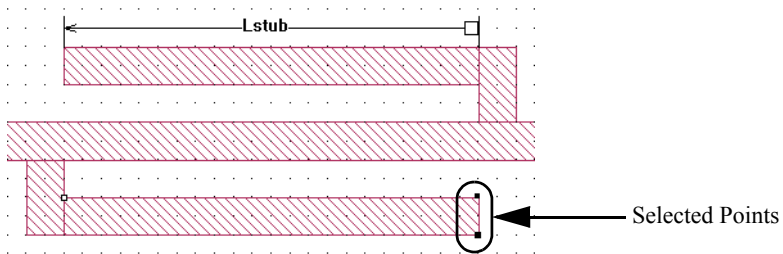
- 12 Click on the top right end of the bottom stub to add the reference point.**



The reference point is indicated by a small square which appears at the point you clicked.

In the next step, you select the rest of the adjustable point set. Points may be selected using any of the edit commands available in the project editor.

- 13 Drag the mouse until both points on the end of the stub are selected.**



These points will be added to the adjustable point set. When the reference point moves, these points move relative to the reference point.

- 14 Once all the desired points are selected, press Enter.**

This completes the parameter creation. The Parameter Properties dialog box appears on your display.

- 15 Enter the name “Lstub” in the Name text entry box in the Properties dialog box and click on the OK button.**

This names the parameter. When you click on the OK button an arrow indicating the length and the name appear on your display. Use your mouse to move the label to the desired location, then click.

Since this parameter’s nominal value is the same as the first parameter you defined, the project editor allows you to use the same name. These parameters are now linked. A change in value of one of the parameters changes both. If the parameters had been of a different length, you would have received an error message when you attempted to use the same name.

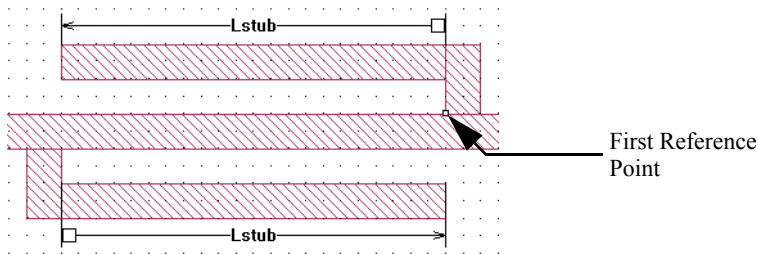
Next, you define the last parameter which is symmetric.

Symmetric Parameters

- 16** Select *Tools* \Rightarrow *Add Parameter* \Rightarrow *Add Symmetric* from the project editor's main menu.

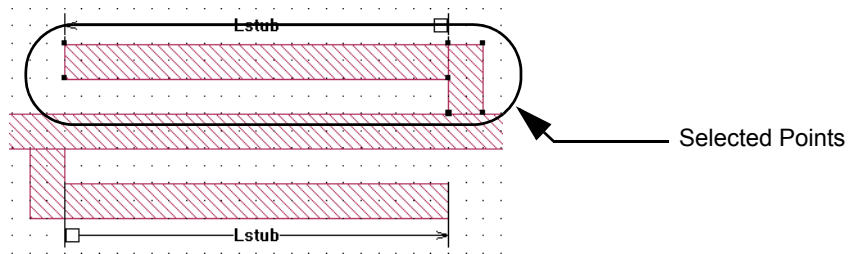
This places the project editor in Add a Symmetric Parameter mode indicated by the change in cursor. Note that the message “Click mouse to specify first reference point” appears in the status bar at the bottom of the project editor window. As you add a parameter, directions for each step appear in the status bar.

- 17** To specify the first reference point for the parameter, click the mouse on the intersection of the inside of the top stub to the transmission line, as shown in the picture below.



The first reference point is indicated by a small square which appears at the point you clicked. The next step is to select the point set you want attached to the first reference point.

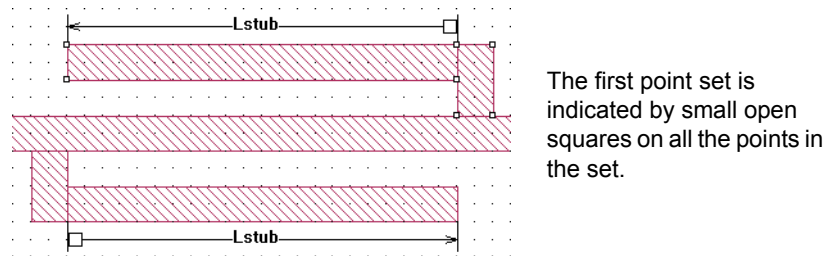
18 Drag the mouse until all points on the upper stub are selected.



These points will be added to the first adjustable point set. When the first reference point moves, these points move in the same direction and distance as the reference point.

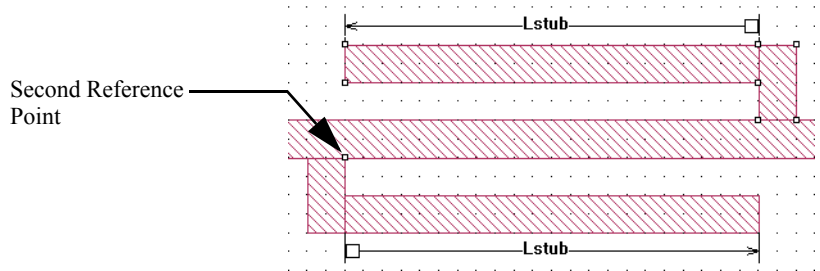
19 Once all the desired points are selected, press Enter.

This completes the first point set. Your circuit should look similar to this:



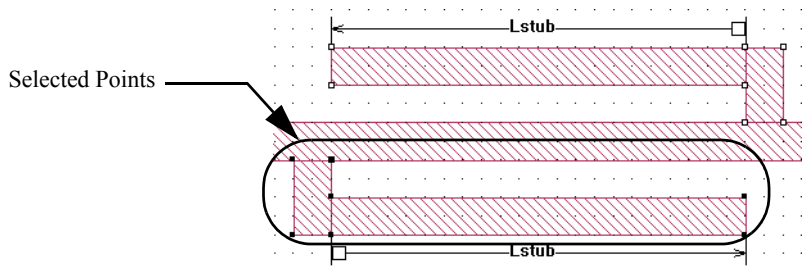
The message “Click Mouse to Specify Second Reference Point” appears in the status bar at the bottom of the project editor window. Next, you will specify the second reference point and its point set.

- 20 To specify the second reference point for the parameter, click the mouse on the intersection of the inside of the bottom stub to the transmission line, as shown in the picture below.**



The second reference point is indicated by a small square which appears at the point you clicked. Note that the first point set continues to be identified by small squares on all its points. The next step is to select the point set you want attached to the second reference point.

- 21 Drag the mouse until all points on the lower stub are selected.**



These points will be added to the second adjustable point set. When the second reference point moves, these points move in the same direction and distance as the reference point.

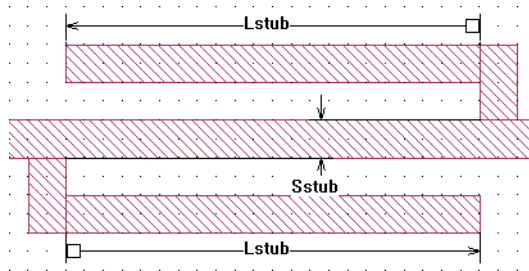
- 22 Once all the desired points are selected, press Enter.**

This completes the second point set and the symmetrical parameter. The Parameter Properties dialog box appears on your display.

- 23 Enter the name “Sstub” in the Name text entry box in the Parameter Properties dialog box and click on the OK button.**

This names the parameter. When you click on the OK button an arrow indicating the length and the name appear on your display. Note that since there is a difference between the reference points in both the x (horizontal) and y (vertical) direction, you may move the parameter name so that the parameter is defined in either the x or y direction.

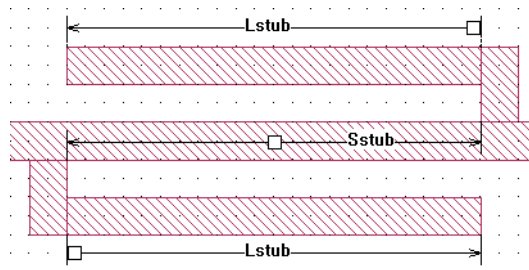
If you were to choose the y direction, moving the mouse to the left or right of both reference points to define your parameter, it would appear like this:



However, for this example, you define the parameter in the x direction, moving your mouse up or down, above or below both reference points.

- 24 Move the mouse until the name is positioned in the middle of the thru line. When the name is in the desired position, click on the mouse.**

This sets the parameter in the x direction. The parameter should now appear as shown below.



This completes entering the parameters. Note that Lstub is affected by Sstub. As Sstub increases, although it does not directly affect the value of Lstub, the two stubs do get further apart. Lstub is dependent on Sstub.

25 Select *File* ⇒ *Save* for the project editor main menu.

This saves the changes you have made to the circuit so that you can analyze it.

The next section of the tutorial teaches you how to setup and run a parameter sweep on the circuit.

Parameter Sweep

The parameter sweep uses only the Lstub variable. You analyze the circuit at two different lengths for Lstub over a frequency band of 2.0 GHz to 10.0 GHz. When the sweep is complete, you view the response curves in the response viewer.

For a detailed discussion of a parameter sweep, please refer to “Parameter Sweep,” on page 164 of the **Sonnet User’s Guide**.

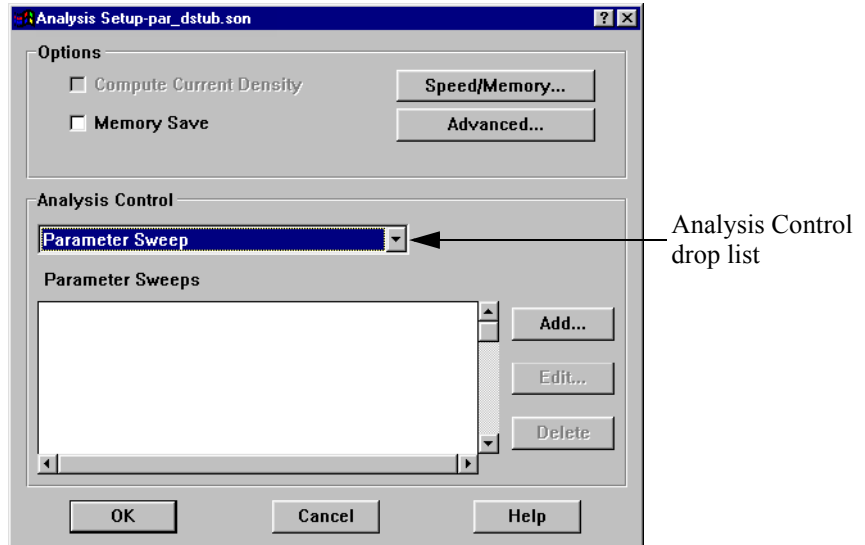
Setting Up a Parameter Sweep

1 Select *Analysis* ⇒ *Setup* from the project editor’s main menu.

The Analysis Setup dialog box appears on your display.

2 Select Parameter Sweep from the Analysis Control drop list.

This selects a parameter sweep as the type of analysis. The dialog box's appearance changes to accommodate the input needed for a parameter sweep. Notice that the Compute Current Density run option is disabled for a parameter sweep.



3 Click on the Add Button in the Analysis Control section of the dialog box.

The Parameter Sweep Entry dialog box appears. This dialog box allows you to add a parameter sweep.

The first step is to specify the analysis frequencies you wish to use for the parameter sweep. For this example, you wish to perform an ABS sweep from 2.0 GHz to 10.0 GHz. Since Adaptive Sweep (ABS) is the default sweep type, you do not need to take any action to select it.

4 Enter 2.0 in the Start text entry box in the Frequency Specification section of the Parameter Sweep Entry dialog box.

This is the starting frequency.

5 Enter 10.0 in the Stop text entry box.

This is the highest frequency. This defines a band of 2 GHz to 10 GHz for the adaptive sweep.

Next, you will select the parameters you wish to sweep.

6 Click on the checkbox in the Sweep column of the entry for the Lstub parameter.

Parameter Sweep Entry-par_dstub.son

Frequency Specification

Sweep Type: Adaptive Sweep (ABS)

Start [GHz]: 2.0 Stop [GHz]: 10.0

X-Cell Size : 10.0 (mils) Y-Cell Size : 10.0 (mils)

Sweep	Start	Stop	Step	Nominal
<input checked="" type="checkbox"/> Lstub	120.0	280.0	160.0	220.0
<input type="checkbox"/> Sstub	220.0	220.0		200.0

OK Cancel Help

It is possible to select multiple parameters for a parameter sweep; however, for this example, only one parameter is used. If you wished to deselect the parameter, you would simply click on the checkbox again. Unchecked parameters are simulated at their nominal value, so Sstub is a constant, fixed at 220 mils for the parameter sweep.

The nominal value, that is the present value of the parameter in the circuit, appears in the Nominal column of the parameter entry. In this case, the nominal value of Lstub is 220 mils, so the project editor shows the length as 220 mils, even though the start value for the parameter is different.

7 Enter 120 in the Start text entry box in the Lstub row.

This sets 120 mils as the first parameter value used for Lstub.

8 Enter 280 in the Stop text entry box in the Lstub row.

This sets 280 mils as the last parameter value used for Lstub.

9 Enter 160 in Step text entry box in the Lstub row.

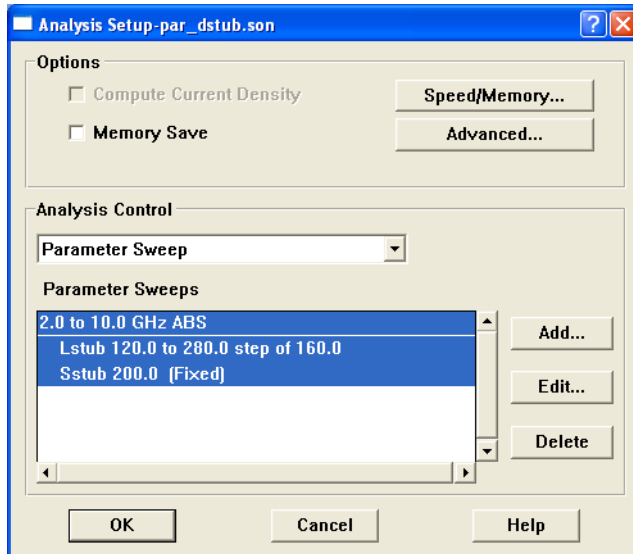
The interval between parameters is 160 mils; therefore, this parameter sweep analyzes at two parameter values, 120 and 280 mils. It is important to note that all three fields: start, stop, and step are required.

This completes the setup for the parameter sweep entry.

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

When the dialog box is closed, the Analysis Setup dialog box is updated with an entry for the parameter sweep that you just defined. In this case, since there are two values for a single parameter, there are two parameter combinations. Each combination is analyzed at each analysis frequency.

If you had a case in which there were two parameters, seven values for the first parameter and eleven values for the second parameter, there would be 77 parameter combinations for the analysis.



- 11 Click on the OK button of the Analysis Setup dialog box.

This completes the entry of the parameter sweep.

Next, you run the analysis and use the analysis monitor to observe the progress.

Executing the Parameter Sweep

- 12 Select *Project* ⇒ *Analyze* from the project editor's main menu to invoke the analysis engine, *em*, and start the analysis.

If you are prompted, save the file. The output window of the analysis monitor appears on your display.

13 Click on the Response Data button in the analysis monitor output window.

This allows you to observe the analysis as it progresses. There is a progress bar at the top of the window which shows what percentage of the total analysis is complete with the number of frequencies analyzed appearing above it. The response data is output in the bottom of the window.

The analysis could take a few minutes to run depending on your computer.

Once the analysis is complete, you open the response viewer to look at your results.

Observing the Parameter Sweep Data

You want to see the data for the S_{21} response at $L_{stub} = 120$ mils and $L_{stub} = 280$ mils.

14 Select *Project* \Rightarrow *View Response* \Rightarrow *New Graph* from the main menu of the analysis monitor output window.

The response viewer window appears on your display with S_{11} displayed.

15 Right click on `par_dstub` in the Curve Group legend.

A pop-up menu appears on your display.

16 Select *Edit Curve Group* from the pop-up menu.

The Edit Curve Group dialog box appears on your display.

17 Double click on `DB[S11]` in the Selected list.

This moves the S_{11} response to the Unselected list. It will no longer appear in your plot.

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

This moves the S_{21} response to the Selected list so that it appears in your plot.

19 Click on the *Select Combinations* button in the Edit Curve Group dialog box.

The Select Parameters dialog box appears on your display.

20 Click on the Select All button above the Selected Parameters list.

There should be only one entry in this list; the Select All button was used to demonstrate its function. This selects all the parameters in the Selected List.

21 Click on the Up Arrow button to move all the parameter combinations to the Unselected list.

22 Double-click on Sstub = 220 mils, Lstub = 120 mils in the Unselected List.

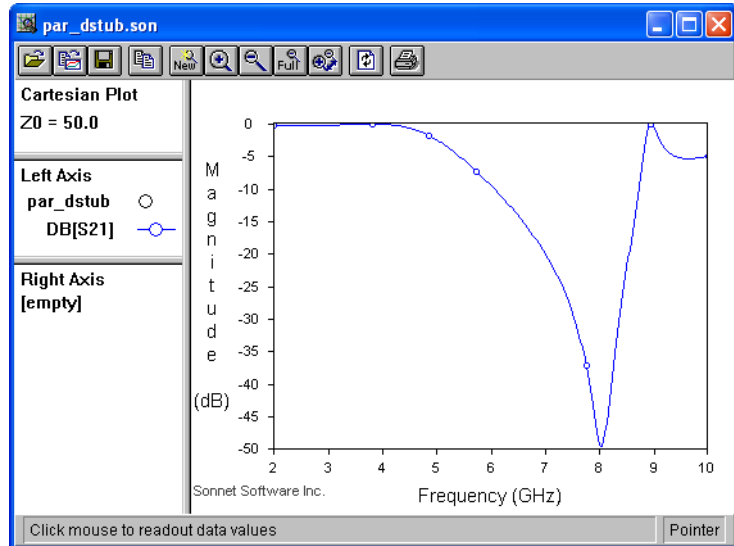
This moves this parameter combination to the selected list so that the data for this combination appears in your plot.

23 Click on the OK button in the Select Parameters dialog box to apply the changes and close the dialog box.

The entry “Lstub = 120.0 Sstub = 220.0” appears in the Parameter Combinations section of the Edit Curve Group dialog box.

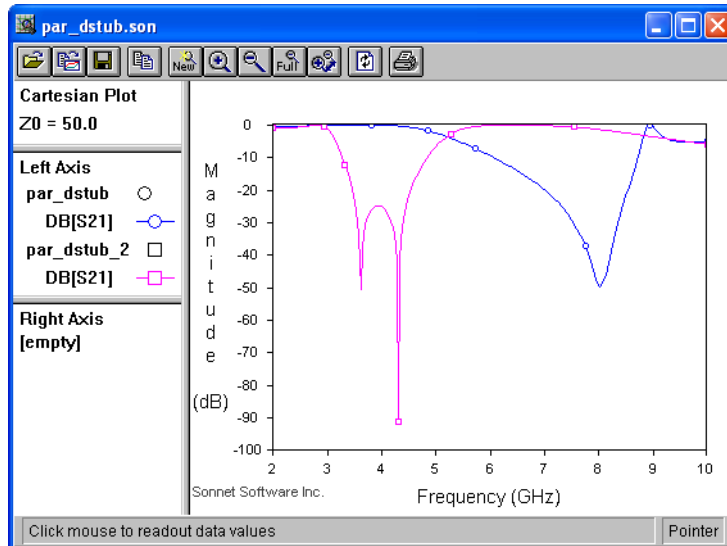
- 24 Click on the OK button in the Edit Curve Group dialog box to apply the changes and close the dialog box.**

The plot is updated with the S21 response for Sstub = 220 mils, Lstub = 120 mils. The entry for the curve group, par_dstub appears in the Curve Group Legend. The graph should be similar to the one shown below.



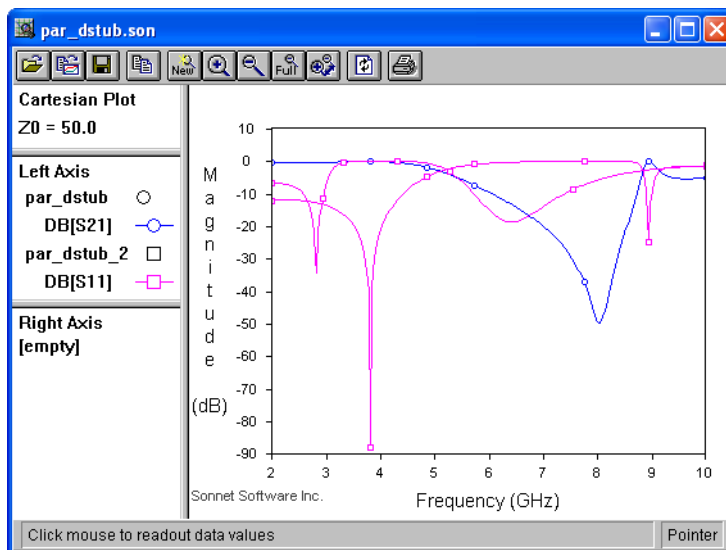
- 25 To add the response at Lstub = 280 mils, select *Curve* \Rightarrow *Add Curve Group* from the response viewer main menu.

The Add Curve Group dialog box appears. This curve group uses the default name of par_dstub_2. Following the same steps you used for par_dstub above, set up this curve group to display the S_{21} response for Lstub = 280 mils, Sstub = 220 mils. Your plot should now look like the one below.



You could also have right-clicked the curve group par_dstub in the Left Axis pane of the legend and selected “Edit Curve Group” from the pop-up menu. Using the Edit Curve group dialog box, you could have added this parameter combination to this curve group. This would result in one curve group with one symbol representing both parameter combinations. This is useful if you want multiple

measurements (S_{21} and S_{11} for example). Each measurement would use a different symbol, but each parameter combination with a measurement would use the same symbol. An example is pictured below.



In the beginning, the goal of the filter design was stated as a stopband between 5.0 and 6.0 GHz. By looking at the graph of Lstub=120 as compared to Lstub=280, you can see that a filter with the required stopband would fall approximately in the middle of the two curves. So a value of 220 mils is chosen for the nominal value for Lstub for the optimization. A nominal value of 220 mils is chosen for Sstub.

Optimization

This next section of the tutorial shows how to set up and execute an optimization.

For a detailed discussion of optimization, please refer to “Optimization,” on page 166 of the **Sonnet User’s Guide**.

If par_dstub.son is not still open in the project editor, open the file in the project editor.

Entering New Nominal Values

Usually for this type of circuit, you would optimize using both the defined parameters, Lstub and Sstub, but for the sake of processing time, the optimization only uses one parameter, Lstub.

The nominal value used for Sstub will be 200 mils. This was arrived at by actually executing the optimization using both parameters and using the closest value on the grid of the optimized parameter.

26 Double-click on the parameter Sstub in the circuit.

The Parameter Properties dialog box for Sstub appears on your display.

27 Change the nominal value in the Nominal text entry box to 200.

28 Click on the OK button to close the dialog box and apply the new nominal value.

The circuit is redrawn using the new nominal value for Sstub.



WARNING

It is recommended that you change the nominal value of parameters by using the properties dialog box, since changing your nominal value this way does not affect any previous response data in your project file. If you change the nominal value of the parameter by changing the circuit through editing commands, such as a reshape, all previous response data is deleted from the project file when you save.

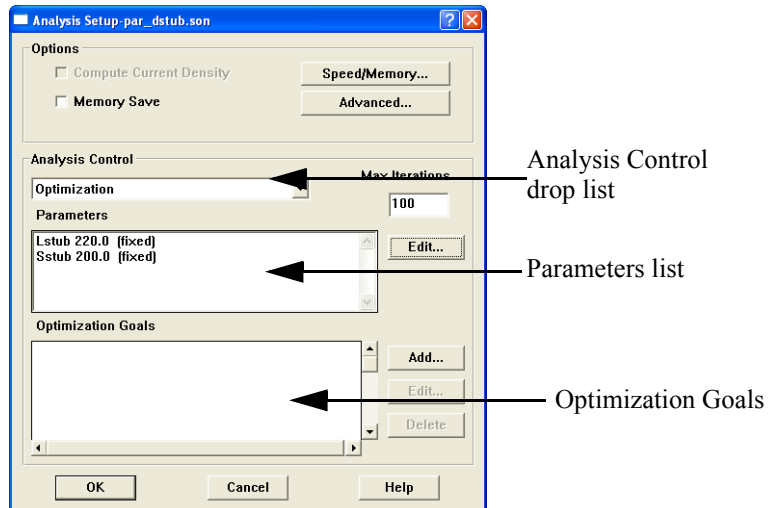
Setting Up an Optimization

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

The Analysis Setup dialog box appears on your display.

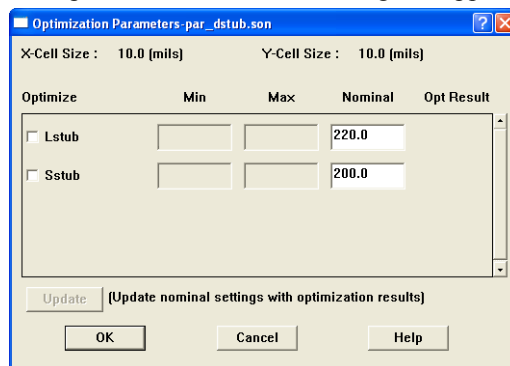
30 Select Optimization from the Analysis Control drop list.

This selects a optimization as the type of analysis. The dialog box's appearance changes to accommodate the input needed for an optimization. Note that initially the nominal values are listed for the parameters, since no range has yet been specified.



31 Click on the Edit button on the right side of the Parameters list.

The Optimization Parameters dialog box appears on your display.



Only one parameter, Lstub, is used for this optimization. The range for Lstub is 100 mils to 300 mils.

32 Click on the Optimize check button next to Lstub.

This selects the Lstub parameter to be used in the optimization. Note that the nominal value appears in the Nominal text entry box. If you wish to change the nominal value, you may do so by entering a new value. The circuit will be redrawn using the new nominal value. Since this is the first optimization run on this project file the Min and Max entries are blank. If a previous optimization had been run, the last entered values would remain.



WARNING

Editing parameter settings or optimization goals causes any pre-existing optimization iterations to be deleted from the project file.

33 Enter 100 in the Min text entry box in the Lstub row.

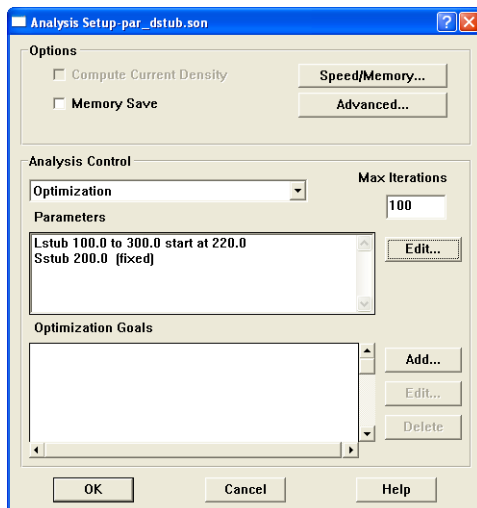
This sets the minimum value for the Lstub parameter to 100 mils for the optimization.

34 Enter 300 in the Max text entry box in the Lstub row.

This sets the maximum value for Lstub to 300 mils for the optimization.

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

When the dialog box is closed, the Analysis Setup dialog box is updated with an entry for the optimization parameters that you just defined.

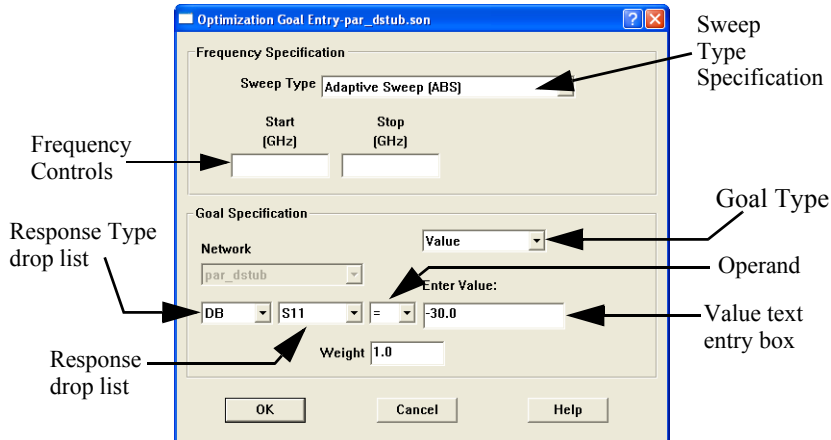


Now that you have identified which parameter to vary and its range, you must specify the optimization goals. Since this is the first optimization for this project, there are no previously defined optimization goals and the list is empty. Having no goals present disables the Edit and Delete buttons. The Edit button allows you to modify an existing goal, and the delete button removes the goal from the list.

As mentioned earlier in the example, our goal for the filter is to have passbands at 1-4 GHz and 7-10 GHz with a stopband at 5-6 GHz. The optimization goals are set up accordingly. Since all three goals in this case are equally important, each uses the default Weight of 1.0. In cases where one goal is more essential, assigning it a higher weight than other goals tells *em* to concentrate more on reaching that particular goal.

36 Click on the Add button to the right of the Optimization Goals list.

The Optimization Goal Entry dialog box appears on your display.



The first goal corresponds to the first passband; therefore, you need to set the ABS frequency range to 1.0 GHz to 4.0 GHz. There are several different types of frequency sweeps available; you will use the default ABS sweep for this optimization.

37 Enter 1.0 in the Start text entry box in the Frequency Specification section of the dialog box.

This sets the beginning of the frequency band for this optimization goal at 1.0 GHz.

38 Enter 4.0 in the Stop text entry box in the Frequency Specification section of the dialog box.

This sets the end of the frequency band for this optimization goal at 4.0 GHz. This completes the specification of the frequency sweep for this optimization goal.

Since this is the first passband, your goal is to have DB[S21] be greater than -1.0 dB.

39 Select DB from the Response Type drop list on the left side of the equation.

This is the default, so DB may already be selected.

40 Select S21 from the Response drop list.

41 Select “>” from the Operand drop list.

42 Select Value from the Goal Type drop list.

This choice allows you to put in a specific value. This is the default; you may also specify another project file or another network in your project (if the project is a Network project). In those cases, you may select a response for that circuit to which you wish to match your selected response.

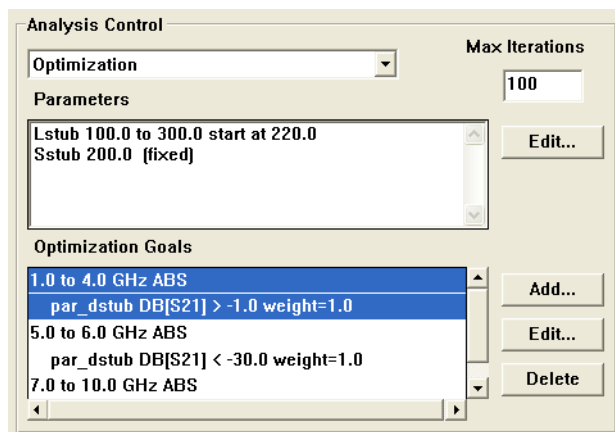
43 Enter -1.0 in the Value text entry box.

This sets your goal of $\text{DB}[S21] > -1.0 \text{ dB}$.

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

The Analysis Setup dialog box is updated. An entry for this optimization goal now appears in the Optimization Goals list.

The other two goals should be entered in a similar manner. The second goal is a adaptive sweep from 5.0 GHz to 6.0 GHz with a desired response of $\text{DB}[S21] < -30.0 \text{ dB}$. This is the stopband. The third goal is an adaptive sweep from 7.0 GHz to 10.0 GHz with a desired response of $\text{DB}[S21] > -1.0$. When you have completed entering these goals, the Optimization Goals list should appear as shown below.



This completes the setup for the optimization.

Running an Optimization

This optimization took approximately 6.5 minutes on a 2 GHz Pentium 4.

- 45 Select *Project* ⇒ *Analyze* from the main menu of the project editor.**

The output window of the analysis monitor appears on your display.



TIP

You can also click on the Analyze button on the project editor tool bar.

- 46 Click on the Response Data button in the analysis monitor output window if it is not already displaying the response data.**

This allows you to observe the optimization as it progresses. The error for the first, present and best iteration are displayed and updated as the optimization progresses. The response data is output in the bottom of the window.

You may notice that some iterations complete more quickly than others. This is because *em* can reuse portions of the response data calculated for previous iterations.

Once the analysis is complete, you open the response viewer to look at your results. Be aware that since this optimization took a number of iterations to conclude, there may be small delays in opening the response viewer and the Edit Curve Group dialog box.



TIP

You may use the response viewer to observe data produced by the optimization while it is still running. The response viewer can chart any data which has been generated. This allows you to stop the optimization and start it over using new parameter ranges or new goals, if the results are not desirable. If you wish to continue viewing data as the optimization runs, use the Freshen Files command in the Response Viewer.

Observing your Optimization Data

Plotting the best iteration will allow you to judge whether or not to use the optimized values of the parameters.

- 47 Select *Project* ⇒ *View Response* ⇒ *New Graph* from the main menu of the analysis monitor.**

The response viewer menu appears on your display with the DB[S11] response for the nominal values parameter combination displayed. There may be some delay while the project loads into the response viewer due to the amount of response data now included.

- 48 Click on the DB[S11] curve group in the legend to select it and select *Curve* ⇒ *Edit Curve Group* from the main menu of the response viewer.**

The Edit Curve Group dialog box appears on your display.

- 49 Select *Optimized* from the Data Collection drop list in the Edit Curve Group dialog box.**

Notice that this drop list is now enabled since the project file now contains both parameter sweep data and optimized data.

- 50 Move DB[S11] to the Unselected list by double-clicking on the entry.**

- 51 Move DB[S21] to the Selected list by double-clicking on the entry.**

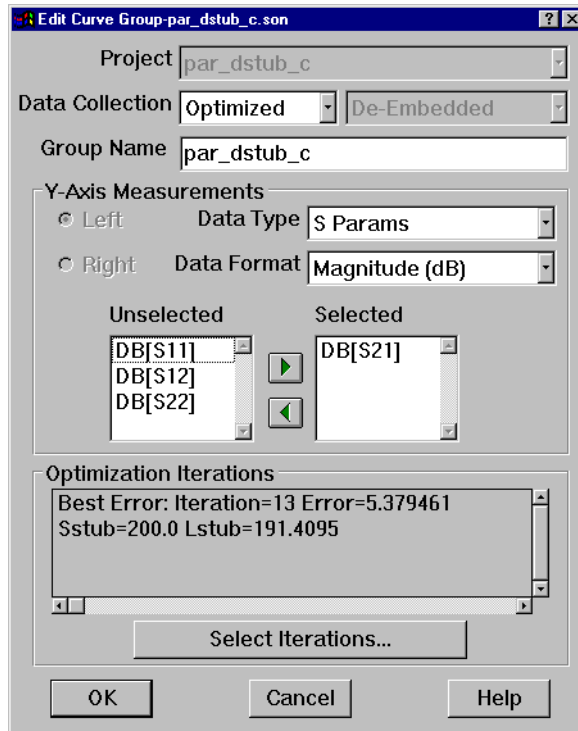
Since your optimization goals were set in reference to the DB[S21] response, you want to plot this response.

Notice that when you selected Optimized data that the parameter combination was updated to read:

**Best Error: Iteration=12 Error=0.096573
Lstub=191.8066 Sstub=200.0**

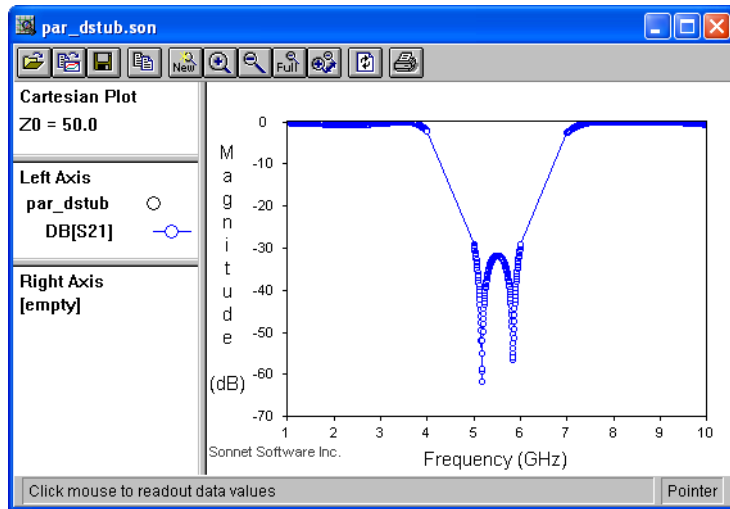
The best iteration is plotted by default when you select optimized data. If the optimization was still running, this is a useful way of always plotting the best iteration calculated so far. Pressing the Freshen Files button on the tool bar of the response viewer will always show the best iteration.

If you were to click on the Select Iterations button, the dialog box would appear with all the parameter values used in all 25 iterations available to plot. Since you wish to see the response at the best iteration, you do not need to change the parameter value for this curve group.



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

The plot is updated showing DB[S21] for the best iteration. It should appear similar to the plot pictured below.



As you can see, the optimized circuit is producing the desired response of a stopband from 5 - 6 GHz and the passbands from 1 - 4 GHz and 7-10 GHz.

Accepting the Optimized Values

Since the desired responses have been achieved by the optimization, you return to the project editor to update the nominal value of your parameters with the optimized values.

If the project `par_dstub` is still open in the project editor, continue at step 55.

- 53 Click on the curve group, `par_dstub` in the Curve Group legend in the response viewer.**

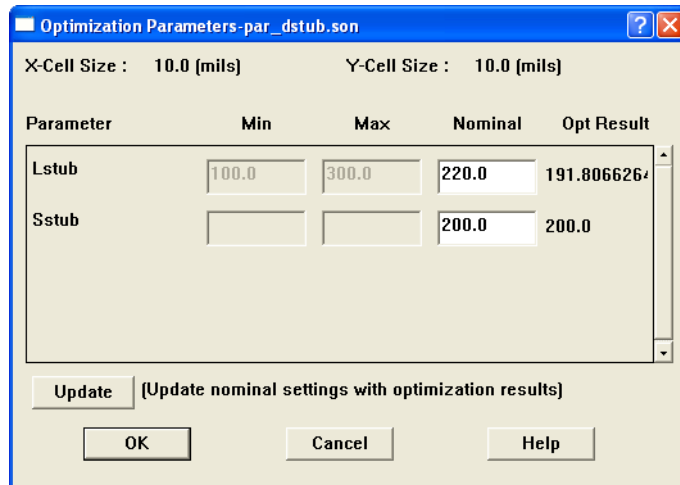
This selects a project file and thereby enables the project menu.

- 54 Select *Project* ⇒ *Edit* from the main menu of the response viewer.**

The project editor appears on your display with `par_dstub` open.

55 Select *Analysis* ⇒ *Optimization Results* from project editor main menu.

The Optimization Parameters dialog box appears on your display.



Notice that the nominal value of both parameters is still the value input at the beginning of the optimization. But the optimization results for both parameters are displayed in the last column. Since Sstub was not used in the optimization, its optimization result is the same as the nominal value.

56 Click on the Update button to replace the present nominal values with the optimization results.

Note that the entries in the Nominal text entry boxes are updated with the optimized values.

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

Notice that the circuit has been redrawn with the new nominal values for the parameters. Since the parameter lengths are not integer multiples of the cell size, the polygons are no longer exactly on the grid. You can see this by pressing Ctrl-M to turn off the cell fill and looking at the actual polygons. The cell fill represents the actual metal *em* analyzes.

The actual metalization analyzed by *em* is not the same as the optimized values. *Em* actually interpolated from data created from analyzing “on grid” versions of the circuit. If your optimization goals did not include a full frequency sweep, it is a good idea to perform a full sweep across your frequency band to ensure that your entire band shows reasonable results. Running a full frequency sweep is detailed below.

In order for *em* to use previously calculated response data, you should edit your parameter value(s) such that they are the closest “on grid” value to the optimization result. For example, in this case the optimized value for Lstub is 191.80662641. You should edit the nominal value of Lstub to change it to 190 which, since the cell size is 10, is on the grid.

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

The Analysis Setup dialog box appears on your display.

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

The appearance of the dialog box is changed to conform with an adaptive sweep. Enter 2.0 GHz in the Start text entry box and 10 GHz in the Stop text entry box to define a frequency band of 2 - 10 GHz for this analysis.

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

61 Click on the Analyze button to start the *em* analysis.

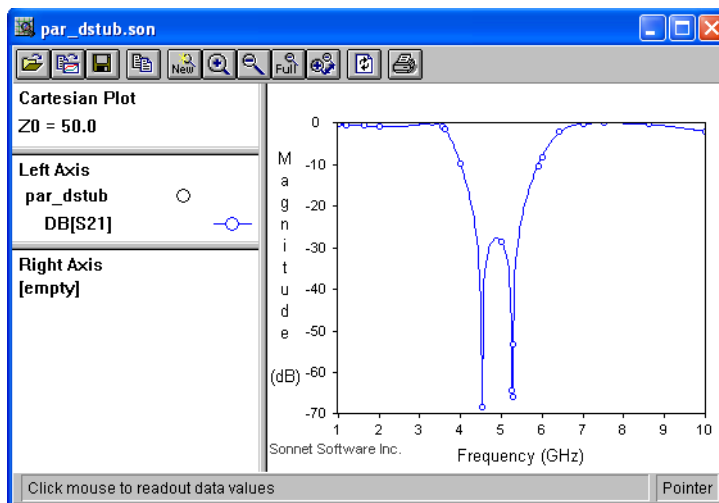
If prompted, save the circuit before analyzing.

62 When the analysis is complete, click on the View Response button on the analysis monitor’s tool bar.

The response viewer appears on your display with the curve group, par_dstub, consisting of DB[S11] displayed.

- 63 Right-click on the curve group, par_dstub, and select Edit Curve Group from the pop-up menu which appears.**

Select DB[S21] for display, move DB[S11] to the Unselected list, and close the dialog box. Your plot should appear similar to that shown below.



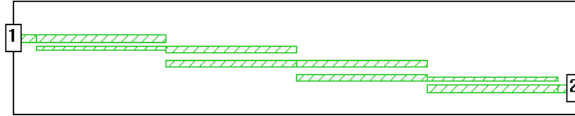
The results conform closely enough to the design criteria that this optimization is considered a success. As was stressed at the beginning of this tutorial, a simple example was chosen in order to clearly demonstrate the optimization process.

You should be aware, however, that most optimization problems are much more complicated and less straightforward. The designer needs to make decisions about parameters, the parameter sweeps and optimization goals based on knowledge of the circuit and design experience. Often, you must observe an optimization while in progress, judge its viability and, as necessary, stop the optimization and start a new one with new nominal values and data ranges.

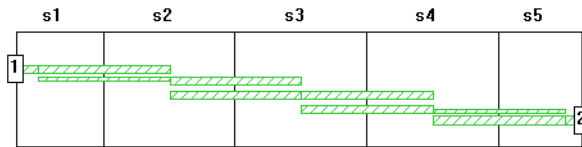
Chapter 3 Circuit Subdivision Tutorial

This tutorial walks you through how to add subdivision lines, subdivide your circuit, and analyze the final netlist. The results of this subdivision are compared to the analysis of the complete circuit in order to demonstrate the accuracy of the results of the subdivision and the savings in memory. For a detailed discussion of circuit subdivision and the use of subdividers, please refer to Chapter 13, “Circuit Subdivision” in the **Sonnet User’s Guide**.

The circuit, an edge-coupled microstrip bandpass filter, is a fairly simple example of a circuit which you might decide to subdivide. In addition, it is not a very good filter design. This circuit was chosen for the purposes of clarity in explaining circuit subdivision.



You will use four vertical subdivision lines to split the circuit into five sections as shown below.

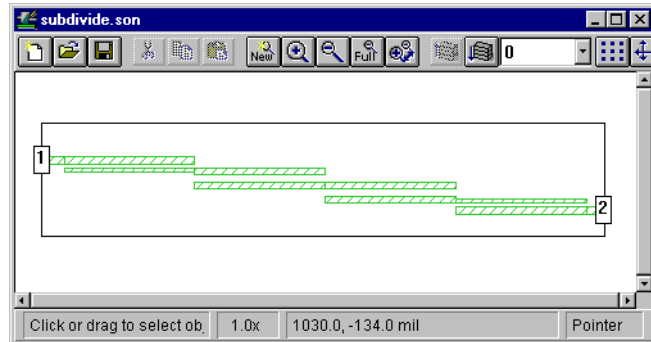


Obtaining the Example File

You use the example file, [subdivide.son](#), for this example. You can obtain a copy of this file from the Sonnet Examples. If you do not know how to obtain a Sonnet example, select *Help* \Rightarrow *Examples* from any program menu, then click on the **Instructions** button. If you are reading this in PDF format, click on the link above.

1 Open the project `subdivide.son` in the project editor.

The circuit appears as shown below.



Adding the Subdivision Lines

The first step in subdividing a circuit, as discussed in “Choosing Subdivision Line Placement,” on page 207 in the **Sonnet User’s Guide**, is to place the subdivision lines that indicate where you wish to split your circuit. Subdivision lines should be placed in locations where there is negligible coupling across the lines. The best place to put subdivision lines in the example used here is at points in the circuit on the coupled lines as far from the discontinuities as possible. Therefore, a vertical subdivision line will be placed in the middle of each coupled pair of polygons.

Each coupled pair of polygons is 595 mils in the x direction. Subdivision lines must be placed on the grid. The closest value to halfway which still remains on the grid is 295 mils. For the first subdivider, you must take into account the feedline polygon which is 100 mils in length. Therefore, the first subdivision line should be placed at $100 \text{ mils} + 295 \text{ mils} = 395 \text{ mils}$ from the left box wall.

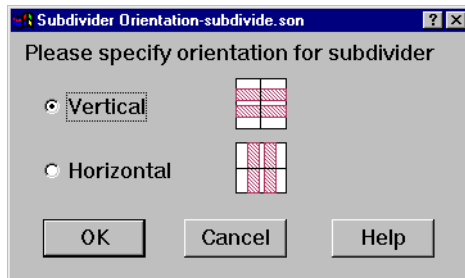
- 2 **Select *Tools* ⇒ *Add Subdivider* from the project editor menu while holding down the shift key.**



TIP

Holding down the shift key allows you to enter multiple subdivision lines without having to select the command multiple times.

Since there were no subdivision lines in the circuit when you selected the Add Subdivider command, the Subdivider Orientation dialog box appears on your display.



All subdividers in your circuit must have either a vertical (up-down) orientation or a horizontal (left-right) orientation on the substrate.

- 3 **Click on the vertical radio button to select the vertical orientation for your subdividers.**

This sets the orientation for all subdividers subsequently added to your circuit.

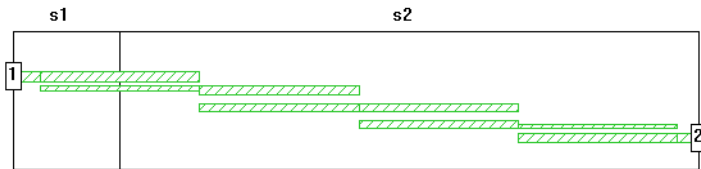
This dialog box does not appear again if you select *Tools* ⇒ *Add Subdivider*. The new subdivider assumes the same orientation. If all the subdividers are deleted from a circuit, then when the Add Subdivider command is used again, this dialog box appears.

- 4 **Click on the OK button to apply your selections and close the dialog box.**

The cursor changes to indicate that you are adding subdivision lines and a line appears which moves with your cursor.

- 5 Move your cursor until the X coordinate of the cursor position in the status bar is 395.0 and click.**

A line representing the subdivider appears in the vertical plane running through the point at which you clicked. The sections of the circuit are now labeled “s1” and “s2”. Subdivision sections are labeled from left to right, or top to bottom, depending upon orientation. These labels are always sequential and are non-editable.

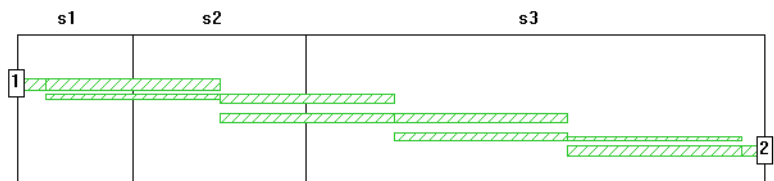


Subdivision lines are always snapped to the grid and may not be placed on top of each other. Once a subdivider has been added to your circuit, you may edit the subdivider as you would any other object in your geometry. You may click on the subdivider and move it. You may also control the display of the subdivider lines and labels in the Object Visibility dialog box, invoked by selecting *View* \Rightarrow *Object Visibility* from the project editor's main menu.

Since each of the coupled line segments are 595 mils long and you wish to place the subdivision lines at the halfway point, each subsequent subdivision line should be placed 595 mils further to the right in the circuit. So the second subdivision line should be placed at 990 mils from the left box wall.

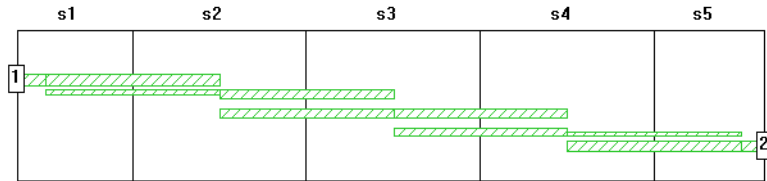
- 6 Move the cursor until the X coordinate is 990.0 in the status bar and click to place the second subdivision line.**

The subdivision line appears on your circuit and the sections are relabeled as shown below.



7 Add subdividers at 1585 mils and 2180 mils from the left box wall.

Once you have completed adding all the subdivision lines, press the Escape key to return to pointer mode. Your circuit should now appear like this:



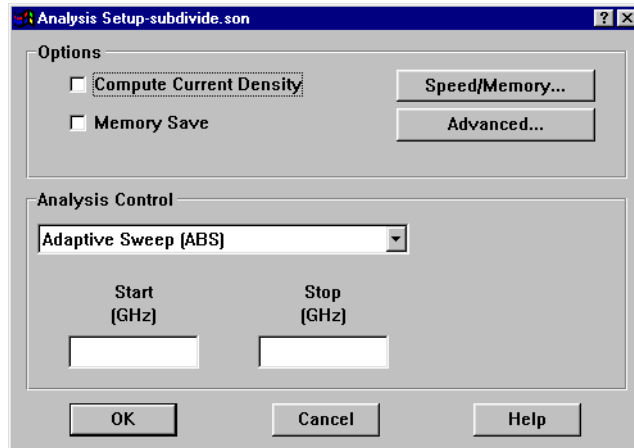
Setting Up Circuit Properties

For this example, the circuit properties such as box size, dielectric layers, metal materials, etc. have already been input in the example circuit. It is important to have the circuit properties input before performing the subdivide since these are the properties used for all the subprojects created as the result of the subdivide. If you do not enter all the desired properties, you will need to enter them individually in each subproject or modify the original source project and execute the subdivide again.

For this example, you will analyze the netlist using an adaptive sweep (ABS) with Hierarchy Sweep turned on. When the Hierarchy Sweep option is used, the analysis control settings for the netlist are used to analyze all the subprojects in the netlist. The desired frequency band for the circuit is 2.3 GHz to 2.5 GHz. An adaptive sweep provides approximately 300 data points. For more information on the Adaptive Band Synthesis technique, see Chapter 9, “Adaptive Band Synthesis (ABS)” in the **Sonnet User’s Guide**.

- 8 Select *Analysis* \Rightarrow *Setup* from the main menu.

The Analysis Setup dialog box appears on your display.

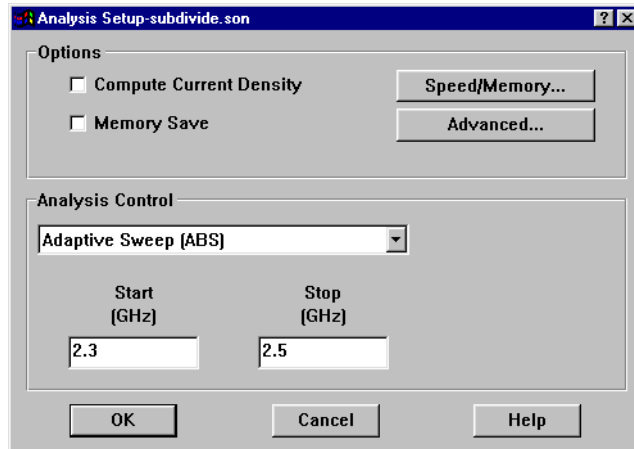


- 9 Select “Adaptive Sweep (ABS)” from the Analysis Control drop list if it is not already selected.

This selects the Adaptive Sweep as your type of analysis. The adaptive sweep provides a fine resolution of response data over the given frequency band. Note that the text entry boxes are updated to reflect your choice of analysis.

10 Enter 2.3 in the Start box and 2.5 in the Stop box.

This sets up the analysis frequency band. This analysis setup is duplicated in all of the geometry subprojects when the subdivide is executed, as well as in the main netlist. The Analysis Setup dialog box should appear as shown below.



11 Click on the OK button to save the analysis setup and close the dialog box.

12 Select *File* ⇒ *Save* from the main menu.

The file must be saved before executing the subdivide. The position of the subdivision lines are saved as part of your source project.

Subdividing Your Circuit

The actual subdivision of the project is executed by the software but you must enter names for the resulting main netlist file and subproject files produced as well as, optionally, defining a feedline length to be added to the subprojects.

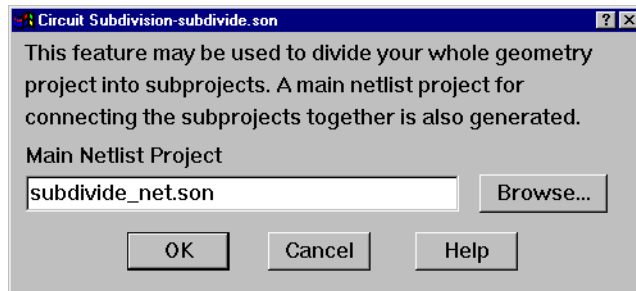
Feedlines should be added to the subprojects if you feel it necessary to move discontinuities in the various sections of the circuit further away from the boxwalls to prevent any interaction between the discontinuities and boxwalls.

This can provide a more accurate analysis result for each section of the circuit. Any added feedlines are of lossless metal, regardless of the metal type to which they are attached.

Sonnet software provides a default recommended value for the feedline or you may enter your own value.

13 Select *Tools* ⇒ *Subdivide Circuit* from the project editor main menu.

The Circuit Subdivision dialog box appears on your display.



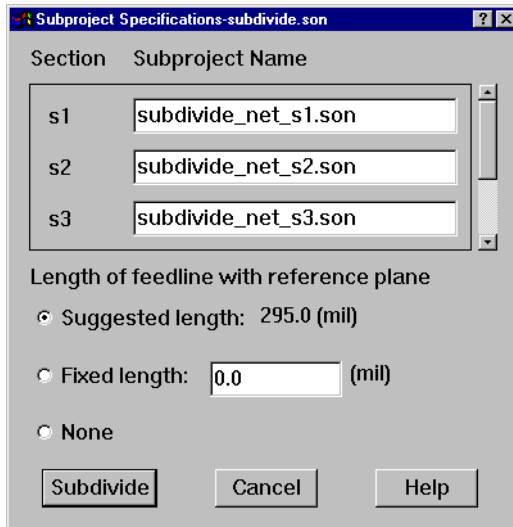
14 The name “subdivide_net.son” is provided by default in the Main Netlist Project text entry box.

This name is used for the main netlist which connects the geometry projects resulting from the subdivide. The default name is the basename of the source project with a “_net” added on. You may use any project name you wish but it must be different than the project name of the source file.

If you wish to change the directory in which the resulting files are created, click on the Browse button to open a browse window. If you select an existing project file, you are prompted if you wish to overwrite the existing file.

15 Click on OK to set the name and close the dialog box.

The Subproject Specifications dialog box appears on your display as shown below. This dialog box allows you to enter names for each of the geometry subprojects that result from performing the subdivide. Default names, consisting of the main netlist project name with the section number added, are provided but may be edited. For this example, use the default names.



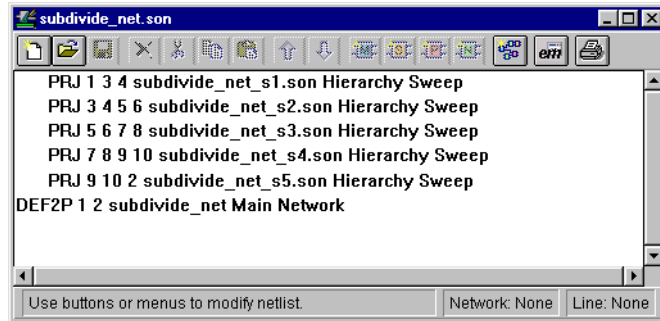
The names for the subprojects must be unique and must be different from the source project name and main netlist name.

The suggested length option is already selected for the feedline length. This feedline of lossless metal is added to ports generated when the subdivide is executed.

To enter your own feedline length, you would select the fixed length radio button and enter the value in the corresponding text entry box. Select the None radio button if you do not wish to add a feedline.

16 Click on the Subdivide button to execute the subdivide.

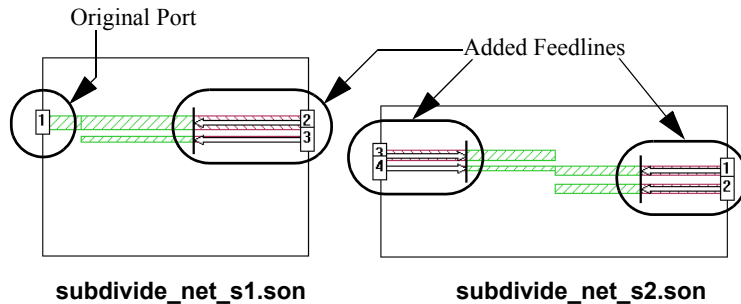
The main netlist and subprojects are created using the names input by you. The main netlist project is opened in the project editor.



The main network is defined as subdivide_net and has two ports. This corresponds to the source circuit. There is a project (PRJ) entry line for each of the subprojects. The project line includes the setting for the source of the analysis frequencies. A Hierarchy sweep, in which the netlist frequency sweep is imposed on all the project elements, is on by default. If you turn this off, the project default setting of using its own sweep is displayed.

Pictured below are the geometries for the first two sections, subdivide_net_s1.son and subdivide_net_s2.son. Note that in subdivide_net_s1.son, feedlines with a reference plane have only been added to ports 2 and 3, the ports created in the

subdivide, but not port 1 which is contained in the source project. All the ports in `subdivide_net_s2` have feedlines since all were created in the subdivide. Note that the feedlines are all of lossless metal.



Analysis of the Network File

The last step to complete the analysis of the filter is to analyze the netlist project created by the subdivide. The analysis controls you entered in the original project are the ones you wish to use to analyze the netlist, so the analysis setup is already complete. An adaptive sweep from 2.3 GHz to 2.5 GHz will be performed on the netlist.

- 17 **Click on the project editor window containing the netlist to make this the active file.**

This is indicated by the title bar on the netlist being highlighted.



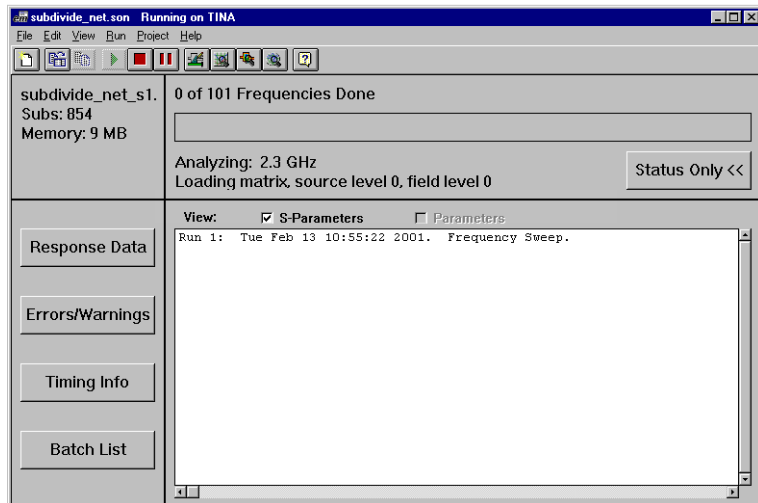
TIP

You can switch the active file in the project editor by clicking on the title bar of the project window or by selecting the project from the Windows menu on the main menu.



18 Click on the Analyze button to launch the netlist analysis.

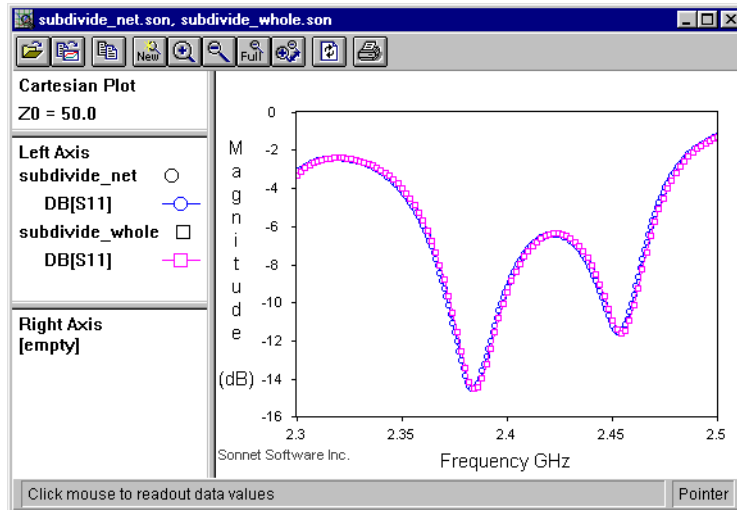
The analysis monitor appears on your display.



The project legend indicates that `subdivide_net_s1.son` is being analyzed. *Em* will perform an adaptive sweep on each of the five subprojects and then use the resulting data to analyze the network. Status messages are output under the progress bar.

There are two results that are significant to observe. A comparison of the netlist analysis data with the analysis data from the source circuit, and a comparison of the amount of time and memory each analysis used. We have provided the source project file including analysis data under the example [sub_whole.son](#) available in the Sonnet Examples.

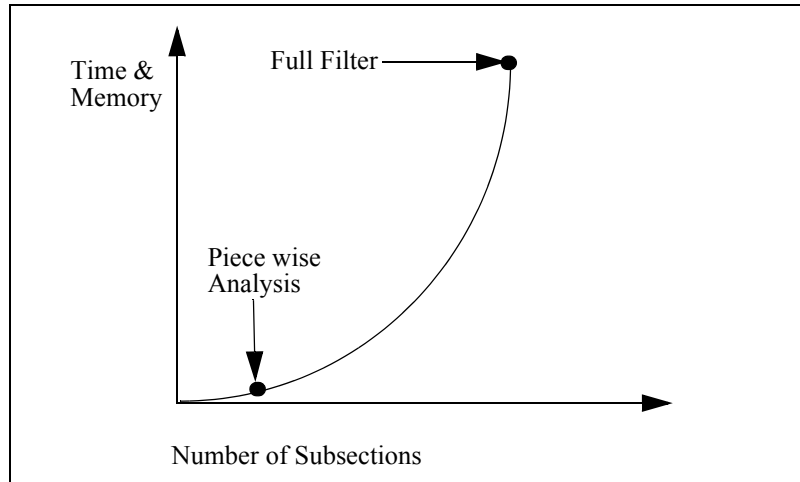
The graph below shows the results of the netlist analysis versus the results of a full analysis of the source project.



As you can see there is very good agreement between the two analysis results. Both files were analyzed on the same computer. The time required for the netlist was actually longer than the time required to analyze the circuit as a whole because this was a simple example chosen for clarity, and the benefits of circuit subdivision are only seen for larger circuits.

Using circuit subdivision reduces your memory requirements for analysis of a large circuit. Each of the subprojects requires less subsections to analyze than the complete circuit. This improvement comes as a result of reducing the number of subsections for any given analysis since both computation time and memory requirements rise sharply as the subsections go up, as shown on the chart below.

For this example, the entire filter circuit used 2006 subsections while the largest individual piece only required 1400 subsections and the smallest only required 854 subsections.



On many larger circuits the use of the automatic circuit subdivision features in Sonnet can greatly improve the efficiency of your *em* usage.

Additional Improvements

There are two other ways this circuit could have been made even more efficient. You could have refrained from adding the automatic feedlines and you could have taken advantage of the fact that some of the subprojects were virtually identical.

For the purpose of illustration, this tutorial added feedlines to all ports generated in the subdivide using the recommended length. Feedlines are added to a circuit to move the discontinuities in the subprojects far enough from the boxwalls to prevent interaction. In the case of this example, either discontinuities were not present or they were already far enough from the box wall that additional feedlines were unnecessary.

If you leave the feedlines out by selecting None in the Subprojects Specifications dialog box, the netlist analysis runs 1.5X faster than previously.

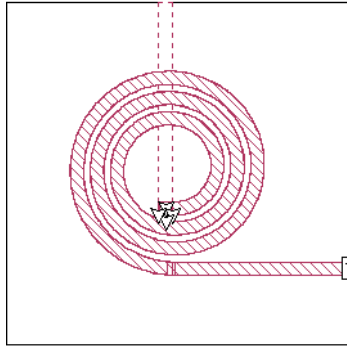
The last method that would allow you to decrease the processing time would be to use fewer subprojects in the netlist to create the circuit. Observation of the circuit geometry and response data shows that subdivide_net_s1.son and subdivide_net_s5.son are virtually identical. The same is true for subdivide_net_s2.son and subdivide_net_s4.son. You could edit the main netlist, subdivide_net.son so that you only use three files: subdivide_s1.son, subdivide_net_s2.son and subdivide_net_s3.son to create the whole circuit. This eliminates the need to calculate data for two out of five subprojects. This analysis is 2X faster than the analysis using feedlines and all five subprojects.

Chapter 4 Conformal Mesh Tutorial

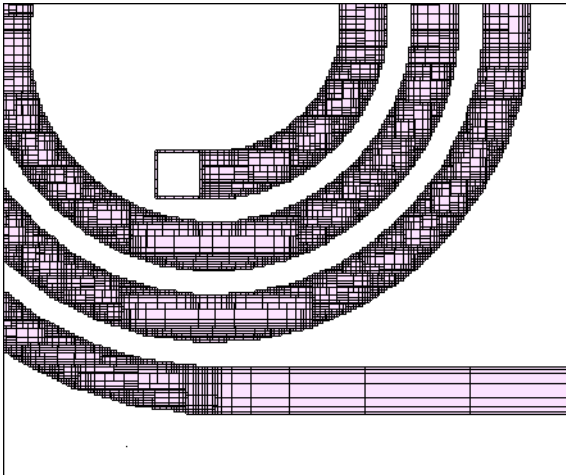
We recommend reading Chapter 11 "Conformal Mesh" on page 169 of the **Sonnet User's Guide** for a background discussion of conformal mesh before performing this tutorial.

You use the example file, [cm_spiral.son](#), pictured below, for this example. You can obtain a copy of this file from the Sonnet Examples. If you do not know how to obtain a Sonnet example, select *Help* \Rightarrow *Examples* from any program menu, then click on the **Instructions** button. If you are reading this in PDF format, click on the link above.

Note that the spiral was added to the circuit using the palette of standard geometries (*Tools* \Rightarrow *Add Metalization* \Rightarrow *Round Spiral*).



Sonnet uses staircase fill by default. Using staircase fill, the memory requirement for this circuit is approximately 7000 Mbytes using 30,000 subsections. This analysis would be impossible or prohibitively time consuming to run on most computers. In this tutorial, you will use conformal meshing where appropriate to reduce the number of subsections so that the circuit uses around 2,800 subsections and requires 69 Mbytes of memory.

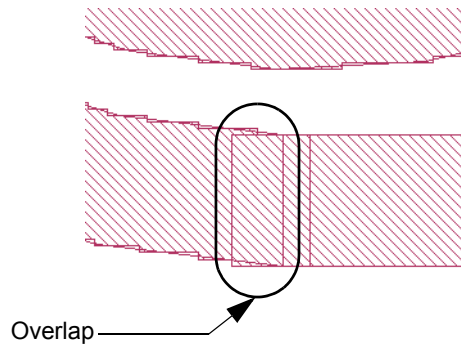


This is a close-up of the circuit which shows the high number of rectangular subsections needed to subsection the curves of the spiral. Note that the feedline is a Manhattan polygon and therefore uses fewer, larger subsections.

As discussed in the rules list in Chapter 11, “Conformal Mesh,” in the **Sonnet User’s Guide**, we want to use conformal meshing on non-manhattan polygons while continuing to use staircase fill on large rectangular polygons. Our spiral consists of three polygons: the three turn spiral inductor and two feed lines. One feed line is on the same level as the spiral and the other, connected to the spiral by a via, is one level down.

No Polygon Overlap

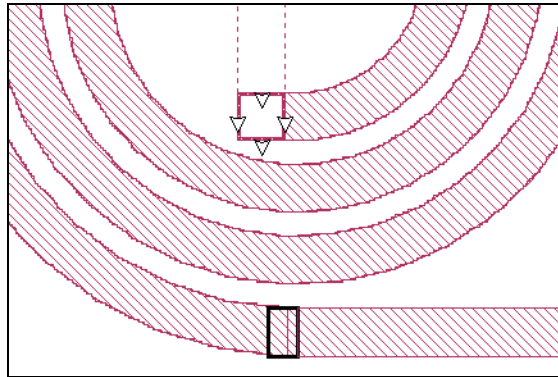
Rule 1; “Polygon Overlap” in the **Sonnet User’s Guide** states that there should be no polygon overlap in a circuit which uses conformal mesh. If you zoom in on the boundary between the spiral conductor and the feedline on level 0, you will observe, as shown below, that the two polygons overlap.



Using the Merge Polygon command is an easy way to remove the overlap from your circuit. To do so, perform the following:

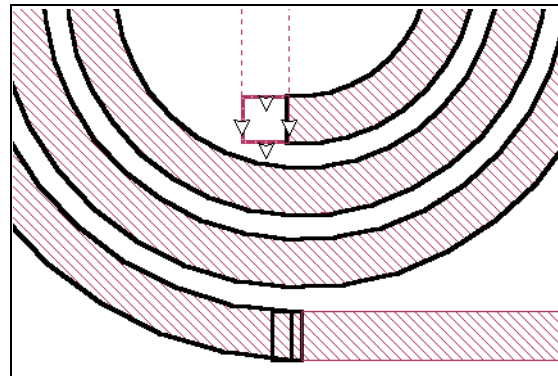
- 1 Click on the small rectangular polygon which overlaps the conductor to select it.

The polygon is highlighted.



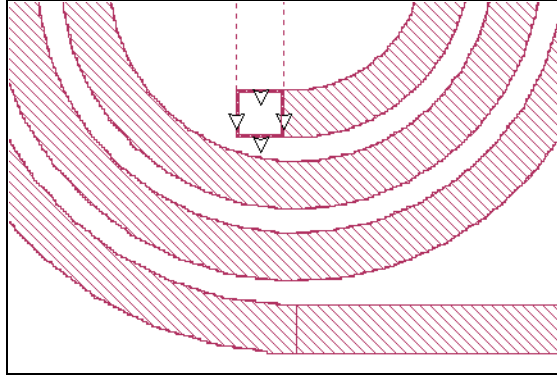
- 2 Hold down the Control key and click on the spiral conductor to select it.

Holding down the control key allows you to select another object without de-selecting any previously selected items. Both polygons are now highlighted.



3 Select Edit \Rightarrow Merge Polygons from the project editor main menu.

The two polygons are merged into one polygon and the overlap is removed.

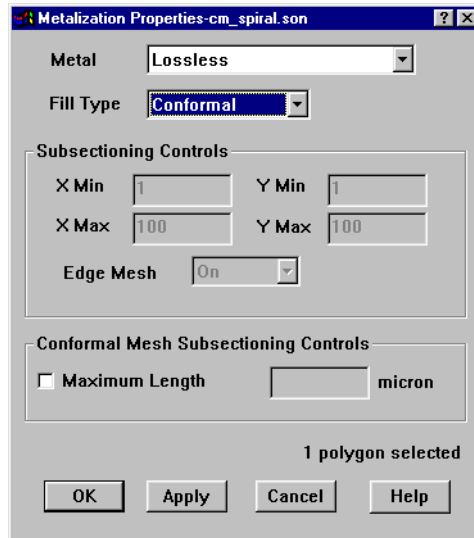


Invoking Conformal Meshing

You need to apply conformal meshing to the three turn spiral. To invoke conformal meshing, you apply the Conformal fill type to the spiral conductor polygon. To do so, perform the following:

- 4 **Double-click on the spiral inductor polygon to open the Metalization Properties dialog box for the polygon.**

The polygon is highlighted and the Metalization Properties dialog box appears on your display.



- 5 **Select Conformal from the Fill Type drop list.**

This applies conformal subsectioning to the spiral polygon. If a Notice appears, click on the OK button to close the message.

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

Note that the appearance of the polygon does not change.

Viewing Conformal Meshing

To see the effect that conformal meshing has on the subsectioning, you must perform an Estimate Memory command on the circuit and view the subsectioning. To do this, perform the following:

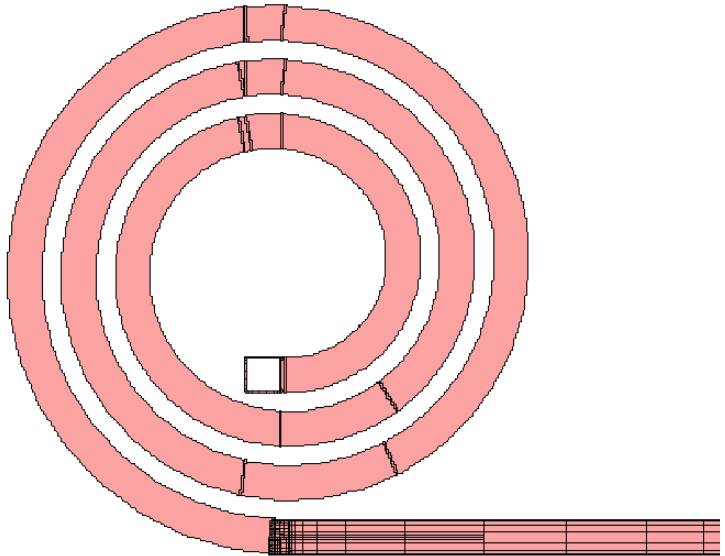
7 Select Analysis \Rightarrow Estimate Memory from the project editor main menu.

A status message appears while the circuit is being evaluated. It may take a few minutes to complete the subsectioning depending on the processing speed and memory available on your computer. Once the subsectioning is complete, the Estimated Memory window appears.

The memory requirement is now around 77 Mbytes and the number of subsections is approximately 2900 subsections.

8 Click on the View Subsections button in the Estimated Memory window.

There is a small delay while the Subsection Viewer is being opened. The circuit is shown with the subsectioning highlighted by black outlines as shown below.



The use of conformal meshing for the spiral inductor reduced the number of required subsections from about 70,000 to 2,800 and required memory from 7,000 Mbytes to about 77 Mbytes. This analysis is now a manageable problem. This concludes the conformal meshing tutorial.

Chapter 5 Microwave Office Interface Tutorial

Introduction

In a significant step towards the inter operability of high frequency design tools, Applied Wave Research (AWR) has provided the EM Socket in Microwave Office 6.0 or higher which allows you to use your completely integrated “solver on request.” Now you can choose the design flow you prefer and maintain synchronicity between your reference design and the EM structures that are part of your layout.

Sonnet has taken advantage of this opportunity by creating the Microwave Office Interface (MOI) which allows for the seamless incorporation of Sonnet’s world class EM simulation engine, *em*, into the Microwave Office 6.0 or higher environment. You can take advantage of Sonnet’s accuracy without having to learn the Sonnet interface. Although, for advanced users who wish to take

advantage of powerful advanced features not presently supported in the integrated environment, the partnership of AWR and Sonnet has simplified the process of moving EM projects between Microwave Office and Sonnet.

For a detailed discussion of the Microwave Office Interface, its modes of operation and translation issues, please see Chapter 14, “Microwave Office Interface” in the **Sonnet User’s Guide**.

The Microwave Office Interface (MOI) tutorial is designed to give you a brief overview of the interface between Sonnet and AWR’s Microwave Office. This tutorial assumes that you are familiar with the basics of using both Sonnet and Microwave Office. If this is not true, we recommend referring to the appropriate documentation. If you are new to Sonnet, we suggest performing the tutorials in the Sonnet Tutorial manual, available as part of the hardcopy manuals set or in PDF format through the Sonnet task bar before using this tutorial.

Tutorial Topics

The following topics are covered in this tutorial:

- Editing in Microwave Office, editing in Sonnet, and working outside of Microwave office.
- Opening Sonnet’s project editor from within Microwave Office.
- Setting up an Adaptive Band Synthesis sweep for the Sonnet analysis engine.
- Running a simple analysis in Microwave Office using Sonnet’s analysis engine, *em*.
- Observing response data in both Microwave Office and Sonnet.
- Exporting an EM structure from Microwave Office to a Sonnet project.
- Importing a Sonnet project into Microwave Office.

Obtaining the Example Project

The Microwave Office example project for this tutorial is supplied with your Sonnet software installation and is available through the PDF manuals on your computer. The Application Examples interface, accessed in PDF format on your computer, allows you to load the example project directly into Microwave Office.

NOTE:

You must have Adobe Acrobat Reader installed on your system to access the example interface 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 [Lowpass](#) 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.

- 2 Click on the Application Examples button in the PDF document.**
- 3 Click on the Complete List button.**

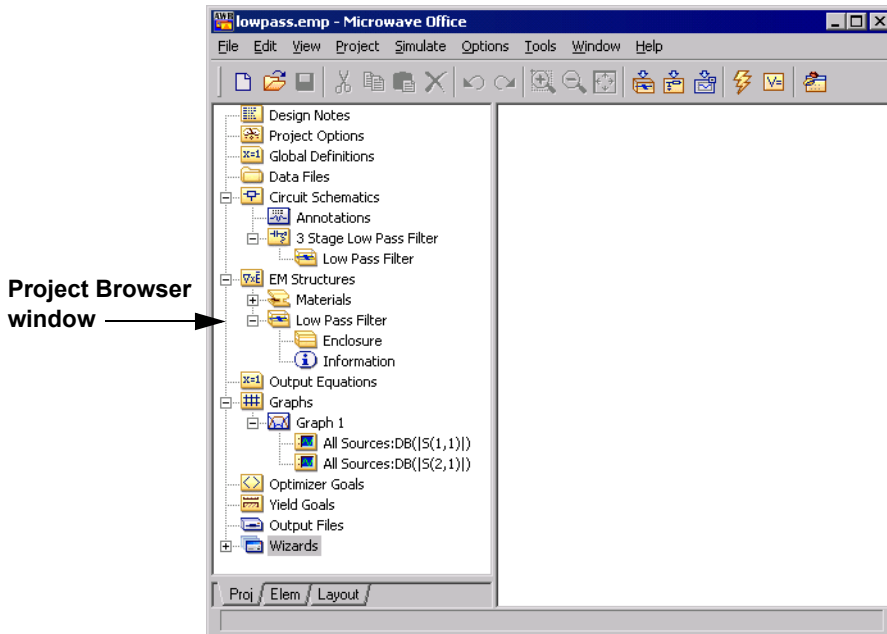
A page appears with a complete list in alphabetical order of the example files.

- 4 Click on Lowpass in the list.**

This will take you to the Microwave Office example project in the Application Examples.

- 5 Click on the **Load into Microwave Office** button at the top of the page.

Microwave Office is invoked on your display with the project “lowpass.emp” opened. It should appear similar to the picture shown below.



- 6 Double-click on “3 Stage Low Pass Filter” under Circuit Schematics in the project browser window.

This circuit uses the EM structure “Low Pass Filter” by connecting together three of the structures.

- 7 Double-click on “Graph 1” under Graphs in the project browser window.

The graph appears in the Microwave Office window. Note that the graph is empty since no analysis has yet been run on your Microwave Office project.

- 8 Double-click on “Low Pass Filter” under EM Structures in the project browser window.

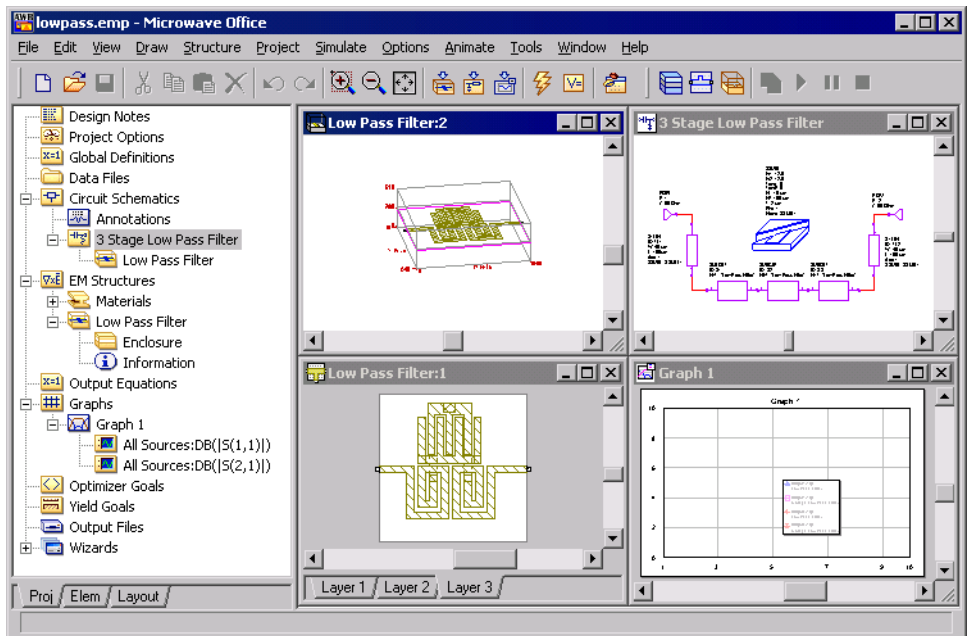
This opens the EM structure “Low Pass Filter” in the Microwave Office window. This is presently the active display indicated by the colored title bar.

- 9 Select View ⇒ New 3D View from the Microwave Office main menu.

A 3-dimensional view of the EM structure appears in the Microwave Office window.

- 10 Select Window ⇒ Tile Vertical from the Microwave Office main menu.

This command makes all four open windows the same size and fills all the space available. Your display should now appear similar to the picture shown below. The order in which the views are displayed may vary.



11 Select File \Rightarrow Save Project As from the Microwave Office 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 and avoid corrupting the example file.

Editing in Microwave Office

You may choose to edit your EM structure in Microwave Office or use the Native Editor, which in this case is Sonnet's project editor. When using Microwave Office to edit your structure, you remain in the Microwave Office environment, editing your EM structures and controlling Sonnet analysis options using options available in Microwave Office. The Sonnet analysis engine, *em*, is invoked from Microwave Office and the analysis results displayed in Microwave Office and, optionally, in Sonnet. This section of the tutorial will demonstrate how to select Sonnet as your analysis engine, how to setup the analysis in Microwave Office, how to execute the analysis and how to observe analysis results in Sonnet Engine Only mode.



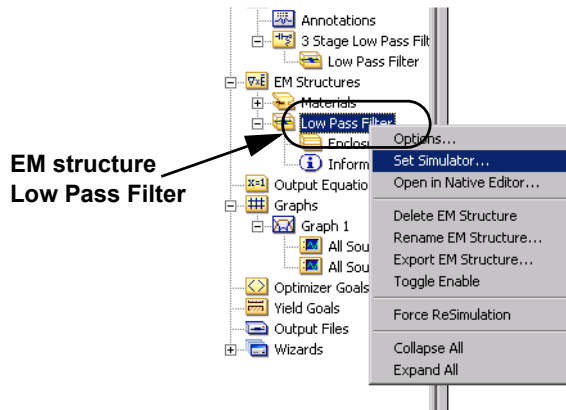
TIP

The Sonnet analysis engine must be selected for each EM structure in the Microwave Office project.

Selecting Sonnet as your EM Analysis Engine

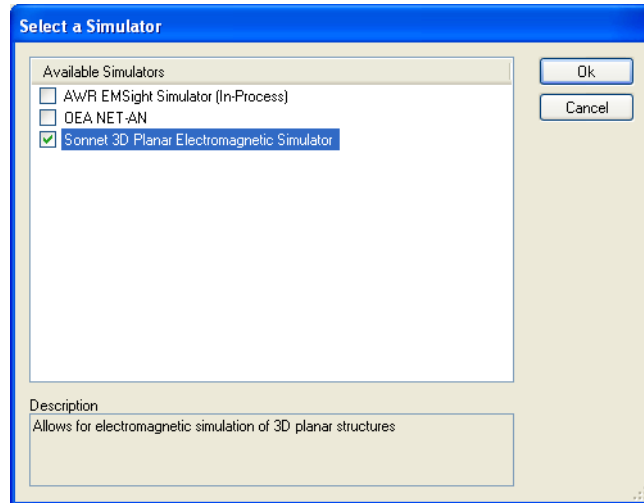
- 1 Right-click on the EM Structure “Low Pass Filter” in the Project Browser section of the Microwave Office window as shown below.

A pop-up menu appears on your display.



2 Select “Set Simulator” from the pop-up menu.

The Select a Simulator dialog box appears on your display.



3 Click on the “Sonnet 3D Planar Electromagnetic Simulator” checkbox.

This selects *em*, Sonnet’s electromagnetic simulator engine, as the engine to use when performing analyses on the Low Pass Filter structure in Microwave Office.

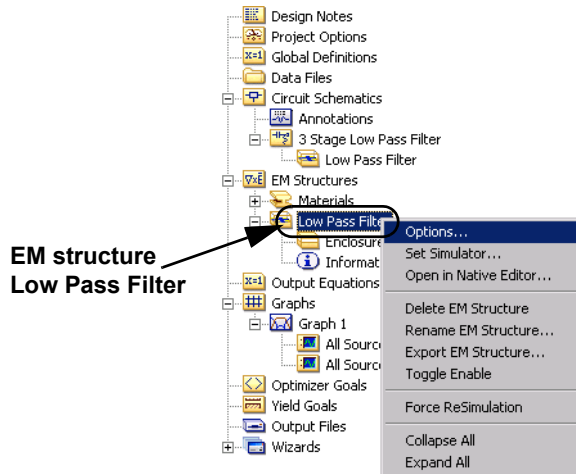
Click on the OK button to close the dialog box and apply the changes.

Selecting Analysis Controls

Sonnet provides some basic analysis controls within the Microwave Office environment for Sonnet Engine Only mode. To view and/or set these controls, do the following:

- 4 Right-click on the EM Structure “Low Pass Filter” in the Project Browser section of the Microwave Office window as shown below.

A pop-up menu appears on your display.

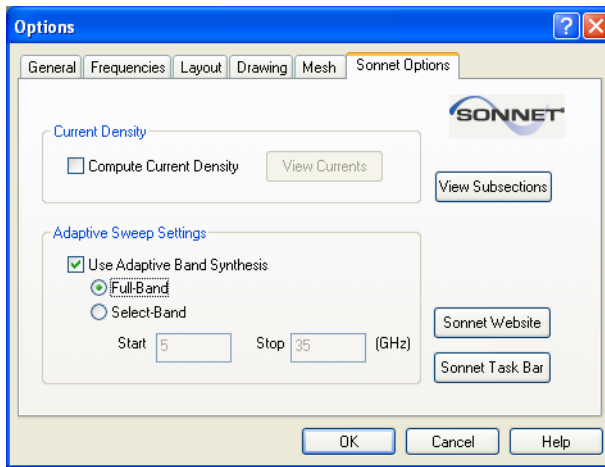


- 5 Select “Options” from the pop-up menu.

The Options dialog box appears on your display with the General Tab selected.

6 Click on the “Sonnet Options” tab in the Options dialog box.

The appearance of the dialog box changes and should appear similar to the picture shown below.



The ABS sweep across the full band is already set up by default, so you will need to take no action to setup the ABS sweep for Sonnet.

The Adaptive Sweep Settings control allows you to specify whether or not you want adaptive processing to be used when generating the S-parameter results. In Sonnet, the adaptive technique is called Adaptive Band Synthesis (ABS). If you disable Use Adaptive Band Synthesis, the analysis engine analyzes the circuit at each frequency specified in Microwave Office. This is recommended if you have fewer than five frequencies requested by Microwave Office.

If, however, you specified five, or more, frequencies in Microwave Office, then it is usually more efficient to enable Use Adaptive Band Synthesis. When enabled, the analysis engine does an ABS sweep over the specified frequency band first, and then it uses the adaptive results to generate S-parameters at each of the Microwave Office frequencies. This allows you to specify a very fine frequency resolution in Microwave Office, with hundreds or even thousands of individual frequencies, and obtain those results in the same time it would take to compute 5-10 frequencies without adaptive processing.

The Global project analysis frequencies for this Microwave Office project are from 5 to 35 GHz in 0.5 GHz steps. In this case, the default setting for Sonnet, full-band ABS, is the most efficient choice.

7 Click on the Compute Current Density checkbox to select this run option.

You check the Compute Current Density control to enable generation of current density information when the analysis engine runs. By default, current density information is not generated. If you have already run the analysis engine with this control enabled, the View Currents button will be active. Later in the tutorial, you use this button to launch the Sonnet Current Density Viewer.

You may also wish to view the Sonnet subsectioning information by pressing the View Subsections button. Experienced users often find it useful to view this information prior to analyzing so that they can assess whether the subsectioning is appropriate for their processing time and accuracy requirements.

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

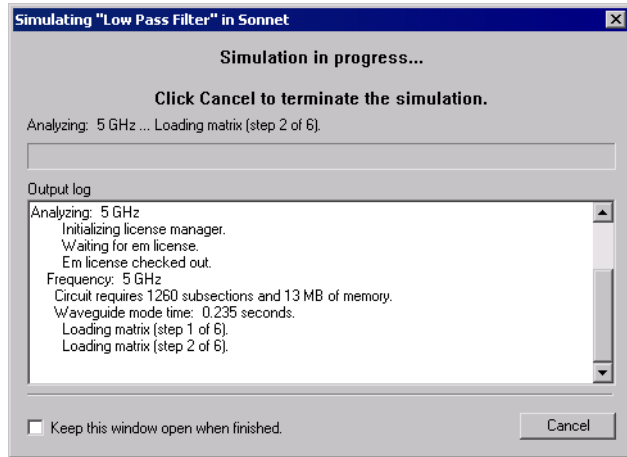
Running the Simulation

When using Microwave Office as the editor and you invoke a simulation, Microwave Office sends the geometry information of the EM structure to Sonnet along with the analysis controls. When the EM simulation is complete, Sonnet returns the requested analysis data.

To run the simulation, do the following:

9 Select *Simulate* \Rightarrow *Analyze* from the main menu of Microwave Office.

A status window indicating the progress of the EM simulation being performed by the Sonnet analysis engine, *em*, appears on your display as shown below. Depending on your computer, the EM analysis may take a few minutes to complete.



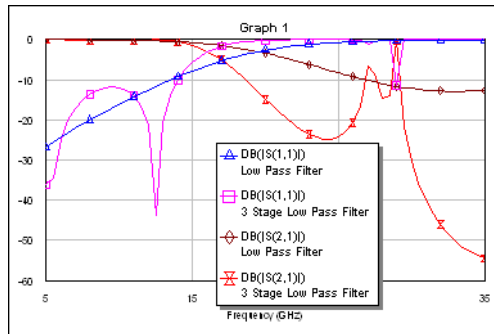
Once the simulation of the EM structure is complete, the status window is closed. The analysis of the complete circuit in the Microwave Office project is completed very quickly. You may now observe the results. Please note again that although Sonnet produced approximately 300 data points as the result of the ABS sweep, it returned data for only the 61 data points requested by Microwave Office.

During an ABS sweep, *em* analyzes at the beginning and end of the desired frequency band. Using an iterative process, *em* then analyzes at other discrete frequencies and determines a rational polynomial fit to the S-parameter data within the frequency band. The data produced by the full analysis at specific frequency points is the discrete data. Once a rational polynomial fit is achieved with an acceptable error, the frequency response across the specified bandwidth is calculated. The data generated using the rational polynomial is the adaptive data. For this particular ABS sweep, *em* performed a full analysis at 7 frequency points.

The S-parameter data returned from Sonnet to Microwave Office may be plotted using the graph capability in Microwave Office. For your convenience, a graph has already been set up in the example project.

- 10 If the graph is not already displayed in Microwave Office, double-click on Graph 1 in the Project Browser.**

The plot should appear similar to the one pictured below.



Microwave Office Plot

It is not possible at this time to display the current density data created by Sonnet within the Microwave Office environment, but you may invoke Sonnet's current density viewer from Microwave Office.

- 11 Once again, right-click on "Low Pass Filter" in the Project Browser section of the Microwave Office window.**

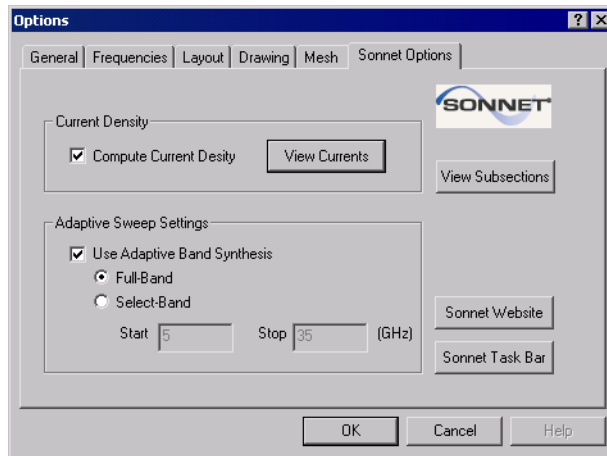
A pop-up menu appears on your display.

- 12 Select "Options" from the pop-up menu.**

The Options dialog box appears on your display with the General tab selected.

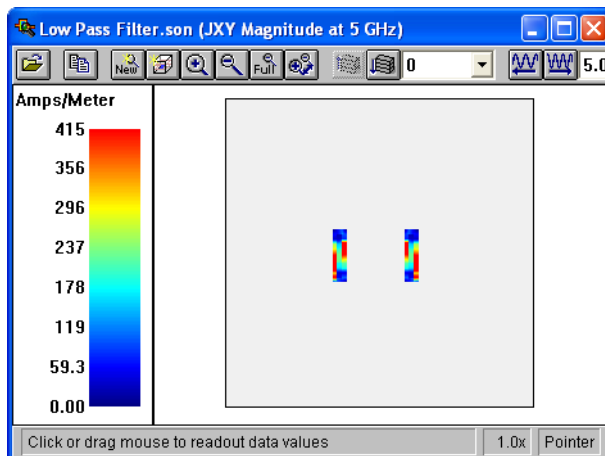
13 Click on the “Sonnet Options” tab in the Options dialog box.

The appearance of the dialog box changes and should appear similar to the picture shown below.



14 Click on the View Currents button to open Sonnet’s current density viewer.

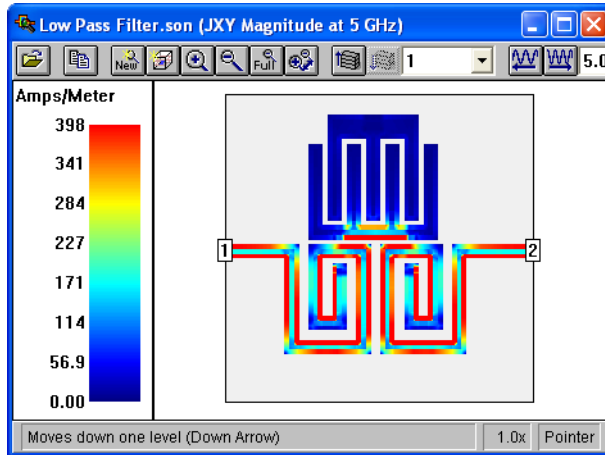
This button is not enabled unless an analysis has been run with the Compute Current Density run option enabled. The current density viewer appears on your display. Note that when you open the current density viewer, Microwave Office is not locked since you are not changing the geometry or producing response data.





- 15 Click on the Down Level button to view the lower metal level.

Level 1 of Low Pass Filter.son is displayed for the frequency 5 GHz.



The default animation setup for the current density viewer is to animate as a function of frequency, so it is a simple operation to view the change in current density in your structure as a function of frequency. It is important to note that current density data only exists for the seven discrete data points where a full analysis was done. The adaptive data does not include current density data.



TIP

For more details on using the current density viewer in Sonnet please refer to the first tutorial in the [Sonnet Tutorial](#) manual and to online help for the current density viewer.

- 16 Select File ⇒ Exit from the current density viewer main menu.

This exits the current density viewer. During the normal design process, observing the response data may lead to making changes in the EM structure. You would edit the EM structure in Microwave Office before once again simulating the circuit.

The next section of this tutorial discusses using the native editor in the Microwave Office Interface in which you use the Sonnet project editor to edit your EM structure.

Native Editor

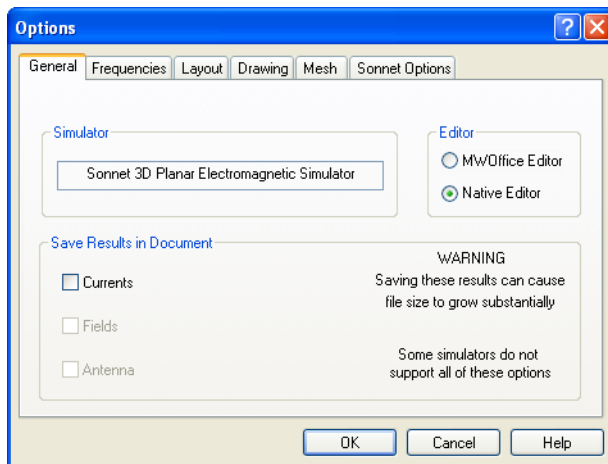
If you choose the native editor, Sonnet's project editor, you not only use Sonnet as your electromagnetic simulator but also EM structure editor. The native editor allows you to take advantage of all the capabilities unique to Sonnet including thick metal, dielectric bricks, autogrounded ports, etc. Using the native editor requires that you are already conversant with Sonnet software or will have to use Sonnet documentation to gain familiarity with the Sonnet environment.

For this tutorial, you will define a physically thick metal in Sonnet using the Thick Metal definition. The use of physically thick metal in this circuit is done solely for purposes of demonstration, not as a design decision. This metal model is a feature not available with Microwave Office and will be used to demonstrate using Sonnet's project editor as the native editor.

In the previous section of the tutorial, you have already selected Sonnet as your EM simulator. Now you will select Sonnet as your editor as well.

- 17 Right-click on Low Pass Filter in the project browser in Microwave Office and select “Options” from the pop-up menu.**

The Options dialog box appears on your display with the General tab displayed.



- 18 Click on the Native Editor radio button to select Sonnet as your editor.**

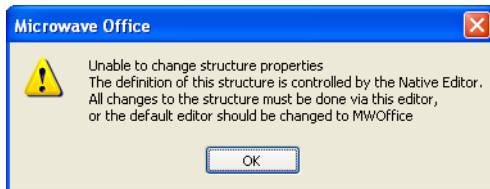
Note that Sonnet is displayed as the simulator to the left of the radio buttons.

- 19 Click on the Currents checkbox in the Save Results in Document section of the dialog box.**

As in the first part of the tutorial, you are going to generate current density data for the EM structure when you perform the analysis. To store this data for future use, you must choose to save the current density data as part of your Microwave Office project. Selecting this checkbox includes the current density data in the Microwave Office project when it is saved. Be aware that saving current density data may increase the size of your project file substantially.

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

Once you have completed this step, you are unable to edit the EM structure in Microwave Office. If you attempt to edit the EM structure in Microwave Office, you will receive the warning pictured below.

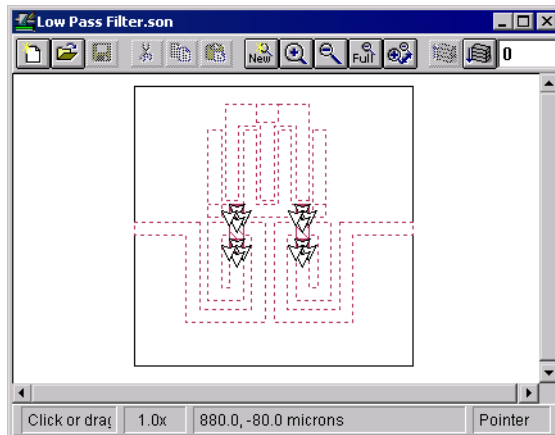


Next, you edit the circuit using the project editor in Sonnet.

Editing your EM Structure in Sonnet

- 21 Right click on Low Pass Filter in the project browser window in Microwave Office and select "Open in Native Editor" from the pop-up menu.**

Sonnet's project editor (*xgeom*) appears on your display with the EM structure Low Pass Filter open displaying the top most metal level 0.

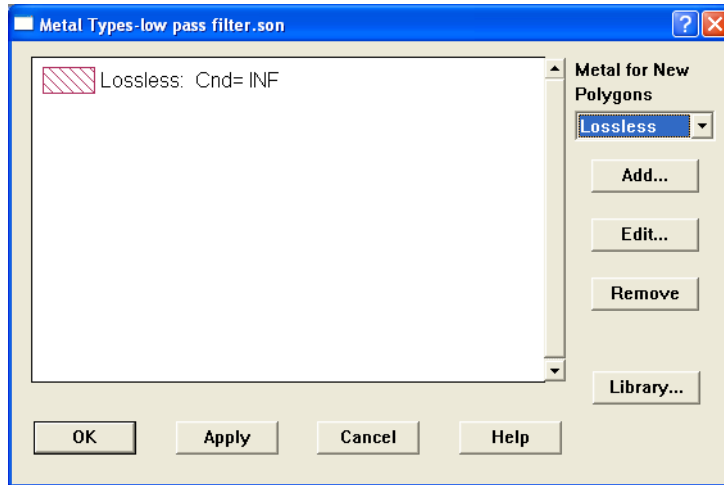


- 22 Click on the Down One Level button on the project editor tool bar.**

Metal level one appears in the project editor's window. You will change all of the metal on this level to thick metal, but first you must define the metal type.

- 23 Select Circuit ⇒ Metal Types from the project editor main menu.**

The Metal Types dialog box appears on your display.



- 24 Click on the Add button to add a new metal type.**

The Metal Editor dialog box appears on your display.

- 25 Enter “Thick Gold” in the Name text entry box.**

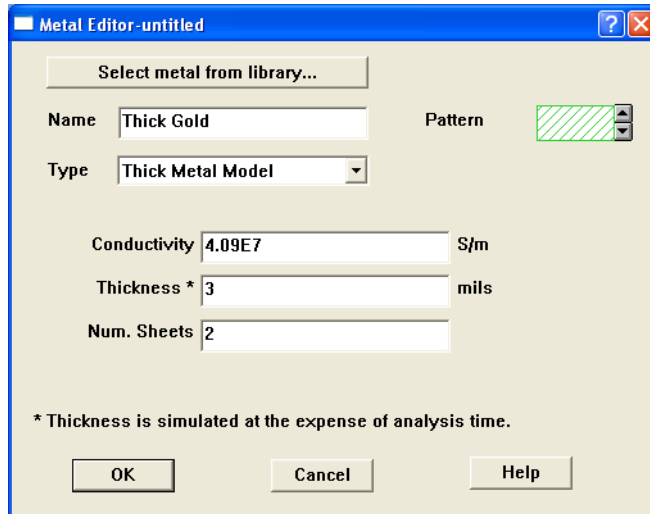
This name is used to select this metal type.

- 26 If it is not already selected, select “Thick Metal Model” from the Type drop list.**

This type is used to model physically thick metal. The other metal types are modeled as zero-thickness metal. For a discussion of thick metal modeling click on the Help button in the dialog box and refer to the “Thick Metal” chapter in the Sonnet User’s Guide.

27 Enter “4.09E7” for the conductivity and “3” for the thickness.


These parameters will be used to calculate the loss of the metal. The default number of two sheets is adequate for this example.



The image shows a screenshot of a software dialog box titled "Metal Editor-untitled". The dialog box has a blue title bar with a question mark icon and a close button. The main area is light beige and contains several input fields and buttons. At the top, there is a button labeled "Select metal from library...". Below this, there are two rows of labels and input fields: "Name" with the text "Thick Gold" and "Pattern" with a green hatched pattern icon. Below these, there is a "Type" dropdown menu showing "Thick Metal Model". Further down, there are three rows of labels and input fields: "Conductivity" with the value "4.09E7" and the unit "S/m", "Thickness *" with the value "3" and the unit "mils", and "Num. Sheets" with the value "2". At the bottom, there is a note: "* Thickness is simulated at the expense of analysis time." and three buttons: "OK", "Cancel", and "Help".

Metal Editor-untitled

Select metal from library...

Name Thick Gold Pattern 

Type Thick Metal Model

Conductivity 4.09E7 S/m

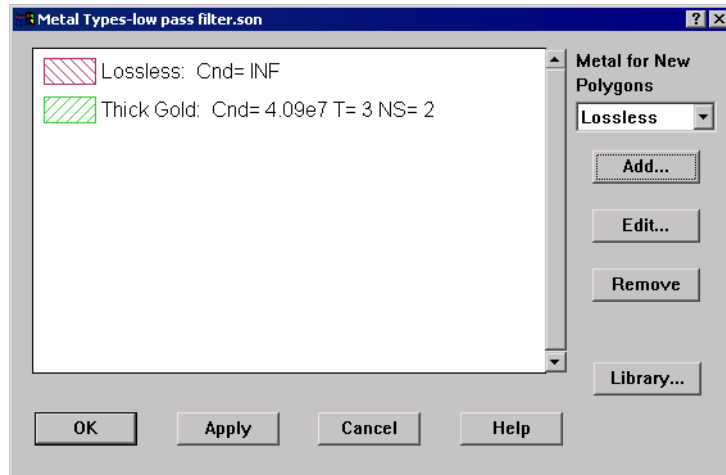
Thickness * 3 mils

Num. Sheets 2

* Thickness is simulated at the expense of analysis time.

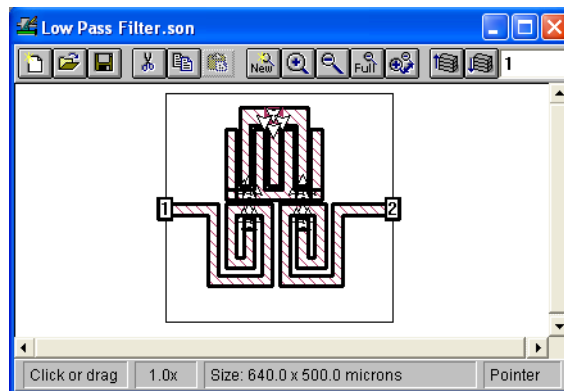
OK Cancel Help

Once you have entered all of the parameters, click on the OK button to close the dialog box and apply the changes. Thick Gold now appears in the list in the Metal Types dialog box.



- 28 Click on the OK button in the Metal Types dialog box to close the dialog box and apply the changes.
- 29 Drag the mouse around all of the metal polygons on level 1 to select all the polygons.

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

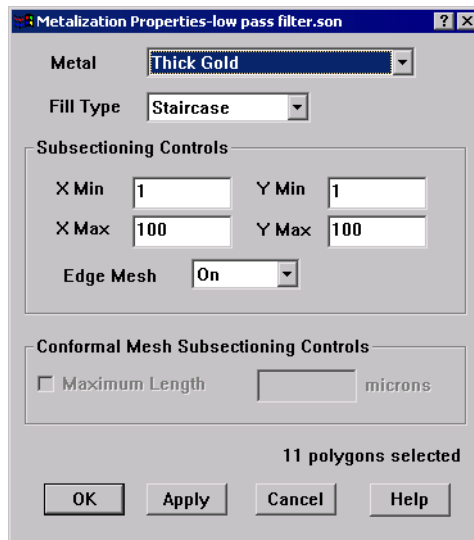


- 30 Select Modify ⇒ Metal Properties from the project editor main menu.**

The Metal Properties dialog box appears on your display.

- 31 Select “Thick Gold,” the metal type you just defined, from the Metal drop list in the Metal Properties dialog box.**

This applies thick metal to all the selected polygons.



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

Note that all the polygons now use a different fill pattern to indicate that they are using the Thick Metal metal type.



- 33 Click on the Up One Level button on the project editor tool bar.**

This changes the view in the project editor to level 0 of the circuit which contains the airbridges for the circuit.

- 34 Change the metal type of all the polygons on this level to “Thick Gold” in the same manner as you changed the metal type on level 1.**

All the metal in the circuit now uses the “Thick Gold” metal type.



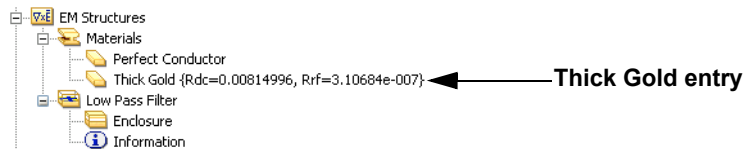
- 35 Click on the Save button on the project editor tool bar to save your changes.**

The project editor will prompt you to remove the analysis results stored in the project, since the changes made in the geometry make that data obsolete.

- 36 Click on the Delete button to delete the inconsistent results.**

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

The project editor disappears from your display. When you open the native editor in Microwave Office, the Microwave Office framework is “locked” and you are unable to make any changes to your project or run any simulations. When the project editor is closed, then Microwave Office is unlocked and you may once again make changes in the Microwave Office framework. When you view Microwave Office note that Thick Metal is now included on the Materials list in the Microwave Office project browser as shown below. The fill pattern in the EM structure has changed to indicate the use of Thick Metal. Since the thick metal



model is not available in Microwave Office, the loss definition is converted to the Microwave Office model which uses the DC resistance, R_{dc} , in ohms/sq and the skin effect coefficient, R_{rf} . The thick metal model definition is maintained as part of the Sonnet project stored in Microwave Office but is not displayed.

Running the Simulation

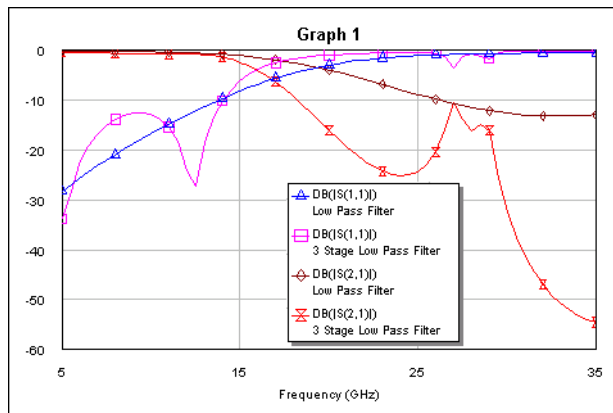
When you complete making your changes to the structure, save the changes and exit back to Microwave Office, the geometry specifications are sent to Microwave Office. In effect, the Sonnet project is stored as part of the Microwave Office project. When the analysis is run, the stored Sonnet geometry information is sent to Sonnet, the EM simulation is executed, then analysis data is returned.

38 In Microwave Office, select **Simulate** \Rightarrow **Analyze** from the main menu.

A status window indicating the progress of the EM simulation being performed by the Sonnet analysis engine, *em*, appears on your display. Depending on your computer, the EM analysis may take a few minutes to complete. The processing time for this part of the tutorial will be increased over the first part, since the use of thick metal does require more processing time in Sonnet.

Once the EM simulation of the EM structure is complete, the status window is closed. The analysis of the complete circuit in the Microwave Office project is completed very quickly. You may now observe the results. Please note again that although Sonnet produced approximately 300 data points as the result of the ABS sweep, it returned data for only the 61 data points requested by Microwave Office.

The S-parameter data returned from Sonnet to Microwave Office may be plotted using the graph capability in Microwave Office. For your convenience, a graph has already been set up in the example project. It should now appear similar to the graph displayed below. You may also observe the current density data in the current density viewer using the same method described starting at step 12.



If you were interested in seeing the effects of design changes on only the EM structure before analyzing your whole circuit, it is possible to remain in Sonnet after making your changes. You may run the EM analysis from Sonnet, and observe the results using the various Sonnet modules. Once you were satisfied that the EM structure was meeting your specifications, you would exit Sonnet and

return to the Microwave Office environment. When you execute the analysis on the whole circuit, the Sonnet analysis results are already available so that processing time is minimal.

Working Outside Microwave Office

Using the export and import functions in Sonnet allows you to run analyses in Sonnet without locking up Microwave Office for the duration of the processing time. However, working outside Microwave Office does require more overhead to keep the data consistent in the Microwave Office project. This section of the tutorial will demonstrate how to analyze an EM structure in Sonnet without locking up Microwave Office for the whole analysis, how to export an EM structure from Microwave Office to a Sonnet project and how to import a Sonnet project as an EM structure.

- 39 If it is not already open in Microwave Office, open lowpass.emp.**
- 40 Right-click on the Low Pass Filter EM structure in the Microwave Office project browser and select “Open in Native Editor” from the pop-up menu which appears.**

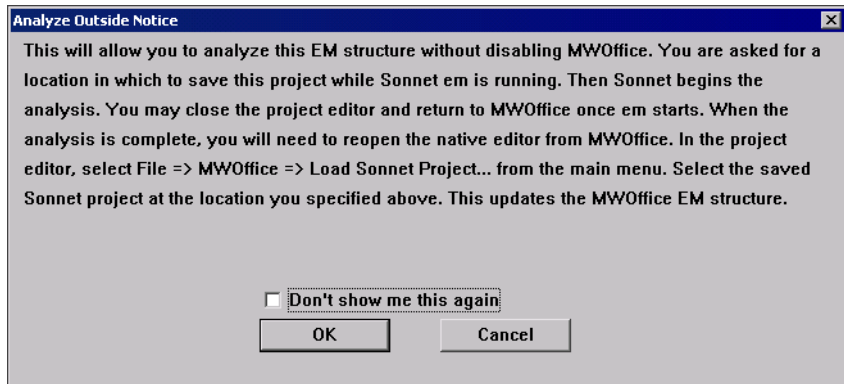
This opens the EM structure in Sonnet’s project editor.

Normally, you would now make changes to the EM structure before again analyzing the circuit. Since we only wish to demonstrate the Export and Import features, no changes will actually be made to the circuit.

When you opened the EM structure in Sonnet, the analysis controls set up in Microwave Office are transmitted as part of the geometry sent to Sonnet. You wish to analyze the circuit but are aware that the processing time is considerable. Therefore, you want to analyze the EM structure without locking up Microwave Office for the whole processing time of the analysis. To do this, perform the following:

- 41 Select File ⇒ MWOOffice ⇒ Analyze Outside MWOOffice from the project editor main menu.**

A message explaining the command appears on your display. The Analyze Outside MWOOffice includes the analysis frequency controls from the Microwave Office project in the export. If you do not wish to run an analysis right away or plan to work on the circuit exclusively in the Sonnet environment, you may use the *File ⇒ MWOOffice ⇒ Save as Sonnet Project* command to export the EM structure to a Sonnet project.



TIP

If you do not want this message to appear each time you select this command, you may select the Don't show me this again checkbox. If you later wish to turn the message back on, you may do so in the Hints tab of the Preferences dialog box. The Preferences dialog box is accessed by selecting *File ⇒ Preferences* from the project editor main menu.

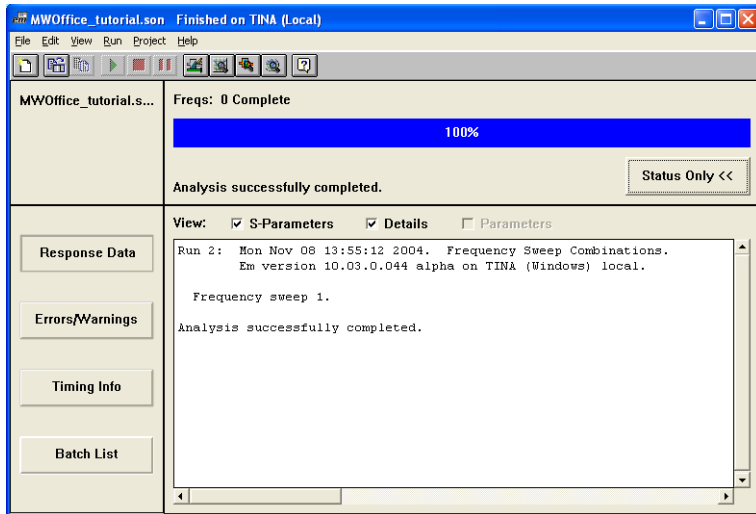
- 42 Once you have read the message, click on OK to close the message box.**

The message box disappears and a Browse window appears on your display.

In order to unlock Microwave Office, you must save the EM structure to a temporary Sonnet project location. This window allows you to specify the location of the Sonnet project.

- 43 Enter the location c:\MWOOffice_tutorial.son in the Browse window and click on the OK button.**

If you wish, you may choose another location on your computer. The Browse window closes, then the analysis monitor appears on your display which indicates the analysis has begun. Notice that the file name displayed in the title bar of the analysis monitor is “MWOOffice_tutorial.son.”



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

The project editor closes and Microwave Office is restored on your display. Microwave Office is now unlocked and is available for use. The EM structure was transmitted back to Microwave Office when the project editor was closed. It is important that you select Sonnet as the Native Editor when you use this mode so that you can not make changes in Microwave Office to the EM structure while the analysis is ongoing in Sonnet. This would cause the geometry to be inconsistent with the analysis data.

- 45 Wait until the analysis in Sonnet is complete.**

Normally, this would take a noticeable amount of time, but since we made no changes to the EM structure, the analysis was able to use the data from the previous simulation so that the job completes very quickly.

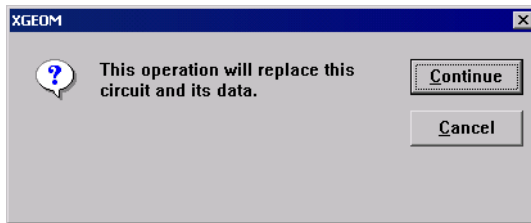
Now you need to import the temporary Sonnet project file in order to make the analysis results available in Microwave Office.

- 46 Right-click on the EM structure Low Pass Filter in the project browser in Microwave Office and select “Open in Native Editor” from the pop-up window which appears.**

The EM structure is opened in the project editor. You want to replace this with the Sonnet project whose analysis just completed.

- 47 Select File ⇒ MWOoffice ⇒ Load Sonnet Project from the project editor main menu.**

A warning message that you are going to overwrite the EM structure appears on your display.



- 48 Click on the Continue button.**

The message window disappears and a Browse window appears which allows you to select the Sonnet project which you wish to load.

- 49 Select “c:\mwoffice_tutorial.son” in the Browse window.**

This is the temporary file to which you saved the EM structure when you executed the Analyze Outside MWOoffice command in step 43. This command loads the project file in, along with the analysis results, in place of the EM structure you opened in Microwave Office.

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

The project editor is closed and the Sonnet project you just loaded is imported into Microwave Office for the EM structure. Microwave Office is unlocked and restored on your display.

51 Select Simulate \Rightarrow Analyze from the main menu of Microwave Office.

The analysis completes very quickly since the results for Low Pass Filter have already been computed. The graph is updated. This should be the same graph as shown on page 94 since the EM structure has not been changed. This completes the Microwave Office Interface tutorial.

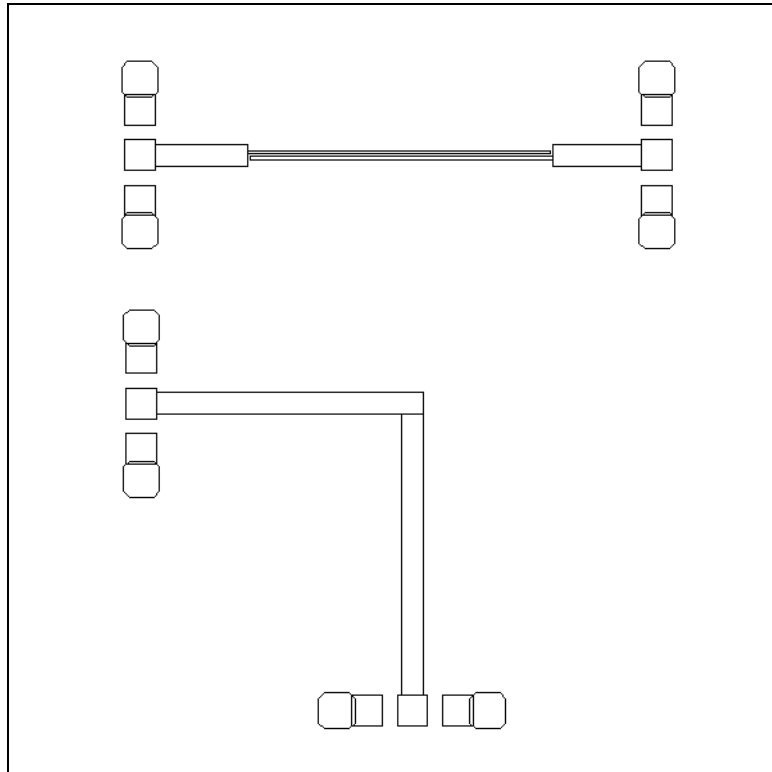
Chapter 6 GDSII and DXF Translator Tutorial

This tutorial teaches you the basics of performing a translation of a GDSII or DXF file into a Sonnet project. Since the operation of the two translators is very similar, the two translators were combined into one tutorial with separate sections for the areas where they differ. For a detailed discussion of the GDSII and DXF translators, please see Chapter 15, “The DXF and GDSII Translators” in the **Sonnet User’s Guide**.

This tutorial describes a simple example. The example is found in the examples directory provided with your software and is called “dcblock.dxf” for the DXF translator or dcblock.gds for the GDSII translator.

Sonnet Supplemental Tutorials

These files are a DXF and GDSII file that contain the same two circuits: a coupled line structure to be used as a DC block, and a bent transmission line as pictured below. The objective of this example is to convert the DXF or GDSII file into a Sonnet project and to tweak the file so that the DC block may be analyzed in *em*.



The DXF or GDSII example file contains two circuits.
This tutorial uses the top circuit.

Obtaining the Translator Example Files

You need to copy the example [Dcblock](#) from the Sonnet examples. Dcblock is a directory which contains the DXF and GDSII files used for input in this example in addition to a layer mapping file for the conversion. If you do not know how to obtain a Sonnet example, select *Help* \Rightarrow *Examples* from any program menu, then click on the **Instructions** button. If you are reading this manual in the PDF format on your computer, click on the link above.

Determine Level Mapping

First you must determine the mapping of DXF layers to project editor levels. To do this you import the DXF file with the default layer mapping. You use the resulting project to determine what layers you wish to retain from your DXF file. You perform a second import of the same file, this time editing the contents of the Layer Mapping dialog box and saving the settings to a layer mapping file for use in subsequent imports.

NOTE:

Although you edit the layer mapping for this example, that will not be necessary for many DXF files; the default map created by the DXF translator will be sufficient.

1 Invoke the project editor.

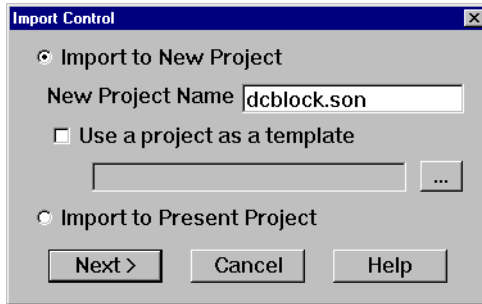
The project editor window appears on your display.

2 Select *File* \Rightarrow *Import* \Rightarrow *DXF* or *File* \Rightarrow *Import* \Rightarrow *GDSII*, depending on which translator you are using, from the main menu of the project editor.

A Browse window appears on your display.

- 3 **Locate the example file “dcblock.dxf” or “dcblock.gds” in your working directory and click on the Open button in the browse window.**

The Import Control dialog box appears on your display.



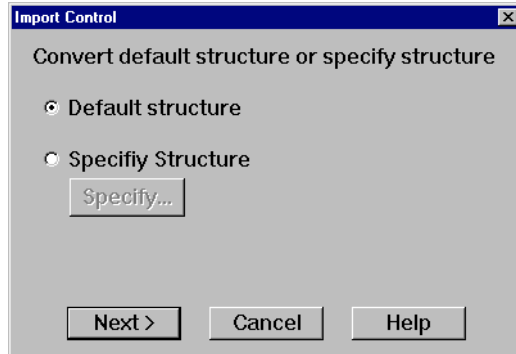
For this example, you want to import the file “dcblock.dxf” or “dcblock.gds” to a new Sonnet project “dcblock.son”. The default option is to import to a new project and the default project name is the basename of the DXF or GDSII file with the “.son” extension of a Sonnet project file. No action needs to be taken since the defaults are what you want.

NOTE:

If you are importing a GDSII file, continue the tutorial at Step 4 below. If you are importing a DXF file, continue the tutorial at Step 7 on page 108.

- 4 Click on the Next button in the Import Control dialog box to continue.

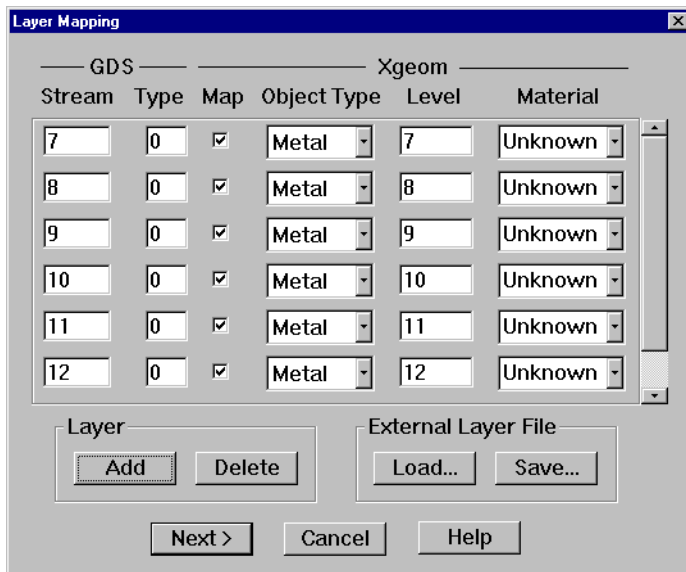
The Structure dialog box appears on your display.



For this example, there is only one structure in the GDSII file; therefore, the default option of converting the default structure is acceptable. If the GDSII file you are importing has multiple structures, you may select the “Specify Structure” option. This enables the Specify button. Clicking on this button opens a list of structures available in the GDSII file. You may select which structure you wish to convert to the Sonnet project.

5 Click on the Next button in the Structure dialog box to continue.

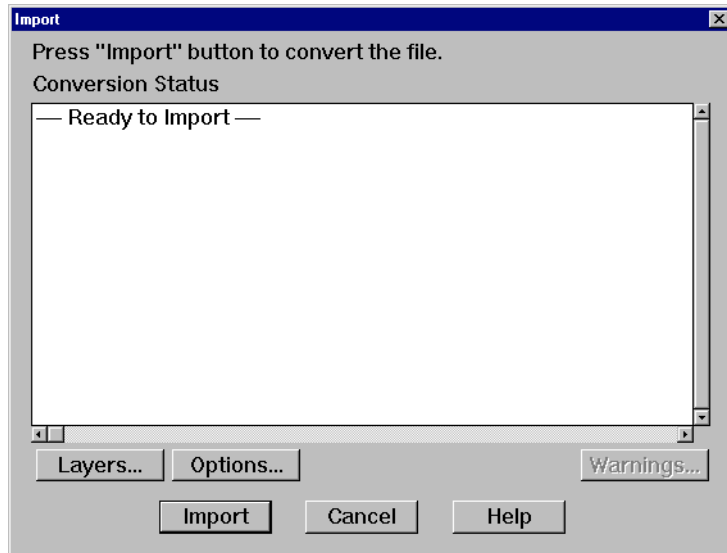
The message “Reading Layers, please wait.” appears in the Import Control dialog box. When the project editor finishes obtaining the layer information from the GDSII file, the Layer Mapping dialog box appears.



You do not know what mapping you want until you observe the converted file in the project editor, so you do not need to change anything in this dialog box at this time.

6 Click on the Next button in the Layer Mapping dialog box to continue.

The Import dialog box appears on your display.



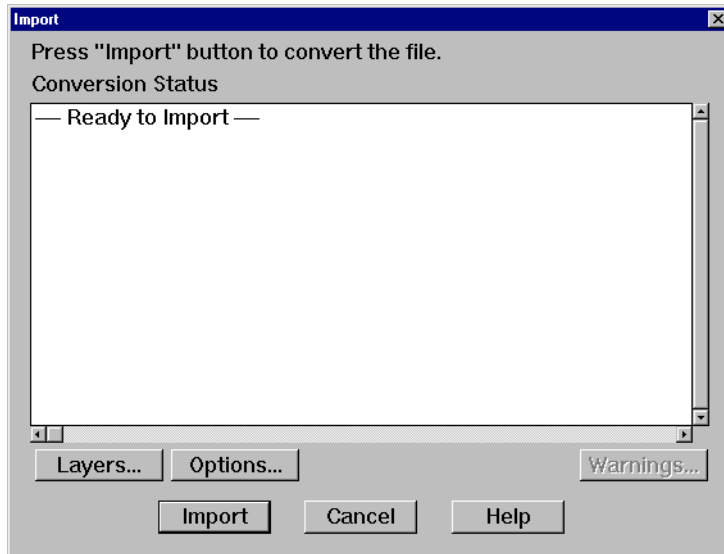
The message “Ready to Import” appears in the output window. Since the default option settings are those you wish to use for this file, you do not need to open the Import Options dialog box. For information about setting import options, refer to the project editor’s Help.

NOTE:

Continue the tutorial at Step 8 on page 109.

- 7 Click on the **Next** button in the **Import Control** dialog box to continue.

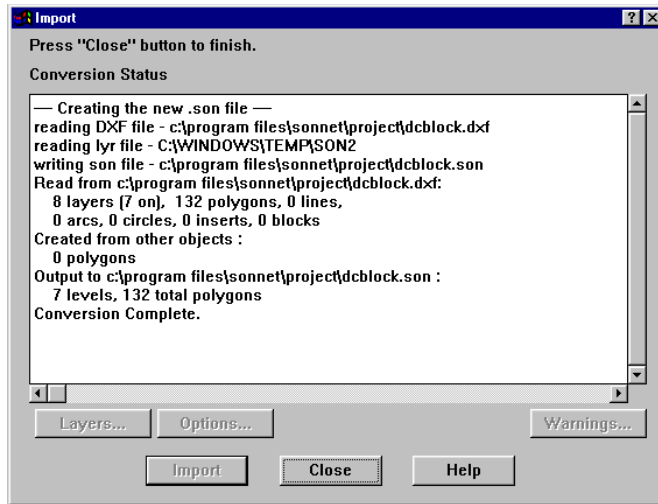
The Import dialog box appears on your display.



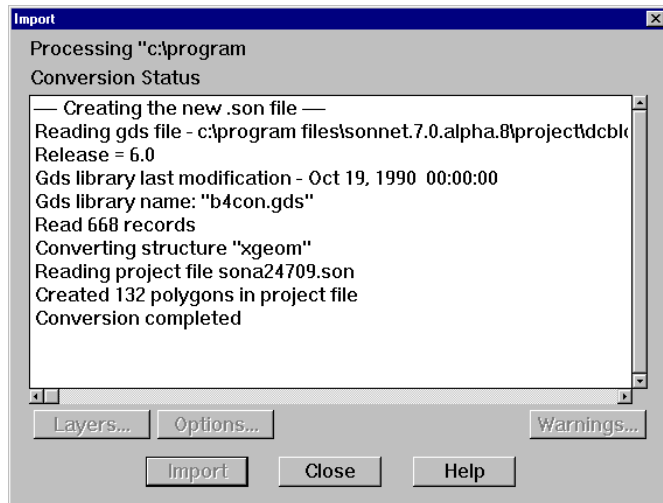
The message "Ready to Import" appears in the output window. Since the default option settings are those you wish to use for this file, you do not need to open the Import Options dialog box. For information about setting import options, refer to the project editor's Help.

- Click on the Import button in the Import dialog box to execute the conversion.

Progress messages appear above the output window. When the import is complete, the DXF or GDSII file information is displayed in the output window.



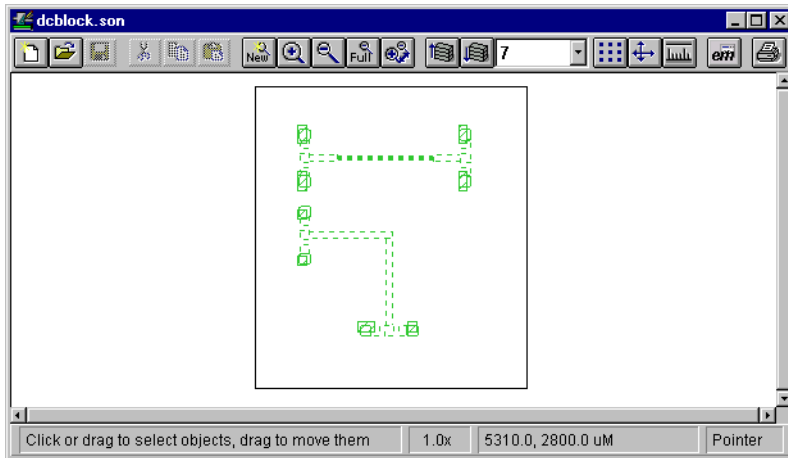
DXF
Information



GDSII
Information

- 9 Click on the Close button to close the Import dialog box.

The target project “dcblock.son” is open in the project editor.



Look at each of the circuit levels in the project editor. Since the cell size has not yet been defined, it may be difficult to see your circuit. Toggle the cell fill on and off at your discretion (using ^M) during the rest of this example.

Notice that there are several layers. The project editor’s level 5 has all of the information that we need to do an *em* analysis. You should now edit the layer mapping in the Layer Mapping dialog box so that the conversion uses only this layer. To do this, import the file again, but before importing edit the Layer Mapping dialog box.

- 10 Select **File** ⇒ **Import** ⇒ **DXF** or **File** ⇒ **Import** ⇒ **GDSII** from the project editor main menu.

This opens the Browse window.

- 11 Once you have located the example file “dcblock.dxf” or “dcblock.gds” click on the **Open** button.

The Import Control dialog box appears on your display.

12 Select the Import to Present Project option.

This imports the DXF or GDSII file to the same project as before.

NOTE:

If you are importing a GDSII file, continue the tutorial at Step 13 below. If you are importing a DXF file, then continue the tutorial at Step 20 on page 113.

13 GDSII Click on the Next button in the Import Control dialog box to continue.

The Structure dialog box appears on your display. The default structure is correct.

14 Click on the Next button in the Structure dialog box to continue.

This opens the Layer Mapping dialog box. Now that you know that layer 8 has all of the information that is needed to do an *em* analysis, you edit the inputs of this dialog box so that the conversion uses only this layer.

15 Click on the Map checkbox for all the rows except for that containing the GDSII Stream number 8.

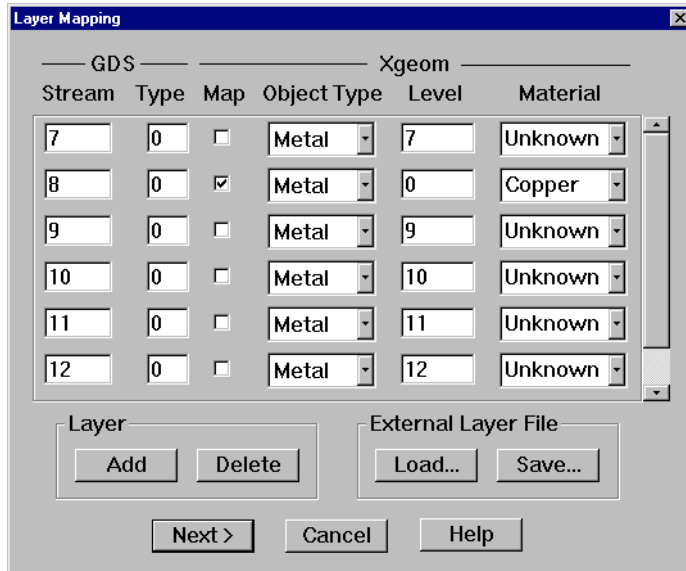
This leaves stream layer 8 as the only layer which is used in the conversion. The other layers are ignored since the Map checkbox is off. Since there are too many layers to display all of them, use the scroll bar to access layers that are not presently displayed.

16 Enter “0” in the Xgeom Level text entry box for the Stream number 8 entry.

Since our example is a single layer circuit, you set the Xgeom Level number to “0” which is the first level in the project editor.

17 Enter “Copper” in the Xgeom Material column in the same row.

Metal is already selected as the Xgeom Object type. The type of metal you wish to use is Copper. Since the metal “copper” is not defined in the project file, you will need to define its attributes later. Your Layer Mapping dialog box should look similar to below.



If you wish to save the settings to use for future imports of this file, you would click on the Save button which allows you to save the settings to an external layer file. To load an external layer file, click on the load button.

18 Click on next in the Layer Mapping dialog box to continue.

19 Click on Import in the Import dialog box to complete the import.

Once the import is complete, click on the Close button in the Import dialog box. The file “dcblock.son” should be open in the project editor.

NOTE:

If you are importing a GDSII file, continue the tutorial at “Define Di-electric Layer and Metallizations,” on page 115.

- 20 To Continue the DXF tutorial, click on the Next button in the Import Control dialog box to continue.**

The Import dialog box appears on your display and messages indicate that the layers have been read from the file by the DXF translator.

- 21 Click on the Layers button to open the Layer Mapping dialog box.**

This opens the Layer Mapping dialog box. Now that you know that layer 5 has all of the information that is needed to do an *em* analysis, you edit the inputs of this dialog box so that the conversion uses only this layer.

- 22 Click on the Map checkbox for all the rows except for that containing the Xgeom level 5.**

This leaves DXF layer “Filter” as the only layer which is used in the conversion. The other layers are ignored since the Map checkbox is off. Since there are too many layers to display all of them, use the scroll bar to access layers that are not presently displayed.

- 23 Enter “0” in the Xgeom Level text entry box for the DXF Layer “Filter” entry.**

Since our example is a single layer circuit, you set the Xgeom Level number to “0” which is the first level in the project editor.

24 Enter “Copper” in the Xgeom Material column in the same row.

Metal is already selected as the Xgeom Object type. The type of metal you wish to use is copper. Since the metal “copper” is not defined in the project file, you will need to define its attributes later. Your Layer Mapping dialog box should look similar to below.



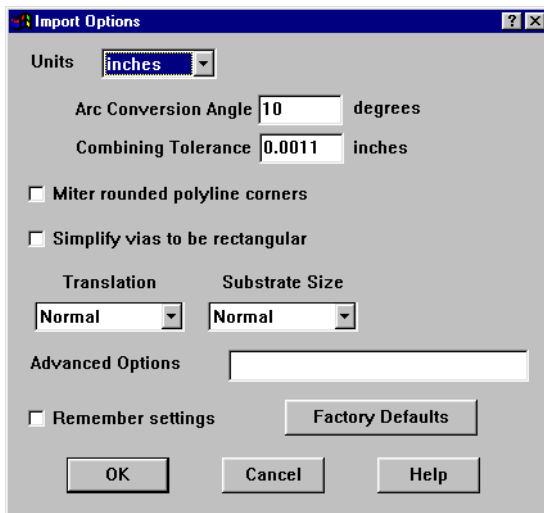
If you wish to save the settings to use for future imports of this file, you would click on the Save button which allows you to save the settings to an external layer file. To load an external layer file, click on the load button.

25 Click on OK in the Layer Mapping dialog box to continue.

You return to the Import dialog box.

26 Click on the Options button in the Import dialog box.

The Import Options dialog box appears on your display. This example was input in microns. Since the default Units value is inches, you need to change this before importing the circuit.



27 Select “microns” from the Units drop list.

The DXF translator will use microns as the length unit when it imports the DXF file.

28 Click on the OK button to apply the changes.

The Options dialog box closes. You are ready to import your DXF file.

29 Click on Import in the Import dialog box to complete the import.

Once the import is complete, click on the Close button in the Import dialog box. The file “dcblock.son” should be open in the project editor.

Define Dielectric Layer and Metallizations

Now define the metallization and dielectric layer parameters in the project editor. The procedure is the same as making these changes for any normal project.

In this example, set the top dielectric layer to 1000 microns of air ($\epsilon_{rel}=1.0$) and the bottom dielectric to 100 microns of GaAs ($\epsilon_{rel}=12.9$) in the Dielectric Layers dialog box. Also, in the Metal Types dialog box, edit the entry for copper using the Normal metal type and setting the Conductivity to $5.8E7$ S/m, the Thickness to 15 microns and the Current Ratio to 0.

Remove Parts of the Circuit Not Being Used

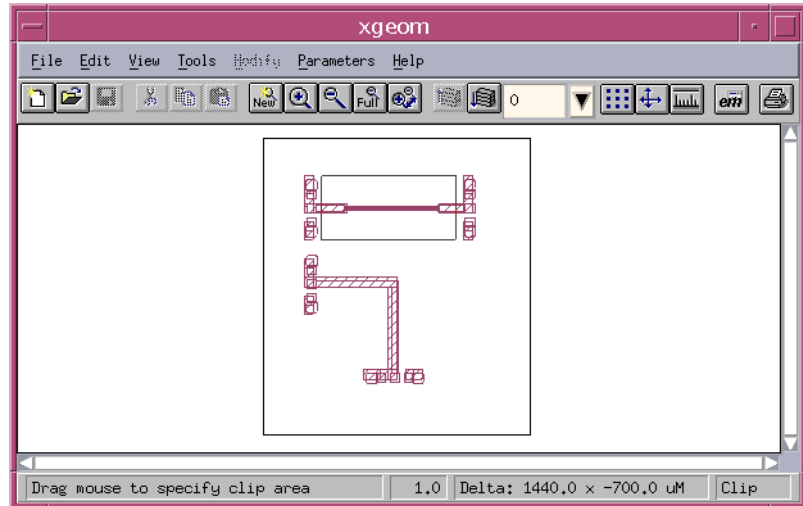
Before you do the deletes and clips, you may want to save your file.

You may have converted a much larger section of the circuit than you want to analyze with *em*. Thus you may want to delete large sections of the circuit from multiple levels. You may even need to delete entire levels using *Circuit* \Rightarrow *Dielectric Layers*. In our example, we used the Layer Mapping dialog box to delete extra metal levels that we did not need, but all the dielectric layers were imported. Either way is acceptable since dielectric layers without metal on them do not affect processing time or memory requirements.

Use *Edit* \Rightarrow *Clip* to window out a part of the circuit you wish to analyze. To window out a precise area, you may need to turn the snap off before selecting *Edit* \Rightarrow *Clip*.

You can also delete several polygons at once by using *Select* \Rightarrow *Single Layer* or *Select* \Rightarrow *Mult. Layer* to select several polygons and points. Press the “Delete” key to delete the selected objects. Remember, any place where you can remove metal that does not affect your answer shortens *em* execution time.

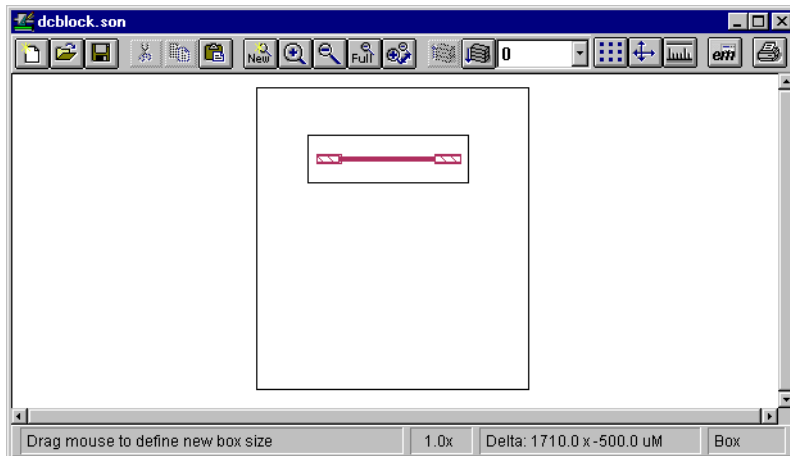
In our example, save your file and then use *Edit* \Rightarrow *Clip* as shown below. The RF probe structure is calibrated out in the measurement so it should not be included in the *em* analysis.



Decide on a Substrate Size and Cell Size

Based on your geometry, you should decide on your cell size and substrate size. Remember to first set the area size to a multiple of the cell size. Since you have deleted a large portion of the circuit, you may now need to resize your substrate.

Next, you need to set the size of the substrate. Select *Circuit* \Rightarrow *Box* to open the Box Settings dialog box. Now click on the *Set Box Size with Mouse* button, then lasso the area around the remaining circuit. This is done to get a box size approximately “close” to the final box size.



Then, precisely set the x dimension of the box to 1800 microns and the y dimension of the box to 800 microns by entering the values in the text entry boxes in the Box Settings dialog box. Now set the cell size to 10 microns by 10 microns. Notice that we selected a substrate size that was a multiple of the cell size. Click on the *OK* button to apply the new box size and close the dialog box.

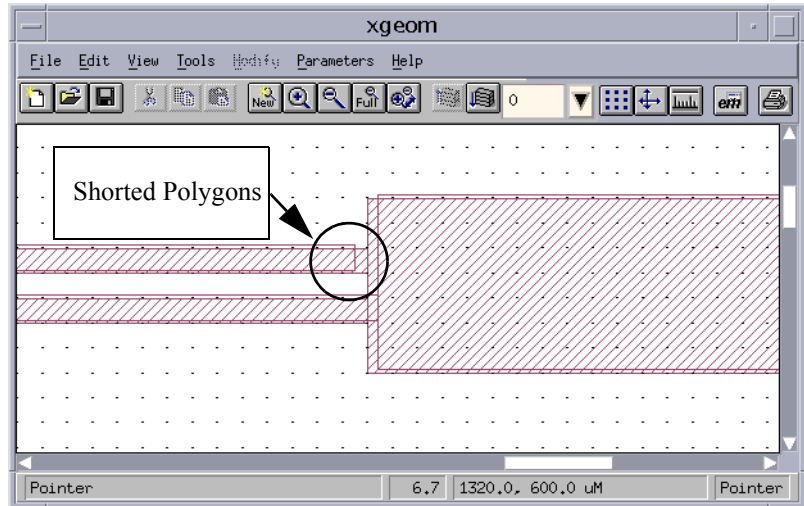
Change Polygons to Have the Proper Fill

Change the polygon fill (subsection type) if needed. For example, you may need to change several polygons from Staircase fill to Diagonal fill. To change several polygons at one time, you can select multiple polygons then use *Modify* \Rightarrow *Metal Properties*.

Our example does not need to have the polygon fill changed.

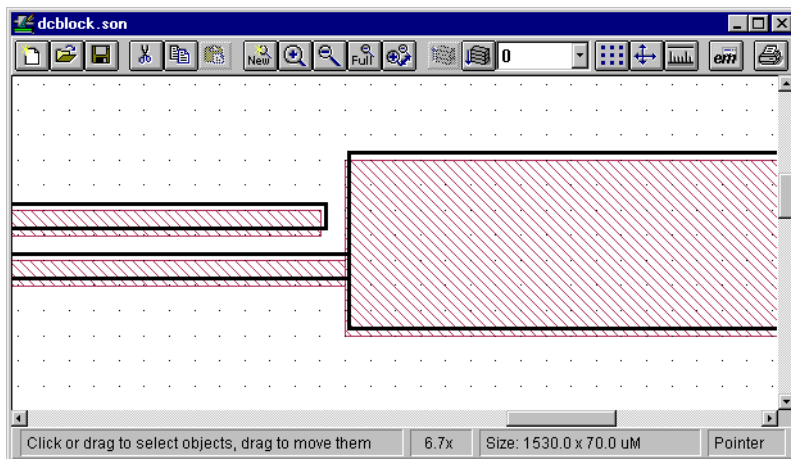
Align the Circuit to Grid Points

Now check to see how *em* will subsection your circuit. To do this, make sure the cell fill is turned “On”. The cell fill shows you how *em* will subsection your circuit. If you did not choose a small enough cell size or if your circuit is not oriented exactly on grid points, the metal may “short” two polygons together. This happens in our example as pictured below. The problem shown in the figure can be fixed by moving the entire circuit by a fraction of a cell.



For our example, turn the cell snap off, using *Tools* \Rightarrow *Snap Setup* to open the Snap Grid Setup dialog box. Click on the *No Snap* radio button to turn the snap off. Click on the *OK* button to apply the changes and close the dialog box. Now,

using *Edit* \Rightarrow *Select All*, move your circuit so that the circuit is aligned to the grid points. Do not worry about getting the circuit exactly aligned. Notice that the polygons are no longer “shorted” together.



TIP

You may use the shortcut key, ctrl-A for the command *Edit* \Rightarrow *Select All*

At this point you may want to snap all of the polygons to grid points. Before you do the snap, you may want to save your file. Select all of the polygons. Now select *Modify* \Rightarrow *Snap to* from the menu. Click on the X and Y radio button in the Cells section of the Snap Objects dialog box which appears. Then select the Preserve Spacing and Shape Relative to the Reference Point radio button. Click on the Select Reference Point button and select a reference point in your circuit. All other vertices in your circuit will be snapped relative to the position of this reference point so that the shape of the imported circuit is maintained. When the dialog box reappears, click on the *OK* button to close the dialog box and snap the circuit to the grid.

If desired, you can select only parts of the circuit to snap to the grid points.

Move Points Around as Needed

Some points may now need to be moved. Pay particular attention to mitered corners and narrow lines and gaps.

Our example needs to be centered. Make sure the snap is set to cell snap (*Tools* \Rightarrow *Snap Setup* and select *Cell Size*) and choose *Edit* \Rightarrow *Select All*. Now move your circuit to the approximate center of your substrate.

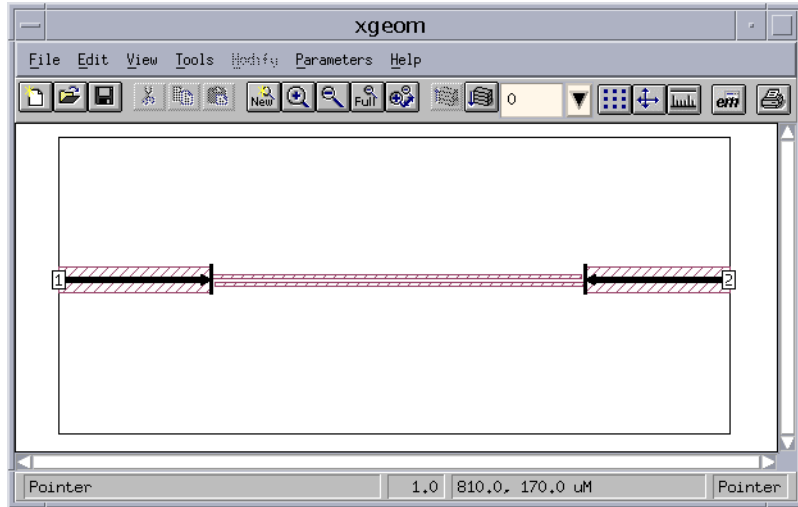
Add Vias

Next, add vias as needed. The metallization being put down in a standard DXF file may only represent the mask dimensions and layers and not the actual physical dimensions or location of the via. You may need to create each via by hand.

Our example does not require vias.

Add Ports and Reference Planes

To finish the example, add ports and reference planes. Typically this entails extending transmission lines to bring the circuit out to the edge by using *Tools* \Rightarrow *Reshape* and *Modify* \Rightarrow *Snap To*. The finished circuit is shown below.



You may now analyze this circuit using *em*. It should have approximately 200 subsections if analyzed at 10 GHz.

Chapter 7 Agilent Interface Tutorial

This tutorial provides you with an overview of the basic use of the Agilent Interface. For a detailed discussion of the Agilent Interface, please refer to Chapter 16 "The Agilent Interface" on page 279 of the **Sonnet User's Guide**. An example project directory, [Ebads_prj](#), has been supplied with your software. This directory contains a few circuits with which you may experiment. The examples are designed to demonstrate most of the processes and features of the Agilent interface in detail. To follow along with these examples, copy the example project directory into your working directory (the directory from which you typically run Agilent).

If you do not know how to obtain a Sonnet example, select *Help* ⇒ *Examples* from any program menu, then click on the **Instructions** button. If you are reading this manual in the PDF format on your computer, click on the blue link above.

To follow along with these examples, start the Agilent software and open the project Ebads_prj.

After it loads, you should see a menu labeled *Ebridge* in the main window. If at any time you need to reset this menu (because of interactions with other programs or if it does not appear) open the command line window by selecting *Option* ⇒ *Command line* from the main menu of ADS. The command line dialog box appears on your display. In the command line text entry box, enter the following:

```
reload_ebridge();
```

This will reload the software and reset the menu.

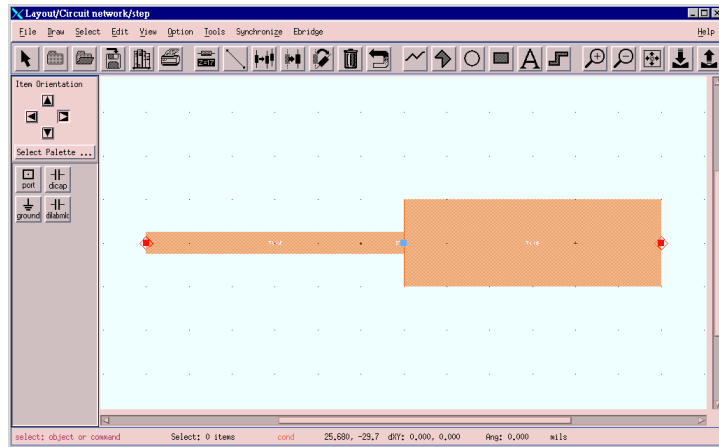
A Simple Example

The example project has a simple circuit to demonstrate the basic operation of the software. This example has been prepared so that the typical adjustments described later are not required.

- 1 Open ADS and open the appropriate project.**

2 Open the layout window and load the design step, a simple step junction.

The design should appear similar to the one shown below.

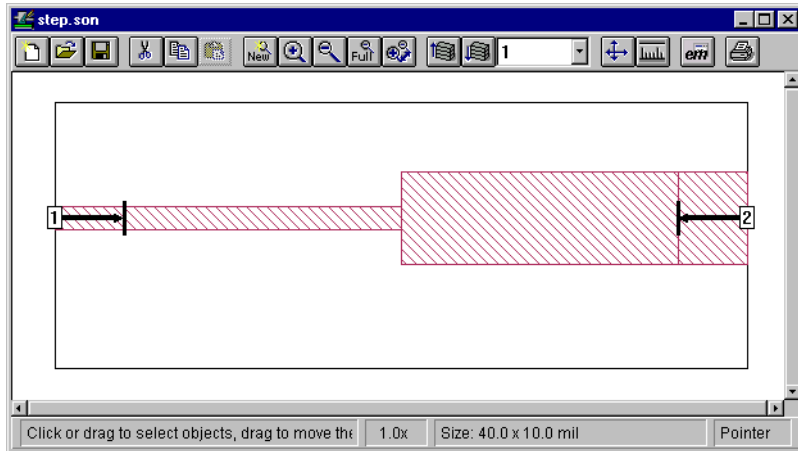


3 Select *Ebridge* \Rightarrow *Rep* \Rightarrow *Geo* from the layout window main menu to translate the circuit.

A pop-up window appears on your display to report that the file "step.son" has been created.

- 4 Select *Ebridge* \Rightarrow *Launch xgeom* from the layout window main menu to start a project editor session of the translated design.

The Sonnet project is shown below.

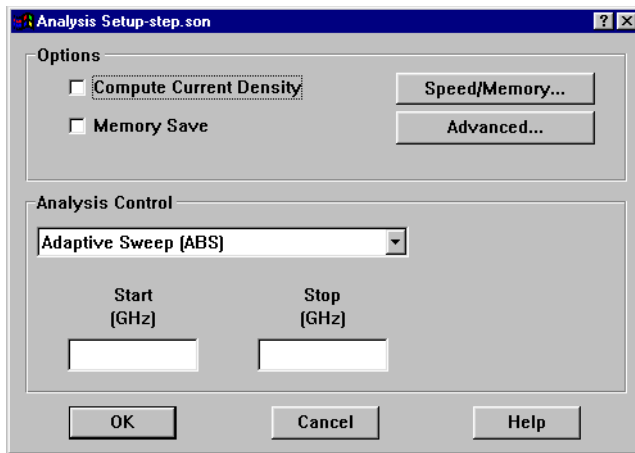


Sonnet project editor session of the translated design "step.son".

Note the automatic extensions of the reference planes to the actual circuit dimensions. Note also that the box and cell sizes work out nicely. This is because of the "step.prf" file which helps control the translation process. These files are described in detail in "The Preference File," on page 286 of the **Sonnet User's Guide**.

- 5 **Analyze the circuit by selecting *Analysis* \Rightarrow *Setup* from the project editor's main menu.**

The Analysis Setup dialog box appears on your display.



- 6 **Select Linear Frequency Sweep from the Analysis Control drop list.**

This selects a Linear Frequency sweep for your analysis. The dialog box's appearance is updated with the appropriate entry boxes for the linear sweep.

- 7 **Enter the analysis frequencies in the Linear Sweep.**

In this example we are analyzing at 5 GHz. To do this, enter 5 GHz in the Start text entry box in the Analysis Control section.

- 8 **Click on the Compute Current Density checkbox.**

Selecting this run option produces current density data which is stored in the project and which may be viewed in the current density viewer.

- 9 **Click on the OK button to apply the Analysis Setup.**

The Analysis Setup dialog box is closed.

The response data is normally stored as part of a project, but when using an Agilent product, you need a separate response file that is readable by your circuit layout program. When you create the Sonnet project with the **Rep** \Rightarrow **Geo**

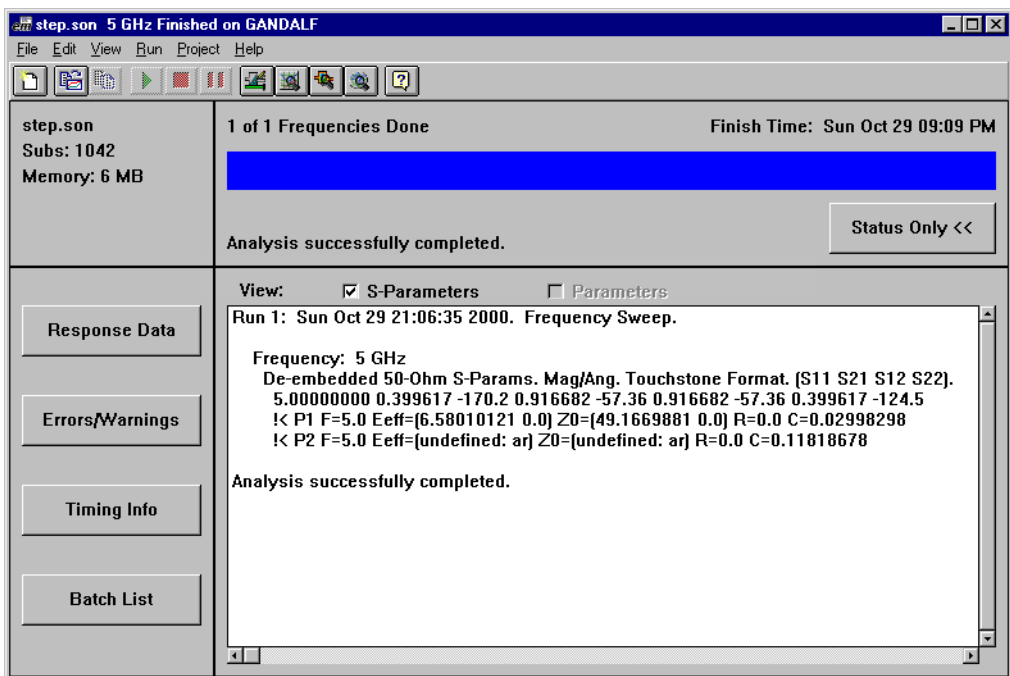
command, an optional output file is automatically specified. The file name is the basename of your project with the extension.sNp where N is the number of ports. For this example, the output file is step.s2p. This file contains de-embedded S-parameters in the Touchstone format.

10 Select *File* ⇒ *Save As* from the project editor’s main menu.

The file must be saved as a project before analyzing. Use the browse window which appears on your display to save the file as “step.son” in your working directory.

11 Select *Project* ⇒ *Analyze* from the project editor main menu or click on the *Analyze* button on the project editor tool bar to launch the *em* analysis.

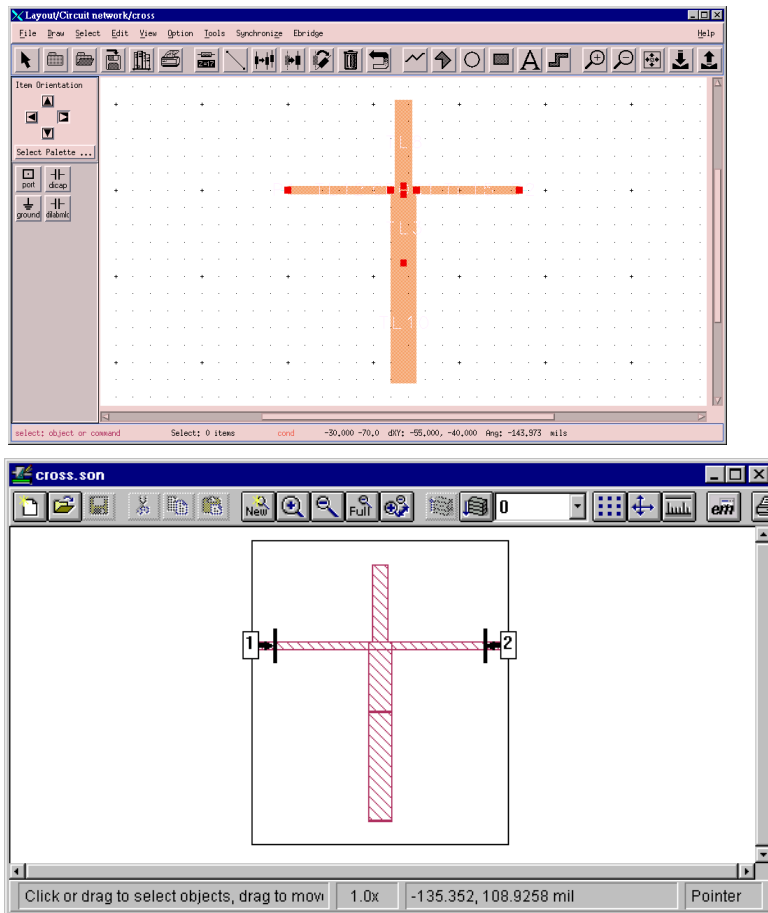
The Analysis monitor appears on your display in response data mode.



After *em* finishes the analysis, the current density can be viewed in the current density viewer or S-parameter data can be viewed by looking at the output file “step.s2p” or with an S2P data element in the Agilent framework. To look at the current density file, click on the View Current button on the analysis monitor tool bar or select *Project* \Rightarrow *View Current* from the menu. A current density viewer session of the “step” project will be started. To view the S-Parameter data, click on the View Graph button on the analysis monitor tool bar. The S-parameter data file will be opened in Sonnet’s response viewer.

Adjusting the Sonnet Project Prior to Analysis

After translation, some adjustments of the project will probably be necessary prior to *em* analysis. It is a good idea to at least review all files in the project editor just to be sure they are exactly what you want before you start an analysis. The next example, “*cross*”, is shown in the figure below.



Picture of the “*cross*” example in the layout window and the translated structure in the project editor.

We will use this and another example to demonstrate how to adjust a Sonnet project prior to analysis.

1 Open the design called “*cross*” in the layout window.

The layout window appears on your display with the file “cross.dsn” displayed.

2 Select *Ebridge* ⇒ *Rep* ⇒ *Geo* from the layout main menu.

You have now created a Sonnet project called *cross.son* in your project directory.

Now we have to adjust the Sonnet geometry project for subsequent *em* analysis.

3 Select *Ebridge* ⇒ *Launch xgeom* from the layout main menu.

This will start up a project editor session of our example file “*cross.son*”.

In most translations, the items that will probably require adjustment are:

- 1** Substrate size and cell size.
- 2** Polygon fill type.
- 3** Circuit alignment to grid points.
- 4** Point location for polygon vertices.

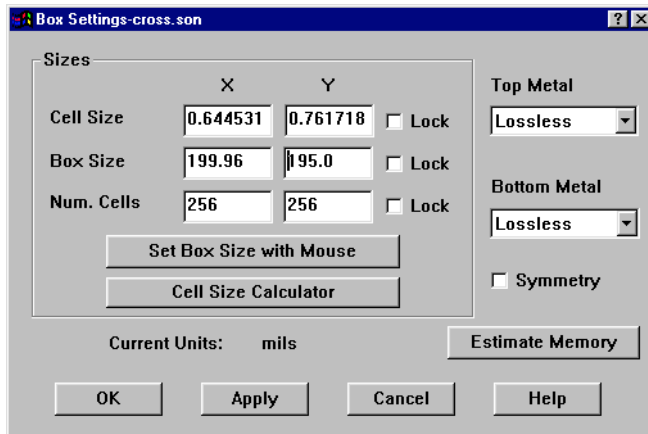
Depending on your circuit, all of these steps may not be needed. However, we recommend that you try to follow the steps in the order given.

Substrate Size and Cell Size

Based on your geometry, decide on your cell size and substrate size. The cell size should be determined first since the area size is a multiple of the cell size.

Remember that the substrate size must be divided into a uniform grid of cells and that all metal will be snapped to that grid. More information on this can be found in Chapter 4 of the **Sonnet User’s Guide**.

The substrate size can be set by either windowing out an area or directly entering the dimensions in the Box Settings dialog box in the project editor. To window out the area, you click on the Set Box Size with Mouse button in the Box Settings dialog box. Shown below is the box settings of our example as translated.



The area is shown as 199.96 by 195 mils. For this problem we would like a cell size of 5 mils by 5 mils and the box 200 by 195 mils.

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

The Box Settings dialog box, as shown above, appears on your display.

2 Enter a value of “5” in both the X and Y text entry boxes in the Cell Size row.

This defines a cell size of 5 mils by 5 mils.

3 Click on the Lock checkbox at the end of the Cell Size row.

This holds the cell size constant while making any other changes to the dimensions.

4 Enter the value “200” in the X text entry box in the Box Size row.

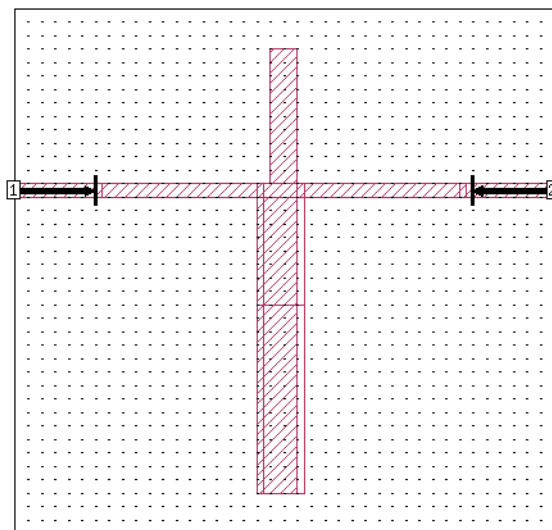
The box will be 200 mils in the X direction.

5 Enter the value “195” in the Y text entry box in the Box Size row.

The box will be 195 mils in the Y direction. Notice that since the cell size is locked, the number of cells in both the X and Y direction are updated to indicate the new box and cell size, i.e. 40 in the X direction and 39 in the Y direction.

6 Click on the Apply command button to apply the changes, but leave the dialog box open.

The new circuit is shown below. Note that the large stub on the bottom is not situated exactly on the grid. This can be solved by changing the X cell dimensions to 2.5 mils.



Sonnet file “cross.son” after changing box settings to 200 by 195 mil area and 5 by 5 mil cells.

7 Unlock the cell size and enter a value of “2.5” in the X text entry box in the Cell Size row. Then click on the OK button to apply the changes and close the dialog box.

The circuit is updated. Notice that the stub is now exactly on the grid.

- 8 **Select *File* \Rightarrow *Save* from the project editor main menu to save the file.**

You are now ready to setup your *em* analysis.

- 9 **Select *Analysis* \Rightarrow *Setup* from the main menu of the layout window.**

The Analysis Setup dialog box appears on your display.

- 10 **Select *Linear Frequency Sweep* from the *Analysis Control* drop list.**

- 11 **Enter the analysis frequencies in the *Start*, *Stop* and *Step* text entry boxes.**

Since this is only an example, analyze only one or two frequencies to save processing time. Also, if you wish to output a current density file from the analysis, click on the Compute Current Density checkbox in the Analysis Setup dialog box.

- 12 **Click on the *OK* button to apply the changes.**

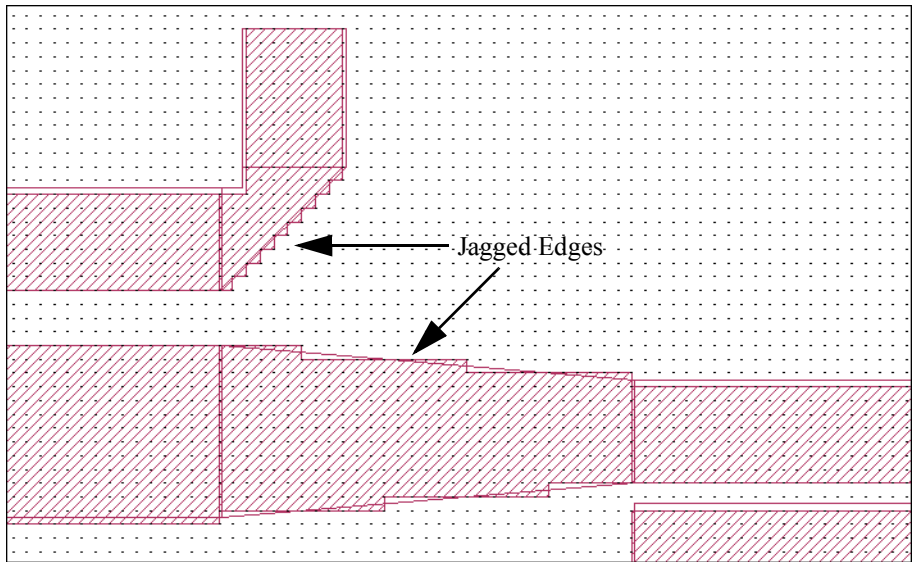
You have completed setting up the analysis. Click on the Analyze button on the project editor tool bar to execute. The analysis monitor appears on your display.

When the analysis is complete, you may observe the current density file by clicking on the View Current button on the analysis monitor tool bar. If you wish to see the S-Parameter data, select *Project* \Rightarrow *View Response* \Rightarrow *New Graph* from the menu of the analysis monitor.

Change Polygons to Have the Proper Fill

Change the polygon fill (subsection type) if needed. For example, you may need to change several polygons from Staircase fill to Diagonal fill. In the previous example, all of our polygons were rectangles, and our default fill, staircase, was sufficient. This is not always the case.

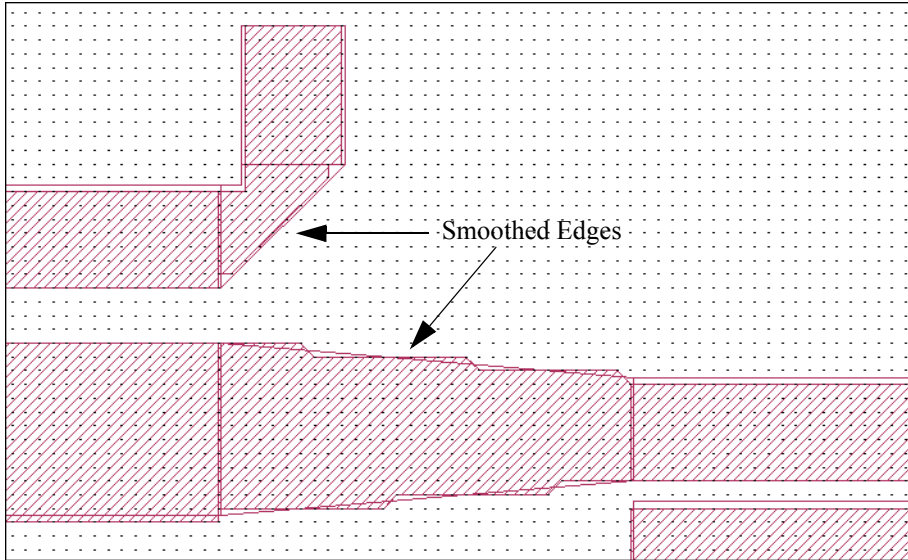
Open the design “*two_coup*” in the layout window and translate it with the **Rep** ⇒ **Geo** menu option. The figure below shows a zoomed area of the translated file.



Zoomed area of “two_coup.son” showing polygons in need of Diagonal fill.

This file has already had its substrate size and cell size adjusted to: 380 by 190 and 2 by 2 mils, respectively, using the procedure described above. Note the diagonal lines have a jagged “sawtooth” pattern to them. This can be changed by modifying the fill for each polygon to Diagonal. To change several polygons at one time, select multiple polygons then select *Modify* ⇒ *Metal Properties* from the project editor main menu. The Metalization Attributes dialog box appears in which you can change the fill type to Diagonal. For more information, consult the project editor Help.

The figure below shows the same area after the fill has been changed on the two polygons highlighted. Note that the top polygon is considerably smoother, except for the two areas at the edges. Fix this by moving points around and adding points to polygons (see below). Save this file now to prepare for this step.

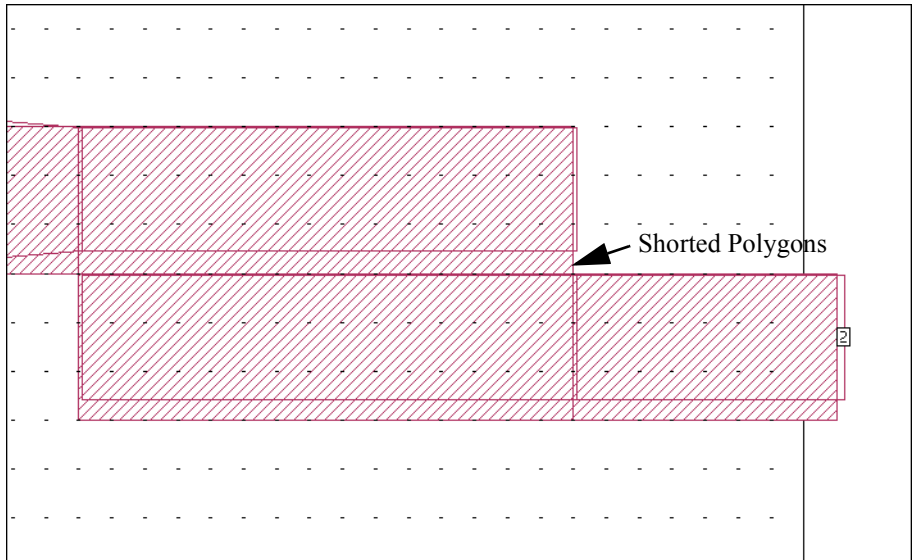


Zoomed area of “two_coup.son” showing polygons after having fill changed to Diagonal.

Align the Circuit to Grid Points

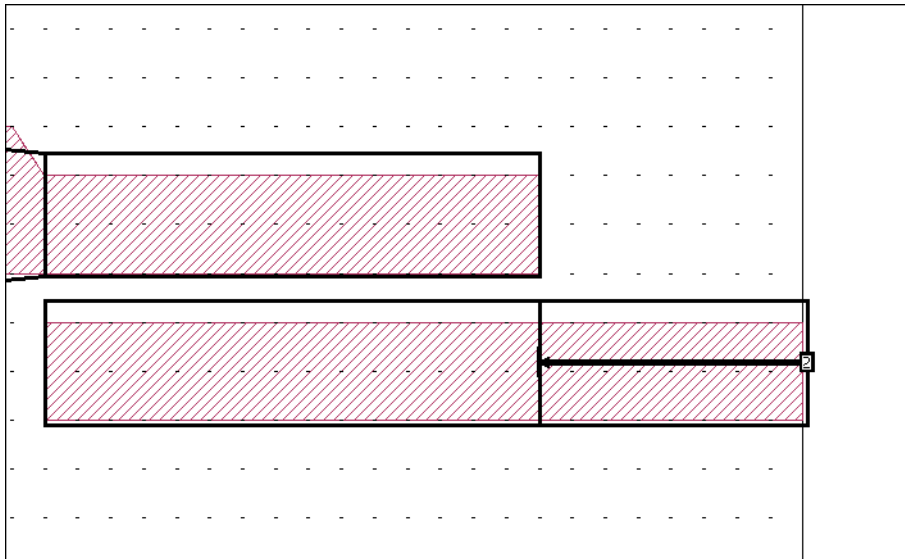
Now check to see how *em* will subsection your circuit. To do this, make sure the cell fill is turned “On” using *View* \Rightarrow *Cell Fill* or Ctrl-M. The cell fill shows you how *em* will subsection your circuit. If you did not choose a small enough cell size or if your circuit is not oriented exactly on grid points, the metal may “short” two polygons together. This can happen in our example, see below, when the cell size is adjusted to an even coarser mesh. Using the circuit “*two_coup*” saved

previously, adjust the cell size to 4 by 5.9375 (95 by 32 cells). The result is shown below. The problem shown in the figure can be fixed by moving the entire circuit by a fraction of a cell.



Example of coupled lines shorted together. This circuit is in need of grid alignment.

In order to do this for our example, turn the cell snap off. Select *Tools* \Rightarrow *Snap Setup* from the project editor main menu. Then click on the No Snap radio button in the Snap Setup dialog box. Now, using *Edit* \Rightarrow *Select All*, select the entire circuit and move it so that the circuit is aligned to the grid points. Don't worry about getting the circuit exactly aligned. Notice that the polygons are no longer "shorted" together, see below.

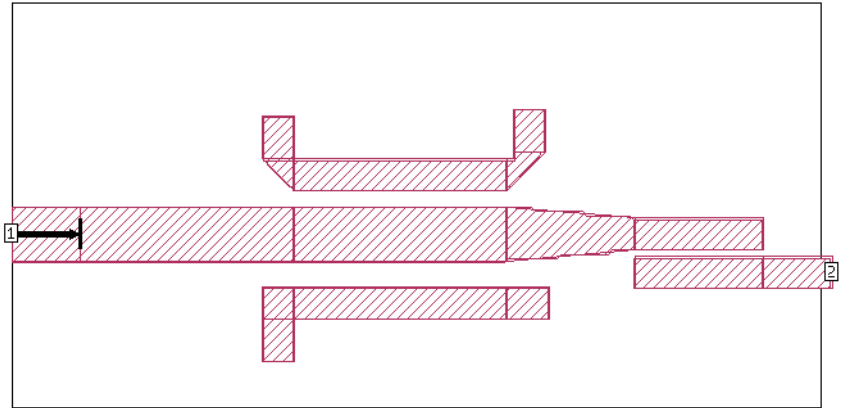


Example of coupled lines no longer shorted together.

Move and add points as needed

At this point you may want to snap all of the polygons to grid points. For this example, since the cell mesh is too coarse, do not snap this circuit. Instead reload the file "*two_coup.son*", discarding the edits performed in the last step. Using the reloaded circuit, modify the cell size to 2 by 2 mils and make sure that the three

diagonal polygons all have Diagonal fill. This modified circuit is shown below. Note that port 2 extends past the box wall and that some polygons are offset slightly.

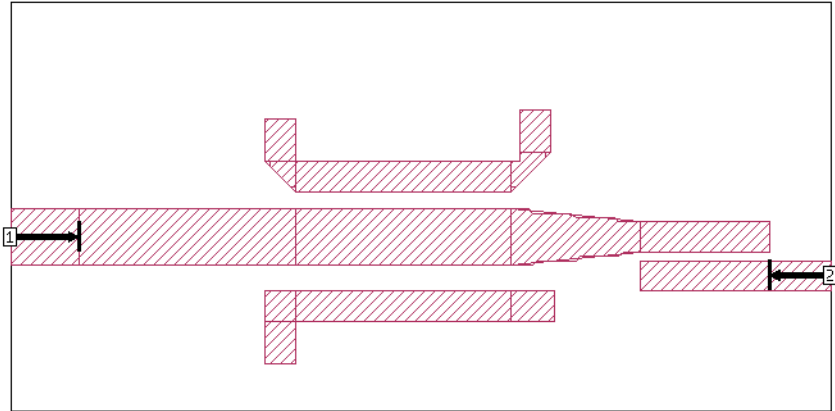


Example file “two_coup.son’ after modifications.

Some points may now need to be moved in order to get the circuit to sit on the grid. This will be done with a cell snap. Before you do the snap, you may want to save your file. Select all of the polygons. Now select *Modify* \Rightarrow *Snap to*. Click on the X and Y button in the Cell Size section of the Snap to dialog box. If desired you can select only parts of the circuit to snap to the grid points. Now select *Tools* \Rightarrow *Reshape* to place the project editor in reshape mode which allows you to select individual points on polygons. Select the two points near port 2. Select *Modify* \Rightarrow *Snap to* and click on the Right box wall radio button to snap these points to the right substrate edge. The reference plane extension will now reappear indicating that the port is now properly placed.

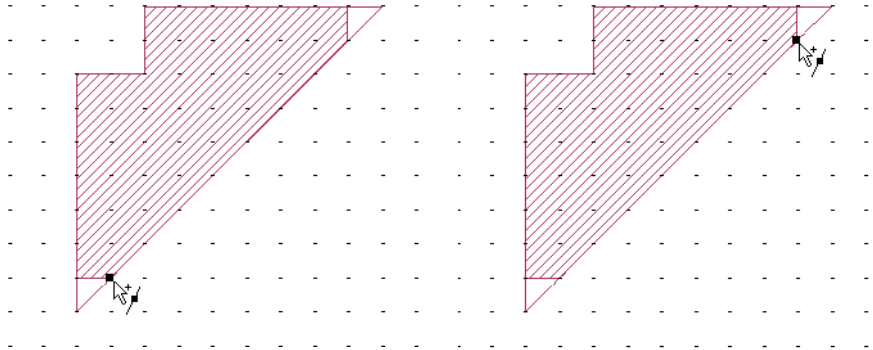
Note that the reference plane is situated too far into the circuit (by the amount that the substrate size was reduced). This can be adjusted by using *Circuit* \Rightarrow *Ref. Planes/Cal. Length*. Click on the Right box wall in the list at the top of the dialog

box. Then click on the Fixed Length radio button. Click on the Mouse button to use your mouse to select the length. Then click on the edge of the coupled line. The modified circuit is shown below.



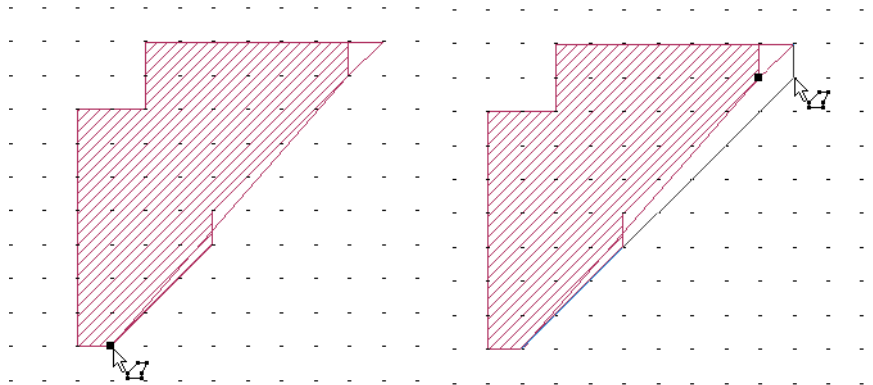
Example file “two_coup.son” after snap, port move and reference plane shift.

Some polygons, such as mitred corners and narrow lines, may need to have points added to help with Diagonal fill. Select and move the right most mitered corner away from the connecting lines in order to allow some working room. Now use *(Shift)Tools* \Rightarrow *Add Points* and *Tools* \Rightarrow *Reshape* to add a one cell extension to the connecting edges of the bend. See the illustrations below.



1) Adding First Point

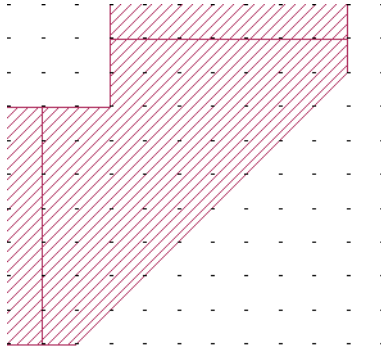
2) Adding Second Point



1) Moving First Point

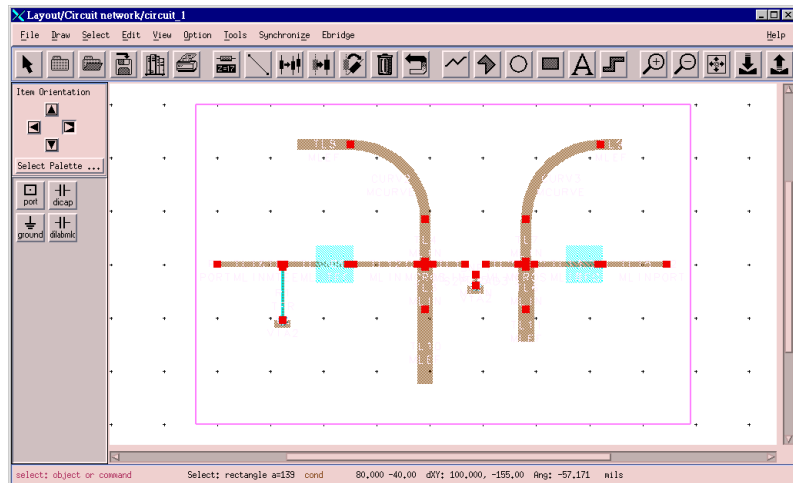
2) Moving Second Point

Note how the diagonal fill now fills the polygon completely. Next, select the modified mitered bend and move it back into position. The figure below shows the new mitered bend in place.



Using a drawn bounding box

A bounding rectangle can be drawn in the layout window to exercise exact control over the box dimensions. This process is described in more detail in Chapter 16 "The Agilent Interface" on page 279 in the **Sonnet User's Guide**. Load the design "*circuit_1*" from the ADS project directory. The *circuit_1* example already has a bounding rectangle drawn on layer 13 (bound), as pictured below.



Example "*circuit_1*" showing the use of a bounding rectangle.

The dimensions of this rectangle are 465 by 300 mils. For UNIX systems, open a command window by selecting *Ebridge* ⇒ *Launch User Window* from the menu of the layout window. For Windows systems, open a text editor, such as NotePad. Edit the file *sonnet.prf* and change the *NUM_X_GRIDS* and *NUM_Y_GRIDS* parameters to 465 and 300 respectively. Now change the *EXT* parameter to 0. Note that the drawn rectangle layer (13) is already specified as being ignored. Save the changes to *sonnet.prf* and translate the example by selecting *Ebridge* ⇒ *Rep* ⇒ *Geo* from the layout window menu. Open the circuit in the project editor by selecting *Launch XGEOM* from the *Ebridge* menu. The translated circuit has a box of the exact size specified by the drawn rectangle (465 by 300 mils) and a cell size of 1 by 1 mil.

Adjustments that may be needed

In addition to the items described above, additional adjustments to the circuit may be required. This depends upon the level of preparation taken prior to translation and the type of circuit being translated. The items that might require adjustment are:

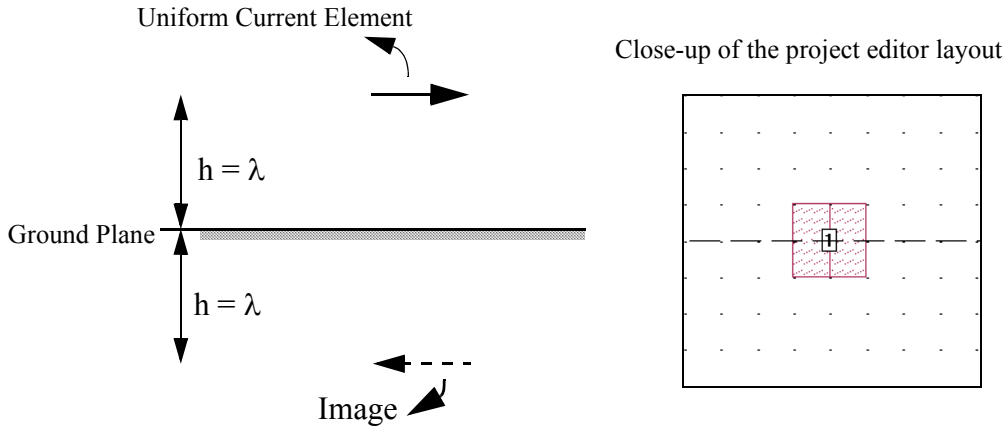
- **Ports and reference planes:** The user might want a port to be in the middle of the circuit rather than extended to the edge. For this an auto-grounded port must be used. These kinds of ports are explained in Chapter 6, “Ports” in the **Sonnet User’s Guide**. Also, translated reference planes might not end up exactly where a user wants them and may need to be set to a more appropriate spot.
- **Metalization and dielectric layer parameters:** In cases where special metalization is required or multiple types of TFR, or TFC instances are used, some adjustment of the metalization and dielectric layers will be necessary. Use *Circuit* \Rightarrow *Dielectric Layers* or *Circuit* \Rightarrow *Metal Types* to adjust and/or add layers and metalization. To change the metal type of a polygon, select them and then use *Modify* \Rightarrow *Metal Properties*. To move polygons to different levels, select them and use *Edit* \Rightarrow *Cut*. Then move to the correct level and use *Edit* \Rightarrow *Paste* to place the objects on the correct level.
- **Vias:** In cases of custom instances, vias may need to be added to the circuit to connect different layers since the Agilent interface software does not include them automatically.

Chapter 8 A Two-Dimensional Far Field Viewer Tutorial

This tutorial describes an example of using the far field viewer to display two-dimensional plots. The far field viewer displays far field radiation patterns using the current density data created during an *em* analysis. In this example, we analyze an infinitesimal dipole antenna above a ground plane, shown below, and compare the results to the exact theoretical antenna pattern shown on page 164, as provided by reference 2.

For more information about modeling antennas and using the far field viewer, please refer to Chapter 21 "Antennas and Radiation" on page 343 in the **Sonnet User's Guide**.

Although this example is not very practical, it is a good example to use for validation because of its simplicity. The infinitesimal electric dipole is placed one wavelength (300 mm at 1 GHz) above the ground plane (an electric field reflection boundary).



Creating an Antenna Pattern File

This tutorial uses an infinitesimal dipole one wavelength above the ground plane. The project, [Infpole](#), is provided in the Sonnet example files. If you do not know how to obtain a Sonnet example, select *Help* \Rightarrow *Examples* from any program menu, then click on the **Instructions** button.

1 Save a copy of “infpole.son” to your working directory.

The file “infpole.son” is the circuit geometry project file for the dipole antenna which was created using the project editor. The dipole geometry can be viewed by using the project editor.

To allow Sonnet Level2 users to view this example, this project was analyzed with de-embedding enabled and the resulting data was included in the project file. However, analyzing with de-embedding enabled results in non-physical de-

embedded S-parameters (the non-de-embedded S-parameters are still valid). This is because the de-embedding algorithm analyzes calibration standards (through lines) with the same dimensions as the polygons connected to the port. In this case, because the distance to the ground plane is a whole wavelength, the calibration standards will contain higher-order transmission line modes. This violates the de-embedding assumptions, and results in incorrect S-parameters (see Chapter 8, “De-embedding Guidelines” in the **Sonnet User’s Guide**). However, since the far field viewer always uses the non-de-embedded results to calculate the far field, the far field calculations are valid with or without de-embedding.

It is important to remember that in order to produce data for input into the far field viewer, the Compute Current Density option must be selected in the Analysis Setup dialog box in the project editor.

Infpole was analyzed at a linear frequency sweep from 0.8 GHz to 1.2 GHz in intervals of 0.1 GHz.

Running the Far Field Viewer



- 2 **Click on the View Far Field button on the Sonnet task bar to invoke the far field viewer.**

A pop-up menu appears on your display.

- 3 **Select “Browse for Project” from the pop-up menu.**

A browse window appears on your display.

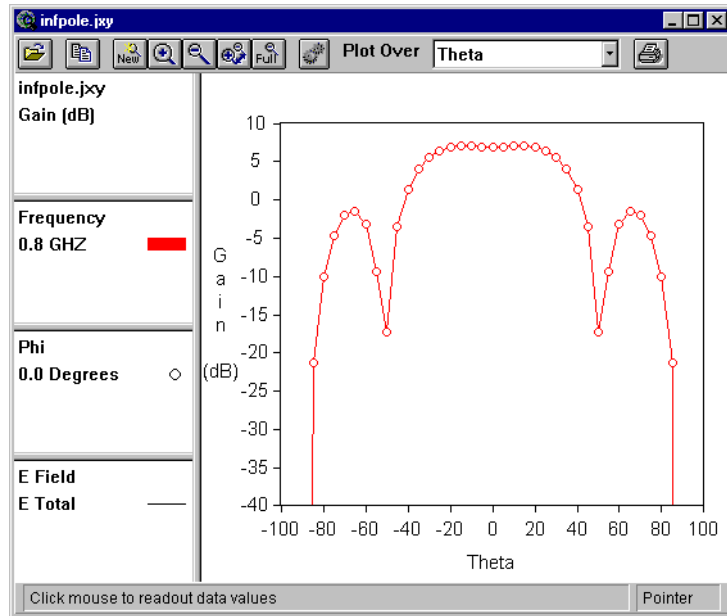
- 4 **Using the browse window, select your saved copy of “inpole.son”.**

The far field viewer window opens on the project file, “inpole.son”.

After the initial calculation is complete, a plot appears on your display as shown below. When a new file is opened, the far field viewer performs an analysis on the first frequency based on a default set of values for directions, port excitations and terminations and displays the Gain (dB) versus theta for the first value of phi. The calculation defaults are as follows:

- There are two values of phi: 0° and 90°

- Theta ranges from -90° to $+90^\circ$ in 5° intervals.
- Port 1 is set to a 1.0 V source magnitude with a $50.0\ \Omega$ load



The far field viewer display defaults to a cartesian plot with theta selected on the X-axis. The polarization defaults to *Theta/Phi*. The Y-axis is set to display the Gain (in dB) of the pattern response and is normalized to power gain of the ideal isotropic antenna.

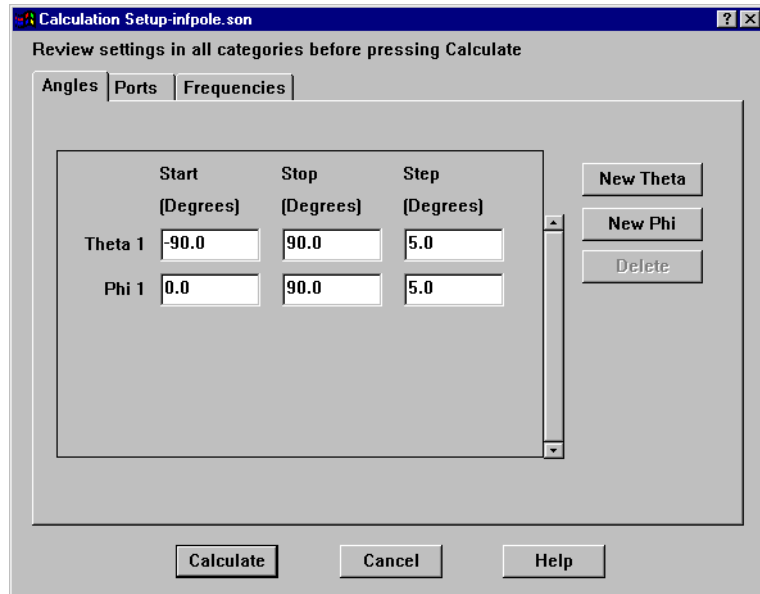
To change the calculation and display defaults, see File - Preferences in the far field viewer's help.

Calculating the Response

As mentioned above, when the far field viewer is invoked, the response data is calculated for only the first frequency in the current response file. To calculate data for the other frequencies at additional angles, perform the following:

- 5 Select *Graph* \Rightarrow *Calculate* from the far field viewer main menu.

The Calculation Setup dialog box appears on your display with the Angles tab selected as shown below.



Selecting Phi Values

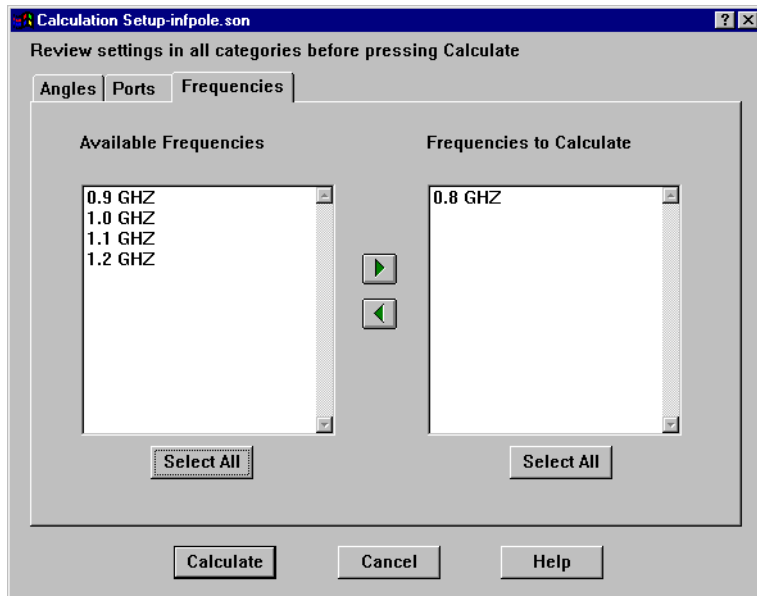
- 6 Enter 0 in Start text entry box, 90 in the Stop text entry box and 5 in the Step text entry box of the Phi line.

This analyzes data points from $\phi = 0^\circ$ to 90° in intervals of 5° .

Selecting Frequencies

- 7 Click on the Frequencies tab in the Calculation Setup dialog box.

The Frequencies tab is now displayed, as shown below.



- 8 Click on the Select All command button under the Available Frequencies.

All of the frequencies are highlighted.

- 9 Click on the Right Arrow button.

This moves all the selected frequencies to the Calculated Frequencies column.

- 10 Click on the Calculate command button.

There is a delay while the far field viewer calculates the requested data. The calculations for each frequency are performed using the defaults cited above for phi, theta, port excitation and exclusions, since none of these items were changed before selecting Calculate.

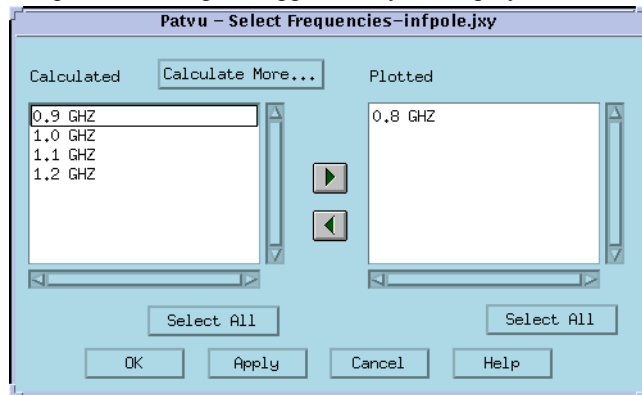
A status box appears on your display to provide updates on calculation progress. The display is updated when the calculation is completed. Be aware that for larger, more complicated circuits, this delay might be a considerable one.

Selecting the Response

The far field viewer allows you to select which data items you wish displayed at any given time. In the next session, you display 1 GHz data at different phi's.

- 11 Select *Graph* \Rightarrow *Select* \Rightarrow *Frequencies* from the far field viewer main menu.

The Select Frequencies dialog box appears on your display.



The Calculated column displays the frequencies for which data has been calculated, but is not presently displayed. The Plotted Column shows those frequencies which are presently displayed. In this case, 0.8 GHz.



TIP

You may also open the Select Frequencies dialog box by right-clicking in the Frequency area of the legend and selecting *Select* from the menu which appears on your display.

12 Double-click on 0.8 in the Plotted column.

This moves 0.8 to the calculated column, i.e., this frequency is not displayed.

13 Double-click on 1.0 in the Calculated column.

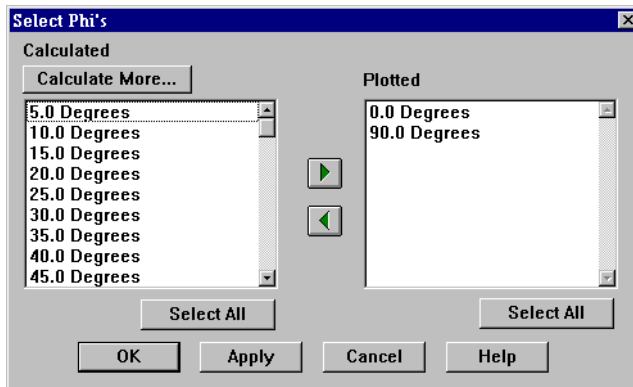
This moves the value 1.0 to the Plotted column.

14 Click on the OK command button.

This closes the dialog box and updates the display with the data for 1.0 GHz at $\Phi = 0^\circ$.

15 Select *Graph* \Rightarrow *Select* \Rightarrow *Phi* from the far field viewer main menu.

The Select Phi's dialog box appears on your display.



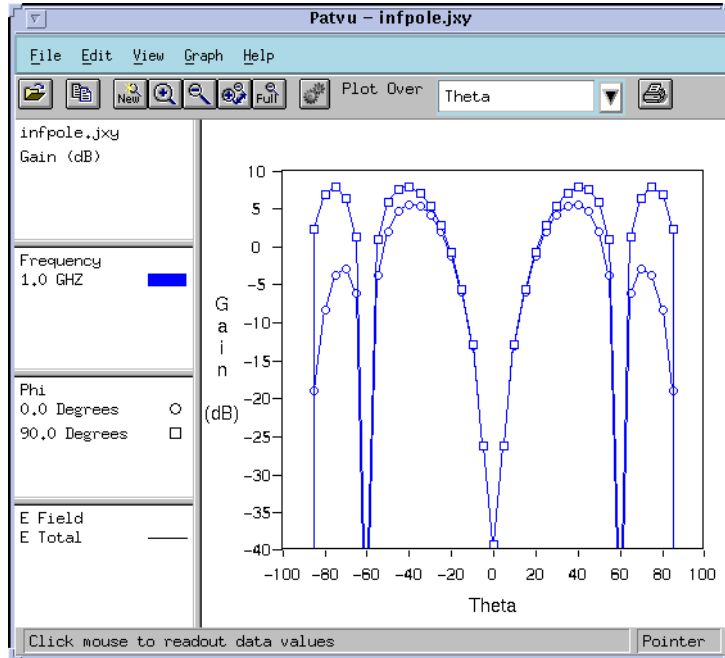
16 Use the scroll bar on the Calculated Column to move down the list until 90.0 Degrees is displayed.

17 Double-click on 90.0 Degrees to move this value to the Plotted column.

Values in this column are displayed in the far field viewer.

18 Click on the OK command button.

The dialog box disappears and the far field viewer display is updated. It should appear similar to the figure below.



The plot is drawn showing two curves: E_{total} at $\phi = 0$ and 90 degrees. The upper curve is the radiation pattern at $\phi = 90$ degrees. The lower curve is the radiation at $\phi = 0$ degrees.

The far field viewer automatically selects an appropriate scale for the plot.

19 Select *Graph* \Rightarrow *Select* \Rightarrow *Phi* from the far field viewer main menu.

The Select Phi's dialog box appears on your display.



TIP

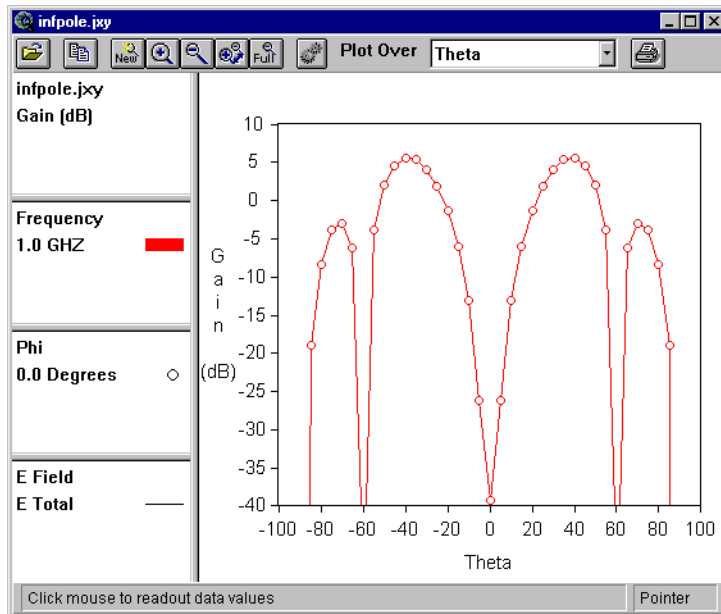
You may also access the Select Phi's dialog box by right-clicking in the Phi box in the legend and selecting *Select* from the pop-up menu which appears.

20 Double-click on 90.0 degrees in the Plotted column to move it from the Plotted column to the Calculated column.

This removes 90.0 degrees from the plot.

21 Click on the OK command button.

The dialog box disappears and the far field viewer plot is updated. It should appear as below.



Zooming

The zoom button, located on the tool bar, may be used to magnify a specific area in the plot.



- 22 Click on the Zoom In button on the Tool bar.**

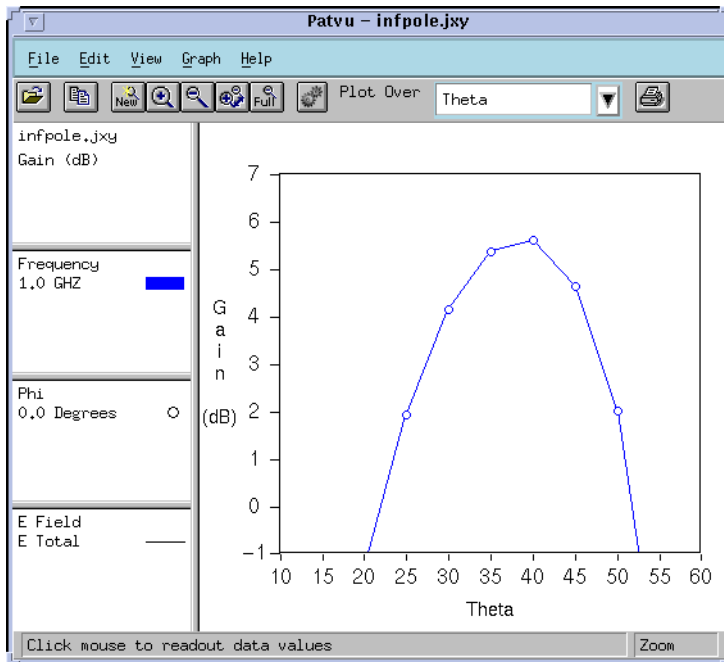
For this example, zoom in on the area from 0 to 10 dB in Gain, where Theta ranges from 20° to 50° .

You may also use *View* \Rightarrow *Zoom In* or the Space Bar for the zoom function.

- 23 Click on the point in the plot corresponding to 0 dB Gain and Theta = 20° , then drag the mouse to the point in the plot corresponding to 10 dB Gain and Theta = 50°**

A rubber band surrounding the area to be magnified follows the mouse.

When the mouse is released, the plot is updated with a magnified view of the selected area, as shown below.



24 Click on the Full View button on the Tool bar.

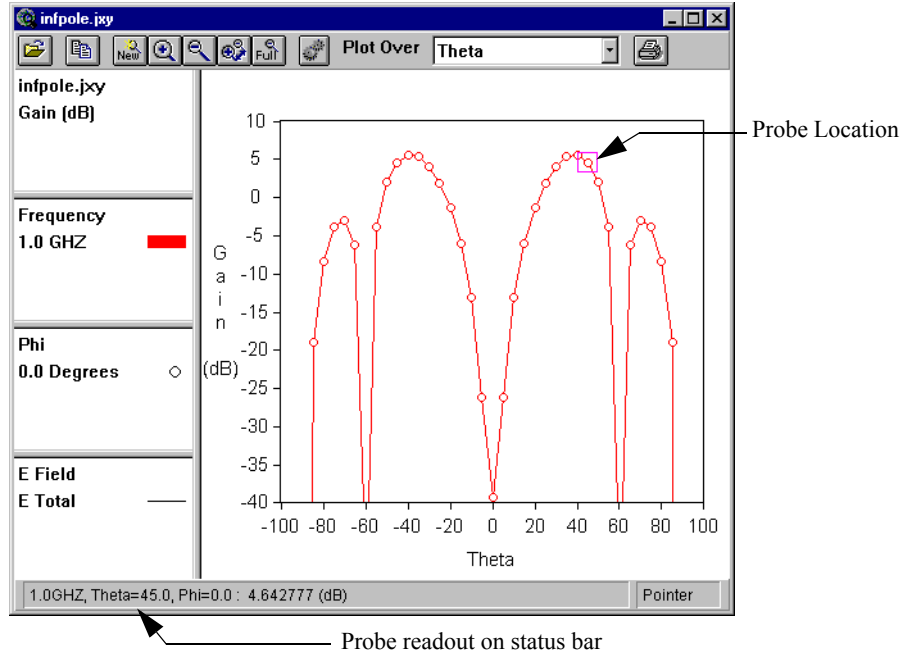
The full plot once again appears on your display.

Probing the Plot

To evaluate the pattern response at any location in your plot you simply click at the desired location.

25 Click on the $\theta = 45^\circ$ point on the plot.

A square appears around the point, as shown below. The readout for the point including the frequency, value of θ and ϕ , and the gain appear in the status bar at the bottom of the far field viewer window.



26 Press the left arrow key, \leftarrow , to move to the $\theta = 40^\circ$ point on the plot, or alternately, click on that point.

The probe box now appears at that point and the data is updated in the status bar.

Note that if there is more than one data curve displayed, the up and down arrow keys, \uparrow and \downarrow , would move the data probe between curves, while the left and right arrow keys, \leftarrow and \rightarrow , move between data points on any given curve.

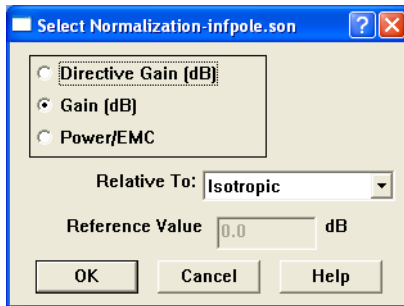
Re-Normalizing the Plot

By default, the far field viewer displays the power gain. The power gain is defined as the radiation intensity divided by the uniform radiation intensity that would exist if the total power supplied to the antenna were radiated isotropically^[1].

We shall now normalize the plot to the maximum value.

27 Select *Graph* \Rightarrow *Normalization* to change the normalization.

The Select Normalization dialog box appears on your display.

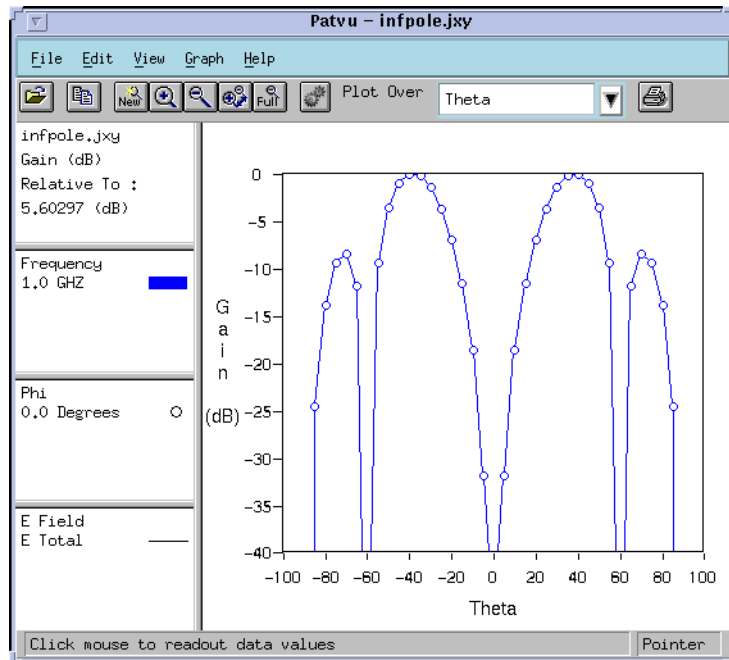


28 Select Max from the Relative To drop list.

This selects the maximum value of radiation for the plot to be the 0 dB point of the plot.

29 Click on the OK command button.

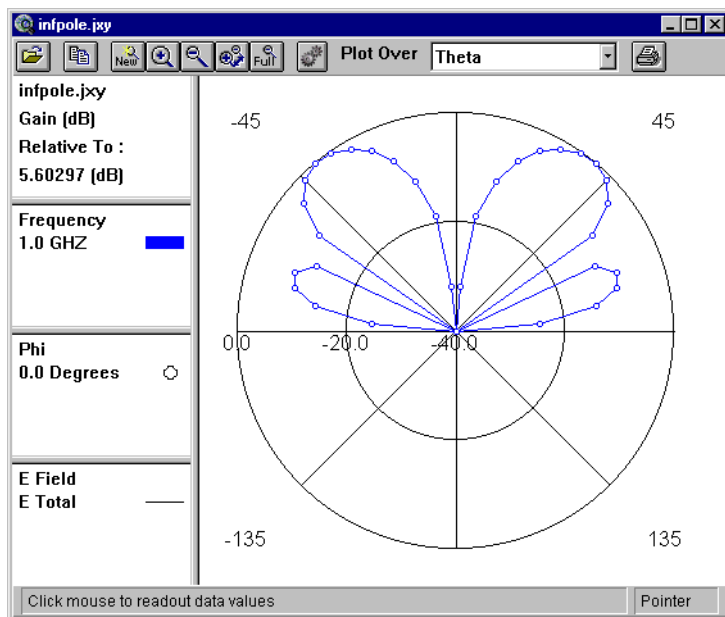
The dialog box is closed and the display is updated with the data normalized to the maximum value, which in this case is 5.60297 dB.



Changing to a Polar Plot

30 Select *Graph* \Rightarrow *Type* \Rightarrow *Polar* to select a polar plot for the display.

A polar plot is chosen since the theoretical data for an infinitesimal dipole is shown in a polar plot. The display is updated using the polar coordinate system. Phi is held constant and theta is swept.



TIP

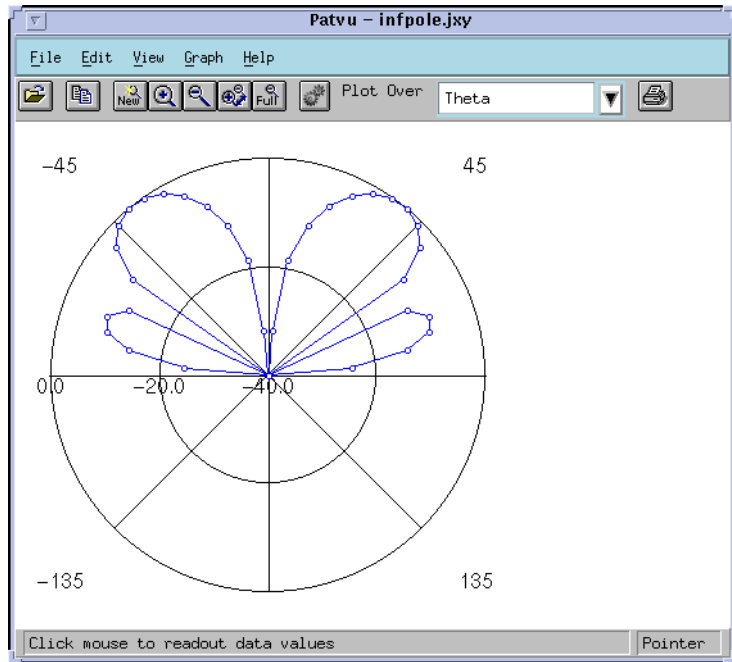
You may select another type of plot by right-clicking in the plot title area of the far field viewer display and selecting *Type* from the pop-up menu which appears.

Turning Off the Legend

Since the legends take up a lot of space on the display, you may turn them off, allowing the plot to fill the extra space.

31 To turn off the legend, select *View* \Rightarrow *Legend*.

This turns “off” the legend and the far field viewer redraws the plot without the legends. The menu item toggles the display state of the legend, so that selecting *View* \Rightarrow *Legend* again displays the legend.

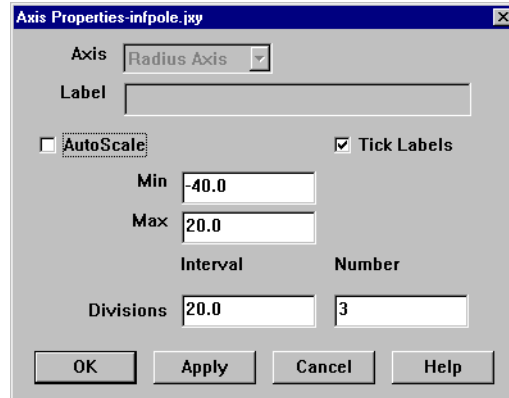


Changing the Radius Axis

You can change the radius axis limits of the plot to another value. For this example, you will change the intervals from 20 dB to 10 dB.

- 32** Select *Graph* \Rightarrow *Axes* from the main the far field viewer menu.

The Axes Properties dialog box appears on your display.



- 33** Click on the AutoScale checkbox to turn it “off.”

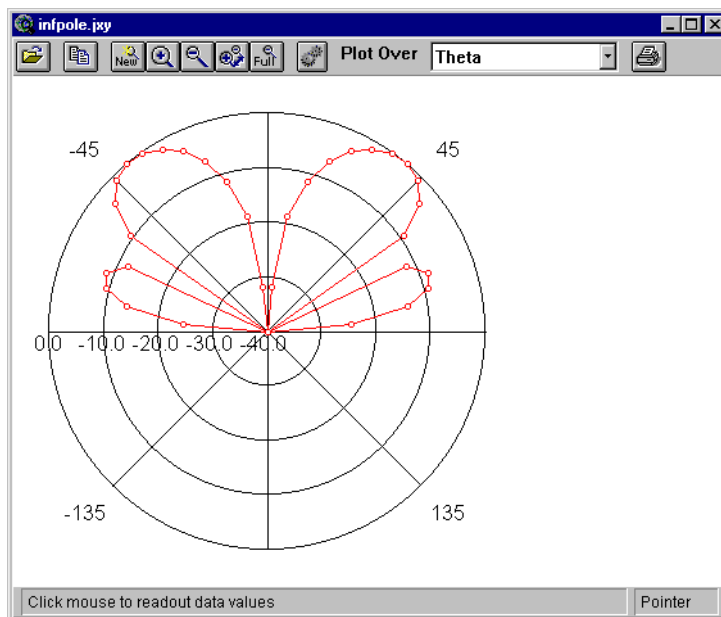
This enables the Min and Max text entry boxes under Autoscale and the Interval and Number text entry boxes under Tick Labels.

- 34** Enter “10” in the Interval text entry box.

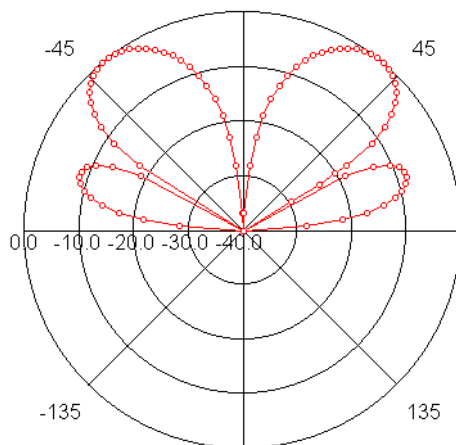
This sets the intervals on the plot grid to 10 dB.

35 Click on the OK command button.

The dialog box disappears and the far field viewer display is updated with the new interval value for the axes. Your display should be similar to the one shown below.



Shown above is the far field viewer calculated far field antenna pattern for the very short dipole in the file infpole.son. The result should be compared with theoretical result in the next figure.



Exact far field antenna pattern from reference [2] of an infinitesimal dipole antenna one wavelength above a ground plane.

Selecting a Frequency Plot

To see how the antenna pattern changes with frequency, you use a frequency plot. Before you can select a frequency plot, you must return to a cartesian plot.

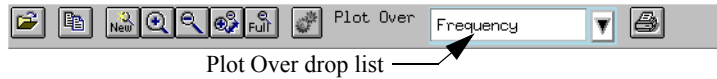
- 36 Select *Graph* \Rightarrow *Type* \Rightarrow *Cartesian* from the far field viewer main menu.**

Your display is updated with a cartesian plot. Note that the autoscale is automatically turned back on when you switch plot types.

- 37 Select *View* \Rightarrow *Legend* from the far field viewer main menu.**

The legend once again appears in your display.

- 38** Select **Frequency** from the **Plot Over** drop list on the far field viewer tool bar.



- 39** Select **Graph** \Rightarrow **Select** \Rightarrow **Theta** from the far field viewer main menu.

The Select Theta's dialog box appears on your display.

- 40** Double-click on **-90.0 degrees** in the **Plotted** list.

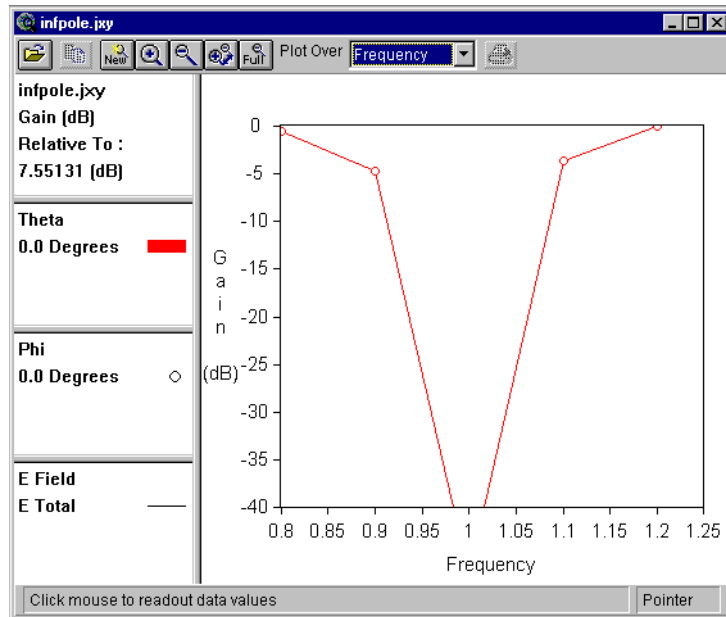
This value moves to the Calculated list which removes it from the display.

- 41** Double-click on **0.0 degrees** in the **Calculated** list.

This value moves to the Plotted list which adds it to the display.

42 Click on the OK command button.

The display is updated with a frequency plot as shown below. Note that the gain is now calculated relative to 7.55131 dB, since the Normalization is relative to Max and this value is the maximum value of radiation for this plot.



43 Select *Graph* ⇒ *Select* ⇒ *Phi* from the far field viewer main menu.

The Select Phi's dialog box appears on your display.



TIP

You may also invoke the Select Phi's dialog box by right-clicking on the Phi box in legend and selecting *Select* from the pop-up window.

44 Double-click on 0.0 degrees in the Plotted list.

This value moves to the Calculated list which removes it from the display.

45 Double-click on 90.0 degrees in the Calculated list.

This value moves to the Plotted list which adds it to the display.

46 Click on the OK command button.

The dialog box disappears and the display window is updated.

47 Select *Graph* \Rightarrow *Select* \Rightarrow *Theta* from the far field viewer main menu.

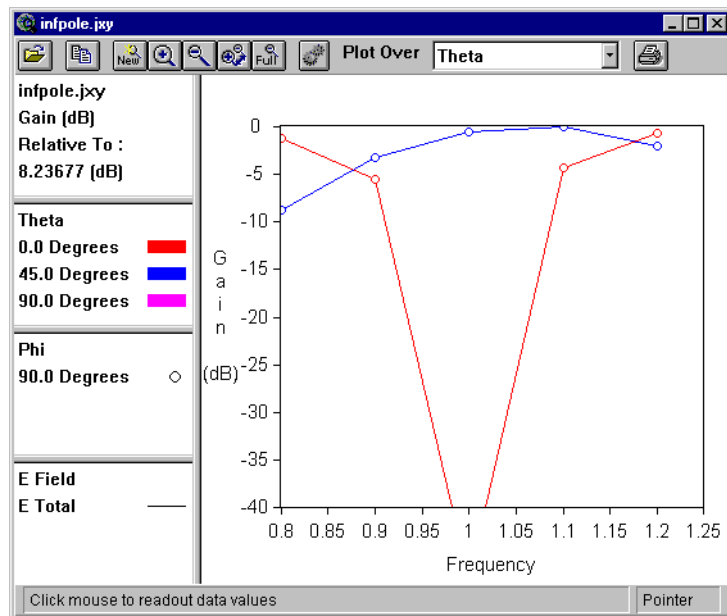
The Select Theta's dialog box appears on your display.

48 Double-click on 45.0 degrees and 90.0 degrees in the Calculated list.

These values move to the Plotted list which adds these values to the display.

49 Click on the OK command button.

The dialog box disappears and the display is updated as shown below.



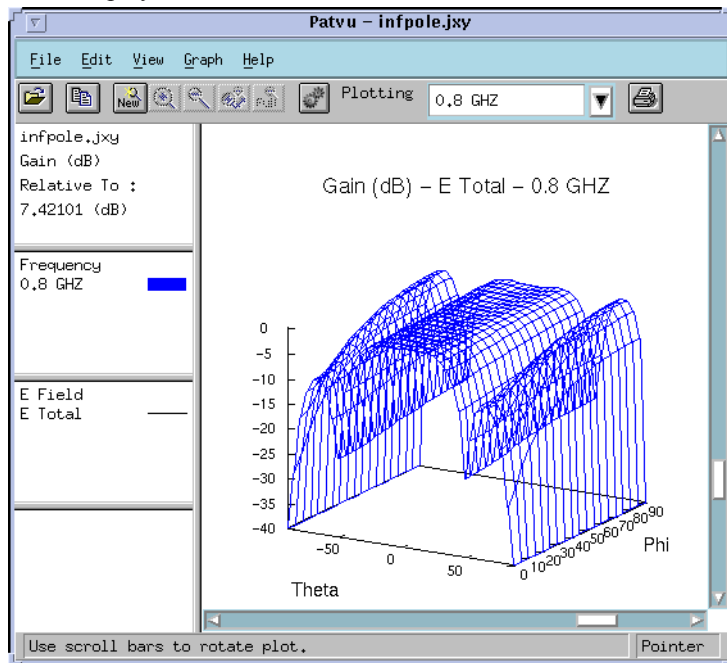
Notice that E-Total at Theta = 90 is shown in the legends, but does not appear on the graph. This occurs because the magnitude is too small to show on the plot.

Viewing a Surface Plot

The surface plot shows all the calculated values of theta and phi plotted against the gain for a single frequency.

- 50** Select *Graph* \Rightarrow *Type* \Rightarrow *Surface* from the main menu.

Your display is updated with a surface plot with the first frequency, 0.8 GHz, selected for display.



Saving the Far Field Viewer File

- 51** Select *File* \Rightarrow *Save* from the far field viewer main menu.

The file is saved to the same filename with a ".pat" extension, i.e., "infpole.pat". This saves any data calculated during this the far field viewer session.

Exiting the Far Field Viewer Program

This is the end of the first example of using the far field viewer.

52 To stop the program, select *File* \Rightarrow *Exit*.

The far field viewer window disappears from your display.

References

- [1] Simon Ramo, John R. Whinnery and Theodore Van Duzer, *Fields and Waves in Communication Electronics*, John Wiley & Sons, Inc. 1994, pg. 601.
- [2] Constantine A. Balanis, ***Antenna Theory Analysis and Design***; New York; Harper & Row, 1982, section 4.7.3

Index

A

- Add Subdivider 50
- Adjustments
 - circuits 144
 - dielectric layer parameters 144
 - metalization 144
 - ports 144
 - reference planes 144
 - vias 144
- analysis
 - starting 26
- analysis frequencies 23
- analysis monitor 26
- Analysis Setup dialog box 22, 32
- analyze 26
- anchored parameters
 - adding 13
- antennas
 - pattern 146
- autoscale 162
- axes 162
- Axis Properties dialog box 162

B

- bandpass filter 12
- best iteration 40
- Bounding rectangle 143

C

- calculate 149
- calculating the response 148
- Calculation Setup dialog box 149
- calculations
 - defaults 147
 - status of 151
- cartesian plot 148, 164
- Cell fill 136
- Cell size 131
- cell size 110, 117, 119
- Cell snap 139
- circuit alignment 119
- circuit subdivision
 - additional improvements 60
 - comparison of results 60
 - executing 54
 - feedlines 56
 - reference planes 56
 - subproject naming conventions 56
 - tutorial 47–62
- Circuit Subdivision dialog box 55
- Commands
 - reload_ebridge() 124
- Cross.geo 131
- Cross.son 131

D

- dcblock 103

dcblock.dxf 101

Designs

- circuit 143

- cross 130

- step 126

- two_coup 135

Diagonal fill 134, 141

diagonal fill of polygons 118

dialog box

- Parameter Properties 15

dialog boxes

- Analysis Setup 22, 32

- Axis Properties 162

- Calculation Setup 149

- Circuit Subdivision 55

- Import Control 104

- Import Options 115

- Layer Mapping 113

- Optimization Goal Entry 36

- Optimization Parameters 33

- Optimization Results 43

- Parameter Sweep Entry 23

- Select Frequencies 151

- Select Normalization 158

- Select Phi's 152, 166

- Subdivider Orientation 50

- Subproject Specifications 56

dielectric

- defining 115

Dielectric layer parameters

- adjustments 144

dipole

- infinitesimal 145, 146, 164

E

E total 153

ebads_prj 123

em

execution time 116

E-Total 167

example files 101

- dcblock 103

- ebads_prj 123

- filtwall 73

- infpole 163

- obtaining 73

- par_dstub 11

- sub_whole 59

- subdivide 48

excitations

- default values 148

exit 169

exiting 169

EXT 143

F

far field 164

Far Field Viewer

- exiting 169

- saving a file 168

- tool bar 165

- tutorial 145

feedlines 56

file

- exit 169

- save 168

Files

- cross.geo 131

- cross.son 131

- sonnet.prj 143

- step.jxy 129

- step.prj 126

- two_coup.geo 138

Fill

- cell 136

filter 12

filtwall 73
 frequencies
 selecting to calculate 150
 selecting to plot 151, 165
 frequency plot 164
 frequency specification 23
 full view button 156

G

gain
 relative to
 max 158
 goals 36
 graph
 axes 162
 calculate 149
 normalization 158
 select
 frequencies 151
 phi 152, 154, 166
 theta 165, 167
 type
 cartesian 164
 polar 160
 surface 168
 grid 120
 grid points 119, 120
 ground plane 145, 164

I

Import Control dialog box 104
 Import Options dialog box 115
 infinitesimal dipole 145, 146, 164
 infpole 163
 Interchange 4
 interdigital 12
 iteration 40
 iterations

 selecting for display 40

L

Launch XGEOM 126, 143
 layer mapping 113
 Layer Mapping dialog box 113
 legend
 selecting frequencies 151
 selecting phi's 154, 166
 turning off 160
 view 161, 164
 level mapping 103
 level number 111, 113

M

main netlist
 default name 55
 mapping
 layer 113
 level 103
 material 114
 Max 158
 Metal fill 136
 Metalization
 adjustments 144
 metallization
 defining 115
 microstrip 12

N

network file
 analysis 58
 nominal value
 changing 32
 nominal values
 update with results 43
 normalization 158
 default 148

NUM_X_GRIDS 143

NUM_Y_GRIDS 143

O

object type 114

opening a graph 27

optimization

- best iteration 40

- data 39

- data range for parameters 34

- example 11, 31

- executing 38

- results 42

- selecting parameters for 34

- setting up 32

- specifying goals 36

- tutorial 11, 31

Optimization Goal Entry dialog box 36

Optimization Parameters dialog box 33

optimization results

- update 43

Optimization Results dialog box 43

optimized values 42

P

par_dstub 11

Parameter Properties dialog box 15

parameter sweep 11

- analysis frequencies 23

- data 39

- setting up 22

- viewing the response 27

Parameter Sweep Entry dialog box 23

parameterization

- tutorial 11

Parameters

- EXT 143

- NUM_X_GRIDS 143

- NUM_Y_GRIDS 143

parameters

- adding 13, 18

- data range 24

- data range for optimization 34

- example 11

- first reference point 18

- nominal value 32

- second reference point 20

- selecting for optimization 34

- selecting for parameter sweep 24

pattern 146

pattern response 156

phi

- default values 147

- selecting values to plot 152, 154, 166

- specifying range for calculation 149

plot

- cartesian 148, 164

- frequency 164

- polar 160

- probing 156

- selecting type 160

- surface 168

- title area 160

- two-dimensional 145

Plot Over drop list 165

polar 160

polarization

- default 148

Polygon fill 134

Polygons

- changing 135

- diagonal fill 134

- staircase fill 134

polygons

- changing of 118

- deletion of several 116

- diagonal fills 118
- shorted 120
- staircase fill 118
- port
 - excitations 148
 - terminations 148
- Ports
 - adjustments 144
- ports
 - adding 122
- probe readout 157
- probing the plot 156
- Project Editor 146
- project file
 - circuit geometry 146

R

- radius axis
 - changing 161
- Reference planes
 - adjustments 144
- reference planes
 - adding 122
- reference point 18, 20
- references 164, 169
- reflection boundary 146
- Rep => Geo 125
- response data
 - calculating 148
 - selecting 151
- response viewer
 - invoking 27
- results of optimization
 - accepting 42
- right-clicking 151, 160, 166

S

- saving a file 168

- scroll bar 152
- select
 - phi 154, 166
 - theta 165, 167
- Select Frequencies dialog box 151
- Select Normalization dialog box 158
- Select Phi's dialog box 152, 166
- selecting the response 151
- shorted polygons 120
- size of substrate 117
- skin effect coefficient 116
- snap 120
- Sonnet level number 111, 113
- Sonnet.prf 143
- space bar 155
- Staircase fill 134
- staircase fill
 - polygons 118
- Step 126
- Step.jxy 129
- Step.prf 126
- sub_whole 59
- subdivide 48
- Subdivide Circuit command 55
- Subdivider Orientation dialog box 50
- subdivision lines
 - adding 49
 - orientation 50
- Subproject Specifications dialog box 56
- subprojects
 - naming conventions 56
- subsections 119
- Substrate size 131
- substrate size 117
- surface plot 168
 - selecting 168
- symmetrical parameters
 - adding 18

T

terminations

 default values 148

TFC 144

TFR 144

theta

 default values 148

 selecting values to plot 165, 167

tool bar 165

 full view button 156

 zoom in button 155

Tools

 Add Subdivider 50

 Subdivide Circuit 55

transmission lines 122

Tutorial 123

tutorial 145

 optimization 11

 parameter sweep 11

Two_coup.geo 135, 136, 138

two-dimensional plots 145

type

 cartesian 164

 polar 160

 surface 168

U

units 115

update nominal values 43

V

validation 146

vertical orientation

 setting 50

Vias

 adjustments 144

vias

 adding 121

view

 legend 161, 164

 zoom in 155

viewing the response 27

X

Xgeom level number 111

xgeom level number 113

Z

zoom in 155

zoom in button 155

zooming 155