Getting Started with Ansoft Designer

New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates which are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Note that pages which are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Version
1.1	08 September 2003	1.1

Table of Contents

4. Discrete Time Domain

1

Introduction

This topic provides step-by-step instructions on how to get started in Circuit, Planar EM, and System design and simulation with Ansoft Designer. As you explore the program, you will:

- Start Ansoft Designer.
- Learn terms and concepts essential to understanding Ansoft Designer's structure and behavior.
- Explore the Ansoft Designer desktop.
- Build, analyze, and report the performance of basic designs with Ansoft Designer's Circuit, Planar EM, and System simulators.

The first step is to start Ansoft Designer.

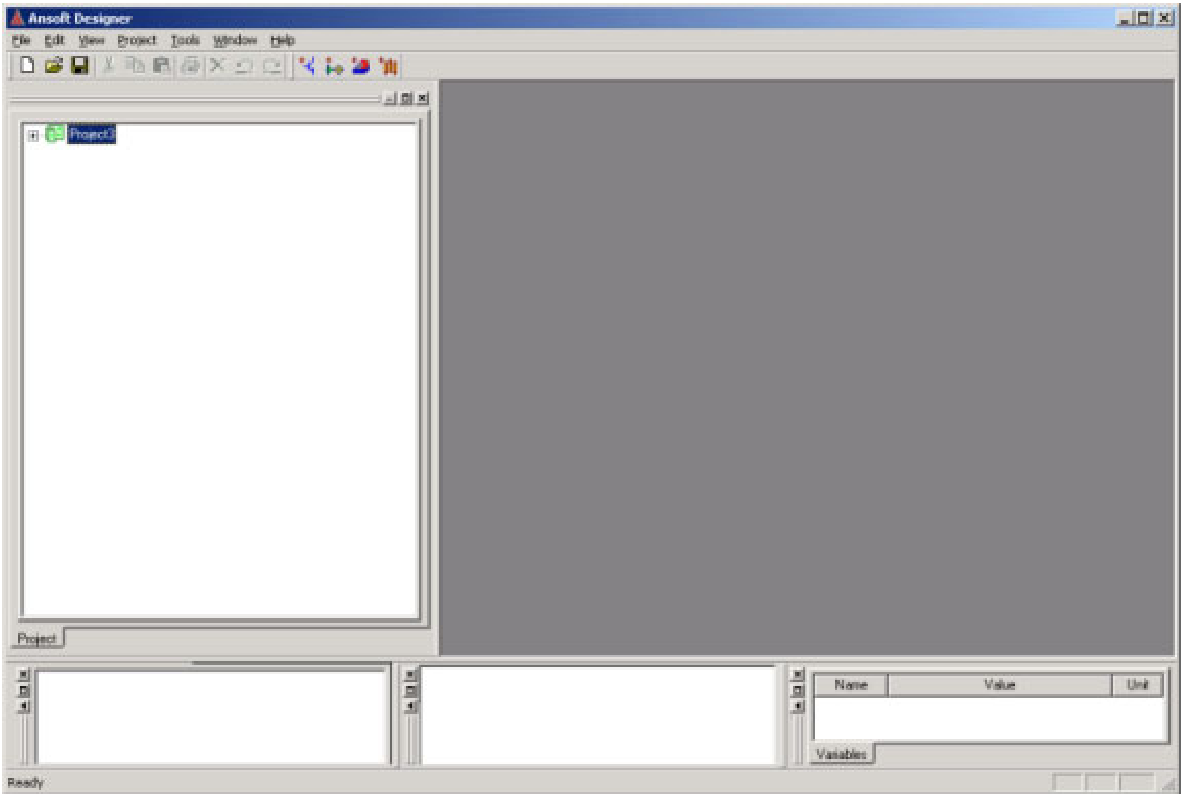
Starting Ansoft Designer

Once you have installed Ansoft Designer you can start the program from the Windows **Start** menu. To do this:

1. Click **Start**.
2. Select **Programs**.
3. Select the **Ansoft Designer** folder.
4. Select Ansoft Designer.

If the program fails to start, make sure you have installed the licensing option provided to you. See the Installation Guide for detailed information on installing the software and licenses.

Depending on your particular window configuration, you will see the Ansoft Designer desktop appear as in the picture below. The windows inside your Ansoft Designer desktop may be located differently compared to the picture.



Opening Example Projects

Ansoft Designer ships with a collection of example projects that you can load, analyze, and explore. To access these files, click **Open** on the **File** menu, and then browse to <*InstallationDirectory*>**/Designer/Examples**, where <*InstallationDirectory*> is the installation directory you specified in Ansoft Designer Setup. The examples are grouped by simulator (Circuit, Planar EM, System), with an additional group (Integrated) that demonstrates Ansoft Designer's co-simulation capabilities.

Terms Used in Ansoft Designer

It is useful to become familiar with some commonly used terms as we progress through the Getting Started guide.

Project

A project is a design or group of designs organized and managed in one file—the ***.adsn** file. Multiple projects can be opened in Ansoft Designer concurrently.

Design

Designs are the building blocks of projects, and can be circuits, systems, or 3D planar EM models. Designs consist of schematics or geometrical models, model data, analysis setup information, output graphs and tables, and other pieces of information that go into describing simulation of electrical circuits.

Component

Components are the items placed on schematics and layouts that represent low-level electrical elements and subcircuits. A component's graphical representation in a schematic is a symbol, while in a layout it is a footprint. Components have pins for connections, bitmaps in the project tree, and properties for simulation. A component can be associated with more than one simulation if it can be analyzed in more than one simulator.

Library

Components are organized into libraries. Libraries are then *configured* by the user (manually) or by loading technology files (automatically) making them available for use in building designs. System libraries are provided with Ansoft Designer. User libraries and Personal libraries are used to add foundry support, user defined models, and any custom set of components or simulation models.

Technology File

A technology file initializes a design with a set of data to avoid repeated entry of commonly used data. This data can consist of layers and stackup information for layout, configured libraries of components, and substrate definition(s) for circuit analysis. Users and foundries can customize Technology files for their own manufacturing process and simulation models.

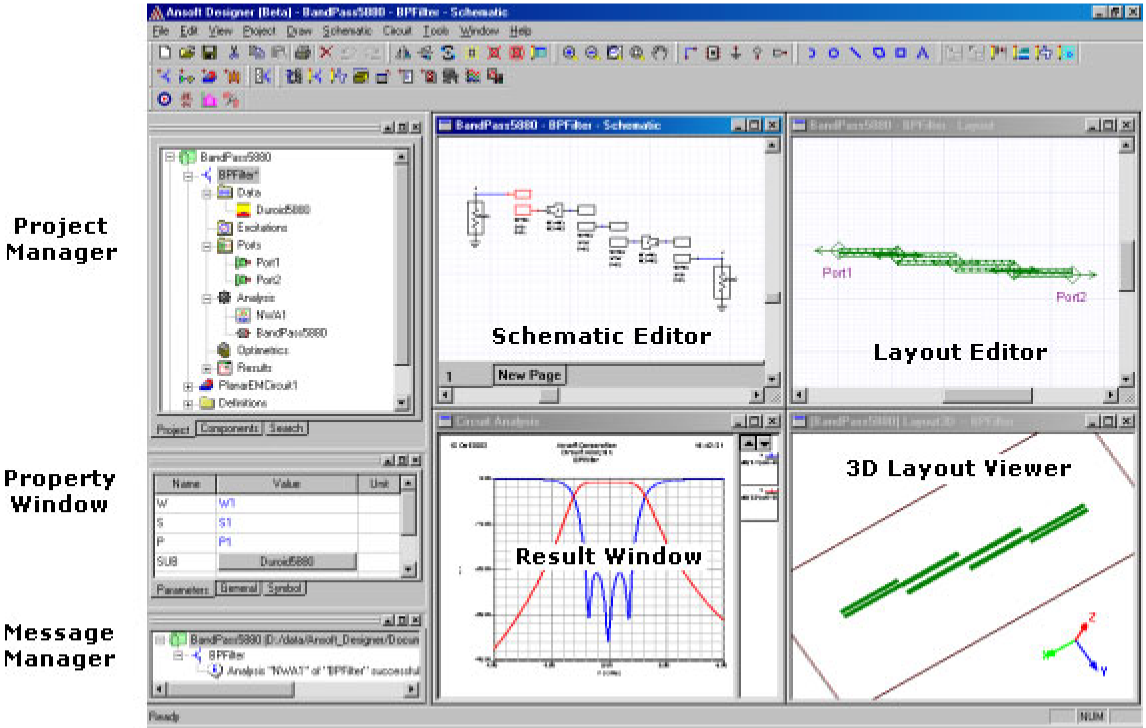
Layers and Stackup

Layers are used in the layout editor to organized and operate on sets of geometry or other visual indicators. Signal, metallized signal, and dielectric are common physical layers, while symbol (to show component symbols in layout), error, and rats (to show connectivity) are non-physical layers. The stackup contains additional properties of the physical layers, such as material, thickness, and elevation. Geometrical information on these layers is used to generate masks for manufacturing.

Solution

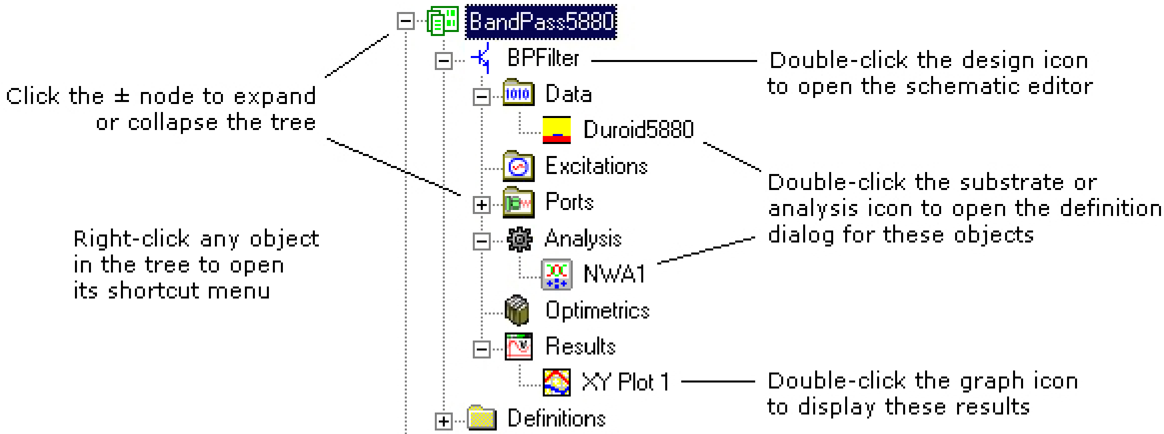
A solution is the successful result of an analysis, or imported results available for plotting.

Windows in Ansoft Designer



Project Manager

The project manager shows the projects loaded in to Ansoft Designer. Each project may consist of one or more circuit, system, or planar EM designs. Each design's data, such as model data information, ports and excitations, analysis setup information, and graphs, are displayed as entries in a separate subtree in the project manager.

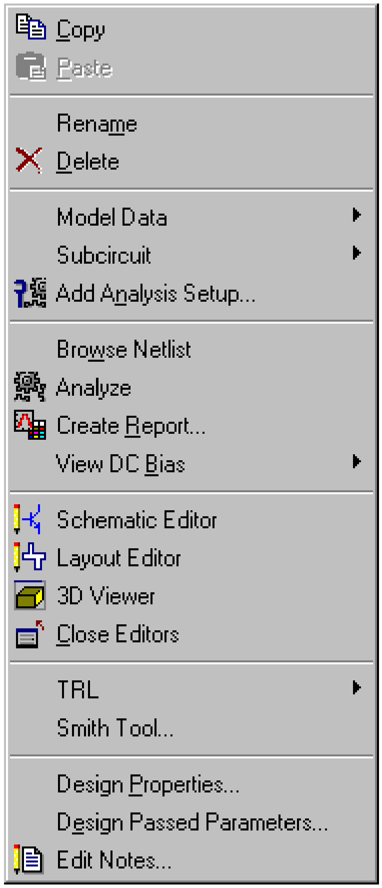


Information contained in the design is organized into folders:

- **Data**—Substrate data, library references, netlist fragments, and so on
- **Excitations**—Sources in the circuit
- **Ports**—Ports in the circuit
- **Analysis**—Setup information to perform various analyses
- **Optimetrics**—Setup information to perform optimization, statistical analysis, and so on
- **Results**—Collection of graphs and tables for the design
- **Definitions**—The collection of components, symbols, footprints, and so on, accessible from Ansoft Designer

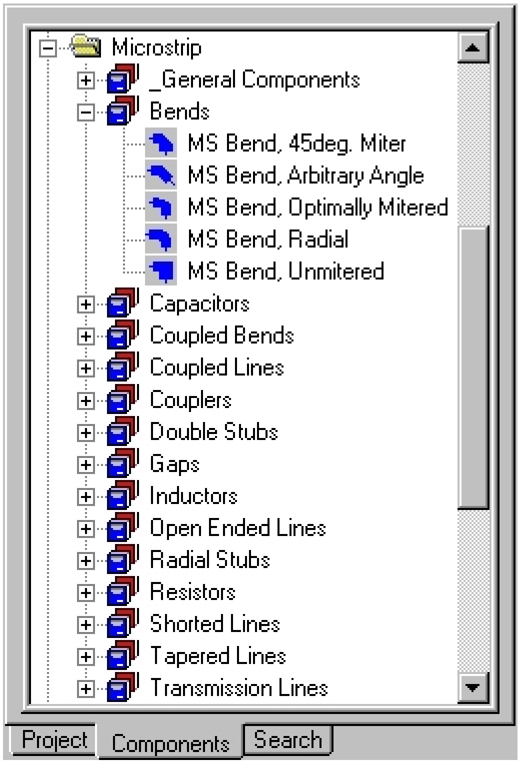
Double-clicking an item in these folders typically opens a dialog used to set the properties of that item.

Right-clicking a folder or item pops up a *shortcut menu* that allows you to perform various operations. For example, right-clicking the design icon for a circuit design displays this menu:



Virtually all project editing and management can be done from the **Project Manager** window using the right mouse button shortcut menus. Many of these menus are also available from the main menu bar in the **Circuit**, **System**, or **Planar EM** menus.

The **Components** tab near the bottom of the **Project Manager** window allows you to browse and select schematic components to place. It is organized by design type (circuit or system) and by component function. For example, the microstrip library has a number of categories and the set of bend components is expanded:



You can launch online help for a component from the **Components** tab of the **Project** window. To do this:

1. Right-click its icon.
2. Click **View Component Help**.

To place a component, you can either:

- Double-click its icon.
- Select, drag, and drop a component in the schematic editor.
- Right-click its icon, and then select **Place Component**.

Move your cursor to the schematic window. Now the cursor is accompanied by the symbol of the component you chose to place. You can rotate the component by pressing the **R** key.

To place the component, click where you want to place it. You can continue to place additional copies of the same component by clicking at additional locations. To stop placing components, do either of the following:

- Press the **ESC** key
- Right-click in the schematic and select **Place and Finish**, **Finish**, or **Cancel**.

If you prefer not to use the multiple-placement feature, you can turn it off as follows:

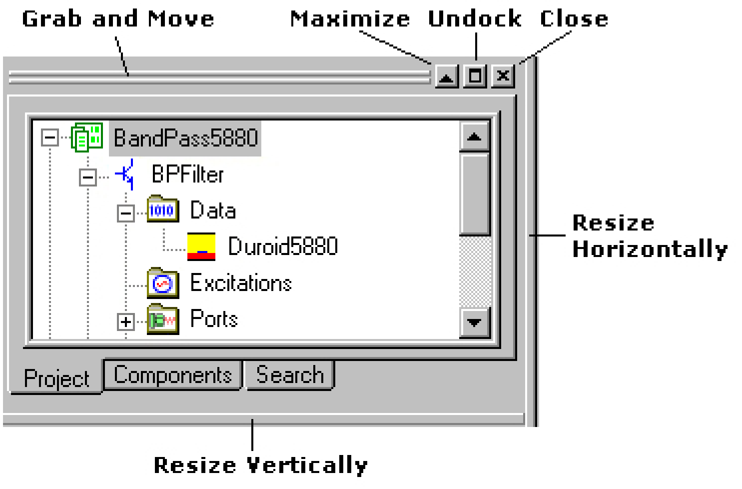
1. On the **Tools** menu, select **Options**, and then click **Schematic Editor Options**.
2. Select the **Multiple Placement** tab.
3. Clear the check box for each schematic item for which you want to turn multiple placement off.
4. Click **OK**.

The **Components** tab also includes a list of **Most Recently Used** (MRU) components and **Favorites**. The MRU list shows the last 10 unique components used.

To add a components to Favorites, right-click it and select **Add to Favorites**. This is a very convenient way of accessing commonly used components rather than navigating through the component tree repeatedly.

The **Search** tab allows you to enter full or partial names of components and search for them. This can be useful if you know some part of the name of the component you want to find.

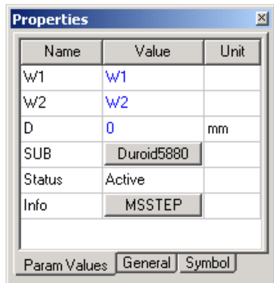
The **Project Manager** window is a dockable window. It can be moved and sized any way you want it and it can attach itself to any edge of the Ansoft Designer desktop. There are controls on the window when it is docked, as shown below:



Property Window

The **Property** window shows the properties of objects selected in the schematic and layout editors, and objects in the **Project Manager** window. It allows you to modify property values. The **Param**

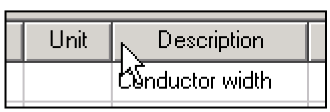
Values tab lists the simulation parameters of the component or components selected. The **General** tab lists the selection's name, symbol name, reference designator, and so on. These are generally not editable.

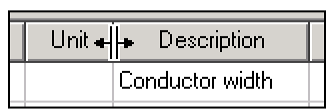


The **Symbol** tab provides information on the location of the component symbol in the schematic. The **Property Displays** tab lists the properties of the component that appear on the schematic, for example, C = 10pf for a capacitor. By clicking in the visibility box, you can choose how this information is displayed: **None** results in no label being shown; **Name** results in just the component name, i.e. C; **Value** results in just the component value, i.e., 10pf; and **Both** results in both the name and the value, i.e. C = 10pf being shown. This can be used to customize what the schematic looks like, which can be useful for very complex circuits where both **Name** and **Value** may be unnecessary.

Moving the Property Window

The **Property** window is a dockable, resizable window and can be positioned elsewhere on the desktop, as you may find convenient. You can change the relative widths of any two adjacent columns in the **Property** window by dragging the header separator between them:





Launching Online Help from the Property Window

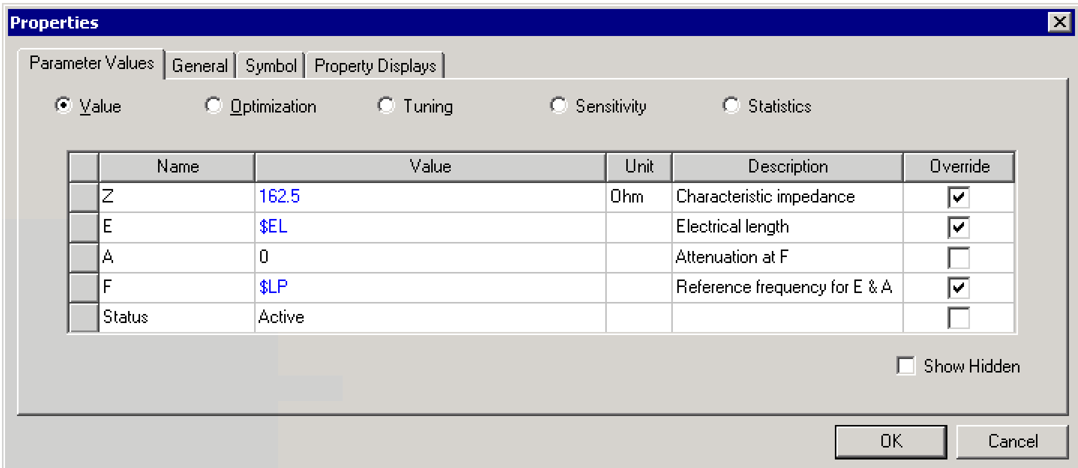
You can launch the online help for a component from the **Property** window. To do this:

1. Click the **Param Values** tab.
2. Locate the **Info** parameter entry, and then click the button in its **Value** cell:

Properties Dialog Box

Like the **Property** window, the **Properties** dialog box shows the properties of selected objects, and, where appropriate, allows editing the values of these properties. It is an expansion on the **Property** window in that it allows access to additional editing commands, and to settings for tuning, optimization, sensitivity, and statistical analysis, that are not available through the **Property** window.

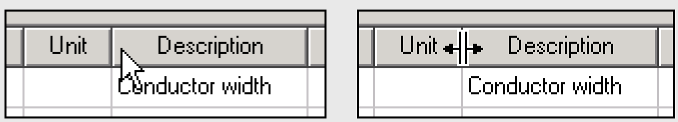
To open the **Properties** dialog for a component, double-click the component.



The **Parameter Values** tab lists the simulation parameters of the component or components selected. The **General** tab lists the selection's name, symbol name, reference designator, and so on. These are generally not editable. The **Symbol** tab provides information on the location of the component symbol in the schematic.

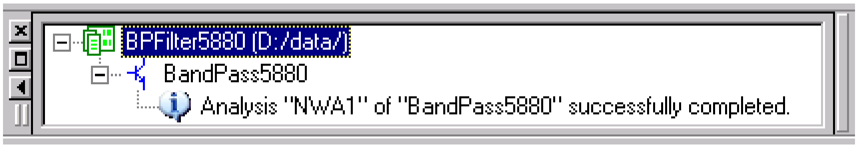
Hints

- You can resize the **Properties** dialog box by dragging its edges.
- You can change the relative widths of adjacent columns in the **Properties** dialog box by dragging the header separators between them:



Message Manager

The **Message** window shows error and informational messages about various processes in Ansoft Designer. These are often related to the status of a simulation, including any errors it may have produced.



Messages are organized by project, and then by circuit. Because a design can contain multiple top-level and subcircuits, and multiple analyses can be set up for each, this organization helps you determine where errors have occurred.

Clearing Messages

The Message Manager clears at the start of each analysis. To manually clear messages, right-click the message tree and select Clear Messages for <ProjectName>, where <ProjectName> is the name of the project for which you want to clear messages.

Hiding the Message Window Until Messages Appear

If you prefer to hide the Message Manager until a message is added:

1. Click **Tools>Options>General Options**.
2. Select **Show message window on new messages**.

You can now close the **Message** window and it will re-open when Ansoft Designer reports any errors, warnings, or successful completion of any simulations.

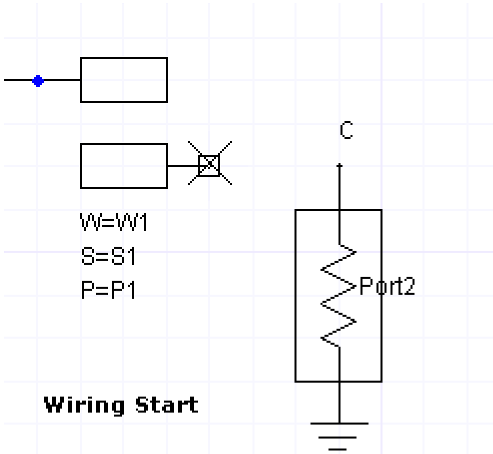
Moving the Message Manager

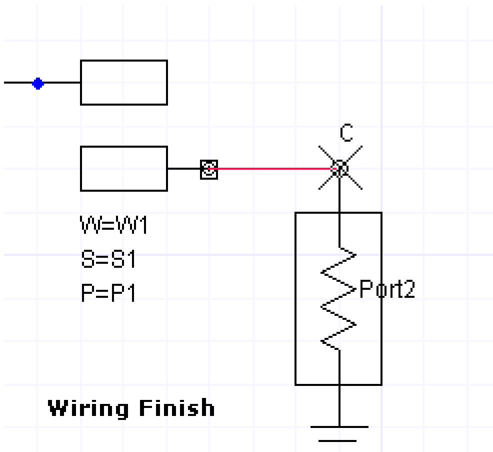
The **Message Manager** window is a dockable window. It can be moved and sized any way you want it and it can attach itself to any edge of the Ansoft Designer desktop. There are controls on the window when it is docked.

Schematic Editor

The **Schematic Editor** window allows you to place components and wire them together. You can move components by simply selecting and dragging them. Copy and paste can be used on components and their wires within the schematic editor. You can also copy and paste to other schematics.

As you place the cursor near a pin of a component, it changes from an arrow to an **X**. This indicates that the schematic editor is in the wiring mode. In the wiring mode, left-click to start drawing a wire. Left-click again to end the wire.





Commonly used items such as ports, *n*-port black boxes, grounds, and page connectors can be placed in the schematic by clicking their toolbar icons or by using the **Draw** menu.

View controls to zoom in, zoom out, and fit the drawing to the editor window are available on the **View** menu, and on the shortcut menu that opens when you right-click in a schematic.

Layout Editor

The layout editor shows the physical realization of the circuit. Components such as transmission lines will be drawn on the metal layer. Components that don't have footprints, such as ideal capacitors, will be represented in the layout as schematic symbols to indicate that no footprints are associated with them. Arbitrary graphical primitives unrelated to modeled elements can also be drawn.

When components with footprints are added to a layout or modified, their footprint connection points may no longer be properly aligned with those of other components. To correct this misalignment, click **Align Mw Ports** on the **Draw** menu.

3D Layout Viewer

Once you have defined a layout and its layer stackup, you can view the entire structure in 3D by doing either of the following:

- On the **Circuit** menu, click **3D Viewer**.
- Right-click the design in the project manager, and then click **3D Viewer**.

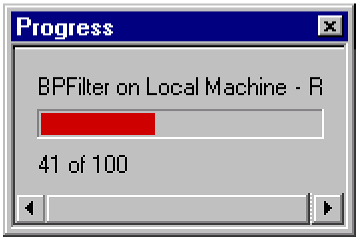
The view can be rotated, zoomed, spun around, and animated.

Results Windows

When a design has been successfully simulated, you can generate results in a wide variety of forms, including XY, polar and 3D graphs; Smith charts; and data tables. Various attributes of each can be customized to your liking.

Progress Window

When a simulation is started, the progress window becomes active. Each simulation run will have its own progress bar. Right-clicking the bar allows you to abort the simulation or view simulation details.



You can configure the **Progress** window to appear only when a simulation is running. To do this:

1. Turn off the **Progress** window display by clearing the **Progress Window** setting on the desktop's View menu.
2. On the desktop's **Tools** menu, choose **Options**, and then click **General Options**.
3. On the **Project Options** tab, select **Show Progress Window when starting a simulation**.
4. Click **OK**.

The progress window is also a dockable window, so you can position it where you like.

Now that you've gained experience with the essentials of Ansoft Designer's desktop and windows, you're ready to explore how to set up linear analysis in Ansoft Designer. The Planar EM and System simulators are described later in this Getting Started guide. Each of the three chapters describing the Circuit, Planar EM, and System simulators can be studied independently if desired.

2

Circuit Design

This topic shows you how to get started with Ansoft Designer's schematic editor and Circuit simulator by building and analyzing basic linear and nonlinear circuits. Its coverage assumes that you are already familiar with the basics discussed in the Introduction to Ansoft Designer topic.

As you work through this Circuit Design topic, you will:

- Gain experience with linear circuit simulation in the Ansoft Designer design environment through constructing and simulating a 5-GHz bandpass filter.
- Learn how to report simulation results using Ansoft Designer's post processing functions.
- Learn how to validate a Circuit design with Ansoft Designer's Planar EM structure simulator.
- Gain experience with nonlinear circuit simulation through constructing and simulating a single-diode mixer.
- Learn how to set up parameterized subcircuits through constructing a two-level circuit hierarchy.

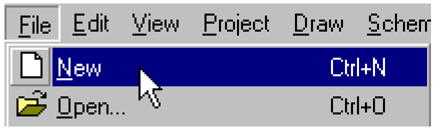
The first step is to start Ansoft Designer if you have not already done so.

Example: An Edge-Coupled Bandpass Filter

This section will guide you through building and analyzing a simple example while demonstrating basic features, and some advanced features, of Ansoft Designer.

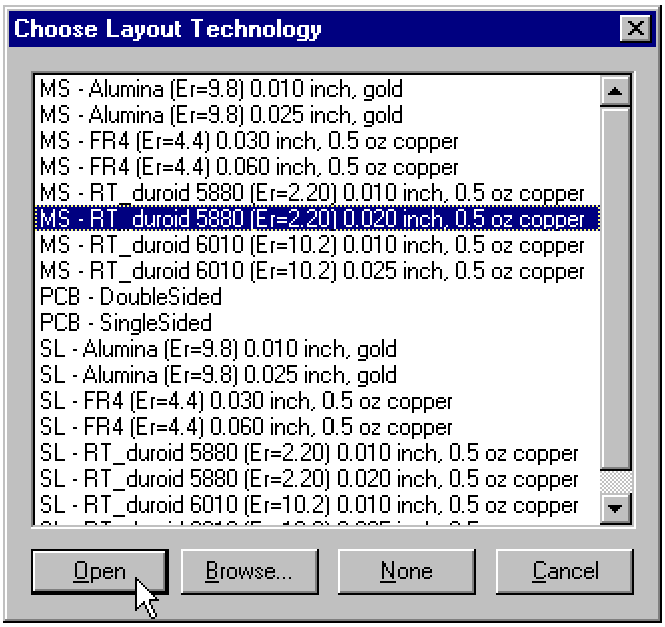
The filter was synthesized in the Filter Design tool within Ansoft Designer, but instead of simply creating the design from the synthesis tool, we will construct it manually to show the basic steps.

1. Start Ansoft Designer: On the **File** menu, click **New** .



   A new project is added to the project tree. It is named Project*n* by default, where *n* is the order in which it was added to the current session of Ansoft Designer.

2. Select the new project's name in the **Project Manager** window.
3. On the **Project** menu, click **Insert Circuit Design** .
4. In the **Choose Layout Technology** dialog box, select **MS - RT_duroid 5880 (ER=2.20) 0.020 inch, 0.5 oz copper**, and then click **Open**.



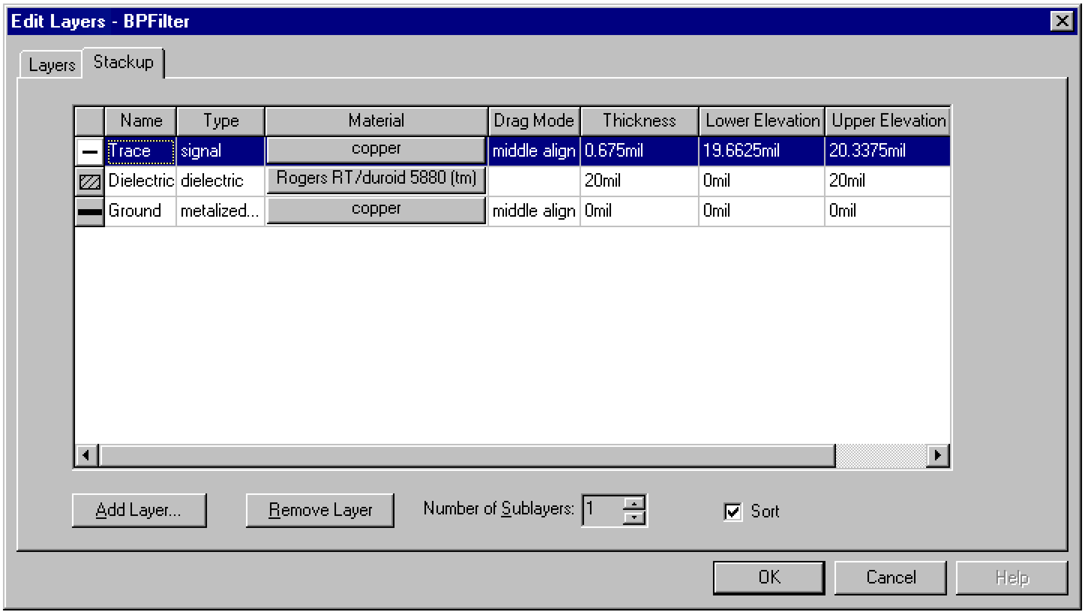
   A new, empty schematic, associated with RT Duroid 5880 technology, opens in the Ansoft Designer schematic editor.

Note If you want to open a new schematic that is not associated with the set of definitions that come with a technology file, you can click **None** rather than selecting a technology and clicking **OK**. This is useful for basic concept designs that do not or need not contain manufacturing or substrate information.

5. On the desktop's **Schematic** menu, click **Layout Stackup**  .

   Note that the stackup has the RT Duroid 5880 substrate material and 0.5-ounce copper for the signal layer:

   The **Stackup** tab shows the material layers upon which the filter will sit. There is a copperinfinite ground (metallized signal) layer with zero thickness. On top of the ground is a



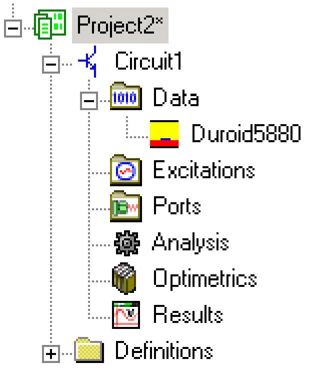
   Rogers substrate (dielectric), which is 20-mil thick. Finally, on top of the substrate is a trace layer that will consist of the metal outlines required for the filter, which is 0.675-mil-thick copper (0.5-oz copper.)

   In this picture, the **Material** column has been resized for better readability.

6. Click **OK**.

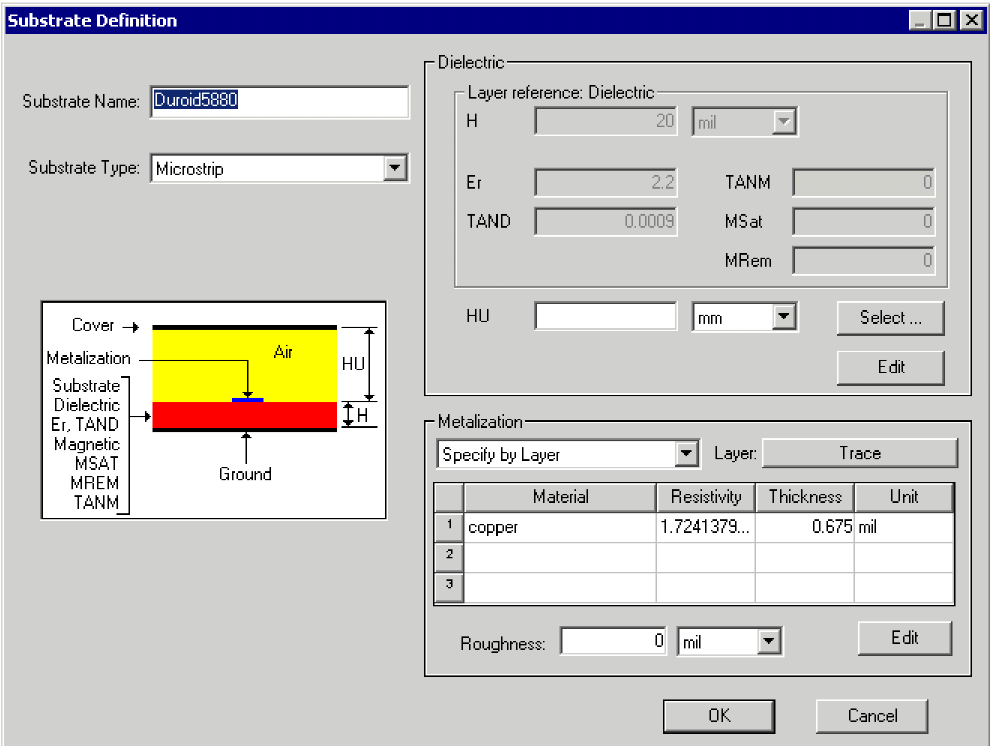
7. You can also expand the default project name in the project tree and the Circuit1 **Data** icons to

reveal the substrate defined in the technology file:



The asterisk after the default project name indicates that the contents of the project have changed—with your addition of Circuit1 at step 4—since it was last opened or saved. Don't save the project just yet; we'll do that shortly.

8. Double-click the **Duroid5880** substrate icon to view the substrate detail:



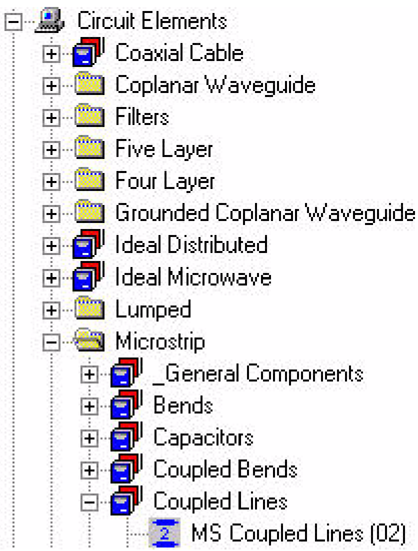
Note that the dielectric information is visible but inactive. This indicates that this substrate definition is referencing a layer in the stackup. The metallization is also referencing the Trace layer. You can edit either of these reference-based definitions by clicking its associated **Edit** button, but doing so will break the connection to the physical stackup.

9. Click **OK**.

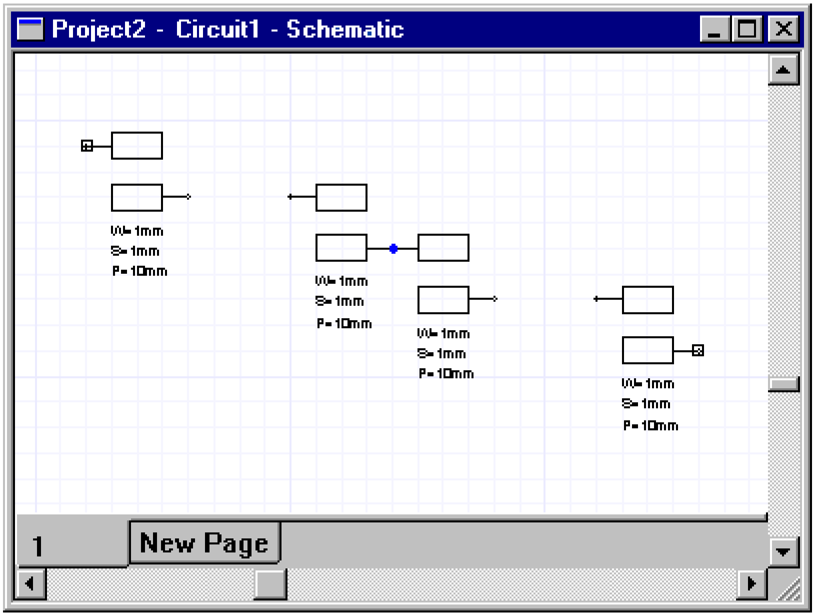
Build the Circuit Design

We will now place the components in the schematic.

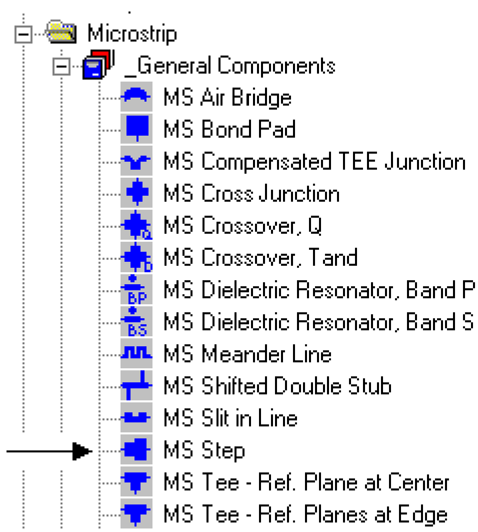
1. Click the **Components** tab of the project manager, and expand the **Microstrip** folder and the **Coupled Lines** icon:

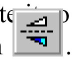


2. Find the component **MS Coupled Lines w/Open Ends, Symmetric**, five up from the bottom of the **Coupled Lines** list. Select the coupled line component and drag it across to the schematic window. Place four of these components in the schematic so the first and second, and the third and fourth, are positioned apart, with the second and third connected:

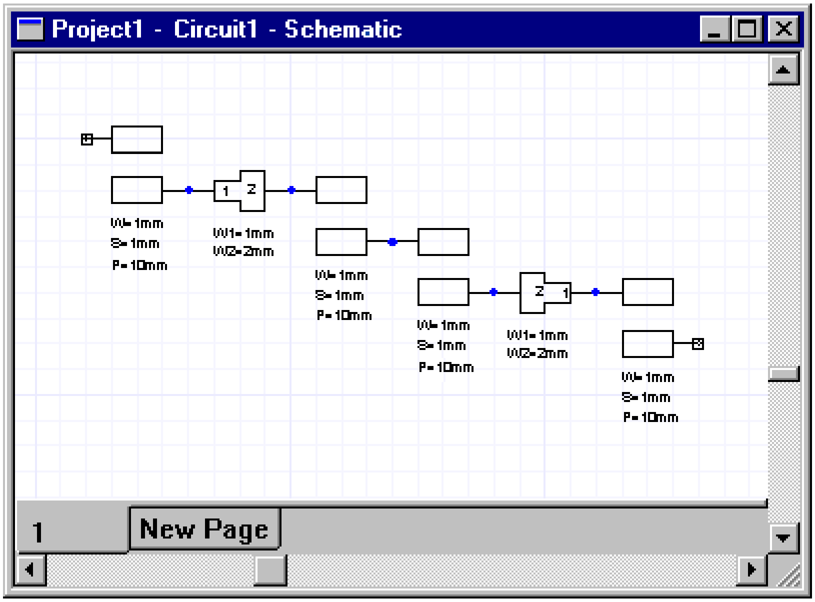


3. Expand the component group called **_General Components**. Select the **MS Step** component and drag it across to the schematic window.



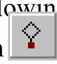
4. Place one step between the first and second couplers, and a second step between the third and fourth couplers. When placing the second step, press the **R** key twice to rotate it so it is turned around. Then, select the second step and click the **Flip about X** toolbar icon .

   The schematic should now look like this:



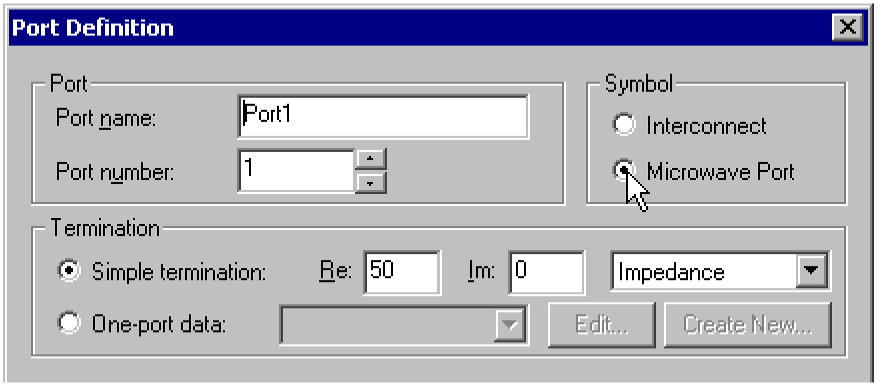
Add Ports

Add two ports by doing the following:

1. Click the **Port** toolbar icon .
2. Click to place the port to the left of the first coupler section.
3. Double-click the port symbol.

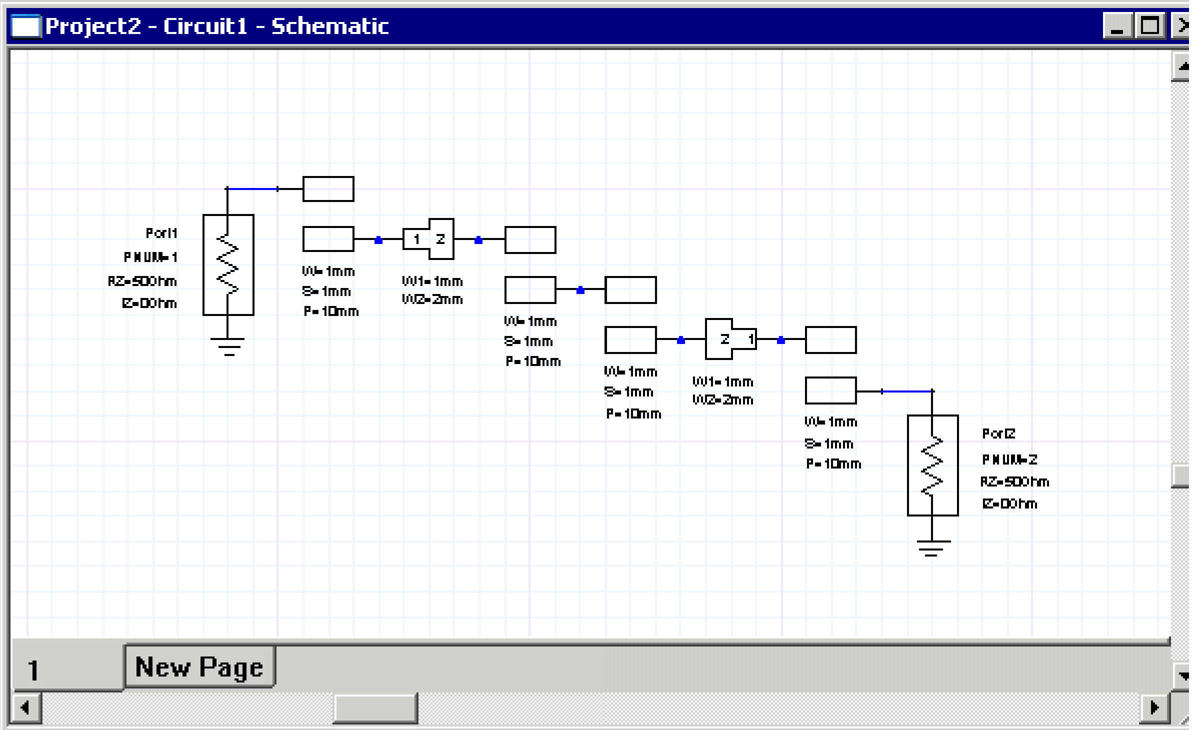
   The **Port Definition** dialog opens. In this dialog, you can modify the port's name, termination, and other characteristics. For now, leave the termination set to 50 ohms.

4. In the **Symbol** group, click **Microwave Port**:



5. Click **OK**.
6. Use this procedure to place a second port at the filter output.
7. Right-click the second port, and select **Flip about Y**.

The schematic should now look like the following:



Add Variables

Since the dimensions of edge-coupled filters are symmetrical, we will use variables to define the transmission line widths, lengths and spacings.

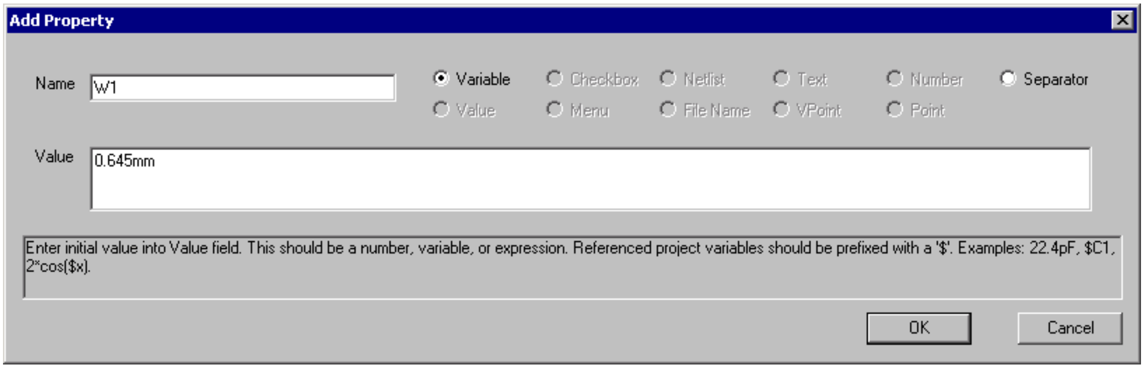
1. From the **Circuit** menu, select **Design Properties**.

   Variables can be defined in two **Design Properties** tabs. The first tab, **Parameter Defaults**, defines parameters for which values can be passed into the design when it is inserted into a parent circuit as a subcircuit. The second tab, **Local Variables**, defines variables that are only accessible in this design.

2. Click the **Local Variables** tab, and then click **Add**.

   The **Add Property** dialog box opens.

3. Type **W1** in the **Name** box and 0.645mm in the **Value** box, and then click **OK**.



4. Keep adding variables until you complete this set:

W1	0.645mm
W2	0.780mm
S1	0.197mm
S2	0.621mm
P1	11.0mm
P2	10.95mm

5. Click **OK** to close the dialog box.

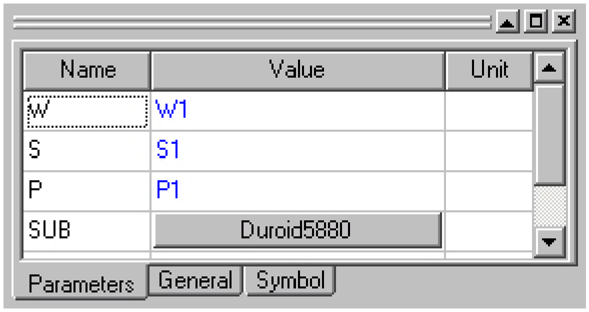
Note Whether you are defining project, local, or parameter variables, intrinsic names (f, freq, lb, etc) are reserved and cannot be used or entered into the source and port diaglogs.

Assign Variables to the Components

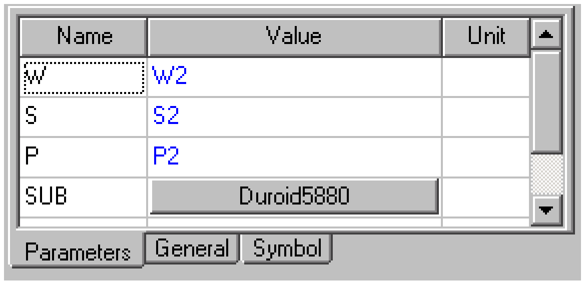
1. Select the first and fourth coupled line components by pressing the **CTRL** key as you click them.
   - Alternatively, right click the two selected components and bring up the **Properties** dialog.

   The **Properties** window now shows the parameters for both components.

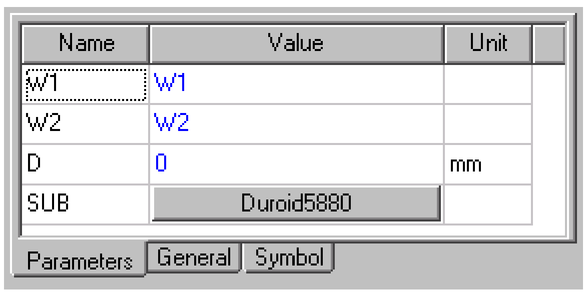
2. Working in the **Properties** window, enter the values W1, S1, and P1 for W, S, and P:



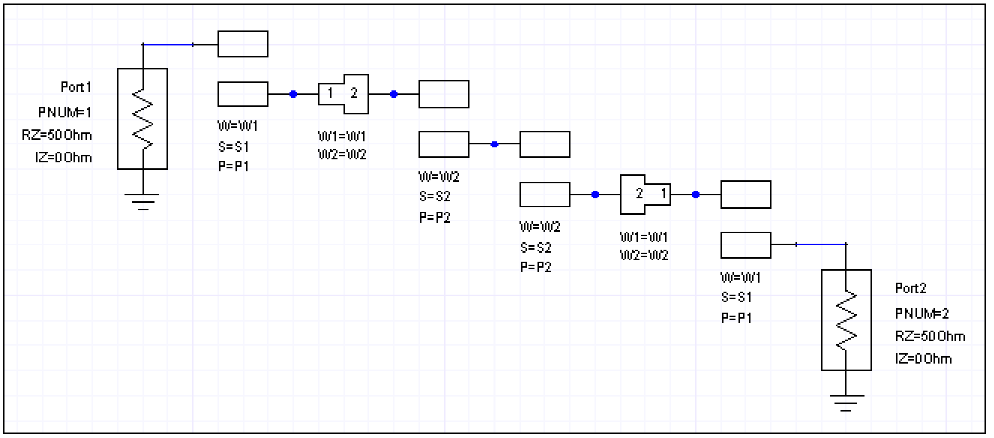
3. Next, select the two inner coupled line sections and change their parameter values to:



4. Select the two step components, and then change their parameter values to:



Now the schematic has all parameters assigned:

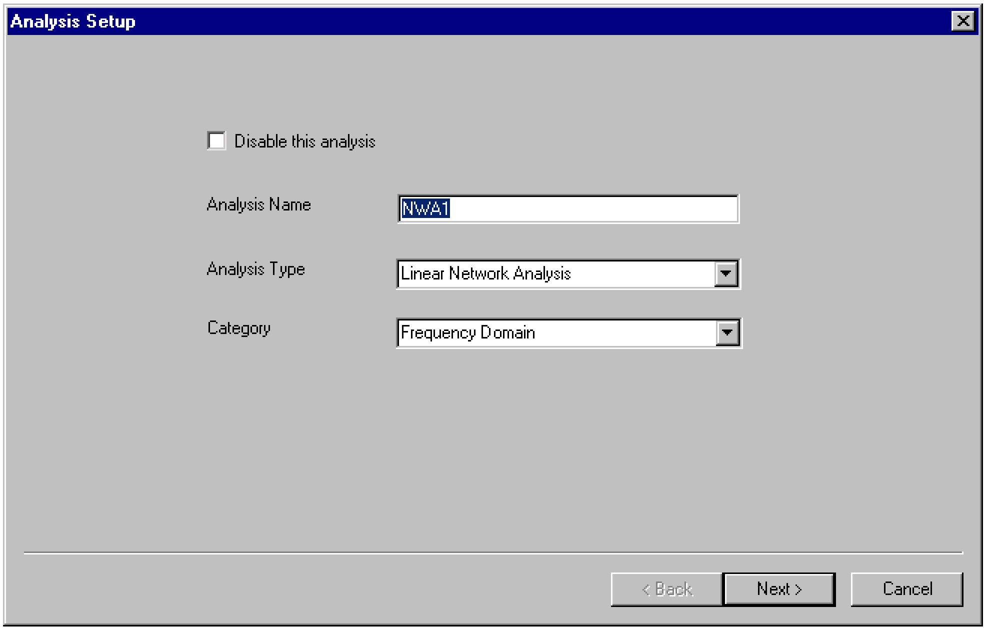


5. Next, we'll change the schematic name from Circuit1 to BandPass5880:
   a. Click the **Circuit1** icon in the project tree, and then click its name.
      The name label turns into a text box, with the existing name text selected for editing.
   b. Type **BandPass5880**, and then press **ENTER**.
6. On the **File** menu, click **Save**. Type **BPFilter5880** into the **File name** box, and then click **Save**.

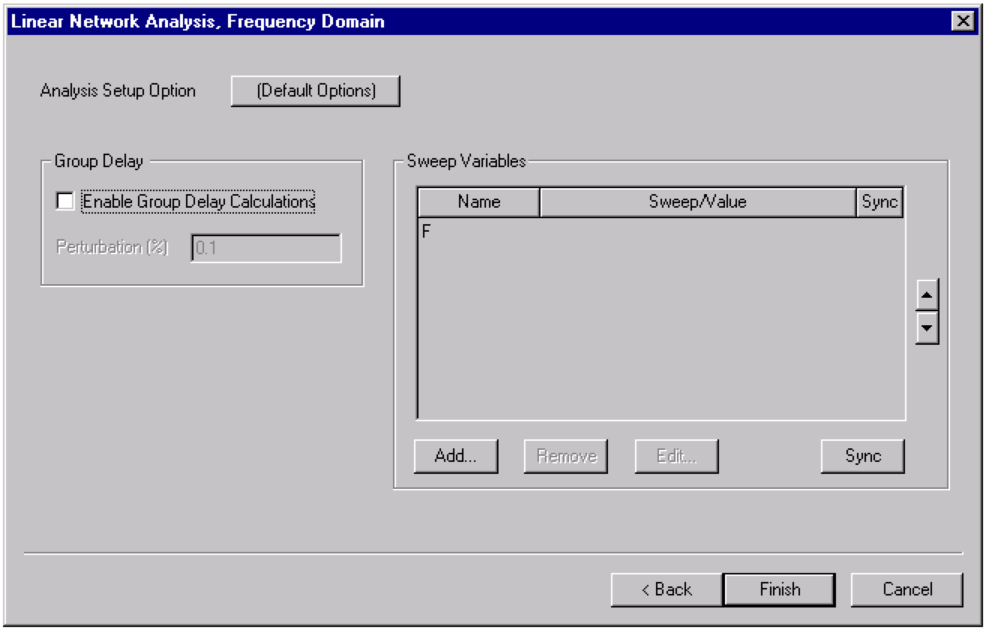
Set up the Circuit Analysis

We must add an analysis setup before we can analyze the filter circuit.

1. On the **Circuit** menu, click **Add Analysis Setup** . The **Analysis Setup** dialog box opens:

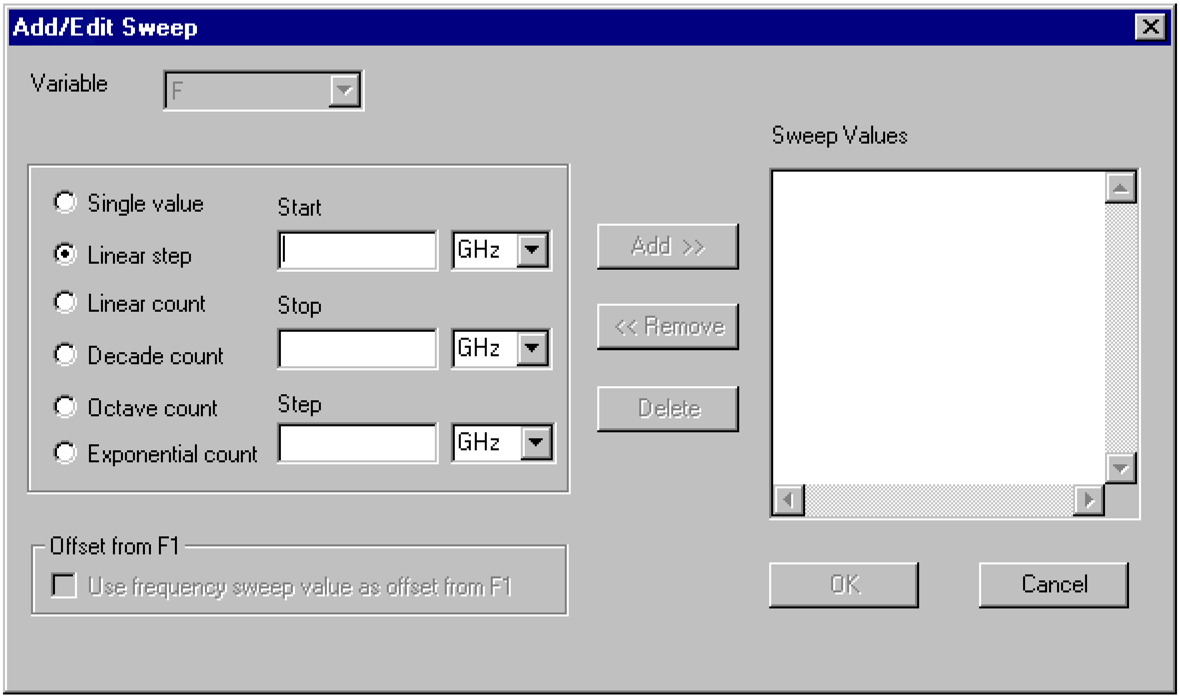


2. We want to perform a linear network analysis in the frequency domain, so click **Next**.



3. Select the sweep variable name **F**, and then click **Edit**:

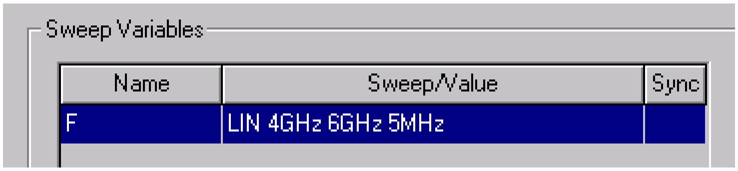
The **Add/Edit Sweep** dialog box opens:



Enter these three frequency sweep values:

Start	4 GHz
Stop	6 GHz
Step	5 MHz

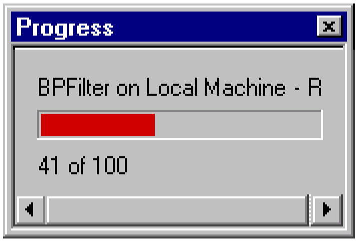
When you’re done, click **Add**, click **OK**, and then click **Finish**.



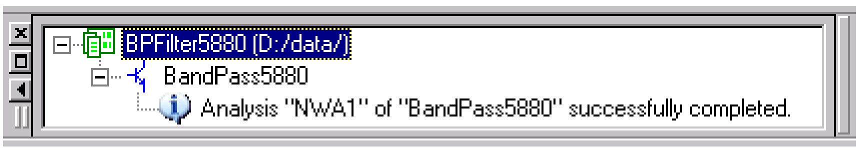
Start the Circuit Analysis

- To start analysis, click **Analyze** on the desktop's **Circuit** menu.

  The **Progress** window activates to show how much of the analysis has been completed:



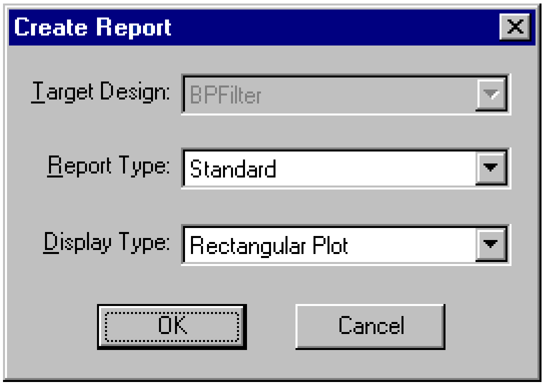
Once analysis is completed, a message is placed in the **Message** window indicating that the analysis was successfully completed. Should any errors occur, messages about them will be shown too.



In this picture, the **BPFilter5880** icon has been expanded to show the message text.

Working with Post Processing

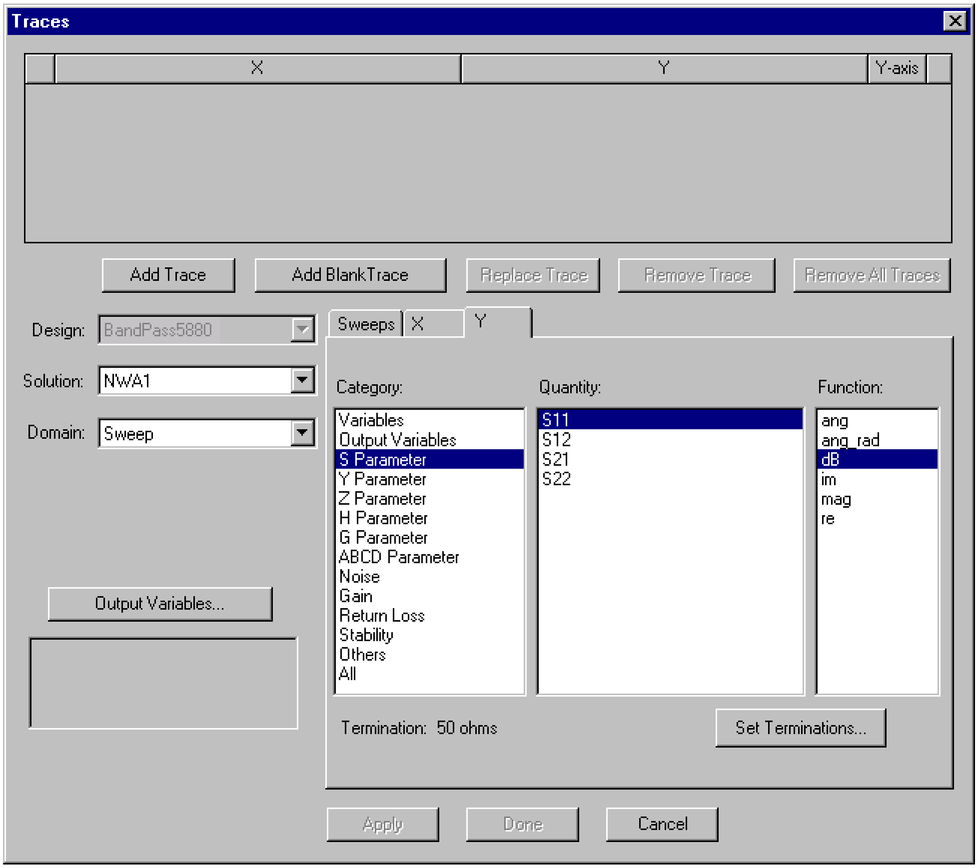
1. To create a graph, click **Create Report** on the **Circuit** menu. The **Create Report** dialog box opens:



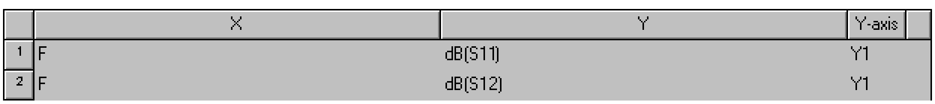
   We will plot *S* parameters in a rectangular (XY) graph, so the default **Display Type** selection is fine. You can also report circuit performance in polar and Smith graphs, and on table listings, by selecting a different display type.

2. Click **OK**.

The **Traces** dialog box opens:



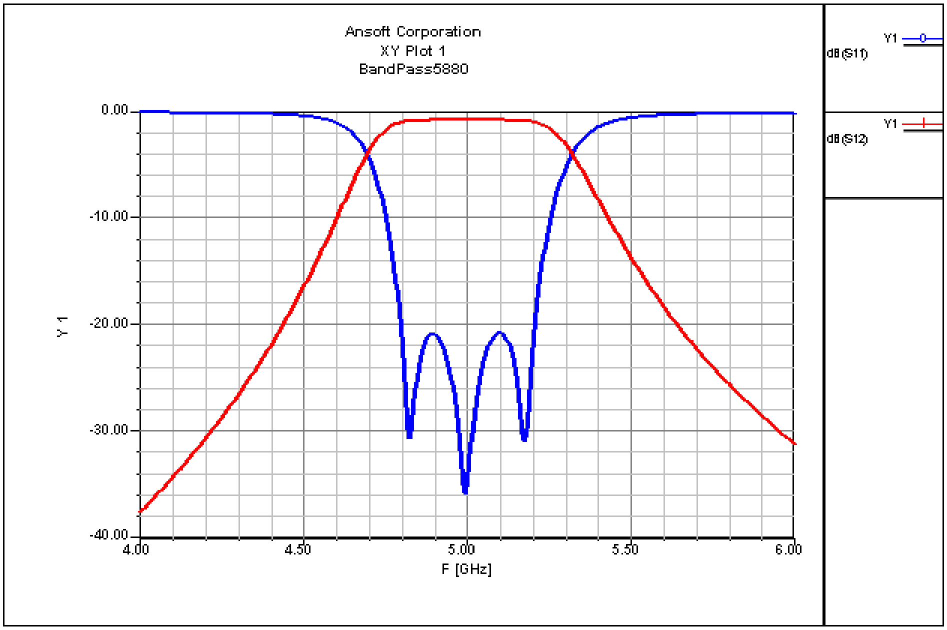
3. To quickly plot *S* parameters for the bandpass filter, multiselect **S11** and **S12** (click **S11**, and then click **S12** while pressing the **CTRL** key), and then click **Add Trace**. This creates two entries in the trace definitions list at the top of the dialog:



	X	Y	Y-axis
1	F	dB(S11)	Y1
2	F	dB(S12)	Y1

4. Click **Done**.

The graph of S_{11} and S_{12} opens:



The Traces Dialog Box

The Traces dialog is organized to provide the flexibility to define a great many different graph types while keeping the task of creating simple graphs quick and direct. The controls and options in the Traces dialog box include:

- **Design**—If the project has multiple top-level designs, selects the desired design.
- **Solution**—If the design has multiple analysis setups, selects the desired analysis.
- **Domain**—Selects a report domain (including frequency, time, spectral, and sweep) from among choices that depend on the analysis type and setup.
- **Sweeps tab**—Allows you to select sweep variables and determine their ordering in trace specification.
- **X, Y tabs**—These specify the calculation to be plotted against the X and Y axes.
- **Category**—Lists groups of responses and calculations so you can easily find them.
- **Quantities**—Lists the responses and calculations available from the simulator for each category.
- **Functions**—Allows you to apply common math functions to response calculations.

The available trace operations include:

- **Add Trace**—Adds a trace as defined in the **Sweeps**, **X**, and **Y** tabs.
- **Add Blank Trace**—Add a blank to the list of traces to be used for editing.

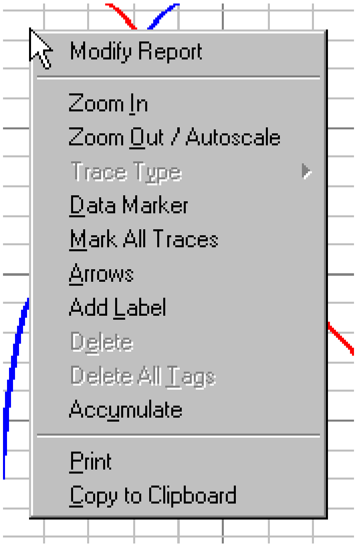
- **Replace Trace**—Replaces the selected trace definition with the trace defined in the **Sweeps**, **X**, and **Y** tabs.
- **Remove Trace**—Deletes the selected trace definition.
- **Remove All Traces**—Deletes all trace definitions.
- **Y-axis**—Selects the Y-axis (Y1 through Y4) to which the selected trace is assigned. Click in the **Y-axis** value cell to select the desired Y-axis.

Note that the text in the **X** and **Y** trace columns is editable, so you can manually type or modify any trace definition. Ansoft Designer validates your manual input as soon as you click out of an edited field.

There are a number of attributes you can control from within a graph, such as changing colors, line widths, axis scaling, and so on. These features can be accessed three ways:

- Double-click a trace. A properties dialog box opens, giving you access to the trace's color, line width, and symbol settings.
- Double-click the graph's X or Y axis. A properties dialog box opens, giving you access to settings to the axis' color, scaling, and unit settings.
- Double-click the graph's background, border, or grids. A properties dialog box opens, giving you access to settings for the colors and, if appropriate, line style of the selected object.

A number of operations are available from the graph shortcut (right-click) menu:



- **Modify Report**—Opens the **Traces** dialog box for the current graph.
- **Zoom In/Out/Autoscale**—Magnifies a portion of the graph, or autoscales the view.
- **Trace Type**—Enabled when a trace is selected. Modifies the method of viewing the trace.

- **Data Marker**—Puts a tag on the trace to mark a specific point.
- **Mark All Traces**—Displays values for corresponding data points of multiple traces. Also finds minimum and maximum values for a single trace or a family of traces.
- **Arrows**—Displays arrows on the trace to show increasing calculation index.
- **Add Label**—Puts a text label in the graph.
- **Delete**—When a trace is selected, deletes the trace.
- **Delete All Tags**—Deletes all data markers and their legends.
- **Accumulate**—Keep prior traces on the graph up to the accumulate depth.

Note You can set the accumulate depth as follows:

1. On the **Tools** menu, point to **Options**, and then select **Report 2D Options**.
2. Select the **General Option** tab.
3. Type a value in the **Accumulate Depth** box.
4. Click **OK**.

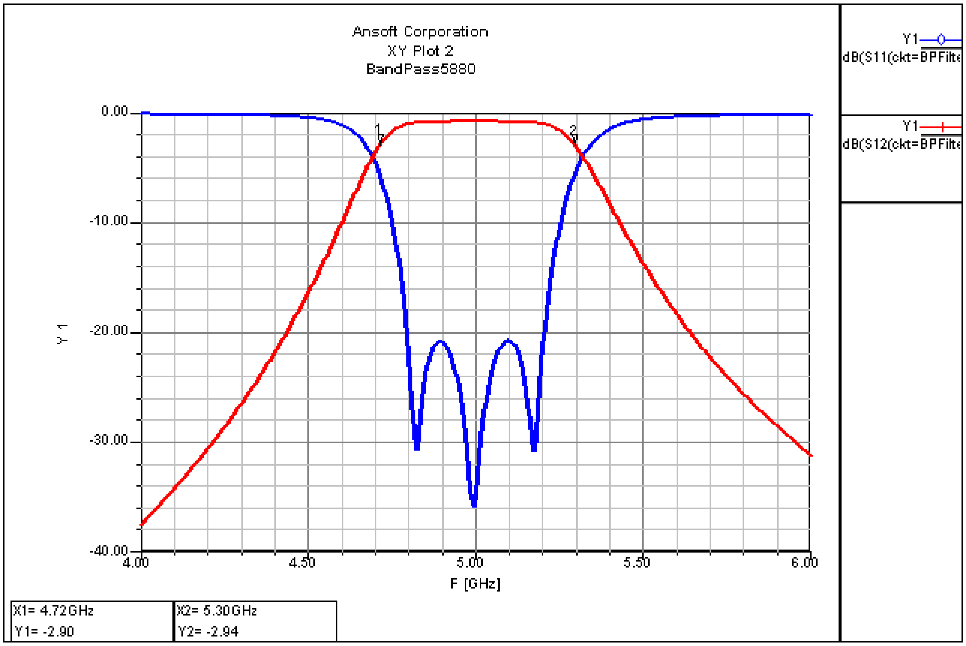
- **Print**—Prints the graph.
- **Copy to Clipboard**—Copies the graph on the clipboard for pasting into other applications.

Numerous other default options, such as table fonts and trace colors, can be modified in the **Report 2D Options** dialog.

Modify the Data Markers

1. Select **Data Marker**, and you'll see a small marker on the first trace.
2. Select the S12 trace legend to put the data marker on that trace. As you move the cursor, the data marker follows it. You can also use the keyboard arrow keys to move the marker left and right and between traces.
3. Position the marker near the lower-frequency –3-dB point and click (or press the **T** key). A tag will appear.

4. Move the marker over to the second –3-dB point and click again; a second tag will appear as shown.



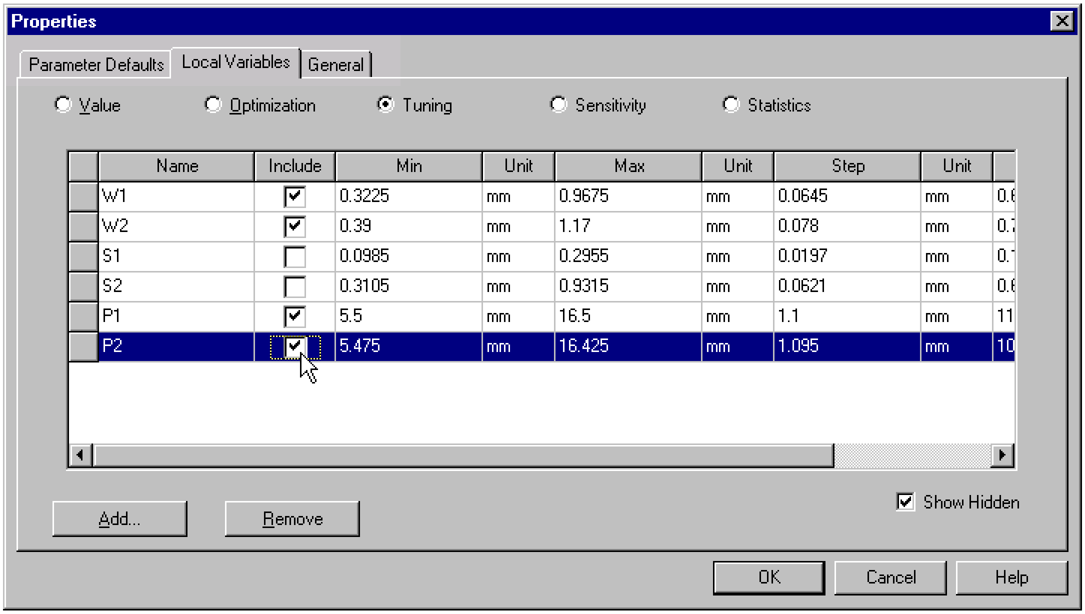
5. Exit marker mode by either clicking the right mouse button to bring up the popup menu and select **Exit Data Marker**, or press the **Esc** key.
6. To delete data markers from the graph, do either of the following:
   - Click a tag legend you want to delete, and press **DELETE**.
   - Right-click in the graph, and select **Delete All Tags**.

Tuning

Tuning provides an interactive means of changing a design's variables or component values and immediately viewing the results. In this example, all our component parameters are set by variables, so we will only tune the variables.

1. On the **Circuit** menu, click **Design Properties**.
2. Click the **Local Variable** tab.
3. Select the **Tuning** option.

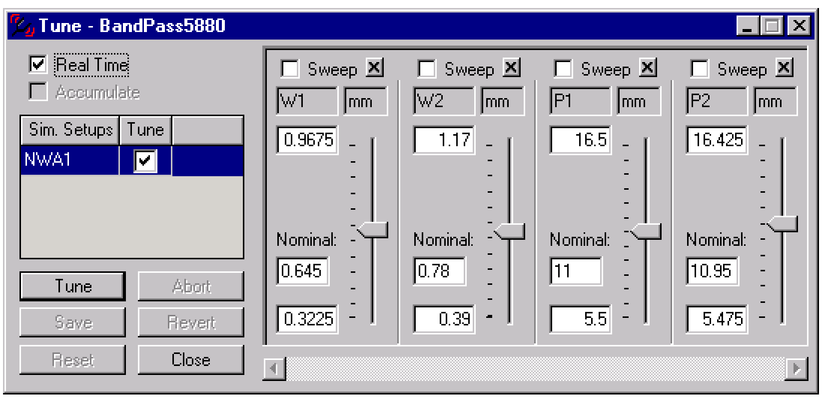
4. You can select any of the variables to include in tuning. For now, select **Include** for W1, W2, P1, and P2.



The minimum and maximum values are automatically set to be 0.5 and 1.5 times the nominal value. You can change them if you want to.

5. Click **OK**
6. On the **Circuit** menu, click **Tune**. 

The **Tune** dialog box appears.



The main part of the dialog consists of the tune variable sliders. With **Real Time** selected, Ansoft Designer performs a new analysis each time you move, and as you move, a slider to a new value. You can also set up a sweep of a tune variable by selecting its **Sweep** check box. To remove a tune variable from the dialog box, click its close button. 

Selecting **Real Time** indicates that the program will do an analysis and update the graphs each time a slider is moved. Clearing **Real Time** allows you to position the slider without performing an analysis until the **Tune** button is clicked.

If the BandPass5880 design included simulation setups other than NWA1, the simulation setups list would include them, and the **Tune** check box would be selected for each by default. The state of the **Tune** check box for an analysis determines whether or not tuning will be performed for that analysis.

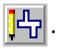
Briefly, the other tune controls include:

- **Tune**–Performs an analysis.
- **Abort**–Stops the analysis.
- **Save**–Saves the current state (values) of the tune variables and names this state. Optionally applies to the design the new values achieved through tuning.
- **Revert**–Restores a saved state.
- **Reset**–Reverts the design to the original tune variable values.
- **Close**–Closes the tune session. As the session closes, you will be asked if you want to apply the tuned values if you have not already done so.

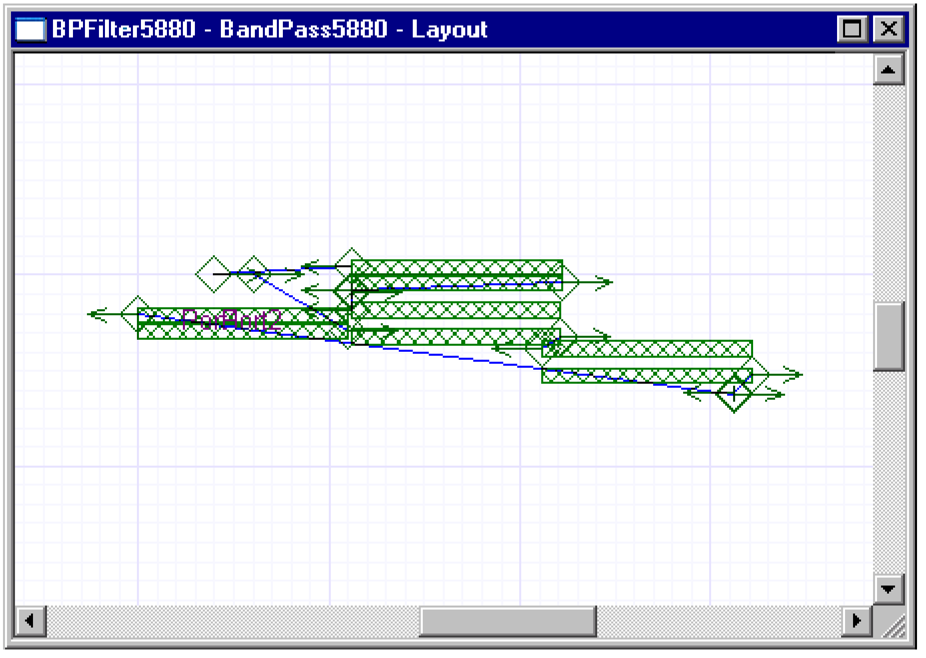
7. Move the sliders and watch the graph change. You can edit the design, change its properties, or change the analysis setup while the tune dialog box is active. So, for example, you may want to reduce the number of analysis frequency points for faster tune updates.
8. When you are finished, click **Close**.
9. You are then given the option to apply the tuned values to your design. For this example, select **Don’t Apply**.

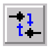
Finalize the Physical Layout

Now that we have entered and analyzed the circuit, and have verified that its behavior is acceptable, we are interested in viewing and finalizing the physical layout of the circuit.

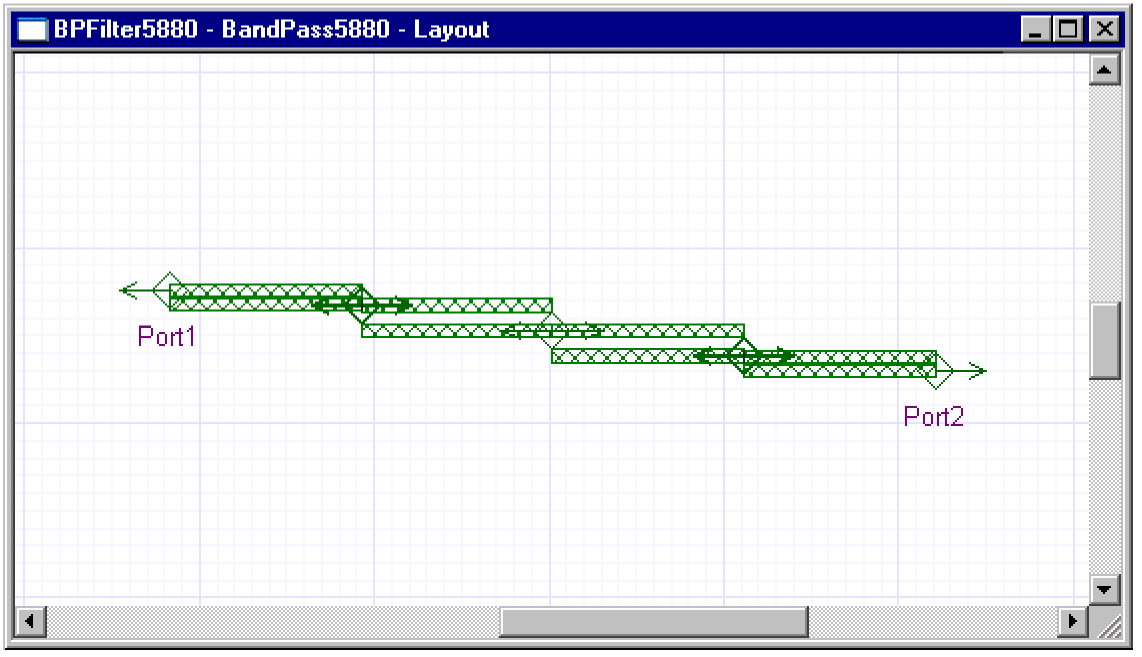
1. On the **Circuit** menu, click **Layout Editor** .

   The **Layout** window opens, displaying a group of primitives as in the picture below:



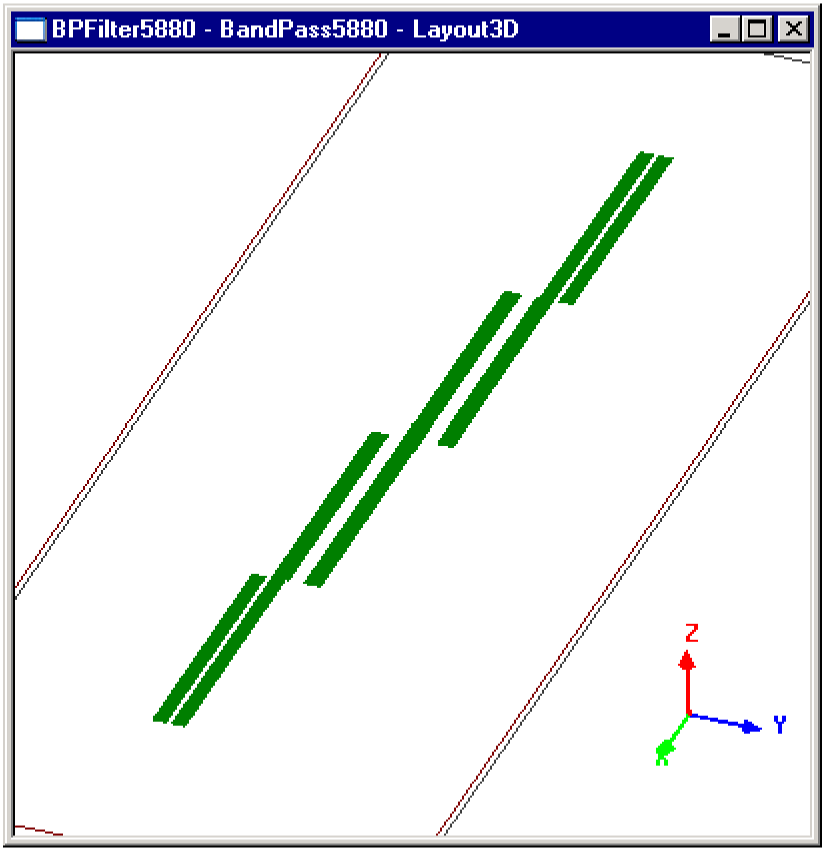
2. All the coupled sections are there, but they are not aligned. That is, their connection points are not physically touching with the correct orientations. To correct this:
   a. Select all the items in the layout (press **CTRL+A** or draw a selection box around all of the sections).
   b. On the **Draw** menu, click **Align Mw Ports**  or press the **Ctrl+M**.

The layout should now look like:



3. You can also view the 3D image of this layout:
   a. On the **Circuit** menu, click **3D Viewer**.
   b. Use commands on the **3D Viewer** right mouse shortcut menu to zoom, rotate, and pan the image. The **3D Viewer** shortcut menu is also available when you right-click the layout window.
      - Alternatively, press **Alt + Shift** and move the mouse to zoom, press **Alt** to rotate, or press **Shift** to pan.
   c. Return to the regular layout view by clicking **Layout Editor** on the **Circuit** menu.

d. Make sure you are viewing the Planar EM design: Click the Planar EM design in the project tree.



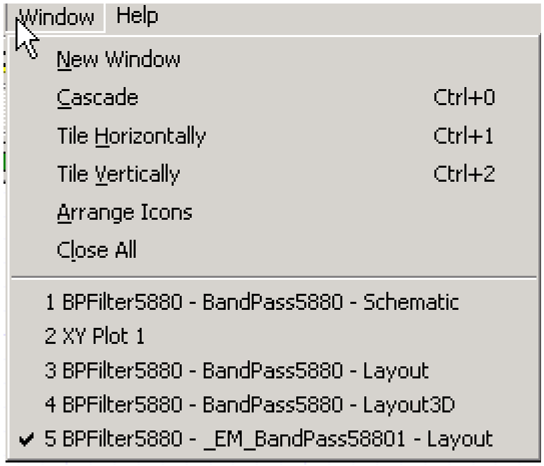
Exporting to Planar EM Analysis

We can verify the design using electromagnetic (EM) planar analysis. To do this, we must first export the circuit layout geometry using Ansoft Designer's **Copy to Planar EM** command. We can copy all of the layout geometry to Planar EM or only a portion. In this example, we will copy everything to Planar EM.

1. Return to the layout editor by selecting the layout window from the **Window** menu.
2. Select all of the items in the layout window by pressing **Ctrl+A** or by drawing a selection box around all the items.
3. On the **Edit** menu, click **Copy to Planar EM**.

   A new Planar EM design is added to the project. It is equivalent to the circuit design just created.

The filter geometry is now available in the Planar EM layout editor. Since both Circuit and Planar EM layouts are identical, care must be taken when viewing and editing the two designs. The Circuit design is known as *BandPass 5880*, while the equivalent Planar EM design is known as *_EM_BandPass5880*. The two layout windows are listed on the **Window** menu.



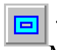
Items to consider when copying Circuit layouts to Planar EM:

- Hierarchical designs will be flattened.
- Parameterized designs will remain parameterized.
- Port definitions will be maintained as much as possible.
- Scripted footprints will be instantiated and will no longer be scripted.

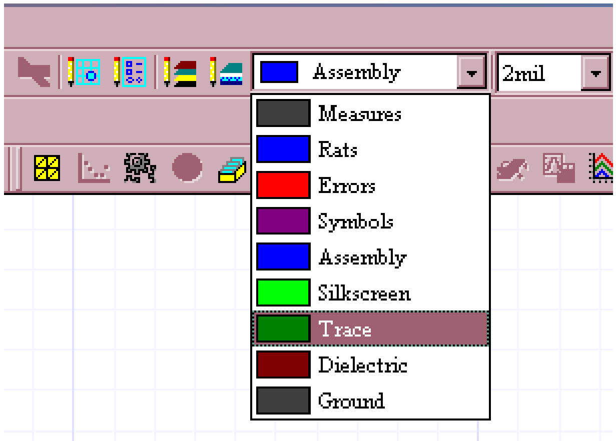
Compare the Planar EM Model to the Circuit Analysis Results

To compare the behavior of this model to the Circuit analysis results, we must add 50-ohm feedlines to each end of the filter. Otherwise, the EM analysis will terminate the ports in their characteristic impedances, giving results that cannot be validly compared with those from the Circuit analysis of the filter.

1. First, remove the ports from the Planar EM design because they are not 50-0hm ports:
   - Select the ports one at a time in the **Excitations** folder of the Planar EM design in the project tree, and then press **Delete**.

2. On the **Draw** menu, select **Primitive**, and then click **Rectangle** to change to the Rectangle tool  . Then:

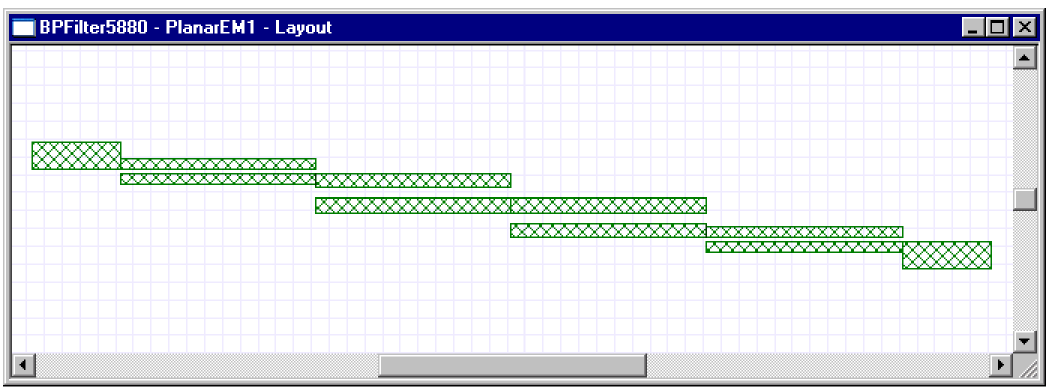
   a. Make **Trace** the active layer by clicking it in the **Active Layer** pull-down list in the menu bar:



   b. Draw a rectangle, and then edit its properties to make it 5 mm wide and 1.58 mm high. This makes 50-ohm line.

   c. Place the line at the input to the filter, carefully aligning it with the input portion of the coupled lines.

   d. Copy the line by pressing **Ctrl+C**, paste it by pressing **Ctrl+V**, and add the copy to the other end of the filter. Your drawing should look like this:



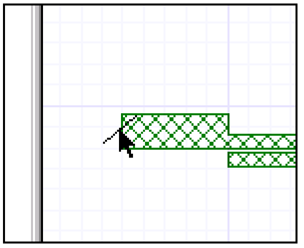
3. The lines just added must be merged with the line segments they adjoin. To do this:

   a. Multiselect the left 50-ohm line and the coupler line it adjoins. (Press and hold **CTRL**, click one structure, and then click the other.)

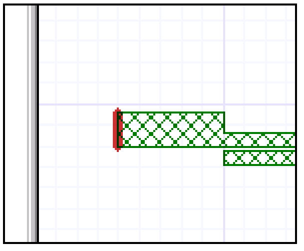
   b. On the **Layout** menu, select **Merge Polygons**, and then click **Union**.

   c. Do the same with the right 50-ohm line and the coupler line it adjoins.

4. Next, we must add ports. To do this:
   a. On the **Edit** menu, click **Select Edges** .
   b. Move the cursor close to the left edge of the filter. As the cursor nears the edge, it changes to a solid arrow touching an oblique line, indicating that you can select the edge:



   c. Click to select the edge:

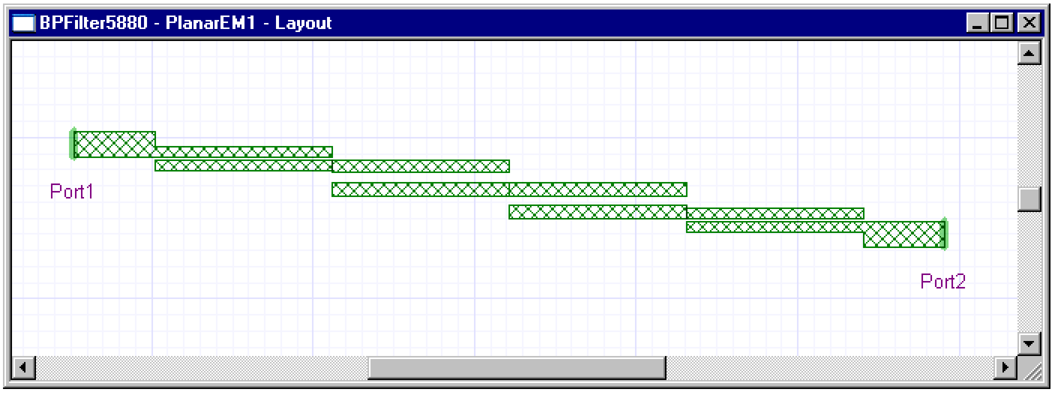


5. On the **Draw** menu, click **Edge Port** .

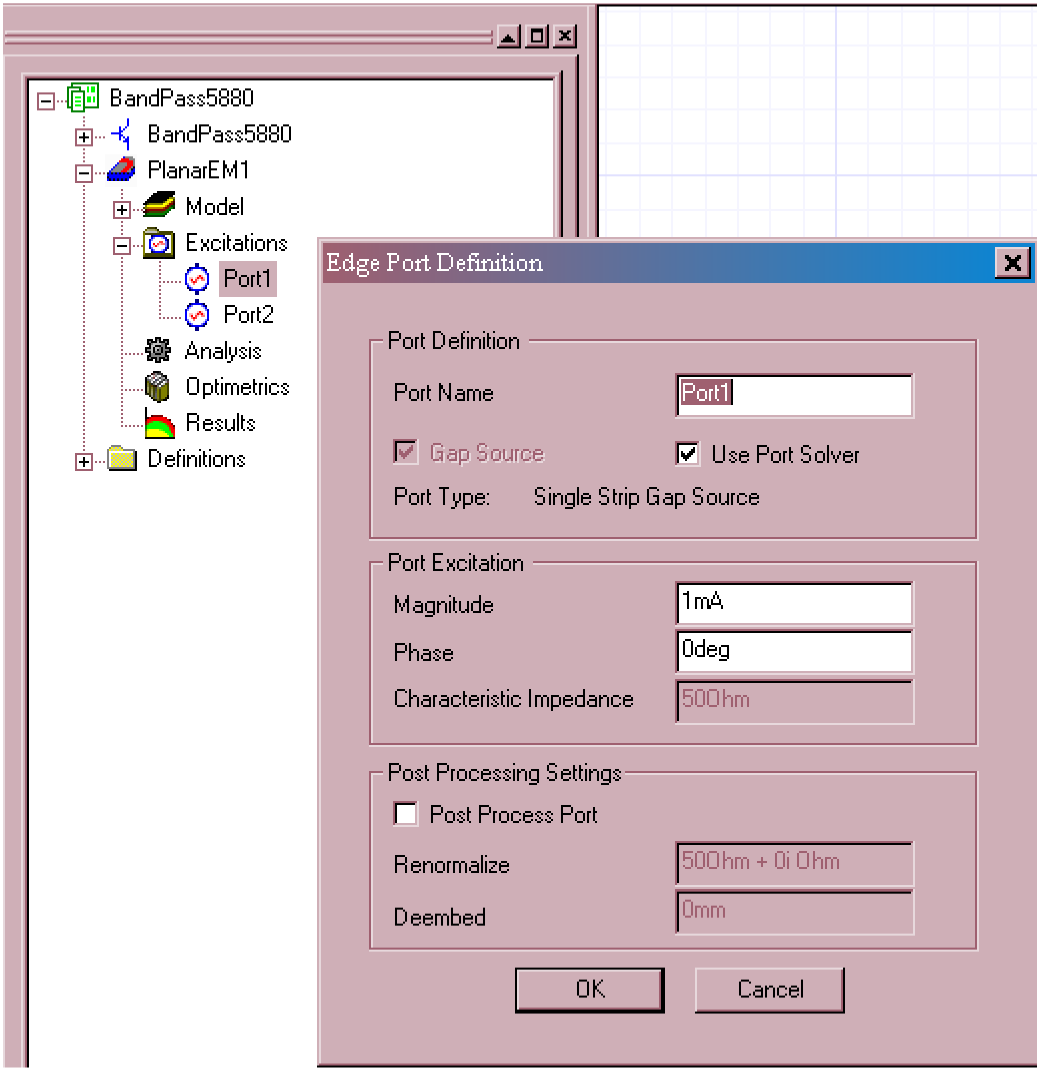
   The selected edge becomes a port.

6. Do the same on the other side so that you have two ports.

Your Planar EM design should now look like this:



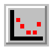
The **Project** window has been updated to show the addition of the two new ports. Expand the PlanarEM design, and then expand the Excitations listing in the tree to see the two new ports listed. The properties of the ports can be accessed by double-clicking on the port name.



Now that we have equipped the filter model with ports, we can analyze it to compare its behavior to its Circuit counterpart.

Set up and Solve the Planar EM Analysis

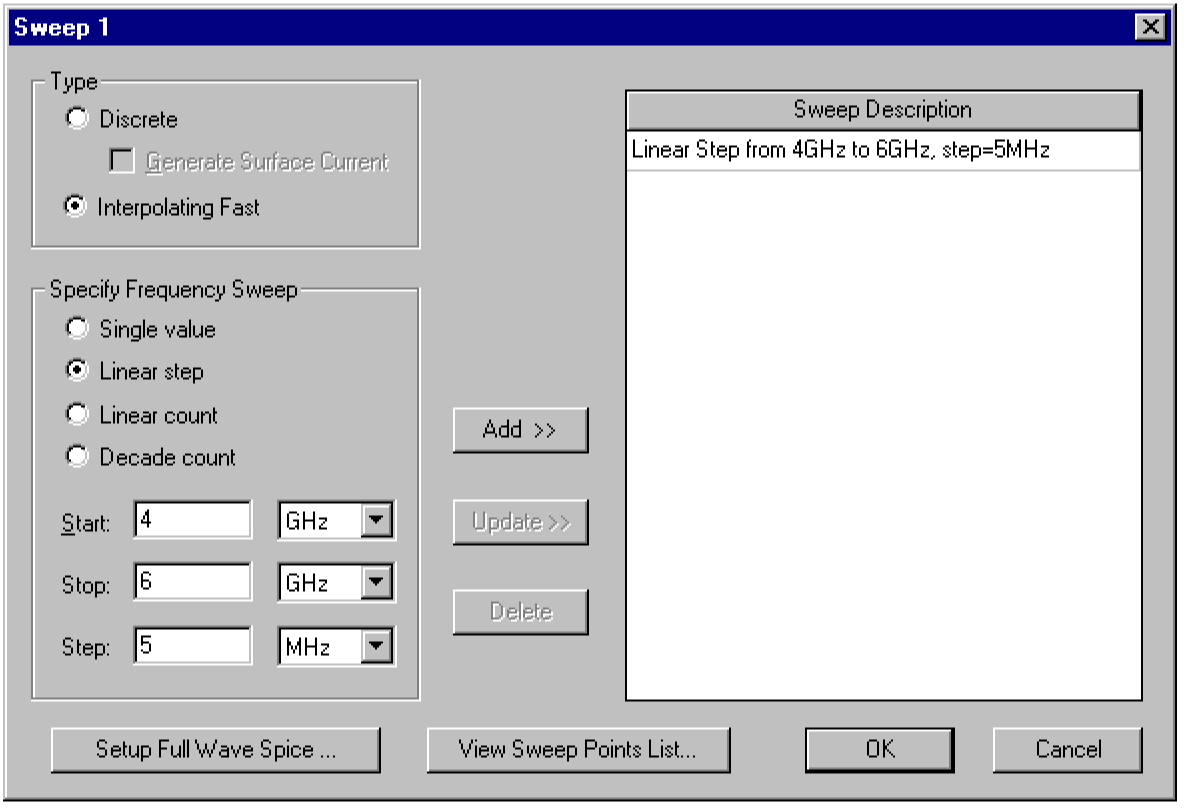
1. On the **Planar EM** menu, point to **Analysis Setup**, and then click **Add Solution Setup**  .

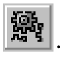
   We will accept the default values shown in the resulting dialog box, **Setup 1**. The most critical parameter is the mesh frequency, for which the default value of 10GHz will be automatically adjusted to 6GHz when we add a frequency sweep in the next step.
2. Click **OK**.
3. On the **Planar EM** menu, point to **Analysis Setup**, and then click **Add Frequency Sweep** .

   The **Sweep 1** dialog box opens.
4. In the **Type** group, select **Interpolating Fast**.
5. In the **Sweep Description** list, select the default entry, and then click **Delete**.
6. In the **Specify Frequency Sweep** group, select **Linear Step**.
7. Enter these sweep definitions:

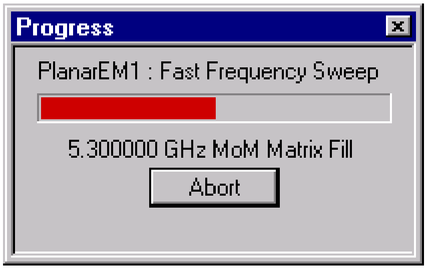
Start	4 GHz
Stop	6 GHz
Step	5 MHz

8. Click **Add**. The sweep specification should look like this:



9. Click **OK**.
10. On the **Planar EM** menu, click **Analyze** .

As the EM solver computes the model, the progress bar indicates its activity:

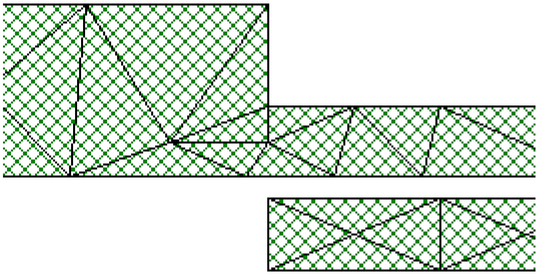


Advanced Note on Geometry Alignment:

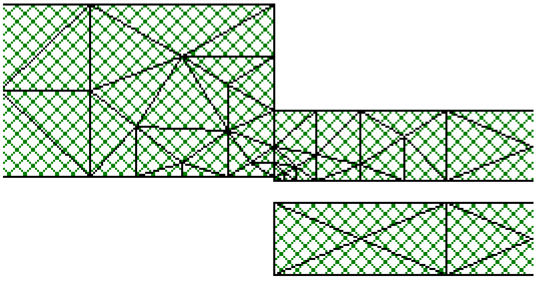
If the analysis deliberates for more than a minute at 4.0 GHz, or if Ansoft Designer displays an error that says that ratio of the mesh size is too large, the 50-ohm line sections are not properly aligned with the coupler input sections. To fix this:

1. Go to the project tree and expand the analysis icon so you can see Setup 1.
2. Right-click the **Setup 1** icon, and then select **Mesh Overlay**.

   Notice that the mesh density varies in regions associated with sudden changes in line width, such as the junctions between coupler lines of different widths.
3. Zoom into the area where one of the added 50-ohm lines adjoins the corresponding coupling input section. If you see a dense mesh, a small misalignment exists.



Normal Mesh



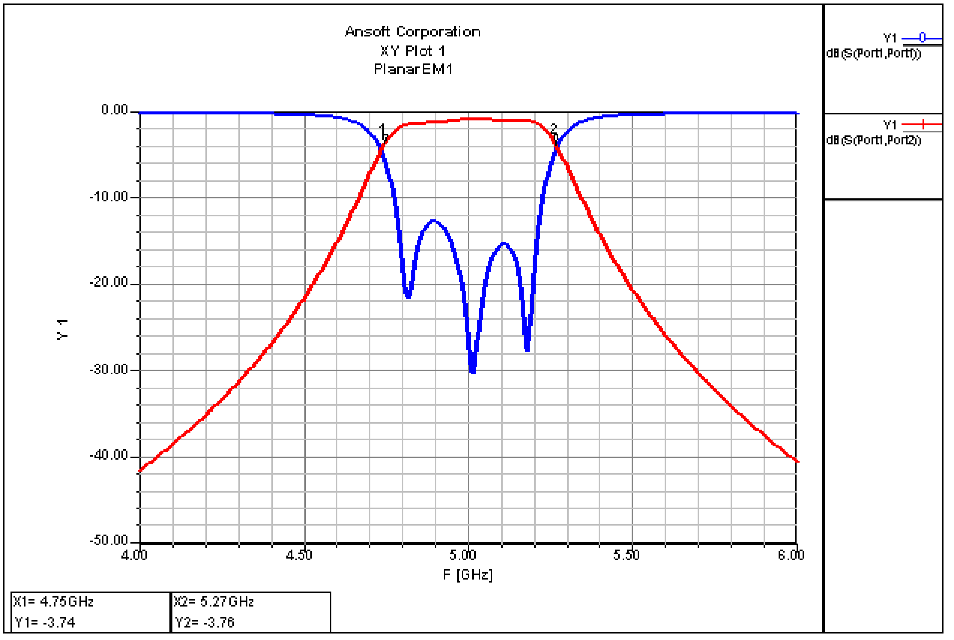
Mesh with Misalignment

4. Use the editor to correct the alignment and check the other 50-ohm line for the same problem.
5. Right-click the **Setup 1** icon, and then click **View Profile**. You can access a history of the current simulation as well as any previous simulations in the design. The profile includes a great deal of useful information, such as which machine the simulation ran on, the start time, stop time, and run time, the number of triangles in the mesh, and what frequencies were solved.

Create a Report of the Planar Results

1. On the **Planar EM** menu, point to **Results**, and then click **Create Report**.
2. In the **Create Report** dialog box, click **OK**.

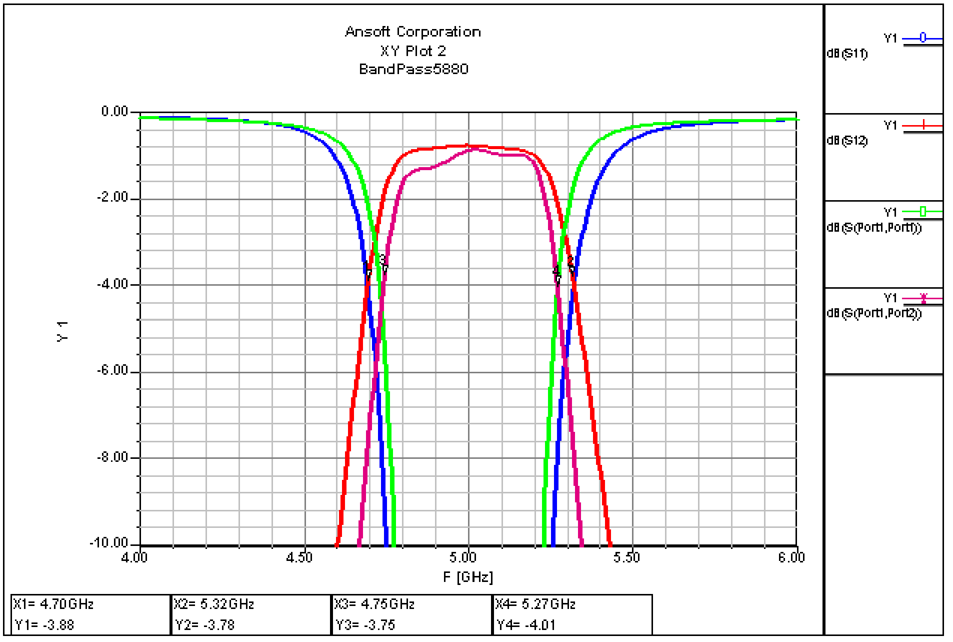
3. Working in the **Traces** dialog box, create a rectangular graph showing dB(S(Port1,Port1)) and dB(S(Port1,Port2)). Add the data markers as we did before. You should see:



The input match is slightly different, due to small differences in impedances in the coupled line section between the circuit and EM solvers. The calculated bandwidths are slightly different also:

	Circuit Model	**EM Model**
Fc1	4.70 GHz	4.75 GHz
Fc2	5.32 GHz	5.27 GHz
Bandwidth	0.58 GHz	0.52 GHz

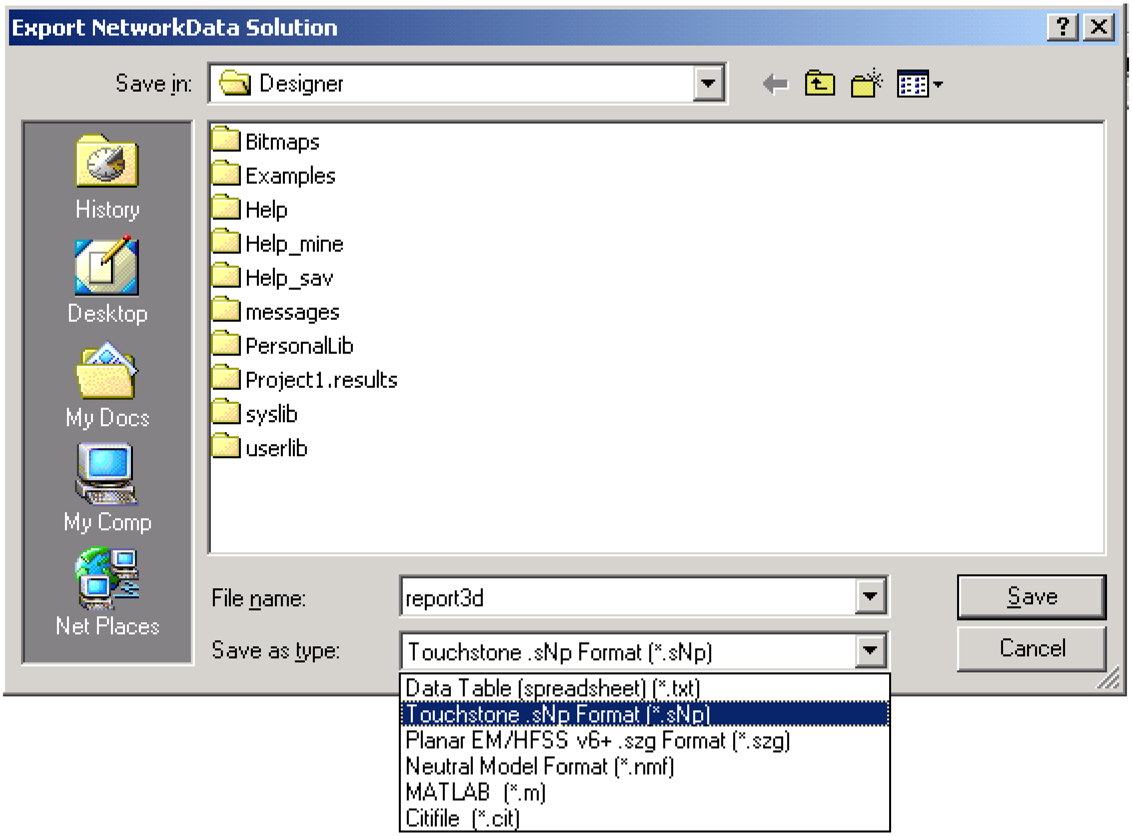
This reduction in bandwidth appears to be from the rounding of the passband in the EM simulation due to the difference in coupled line impedances from the Circuit analysis. Looking closely at the passband, we see:



The preceding plot was made by exporting the Planar EM solution, importing it into the Circuit analysis, and selecting the imported solution in the **Traces** dialog.

Export the Planar EM Solution

1. On the **Planar EM** menu, point to **Results**, and then click **View Profile** .
2. In the **Solution Data** dialog box, select the **Matrix** tab, and then click **Export**.



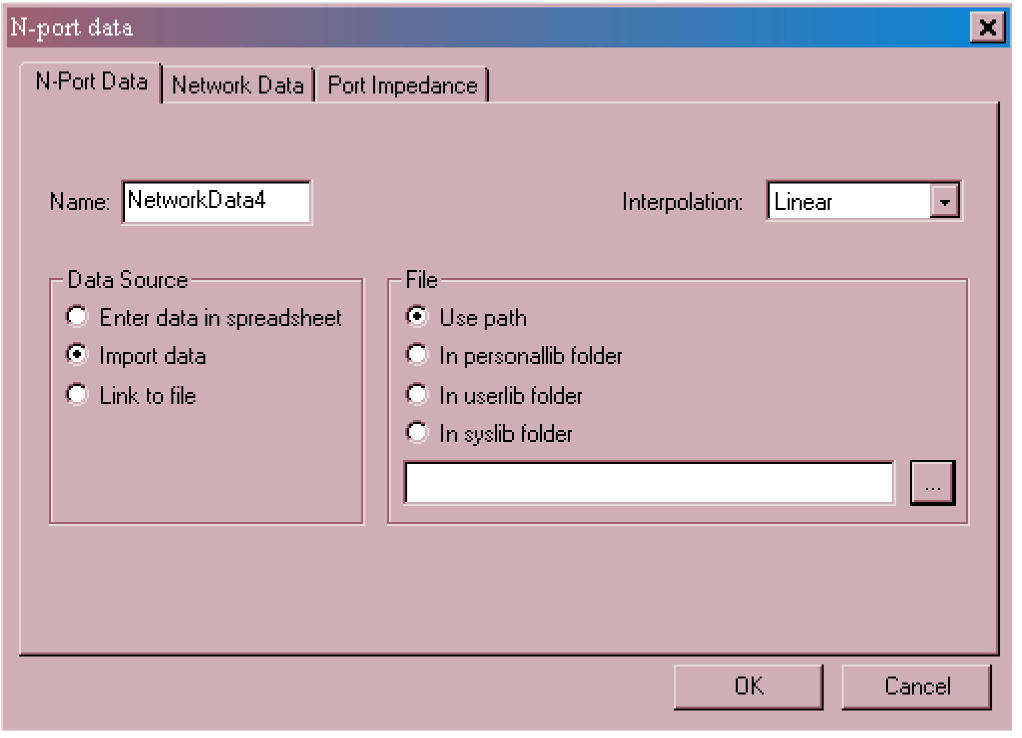
3. In the **Export NetworkData Solution** dialog box, select **Touchstone .sNp format (*.sNp)** in the **Save As** type list, and then click **Save**.
4. Note the full pathname of the file you saved.
5. Click **Close**.

Note the several alternate formats that are available in the **Save as type** list for exporting the Planar EM solution.

Import the Planar EM Solution into the Circuit Analysis

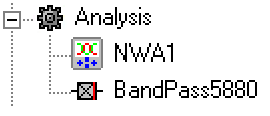
1. Bring the filter circuit schematic to the top by clicking it, selecting its entry in the **Window** menu, or double-clicking its icon in the project tree.
2. On the **Circuit** menu, click **Import Solution** to import network data.
3. In the **Import Solution** dialog box ensure that **Network Data** is selected, and then click **OK**.

The **N-port Data** dialog box appears:

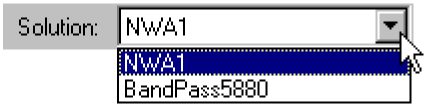


4. Under **Data Source** leave **Import Data** selected.
5. Under **File**, leave **Use Path** selected.
6. Click the browse button to browse to the EM-exported data.
7. Browse to the file you saved, and then click **Open**.
8. Click **OK**.

   The solution object appears beneath the NWA1 object in the circuit project tree:



The imported solution is now available as an option on the **Solutions** menu in the **Traces** dialog box:



Now that you've gained experience with linear analysis, you're ready to explore how to set up nonlinear analysis in Ansoft Designer.

Example: A Nonlinear Circuit

This section describes building a simple nonlinear circuit and setting up analyses for it. The circuit is kept simple so we can concentrate on the aspects specific to nonlinear circuits. The circuit is a diode detector with bias.

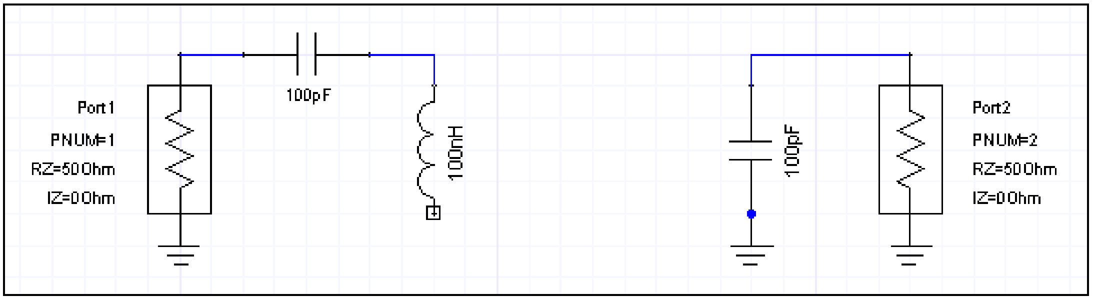
Build the Design

1. Start Ansoft Designer, if it is not already running.

   If Ansoft Designer is already running, create a new project by clicking **New** on the **File** menu.
2. Insert a Circuit design in the default new project or the project you created by selecting **Insert Circuit Design** on the **Project** menu.
3. In the **Choose Layout Technology** dialog box, click the **None** button. (We need not consider the physical layout of this circuit to explore nonlinear analysis, so the simplicity of a purely electrical simulation is acceptable.)
4. Click the **Components** tab in the **Project** window, and then locate and place the components below in the schematic window. Select and drag a component to place it in the schematic window. Press **ESC** to stop placement.

Component	Value	Location	Number
Capacitors	100pF	Lumped, Capacitors, Capacitor	2
Inductor	100nH	Lumped, Inductors, Inductor	1
Ground		**Draw** menu and **Schematic Draw** toolbar	1

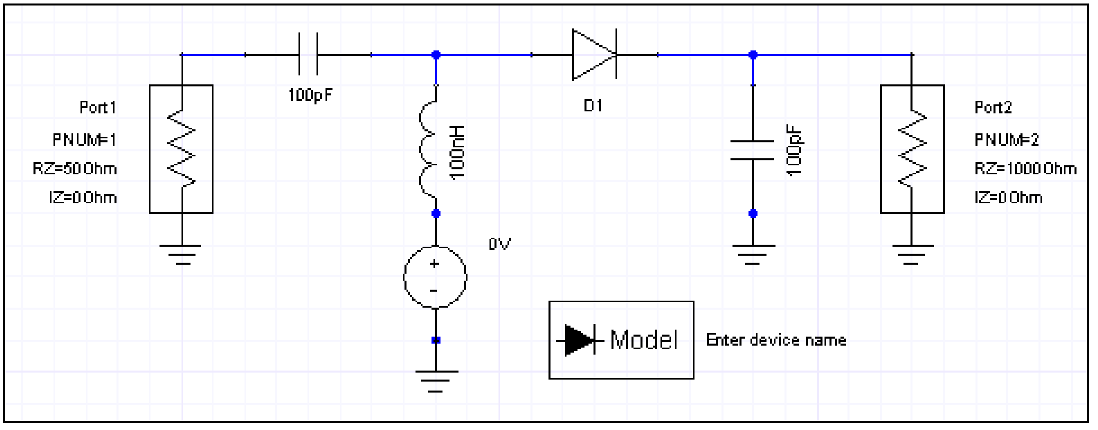
5. Also place ports as shown in the schematic below.



In this picture, the **Symbol** property for both ports has been set to **Microwave Port**, and **Flip About Y Axis** has been done for port 2.

6. Set the Port 2 termination to 1000 ohms, resistive, as follows:
   a. Double-click the **Port 2** symbol to open the **Port Definition** dialog box.
   b. In the **Termination** group, type **1000** in the **Re** box.
   c. Click **OK**.

7. On the **Component Tab**, expand the **Nonlinear** folder, then the **Diodes** folder.
8. In the **Diodes** folder, select and drag the **Microwave Model** component to place it between the shunt inductor and shunt capacitor in the schematic window, with its cathode end toward port 2
9. In the **Diodes** folder, select and drag **Microwave Model Data** to place this connectorless component somewhere on the schematic.
10. Open the **Sources** folder, and then the **Independent Sources** folder.
11. Select and drag the **Voltage** source and place it on the schematic with its positive terminal on the lower terminal of the inductor. Connect the other source terminal to ground.
12. Wire the diode as shown below.



The Model Data component specifies the device parameters for the diode. These parameters can be common to more than one instance of a diode, so they need only be specified once and can be referenced by as many diodes as desired.

13. Double-click the Model Data component, and then enter the following values:

Parameter	Value
DeviceName	demo
CT0	0.2p

The other parameters will take their default values.

14. Double-click the diode and enter the following values:

Parameter	Value
MOD	demo

This links the diode to the model data block of the same name, *demo*.

Set up the RF and DC Sources

Next, we'll set up the RF and DC sources. We want to be able to sweep the RF power and the DC bias, so we need to set up two parameters, which we can then sweep.

To set up the sources:

1. On the **Circuit** menu, click **Design Properties**.
2. Ensure that the **Value** option is selected.
3. Click the **Add** button, and then enter this parameter:

Name	Value
Pavs	10dBm

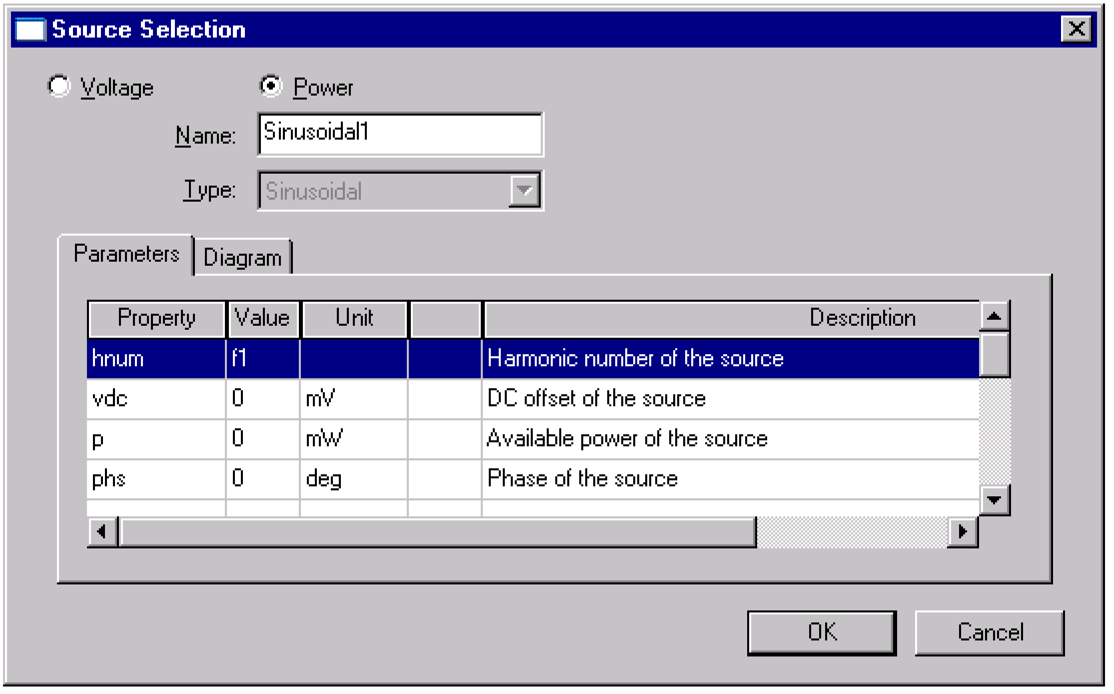
4. Click **OK** to accept the entry.
5. Click the **Add** button again and enter this parameter:

Name	Value
Vbias	0V

6. Click **OK** to accept the entry.
7. Click **OK** to close the **Properties** dialog box.
8. Double-click the left port component.

   This opens the **Port Definition** dialog box, through which you can add sources to the port. The default source type is Power.

9. Click the **Add** button to add a source.



We'll keep the **hnum** (harmonic number) property set to **f1**, signifying that this source will be considered as tone 1 in harmonic-balance analysis.

10. For the value of **p**, type Pavs.

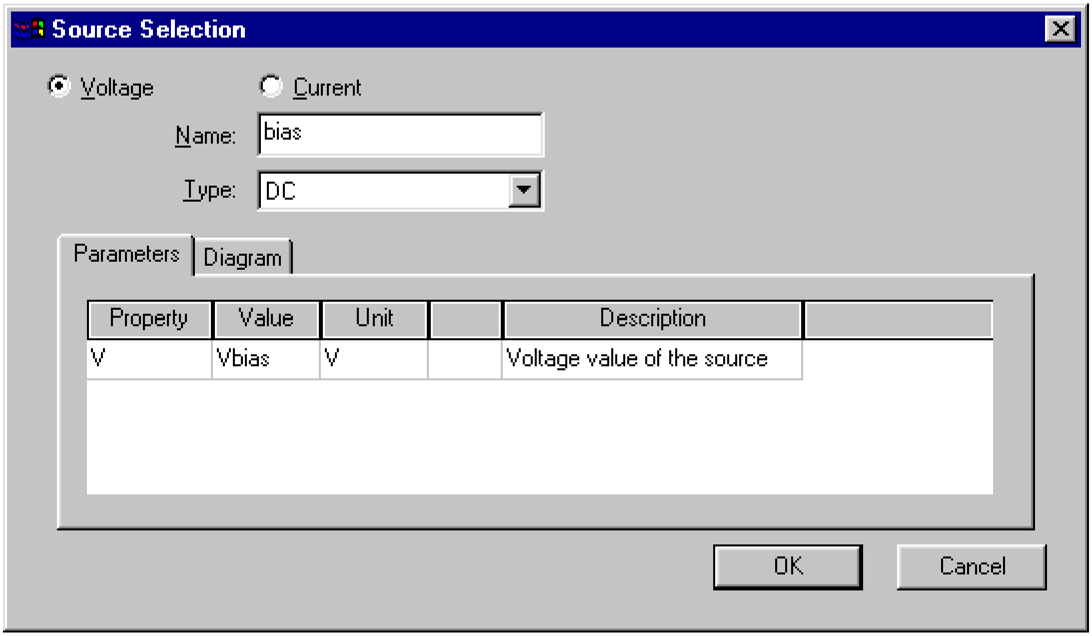
    This sets the source power equal to the value of the **Pavs** parameter defined earlier.

11. Click **OK** in the **Source Selection** dialog box, and then click **OK** in the **Port Definition** dialog box.

12. Double-click the voltage source in the circuit.

    The **Source Selection** dialog box opens.

13. Set **V** to **Vbias**, and set the name of the source to **bias**:



Multi-Tone

In addition to the Sinusoidal source type, a Multi-Tone type exists in the **Source Selection** dialog for use in defining ports. The Multi-Tone type has the same parameters as the Sinusoidal type with three additions:

- ncarries (number of carriers)
- cp (power per carrier)
- deltaf (frequency separation between carriers)

Sampling Rate and Number

The Source Selection dialog also supports the following system port parameters for use in discrete time analysis:

- nsamp (number of samples)
- sample_rate (sampling rate)

Set up a Harmonic-Balance Analysis

1. On the **Circuit** menu, click **Add Analysis Setup**.
2. In the **Analysis Type** list, choose **Harmonic Balance.**

3. In the Category list, click **1-Tone**, and then click **Next**.

   The **Harmonic Balance Analysis, 1-Tone** dialog box opens.
4. Set **No. of Harmonics** to **8**.
5. In the **Sweep Variables** list, select **F1**, and then type **2GHz** in the **Sweep/Value** cell.
6. Click **Add**.

   The **Add/Edit Sweep** dialog box opens.
7. In the **Variable** list, select **Pavs**.
8. Ensure that **Linear Step** is selected, and then type **–10dBm** for **Start**, **10dBm** for **Stop**, and **1dB** for **Step**.
9. Click **Add**.
10. Click **OK**.
11. Click **Finish**.

Rename and Save the Project

Now that the project's setup is complete, rename the project and save it:

1. Right-click the circuit design in the **Project** window, and then click **Rename**.
2. Type **NLdiode**, and then press **Enter**.
3. On the **File** menu, click **Save As**.
4. Type **NLdiode** in the **File name** text box, and then click **Save**.

Analyze the Project

1. Expand the **Analysis** icon in the project tree.
2. Right-click the **HB1Tone1** icon and then click **Analyze HB1Tone1** on the shortcut menu.

Create Graphs of Results

With the analysis completed, we will create three graphs:

- Port 2 DC voltage at versus Pavs
- Time-domain voltage waveform across the diode for each value of Pavs
- Time-domain current waveform into the diode for each value of Pavs

To create the graph of Port 2 DC voltage versus Pavs:

1. In the project tree, right-mouse click this project's **Results** icon, and then select **Create Report**.

   The **Create Report** dialog opens.
2. Click **OK**.

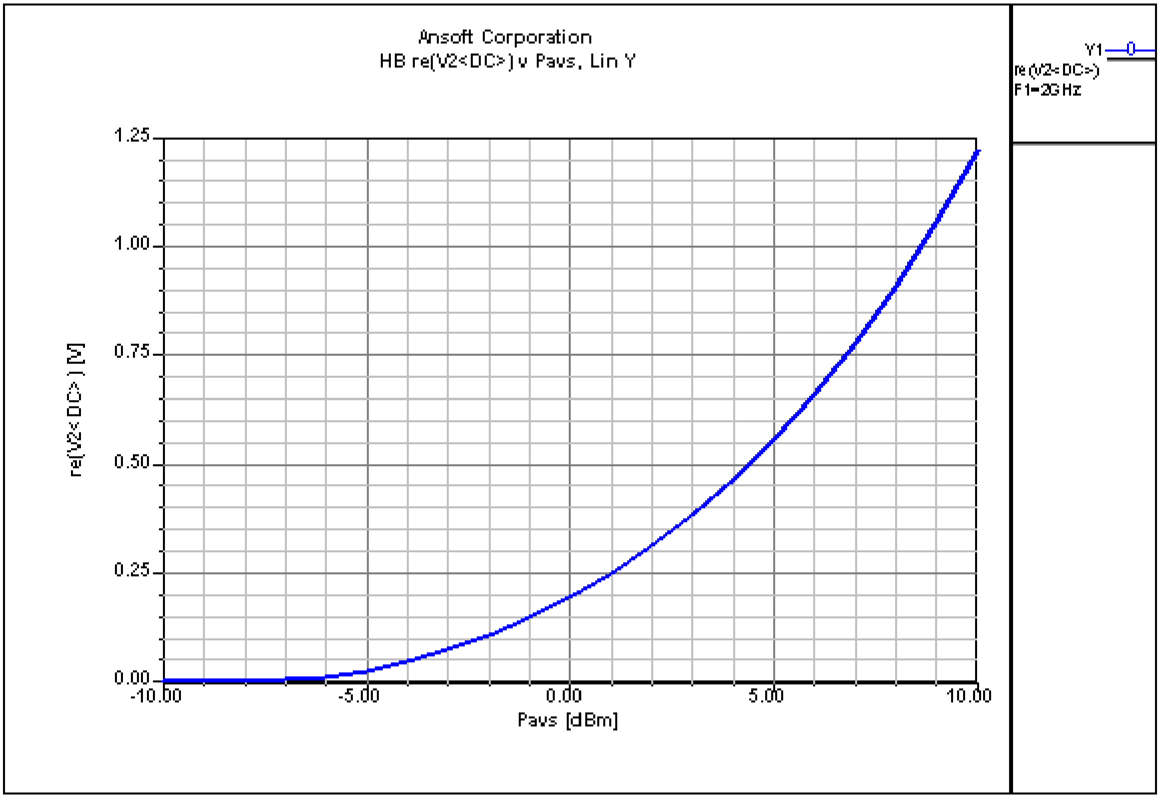
3. Select the following in the **Traces** dialog:

Domain	Sweep
Category	Voltage
Quantity	V2<DC>
Function	re

You must graph the real part of V2<DC> quantity because all harmonic phasor quantities are handled as complex numbers, including their DC components.

4. Click **Add Trace**.
5. Click **Done**.

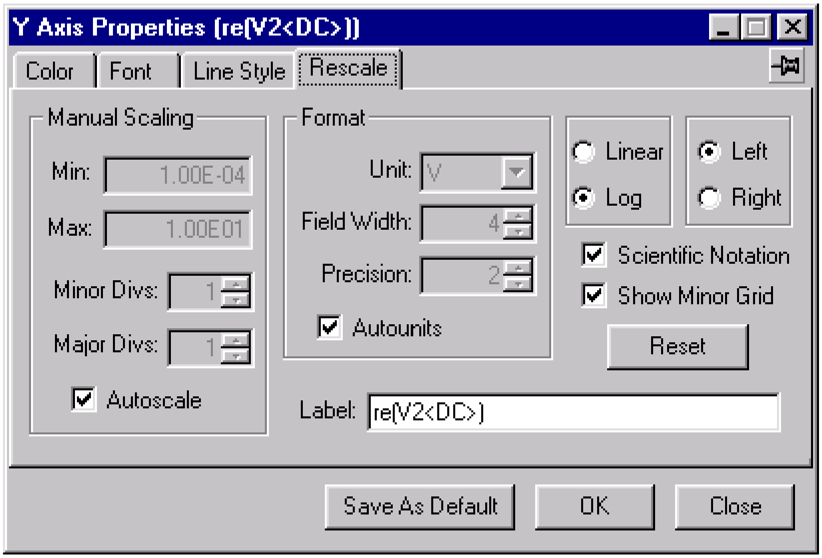
You should see the following graph:



For this picture, we have renamed the graph by selecting its icon in the project tree, right-clicking, selecting **Rename**, typing a new name, and pressing **ENTER**.

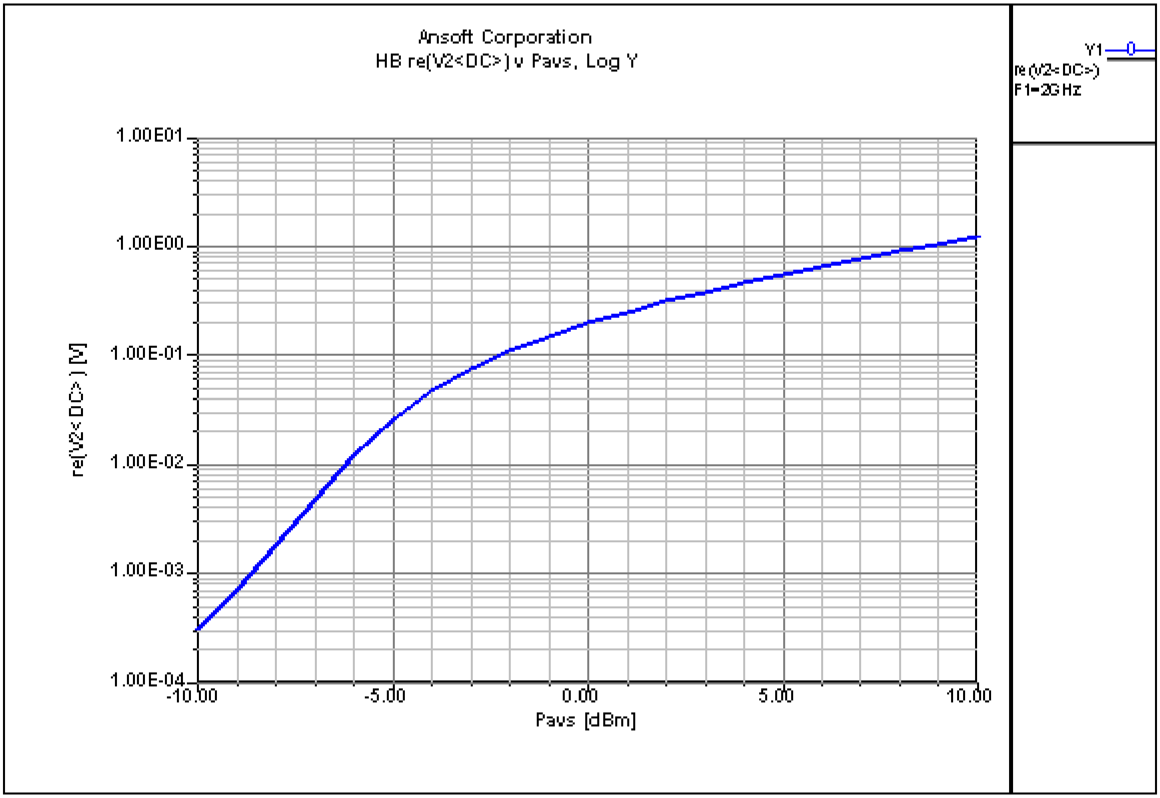
To view the linear region (in the logarithmic sense) of the detector's operation, change the Y axis scaling to logarithmic:

1. Double-click the Y axis to open the **Y Axis Properties** dialog box.
2. Select the **Rescale** tab, and then click **Log**.



3. Click **OK**.

The rescaled plot should look like this:



The linear region extends to about −5dBm.

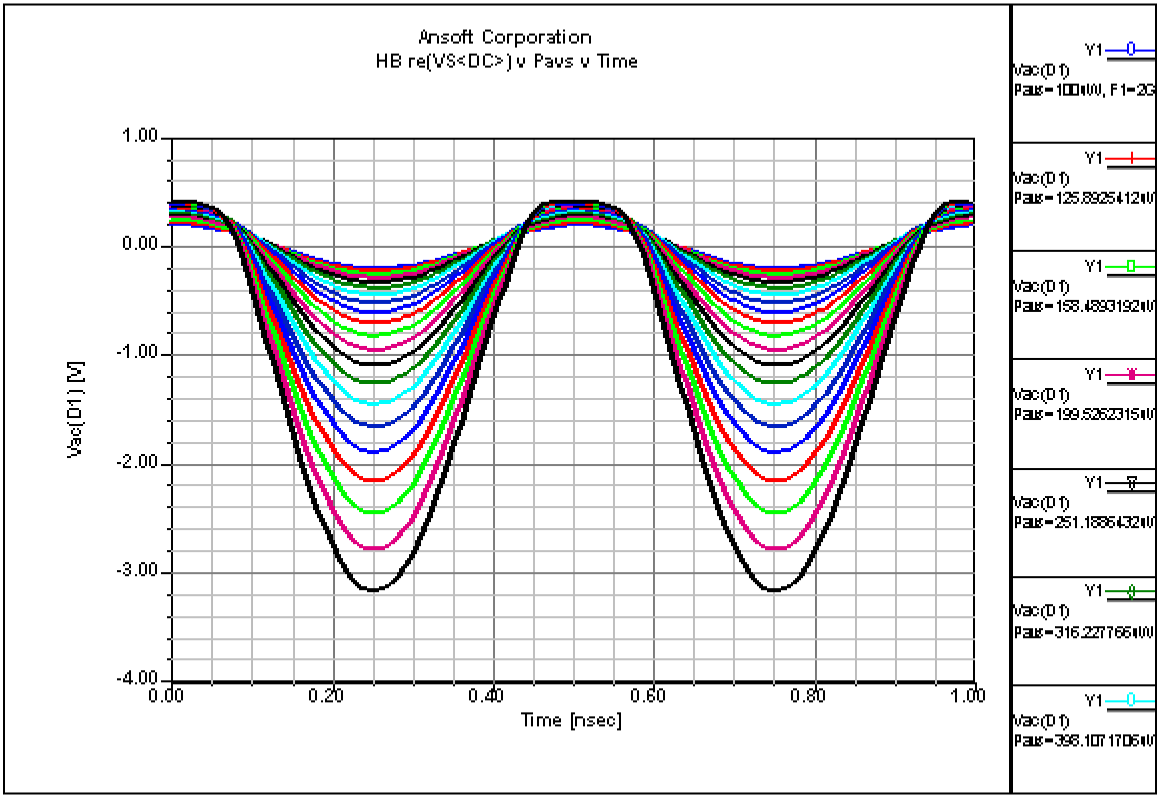
To create the graph of time-domain voltage waveform across the diode for each Pavs:

1. Start creating a standard, rectangular 2D report as described earlier.
2. In the **Traces** dialog box, select the following settings:

Domain	Time
Category	Voltage
Quantity	Vac(D1)
Function	none

3. Click **Add Trace**.
4. Click **Done**.

The resulting graph should look like this:



The graph clearly shows the clipping of the voltage across the diode at higher power levels. On the graph legend on the right-hand side, you can select the line for each level of Pavs, from -10 dbm to 10 dbm, and it will be highlighted on the graph. You can scroll down through the Pavs levels by clicking the black rectangle at the top of the legend window. The legend window can also be resized by clicking the vertical divider and be moved left and right across the **Results** window.

To create the graph of the time-domain current waveform into the diode for each Pavs:

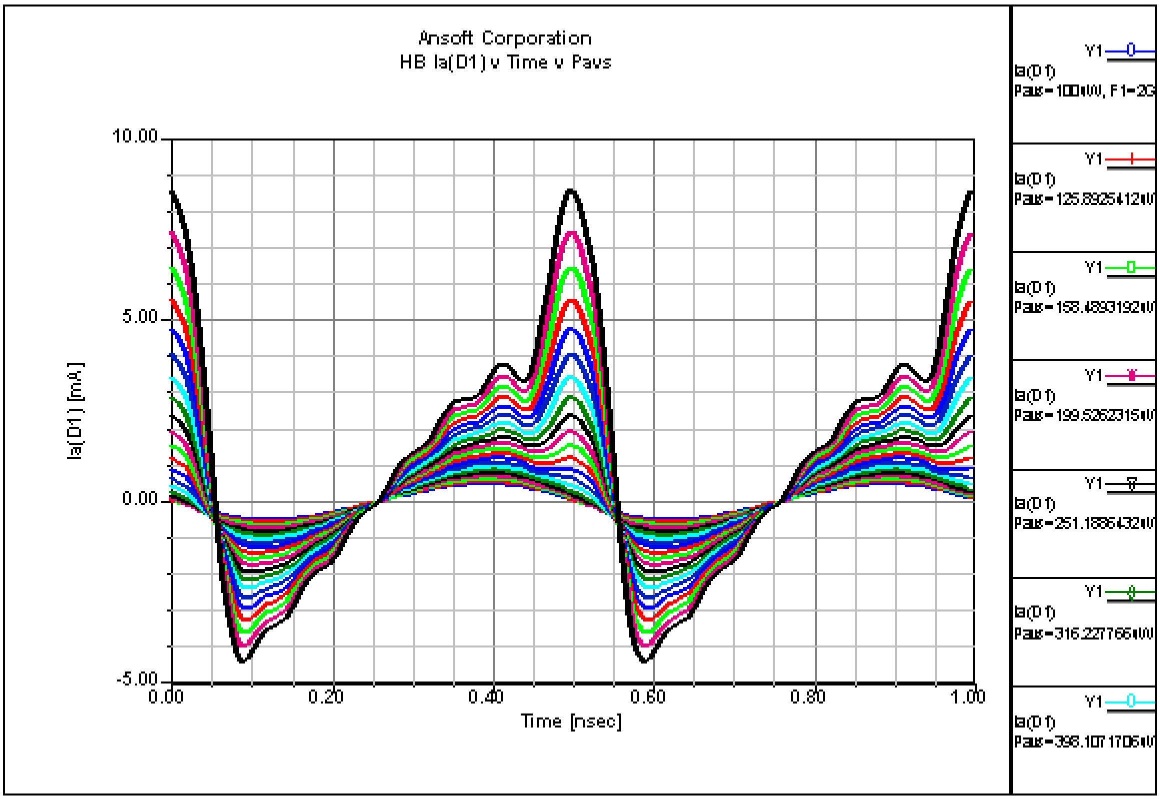
1. Right-mouse click the **Results** icon, select **Create Report**, and then click **OK**.

2. Select the following in the **Report** dialog:

Domain	Time
Category	Current
Quantity	Ia(D1)
Function	none

3. Click **Add Trace**.
4. Click **Done**.

The resulting graph should look like this:



The forward current through the diode is clearly visible during the time intervals in which the diode voltage is positive. The negative current and initial rise in forward current are due to energy storage in the diode capacitance.

Re-analyze the Circuit

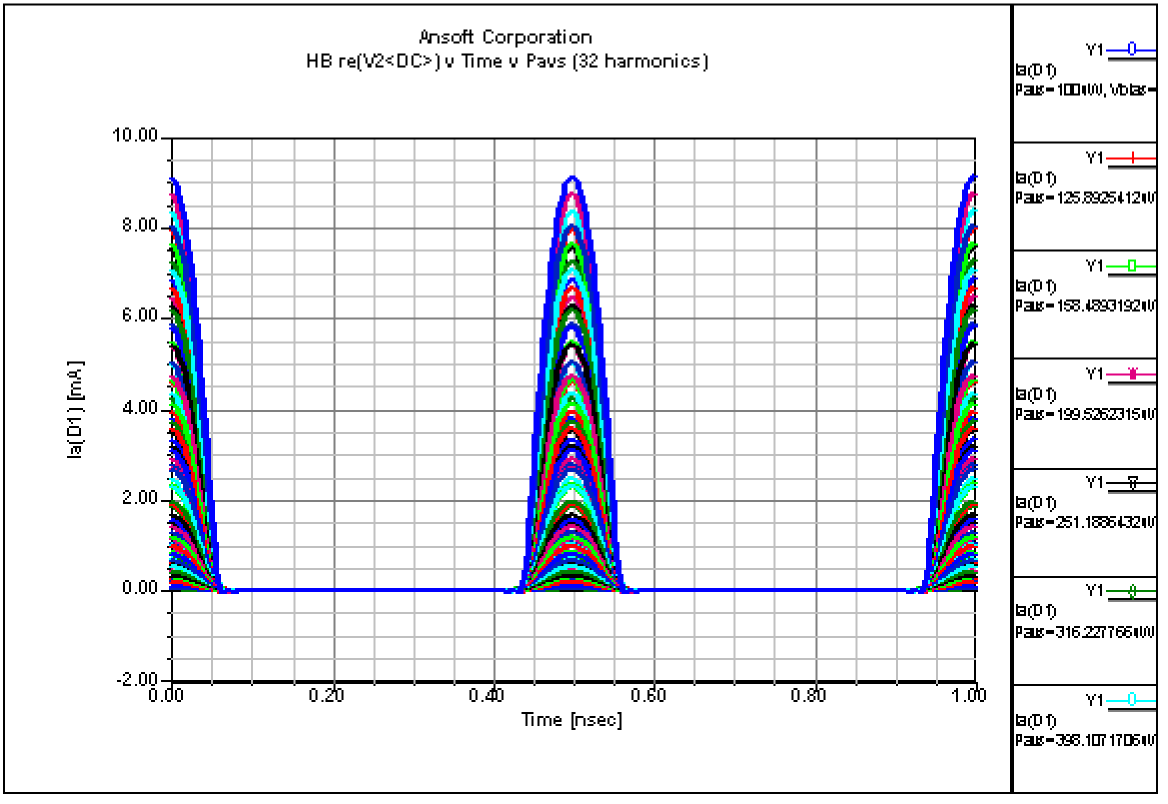
First, make these changes:

- In the Diode Model Data component, set **CT0** equal to 0.

- In the HB1Tone1 analysis setup, change the **No. of Harmonics** setting from 8 to 32.
- Re-analyze the circuit.

  The previously created graph of current into the diode still exists and has been updated automatically with the new simulation data. To bring the graph up to the front, you can double-click on the third graph under the **Results** item in the **Project** window.

  The current plot changes to:



  This shows the ideally rectified current in the diode.

- Change **CT0** back to 0.2p, and change **No. of Harmonics** back to 8.

Add a Sweep of the Bias Voltage

1. Double-click the **HB1Tone1** analysis setup to open the **Harmonic Balance Analysis, 1-Tone** dialog.
2. Click **Add**.

3. In the **Variable** list, select **Vbias**.
4. Specify a **Linear Step** sweep:.

Start	–200mV
Stop	200mV
Step	100mV

5. Click **Add**, and then click **OK**.

   The graph is now the original graph of current into the diode, which shows a linear frequency, a power sweep, and a voltage sweep.
6. Right-click the **HB1Tone1** icon, and then select **Analyze HB1Tone1**.
7. Click the V2<DC> graph to bring it to the top, right-click in the graph, and then select **Modify Report**.
8. Click the **Sweeps** tab. If you don't see a **Vbias** entry, reselect the **Solution** entry for HB1Tone1 to update the listing.
9. Click the **Apply To All Selected Traces** button to update the trace with the new sweep information.
10. Click **Done**.

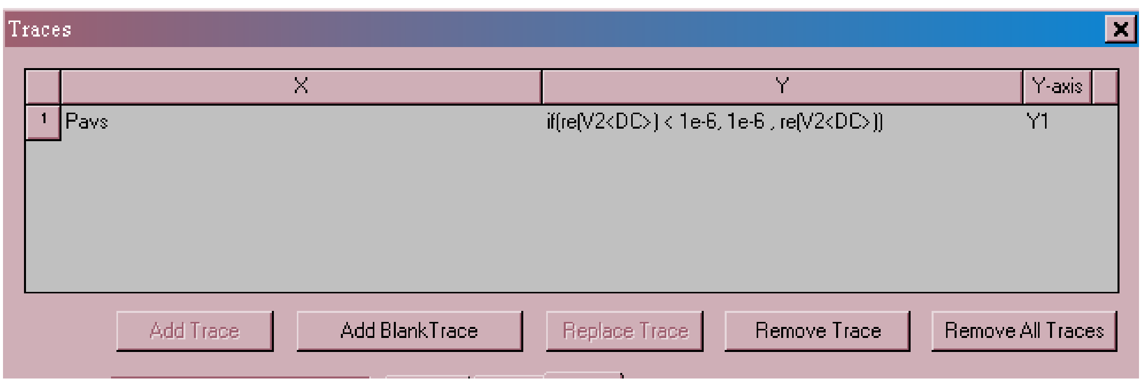
    You should see a set of curves now, but the axis changed back to a linear scale. This is because some of the data is negative and cannot be plotted on a logarithmic scale. To remedy this, we can define a function that avoids negative voltages.

    a. Right-mouse click on the graph and select **Modify Report**.

    b. Enter the following expression in place of re(V2<DC>):

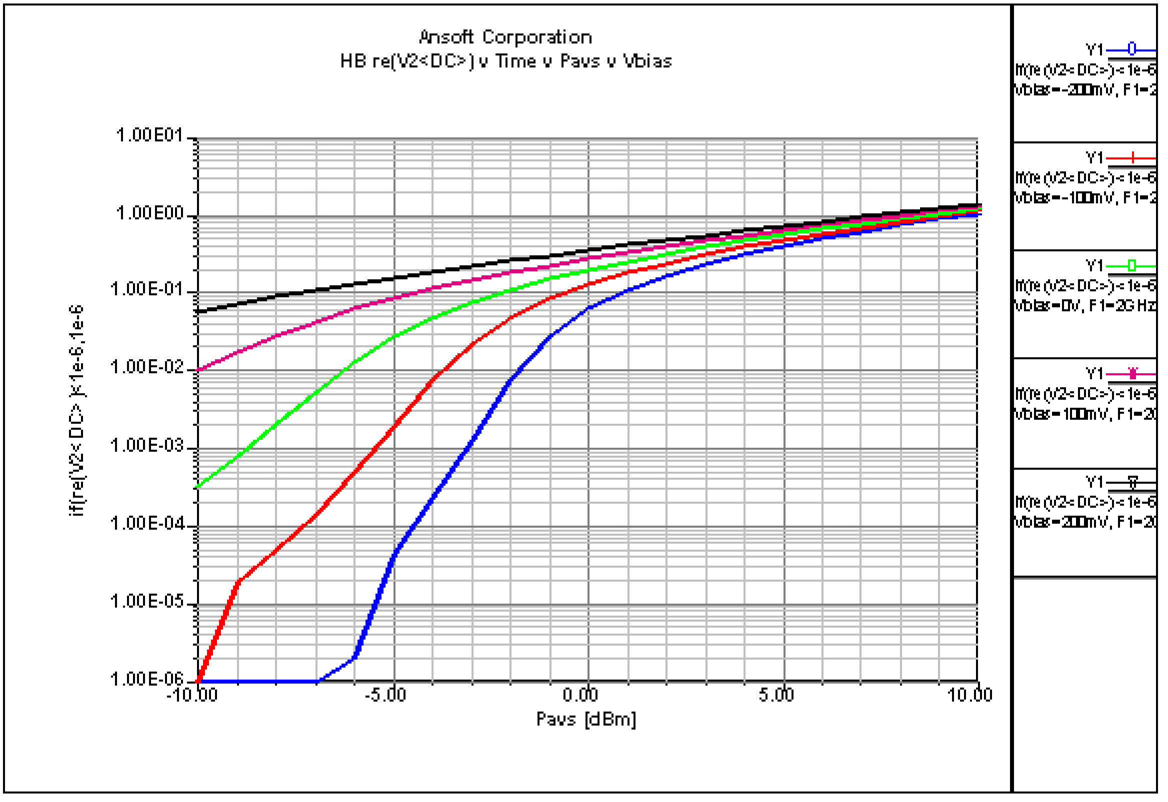
    ```
    if(re(V2<DC>) < 1e-6, 1e-6, re(V2<DC>))
    ```

    The IF function tests the condition in the first argument and if true, returns the value of the second argument; otherwise, it returns the value of the third argument. This expression states, “If re(V2<DC>) is less than 1E–6 (1 microvolt), then let the value be 1E–6, otherwise use re(V2<DC>).”



c. Click **Done**.

Now we can apply the log scaling to obtain this graph:

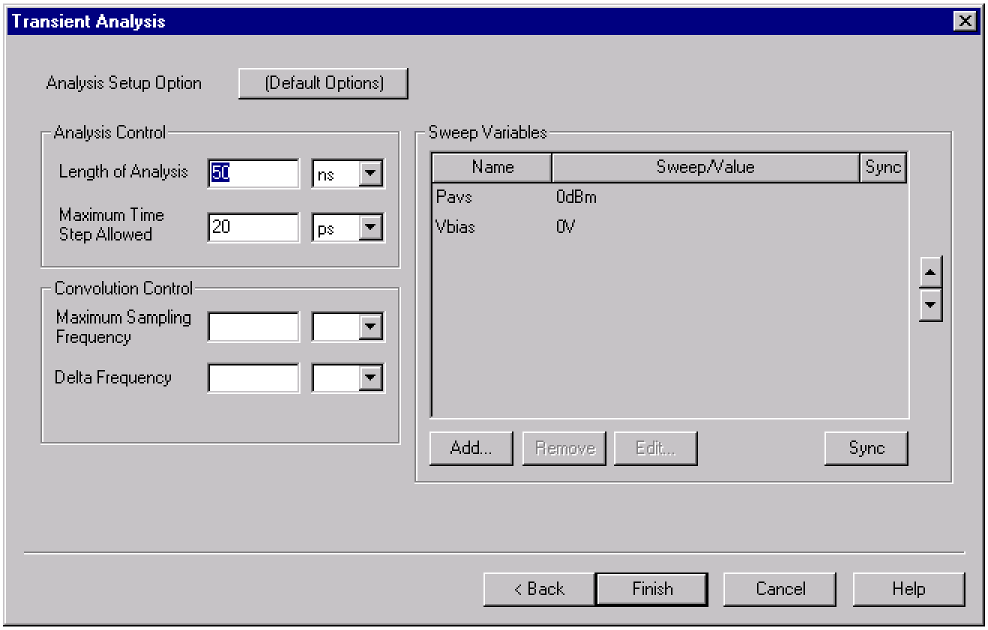


Perform a Transient Analysis

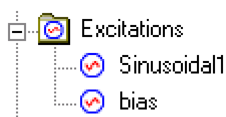
1. Right-click the **Analysis** icon in the project tree, and then select **Add Analysis Setup**.

   The **Analysis Setup** dialog opens.
2. In the **Analysis Type** list, select **Transient Analysis**, and then click **Next**.
3. Type **50ns** in the **Length of Analysis** box, and then type **20ps** in the **Maximum Time Step Allowed** box.
4. In the **Sweep Variables** group, click **Add**.
5. In the **Variable** list, select **Pavs**.
6. Click **Single Value**, type **0dBm** in the **Value** box, click **Add**, and then click **OK**.
7. In the **Sweep Variables** group, click **Add**.
8. In the **Variable** list, select **Vbias**.

9. Click **Single Value**, type 0V in the **Value** box, click **Add**, and then click **OK**.
10. Click **Finish**.



11. When we set up the excitation source, we left blank the frequency field for transient analysis. To edit the source, do the following:
    a. Expand the **Excitations** icon:



    b. Double-click the **Sinusoidal1** icon.

       The **Source Selection** dialog opens.
    c. Scroll to the **f** parameter, type **2GHz** in its **Value** cell, and then click **OK**.
12. Right-click the **Transient1** icon, and then select **Analyze Transient1**.

Create a Rectangular Plot of Results

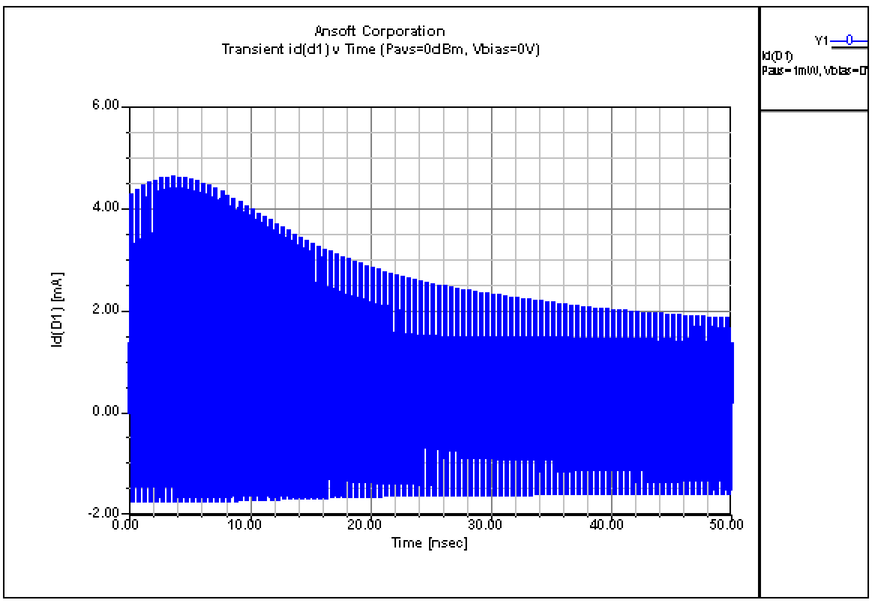
Create a standard rectangular plot after the analysis concludes.

1. Right-click the **Results** icon, and then select **Create Report**.
2. In the **Traces** dialog box, make these selections:

Solution	Transient1
Category	Current
Quantity	Id(D1)
Function	None

3. Click **Add Trace**.
4. Click **Done**.

   The following graph will appear:



The solution reaches steady state at around 50 ns. To view a close-up of the waveform, right-click, select **Zoom In**, and then drag a box around the region you want to magnify.

Now that you've gained experience with nonlinear analysis, you're ready to explore how to set up and use hierarchical designs in Ansoft Designer.

Creating Hierarchical Designs

A hierarchical design is a Circuit or System design that contains *subdesigns*—designs placed in it. Any number of hierarchical levels can be created, and any number of subdesigns can be placed in another design. The design at the top of such a structure is commonly called the *top-level design* or *top-level circuit*. A design that contains a subdesign is called the *parent* of that subdesign. Subdesigns can be *subcircuits* (Circuit simulations), subsystems (System simulations), or Planar EM simulations, depending on the parent design type:

- Circuit designs may contain subcircuits and Planar EM simulations
- System designs may contain subcircuits, subsystems, and Planar EM simulations

You can set up analyses at any level in a hierarchy, analyze, and then view results of the analyses. An analysis performed on a design at a level other than top includes that design and all of its subdesigns, if any, but not its parent design or designs at any higher levels.

All of the many possible analyses you may specify for various designs at different levels of a hierarchy are entirely independent. For example, you can set up a linear network analysis for a filter at an intermediate level of a hierarchy to obtain its *S*-parameters over a frequency range, and you can set up a spot-frequency nonlinear harmonic-balance analysis for the hierarchy's top-level circuit. You can then analyze only the filter subcircuit without having to analyze the entire hierarchical design over the filter frequency sweep—or you can run just the harmonic-balance analysis without sweeping the filter. (Analyzing a subcircuit that contains nonlinear devices requires special attention, as the devices' biasing and excitation may originate in higher-level circuitry not included in the analysis.)

There are two approaches to creating a hierarchical design: bottom-up and top-down. We will use Circuit designs to illustrate each.

The Bottom-Up Approach

1. Start Ansoft Designer, if it is not already running.

   If Ansoft Designer is already running, create a new project by clicking **New** on the **File** menu.
2. Insert a Circuit design in the default new project or new project you created by selecting **Insert Circuit Design** on the **Project** menu.
3. Create a design at the project level by selecting **Insert Circuit Design** on the **Project** menu.
4. Repeat step 3 for each subdesign you will need.
5. To build the hierarchy:
   a. Right-click a design icon in the project tree, and then select **Copy**.
   b. Select the icon for the design into which you want to copy the subcircuit, right-click, and then select **Paste**.

      The **Synchronize Design** dialog box opens. In this dialog box, you can select whether the subdesign will be incorporated into the hierarchy of which the parent is part (**Incorporate**, the default choice) or will merely be linked to it [**Keep independent (black box)**]. When choosing **Incorporate**, you can also specify a name for the new subdesign.

c. Click **OK**.

A symbol representing the subcircuit appears in the parent schematic. You can place and wire it as desired.

The Top-Down Approach

1. Start Ansoft Designer, if it is not already running.

   If Ansoft Designer is already running, create a new project by clicking **New** on the **File** menu.

2. Insert a Circuit design in the default new project or new project you created by selecting **Insert Circuit Design** on the **Project** menu.

   This will be the top-level circuit.

3. For each subcircuit you will need in the top-level circuit:

   a. Make sure the top-level circuit is active in the schematic editor.

   b. On the **Circuit** menu, point to **SubCircuit,** and then click **Add SubCircuit**.

4. Repeat step 3 at each level of the hierarchy as needed.

5. To reuse subcircuits at multiple places in a design:

   a. In the project tree, right-click the icon for a design you want to reuse, and then select **Copy**.

   b. Right-click the icon for the design into which you want to copy the subcircuit, and select **Paste**.

      A symbol representing the subcircuit appears in the parent schematic. You can place and wire the subcircuit as desired.

As you create subcircuits, you will probably want to rename them. To do this:

1. In the project tree, click the name of the icon for the design that you want to rename.

2. Click the icon name again.

   The icon name changes to an editable string that you can modify. When you renamed the design name to your liking, press **ENTER**.

Ansoft Designer creates a generic rectangular symbol for each new subcircuit. You can customize the symbol for a subcircuit as follows:

1. Expand the folder in the project tree labeled **Definitions**, and then expand the **Symbols** subfolder.

2. Double-click the symbol icon that corresponds to the subcircuit symbol you want to edit.

   The symbol opens for editing in Ansoft Designer's symbol editor.

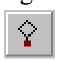
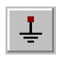
3. When you're finished editing the symbol, click **Update Project** on the **Symbol** menu to propagate your changes through the project.

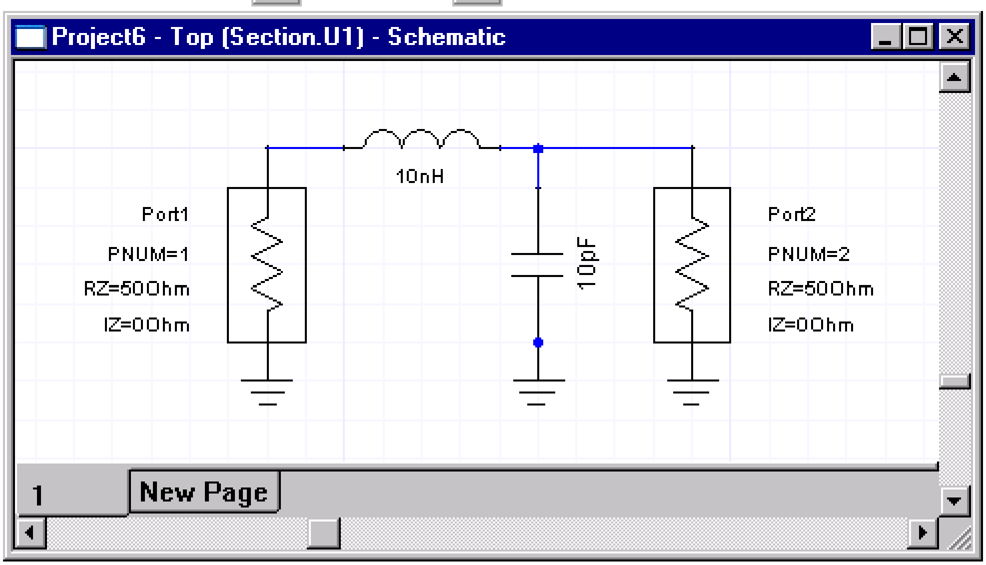
Example: Hierarchical Schematic Editing

This exercise shows you how to build a simple two-level schematic using the top-down approach. The circuit is a simple design that demonstrates the basic procedure.

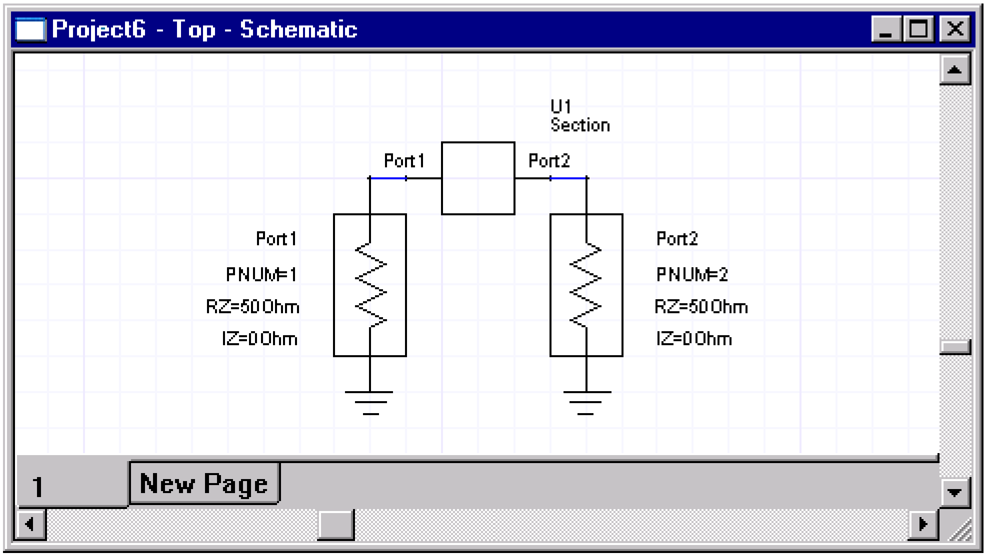
1. Start Ansoft Designer if it is not already running.

   If Ansoft Designer is already running, create a new project by clicking **New** on the **File** menu.
2. Insert a Circuit design in the default new project or new project you created by selecting **Insert Circuit Design** on the **Project** menu.
3. In the **Choose Layout Technology** dialog box, click **None**.
4. Rename the design as **Top** by clicking twice on the design name in the project window.
5. Add a subcircuit to the top-level circuit:

   On the **Circuit** menu, point to **SubCircuit**, and click **Add SubCircuit**.
6. Rename the subcircuit to **Section** by clicking twice on the name of the corresponding design icon in the **Project** window.
7. Click the subcircuit schematic window to make sure that the focus is in that window.
8. Click the **Components** tab in the **Project** window, expand the **Lumped** folder, and then expand the **Inductors** library.
9. Select and drag the **Inductor** component to place one inductor in the schematic. (Press **ESC** to end the multiple-placement mode if it's enabled.)
10. Expand the **Capacitors** library in the **Lumped** folder.
11. Select and drag the **Capacitor** component to place a capacitor in the schematic, rotating it to vertical by clicking the **R** key.
12. Place two ports  and a ground , and wire the circuit so it looks like this:



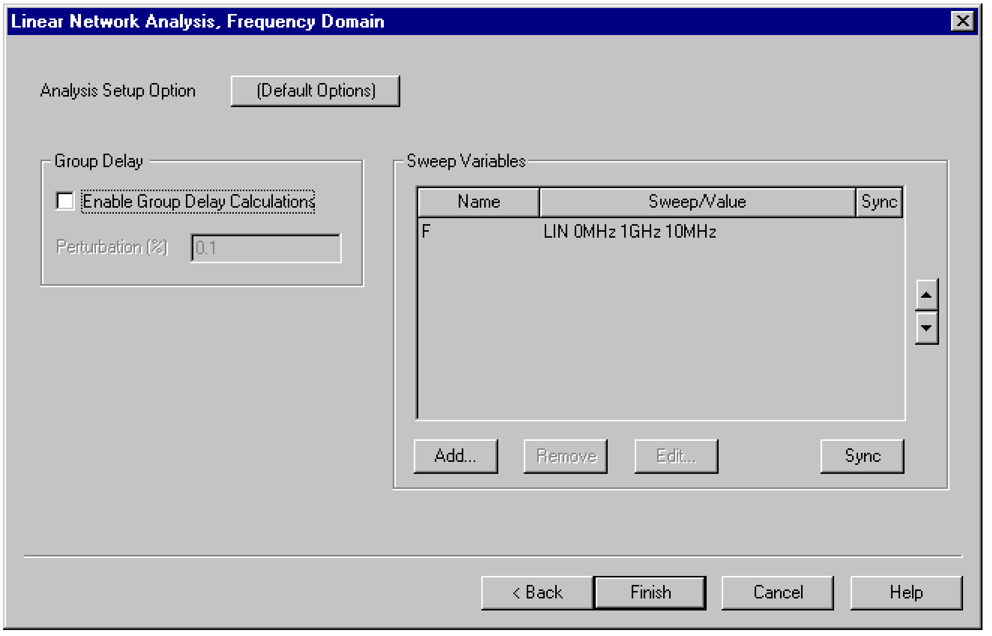
13. Switch to the Top (parent) schematic. The addition of the two ports to the subcircuit makes two corresponding pins appear on the Section subcircuit symbol in the Top (parent) schematic. Add two ports and connect each to a pin so the Top schematic looks like:



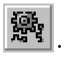
Add an Analysis Setup

1. Make sure the Top circuit is active in the schematic editor.
2. On the **Circuit** menu, click **Add Analysis Setup**.
3. Select **Linear Network Analysis** for **Analysis Type** and accept the default **Category** setting

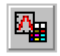
of **Frequency Domain**, and set up a linear sweep from 0 to 1 GHz in steps of 10 MHz:

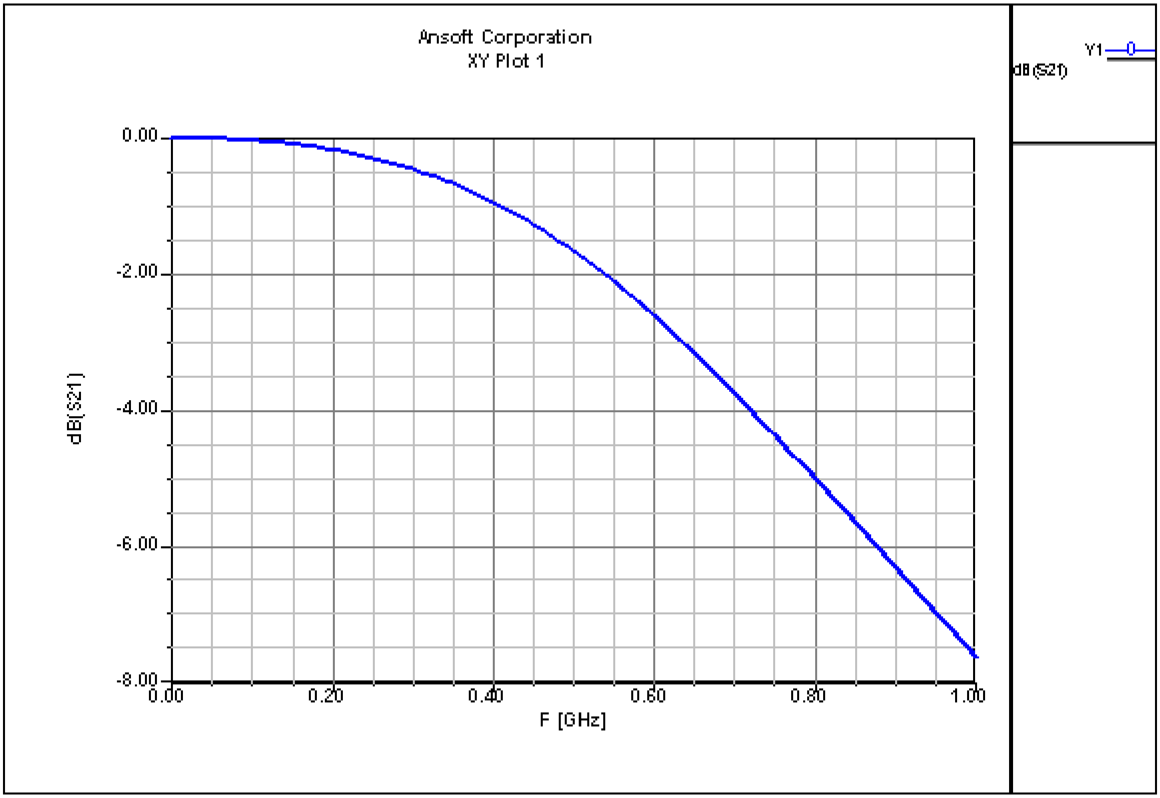


Analyze the Circuit

- On the **Circuit** menu, click **Analyze** .

Create a Report of Results

1. On the **Circuit** menu, click **Create Report**  .
2. Create a rectangular graph of dB(S21):



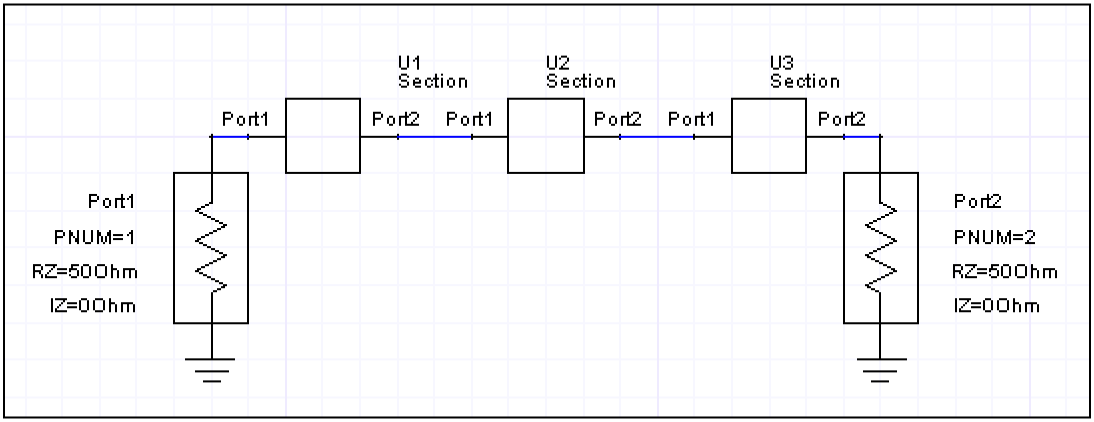
Now you will put more of these subcircuits into the Top circuit.

1. Right-click the subcircuit **U1: Section** in the project tree, and then click **Copy** on the shortcut menu.
2. Right-click the Top circuit, and then click **Paste** on the shortcut menu.

   If multiple placement is enabled for components, you'll be able to place another copy of U1:Section in the schematic after placing the first copy. Click in the schematic window to place a second copy, and then end placement by pressing ENTER, SPACE, or ESC, or by right-clicking and selecting **Place and Finish**, **Finish**, or **Cancel**.

   If multiple placement is not enabled, placement will end after you place the first copy of U1:Section into Top. To place a second copy, display the **Edit** menu, and then click **Paste**.
3. You should now see three instances (copies) of the Section subcircuit in the Top circuit.

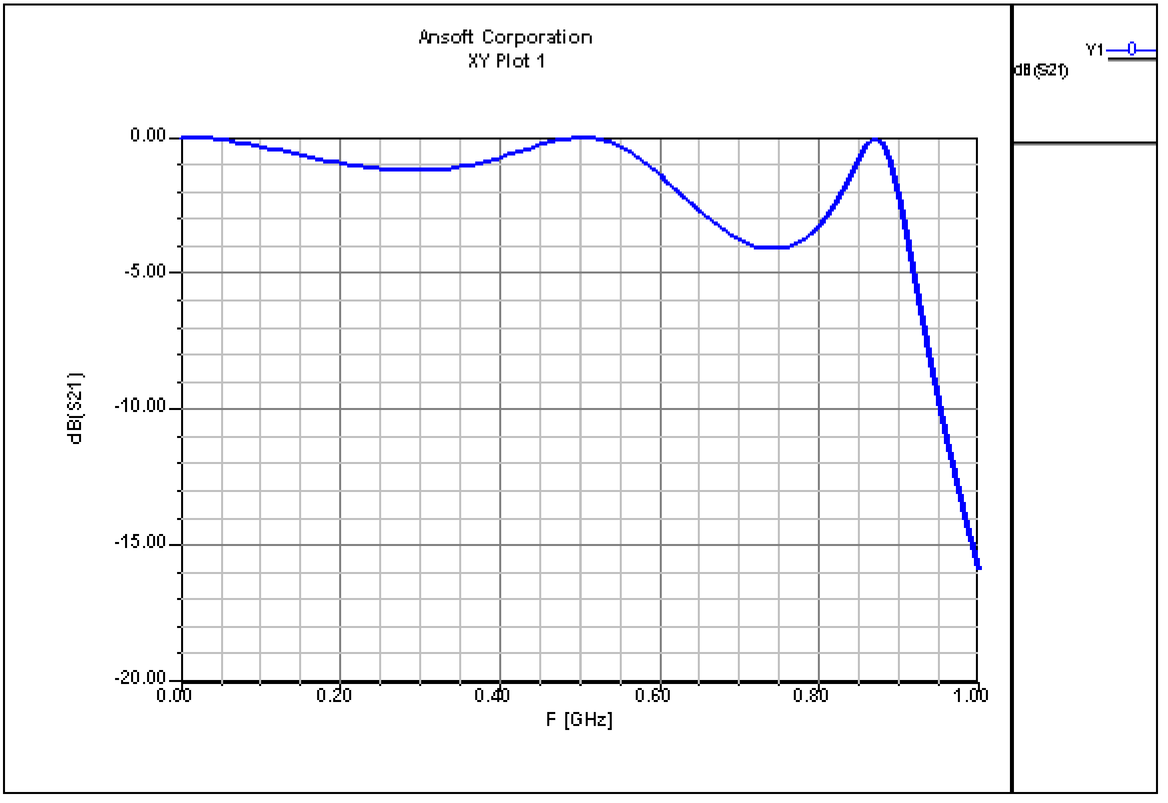
Arrange them between the ports in a cascaded fashion to look like this:



Re-Analyze the Circuit and Save the Project

1. On the **Circuit** menu, click **Analyze** .

   Now the graph looks like this:



2. Save the project by clicking **Save** on the **File** menu. Type “Hierarchial” into the **File name** box, and then click **Save**.

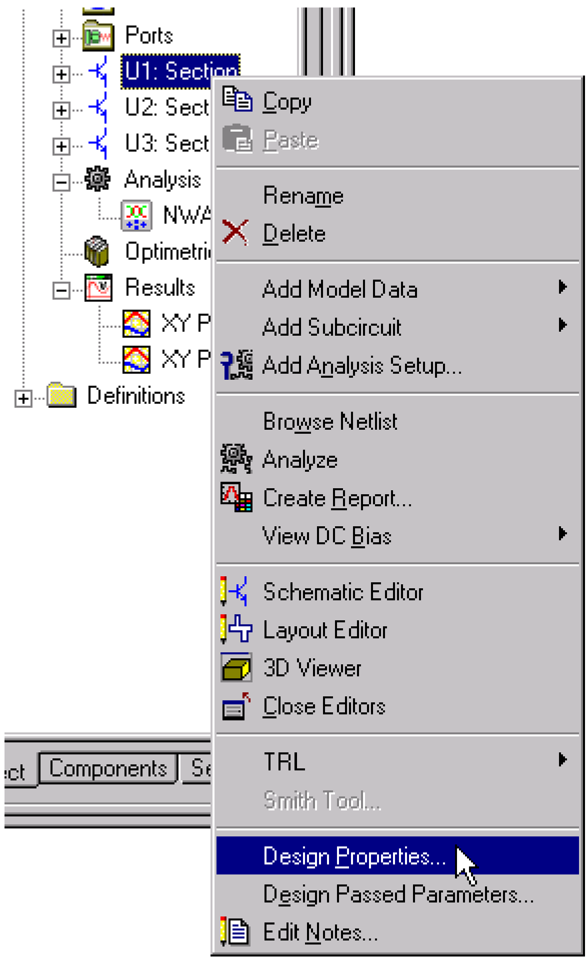
By selecting a design in the **Project** window, you can copy and paste single subcircuits or hierarchical circuits into other designs *within a project or into other projects*. It is all the same procedure.

Working with Parameter Defaults and Passed Parameters

Subcircuits can be parameterized, meaning that values for properties of components within them can be assigned to alphabetic or alphanumeric placeholders (*parameters*) for which values can be inherited (passed in) from their parents. A property value inherited in this way is known as a *passed parameter.*

Subcircuits cannot accept parameter values from their parents by default. You must enable parameter passing for every subcircuit component property the value of which you want set by the subcircuit parent. To do this for a given property value, you must create a special placeholder called a *parameter default*, assign it a valid default value, and then use that parameter default as a value placeholder in a subcircuit component or components. This exercise demonstrates how to enable and use passed parameters in Ansoft Designer.

1. Select one of the subcircuits in the project tree, right-click it, and then select **Design Properties**:



The **Properties** dialog box opens.

2. Add a parameter default as follows:

   a. In the **Parameter Defaults** tab, ensure that **Value** is selected, and then click **Add**.

      The **Add Property** dialog box opens.

b. Enter **L1** for **Name** and **10nH** for **Value**.

c. Click **OK**.

You have just defined a parameter called L1, with a default value, 10nH, that can be overridden with values passed in from the parent circuit on an instance-by-instance basis.

Before proceeding further, right-click each of the three Section subcircuit instances and select **Design Properties**, and then **Design Passed Parameters**, for each in turn. Note in the resulting **Properties** dialogs that L1 is listed as a parameter default, and as a passed parameter, for all three instances. This is so because all three Section instances are *instances of the same subcircuit*, and adding a parameter default to any one of the three therefore populates that parameter default to all three.

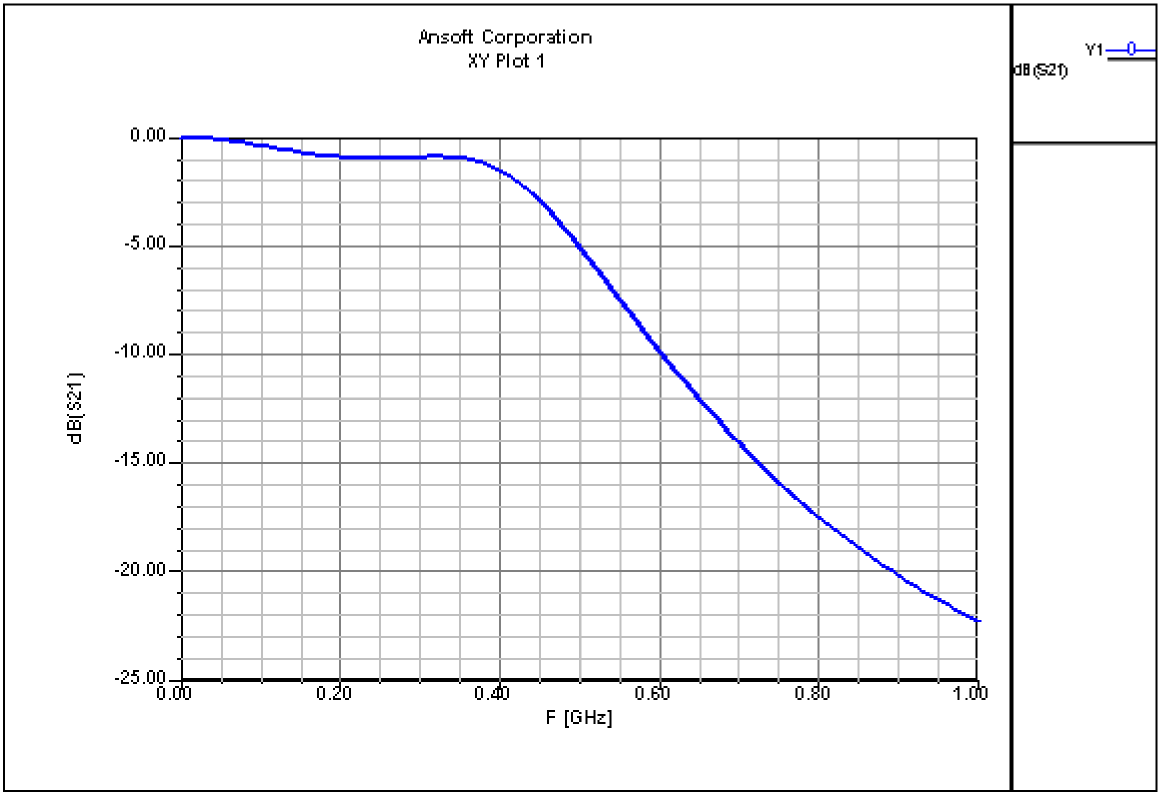
At this point, the Design Passed Parameters value of L1 is shown as the default value of 10nH for all three instances, but we will change that now.

3. Double-click the inductor in the subcircuit and edit its properties. In the **Value** cell for the **L** property, type L1. The value of L1 will now be used as the value of **L** for this inductor.
4. Click in the schematic window for the Top circuit—the parent circuit—to bring that window to the top of the Ansoft Designer desktop.
5. Double-click on the leftmost instance of the subcircuit to open its properties. The **Parameter Values** tab lists the passed parameter L1 with its default value of 10nH.
6. Change L1 to 2nH. Doing so checks the **Override** box, indicating that you have overridden the default value. Click **OK**.
7. Similarly, change L1 to 20nH for the middle Section instance and to 2nH for the rightmost Section instance.

   To view the value of L1 being passed into a subcircuit instance, do any one of the following:

   - Select the subcircuit instance in the parent schematic. The **Properties** window displays the local values of passed parameters in its **Param Values** tab.
   - Double-click the subcircuit in the parent schematic. The resulting **Properties** dialog box displays the local values of passed parameters in its **Parameter Values** tab.
   - Right-click the design icon for the subcircuit in the project window, that is, **U1:Section**, **U2:Section**, and **U3:Section**, and then select **Design Passed Parameters**. The resulting **Properties** dialog box displays the local values of passed parameters in its **Parameter Values** tab.

8. Select the design icon for the Top circuit, right-click, and select **Analyze**. The graph updates with the new results.



Now that you've gained experience with hierarchical designs, you're ready to explore Ansoft Designer's planar electromagnetic simulator.

3

Planar EM Design

The Planar EM simulator is the ideal tool for projects that involve full-wave or radiative effects. For example, a designer might use Planar EM to draw the physical layout, and simulate the electromagnetic properties, of a patch antenna or a millimeter-wave integrated circuit. You can use the Planar EM simulator to compute and display quantities such as:

- Characteristic port impedances and propagation constants.
- Generalized *S* parameters, and *S* parameters renormalized to specific port impedances.
- Basic far-field parameters for electromagnetic fields and antennas.

Example: A Low-Pass Filter

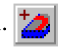
This section will guide you through the physical design and EM analysis of a low-pass filter. As you follow this example step by step, you will:

- Start **Ansoft Designer** and explore the Planar EM tools (all the basics, as well as a few advanced features).
- Use each of the Planar EM toolbars.
- Learn some terms and concepts essential to planar EM simulation. These include selecting a **Layout Technology File**; inserting signal, ground, and substrate layers; drawing a layout; and assigning ports.
- Add a custom-defined dielectric material to the design.
- Use **Create Report** to display simulation results.

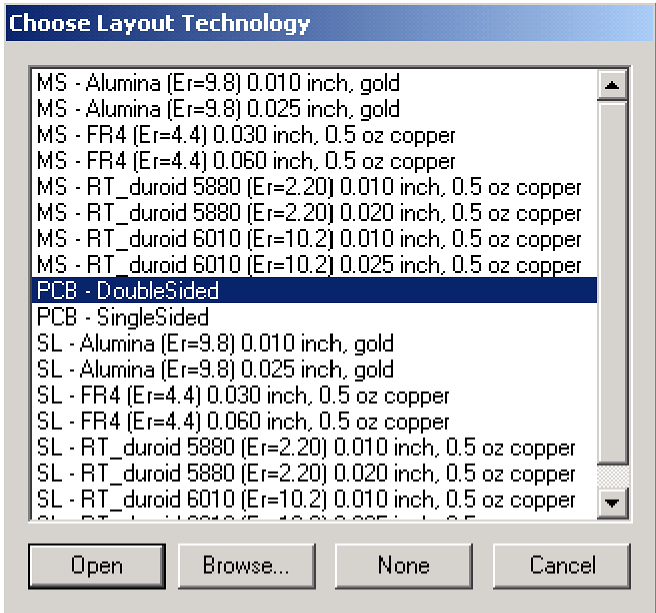
Set up the Planar EM Design

1. On the Windows taskbar, click **Start**, point to **Programs**, point to the **Ansoft Designer** folder icon, and then click **Ansoft Designer**. Alternatively, you can double-click the **Ansoft Designer** desktop icon.
2. On the **Project** menu, click **Insert Planar EM Design**. Alternatively, you can insert a Planar

EM circuit using either of these options:

- On the **Project** toolbar, click the **Insert Planar EM Design** icon. 
- In the **Project Manager** window, right-click the **Project** icon, point to **Insert**, and then click **Insert Planar EM Design**.

3. When the **Choose Layout Technology File** dialog box appears, click **None**.



The **Layout Editor** window appears.

- All Planar EM designs must define a process topology (also called a stackup).
- Layout technology files provides a quick way to store frequently used process topologies. For example, a designer who frequently uses a specific substrate can pre-specify the correct materials and stackups with a single click.
- In this example, we will not use a layout technology file, but will define the topology or stackup manually.

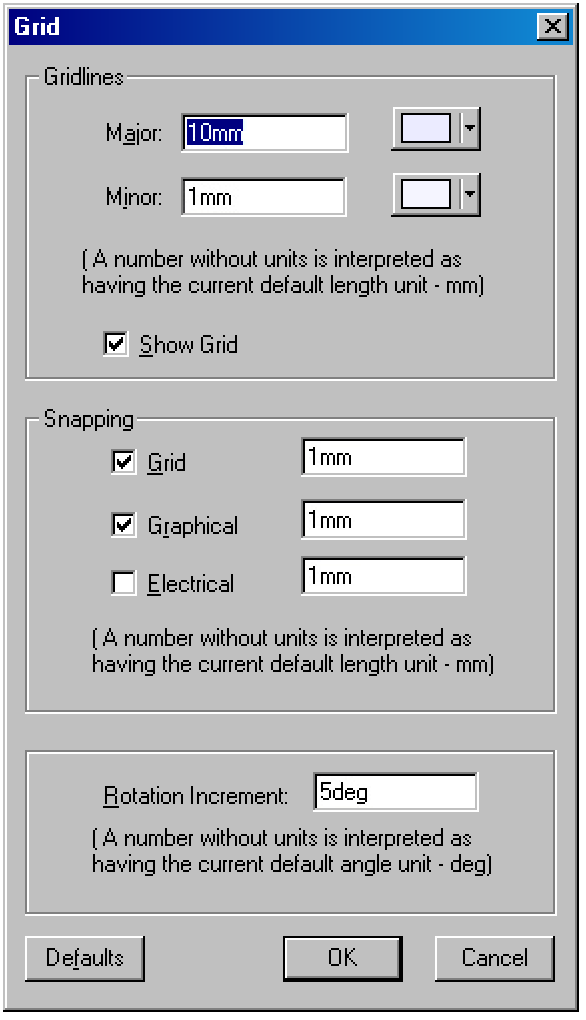
4. Hide the **Message Manage**r and **Progress** windows:
   - On the **View** menu, make sure that the checkboxes for **Message Manager** and **Progress** are cleared.
5. Set the default units:
   a. On the **Tools** menu, point to **Options**, and then click **General Options**.
   b. Select the **Default Units** tab.
   c. In the **Length** list, select **mm**, and then click **OK**, leaving the other settings unchanged.
6. Configure **Grid & Snap** for the **Layout Editor**:

a. On the **Layout** menu, click **Grid & Snap**.

   The **Grid** dialog box opens.

b. In the **Gridlines** group, enter 10 for **Major** and 1 for **Minor**, making sure that the unit used for each is the millimeter (**mm**, the default setting).

c. In the **Snapping** group**,** enter 1 for **Grid** and 1 for **Electrical**, making sure that the unit used is the millimeter (**mm**, the default setting).



Insert Layers

1. Access the **Stackup** tab of the **Edit Layers** dialog box by doing either of the following:
   - On the **Layout** menu, click **Layers** .

     The **Edit Layers** dialog box opens.

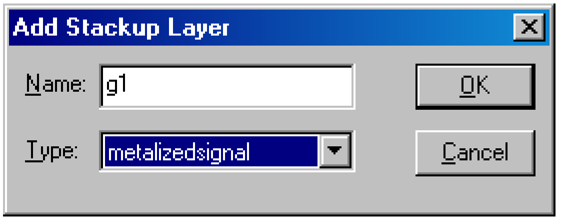
     Click the **Stackup** tab.
   - On the **Layout** toolbar, click the **Stackup dialog** icon.
2. Insert an infinite ground layer:

   a. In the **Stackup** dialog box, click **Add Layer**.

   The **Add Stackup Layer** dialog box appears.

   b. In the **Name** box, type g1.

   c. In the **Type** list, select **metalizedsignal**.

   d. Click **OK**.
3. In the **Stackup** tab of the **Edit Layers** dialog box:

   a. Click **Add Layer**.

   The **Add Layer** dialog box appears.

   b. In the **Name** box, type d1.

   c. In the **Type** list, select **dielectric**.

   d. Click **OK**.
4. In the **Stackup** tab of the **Edit Layers** dialog box, change the thickness of d1 to 1.5 mm.
5. Insert a trace layer by doing the following:

   a. In the **Stackup** dialog box, click **Add Layer**.

   The **Add Layer** dialog box appears.

   b. In the **Name** field, type t1.

   c. In the **Type** list select **signal**.

   d. Click **OK**.

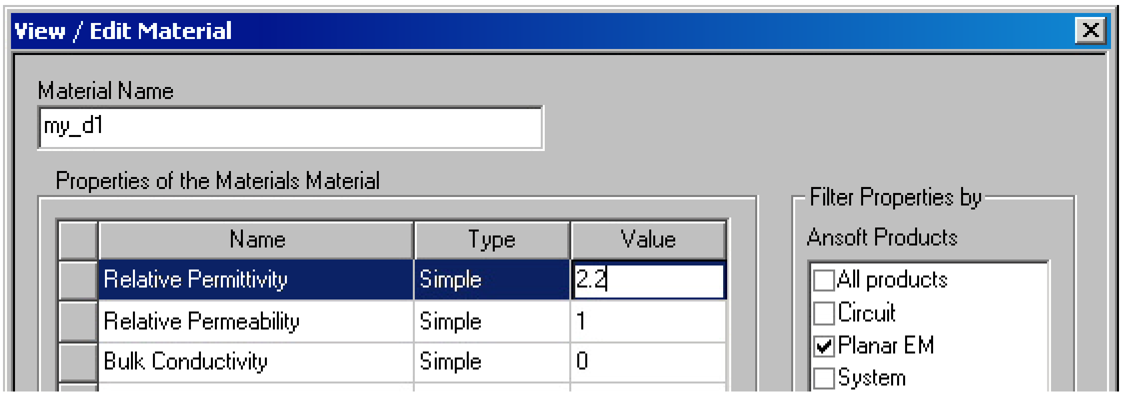
   e. If the stackup created is not in the correct order, you can rearrange it by selecting and dragging the selection handle in the left-most column using the mouse. Select and drag layer t1 above the dielectric layer.

The **Stackup** tab of the **Edit Layers** dialog box should now look like this:

	Name	Type	Material	Drag Mode	Thickness	Lower Elevation	Upper Elevation	Roughness
	t1	signal	copper	middle align	0mm	1.5mm	1.5mm	0mm
	d1	dielectric	Rogers...		1.5mm	0mm	1.5mm	
	g1	metalized...	copper	middle align	0mm	0mm	0mm	0mm

Next, create a new dielectric material and assign it to d1:

1. In the **Stackup** tab of the **Edit Layers** dialog box, locate the d1 row, and click **Rogers**.
   The **Select Definition** dialog box appears.
2. Click **Add Material**.
   The **View/Edit Material** dialog box opens.
3. For **Material Name**, enter my_d1, and as the **Value** for **Relative Permittivity**, enter 2.2.
4. Leave all other settings unchanged.



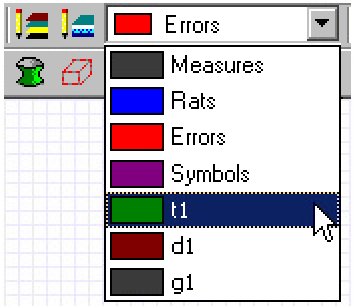
5. Click **OK** in the **View/Edit Material** and **Select Definition** dialog boxes to close each in turn.
   The **Stackup** tab of the **Edit Layers** dialog box should now look like this:

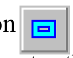
	Name	Type	Material	Drag Mode	Thickness	Lower Elevation	Upper Elevation	Roughness
	t1	signal	copper	middle align	0mm	1.5mm	1.5mm	0mm
	d1	dielectric	my_d1		1.5mm	0mm	1.5mm	
	g1	metalized...	copper	middle align	0mm	0mm	0mm	0mm

6. Click **OK** to close the **Edit Layers** dialog box.

Draw the Model

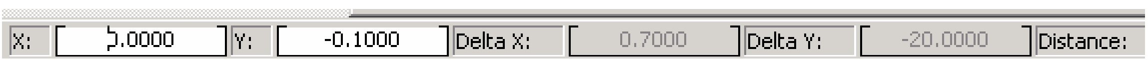
1. Switch the active layer to t1:
   - On the **Layout** toolbar, select t1 from the list of layers:



2. Draw a rectangle:
   a. Do either of the following:
      - On the **Draw** menu, point to **Primitive**, and then click **Rectangle**.
      - On the **Layout Draw** toolbar, click the **Draw Rectangle** icon  .

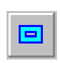
   b. Drag the cursor into the **Layout Editor**, and use the keyboard to enter the data: Start by entering its lower left-hand coordinates. In the **Status Bar** (located at bottom of screen) enter 0.0 for **X** and 0.0 for **Y**. Use the TAB key to move between entries and press ENTER to apply the new settings.

   c. Note that when entering data using the **TAB** key, press **ENTER** as soon as you enter the data in **Delta Y**, and do not tab into the **Distance** box. (Otherwise, the program may capture extraneous mouse movements that can impair your results.)



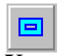
   d. Complete the rectangle by doing either of the following:
      - Enter **Delta X** = 20, **Delta Y** = 4.6, and then press **ENTER**.
      - Drag the upper-right corner until **Delta X** = 20, **Delta Y** = 4.6.

   e. Fit the drawing to the **Layout** window by doing either of the following:
      - Press **CTRL +D**.
      - On the **View** menu, click **Fit Drawing**.

3. Draw another rectangle:
   a. Click the **Draw Rectangle** icon  , and snap the cursor to the lower-right corner of the first rectangle.

   b. Drag the rectangle until **Delta X** = 2.1 and **Delta Y** = –10.5.

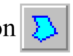
4. Draw a third rectangle (here we show how to enter the coordinates directly):

a. Click the **Draw Rectangle** icon  .

b. In the status bar, enter 21.05 for **X** and 0.0 for **Y**, and then press **ENTER**.

c. Enter 21.7 for **Delta X** and 0.7 for **Delta Y**, and press **ENTER**.

5. Draw a polygon:

a. Click the **Draw Polygon** icon  .

b. In the **Layout** editor, snap the cursor to the upper-right corner of the first rectangle (20.0, 4.6). Note: If you let the mouse hover briefly over the target point, you will see the cursor jump to the snap point.

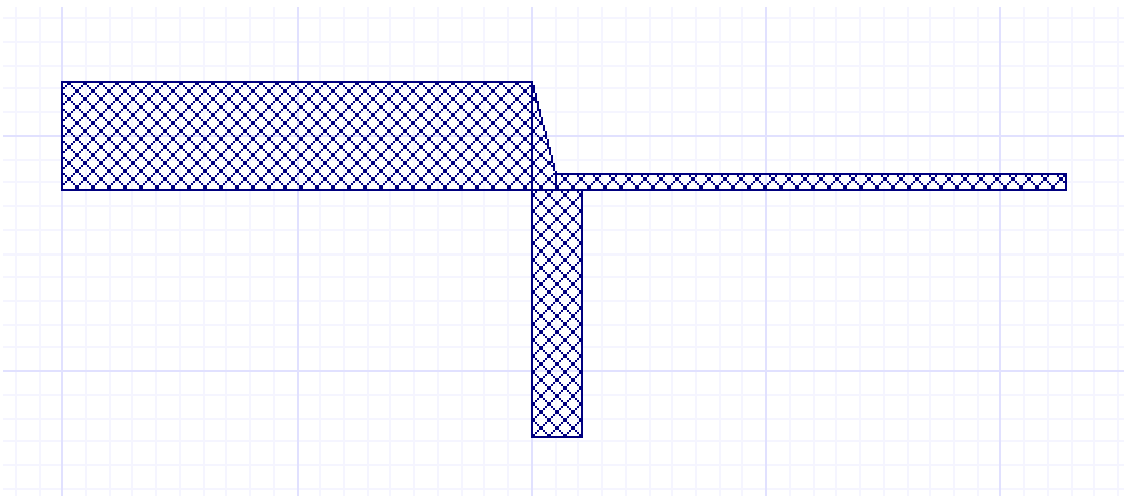
c. Click the upper-left corner of the third rectangle (21.05, 0.7).

d. Click the lower-left corner of the third rectangle (21.05, 0).

e. Double-click the lower-right corner of the first rectangle (20.0, 0).

f. Press **CTRL+D** to fit the drawing in the **Layout** window.

At this point, your layout should look similar to this:



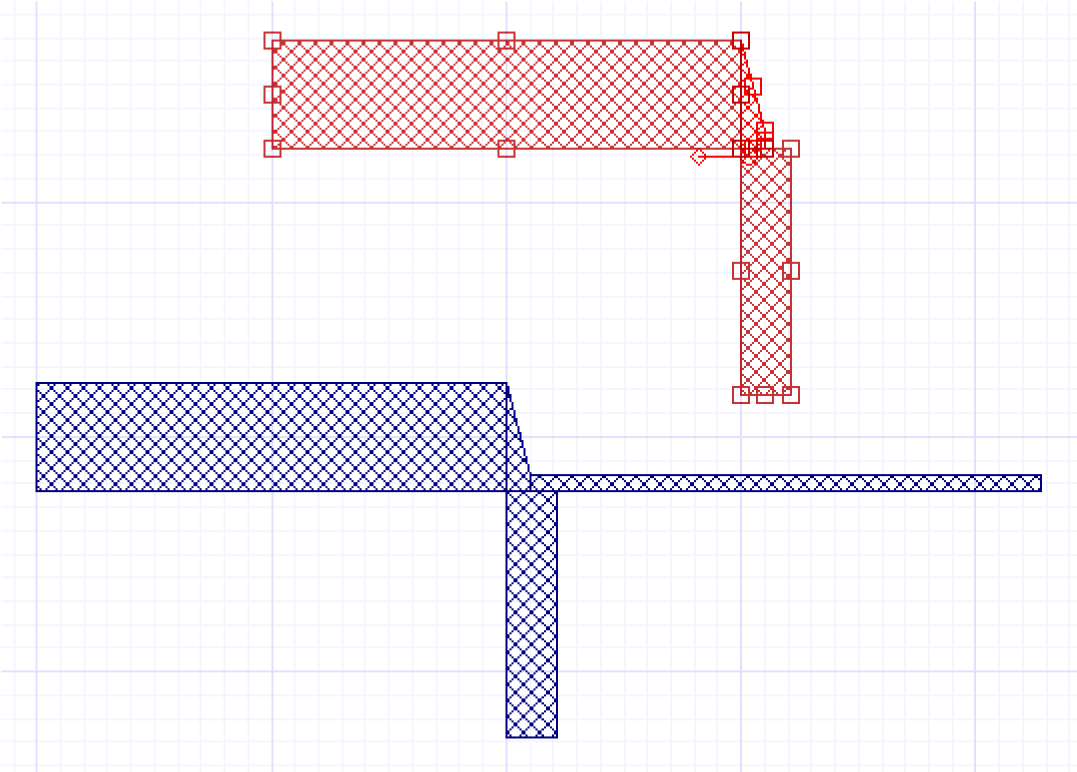
The fill pattern within the objects may be different.

6. Copy and paste the selected elements:

a. Select the polygon, first rectangle, and second rectangle by pressing and holding **CTRL** while clicking the three objects in turn.

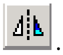
b. Copy the objects by pressing **CTRL+C**. Then click in the **Layout** window and press

CTRL+V to paste the objects into the layout, as shown.



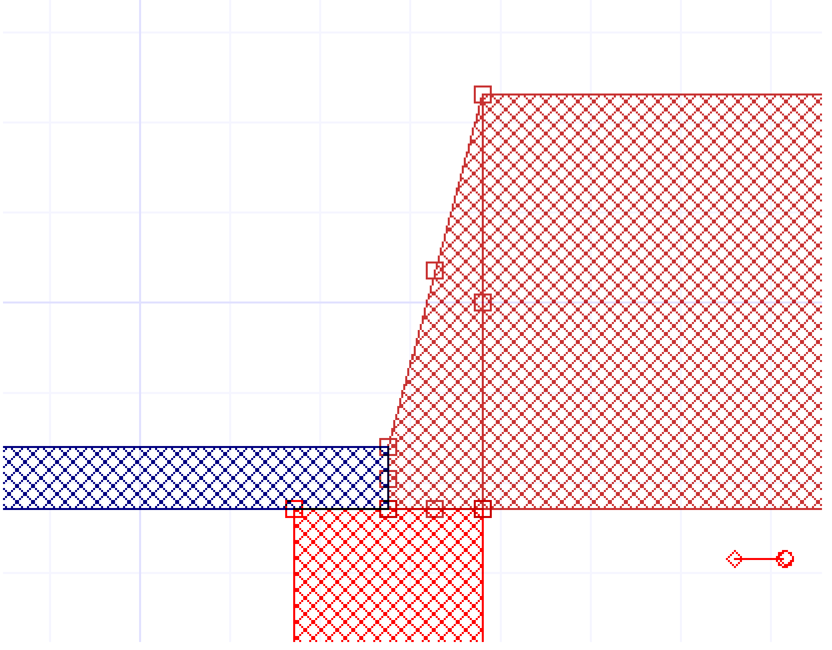
7. Mirror and move the elements that you just pasted:

   a. Make sure that all of the copied elements are still selected.

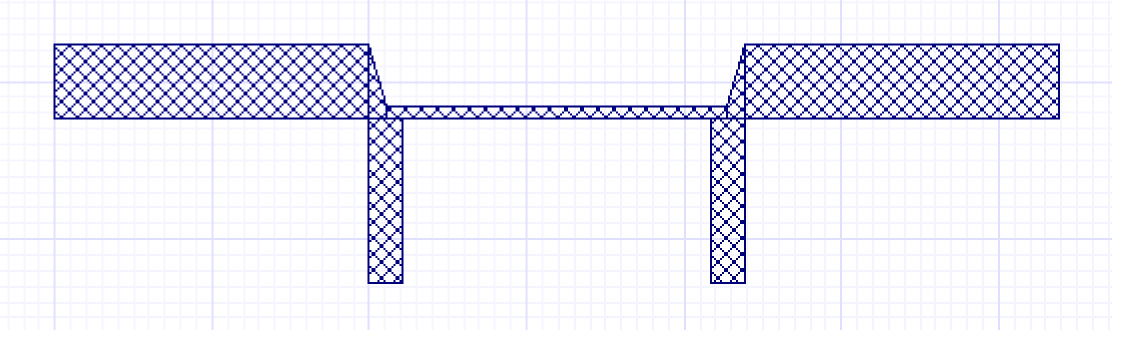
   b. On the **Layout Edit** toolbar, click the **Flip about Y** icon .

c. Move the mirrored objects so that they snap to the lower-right corner of the third rectangle.

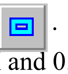
To position the objects exactly, click the **Zoom Area** icon  and drag a box around the portion of the layout you want to magnify.



d. Press **CTRL+D** to view the full geometry. It should look like this:



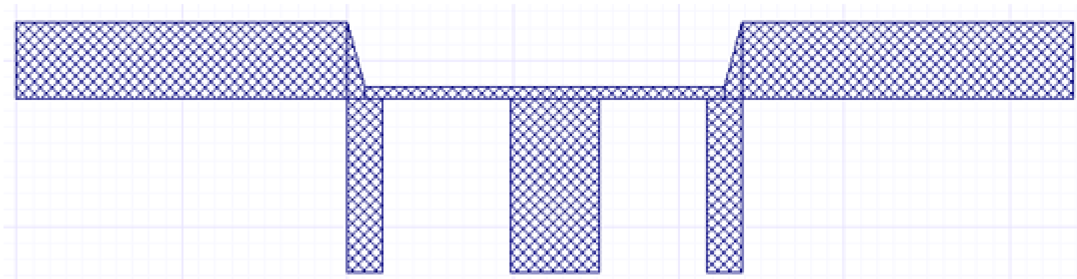
8. Draw the fourth rectangle:

   a. Click the **Draw Rectangle** icon .

   b. In the status bar, enter 29.3 for **X** and 0.0 for **Y**, and then press **ENTER**.

   c. Also in the status bar, enter 5.3 for **Delta X** and–10.5 for **Delta Y**, and then press **ENTER**.

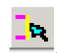
d. At this point, your layout should look like this:

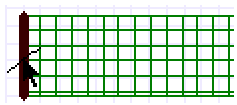


Save the project:

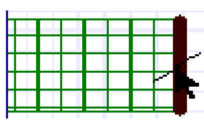
1. On the **File** menu, click **Save**.
2. Type **planarEM1** in the **File name** box, and then click **Save**.

Assign the Ports

1. Create **Port1**:
   a. On the menu bar, click **Edit**, and then click **Select Edges**.
      Alternatively, on the **Layout Draw** toolbar, click the **Select Edges** icon .
   b. Point the cursor to the left edge of the first rectangle and click.



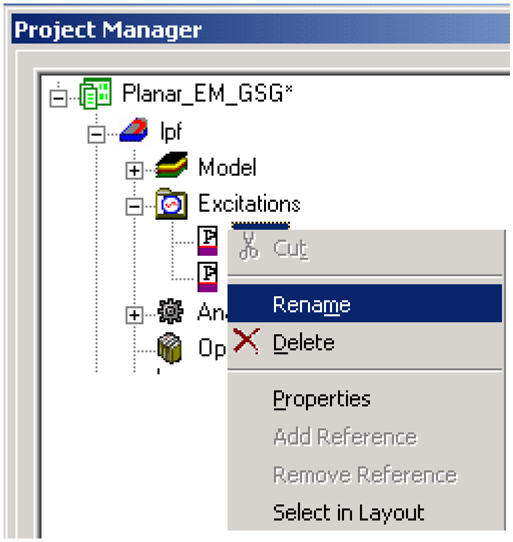
   c. Click **Draw** and then click **Edge Port**. Note that **Port1** appears in the layout editor. Also, in the **Project Manager**, an icon labeled *Port1* appears under **Excitations**.
   d. Alternative way to add an excitation (port): First select the edge and, in the **Project Manager**, right-click **Excitations** and then click **Add Port**.
2. Create **Port 2**:
   a. Click the **Select Edges** icon  , point the cursor to the right edge of the right-most rectangle, and click.



   b. Click the **Draw port** icon (located on the **Layout Draw** toolbar :
3. Rename **Port 1**:
   a. In the **Project Manager**, expand **Excitations**, and then right-click **Port1**.
   b. Click **Rename**, enter p1, and then press **ENTER**.
4. Rename **Port 2**:

a. In the **Project Manager**, expand **Excitations**, and then right-click **Port2**.

b. Click **Rename**, enter p2, and then press **ENTER**.

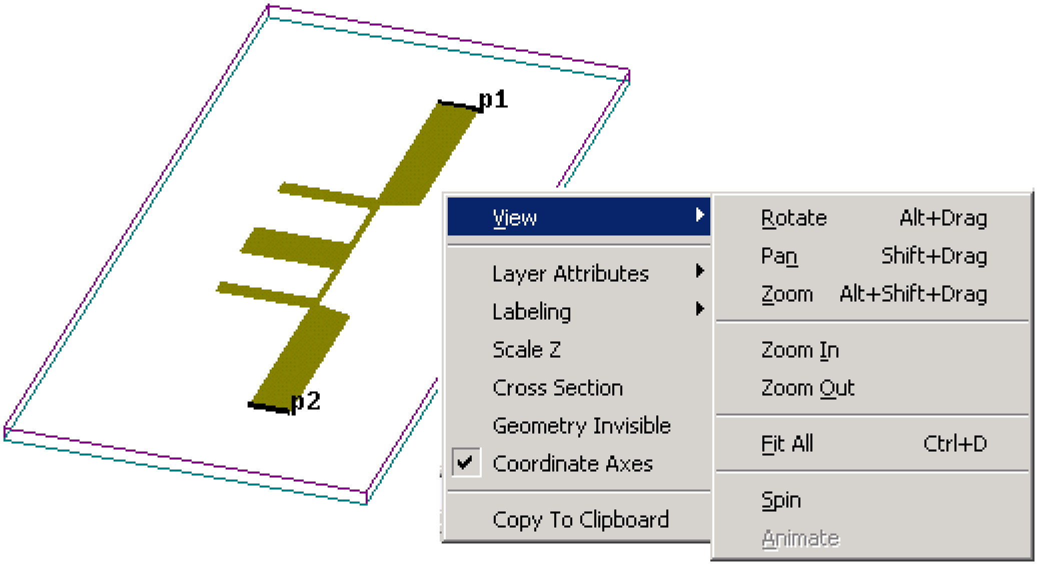


Display the Circuit Using the 3D Viewer

Explore the functionality of the 3D Viewer:

1. In the menu bar, click **Planar EM** and then click **3D Viewer**.
2. Right-click anywhere in the **3D View** window, point to **View**, and then check out the various

options.

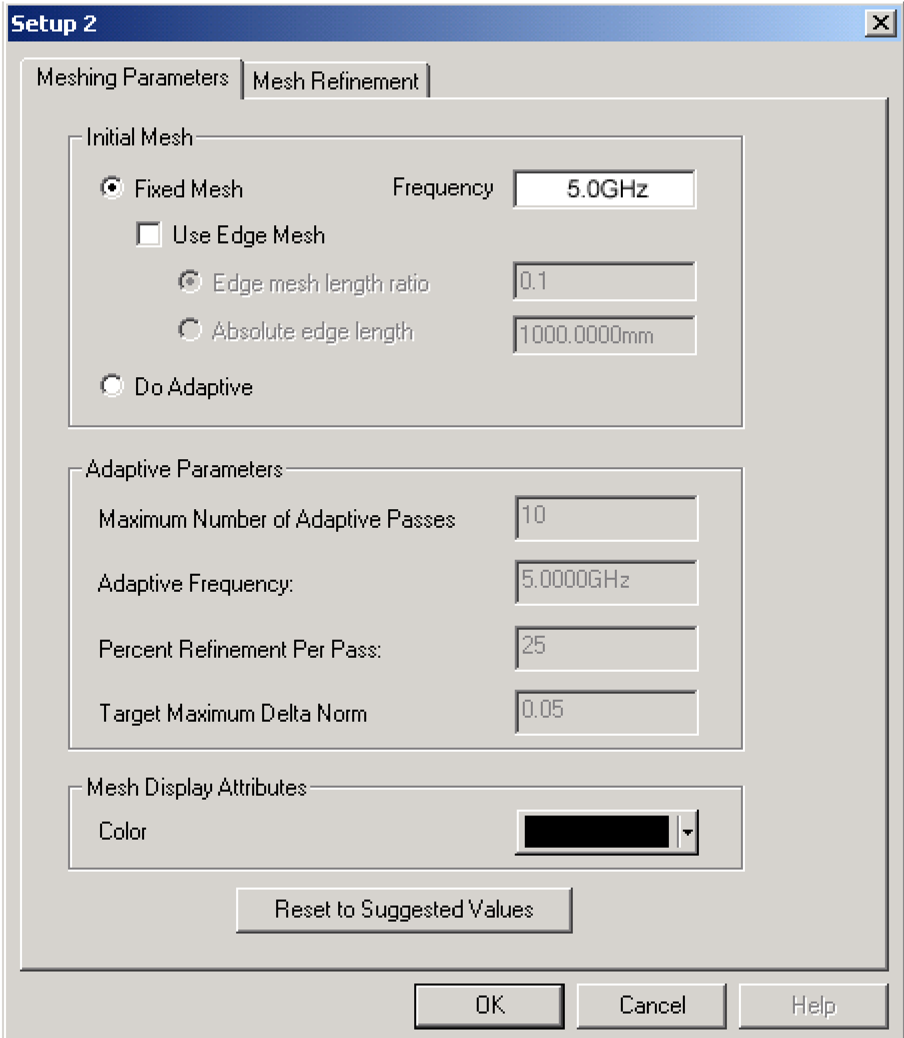


Set up the Planar EM Analysis

Add an analysis-setup:

1. You can add an analysis setup in either of two ways:
   - On the menu bar, click **Planar EM**, point to **Analysis Setup** and then click **Add Solution Setup**.
   - Or, in the **Project Manager**, right-click **Analysis** and then click **Add Solution Setup**.
2. When the **Setup** dialog box appears, select **Fixed Mesh** and enter 5.0 GHz in the **Frequency**

text box. Note: There is no space between “5.0” and “GHz.”



c. Click **OK** to close **Setup**.

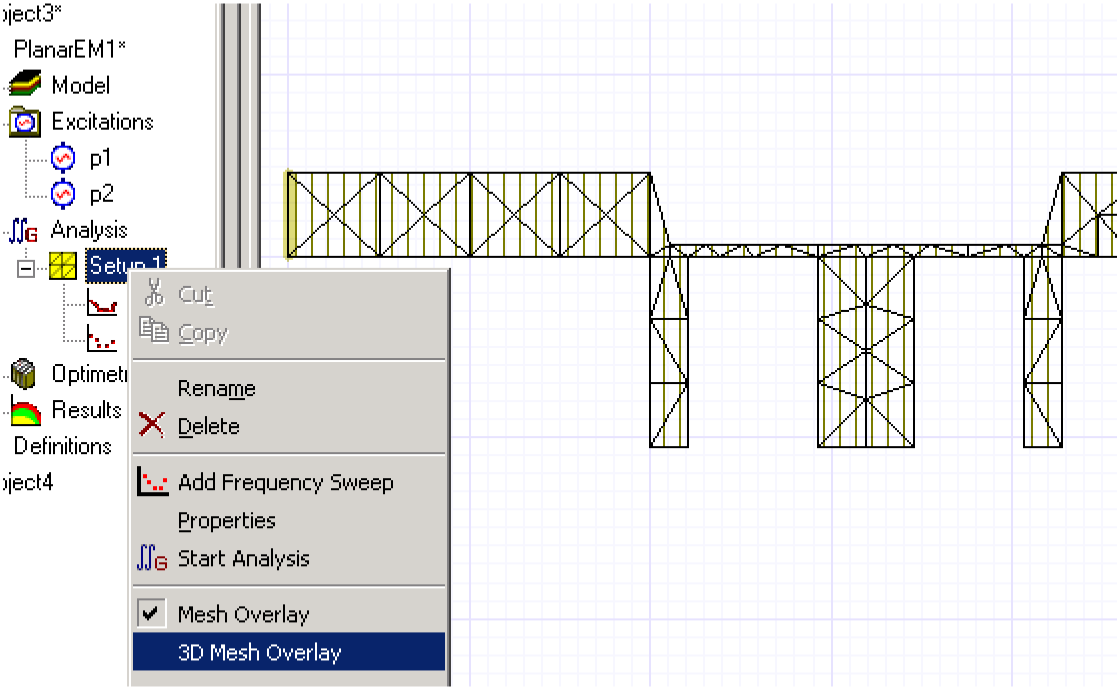
View the Mesh and Set Dynamic Mesh Updates

You can view the layout of the design in a planar or 3D view. On the **Planar EM** menu, click **Layout Editor** to view the planar view of the design.

In the **Layout Editor**, you can view the mesh in either of two ways.

- Click the setup you added. Then, on the **Planar EM** menu, point to **Analysis Setup** and then click **Mesh Overlay**. **Mesh Overlay** is available when a setup has been selected in the **Project Manager**.

- In the **Project Manager**, expand the **Analysis** folder (by clicking the + sign next to its icon), right-click the setup you added, and then click **Mesh Overlay**.



Set up the analysis for **Dynamic Mesh Updates** in either of two ways:

- On the **Planar EM** menu, point to **Analysis Setup**, and then click **Dynamic Mesh Updates**.
- In the **Project Manager**, expand the **Analysis** folder (by clicking the + sign next to its icon), right-click **Setup1**, and then click **Dynamic Mesh Updates**.

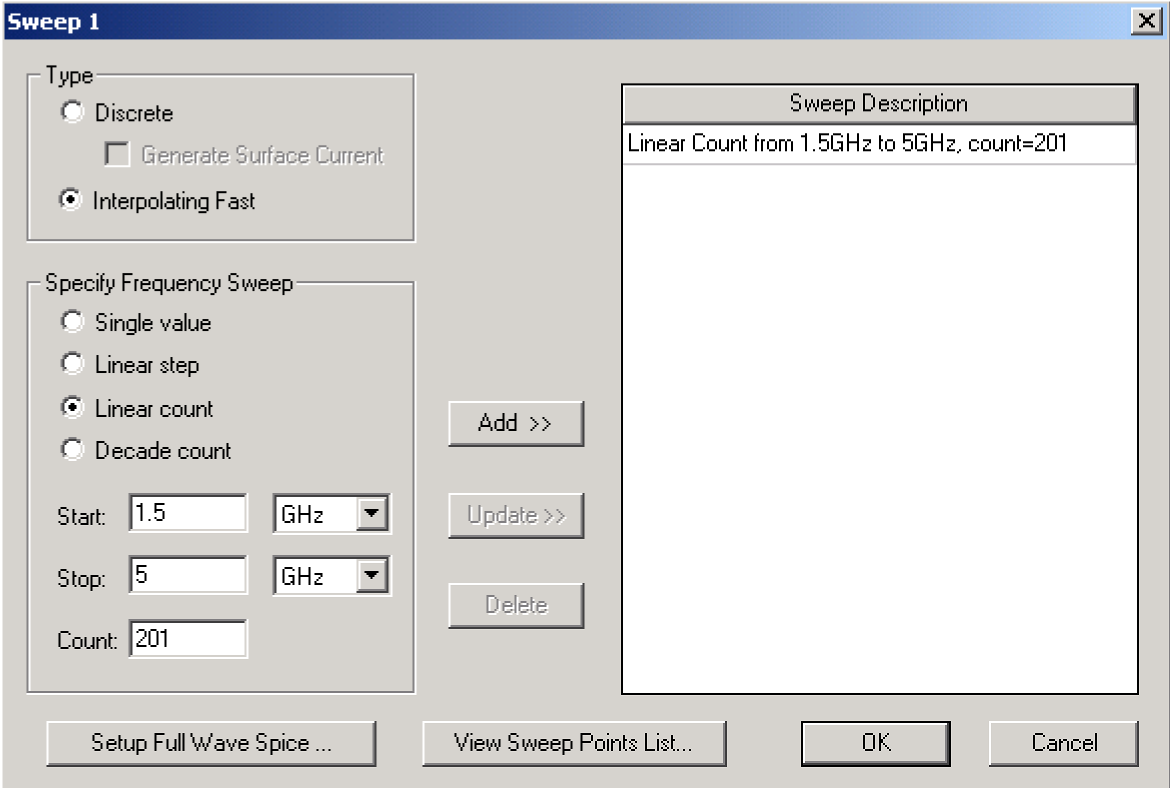
Set up Sweeps

Add an **Interpolating Fast** sweep:

1. Open the **Sweep1** dialog box in either of two ways:
   - On the **Planar EM** menu, point to **Analysis Setup** and then click **Add Frequency Sweep**.
   - In the **Project Manager**, expand the **Analysis** folder, right-click **Setup1**, and then click **Add Frequency Sweep**.
2. When the **Sweep1** dialog box appears, select the default sweep and then click **Delete**.
3. Select **Interpolating Fast**.
4. Select **Linear Count**.
5. Enter the sweep parameters:
   a. In the **Start**, **Stop**, and **Count** boxes enter 1.5 GHz, 5 GHz, and 201.
   b. Click **Add**.

- Alternatively, to update frequencies for an existing sweep, enter the new data in the **Start**, **Stop**, and **Count** boxes, and then click **Update**.

6. Click **View Sweep Points List** to display a tabular list of the frequencies used during the analysis.
7. Click **OK** to close the **Sweep1** dialog box.



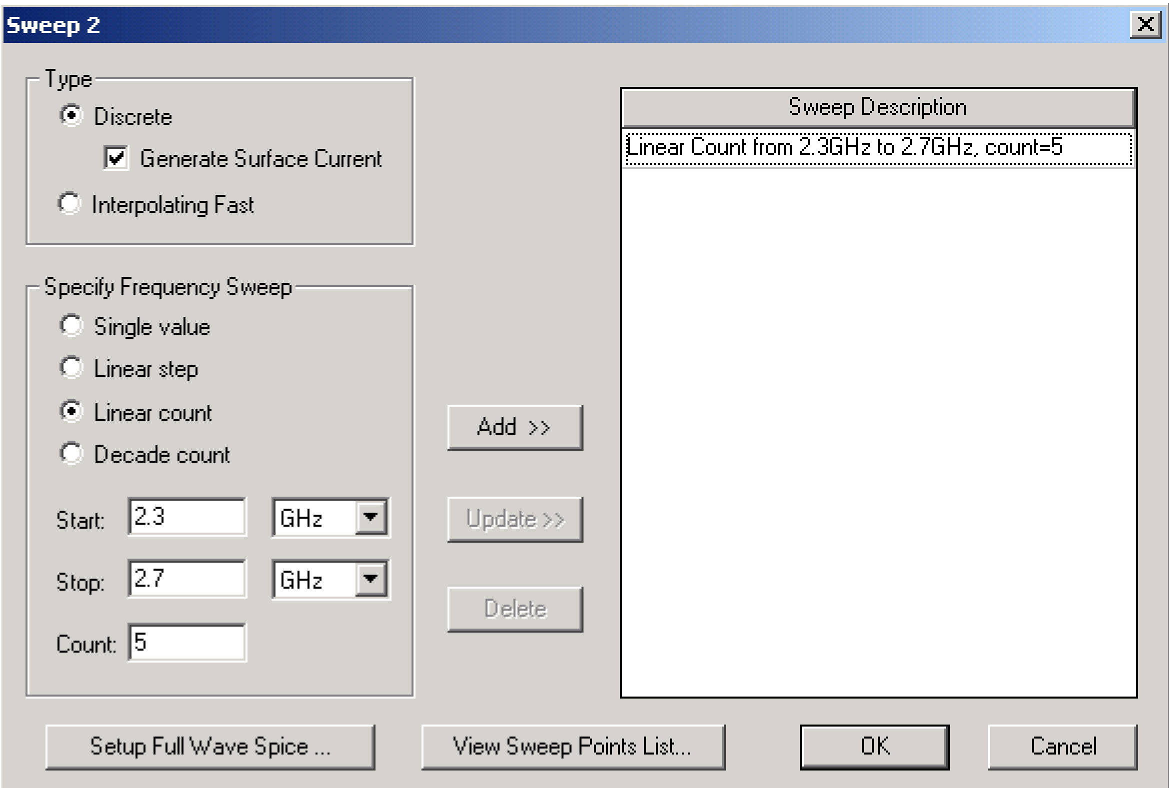
Add a **Discrete** sweep:

1. Open the **Sweep2** dialog box in either of two ways:
   - On the **Planar EM** menu, point to **Analysis Setup,** and then click **Add Frequency Sweep**.
   - In the **Project Manager**, expand the **Analysis** folder, right-click **Setup1**, and then click **Add Frequency Sweep**.
2. When the **Sweep2** dialog box appears, select the default sweep and then click **Delete**.
3. Select **Discrete**, **Generate Surface Current**, and **Linear Count**. Generating surface currents enables you to view currents and field quantities in later steps.
4. Enter the sweep parameters:
   a. In the **Start**, **Stop**, and **Count** boxes enter 2.3 GHz, 2.7 GHz, and 5.

b. Click **Add**.

- Alternatively, to update frequencies for an existing sweep, enter the new data in the **Start**, **Stop**, and **Count** boxes, and then click **Update**.

5. Click **View Sweep Points List** to display a tabular list of the frequencies used during the analysis.
6. Click **OK** to close the **Sweep2** dialog box.

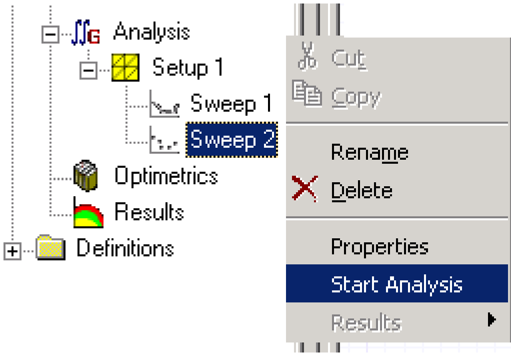


Run the Analyses

Run the analysis:

1. To sequentially run all of the sweep setups that exist in the design, do either of the following:
   - On the **Planar EM** menu, click **Analyze**. (Alternatively, in the **Project Manager**, right-click **Analysis** and then click **Start Analysis**.)
   - Or, you can run a single sweep: In the Project Manager, right-click the appropriate sweep

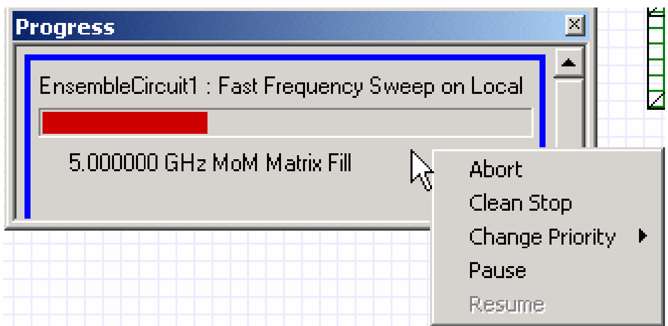
icon and then click **Start Analysis**.



Note that it is also possible to run a single sweep: In the **Project Manager**, right-click the appropriate sweep icon (in this example, either **Sweep1** or **Sweep2**) and then click **Start Analysis**.

2. Check the status while the analysis is running:

   a. While the simulation is running, you can pause or abort the simulation by right-clicking the **Progress** window. (If the window is not visible, click **View** on the menu bar, and then then select **Progress Window**.)



   b. To alter the priority of a particular simulation: Right-click the **Progress Window**, click **Change Priority**, and then select the appropriate setting (**Normal**, **Lowest Priority**, **Highest**, **Above Normal**, **Below Normal**).

   c. The **Clean Stop** selection terminates the simulation when simulation for the current frequency point is completed.

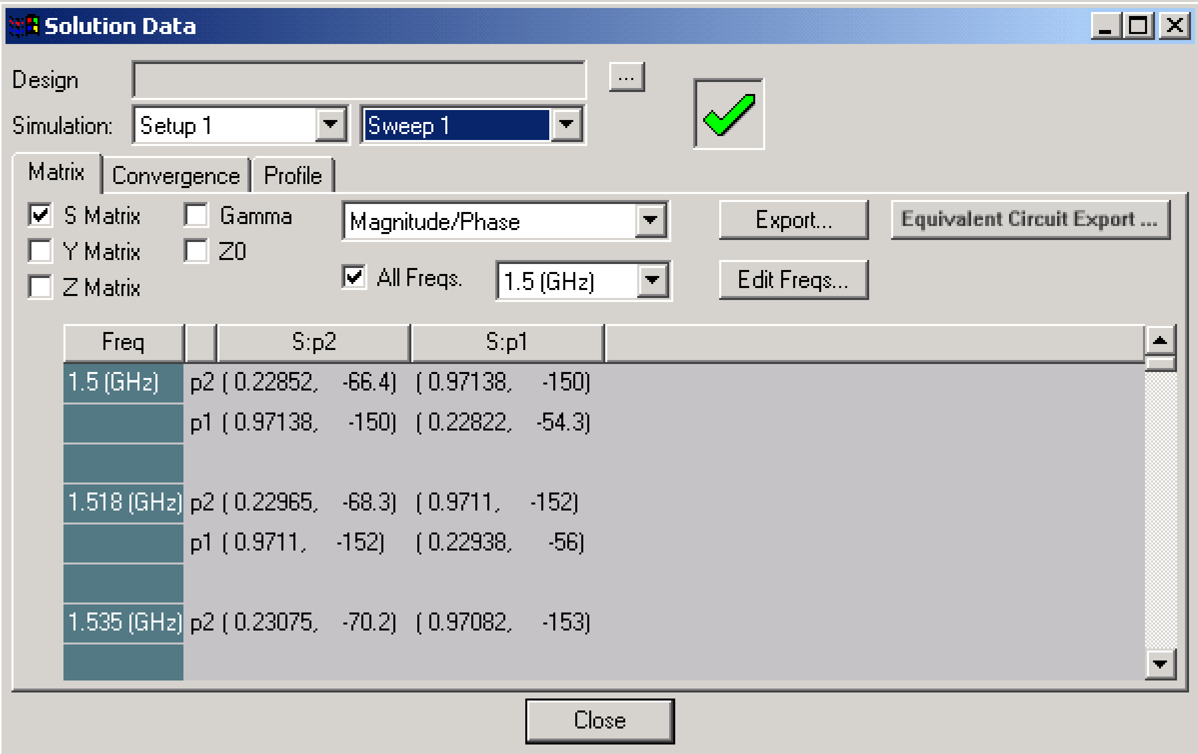
Work with Post Processing

Use the post processing capabilities of Ansoft Designer to display the results of a simulation. You can also use the export features to save the analysis data (and an equivalent circuit) in various industry-standard file formats.

View Tabular *S* Parameters

Open the **Solution Data** dialog box:

1. In the **Project Manager**, right-click **Sweep1**, point to **Results** and then click **Matrix Data**.
   The **Solution Data** dialog box appears. Your results will be similar to the following:



2. Select **S Matrix** and then select the appropriate format (for example, **Magnitude/Phase**).
3. Click **Export**, and the **Export Network Data Solution** dialog box appears. Select the appropriate format (for example, Touchstone, .SZG or .NMF) and then click **Save**.
4. To export the circuit as an equivalent SPICE model, click **Equivalent Circuit Export**, select an appropriate file format and click **Save**. If the **Equivalent Circuit Export** command is unavailable, you need to obtain a license from your Ansoft representative.
5. To see runtime data pertaining to the analysis, click the **Profile** tab.

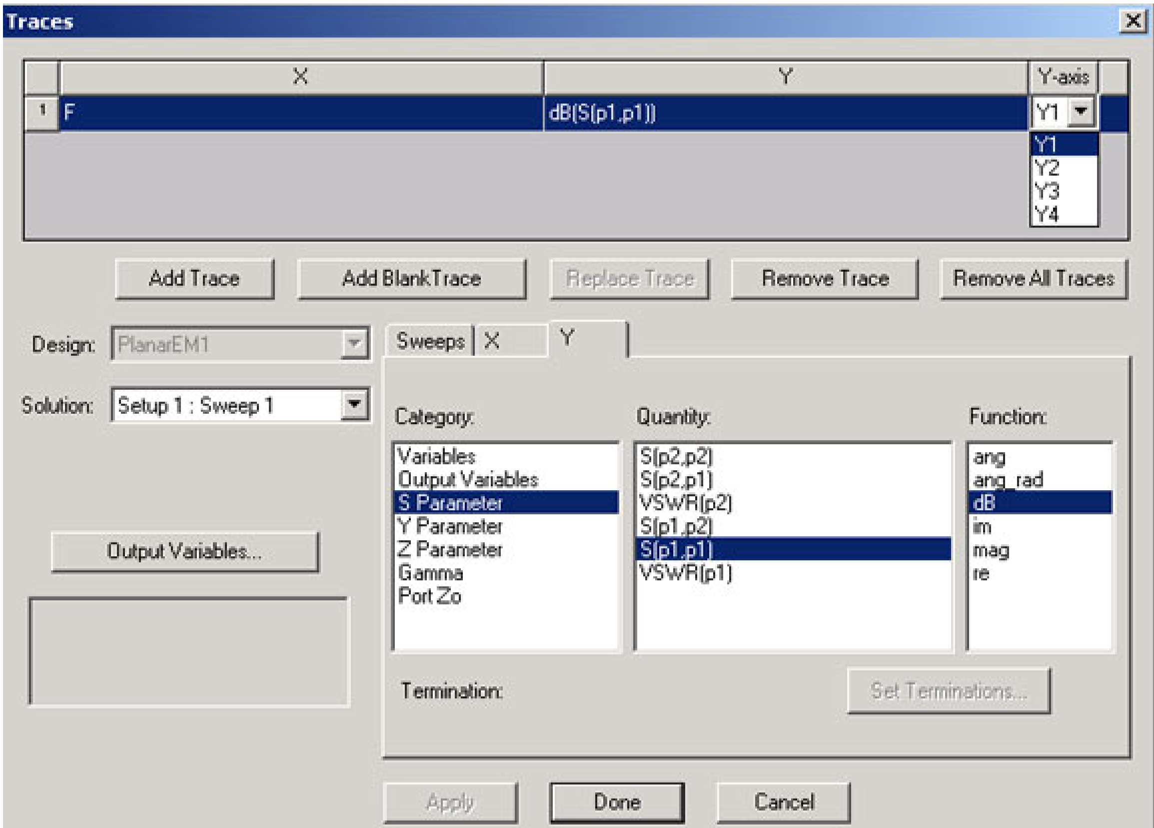
Plot Return Loss (S_{11})

To create a report:

1. First open the **Create Report** dialog box by doing either of the following:
   - On the **Planar EM** menu, point to **Results**, and then click **Create Report**.

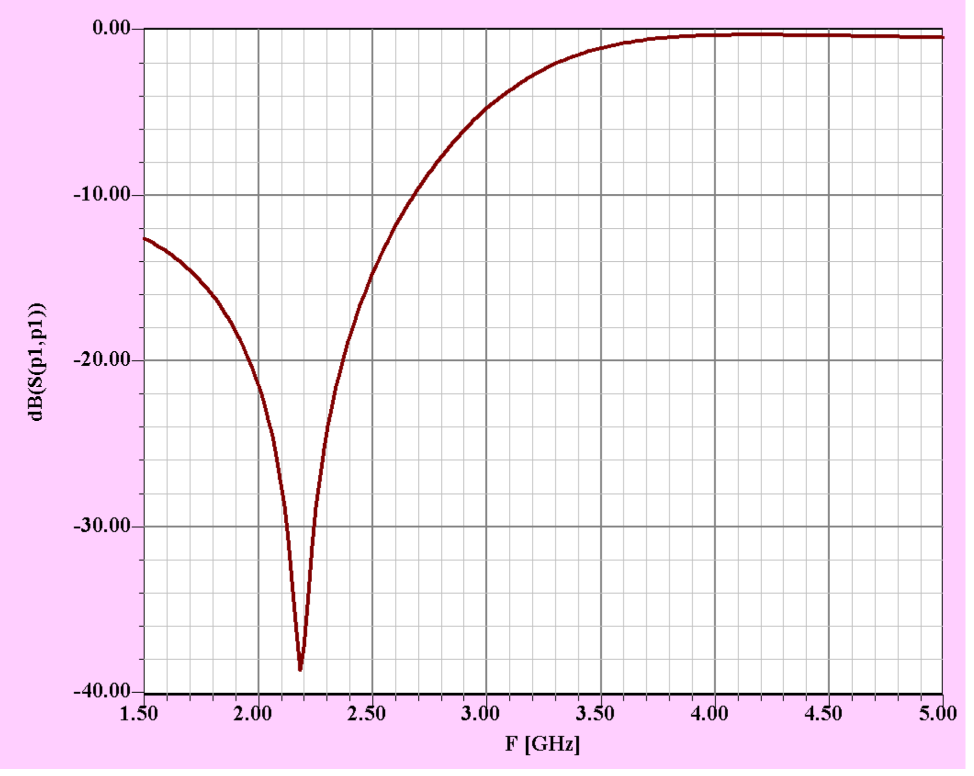
- In the **Project Manager**, right-click **Results**, and then click **Create Report**.

2. Select **Standard** and **Rectangular Plot.**
3. Click **OK**, and the **Traces** dialog box opens:



4. Plot the return loss:
   a. In the **Traces** dialog box, select **Setup 1: Sweep 1** in the **Solution** list.
   b. In the **Category**, **Quantity**, and **Function** lists select **S Parameters**, **S(p1, p1)** and **dB**.

- Click **Add Trace**, then click **Done**, and the plot appears.



- Note that certain results (for example, *S* parameters) have **Plot Templates** that simplify the report-creation process. To use a template, select the appropriate sweep in the project tree, click **Planar EM**, point to **Results**, and then point to **Display Templates** and click the appropriate parameter.
- For additional details about formatting the plot (for example, labeling the axes and changing trace color) see *Post Processing and Generating Reports* in the online help topics.

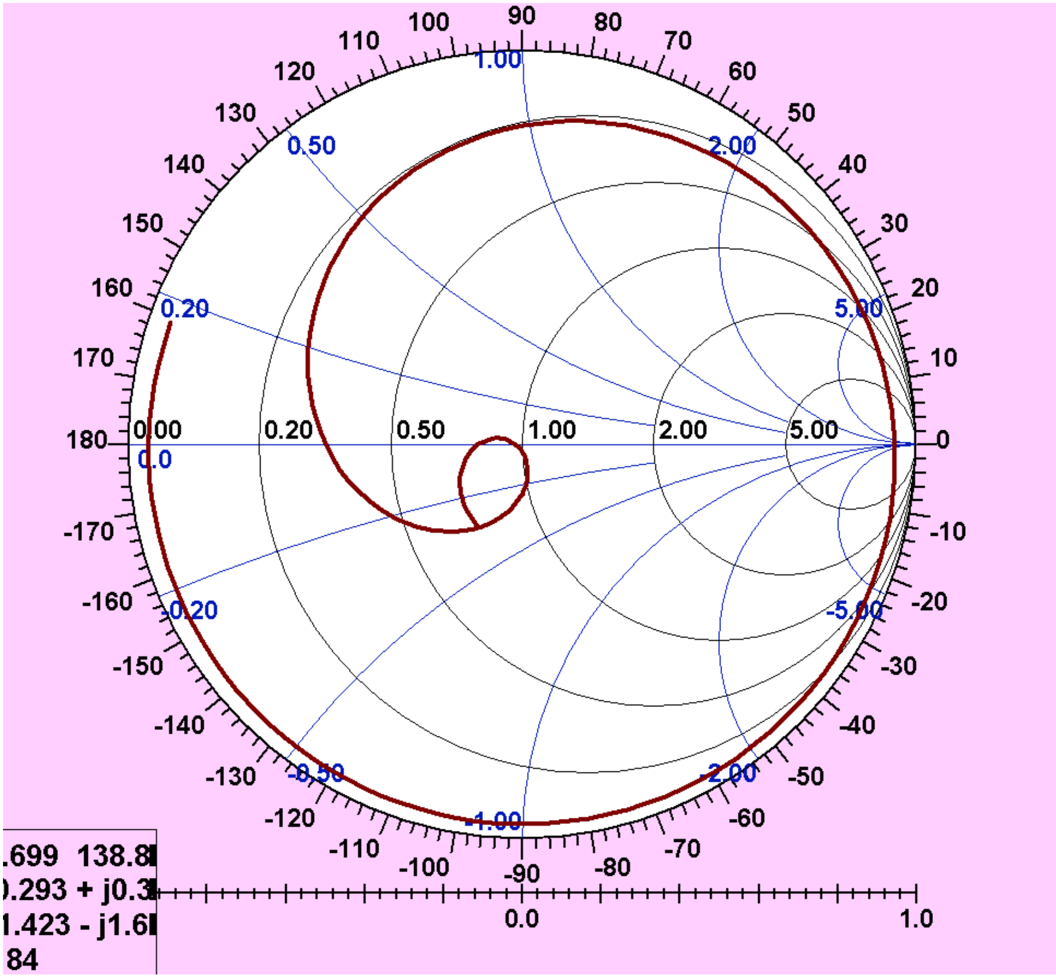
Plot a User Defined Graph

Display a Smith chart for S(p1, p1):

1. On the **Planar EM** menu, point to **Results**, and then click **Create Report**.
   - Alternatively, in the **Project Manager**, right-click **Results**, and then click **Create Report**.
2. When the **Create Report** dialog box appears, select **Standard** and **Smith Chart**.
3. Click **OK**, and the **Traces** dialog box appears.
4. In the **Solution** list, select **Setup1:Sweep1**.

5. In the **Category** and **Quantity** lists, select **S Parameters** and **S(p1, p1)**, and make sure that **None** is selected in the **Function** list.
6. Click **Add Trace**.
7. Click **Done**.

   The Smith chart appears.



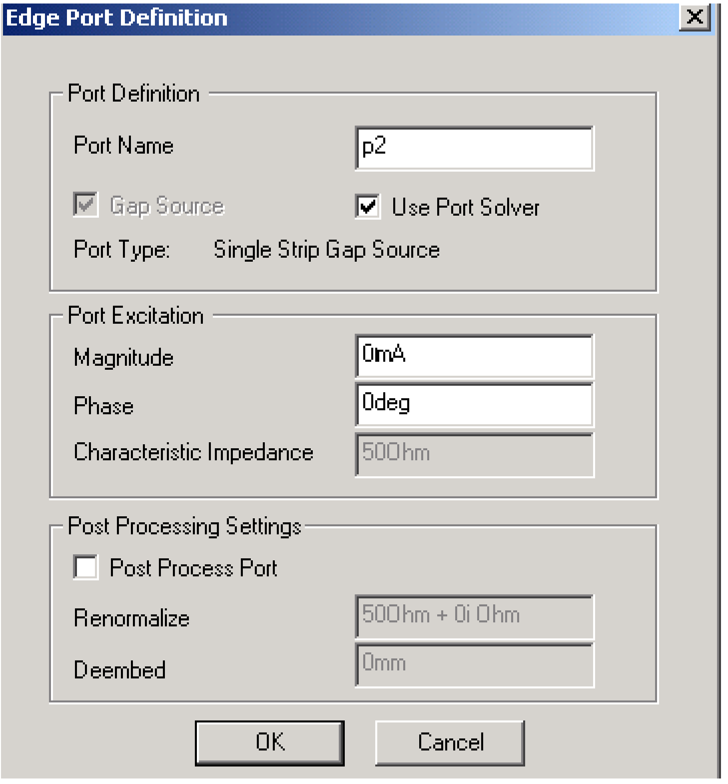
Rescale and Phase Animate the Currents

Setup excitations:

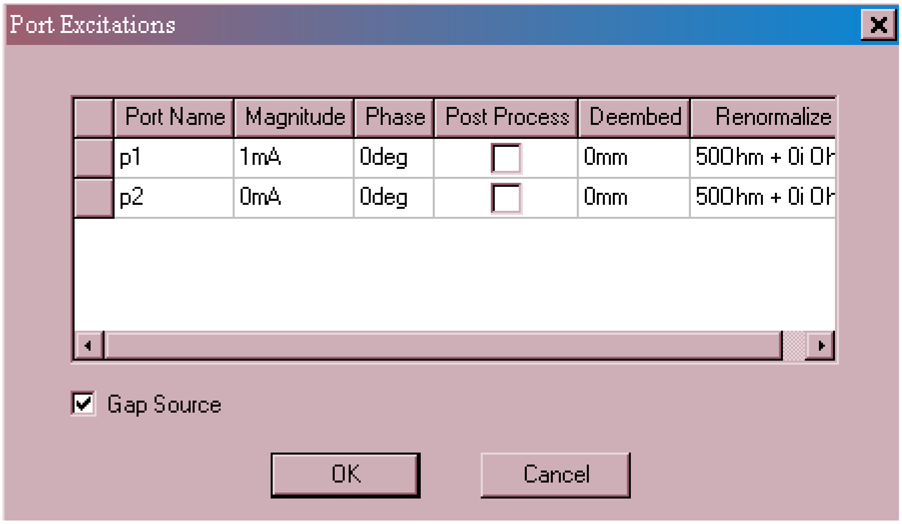
1. In the **Project Manager**, under **Excitations**, right-click **p2**, and then select **Properties**.

   The **Edge Port Definition** dialog box appears.

2. In **Port Excitation Magnitude**, enter 0 ma and 0 degrees.

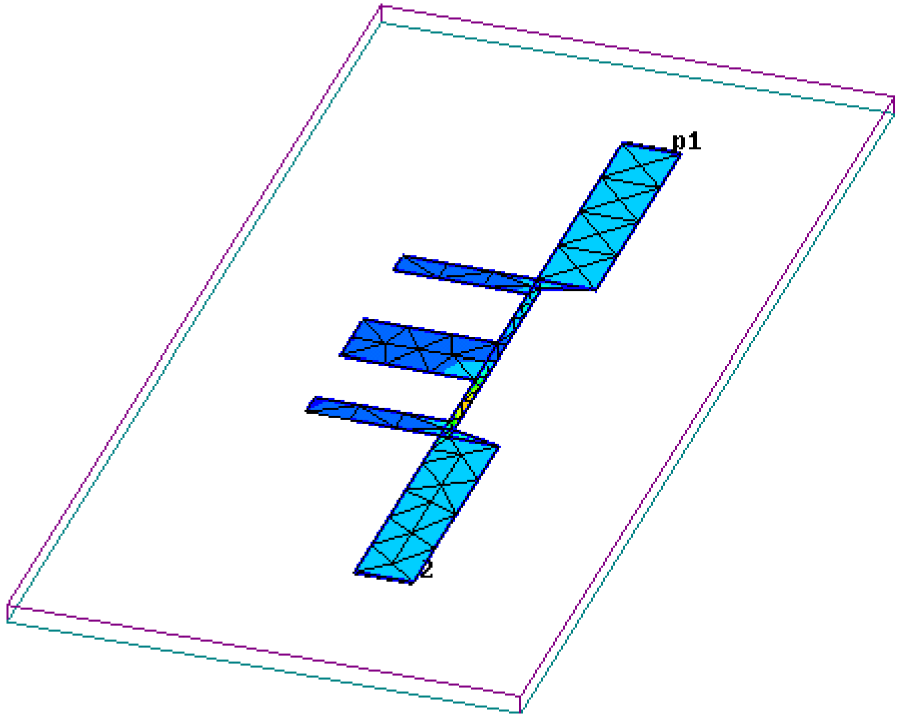


Alternatively, right-click **Excitations** in the project tree, and then click **Port Excitations** on the shortcut menu. The **Port Excitations** dialog box appears, in which you can modify port excitations. This method would be convenient for multiple-port designs.



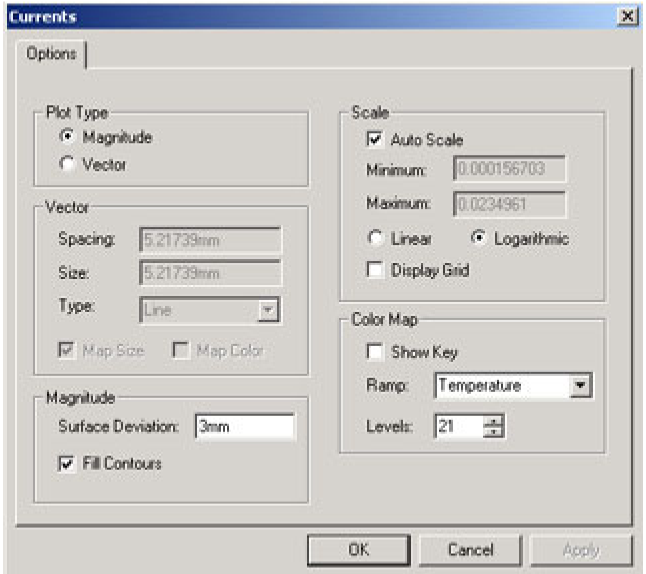
Animate the currents:

1. Display currents: In the **Project Manager**, right-click **Sweep2**, point to **Results** and then click **Display Currents**.

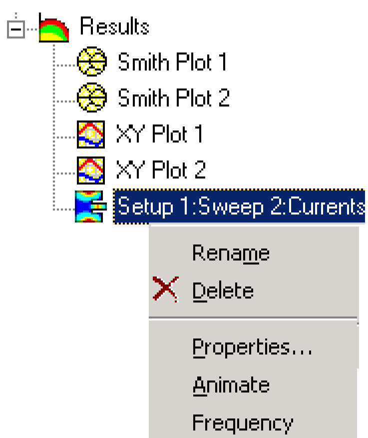


2. In the **Project Manager** window, in the **Results** folder, right-click **Setup 1: Sweep2:Currents**, and then click **Properties**.

The **Currents** dialog box appears:

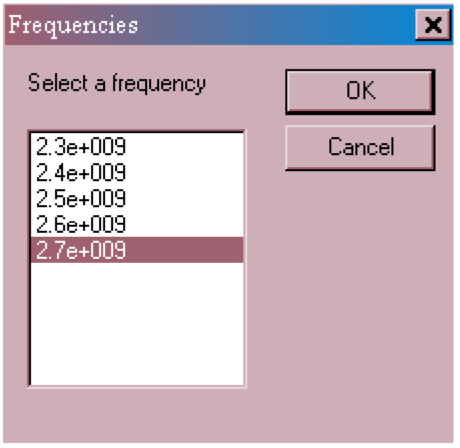


3. Do the following:
   - Under **Scale**, select **Auto Scale** and **Logarithmic**.
   - Under **Color Map**, in the **Ramp** and **Levels** lists, select **21** and **Temperature**.
   - Under **Magnitude**, in the **Surface Deviation** list, enter 3mm.
   - Leave the other settings unchanged, and click **OK**.
4. Phase animate the currents:
   a. Under **Results** in the project tree, right-click **Setup 1: Sweep2 Currents**, and then click



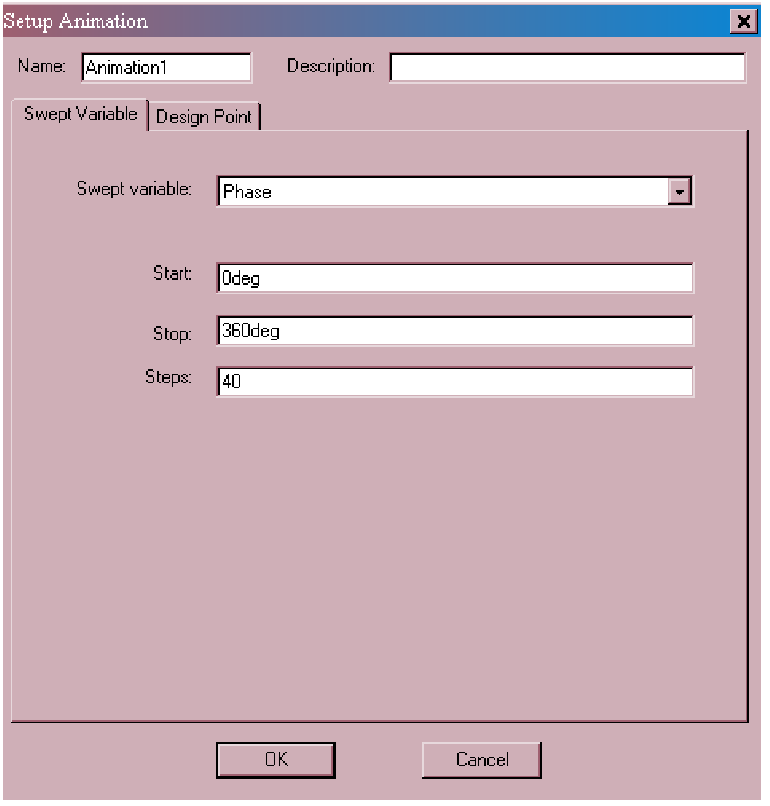
Frequency on the shortcut menu. The **Frequencies** dialog box appears.

b. Select 2.7 GHz from the list.



c. Right-click **Setup1:Sweep2:Currents** and then select **Animate**.

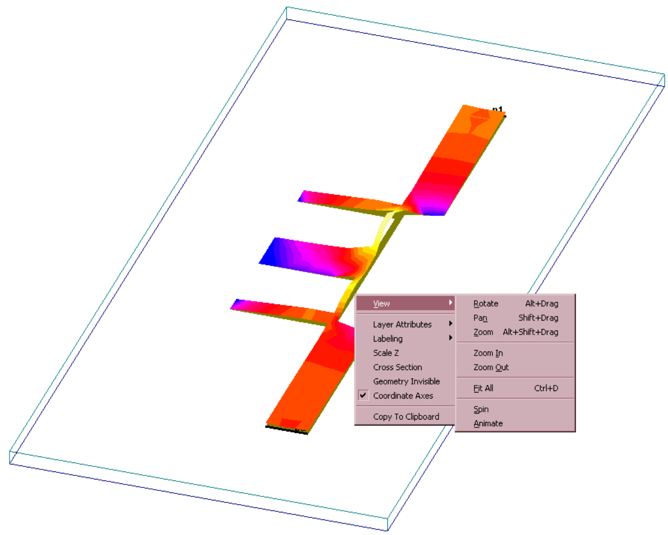
The **Setup Animation** dialog box appears:



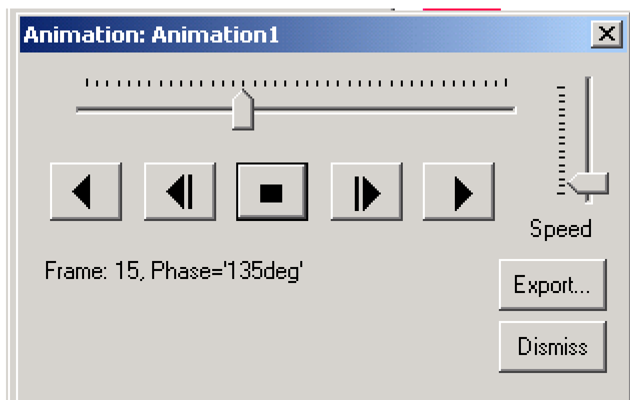
d. Do the following:
- Accept the defaults for **Name** (**Animation1**) and make sure that **Phase** is selected as the **Swept Variable**.
- In the **Start**, **Stop**, and **Steps** boxes, enter 0deg, 360deg, and 40.

e. Click **OK**.

The resulting display looks like this:



5. In the **Layout 3D** window, right-click to **Zoom**, **Rotate**, or **Pan** the animation.
6. Click **Dismiss** when finished.

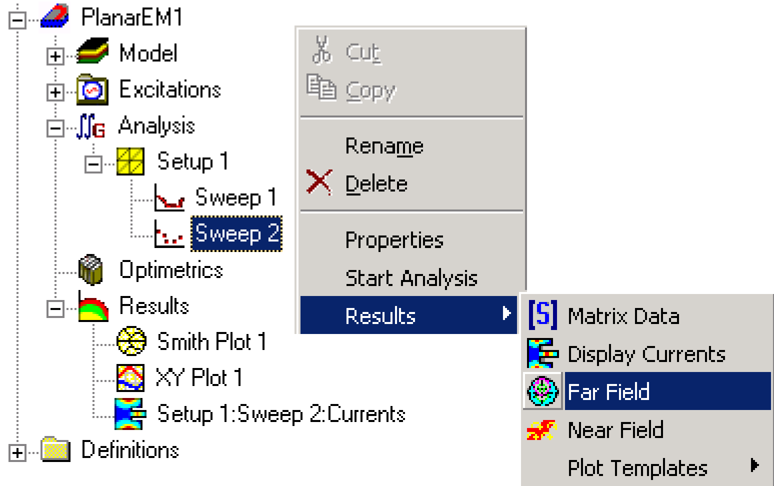


Frequency Animated Far Field Plots

Far fields are not usually of interest when designing a filter, but the procedure is shown here for reference. To be able to plot the currents and far field, a discrete sweep must have been run, with **Generate Surface Current** selected in the solution setup, so that the currents are saved and are available to be plotted. As with the current animation, you can define the frequency at which you view the far field: Right-click **Setup1:Sweep2:Far Field** and click **Frequency**. Click 2.4 GHz.

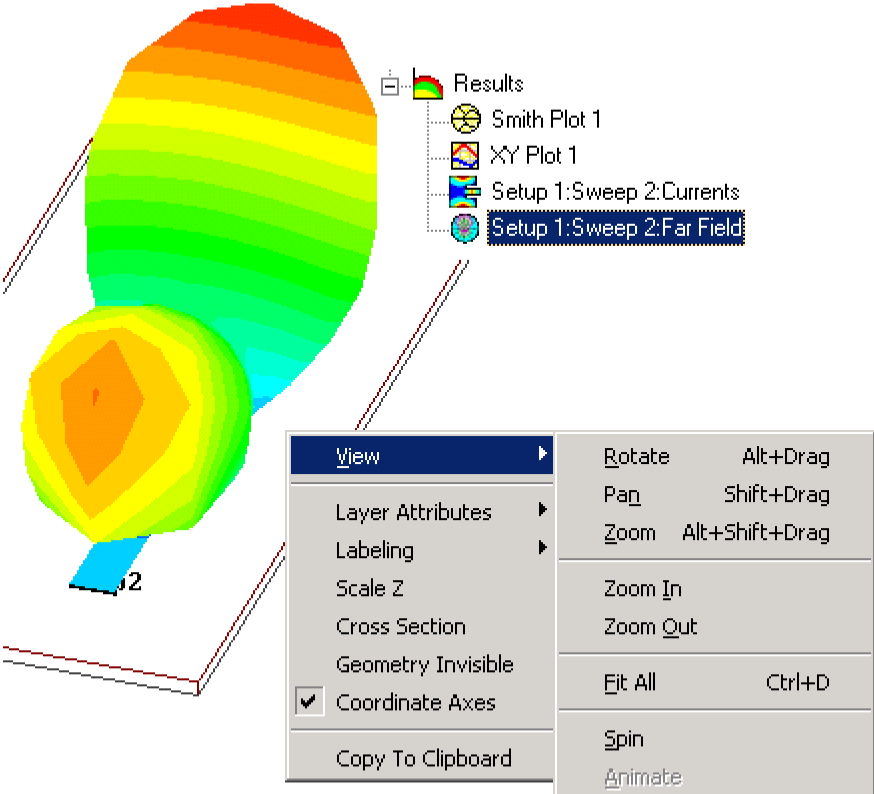
Plot the far field:

1. In the **Project Manager**, right-click **Setup 1: Sweep2**, point to **Results**, and then click **Far Field**:



The **Layout 3D** window appears and the far-field pattern is displayed. Also note that an icon is added in the **Project Manager**, under **Results** 

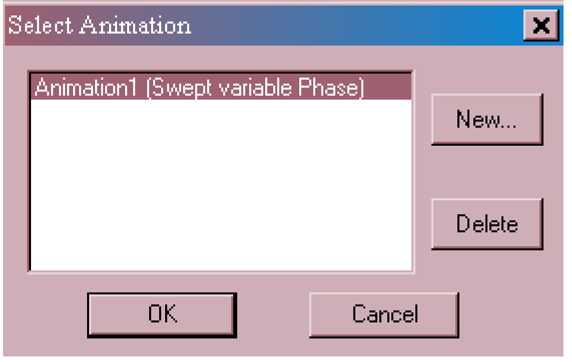
2. Right-click anywhere in the **Layout3D** window, point to **View**, and then check out the various options.



Animate the far field:

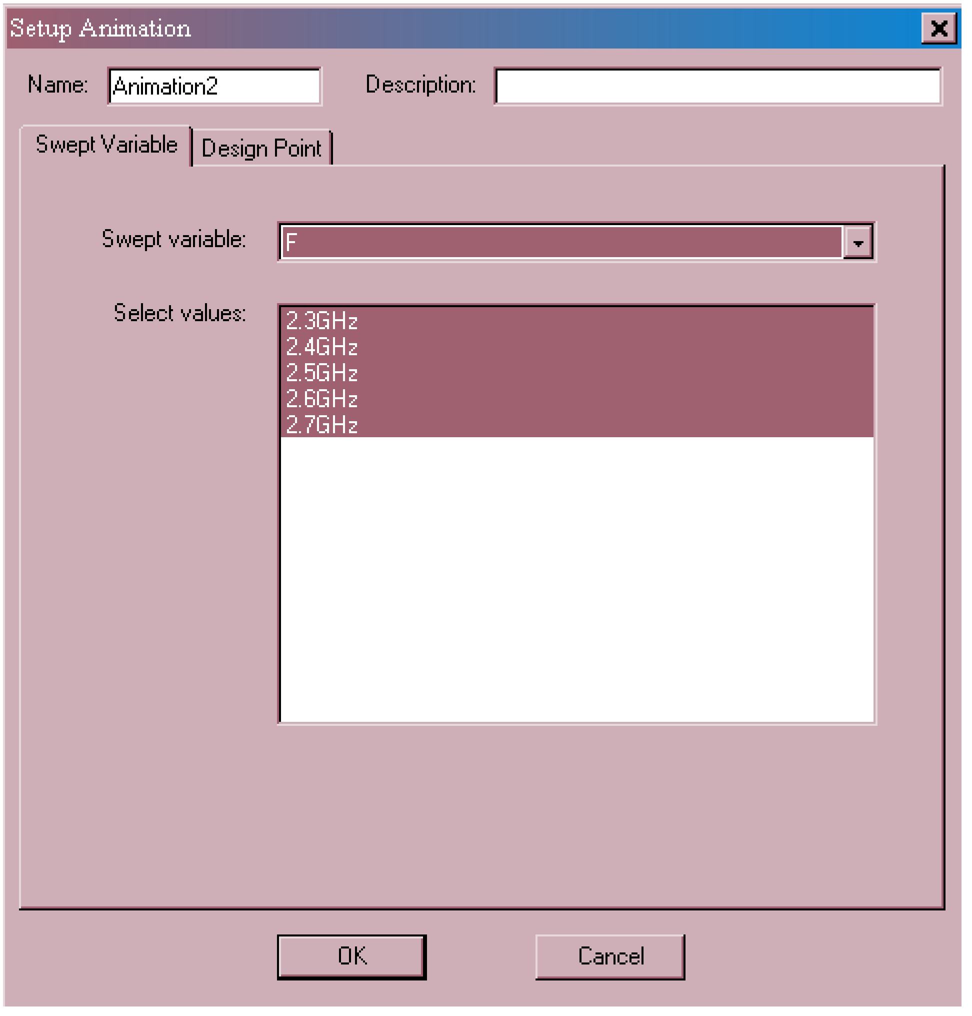
1. In the **Project Manager**, under **Results**, right-click the **Setup1: Sweep2: Far Field** icon, and then select **Animate**.

   The **Select Animation** dialog box appears:



2. Click **New** because you will view an animation of a varying frequency against a fixed input phase of 0 degrees.

The **Setup Animation** window appears:



3. Accept the default name and swept variable.
4. Click **OK** and the animation begins.
5. Click **Dismiss** when finished.

Discrete Time Domain

Discrete time domain analysis yields the output signal response at arbitrary nodes in the system. This type of analysis can be applied to functional or mixed systems and is defined as follows:

- Functional system: contains only functional components
- Mixed system: contains both functional and electrical components

During discrete time domain analysis, all electrical elements and subcircuits are converted to time-domain models (uni-directional). The resulting time-domain model is extracted from the corresponding frequency response that is evaluated during the simulation. To do a system simulation, first create a project, then create a schematic within that project, and then run the simulation.

The example presented in this section is a communications receiver with frequency and discrete time analysis setups. You will analyze and create output reports for each schematic after first running a simulation for each.

Example: A Double Down-Conversion Receiver

This section shows you how to build the schematic, set up an analysis, and then create reports and display results. Along the way you will:

- Start **Ansoft Designer** and explore the system tools (all the basics, as well as a few advanced features).
- Use each of the system tools.
- Learn some terms and concepts essential to system simulation.
- Use **Create Report** to display the simulation results.

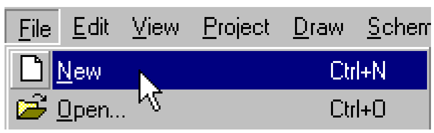
Set up the System Design

To start Ansoft Designer:

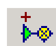
1. Click **Start**.
2. Select **Programs**.
3. Select the **Ansoft Designer** folder.
4. Select **Ansoft Designer**.

5. On the Windows taskbar, click **Start**, point to **Programs**, and then point to the **Ansoft Designer** folder icon and click **Ansoft Designer** (alternatively, you can double-click the **Ansoft Designer** desktop icon).

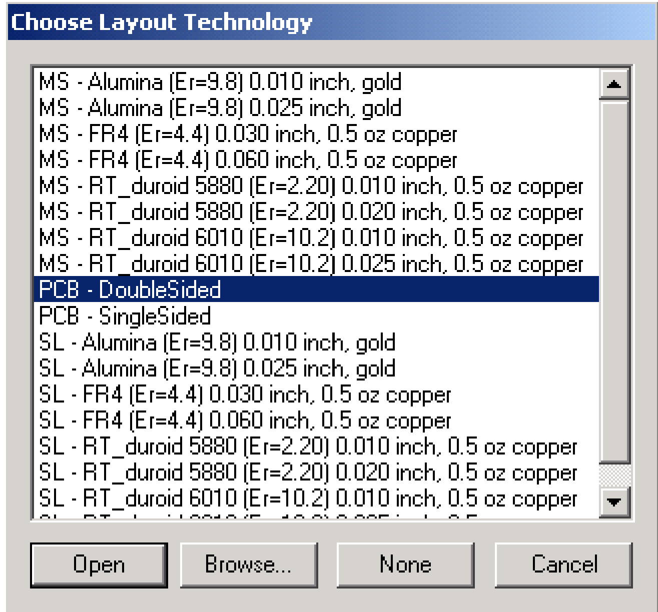
6. Start Ansoft Designer. On the **File** menu, click **New.** 



7. Select the **Project1** icon in the **Project Manager** window.

8. On the **Project** menu, click **Insert System Design**. 

9. In the **Choose Layout Technology** dialog box, select **PCB - DoubleSided** and then click **Open**.



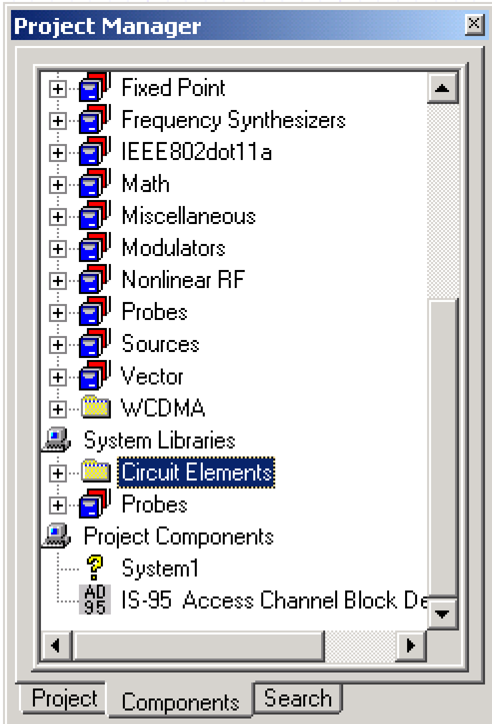
A new, empty schematic, associated with PCB DoubleSided technology, opens in the Ansoft Designer schematic editor.

If you want to open a new schematic that is not associated with the set of definitions that come with a technology file, you can click **None** rather than selecting a technology and clicking **OK**. This is useful for basic concept designs that do not or need not contain manufacturing or substrate information.

Configure the Design Libraries

1. In the **Project Manager** window, click the **Components** tab and then determine if the **Circuit**

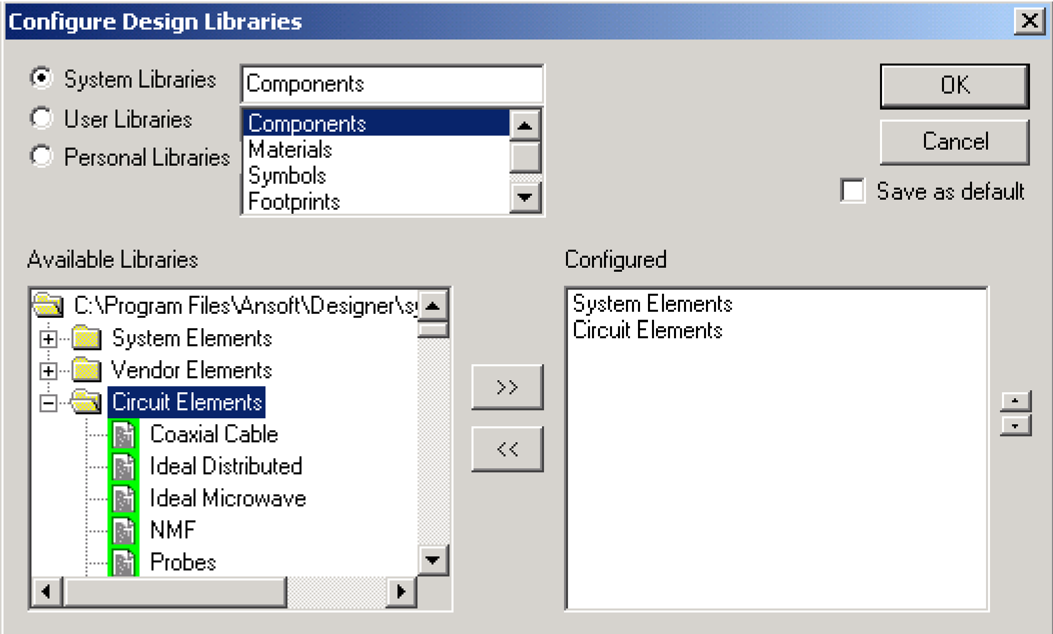
Elements folder is installed (if already installed, it appears under **System Libraries** icon):



a. If the **Circuit Elements** folder is already installed in the **Project Manager**, then go to the next step.

b. If the **Circuit Elements** folder is not installed, you must configure the **Circuit Elements**: On the menu bar, click **Tools** and then click **Configure Libraries**. When the **Configure Design Libraries** dialog box appears, make sure that **System Libraries** is selected (default setting). In the **Available Libraries** list, scroll down the list and select **Circuit Elements**. Click the **Insert** button >>, and then click **OK**. If **Vendor Elements** is not listed in the **Configured** area, you should move this across at the same time, allowing access to manufacturers' component data.

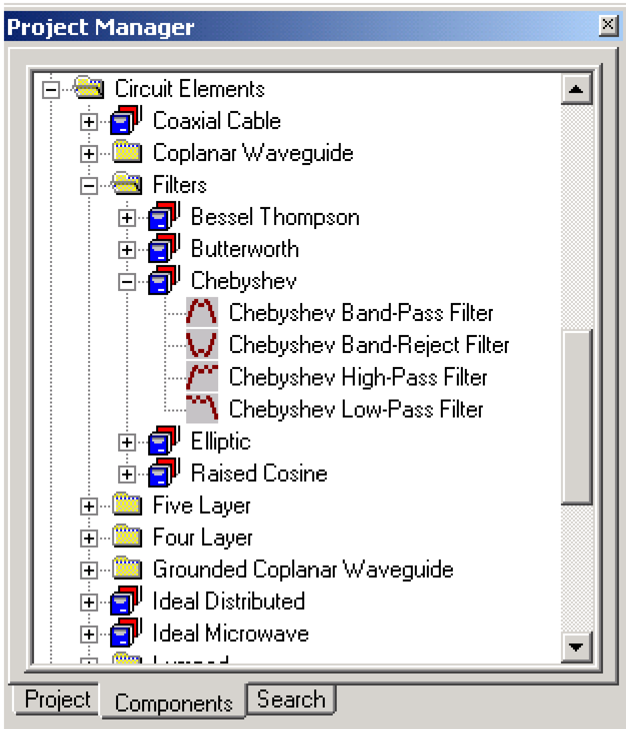
c. In the **Components** tab of the **Project Manager**, confirm that the **Circuit Elements** folder has been configured and is available for use in the design.


Configure Design Libraries
System Libraries
User Libraries
Personal Libraries
Components
Components
Materials
Symbols
Footprints
OK
Cancel
Save as default
Available Libraries
C:\Program Files\Ansoft\Designer\s
System Elements
Vendor Elements
Circuit Elements
Coaxial Cable
Ideal Distributed
Ideal Microwave
NMF
Probes
>>
<<
Configured
System Elements
Circuit Elements

Place the Components in the Schematic

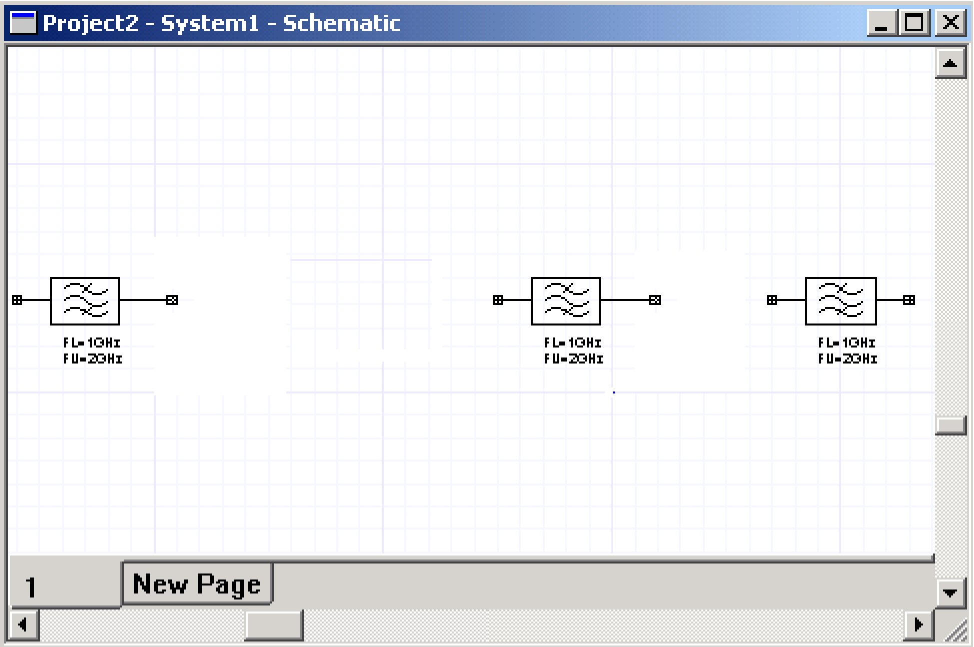
We start by placing the components in the schematic.

1. Click the **Components** tab in the **Project Manager**, and then expand the **Circuit Elements/ Filters** folder and the **Chebyshev** icon.



2. Select **Chebyshev Band-Pass Filter**, and then right-click **Place Component**.
3. Now drag the cursor into the **Schematic Editor** window, and note that the cursor changes into the filter's schematic symbol.
4. Move the symbol near the left side of the screen, then right-click and select **Place and Finish**.

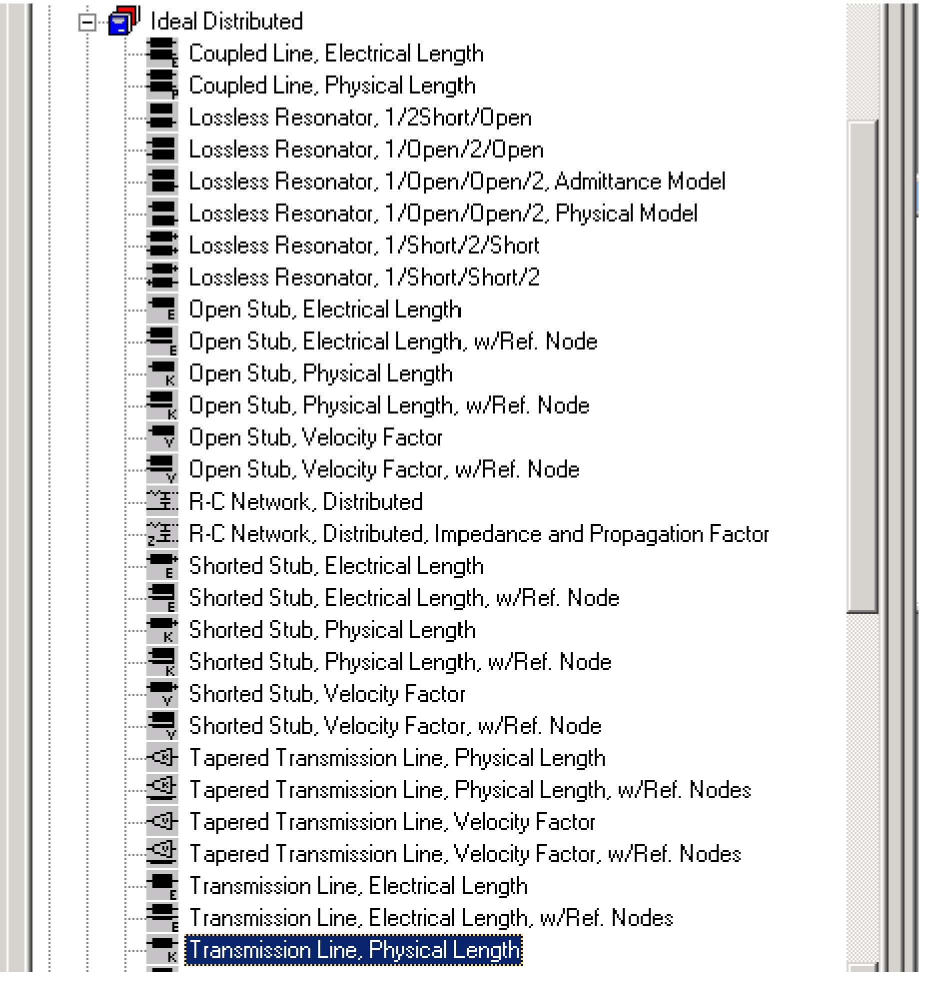
5. Place two additional Chebyshev Band-Pass filters so that your schematic looks like this:



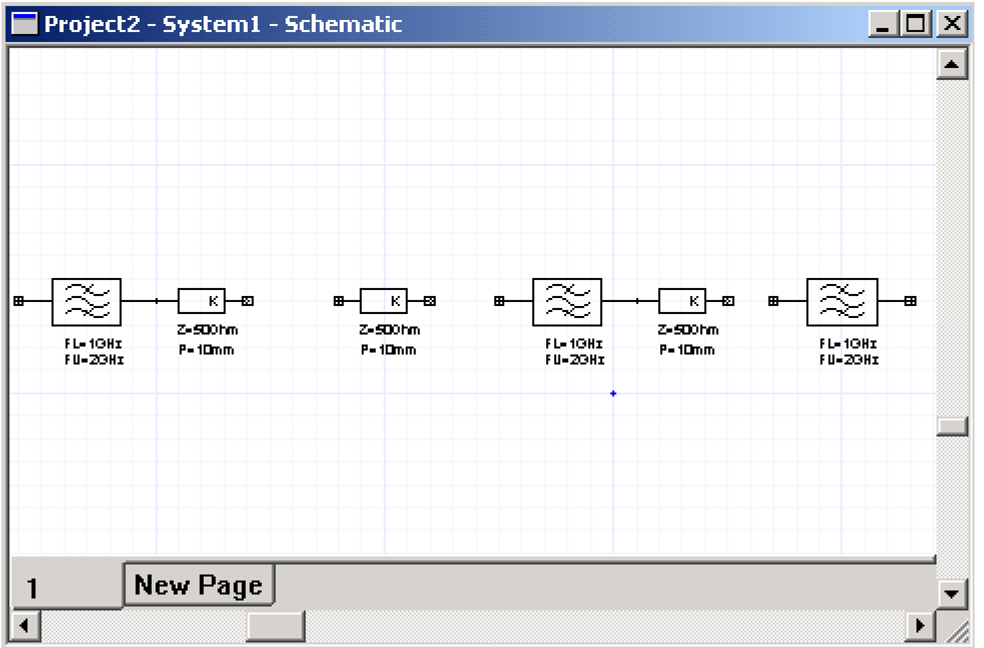
Note that to make multiple copies of a component, you can use any of these shortcut techniques to place multiple components:

- To make multiple copies: Drag the cursor into the Schematic Editor (same procedure as before), left-clicking to place as many copies as needed. As soon as you position the final schematic symbol, right-click and then select **Place and Finish**.
- To copy-and-paste: In the **Schematic Editor**, select a schematic symbol (one click). On the main menu bar, click **Edit** and **Copy**, and then click **Edit** and **Paste**. Drag the cursor to the Schematic Editor, position the component, and then right-click **Place and Finish**.
- Alternatively, you can copy-and-paste by selecting the component and then applying the standard Windows shortcut keys (**Ctrl +C** and **Ctrl +V**).

6. In the Project Manager, expand **Ideal Distributed** (under **Circuit Elements**) and select **Transmission Line, Physical Length**:

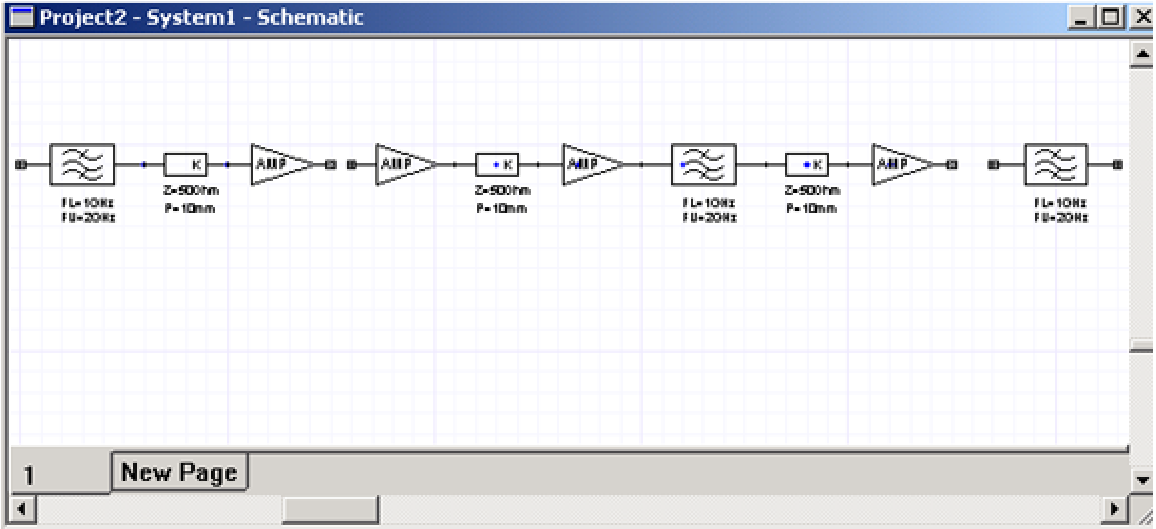


7. Drag the cursor into the **Schematic Editor**, and—using any of the techniques described earlier—place two copies of the transmission line, as shown. At this point your schematic should look like this:



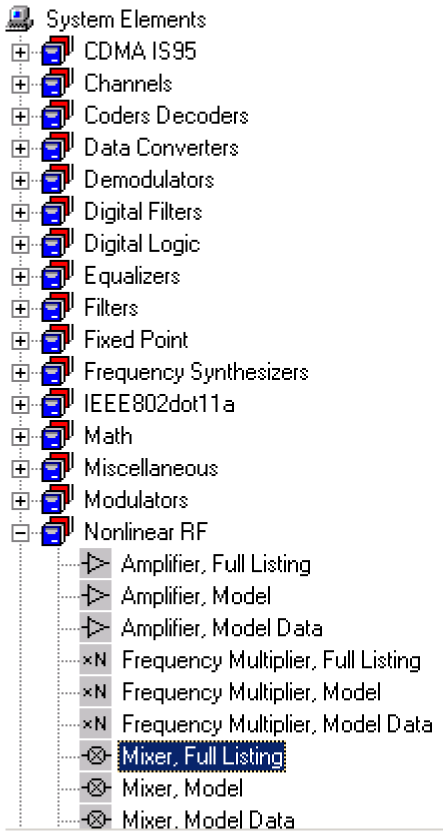
8. In the **Project Manager**, under the **System Elements** icon, expand the **Nonlinear RF** folder.

9. Select **Amplifier**, **Full Listing**, and then place four instances in the **Schematic Editor**:



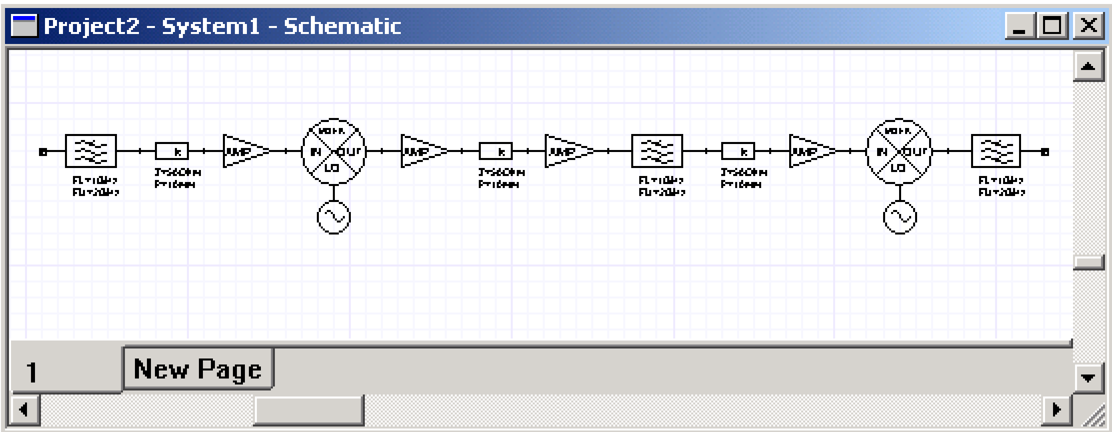
10. Place the mixers:

    a. In the **Project Manager**, under the **System Elements** folder, expand the **Nonlinear RF icon** and select **Mixer**, **Full Listing**:

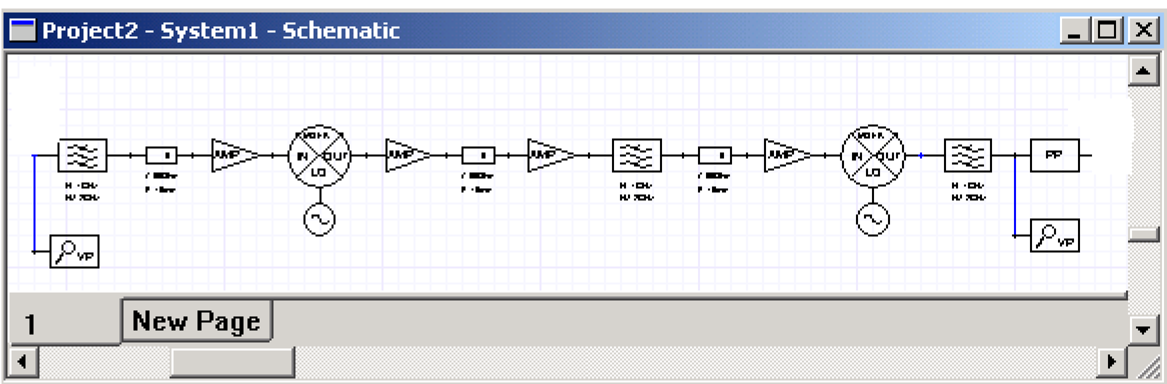


    b. Now place two instances of the mixer in the **Schematic Editor**, as shown.

c. If you need additional space between unconnected sections, create a bounding box around a group of elements and then move them. (Click the left mouse button, drag the cursor to select a group of components, and then move the group.)

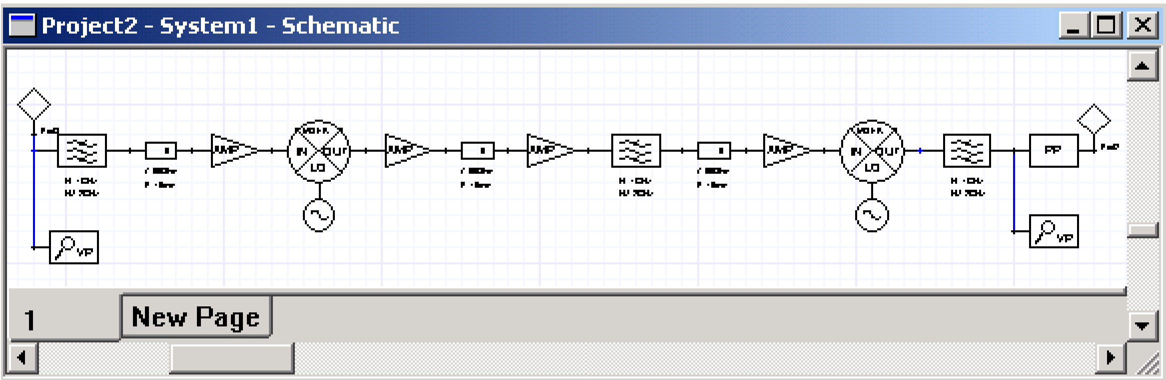


11. Add the voltage and power probes:
   a. In the **Project Manager**, under the **System Elements** folder, expand the **Probes** icon and select **Voltage Probe** (located at the bottom of the list). Insert two copies of the voltage probe (input and output) as shown.
   b. Select **Power Probe** and insert it as shown:

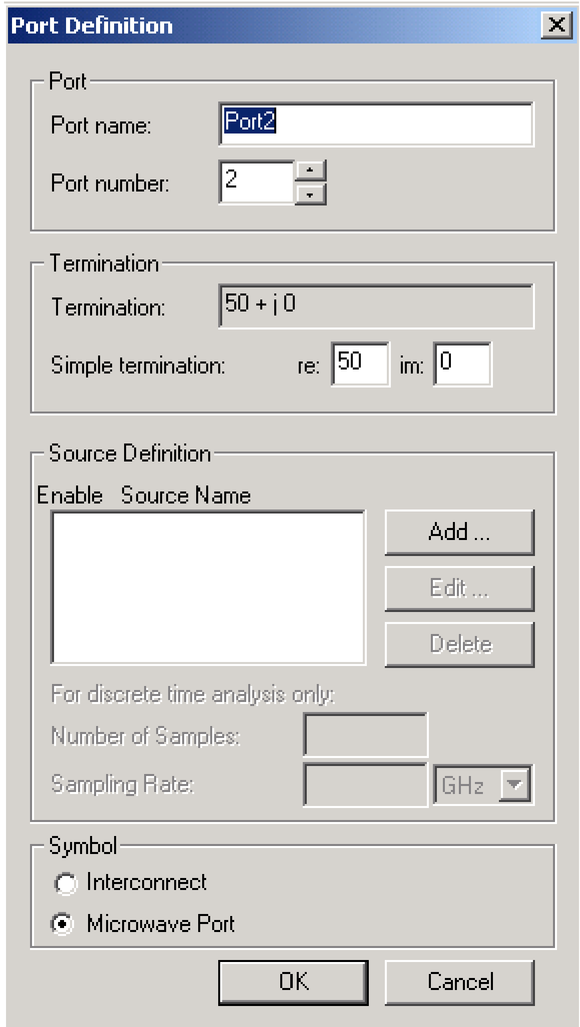


12. Add the input and output ports and define their parameters:

    a. On the **Schematic Draw** toolbar, click the **Interface Port** icon.  and place ports at the input and output, as shown.



    b. After a port is placed, you can open the **Port Definition** dialog box by double-clicking the port symbol.



13. Define the source (excitation) for Port1:

    a. In the schematic editor, double-click the input port. The **Port Definition** dialog box appears. (Same procedure as previous step.)

b. Make sure that the following values appear:

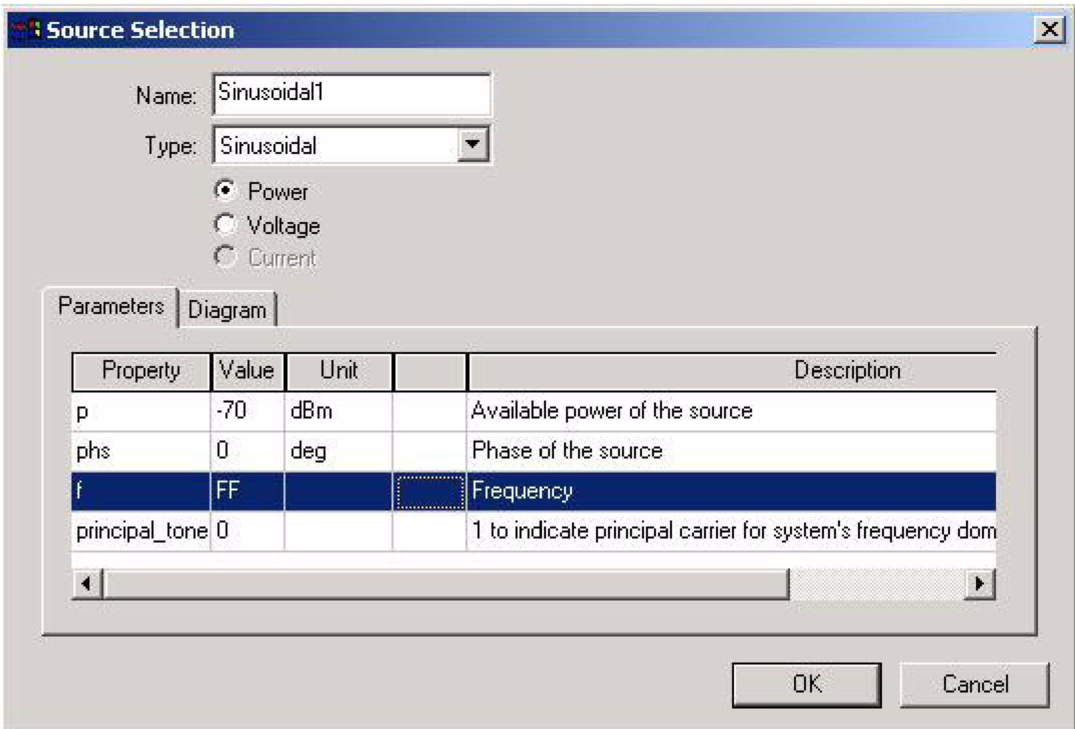
Port name	Port1
Port Number	1
re:	50
im:	0
Microwave Port	Selected

c. Under **Source Definition**, click **Add**, and the **Source Selection** dialog box appears.

d. Set power (**p**) to **-70 dBm**, and select **dBm** in the **Unit** list, if desired (variables may be added without specifying a unit).

e. Specify a variable for frequency:

- Type a new variable name, **FF**, in the **Property** text box in the **Frequency** row.

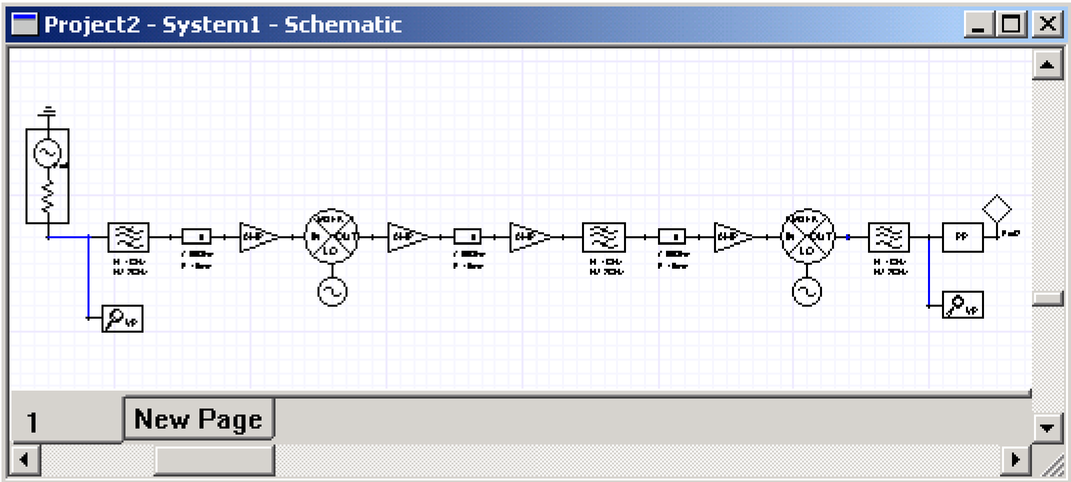


The **Add Variable** dialog appears.

- Type **3.9GHz** in the **Value** box and verify that the **Local Variable** button is selected, then click **OK**.

f. Click **OK** to close the **Source Selection** and **Port Definition** dialog boxes.

- Note that schematic symbol for **port1** has changed into a source symbol.
- Note that the microwave port has been flipped in the x direction.
- Also note that you can add additional sinusoidal excitations. At this point, your schematic should look like this:

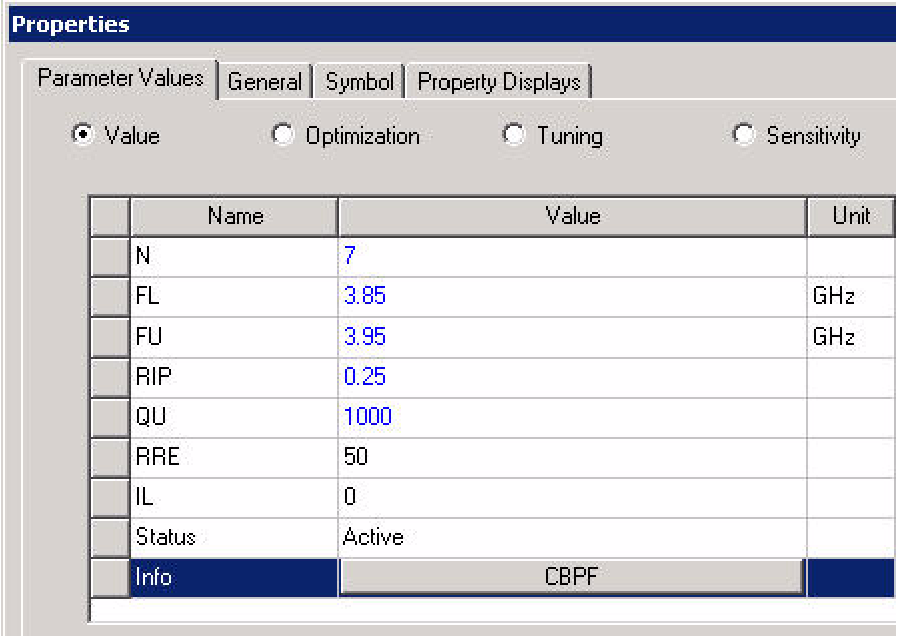


Edit Component Properties

Define the parameters for Filter1:

1. Double-click the schematic symbol for the first filter, and the **Properties** dialog box appears.
2. Enter the values for **N**, **FL**, **FU**, **RIP**, **QU**, and **RRE** as shown. (If a property has units, do not

insert a space between the number and units, for example, 3.8GHz).



3. To display additional information about this component (for example, model notes and netlist syntax) click the **CBPF** button and the **Ansoft Designer Help** window opens.
4. Now enter the following values for **Filter2** and **Filter 3**:

	Filter2	**Filter3**
N	5	5
FL	1.15GHz	0.25GHz
FU	1.25GHz	0.35GHz
RIP	0.2	0.2
QU	1000	1000
RRE	50	50

5. Enter the following values for the transmission lines (leave **Z** and **F** unchanged):

	TRLK1	**TRLK2**	**TRLK3**
P	1in	240 in	1in
K	9.8	2.1	9.8
A	1.9685	0.98425	1.9685
F	3.9GHz	1.2GHz	1.2GHz

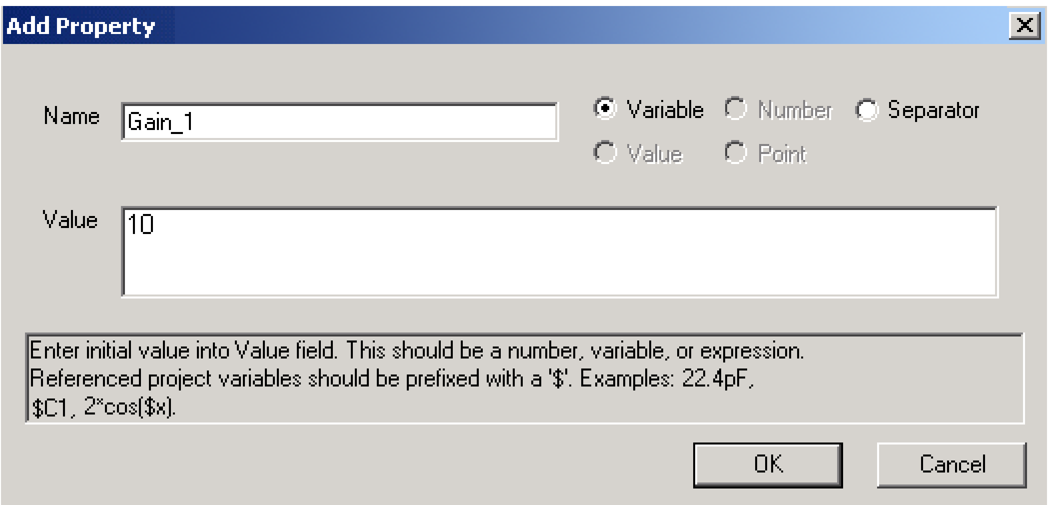
Create a Variable to Define Gain

Create a local variable that will be used to define the gain for the fourth amplifier:

1. On the **System** menu, click **Design Properties**.

   In the **Properties** dialog box, click the **Local Variables** tab and then click **Add**.

   The **Add Property** dialog box appears.
2. In the **Name** box, enter **Gain_1**.
3. In the **Value** box, enter **10**.



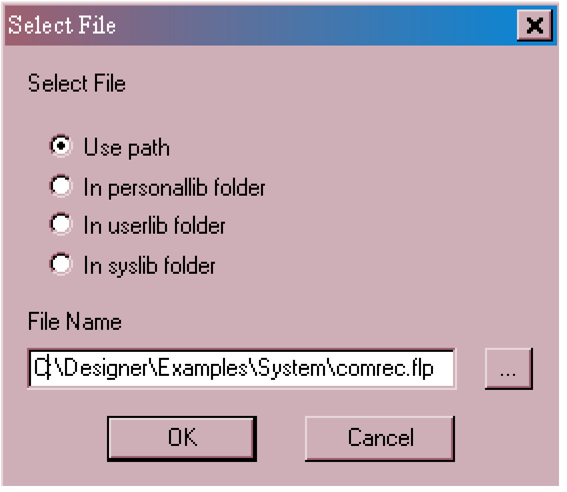
4. Click **OK**.

   For additional explanation of parameters and variables, see Chapter 1, *Terms Used in Ansoft Designer*.

Use an external file to specify compression characteristics for the second amplifier:

1. In this example, we use an external data file, *Comrec.flp*, to define the compression characteristics for the second and fourth amplifiers. (Later, in step 27, we use this file to define the intermodulation table for the two mixers.)
2. Double click the second amplifier, and the **Properties** dialog box appears.

3. In the **File** row, under value, click the blank button.
4. In the **Select File** dialog box, select **Use Path** and click the browse button ... to browse to the directory where Ansoft Designer is installed. Browse to: **Examples\System\comrec.flp**.
5. Click **Open**, and then **OK**.



6. Click **OK** to close the **Select File** and **Properties** dialog boxes.
7. Set values for **Amplifier1**, **Amplifier2**, **Amplifier3**, and **Amplifier4** as shown in the table (leave all other values unchanged):

	Amplifier1	**Amplifier2**	**Amplifier3**	**Amplifier4**
FILE		comrec.flp	comrec.flp	
DEVICE		MID_GAIN	GAIN_2	
MS11	-12	-12	-12	-15
MS12	-60	-60	-50	-50
MS21	30			Gain_1
MS22	-18	-18	-15	-15
NF	2	6	8	6
OIP3	0dBm		10dBm	
P1DB				5dBm

8. Set values for the **Mixer1** and **Mixer2**, as shown in the table:

	Mixer1	**Mixer2**
FILE	comrec.flp	comrec.flp
DEVICE	MIX_1	MIX_2
TEMP		
MS11	-10	-12
MS22	-10	-15
CONVGAINMAG	-8.5	-10
NF	8	10
OIP3		20dBm
MAXO	10	10
MINF	1GHz	0.1GHz
MAXF	6GHz	1GHz
FLO	2.7GHz	0.9GHz

9. Assign a name to each of the probes:
   a. Double-click the first voltage probe, and its **Properties** dialog box appears. Enter **Vin** for **Name** and then click **OK** to close.
   b. Double-click the second voltage probe, and its **Properties** dialog box appears. Enter **Vout** for **Name** and then click **OK** to close.
   c. Double-click the power probe, and its **Properties** dialog box appears. Enter **Pout** for **Name** and then click **OK** to close.
   d. At this point, the schematic has all of the necessary parameter values assigned.
10. Change the schematic name from **System1** to **SingleTone**:
    a. In the **Project Manager**, right-click the **System1** icon, select **Rename**, and then type **SingleTone**.
    b. On the menu bar, click **File** and then click **Save**. When the **Save** dialog box opens, enter **ComReceiver** in the **Name** field.

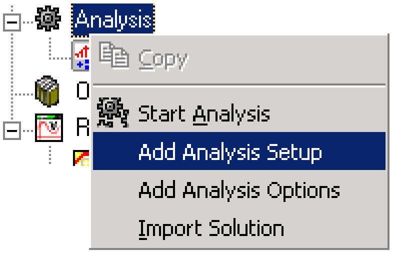
Set up the System Analyses

Two separate analyses will be set up for this project: A simple frequency-domain sweep, and a discrete time-domain analysis (which requires additional data in the **Source Selection** dialog box).

Frequency Domain Example

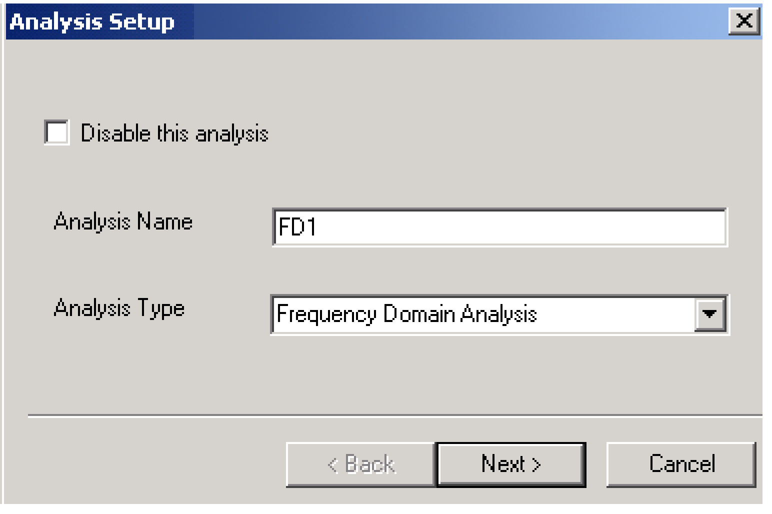
To set up a frequency-domain analysis:

1. On the **System** menu, click **Add Analysis Setup**.
   - Alternatively, in the **Project Manager**, right-click the **Analysis** icon and then click **Add Analysis Setup**.



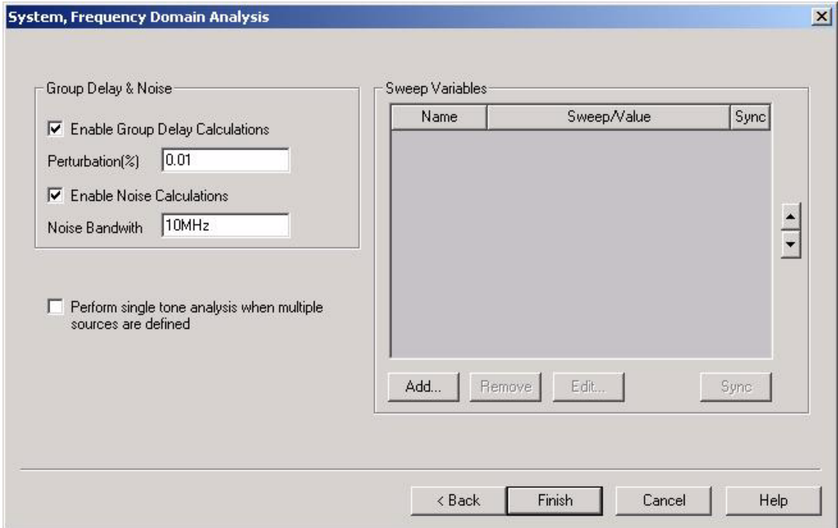
   The **Analysis Setup** dialog box appears.

2. Accept the defaults for **Analysis Name** and **Analysis Type**. Click **Next**.



3. In the **System, Frequency Domain Analysis** dialog box:
   a. Make sure that **Enable Group Delay Calculations** and **Enable Noise Calculations** are selected.

b. Make sure that **Perturbation (%)** and **Noise Bandwidth** are set to 0.01 and 10MHz.



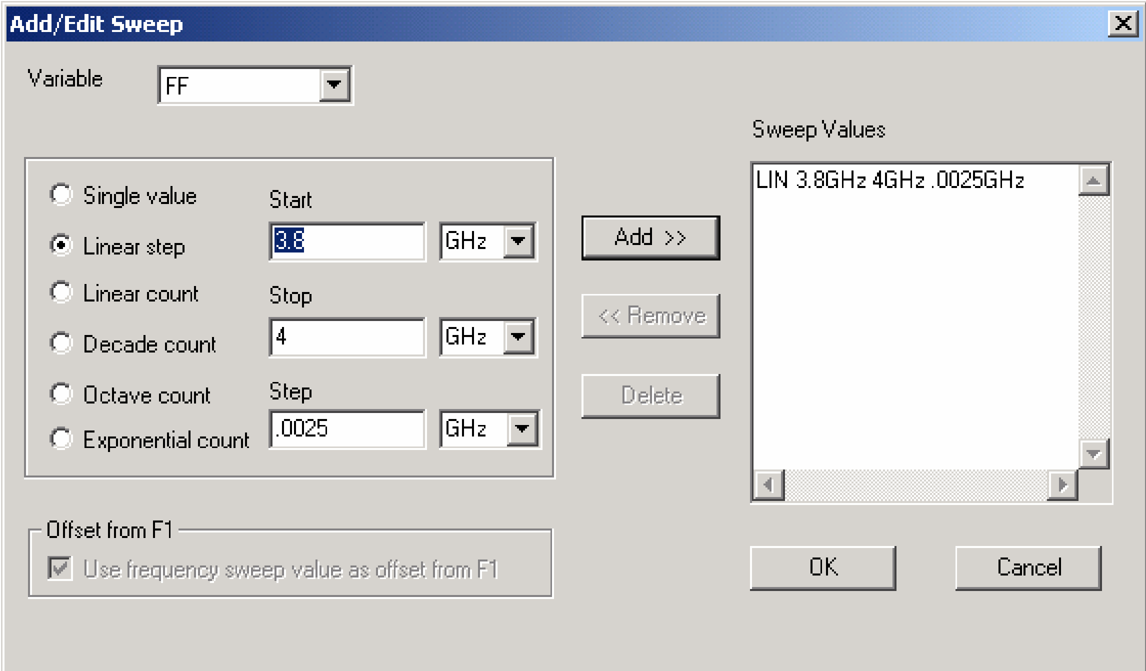
4. In the **System, Frequency Domain Analysis** dialog box, click **Add** and the **Add/Edit Sweep** dialog box appears:

   a. In the **Variable** list, select **FF**.

   b. Select **Linear Step** and enter **3.8 GHz** for **Start**, **4 GHz** for **Stop**, and **.0025 GHz** for **Step**. (Make sure that the units are set to **GHz**).

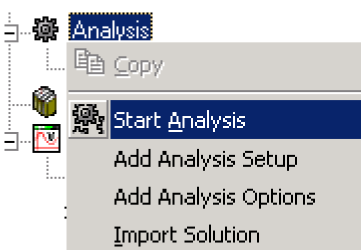
   c. Click **Add** and then click **OK** to close the **Add/Edit Sweep** dialog box.

   d. Click **Finish** to close the **System, Frequency Domain Analysis** dialog box.

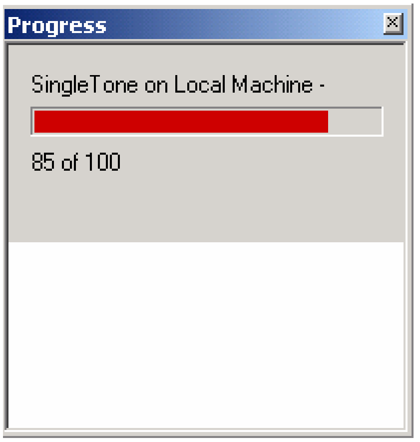


Start the analysis:

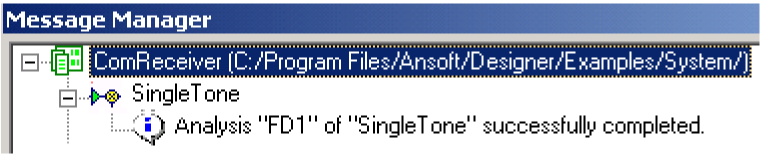
- In the **Project Manager**, right-click the **Analysis** icon and then click **Analyze**.



As soon as the analysis starts, the **Progress** window becomes active:



- Check the **Message Manager** for status:
  In this example the message confirms a successful analysis, but if an error occurs it will be reported here.

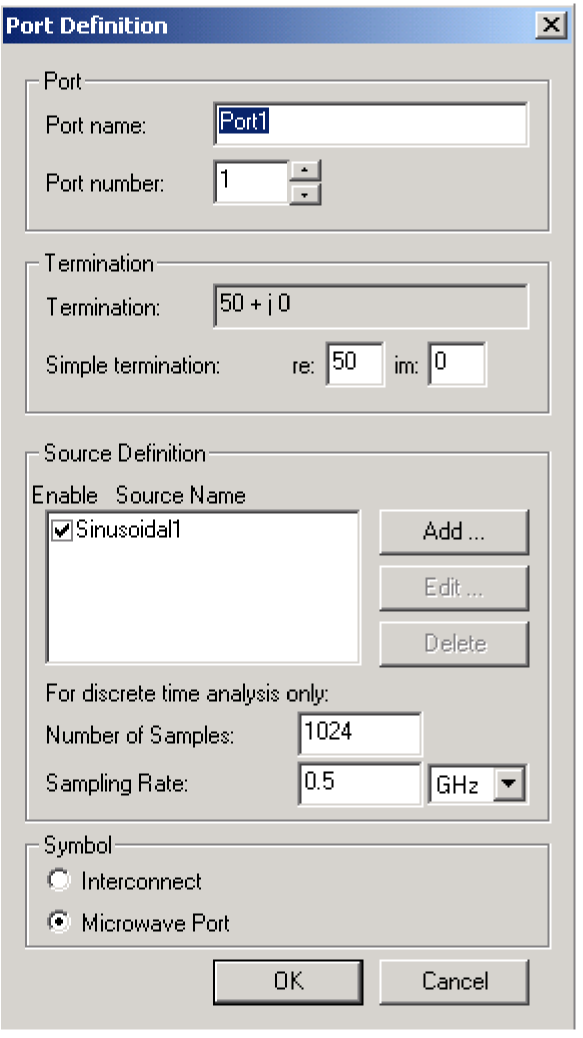


Time Domain Example (TD1)

To analyze the receiver in the discrete time domain:

1. First, you must provide additional timing information for the input source: Double-click on the input port, and the **Port Definition** dialog box appears.
2. In the **Port Definition** dialog box, under **Source Definition**., select **Sinusoidal1**.

3. Enter **1024** for **Number of Samples**, and enter **0.5GHz** for **Sampling Rate**. (These additional parameters are required for discrete time analysis).



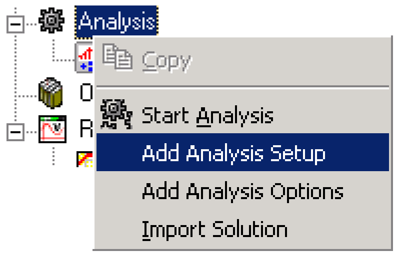
- Note that the total duration of the discrete-time simulation will be **Number of Samples/ Sampling Rate** = 2.048 µs.
- Also note that since **Sampling Rate** is less than the source frequency (**f**), this discrete time simulation is based on envelope analysis, so the source fundamental carrier (**f**) is not sampled. In this case, a single sinusoid has a constant envelope proportional to the source power. Envelope simulation is usually a more efficient choice when we are only interested in examining in-band responses.

4. Click **Edit**. The **Source Selection** dialog box appears.

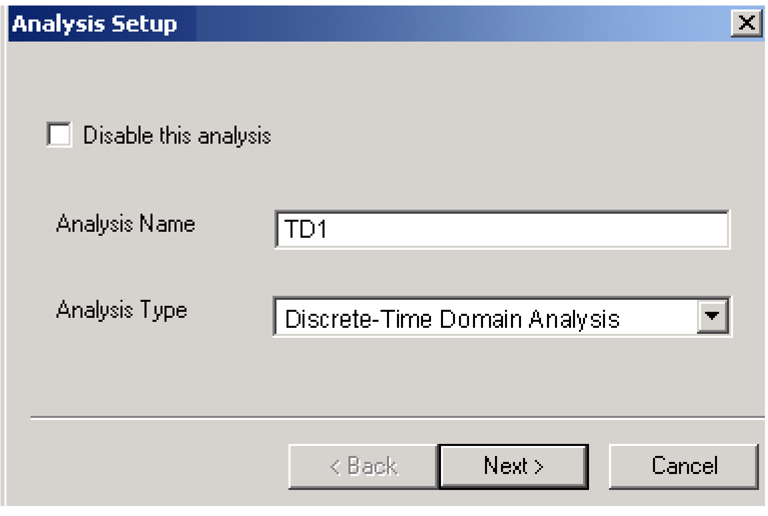
5. Verify that **f** is set to **FF**, and leave **p** set to **-70 dBm**. Leave the other parameters unchanged.
6. Click **OK**.

To set up a time-domain analysis:

1. On the **System** menu, click **Add Analysis Setup**.
   - Alternatively, in the **Project Manager**, right-click the **Analysis** icon and then click **Add Analysis Setup**.



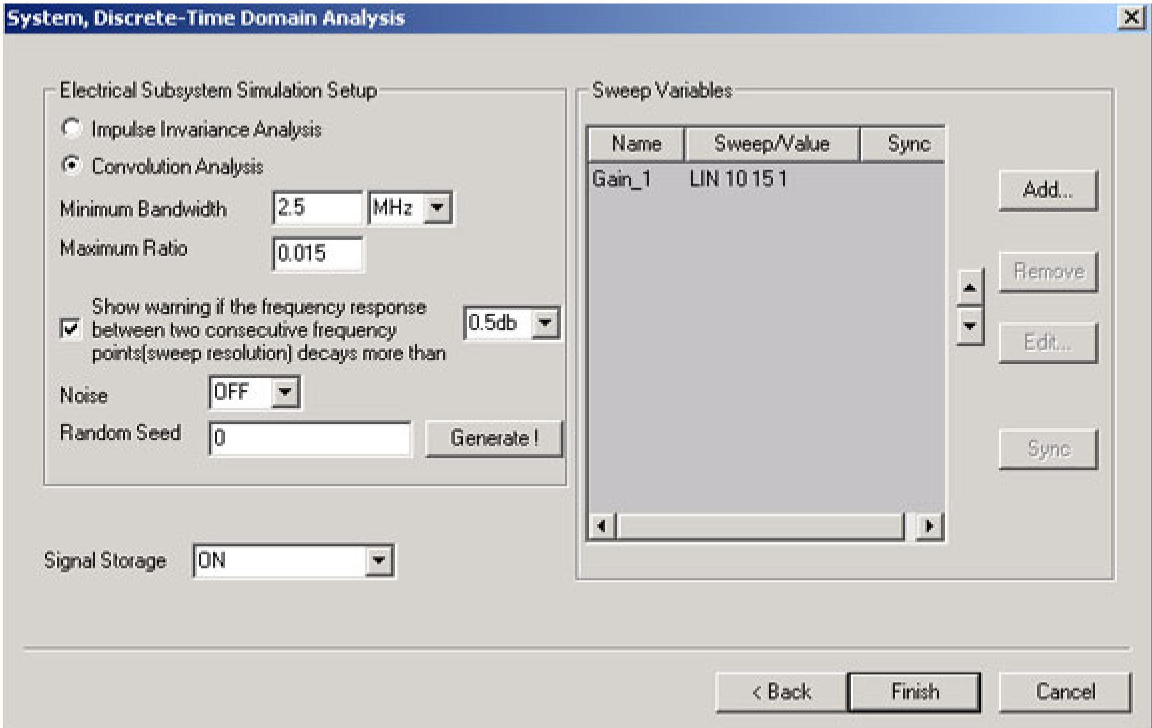
2. When the **Analysis Setup** dialog box appears, accept the default for **Analysis Name** and select **Discrete-Time Domain Analysis** as the **Analysis Type**, and then click **Next**.



   The **System, Discrete-Time Domain Analysis** dialog box appears.

3. Select **Convolution Analysis**.

4. Set **Minimum Bandwidth** to 2.5 MHz and **Maximum Ratio** to .015, as shown.

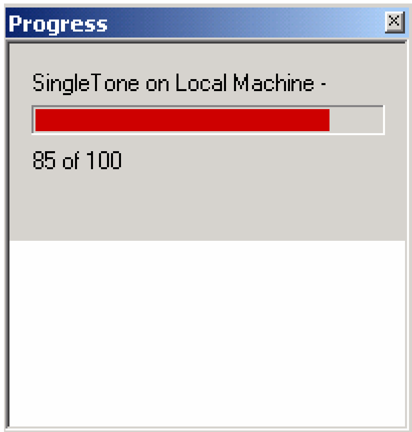


5. To set the **Sweep Variables**: Click **Add**, and the **Add/Edit Sweep** dialog box appears.
6. In the **Variable** list, select **Gain_1**.
7. Select **Linear Step** and enter, 10, 15, and 1 in the **Start**, **Stop**, and **Step** fields. Click **Add**.
8. Click **OK** and then click **Finish** to close the **Add/Edit Sweep** and **System, Discrete-Time Domain Analysis** dialog boxes.

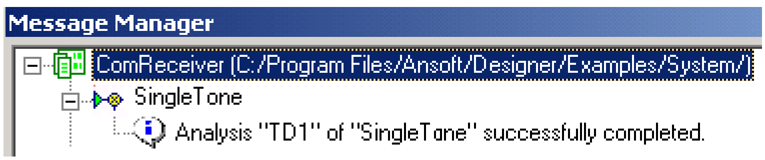
Start the analysis:

- In the **Project Manager**, right-click the **TD1** icon and then click **Analyze.**

  Note that the **Start Analysis** command sequentially runs all of the analysis-setups in the design. To run an analysis for **TD1** only, right-click the **TD1** icon and select **Analyze TD1**. Note that as soon as an analysis starts, the progress bar appears:



Check the **Message Manager** for status:
In this example the message confirms a successful analysis, but if an error occurs it will be reported here.



Display the Results

At this point we've successfully run a frequency-domain analysis (FD1) and a time-domain analysis (TD1). Now we'll display the results for both cases.

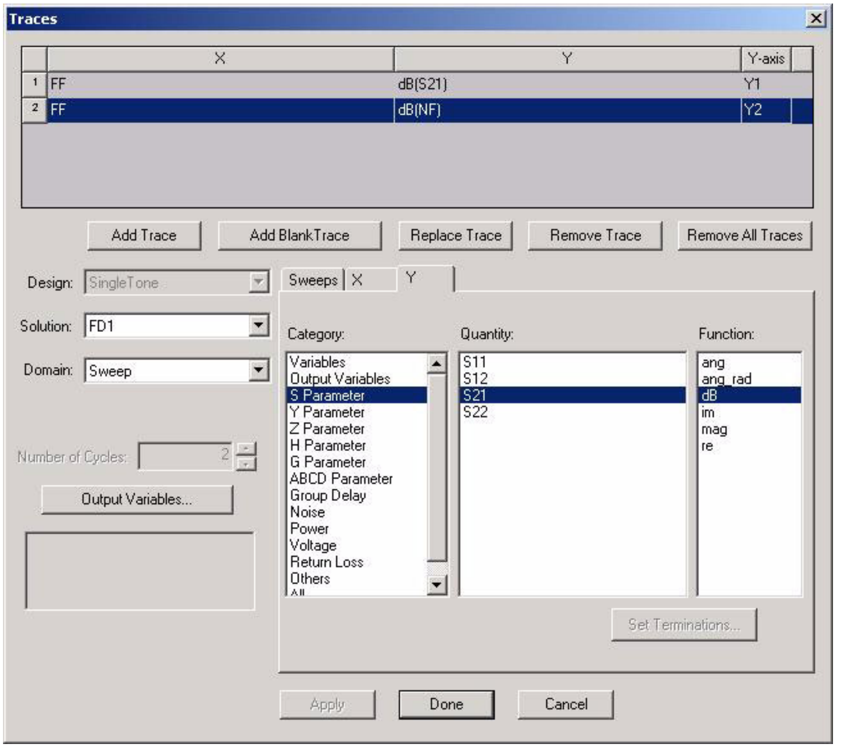
Display Results for the Frequency-Domain Analysis

To report the results for FD1:

1. On the **System** menu, click **Create Report**. Also note these two alternative ways to insert a report:
   - On the **Systems Solutions** toolbar, click the **Create Report** icon. 
   - In the **Project Manager** window, right-click the **Project** icon, point to **Create Report**.
2. When the **Create Report** dialog box appears, click **OK**. Select Standard and Rectangular Plot.
3. In the **Traces** dialog, plot S21 and NF:
   a. In the **Solution** list, make sure that **FD1** is selected (default setting).
   b. In the **Domain** list, make sure that **Sweep** is selected.
   c. Add the first trace: In the **Category**, **Quantity**, and **Function** lists, select **S Parameter**, **S21**, and **dB**. Click **Add Trace**, and the data is displayed in the first row.

d. Add the second trace: In the **Category**, **Quantity**, and **Function** lists, select **Noise**, **NF**, and **dB**.

e. Click **Add Trace**, and the data for the second trace is displayed in the second row, as shown.



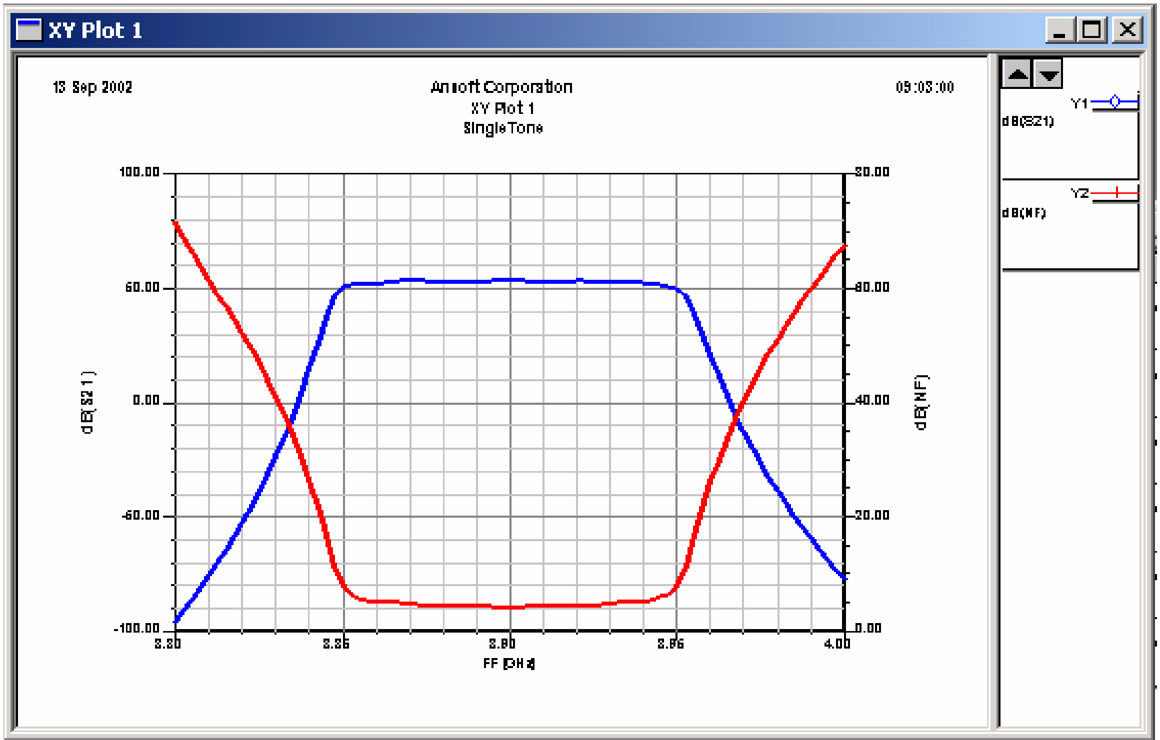
4. To plot **NF** on a separate Y-axis:

   a. In the trace-display area (located at the top of the **Traces** dialog box), move the cursor to rightmost column and click the area under the **Y-Axis** column. A list of Y-Axis labels appears (**Y1**, **Y2**, and so on).

   b. Select **Y2** for the NF so that the noise figure is plotted on a separate Y-axis to the S-parameter.

   c. Click **Done** and the graphs for **S21** and **NF** will appear as shown.

   d. To generate a variety of other responses, just repeat this procedure. (For additional details about the Traces dialog, see the first chapter of this Getting Started Guide.)

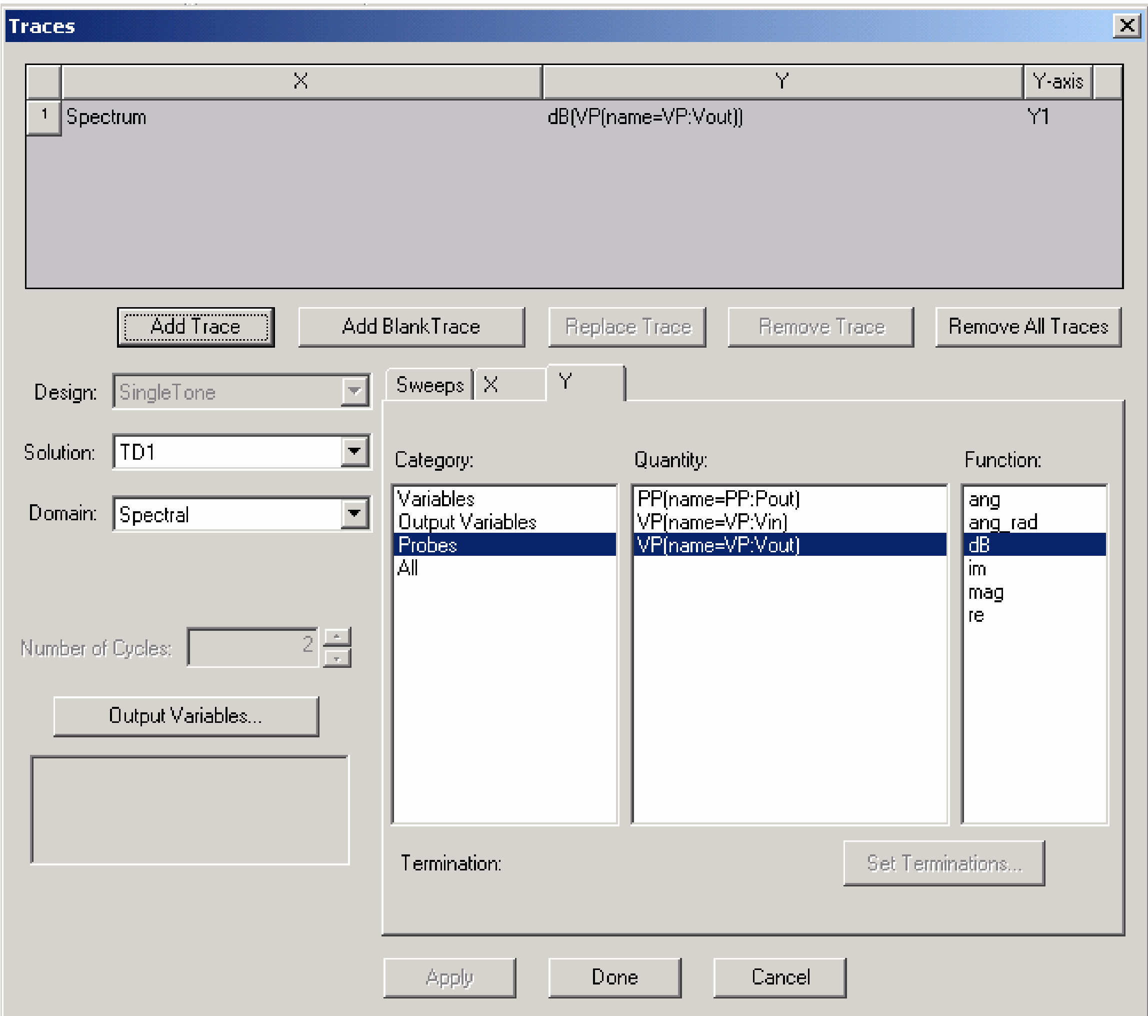


Display Results for the Time-Domain Analysis

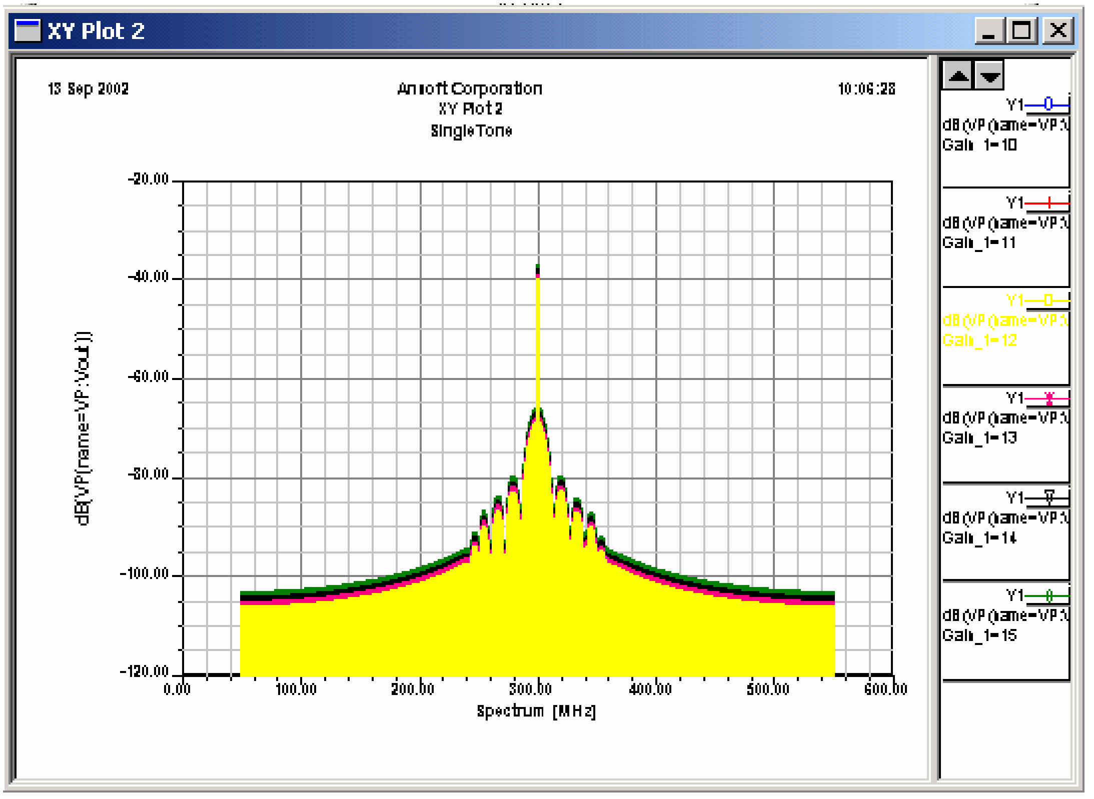
In this case, we will plot the spectrum variations when the gain of the fourth amplifier is swept (a local variable, Gain_1, was set up and assigned to in steps 18 and 21).

To report the results for TD1:

1. On the **System** menu, click **Create Report**. Also note these two alternative ways to insert a report:
   - On the **Systems Solutions** toolbar, click the **Create Report** icon. 
   - In the **Project Manager** window, right-click the **Project** icon, point to **Create Report**.
2. When the **Create Report** dialog box appears, click **OK**. Select Standard and Rectangular Plot.
3. In the **Traces** dialog, plot the spectrum for the output-voltage probe (**Vout**):
   a. In the **Solution** list, select **TD1**.
   b. In the **Domain** list, select **Spectral**.
   c. Add the trace data: In the **Category**, **Quantity**, and **Function** lists, select **Probes**, **VP(name=VP:Vout**, and **dB**. Click **Add Trace**, and the data is displayed in the first row.



4. Click **Done**.
5. To see how the spectral plot varies for various values of gain (for example, 10 dB or 13 dB), click the thumbnails on the right side of the display. Zoom in on areas of interest.


XY Plot 2
13 Sep 2002
Ansoft Corporation
XY Plot 2
SingleTone
10:06:28
-20.00
-40.00
-60.00
-80.00
-100.00
-120.00
dB(VP(name=VP:Vout))
0.00
100.00
200.00
300.00
400.00
500.00
600.00
Spectrum [MHz]
Y1
Gain_1=10
Y1
Gain_1=11
Y1
Gain_1=12
Y1
Gain_1=13
Y1
Gain_1=14
Y1
Gain_1=15