**Agilent Technologies**

**Advanced Design System 2002**

# Momentum

**February 2002**

## Notice

The information contained in this document is subject to change without notice.

Agilent Technologies makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

### Warranty

A copy of the specific warranty terms that apply to this software product is available upon request from your Agilent Technologies representative.

### Restricted Rights Legend

Use, duplication or disclosure by the U. S. Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 for DoD agencies, and subparagraphs (c) (1) and (c) (2) of the Commercial Computer Software Restricted Rights clause at FAR 52.227-19 for other agencies.

Agilent Technologies
395 Page Mill Road
Palo Alto, CA 94304 U.S.A.

### Acknowledgements

MS-DOS®, Windows®, and Windows NT® are U.S. registered trademarks of Microsoft Corporation.

UNIX® is a registered trademark of the Open Group.

# Contents

**7   Simulation**

**8   Radiation Patterns and Antenna Characteristics**

# Chapter 1: Momentum Basics

Momentum is a part of Advanced Design System and gives you the simulation tools you need to evaluate and design modern communications systems products. Momentum is an electromagnetic simulator that computes S-parameters for general planar circuits, including microstrip, slotline, stripline, coplanar waveguide, and other topologies. Vias and airbridges connect topologies between layers, so you can simulate multilayer RF/microwave printed circuit boards, hybrids, multichip modules, and integrated circuits. Momentum gives you a complete tool set to predict the performance of high-frequency circuit boards, antennas, and ICs.

Momentum Optimization extends Momentum capability to a true design automation tool. The Momentum Optimization process varies geometry parameters automatically to help you achieve the optimal structure that meets the circuit or device performance goals.

Momentum Visualization is an option that gives users a 3-dimensional perspective of simulation results, enabling you to view and animate current flow in conductors and slots, and view both 2D and 3D representations of far-field radiation patterns.

If you are unfamiliar with Advanced Design System, refer to the *Quick Tour* in the online documentation, and to the *User's Guide*. To learn how to create a physical design, refer to *Layout*. For information on the interactions between Layout and Momentum, refer to Appendix B, Drawing Tips.

## Momentum Major Benefits

Momentum enables you to:

- Simulate when a circuit model range is exceeded or the model does not exist
- Identify parasitic coupling between components
- Go beyond simple analysis and verification to design automation of circuit performance
- Visualize current flow and 3-dimensional displays of far-field radiation

## Momentum Major Features

Key features of Momentum include:

- An electromagnetic simulator based on the Method of Moments

- Adaptive frequency sampling for fast, accurate, simulation results

- Optimization tools that alter geometric dimensions of a design to achieve performance specifications

- Comprehensive data display tools for viewing results

- Equation and expression capability for performing calculations on simulated data

- Full integration in the ADS circuit simulation environment allowing EM/Circuit co-simulation

# Momentum Overview

Momentum commands are available from the Layout window. The following steps describe a typical process for creating and simulating a design with Momentum:

1. **Create a physical design**. You start with the physical dimensions of a planar design, such as a patch antenna or the traces on a multilayer printed circuit board. There are three ways to enter a design into Advanced Design System:

   - Convert a schematic into a physical layout

   - Draw the design using Layout

   - Import a layout from another simulator or design system. Advanced Design System can import files in a variety of formats.

   For information on converting schematics or drawing in Layout, refer to the *Layout* manual. For information on importing designs, refer to the manual, *Importing and Exporting Designs*.

2. **Choose Momentum or Momentum RF mode.** Momentum can operate in two simulation modes: microwave or RF. You can select the mode based on your design goals. Use *Momentum* (microwave) mode for designs requiring full-wave electromagnetic simulations that include microwave radiation effects. Use *Momentum RF* mode for designs that are geometrically complex, electrically small, and do not radiate. You might also choose Momentum RF mode for quick simulations on new microwave models that can ignore radiation effects, and to conserve computer resources. For more information comparing the Momentum and Momentum RF modes, see "About the Simulation Modes" on page 1-4.

3. **Define the substrate characteristics.** A substrate is the media upon which the circuit resides. For example, a multilayer PC board consists of various layers of metal, insulating or dielectric material, and ground planes. Other designs may include covers, or they may be open and radiate into air. A complete substrate definition is required in order to simulate a design. The substrate definition includes the number of layers in the substrate and the composition of each layer. This is also where you position the layers of your physical design within the substrate, and specify the metal characteristics of these layers. For more information, refer to the chapter, Chapter 3, Substrates.

4. **Solve the substrate.** Momentum calculates the Green's functions that characterize the substrate for a specified frequency range. These calculations are stored in a database, and used later on in the simulation process. For more information, refer to Chapter 3, Substrates.

5. **Assign port properties.** Ports enable you to inject energy into a circuit, which is necessary in order to analyze the behavior of your circuit. You apply ports to a circuit when you create the circuit, and then assign port properties in Momentum. There are several different types of ports that you can use in your circuit, depending on your application. For more information, refer to Chapter 4, Ports.

6. **Add a box or a waveguide.** These elements enable you to specify boundaries on substrates along the horizontal plane. Without a box or waveguide, the substrate is treated as being infinitely long in the horizontal direction. This treatment is acceptable for many designs, but there may be instances where a boundaries need to be taken into account during the simulation process. A box specifies the boundaries as four perpendicular, vertical walls that make a box around the substrate. A waveguide specifies two vertical walls that cut two sides of the substrate. For more information, refer to Chapter 5, Boxes and Waveguides.

7. **Create Momentum components.** Momentum components can be used in the schematic design environment in combination with all the standard ADS active and passive components to build and simulate circuits including the parasitic layout effects. The Momentum engine is automatically invoked to generate an S-parameter model for the Momentum component during the circuit simulation. For more information on the Momentum Components and EM/Circuit co-simulation feature, refer to "Momentum Components" on page 7-24.

8. **Set up and generate a circuit mesh.** A mesh is a pattern of rectangles and triangles that is applied to a design in order to break down (discretize) the design into small cells. A mesh is required in order to simulate the design effectively. You can specify a variety of mesh parameters to customize the mesh to your design, or use default values and let Momentum generate an optimal mesh automatically. For more information, refer to Chapter 6, Mesh.

9. **Simulate the circuit**. You set up a simulation by specifying the parameters of a frequency plan, such as the frequency range of the simulation and the sweep type. When the setup is complete, you run the simulation. The simulation process uses the Green's functions computed for the substrate, plus the mesh pattern, and the currents in the design are calculated. S-parameters are then computed based on the currents. If the Adaptive Frequency Sample sweep type is chosen, a fast, accurate simulation is generated, based on a rational fit model. For more information, refer to Chapter 7, Simulation.

10. **View the results.** The data from an Momentum simulation is saved as S-parameters or as fields. Use the Data Display or Visualization to view S-parameters and far-field radiation patterns. For more information, refer to Chapter 9, Viewing Results Using the Data Display and Chapter 10, Momentum Visualization Basics.

# About the Simulation Modes

With ADS 1.5, Momentum added an RF simulation mode to its existing microwave mode. The microwave mode is called Momentum; the new RF mode is called Momentum RF. Momentum RF provides accurate electromagnetic simulation performance at RF frequencies, typically up to 10 GHz; however, this upper limit depends on the size of your physical design. At higher frequencies, as radiation effects increase, the accuracy of the Momentum RF models declines smoothly with increased frequency. Momentum RF addresses the need for faster, more stable simulations down to DC, while conserving computer resources. Typical RF applications include RF components and circuits on chips, modules, and boards, as well as digital and analog RF interconnects and packages.

When compared to the Momentum mode, the Momentum RF mode uses new technologies enabling it to simulate physical designs at RF frequencies with several useful benefits. The RF mode is based on quasi-static electromagnetic functions enabling faster simulation of designs. The star-loop technology provides low-frequency stability down to DC. The mesh-reduction algorithms allow quick simulation of complex designs, and require less computer memory and CPU time.

Additionally, Momentum RF has the same use-model as Momentum in ADS, and works with Momentum Visualization and Optimization. The following table shows how each mode supports Momentum product features. For a detailed comparison of the two simulation modes, see "Comparing the Microwave and RF Simulation Modes" on page A-20.:

Table 1-1. Features Supported in Momentum and Momentum RF Modes

| Feature | | Momentum Mode | Momentum RF Mode |
|---|---|---|---|
| Substrate | Database | Yes | Yes |
| | Functions | Fullwave | Quasi-static |
| Ports | Port Types | Yes | Yes |
| | Calibration | Yes | Yes |
| Box Waveguide | | Yes | No |
| Mesh | Mesh Options | Yes | Yes |
| | Cells | Rectangle / Triangle | Polygonal |
| Simulation | Sweep Types | Yes | Yes |
| | Base Functions | Rooftop | Star/Loop |
| Optimization | | Yes | Yes |
| Post Processing | Visualization | Yes | Yes |
| | Radiation Pattern | Yes | No |

## Selecting the Correct Mode

In the Layout window, the *Momentum* menu label displays the current simulation mode. To select the mode, toggle the mode setting.

- To switch the mode from *Momentum* to *Momentum RF*, choose **Momentum** > **Enable RF Mode**.

- To switch the mode from *Momentum RF* to *Momentum*, choose **Momentum RF** > **Disable RF Mode**.

    In each case, the menu label in the Layout window changes to the current mode.

Deciding which mode to use depends on your application. Each mode has its advantages. In addition to specifically RF applications, Momentum RF can simulate microwave circuits. The following graph identifies which mode is best suited for various applications. As you can see, some applications can benefit from using either mode depending on your requirements. As your requirements change, you can

quickly switch modes to simulate the same physical design. As an example, you may want to begin simulating microwave applications using Momentum RF for quick, initial design and optimization iterations, then switch to Momentum to include radiation effects for final design and optimization.
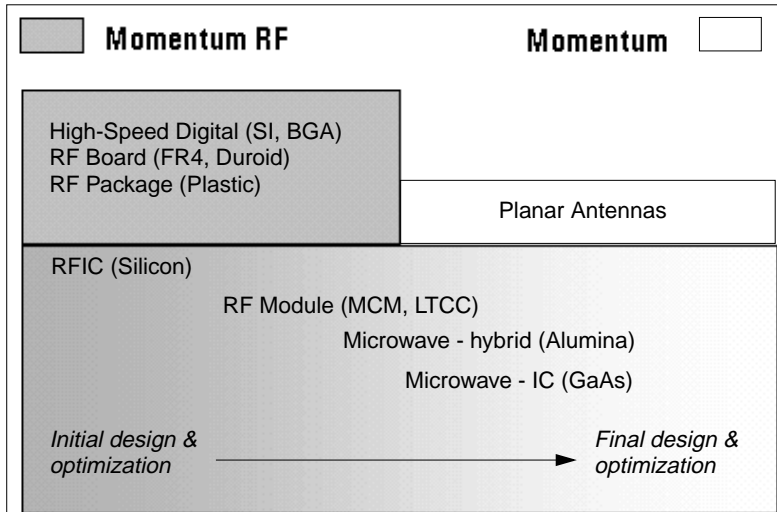


Figure 1-1. Choose the mode that matches the application.

Momentum RF is usually the more efficient mode when a circuit

- is electrically small.
- is geometrically complex.
- does not radiate.

For descriptions about electrically small and geometrically complex circuits, see "Matching the Simulation Mode to Circuit Characteristics" on page A-21.

# Locating Example Projects

Examples of designs that are simulated using Momentum and Momentum RF can be found in the Examples directory.

To open an example project:

1. From the Main window, choose **File** > **Example Project**.

2. From the *Open Example Project* dialog box, position the mouse in the *Directories* field and double-click *Momentum*.

3. In the *Directories* field, double-click one of the example directories: *Antenna*, *Microwave*, *Optimization*, or *RF*.

4. Select a project from the *Files* field, then click **OK**.

Several of these examples are referred to in the documentation to highlight applications of various Momentum features.

# Chapter 2: Getting Started with Momentum

This chapter is intended to help you get started with Momentum. It illustrates basic tasks and exercises using Momentum. The subsequent chapters contain more in-depth reference information and examples.

## Layout Basics

There are two basic ways to create a layout:

- From a schematic (in a Schematic window)
- Directly (in a Layout window)

The best approach to creating a layout depends on the design and the designer.

This section describes how to create a layout automatically from a finished schematic, and how to use the basic features in a Layout window to create a layout directly.

---

**Note**   When you are through working in layout, release the Layout license so that it is available to another user. To do this, select **File** > **Release Layout License** from the Layout window.

---

## Creating a Layout from a Complete Schematic

If all data are contained in the schematic, it is very simple to create a layout.

1. In the Schematic window, build the schematic shown here, then choose the menu command **Layout** > **Generate/Update Layout** (the schematic is the source representation). The Generate/Update Layout dialog box appears.
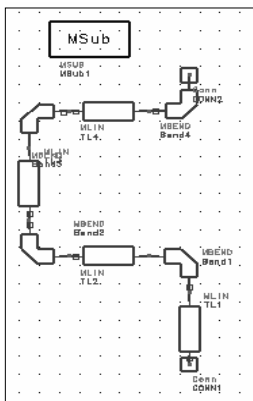


By default, the layout will begin with P1, at 0,0, with an angle of 0 degrees. There is no existing layout, so the Equivalence (the layout component that

corresponds to the starting component in the schematic) is shown as not created.

Starting item for generation
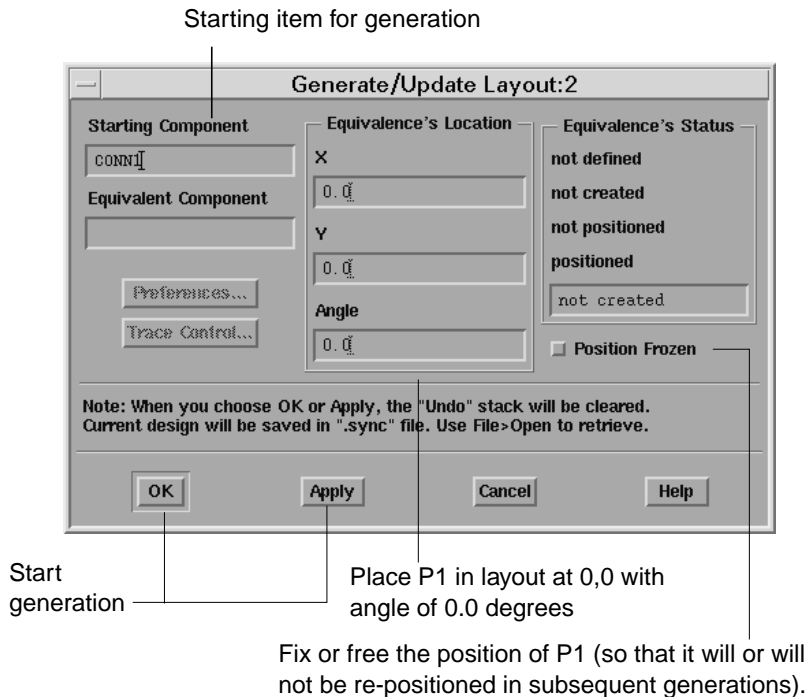


At this point, all of the elements in the schematic are highlighted, indicating that they all need to be generated.

2. Click **OK**.

The Status of Layout Generation dialog box appears. It displays the number of designs processed, the number of items regenerated (created) in the layout, the

number of items that are oriented differently in the layout than in the schematic, and the number of schematic components that were not placed in the layout.
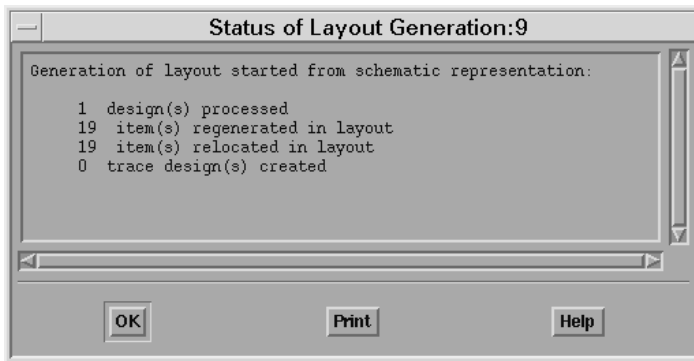


The program automatically opens a Layout window and places the generated layout in it. The orientation of the layout is different from that of the schematic, because the layout is drawn from left to right across the page, beginning at the **Starting Component**.
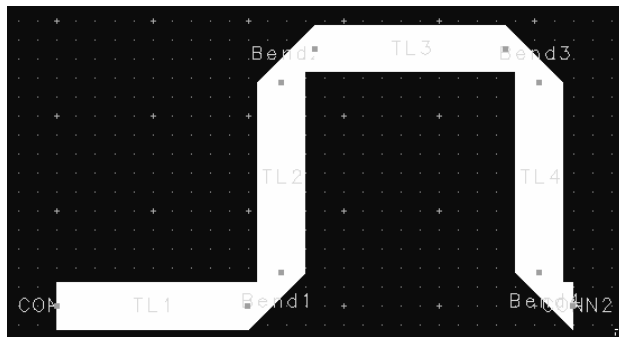
## Creating a Layout Directly

Launch Advanced Design System and create a project. To display a Layout window, choose **Window** > **Layout** from a Schematic window.
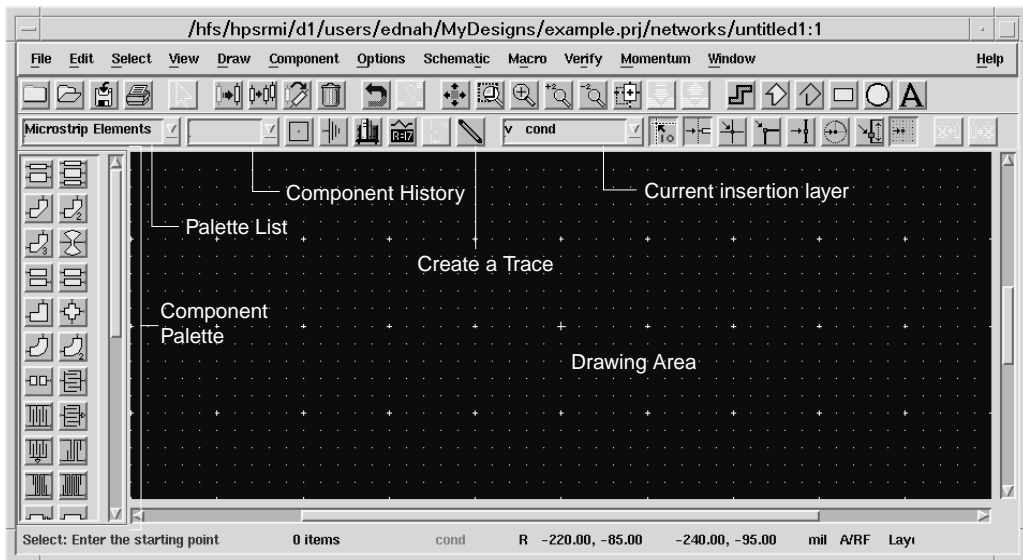


The following tasks are performed the same way in a Layout window as they are in a Schematic window:

- Selecting components
- Placing and deselecting components
- Changing views
- Hiding component parameters
- Coping and rotating components
- Using named connections

## Connecting Components with a Trace

As in a Schematic window, you can connect components without having them actually touch. In a Layout window, this is done by placing a *trace* between the component (in

the same way a wire is used in the Schematic window). Either choose **Component** > **Trace**, or click the Trace button in the toolbar.

Unlike wires in a Schematic window, a trace in a Layout window may be inserted alone (click twice to end insertion).

## Shapes

In Layout you can insert shapes, using either Draw commands or toolbar icons:



Draw commands     polygon     rectangle

polyline

path     circle
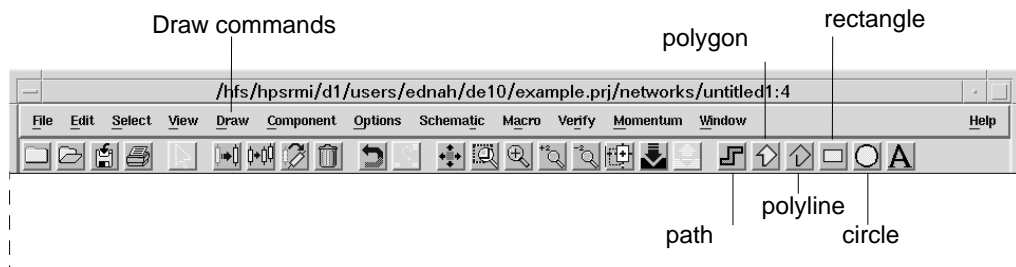
With each shape, you can either click and drag to place it, or define points by coordinate entry (choose the shape, then choose **Draw > Coordinate Entry**).

- **path**: starting point, segment end (click), and end point (double-click)
- **polygon**: starting point, vertex (click), and end point (double-click) that closes the shape
- **polyline**: starting point, segment end (click), and end point (double-click)
- **rectangle**: two diagonal corners
- **circle**: center and circumference point
- **arc**: (Draw command only) center and circumference point

Experiment by drawing different shapes to get the idea of how each is created.

## Layers

In a Layout window, items are placed on a layer. The name of the current insertion layer is displayed in the toolbar and in the status bar.

Current Insertion Layer



Current Insertion Layer
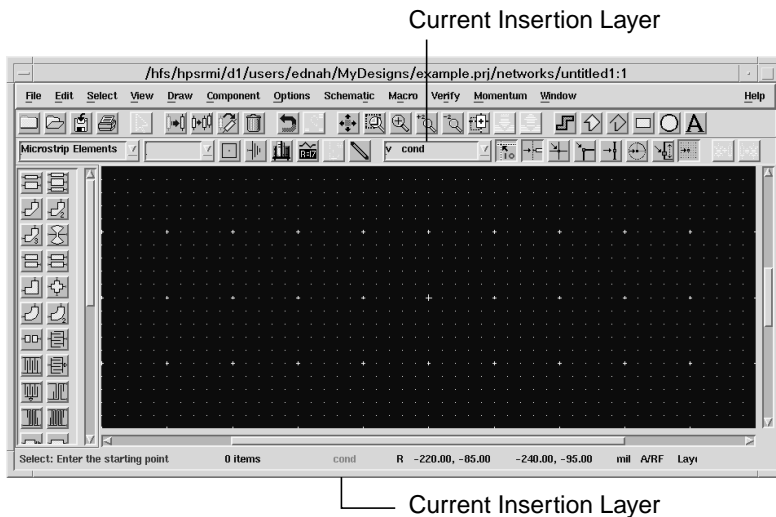
### Changing the Insertion Layer

There are *many* ways to change the insertion layer:

- In the toolbar, retype the name of the layer and press Enter.

- In the toolbar, click the arrow next to the layer name. Choose a name from the list of currently defined layers.

- Choose the command **Draw** > **Entry Layer** and select a layer from the list.

- Choose the command **Options** > **Layers** and select a layer from the list of defined layers in the Layer Editor dialog box.

- Choose the command **Draw > Change Entry Layer To**, and click an object whose layer you wish to make the current insertion layer.

Experiment with placing shapes on different layers. Remember to click **OK** to accept a change in a dialog box.

## Copy to a Different Layer

Experiment with copying shapes from one layer to another; use the command
**Edit > Copy/Paste > Copy to Layer** Note that the copied shape is placed at exactly the
same coordinates as the original. Move one to see them both.

## Default Layer Settings

Choose the command **Options** > **Layers** to display the Layer Editor. This is where you
can edit the parameters of any defined layer, add layers, or delete existing layers.
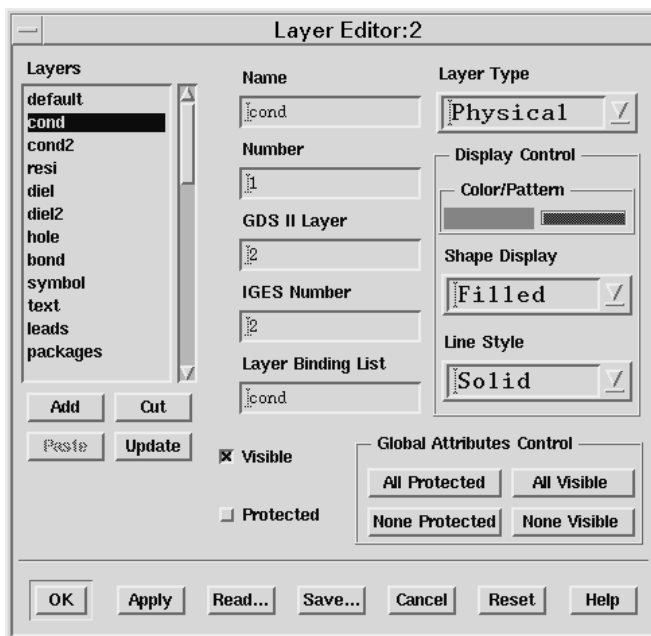


Clicking **Apply** updates layer definitions but does not dismiss the dialog box.

Experiment with layer parameters. Note that you can toggle the visibility of all items
on a layer. Protected means you can not select items on that layer.

## Other Layout Defaults

The **Options** > **Preferences** command displays the Preferences for Layout dialog box. This dialog box has 12 tabs; clicking a tab brings the corresponding panel to the front.
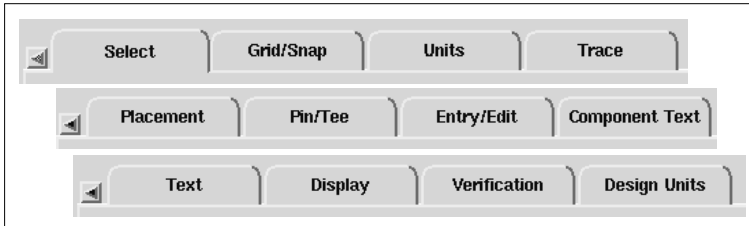


Click the Grid/Snap tab.
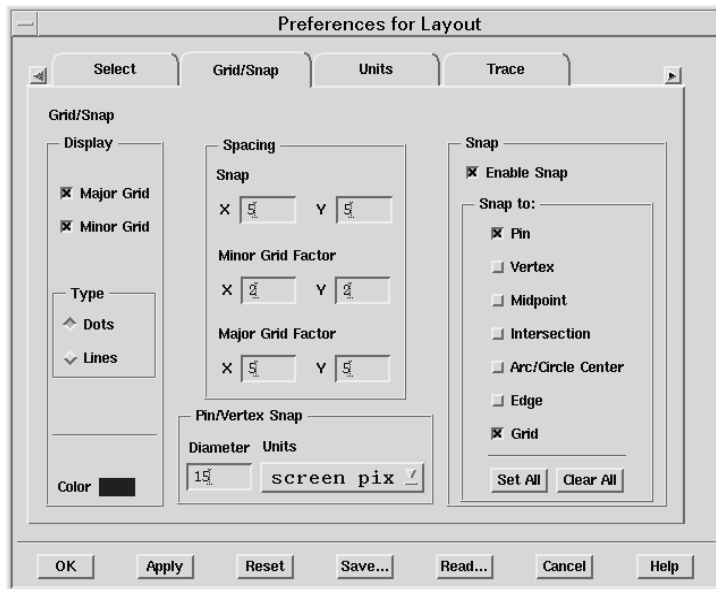


This panel is where you set the snap grid and display grid parameters.

The display grid appears on the screen as a series of vertical and horizontal lines or dots that you can use for aligning and spacing items in the drawing area.

**Adjust Grid Visibility and Color**

1. In the Display area, choose Major, Minor, or both.

2. Choose the Type of display (Dots or Lines). You may have to zoom in to see the grid display.

3. Click the colored rectangle next to the word Color, and choose the desired color for the grid. Click **OK** to dismiss the color palette.

4. Click **Apply**. Experiment with different settings

---

**Note**   The drawing area color is the *Background* color under the **Display** tab.

---

**Adjust Snap and Grid Spacing**

The ability to display a major grid as an increment of the minor grid enables you to gauge distances and align objects better in a layout.

1. In the Spacing area, change the Minor Grid display factors for both X and Y. The larger the number, the wider the grid spacing.

2. Click **Apply**. Experiment with different settings. If a display factor makes the grid too dense to display, it is invisible unless you zoom in.

3. Now experiment with the Major Grid.

**Adjust Pin/Vertex Snap Distance**

Pin/vertex snap distance represents how close the cursor must be to a pin of a component or a vertex of a shape before the cursor will snap to it.

A large value makes it easier to place an object on a snap point when you are unsure of the exact location of the snap point. A small value makes it easier to select a given snap point that has several other snap points very near it.

Place several components and several shapes in the drawing area and experiment with different settings of Pin/Vertex Snap.

**Screen pix** specifies sizes in terms of pixels on the screen. For example, if you choose 15, the diameter of the snap region is 15 pixels.

**User Units** specifies sizes in terms of the current units of the window. For example, if you are using inches and choose 0.1 user units, the diameter of the snap region is 0.1 inch.

**Experiment with Snap Modes**

Snap modes control where the program places objects on the page when you insert or move them; you can change snap modes when inserting or moving a component, or drawing a shape. When snap is enabled, items are pulled to the snap grid.

Experiment with different snap modes turned on or off to see how they affect the placement of items in a Layout window.

You can restrict or enhance the manner in which the cursor snaps by choosing any combination of snap modes. This table lists the snap modes, and their priorities.

| Snap Mode | Priority |
|---|---|
| Pin | 1 |
| Vertex | 2 |
| Midpoint | |
| Intersect | |
| Arc/Circle Center | |
| Edge | 3 |
| Grid | 4 |

**Angle Snapping** automatically occurs when *only* Pin snapping is enabled and you place a part so that the pin *at the cursor* connects to an existing part. The placed part rotates so that it properly aligns with the connected part.
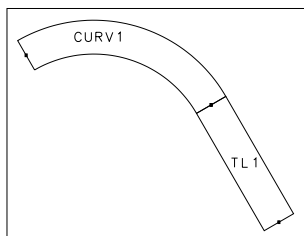


For example, if you have a microstrip curve at 30° and place a microstrip line so that it connects to it, the microstrip line will snap to 30° so that it properly abuts the curve.

**Enable Snap** toggles snap mode on and off. You can also toggle snap mode from the Options menu itself, and there are snap mode buttons on the toolbar.

Except for pin snap, the pointer defines the selected point on the inserted object.

When you set all snap modes OFF, you can insert objects *exactly* where you release them on the page. This is sometimes called raw snap mode. Like other snap modes, the raw snap mode also applies when you move or stretch objects.

**Pin**   When a *pin* on an object you insert, move, or stretch is within the snap distance of a pin on an existing object, the program inserts the object with its pin connected to the pin of the existing object. Pin snapping takes priority over all other snapping modes.

**Vertex**   When the selected location on an object you insert, move, or stretch is within the snap distance of a vertex on an existing object, the program inserts that object with its selected location on the vertex of the existing object.

In vertex snap mode, a vertex is a control point or boundary corner on a primitive, or an intersection of construction lines.

**Midpoint**   When the selected location on an object you insert, move, or stretch is within the snap distance of the midpoint of an existing object, the program inserts that object with its selected location on the midpoint of the existing object.

**Intersect**   When the selected location on an object you insert, move, or stretch is within the snap distance of the intersection of the edges of two existing objects, the program inserts that object with its selected location on the intersection of the existing objects.

**Arc/Circle Center**   When the selected location on an object you insert, move, or stretch is within the snap distance of the center of an existing arc or circle, the program inserts that object with its selected location on the midpoint of the existing arc or circle.

**Edge**   When the selected location on an object you insert, move, or stretch is within the snap distance of the edge of an existing object, the program inserts that object with its selected location on the edge of the existing object. Once a point snaps to an edge, it is captured by that edge, and will slide along the edge unless you move the pointer out of the snap distance.

Because edge snapping has priority 3, if the cursor comes within snap distance of anything with priority 1 or 2 while sliding along an edge, it will snap the selected location to the priority 1 or 2 item.

**Grid**   When the selected location on an object you insert, move, or stretch is within the snap distance of a grid point, the program inserts that object with its selected location on the grid point.

All other snap modes have priority over grid snap mode.

**Tip**  Whenever possible, keep grid snapping on. Once an object is *off* the grid, it is difficult to get it back on.

Use 45 or 90° angles to ensure that objects are aligned evenly, and to reduce the probability of small layout gaps due to round-off errors.

# Designing a Microstrip Line

This chapter is made up of an exercise that takes you through the process of creating a schematic, converting to a planar (Layout) format, preparing the layout for simulation, simulating, and generating analysis plots.

This exercise uses many default settings and a simple circuit (a microstrip line with step in width), and illustrates how quickly a design and analysis can be accomplished.

In this exercise, you will:

- Draw a simple microstrip line with step in width as a schematic, then generate a corresponding layout
- Create a simple substrate
- Define a mesh
- Perform a simulation
- Examine the results

Terms such as *substrates* and *meshes* may be unfamiliar, so they are explained in the course of the exercise.

This and later exercises assume that you have an introductory working knowledge of Advanced Design System, such as understanding the concept of projects, and being familiar with Schematic and Layout windows and placing components.

## Drawing the Circuit

The basic steps to making the microstrip line with step in width circuit include:

- Creating a new project

- Adding microstrip components to the schematic
- Converting the schematic to a layout

The schematic and layout representations are shown here. The sections that follow describe how to create both.
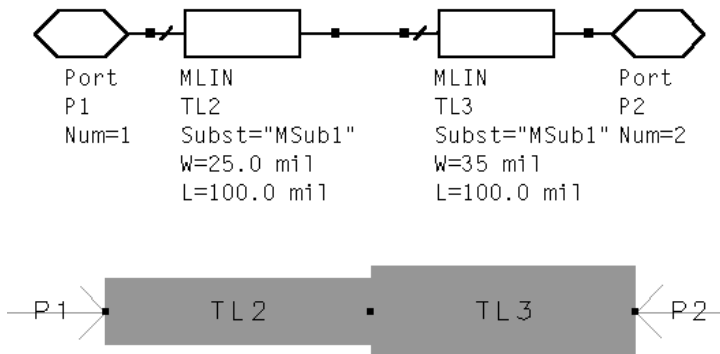


## Creating a New Project

You should start this exercise in a new project.

1. From the Main window, choose **Options > Preferences**. Ensure that **Create Initial Schematic Window** is enabled. Click **OK**.

2. From the Main window, choose **File** > **New Project**. The New Project dialog box appears.

3. In the *Name* field, type **step1**.

4. Click **OK**.

A Schematic window appears, which is where you will enter the design.

## Adding Microstrip Components to the Schematic

The steps in this section describe how to select a component. In the next section, it will be placed in the Schematic window.

Refer to the this figure to select the microstrip line component from the Microstrip Transmission Lines palette:



1 Click Palette List arrow once. The Palette List drops down.

2 Click TLines-Microstrip. The palette list closes and the Microstrip Component palette is displayed.

3 Click the MLIN button. This selects the Libra Microstrip Line component. Crosshairs and a ghost icon of the component appear as you move the cursor over the design window.
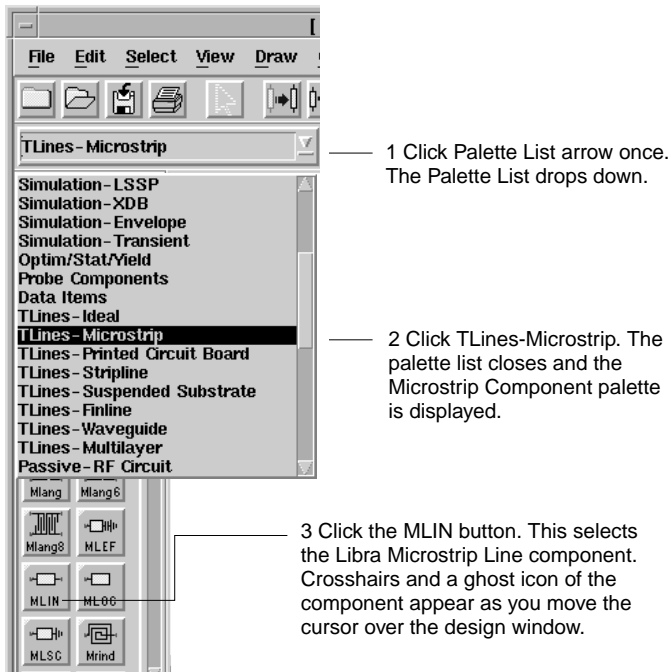
**Placing the Components**

The steps in this section describe how to place two microstrip lines in the Schematic window.

1. Move the crosshairs to the Schematic window and click once to place the component. A schematic representation of the component is placed in the Schematic window.

Click here to place a component
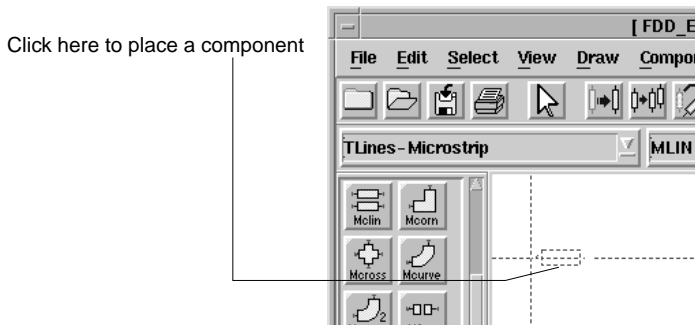


2. Move the cursor so that the crosshairs are directly over the right pin of the first component, and then click once to place a second component.



Position cursor here and click to add another component
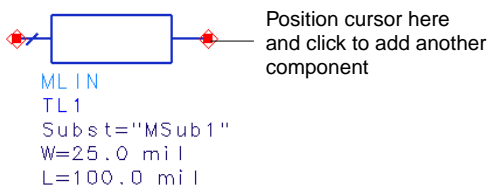
**Cancelling Commands**

If you continue to click without ending the current command, you will add another component with each click.

1. Click the arrow button or choose **Draw** > **End Command**. The crosshairs disappear.

2. You can also end a command by pressing the **Esc** key.

   You will use the end command frequently in this and other exercises, so be sure you are familiar with it.

**Editing Component Parameters**

Below the schematic representation of each component are some of the editable parameters of the component. This section describes how to change the width of one of the strips. The result is a microstrip step in width transmission line.

1. Click twice on the second component that you placed. The Libra Microstrip Line dialog box opens.
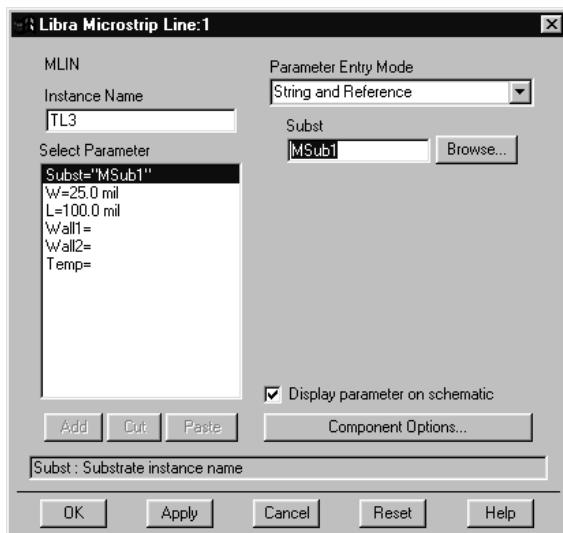


2. In the *Select Parameter* field, select the *W* (width) parameter. When the field to the right shows the value of the width, change the value in this field to 35 mil. Click **Apply.**

3. Click **OK** to dismiss the dialog box.

---

**Note**   If a parameter for a component is displayed in the Schematic window, you can also edit that parameter by clicking on the value and entering a new value.

---

4. Verify that the width of component on the left is set to 25 mil, and if needed, change the value of this parameter.

**Adding Ports to the Circuit**

To complete the circuit, you must add ports, one at the beginning of the microstrip step in width and one at the end. For Momentum, ports identify where energy enters and exits a circuit. This section describes how to add ports.

1. In the menu bar, click the Port button. Move the cursor over the Schematic window and note the orientation of the ghost icon of the port. The ports should be positioned as shown in the schematic below at the end of these steps.

2. You may need to rotate the port to the necessary orientation. If so, click the Rotate button and move the cursor back into the Schematic window. Note the rotation of the port outline, and repeat until it is correct.

3. Move the cursor over the open pin on the left side of the left component, then click.

4. The command to insert a port remains active. To insert a second port, change the orientation appropriately, move the cursor over the open pin on the right side of the right component, then click.

5. End the current command. Your schematic should now look like this figure. It is a microstrip line step in width. The width of the first part of the line is 25 mil, and it increases to a width of 35 mil. The overall length is 200 mil.

---

**Note** All of the components are connected. Diamond-shaped pins indicate that pins are not connected, and you will need to select and move components to make complete connections.

---



```
Port       MLIN                 MLIN                 Port
P1         TL2                  TL3                  P2
Num=1      Subst="MSub1"        Subst="MSub1" Num=2
           W=25.0 mil           W=35 mil
           L=100.0 mil          L=100.0 mil
```
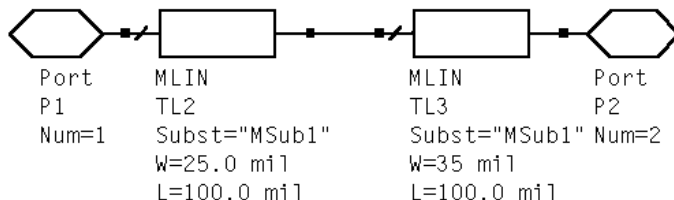
**Saving the Design**

It is good practice to save your work periodically. This section describes how to save the schematic.

1. Choose **File** > **Save**. When the Save dialog box appears, enter the name of the project, in this case, type **step1**.

2. Click **OK**.

## Generating the Layout

A powerful feature of Advanced Design System is the ability to convert a schematic to a layout automatically. Since Momentum requires a circuit be in Layout format, this gives you the option of drawing your circuits either as schematics or as layouts. Note that if you do choose to draw in a Schematic window, footprints of the components you use must also be available in Layout. Components that are available in Layout include transmission lines and lumped components with artwork.

This section describes how to covert the microstrip line step in width schematic that you just finished to a layout.

1. In the Schematic window, choose **Layout** > **Generate/Update Layout**. The Generate/Update Layout dialog box appears. It is not necessary to edit fields.

2. Click **OK**.

3. A Status of Layout Generation message appears indicating that the conversion is complete

4. Click **OK**.

5. A Layout window appears, showing a layout representation of the schematic. This window may be hidden by the Schematic window, so you may need to move some windows to locate the Layout window.



6. From the Layout window, choose **File** > **Save**. Name the layout **step1**. You now have a layout and a schematic as part of your project.

## Creating a Simple Substrate

A *substrate* is required as part of your planar circuit. The substrate describes the media where the circuit exists. An example of a substrate is the substrate of a multilayer circuit board, which consists of:

- Layers of metal traces
- Layers of insulating material between the traces
- Ground planes
- Vias that connect traces on different layers
- The air that surrounds the circuit board

Momentum includes predefined substrates that you can add to a layout, or you can make your own. Complete details about substrates, including where substrates are saved, are in Chapter 3, Substrates.

The steps in this section describe how to define a substrate.

For the microstrip line step in width example, a substrate with the following layers will be used:

- A ground plane
- A layer of insulation, such as Alumina
- A metal layer for the microstrip
- An air layer above the microstrip

1. From the Layout window, chose **Momentum** > **Substrate** > **Create/Modify**. The Create/Modify dialog box opens, showing the Substrate Layers options and parameters.
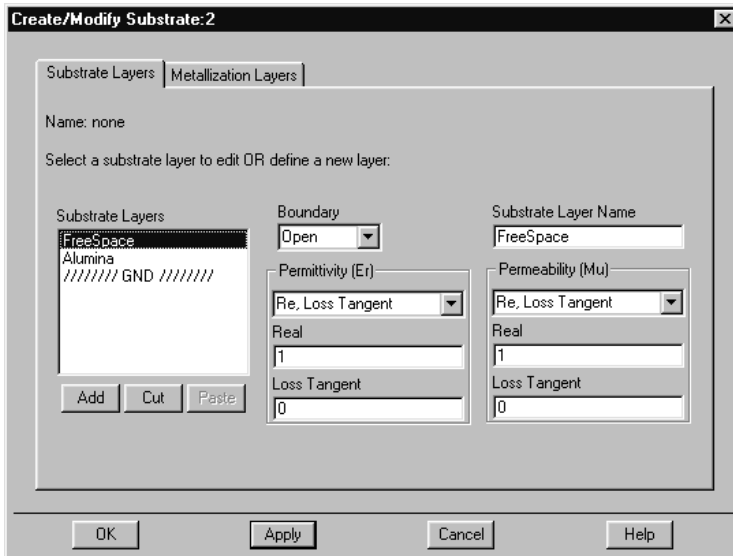


2. In the *Substrate Layers* field, select *FreeSpace*. Move to the *Substrate Layer Name* field and change it to read *Air*. Leave all other parameters at their default values and click **Apply**.

3. Keep the default layers *Alumina* and *///GND///*, but highlight and **Cut** any other layers that may be showing. Click **Apply**.

   You currently see three of the four substrate layers that you need. The fourth (metal) layer can be found by clicking the **Metallization Layers** tab.

The metal layer is automatically positioned between the layers of Alumina and air. The microstrip line is assumed to be on this layer.
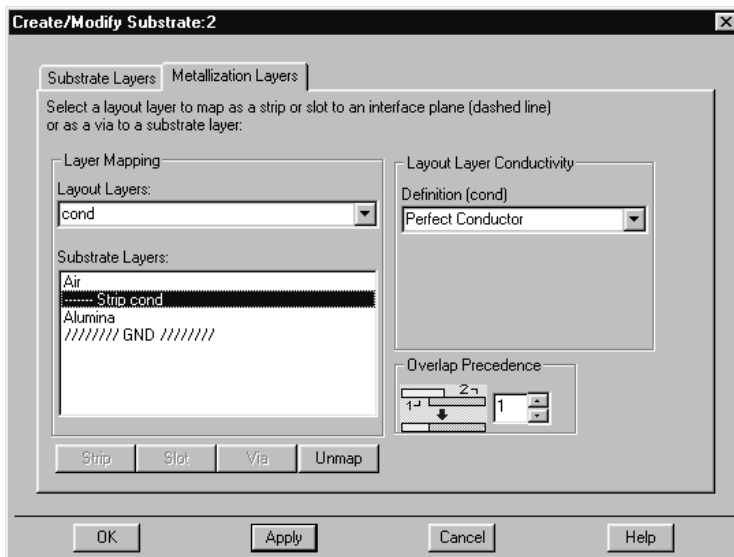


4. Click **OK** to dismiss the dialog box.

5. To save the substrate with the project, choose **Momentum** > **Substrate** > **Save As**. Type **step1** in the *Selection* field and click **OK**. The substrate *step1.slm* is saved in the project *networks* folder.

   Some other information about saving substrates:

   • A substrate definition is automatically saved with the design when **File > Save** is invoked from the Layout window

   • The command **Substrate > Save As** enables you to save the substrate definition in
     *slm* that can be saved anywhere and used with another design

   • When a design is opened, the substrate definition that was saved with the design is automatically loaded

## Precomputing the Substrate

In order to perform a simulation, Green's functions that characterize the behavior of the substrate must be computed. Substrates that are supplied with Momentum already include the computations. If you are:

- Reusing a substrate
- Have previously computed Green's functions for the substrate
- Created a substrate that is identical to one that already exists

the attempt to perform the computations will be ignored, since the calculations are saved in a database. In these cases, only if you extend the computation frequency range will additional calculations be performed.

The steps in this section describe how to precompute the substrate. Note that for this example, computations will not be performed. The substrate that was defined in the previous example is, in fact, identical to one that is supplied with Momentum.

1. Choose **Momentum** > **Substrate** > **Precompute Substrate Functions**. Set the Minimum Frequency to 1 GHz and Maximum Frequency to 4 GHz. This is the range that the simulation will cover. You should check that computations exist for the entire range that you intend to simulate, but if you do not, an error will be displayed if you need to run any computations.

2. Click **OK**.

   - If a dialog box opens to ask if you want to compute the substrate, click **Yes**. Calculations will be performed.

   - If a dialog box informs you that substrate calculations exist, click **OK**. The calculations will not be recomputed.

   A status window also opens. You can leave this status window on the screen as you proceed with this exercise.

This completes the process of defining a substrate.

## Setting up Mesh Parameters

A mesh is required in order to perform a simulation. The simplest method for generating a mesh is to skip the Mesh menu entirely and let Momentum automatically generate the mesh for your circuit, but you can choose to edit parameters that control how the mesh is generated.

A *mesh* is a grid-like pattern of triangles and rectangles that is applied the surfaces of a circuit. Using the mesh, the current within each triangular or rectangular area is calculated, and any coupling effects in the circuit can also be calculated during the simulation. From these calculations, S-parameters are then calculated for the circuit.

Complete details about mesh definitions and generation are in Chapter 6, Mesh.

The steps in this section describe how to set the frequency range for mesh computations. All other options will remain at the default values.

1. In the Layout window, select **Momentum** > **Mesh** > **Setup Controls**. A Create/Modify dialog box appears.
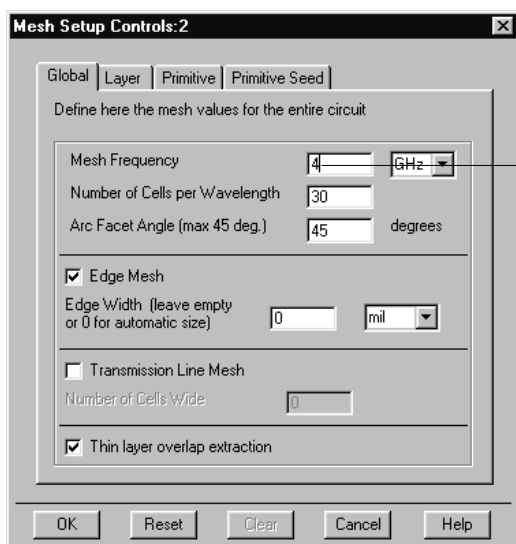


All that is required is the mesh frequency. Momentum uses defaults to generate a valid mesh for the circuit.

2. Set the value in the *Mesh Frequency* field to **4 GHz** and click **OK**.

## Precomputing the Mesh

If you want to view the mesh before simulation, you can precompute the mesh. The mesh will not be precomputed for this exercise.

This completes the process for generating a mesh.

## Performing the Simulation

The Momentum simulation process combines the Green's functions that were calculated for the substrate, plus the mesh information, and solves for currents in the circuit. Using the current calculations, S-parameters are then calculated for the circuit.

There are a variety of options to select to set up a simulation. Details are in Chapter 7, Simulation. This section describes the minimal steps for running a simulation.

1. From the Layout window, choose **Momentum** > **Simulation** > **Simulate Circuit**. The Simulation Control dialog box appears.

2. Set the Sweep Type to *Linear* and confirm that the following parameters are set:

   - Start = 1 GHz

   - Stop = 4 GHz

   - Frequency Step = 1 GHz

   By choosing this sweep type, a linear simulation will be performed over the 1-4 GHz frequency range, selecting the frequency points to be simulated based on the step size.

3. Click **Add to Frequency Plan List**. The dialog box should resemble the figure here.
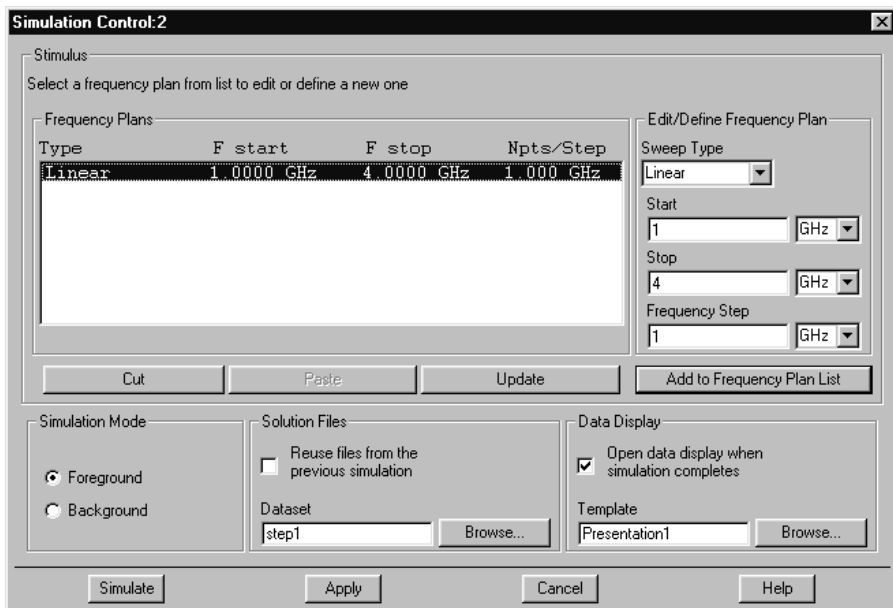


4. Ensure that the field *Open data display when simulation completes* is enabled.

5. Click **Simulate**. The simulation will be performed. The simulation progress and its completion will be indicated in the Simulation Status window.

6. When the simulation is complete:

   • The layout is overlaid with the mesh that was generated and used during the simulation process.
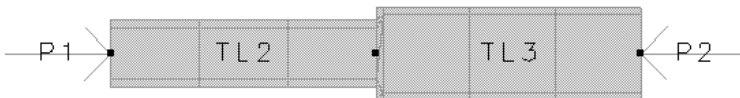


   • The S-parameter simulation results are automatically displayed on both rectangular plots and Smith charts in a Data Display window. This is because the field *Open data display when simulation is complete* was enabled as a default in the simulation setup. Choose **View > View All** or click the *View*

*All* button to view all of the plots, then zoom in on individual plots. An example of one S-parameter result is illustrated here.
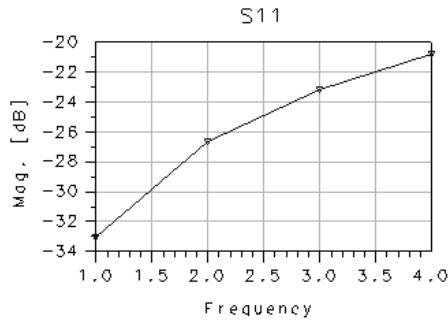


This completes the first Momentum design exercise.

# Designing a Microstrip Filter

This section is made up of an exercise that takes you through the process of designing a microstrip coupled-line filter. It is similar to the previous exercise, but uses fewer defaults and explores how to:

- Create a more complex substrate
- Specify port properties
- Use a box to simulate the filter within a metal enclosure
- Specify mesh parameters

## Drawing the Circuit

This section is very similar to the drawing instructions in the previous exercise. The circuit is drawn as a schematic, then converted to a layout. If you need more information on how to perform certain steps, refer to the previous exercise.

### Opening a New Project

Like the previous exercise, start the filter design in a new project.

1. From the Main window, choose **File** > **New Project**.

2. Name the project **filter.**

3. Click **Length Unit**. Select *millimeter* to set the layout units for this circuit. Click **OK** to set the units.

4. Click **OK** in the New Project dialog box to create the new project.

## Adding Components to the Schematic

The steps in this section describe how to select microstrip filter components.

1. In the Schematic window, click the Palette List arrow. The Palette List drops down.

2. Scroll and select **T-lines Microstrip** from the Palette List.

3. In the component palette, locate and click **Mcfil**. This selects the Libra Microstrip Coupled-Line Filter Section component.

4. Crosshairs and a ghost icon of the component appear as you move the cursor over the Schematic window. Position the cursor and click.

5. End the command either by clicking the **Cancel Command** button or by choosing **Draw > End Command**.

### Editing the Component

This section describes how to edit several component parameters, such as the length and width, and also how to change the parameters that are displayed below the component.

1. To edit the component parameters, double click the component.

2. In the dialog box that appears, select one of the parameters listed below, and edit the value. Also, enable *Display parameter on schematic* so that the parameter will be displayed below the component in the Schematic window. Select another parameter, and continue to set the values for these parameters:

   - W = line width = 0.25 mm

   - S = spacing between lines = 0.044 mm

   - L = line length = 1.8 mm

   - W1 = width of the line that connects to pin 1 = 0.25 mm

   - W2 = width of the line that connects to pin 2 = 0.25 mm

3. Click **OK** to accept the edits and dismiss the dialog box.

**Copying and Placing another Component**

This section describes how to make a copy of the filter section and add it to the schematic.

1. From the Schematic window, choose **Edit** > **Copy/Paste** > **Copy**.

2. Click the filter section component that is on the schematic. A black outline appears around it.

3. Move the crosshairs so that they are directly over the left connector of the first component and click.

4. End the current command.

**Adding Ports**

Ports are required on a circuit. If you forget to add ports to the schematic, you can always add them later after the schematic is converted to a layout. In this exercise, ports will be added to the layout.

With Momentum, you can define additional characteristics to a port. This also will be performed later in this exercise.

Your schematic should now look like the figure here. Be sure that all components are connected correctly.



```
MCFIL
CLin1
Subst="MSub1"
W=0.25 mm
S=0.044 mm        MCFIL
L=1.8 mm          CLin2
W1=0.25 mm        Subst="MSub1"
                  W=0.25 mm
                  S=0.044 mm
                  L=1.8 mm
                  W1=0.25 mm
```
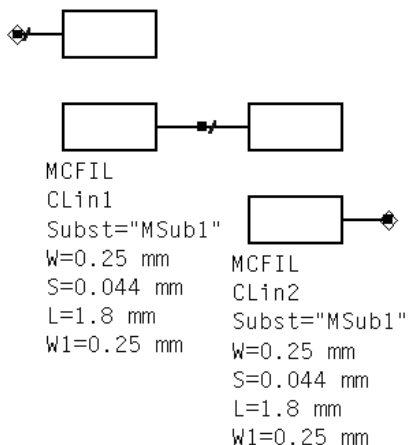
**Saving the Design**

It is good practice to save your work periodically.

1. Choose **File** > **Save**.

2. Enter **filter** as the file name.

3. Click **OK**.

# Generating the Layout

This section describes how to covert the microstrip coupled-line filter schematic that you just finished to a layout.

1. From the Schematic window, choose **Layout** > **Generate/Update Layout**. The Generate/Update Layout dialog box and a Layout window appear.

2. From the *Layout* window, choose **Options** > **Preferences**. Click the **Layout Units** tab and set the Resolution to 0.001. Click **OK**.

3. In the Generate/Update Layout dialog box, enter **-1.8** in the X field and **0.294** in the Y field. This step identifies the placement of P1 on the layout. This step is usually optional, but specifying the location will be helpful later when the box is added to enclose the filter. Click **OK**.

4. A window displaying the results of the conversion is displayed. Click **OK**.

A layout representation of the schematic appears in the Layout window. It should resemble the figure here.
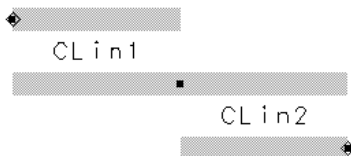


5. From the layout window, choose **File** > **Save**. Name the layout **filter**. You now have a layout and a schematic as part of your project.

## Defining a Substrate

This circuit uses a relatively simple substrate. In this exercise, the thickness of some of the layers is modified. A more important task is that ground planes are added to the top and bottom of the substrate. These ground planes form the top and bottom of the metal box in which the filter will be enclosed. The substrate will have the following layers:

- A ground plane representing the top of the box
- A layer of air
- The microstrip filter traces
- A layer of alumina
- A ground plane representing the bottom of the box.

1. From the Layout window, chose **Momentum** > **Substrate** > **Create/Modify.**
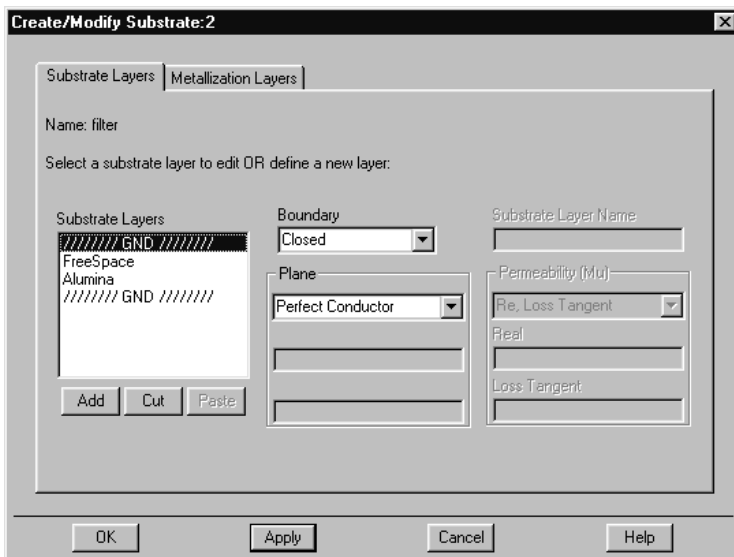


2. Select the top layer in the Substrate Layers field. In the Boundary list, select *Closed*. The top ground plane is added to the substrate.

3. In the Substrate Layers list, select *Free_Space*. Change the Thickness to **2846 um**. This sets the layer of air to an appropriate thickness.

4. In the Substrate Layers list, select *Alumina*. Change the Thickness to **254 um**.

5. Keep the bottom ground plane, but select and **Cut** any other layers that may appear in the substrate.

6. Click **Apply.**

---

**Note**   The top and bottom of the box enclosure are defined with groundplanes, the sides of the enclosure will be defined later in this exercise.

---

You currently see four of the five substrate layers that you need. The metal layer for the microstrip filter can be found by clicking the **Metallization Layers** tab.
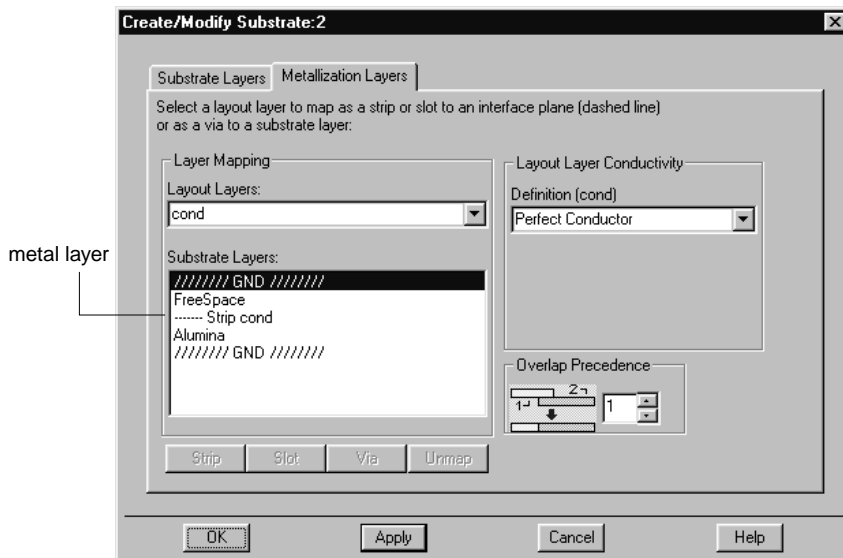


In this instance, the metal layer is automatically positioned between the Alumina and air.

Note the following points about this layer:

• Metal layers are identified by a dashed line.

- The word *cond* identifies the layout layer that is mapped to this position. Refer to the Layout window, and you will see that the microstrip circuit was automatically applied to the layout layer named *cond* during the translation process.

- The word *strip* defines the layer such that the microstrips are metal and what surrounds the microstrips on that layer is air or dielectric. Other choices are slot and via. These are described in the exercise in the next chapter.

7. Choose **Momentum** > **Substrate** > **Save As**. Use the file name **filter** and click **OK**. The substrate file *filter.slm* is saved as part of the project.

8. Choose **Momentum** > **Substrate** > **Precompute Substrate Functions**. Set Minimum Frequency to **1 GHz** and Maximum Frequency to **40 GHz**. Click **OK**. If a dialog box appears and asks if you want to compute the substrate, click **Yes**.

There are instances when the computations will not be performed. This was discussed in the previous exercise. If they are performed, a window opens that indicates the status of the computations and when they are complete.

You can choose **Momentum** > **Substrate** > **View Substrate Summary** at any time and see a summary of the substrate calculations that have been completed.

## Adding Ports to the Layout

In Momentum, you can define the ports in your circuit to be one of several types. Depending on the type you choose, the ports will be characterized in different ways. This can impact the result of a simulation, because the simulation can take these characteristics into account. With different port types, you can select one that best matches the intended application of your layout.

More detailed information about the various port types can be found in Chapter 4, Ports.

Ports are defined in a two-step process. First, ports are added to a circuit when the circuit is drawn. Then, in Momentum, you specify the port type in order to tailor the port to your circuit. Note that you can add the port components at any time as you draw the circuit, but before a port type can be specified, a substrate must be defined for the circuit.

This section describes how to add ports to a layout and how to specify a port type for them.

1. From the Layout window, click the port icon. A Port dialog box appears. If necessary, set **layer** = **cond**. The ports must be on the same layout layer as the microstrip filter. Click **OK**.

2. Move the cursor over the open port on the left side of the layout and click. Move the cursor over the open port on the right side of the layout and click. End the command. The layout should resemble the figure here.
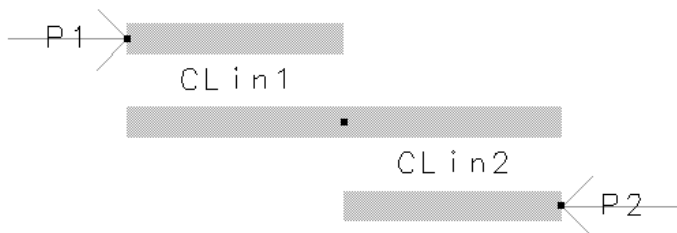


3. From the Layout window, choose **Momentum** > **Port Editor**. The Port Properties Editor dialog box.

4. If necessary, drag the dialog box away from the Layout window so that both ports are visible. Click the connector P1.

5. The Port Properties dialog box will change so that you can select a port type to be applied to P1.



Port Properties Editor:2

Port 1 selected on strip layer cond .

Port Type
[Internal ▼]

Polarity
○ Normal    ○ Reversed

Impedance
Real
[           ] [Ohm ▼]

Imaginary
[           ] [Ohm ▼]

Reference Offset (+ = inward)
[           ] [mm ▼]

Associate with port number
[           ]

[ OK ] [ Apply ] [ Cancel ] [ Help ]
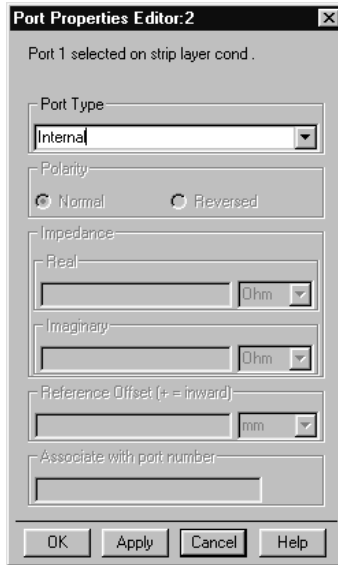
6. Select *Internal* from the Port Type drop down list and click **Apply**.

7. Click the connector P2. Select *Internal* from the Port Type drop down list. Click **OK** to accept the port type and dismiss the dialog box.

The internal port type is selected because it can be applied to the interior of a circuit. While normally this placement of the ports on the filter would not be considered internal, they are in this case because the filter is enclosed within the box. More information on this and other port types is in Chapter 4, Ports.

## Finishing the Box around the Filter

Boxes and waveguide enclosures are useful because they introduce horizontal boundaries to the design. Adding a box enables you to look at box resonance, which can have a significant effect on S-parameters in a small band centered around the box resonance frequency. More information about boxes, waveguides, and their uses are in Chapter 5, Boxes and Waveguides.

Recall that when you defined the substrate, you defined ground planes as the top and bottom of the metal box that will enclose the filter. This section describes how to specify the dimensions and location of the sidewalls of the box.

1. From the Layout window, choose **Momentum** > **Box & Waveguide** > **Add Box**.

2. You will enter specific x, y coordinates for the left and right sides of the box. Choose **Draw** > **Coordinate Entry**. Type **-4.5**, **-4.5** in the X,Y fields of the Coordinate Entry dialog box and click **Apply**.

3. Type **+4.5**, **+4.5** in the X,Y fields of the Coordinate Entry dialog box and click **Apply**. The box appears as shown below (you may need to click **View All** to see the box). Click **Cancel** to close the dialog box.
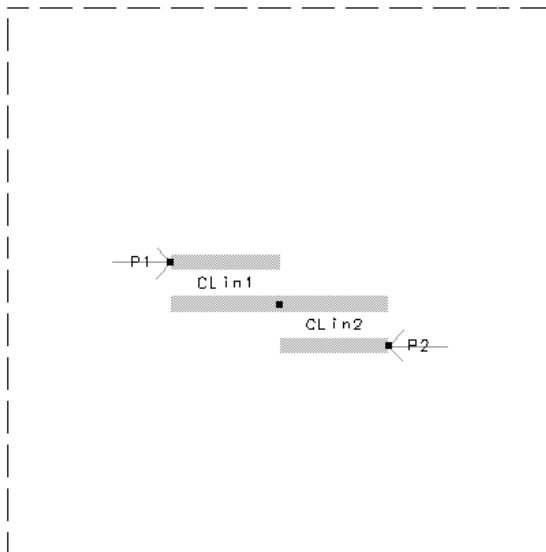


## Generating a Mesh

This section describes how to specify additional mesh parameters.

1. From the Layout window, choose **Momentum** > **Mesh** > **Setup Controls**.

2. Set the following options and values:

   • Mesh Frequency = 40.0 GHz. You should set the mesh frequency to the highest frequency to be simulated.

- Number of Cells per Wavelength = 20. This will be used to determine the density of the mesh. A mesh needs to be made up of small cells, but a mesh that is too dense over the entire circuit slows a simulation without adding much accuracy to results.

- Edge Mesh is enabled, and Edge Width = 0.05 mm. The edge mesh adds a relatively dense mesh along the edges of objects. Since most current flows along the edges of objects, using a denser mesh only on the edges can improve the speed and accuracy of a simulation.

Leave the other settings as they appear.

3. When these selections are completed, click **OK**.

4. Choose **Momentum** > **Mesh** > **Precompute Mesh**. The mesh will be computed and displayed on the layout, like the figure here. If you have trouble seeing the mesh, choose **Options** > **Layers** and reset the color and pattern values for the **cond** layer.
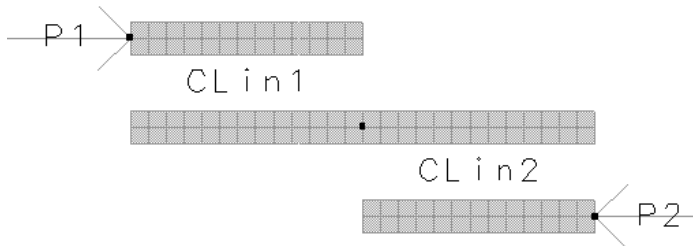


5. Save your work by choosing **File > Save** from both the Layout window and the Schematic window. Use the file name *filter* for both of them; Advanced Design System can distinguish the layout from the schematic and will not overwrite one with the other.

## Performing the Simulation

This section describes how to set up and run the simulation. In this exercise, the adaptive sweep type is used in the simulation process. This the preferred sweep type, because it used a fast, highly accurate method of comparing simulated data points to a rational fitting model. Unlike the linear sweep type, where simulated data points are chosen in a linear fashion based on step size, adaptive sweep type data points are

chosen based on where the most variance seen. Wherever S-parameters vary from the rational fitting model the most, more samples are taken.

1. From the Layout window, choose **Momentum** > **Simulation** > **Simulate Circuit**.

2. Select the **Adaptive** Sweep Type and set these parameters to the following values:

    • Start = 1 GHz

    • Stop = 40 GHz

    • Sample Points Limit = 50

3. Click **Add to Frequency Plan List**.

4. Make sure that *Open data display when simulation completes* is enabled The S-parameters will automatically be plotted and displayed at the end of the simulation.

5. Click **Simulate**. The simulation will be performed and its progress and completion will be indicated in the status window.

6. Click **Momentum** > **Simulation** > **View Summary** and saved simulation details will be displayed. You can print the report if you wish.

## Viewing Simulation Results

The S-parameter simulation results are automatically displayed on both rectangular plots and Smith charts in a Data Display window. An example of S12 plotted on a rectangular plot is shown here.
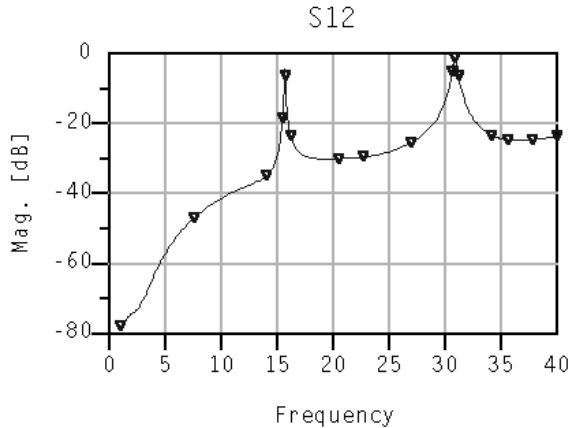


# Designing a Coplanar Waveguide Bend

This exercise, like the previous exercises, takes you through all of the steps that are necessary when designing with Momentum: creating a layout, defining a substrate, adding ports to the circuit, defining a mesh, simulating, and viewing results. Unlike the previous exercises, this one explores the following design topics:

- Creating a layout that is drawn on more than one layer
- Using vias as connections between layers
- Specifying new port types
- Working with new mesh parameters

This circuit drawn in this exercise is a coplanar waveguide bend. A coplanar waveguide is, generally, slots filled with a dielectric or air, surrounded by metal, as shown in the figure here.
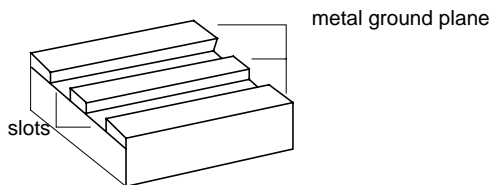


There are two basic methods for constructing coplanar waveguides in Layout, one is used in this exercise, the other is discussed in Chapter 5, Boxes and Waveguides.

## Creating the Layout

In this example, the circuit is created entirely within the Layout window, and a schematic representation is not used. Be sure to save your work periodically.

The layout consists:

- Coplanar waveguide slots
- Bridges that span the two slots
- Vias that vertically connect the bridges to the slots

The slots will be drawn on one layer, the bridges will be drawn on a second layer, and the vias on a third layer. This will be important for when these layers are mapped to the substrate. An illustration of the circuit is shown on the left, the layout is on the right.
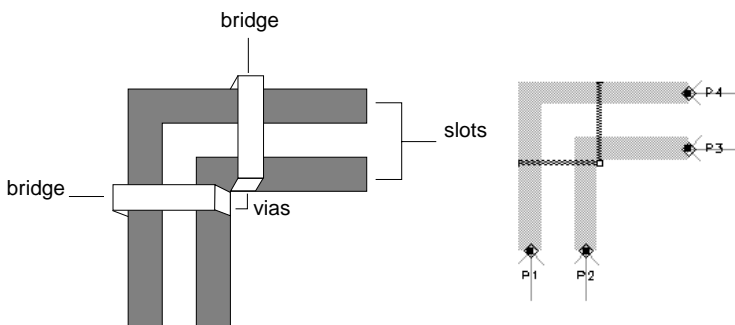
1. From the Main window, choose **File** > **New Project**. Name the project **cpw_bend**. Click **Length Unit**, and set the units to **millimeter**. Click **OK**.

2. Set your preferences so that a Layout window will open instead of a Schematic window. From the Main window, choose **Options > Preferences** and enable *Create Initial Layout Window*. Disable *Create Initial Schematic Window*.

3. Click **OK** in the open dialog boxes**.**

4. From the Layout window, choose **Options** > **Preferences**. Scroll to the **Layout Units** tab. Set the layout units to **um** and the resolution to 0.001. Click **OK**.

5. There is a difference between the length units that you chose when you created the new project, and the units that you just set. The units chosen when you created the project set the units of the layout window in which you are working. The units chosen through *Options > Preferences* sets the units for objects that are drawn in the Layout window.

6. You will rename the layout layers that are used for drawing the slots, bridges, and vias of the coplanar waveguide bend. Using more descriptive names can help identify which layer that each part of the circuit is drawn on. Choose **Options > Layers**. From the *Layers* list, select **cond**. In the *Name* and *Layer Binding List* fields, enter **slot**. Click **Update** (located just below the Layers list). Select **cond2**. In the *Name* and *Layer Binding List* fields, enter **bridge**. Click **Update**. Select **hole**. In the *Name* field only, enter **via** and click **Update**. Click **OK**.

## Drawing the Waveguide Slots

The section describes how to draw the coplanar waveguide slots.

1. Verify that *slot* is the current layer. The slots will be drawn on this layer. The name of the current layer is displayed in the tool bar and at the bottom of the Layout window. If slot is not displayed, select it from the list.



select a layer

current layer and its color
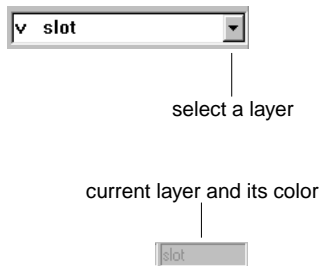
2. Choose **Draw** > **Coordinate Entry,** then choose **Draw** > **Rectangle**. The toolbar arrow is activated and the Coordinate Entry dialog box opens.

3. You will define four rectangles to make two waveguide bends. Establish the following values in the X,Y coordinate window, and click **Apply** after each entry:

| X, Y coordinates | Define |
| --- | --- |
| **0, 0** | First corner of first rectangle |
| **+45, +290** | Second corner of first rectangle |
| **0, +335** | First corner of second rectangle |
| **+335, +290** | Second corner of second rectangle |
| **+110, 0** | First corner of third rectangle |
| **+155, +180** | Second corner of third rectangle |
| **+110, +180** | First corner of fourth rectangle |
| **+335, +225** | Second corner of fourth rectangle |

4. Click **View All** to see your drawing.

5. End the command. The drawing in the layout window should resemble the figure here.



## Drawing the Bridges

The slots that make up the coplanar waveguide have been drawn on one layer. On a different layer, you will draw the bridges that span the two slots. This is important, because the slots and bridges will be treated differently in Momentum. This will become more clear when the substrate is applied to the circuit, later in this exercise.

1. From the layer list on the toolbar, select the layer **bridge** and click **OK**.

---

**Note**   The **slot** layer is still visible. You will not do so here, but it can be made invisible if you go to the Layout window and choose **Options** > **Layers**, select the slot layer, and turn Visible off.

---

2. Choose **Draw** > **Rectangle**.

3. You will define two rectangles that make the bridges. Establish the following values in the X,Y coordinate window, and click **Apply** after each entry:

| X, Y coordinates | Define |
| --- | --- |
| **0, +170** | First corner of first bridge |
| **+155, +180** | Second corner of first bridge |
| **+155, +180** | First corner of second bridge |
| **+165, +335** | Second corner of second bridge |

4. End the current command. The drawing in the Layout window should now look like the figure here.
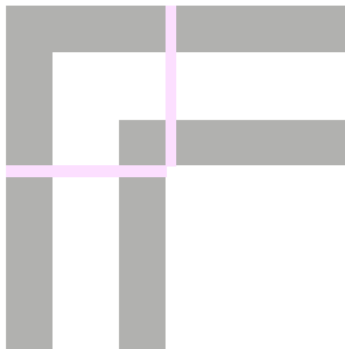


## Drawing a Via

The vias must be drawn on a third layer. This is necessary for when the vias are mapped to the substrate, later in this exercise.

The position and size of a via is very important. The various snap modes in Layout can aid in drawing vias, as shown in this part of the exercise. More information on how to draw vias and other drawing tips are in Chapter B, Drawing Tips.

1. From the layout layer list on the toolbar, select **via.**

2. Choose **Options** > **Preferences**. Scroll to and select **Grid/Snap**. Under Active Snap Modes, enable **Vertex** and clear **Grid**. Click **OK**.

   By using the vertex snap mode, the vias that you draw will snap to the vertex of a nearby object (a bridge) and not to a grid point. This ensures that the vias will be positioned precisely on the edges of the rectangles.

3. Choose **Draw** > **Polyline**. Referring to the points identified on the drawing below, create the line segments shown by performing these steps:

   • Click once on A1 and click twice on A2.

   • Click once on B1 and click twice on B2.

   • Click once on C1, click once on C2, and click twice on C3. Then click the toolbar arrow. The drawing in the Layout window should now appear as shown below.
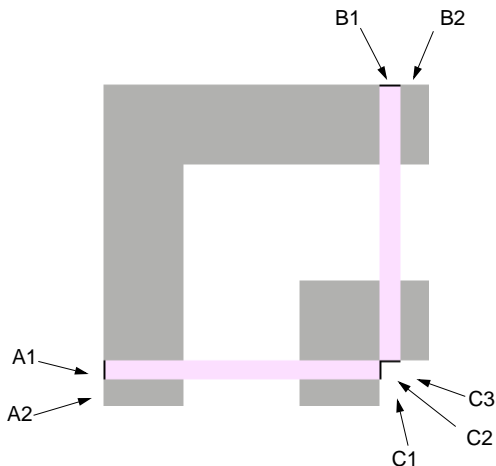
## Opening a Substrate

This exercise uses a substrate that is supplied with Momentum. It will not be necessary to make any changes to it, but it will be reviewed to clarify how layout layers can be mapped to a substrate.

1. From the Layout window, choose **Momentum** > **Substrate** > **Open**. To the question, *Open a supplied substrate?,* click **Yes**.

2. Locate and select the substrate file *cpw_bend_example.slm*, then click **OK**. You are notified that the substrate has been opened. Click **OK**.

3. Choose **Momentum** > **Substrate** > **Create/Modify.** Select the **Metallization Layers** tab.
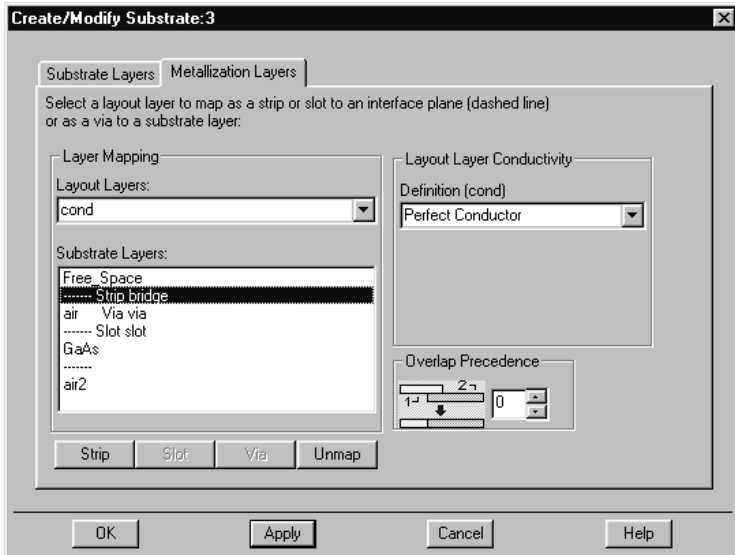


The metallization layers are identified by dashed lines, except in the case of vias. For the used metallization layers, you see the type of layer (Strip, Via, or Slot) and the name of the layout layer that is mapped to it (bridge, via, and slot). You can see how the metallization/layout layers are mapped in between the Free_Space, air, and GaAs layers:

• There is a small layer of air between the bridges and the slots.

- The vias are mapped the directly to the air substrate layer. The vias, although they are drawn as lines, are extruded by Momentum and effectively cut vertically through this layer of air, connecting the edges of the slots and the bridges.

- All three types of metallization layers, strip, slot, and via, are used here. A *strip* defines the layer such that the bridges are metal and what surrounds the microstrips on that layer is air. A *slot* is the opposite of a strip: the layout layer would be treated as a ground plane, and the bends that you have drawn are treated as open areas in the ground plane. *Vias* represent vertical sections that cut through other layers vertically.

## Adding Ports

This section describes how to add ports to the layout and how to apply the coplanar port type to them.

Coplanar is one of the several port types that are available in Momentum. It is designed specifically for coplanar waveguide circuits, where an electric field is likely to build up between two ports. Each of the two ports is excited with the same absolute electrical field, but with opposite polarity. The currents are equal but in opposite in direction. For more information, refer to Chapter 4, Ports.

1. The ports must be added to the waveguide slots and so they must be drawn on the same layer as the slots. From the Layout Layers list box, select **slot**.

2. The ports must also be positioned in the center of each slot edge. To help position the ports, set the following snap modes:

   - Enable **Options** > **Midpoint Snap**

   - Enable **Options > Edge Snap**

   - Disable **Options > Vertex Snap** and other enabled snap modes (but make sure that **Options > Enable Snap** remains enabled)

3. Click the Port icon. The Port dialog box appears.

4. Verify that **layer=cond** is displayed in the *Select Parameter* field. If it is not, set **layer** to **cond**. Verify that **Num=1**. If it is not, select **Num** and set it to **1**. Click **OK**.

5. Note the ghost image of the port. If it appears very large, or if it cannot be seen, cancel the command. Depending on your system defaults, you may need to change the value of the Port/Ground Size (**Options** > **Preferences** > **Placement**)

and the value of the Font Definition Size (**Options** > **Preferences** > **Component Text)** before your ports and labels will be an appropriate size for this layout.

6. When you are ready to add ports, apply them in the order and direction as shown here, then end the port command.
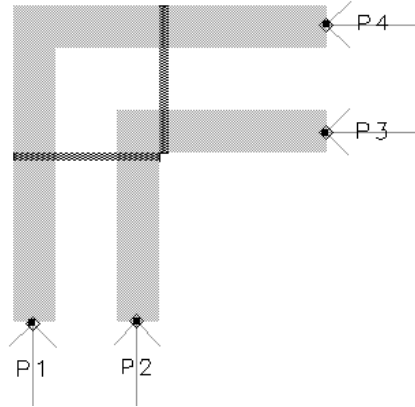


7. From the Layout window, chose **Momentum** > **Port Editor**. The Port Properties Editor dialog box appears. You may need to drag the box away from the layout in order to have access to the ports on the layout.

8. In the Layout window, click P1. In the Port Editor, select **Coplanar** from the Port Type list. Set the Polarity to **Normal**. Do not change any other values. Click **Apply**.

9. Click P2. Select **Coplanar** and set the Polarity to **Reversed**. Type **1** in the field *Associate with port number*. Click **Apply**.

10. Click P3. Select **Coplanar** and set Polarity to **Reversed**. Click **Apply**.

11. Click P4. Select **Coplanar** and set Polarity to **Normal**. Type **3** in the field *Associate with port number*. Click **Apply**.

12. Click **OK**.

## Generating the Mesh

This section describes how to use additional mesh parameters. Because this circuit is on multiple layers, it is possible to set mesh parameters for each layer. Thus, you can set a relatively dense mesh on a specific layer. In this example, you will apply an edge mesh to the slot and via layers, but you will not use an edge mesh on the bridge layer.

1. From the Layout window, choose **Momentum** > **Mesh** > **Setup Controls**.

2. Set the following Global parameters:

   • Mesh Frequency = 40 GHz

   • Number of Cells per Wavelength = 100

   • Enable Edge Mesh

   Global parameters will be applied to all layers, unless other values are specified from the **Layer** tab.

3. Select the **Layer** tab.

4. In the Layout Layers list, select **bridge**. Set the Number of Cells per Wavelength to **100** and disable Edge Mesh. These mesh parameters will be applied to the objects drawn on the bridge layer only.

5. Click **OK**.

6. Select **Momentum** > **Mesh** > **Precompute Mesh**. Click **OK** to start the computations. The mesh will be computed and displayed on the layout, as shown here. Zoom in for a closer look at the mesh.
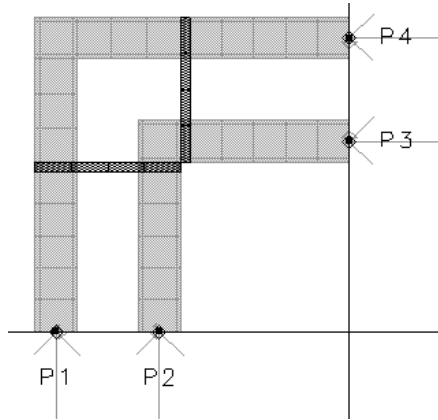


7. Zoom in on the ports. You can see that additional arrows have been added to each port. The new arrow indicate the direction of the voltage over the slot.

## Performing a Simulation

1. From the Layout window, choose **Momentum** > **Simulation** > **Simulate Circuit**.

2. From the Sweep Type list, select **Adaptive** and set these parameters:

   • Start = 1 GHz

   • Stop = 40 GHz

   • Sample Points Limit = 50

3. Click **Add to Frequency Plan List**.

4. Make sure that *Open data display when simulation completes* is enabled.

5. Click **Simulate**.

# Viewing Simulation Results

The figure shows some of the automatically plotted simulation results.

S11

S13
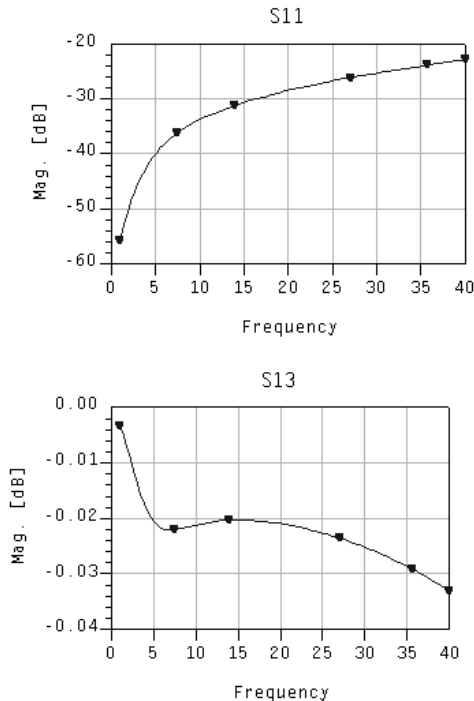
To save your work, choose **Yes to All** when you are asked to save your work when you close the project.

This concludes the third exercise.

# Chapter 3: Substrates

A *substrate definition* describes the media where a circuit exists. An example is the substrate of a multilayer circuit board which consists of layers of metal traces, insulating material, ground planes, vias that connect traces, and the air that surrounds the board. A substrate definition enables you to specify properties such as the number of layers in the substrate, the dielectric constant, and the height of each layer for your circuit.

A substrate definition is made up of *substrate layers* and *metallization layers*. Substrate layers define the dielectric media, ground planes, covers, air or other layered material. Metallization layers are the conductive layers in between the substrate layers, and they are used in conjunction with the layout layers. By mapping layout layers to metallization layers, you can position the layout layers that your circuit is drawn on within the substrate. An example of a substrate is shown below. It contains four substrate layers and two microstrips with a via.
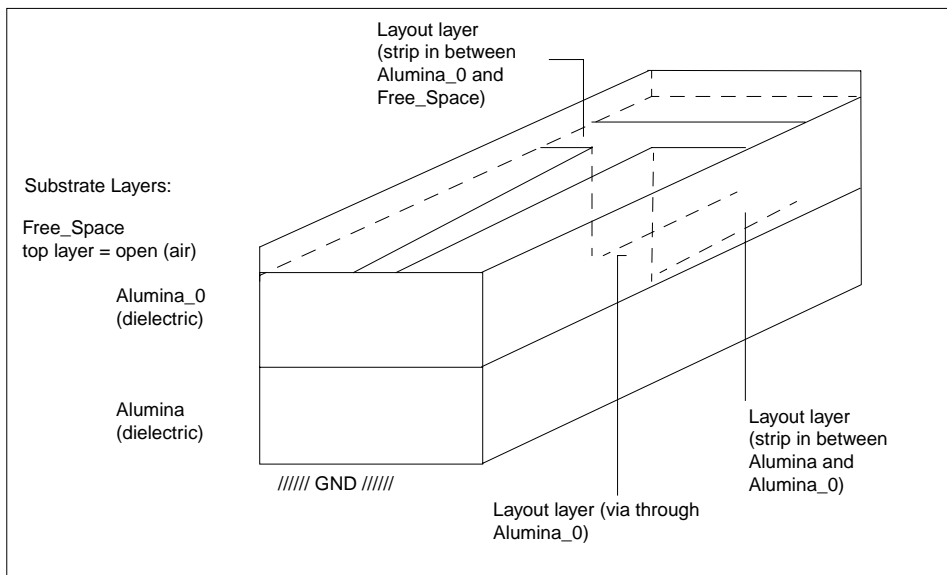


Substrate definitions can be saved and used with other circuits. A variety of predefined substrates are included with Advanced Design System, which can be used "as is" or modified to your design specifications.

The steps for defining a substrate include:

- Defining the substrate layers
- Mapping the layout layers to metallization layers
- Specifying metallization layer conductivity
- Solving the substrate

Details on how to work with existing substrate definitions and create new ones follow.

# Selecting a Predefined Substrate

Momentum includes a number of predefined substrate definitions, so it may not be necessary to create a substrate from scratch. A predefined substrate may be suitable so that it can be used it as is or modified to suit your design. Substrate files end with the extension *.slm*.

To open a predefined substrate:

1. In the layout window, select **Momentum** > **Substrate** > **Open.**

2. If you want to open a supplied substrate, click **Yes**. A list of supplied substrate files appears.

   If you want to open a substrate that has been saved as part of the project, click **No.** A list of substrate files that are saved as part of the project appear.

   For more information on locating substrate files, refer to "Identifying Where Substrates are Saved" on page 3-4.

3. Select the substrate file, then click **OK**.

To edit the substrate, refer to "Defining Substrate Layers" on page 3-5.

# Reading a Substrate Definition from a Schematic

Choose *Momentum > Substrate > Update From Schematic* to update a substrate under the following conditions:

- You have used a substrate component on a schematic, such as MSUB or SSUB

- You have already generated a layout from the schematic using *Layout > Generate/Update Layout*

- You then change the substrate definition on the schematic, and need to transfer it to the layout

In order to retrieve the new substrate information, you must use *Momentum > Substrate > Update From Schematic*. Using *Layout > Generate/Update Layout* will not transfer the new substrate information to the layout.

To update a substrate definition from a schematic:

1. Choose **Momentum > Substrate > Update From Schematic**. The substrate on the correct schematic is read and the new information is entered in the Momentum substrate definition.

# Saving a Substrate

A substrate definition can be saved to a file. This enables you to store the definition outside of the current project, making it easier to use the substrate definition in other designs. To save a substrate:

1. Choose **Momentum > Substrate > Save As**.

2. Type a name for the file and click **OK**. Substrate files end with the extension *.slm*.

To save a named file, choose **Momentum > Substrate > Save**. The substrate information is saved in a file called <substrate_name>.*slm* under */networks* in the current project directory.

### Identifying Where Substrates are Saved

Substrates are saved in one of two locations:

- In the directory containing supplied substrates, which is *$HPEESOF_DIR/momentum/lib/substrates*. You should not use this location for substrates that you have created or modified.

- The location that the environment variable $HOME points to. If this variable is not defined, substrates are saved in the current project folder.

# Creating/Modifying a Substrate

To create or modify (edit) a substrate definition, use the procedures in "Defining Substrate Layers" on page 3-5 and "Defining Metallization Layers" on page 3-12.

# Changing Substrates

If you no longer want to use the substrate definition assigned to your design, or to choose a different substrate use the procedure in "Selecting a Predefined Substrate" on page 3-2.

# Deleting a Substrate

This command erases the substrate file that you select. If the substrate has been precomputed, the calculations remain in the database.

To delete a file:

1. Choose **Momentum > Substrate > Delete**.

2. Select the name of the file that you want to delete.

3. Click **OK**.

# Defining Substrate Layers

You can define a substrate from scratch or edit an existing one. A substrate must have, at the minimum, a top plane and a bottom plane.

---

**Note**   Do not use substrate components that might appear on layout component palettes. You must use the dialog box under *Momentum > Substrate > Create/Modify* to create or edit substrate definitions.

---

To define substrate layers:

1. Choose **Momentum > Substrate > Create/Modify**.

2. If this is a new definition, three default layers appear in the Substrate Layers Field:

   - Free_space (top plane)

   - Alumina (dielectric)

   - GND (bottom plane)

   These layers represent a basic substrate definition that includes the three types of substrate layers in Momentum.

   - *Free_Space* represents the top plane of the substrate. In this definition, it is defined as an *open boundary*.

   - *Alumina* represents a dielectric layer of finite thickness. These layers are also referred to as *interface layers*.

   - *GND* represents the bottom plane of the substrate. GND defines a *closed boundary*. Closed boundaries define ground planes or other closed boundaries, such as the lid or bottom of an enclosure.

   For an existing substrate definition, you may see more layers with different names, but they will all be one of these three basic types of layer, and the substrate will have a top and bottom plane. A substrate definition must have a top plane and a bottom plane, and these planes can be defined either as open boundaries or closed boundaries.

3. Select the layer of interest.

4. To create a substrate, you will edit these layers, rename them, and add to them as desired. For more information on how to perform these tasks, refer to the sections below.

5. When you are finished defining the substrate, click **OK** to dismiss the dialog box.

## Defining an Open Boundary

An open boundary represents a layer of infinite thickness, such as air. An open boundary can be used to define other gases or infinitely-thick materials by editing the relative permeability and permittivity values of the boundary.

To define an open boundary:

1. Select the Free_Space layer or another open boundary layer.

2. Select **Open** from the Boundary list.

3. From the Permittivity (Er) listbox, select a format for the relative permittivity of the boundary. You can enter the components of relative permittivity as:

   • Real and imaginary

   • Real and loss tangent

   • Real and conductivity, conductivity in Siemens/meter

   The real portion of the relative permittivity, $\varepsilon'$, is a dimensionless quantity and is identical to the material's relative dielectric constant, $\varepsilon_r$. For more information on this parameter, refer to "Dielectric Permittivity" on page 3-22.

   To represent a dielectric that dissipates the power of a high-frequency electric field, enter the dielectric loss tangent, $\varepsilon''/\varepsilon'$, of the material in the Loss Tangent field. The smaller the loss tangent, the less lossy the material. For more information on this parameter, refer to "Dielectric Loss Tangent" on page 3-23.

   Enter the components of the relative permittivity in the fields below the listbox.

4. From the Permeability (Mu) listbox, select a format for the relative permeability of the boundary. You can enter the components of relative permeability as:

   • Real and imaginary

   • Real and loss tangent

   The real portion of the relative permeability, $\mu'$, is a dimensionless quantity and is identical to the material's relative permeability constant. For more information on this parameter refer to "Dielectric Relative Permeability" on page 3-24.

   To represent a dielectric that dissipates the power of a high-frequency magnetic field, enter the magnetic loss tangent, $\mu''/\mu'$, of the material in the Permeability Loss Tangent field. The smaller the loss tangent, the less lossy the material. For more information on this parameter, refer to "Dielectric Magnetic Loss Tangent" on page 3-25.

   Enter the components of the relative permeability in the fields below the listbox.

5. Click **Apply** to accept the open boundary definition.

---

**Tip**   You can create a ground plane from a open boundary layer by selecting the layer and choosing *Close* from the Boundary list.

---

## Defining an Interface Layer

Interface layers have finite thickness, and they can be characterized using relative permittivity and permeability values. The thickness of a layer can be an arbitrary value, with these considerations:

   • *Thin substrates* are substrates less than one micron in thickness, and they require special meshing considerations. Substrates less than 0.1 micron should be avoided.

   • *Thick substrates* should be less than 0.5 wavelength in thickness. The recommendation for thick substrates is based upon typical design values. For example, a 10 mil substrate with a cover height (air) of 300 mils would be acceptable for frequencies up to 20 GHz, which is 0.5 wavelength.

**To edit an interface layer:**

1. Select the Alumina layer or other interface layer of interest.

2. Enter the thickness of the layer in the Thickness field and select the appropriate units in the adjacent listbox.

3. From the Permittivity (Er) listbox, select a format for the relative permittivity of the boundary. You can enter the components of relative permittivity as:

   • Real and imaginary

   • Real and loss tangent

   • Real and conductivity, conductivity in Siemens/meter

   The real portion of the relative permittivity, $\varepsilon'$, is a dimensionless quantity and is identical to the material's relative dielectric constant, $\varepsilon_r$. For more information on this parameter, refer to "Dielectric Permittivity" on page 3-22.

   To represent a dielectric that dissipates the power of a high-frequency electric field, enter the dielectric loss tangent, $\varepsilon''/\varepsilon'$, of the material in the Loss Tangent field. The smaller the loss tangent, the less lossy the material. For more information on this parameter, refer to "Dielectric Loss Tangent" on page 3-23.

   Enter the components of the relative permittivity in the fields below the listbox.

4. From the Permeability (Mu) listbox, select a format for the relative permeability of the boundary. You can enter the components of relative permeability as:

   • Real and imaginary

   • Real and loss tangent

   The real portion of the relative permeability, $\mu'$, is a dimensionless quantity. For more information on this parameter refer to "Dielectric Relative Permeability" on page 3-24.

   To represent a dielectric that dissipates the power of a high-frequency magnetic field, enter the magnetic loss tangent, $\mu''/\mu'$, of the material in the Permeability Loss Tangent field. The smaller the loss tangent, the less lossy the material. For more information on this parameter, refer to "Dielectric Magnetic Loss Tangent" on page 3-25.

Enter the relative permeability components in the fields below the listbox.

5. Click **Apply** to accept the layer definition.

## Defining a Closed Boundary Layer

A closed boundary represents a plane, such as a ground plane. It is a layer with zero thickness. It can be defined as a perfect conductor, or you can specify bulk conductivity or sheet impedance to characterize it as a lossy conductor.

To edit a ground plane:

1. Select the **///GND///** layer of interest.

2. Select **Closed** from the Boundary list.

3. From the Plane listbox, select a format for the ground plane. You can specify the ground plane using these parameters:

   • Perfect conductor

   • Bulk conductivity in Siemens/meter

   • Sheet impedance in Ohms/square

   Conductivity is entered as a real number. Impedance is entered as the real and imaginary components of a complex value.

   Enter the ground plane parameters in the field below the listbox.

4. Click **Apply** to accept the closed boundary definition.

---

**Tip**   You can create an open boundary layer from a ground plane by selecting the ground plane and choosing *Open* from the Boundary list.

---

## Renaming a Layer

To rename a layer:

1. Select a substrate in the Substrate Layers list.

2. The name appears in the Substrate Layer Name field. Edit the name as desired.

3. Click **Apply**.

---

**Note** A ground plane is identified as *///GND///* and its name cannot be changed.

---

## Deleting, Adding, and Moving Layers

To organize your substrate layers in the correct order, you can delete, add, and move layers.

To delete a substrate layer:

1. Select a substrate in the Substrate Layers list.

2. Click **Cut**. The layer definition is deleted.

To add a substrate layer:

1. Select a substrate in the Substrate Layers list that has the same basic property (open boundary, closed boundary, or finite thickness) as the layer you are adding.

2. Click **Add**.

3. The new layer is highlighted. To rename the new layer, edit the Substrate Layer Name field.

4. To move the layer, refer to the steps below.

To move a substrate layer:

1. Select the substrate that you want to move from the Substrate Layers list.

2. Click **Cut**.

3. Select the substrate that you want positioned below the substrate that you are moving.

4. Click **Paste**.

# Defining Silicon Substrate Layers

The electrical behavior of Silicon material is usually specified using the dielectric constant and a resistivity (or conductivity value).

For the dielectric constant, a value for $\varepsilon'$ of 11.8 or 11.9 is usually used.

Resistivity ($\rho$) values are normally specified in ohm cm, and a typical value is 10.

In Momentum, you can specify the conductivity ($\sigma$) in S/m (inverse of the resistivity).

Foe more information, see "Dielectric Conductivity" on page 3-23.

## Calculating Conductivity from Resistivity

Conductivity is simply the inverse of resistivity. Make certain that the units of resistivity are ohms m before inverting.

Example:

1. $\rho$ = 10 ohm cm

2. $\rho$ = 10 ohm cm = 0.1 ohm m

3. $\sigma$ = 1/rho = 10 S/m

## Defining Conductivity

To define conductivity for silicon substrates:

1. Choose **Momentum > Substrate > Create/Modify.**

2. In the **Create/Modify** dialogue box, select *RE, Conductivity* under *Permittivity (ER).*

3. Specify the substrate dielectric constant and conductivity (S/m).

4. When you are finished, click **OK** to dismiss the dialog box.

# Defining Metallization Layers

Metallization layers enable you to:

- Define the position of layout layers in the substrate
- Specify which parts of the layer are conductive
- Define the conductivity of the layout layers

In order to identify which areas of a layout layer are conductive, a layer can be specified as:

- Strip—The objects on the layout layer are conductive, the rest of the layer is not

the hairpin filter objects are conductive and
mapped to a strip metallization layer

- Slot—The inverse of a strip, the objects on the layout layer are not conductive, but the rest of the layer surrounding the objects is conductive. When simulated, Momentum considers the electric field distribution (the equivalent magnetic current flow) in the slot.
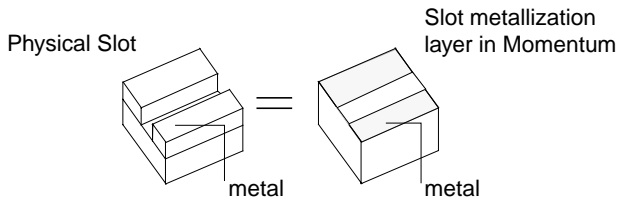
- Via—The objects on the layout layer are conductive and cut vertically through one or more substrate layers. For more information on how to draw and apply vias, refer to "Applying Vias" on page 3-30 and "Drawing Vias in Layout" on page B-8.
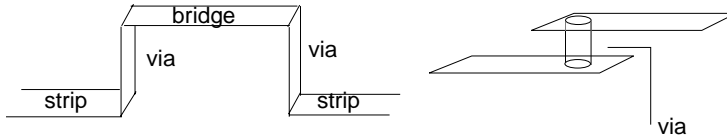


Examples of vias as sheets and cylinders

Some examples of using strips and slots include:

- The patches of the Double_Patch antenna example. They are drawn on the layout layer named top_met, and are conductive. This layer is mapped to a metallization layer that is defined as a strip.

- The slots of the Slot_dipole antenna example. The slots are drawn on the layout layer named slot. The slots are not conductive, but the area surrounding the slots is. This layer is mapped to a metallization layer that is defined as a slot.

Note that a layout can have many other layers that are not part of the actual circuit, such as text or error reporting layers. For the purposes of an Momentum simulation, they are ignored. If layout layers containing parts of the circuit are not mapped to metallization layers, they are ignored as well.

The steps for defining metallization layers are:

- Mapping a layout layer to a metallization layer
- Defining the conductivity of the layer
- Setting overlap precedence

Details for performing these steps are presented in the following sections.

## Mapping a Layout Layer

Layout layers that contain any shapes or components that are part of the circuit must be mapped to metallization layers. If layout layers are not mapped to metallization layers, they will not be included in the simulation.

To map a layout layer:

1. Choose **Momentum > Substrate > Create/Modify**.

2. Click the Metallization Layers tab.

3. Select a layer from the Layout Layers list box. Note that the layout layer *default* cannot be mapped to a metallization layer, the first valid layer is *cond*.

4. To map the layer as a slot or strip, select from the Substrate Layers list a dashed line to position the slot or strip in between two substrate layers, then click **Strip** or **Slot**.

   If you make a mistake, click **Unmap**, then click the correct choice.

   • A *Strip* defines the objects on the layout layer as conductive, the area surrounding the objects is not.

   • A *Slot* is the inverse of a strip, it defines the objects on the layout layer as not conductive, but the area of this layer surrounding the objects is conductive. When simulated, Momentum considers the electric field distribution (the equivalent magnetic current flow) in the slot.

5. To map the layer as a via, select the substrate layer that you want the via to cut through, then click **Via**.

   If you want the via to cut through more than one layer, select the next layer, then click **Via** again.

   • A *Via* defines the objects on the layout layer as conductive and they vertically cut through one or more substrate layers.

6. To define the conductivity characteristics of this layer, refer to "Defining Conductivity" on page 3-15.

7. If you have or suspect that you have overlapping layers, you need to set overlap precedence. Refer to "Setting Overlap Precedence" on page 3-17.

8. Click the **Apply** button.

## Unmapping a Layer

If you want to change the location of a layer or remove one from the substrate definition, use *Unmap*.

To unmap a layer:

1. Choose **Momentum > Substrate > Create/Modify.**

2. Click the Metallization Layers tab.

3. In the Substrate Layers list, select the substrate layer or the interface where the layout layer is mapped.

4. Click **Unmap**. This removes all layout layers assigned to this position.

5. To remap a layer, refer to the steps in .

## Defining Conductivity

You can define a layout layer as a conductor or a resistor. If the layout layer is defined as a strip or via, you define the conductivity of the strip or via. If the layout layer is defined as a slot, you define the conductivity of the metal around the slot.

To define conductivity:

1. Choose **Momentum** > **Substrate** > **Create/Modify.**

2. Choose **Metallization Layers**.

3. Select a mapped layout layer from the *Layout Layers* list.

4. Select the conductivity definition from the *Definition* list:

   • Perfect conductor

   • Sigma, using real and conductivity values

   • Sigma, using real and imaginary values

   • Impedance

   *Perfect Conductor* means the strip, via, or metal around the slot is a perfect conductor and is lossless. No further definition is required.

*Sigma (Re, thickness)* enables you to specify conductivity in terms of bulk conductivity and thickness. Enter bulk conductivity as a real value in the Conductivity field, in Siemens/meter or Siemens/centimeter. Enter the metal

thickness in the Thickness field and select the appropriate units. These values will be used to calculate the dc and high frequency loss mechanisms when determining the equivalent surface impedance. For example, the bulk conductivity of gold is 41,000,410 Siemens per meter with any arbitrary thickness. The equivalent DC sheet impedance that will be used when conductivity and thickness is specified as:

$$Z_{DC} = \frac{1}{\sigma t}$$

where $\sigma$ is the conductivity (S/m) and t is the thickness of the conductor (m).

**At high frequencies, the equivalent sheet impedance will be equivalent to:**

$$Z_{HF} = \frac{1+j}{2\sigma\delta}$$

where $\delta$ is the skin depth given by:

$$\delta = \sqrt{\frac{1}{\pi f \mu_0 \sigma}}$$

with f as the frequency (Hz) and $\mu_0$ the free space permeability. This high frequency loss specification corresponds to a single sheet skin-effect loss model, meaning that it assumes the current at high frequency is assumend to be concentrated on one side of the finite thickness conductor.

---

**Note** A double sheet skin effect loss model can be activated by setting the following variable in the momentum.cfg file:

MOM3D_USE_3DSURFACEIMPEDANCE=FALSE

---

*Sigma (Re, Im)* specifies bulk conductivity as a complex number. Enter the real and imaginary component of the value in the Real and Imaginary fields, respectively, in Siemens/meter or Siemens/centimeter. The bulk conductivity will be used to calculate the high frequency loss mechanisms, which are then applied to all frequencies when determining the equivalent surface impedance. The imaginary value is important for super-conductivity applications. Note that the single sheet skin-effect model is used and that this model will result in a equivalent sheet impedance equal to zero (or lossless) at DC. The ZHF formula is basically used at all frequencies.

*Impedance (Re, Im)* specifies conductivity in terms of sheet impedance, in Ohms/square. Enter the real and imaginary component of the value in the Real and Imaginary fields, respectively.

5. When you are finished with the definition, click **Apply**.

6. Repeat these steps for the remaining mapped layout layers.

7. When you are finished, click **OK** to dismiss the dialog box.

## Setting Overlap Precedence

Overlap precedence specifies which layout layer has precedence over another if two or more layout layers are assigned to the same metallization layer and objects on the layers overlap. Precedence is used by the mesh maker so that objects on the layer with the greatest precedence number are meshed and any overlap with objects on layers with lesser numbers is logically subtracted from the circuit. If you do not set the precedence, and there are overlapping objects, a mesh will automatically and arbitrarily be created, with no errors reported. Resistive layers generated from schematics are automatically set to the highest precedence.
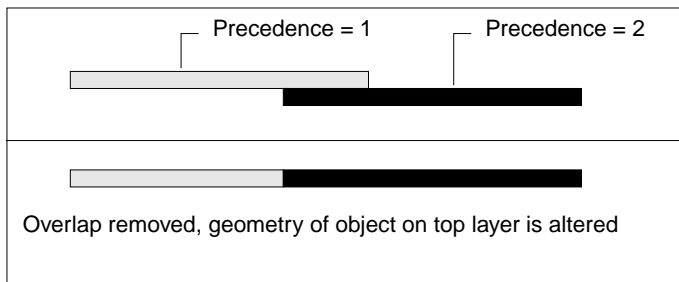


In some cases, you may be designing with an intentional overlap because of manufacturing layout guidelines. In this case, assign a precedence number to the layout layers that overlap with precedence order in reverse numerical order (largest to smallest). The system will draw the boundary at the edge of the higher numbered layer without returning an error.

Precedence affects only how the mesh is created, it does not affect or alter the layout layers in your design.

To specify overlap precedence:

1. Choose **Momentum > Substrate > Create/Modify**.

2. Choose Metallization Layers.

3. Select one of the overlapping layout layers in the list of Substrate Layers.

4. Assign its order of precedence in the Overlap Precedence field. Either type a value directly into the field or use the arrow keys to select a value.

5. Click **Apply**.

6. Repeat these steps for the remaining overlapping layers.

7. When you are finished, click **OK** to dismiss the dialog box.

# Precomputing the Substrate Function

In order to perform a simulation and to calculate a mesh, Green's functions characterizing the behavior of the substrate must be computed:

- If you intend to precompute a mesh before you simulate, you must precompute the substrate first. For more information on precomputing a mesh, refer to "Precomputing a Mesh" on page 6-11.

- If the Green's functions are not computed before you start a simulation, they will be calculated and then the simulation will run.

If you are using a substrate that is supplied with Advanced Design System, you do not need to precompute the substrate functions. If you have edited the substrate or created a new one, you must precompute the substrate function.

When the computations (Green's functions) are complete, the data is stored in a database, and it will be used in the simulation of any design that uses this substrate. If the substrate has been precomputed before and there has been no changes to the substrate, it will not be recomputed unless the frequency range has been extended. The frequency range can be set when using the Momentum mode, but not in Momentum RF mode. In RF mode, the frequency range (start-stop) is not requested. Since the RF mode uses the quasi-static Green's functions, they are calculated at quasi-static frequencies automatically chosen by the program. Substrate functions calculated in Momentum mode are reusable in RF mode if they are calculated from DC to some upper frequency.

For more information about the precomputation process, refer to Appendix A, Theory of Operation.

To precompute the substrate function:

1. Choose **Momentum** > **Substrate** > **Precompute**.

2. To specify the frequency range of the computations, enter starting frequency in the Minimum field and select the frequency units. Enter the stopping frequency in the Maximum field and select the frequency units. (Not available in RF mode.)

    Be sure that the frequency range includes all frequencies that you want to simulate.

3. Click **OK** to start the computations.

Once the computations are completed, you do not need to perform this step again unless the substrate is edited.

---

**Note**   Be aware that the greater the number of layout layers in the substrate, the more time required to perform the calculation.

---

### Viewing Substrate Computation Status

After the substrate calculations are started, any messages regarding the calculations will appear in the Momentum Status window. Messages usually refer to any errors found, and indicate when the calculations are complete.

If you close the status window and want to reopen it, from the Layout menu bar choose *Window > Restore Status*.

## Stopping Substrate Computations

To stop the substrate calculations:

1. From the Momentum Status window, choose **Simulation/Synthesis > Stop Simulation**.

The computations will stop and no information will be saved.

## Viewing the Substrate Summary

If the simulation is successful, you can view substrate, mesh, and solution statistics. Some of the information returned includes time to solve, the resources required, and cell information. An example of solution statistics is given next.

To view the summary, choose *Momentum > Substrate > Summary*.

```
====================================================================
                   --- STATUS SERVER MESSAGES ---
====================================================================
 Simulation / Synthesis Messages

Substrate to be computed from 9 GHz to 11 GHz!

Status / Summary

---------Momentum Simulation ------------------

Version: Momentum 4.0 (*) 150.100 Sep 25 2000
Simulation mode: microwave
Design: circular_polarization

Start compute substrate functions on Tue Oct 03 10:11:16 2000
Computation 0.0% complete.
Computation 2.6% complete.
====================================================================
```

# Reusing Substrate Calculations

Substrate definitions and substrate computations can be reused for other designs, if the substrate database is accessible. If you are unsure about access to the database, consult your system administrator.

Before a calculation begins, several folders are searched to determine if a solution has already been generated. The folders searched, in order, are:

- The project *networks* folder
- The home directory or the site directory (made available by your system administrator)
- The folder containing the Momentum supplied substrates (*$HPEESOF_DIR/Momentum/lib*)

If the Substrate Calculator finds the same substrate definition, regardless of the name, it uses the one it finds instead of recomputing the same information.

# About Dielectric Parameters

The sections below give additional detail about the following dielectric parameters:

- "Dielectric Permittivity" on page 3-22
- "Dielectric Loss Tangent" on page 3-23
- "Dielectric Conductivity" on page 3-23
- "Dielectric Relative Permeability" on page 3-24
- "Dielectric Magnetic Loss Tangent" on page 3-25

## Dielectric Permittivity

The relative permittivity of all dielectrics is assumed to be complex:

$$\varepsilon = \varepsilon' + j\varepsilon''$$

which can also be expressed as:

$$\varepsilon = \varepsilon'\left(1 + j\frac{\varepsilon''}{\varepsilon'}\right)$$

where $\varepsilon'$ is the real portion of $\varepsilon$ and $\varepsilon''/\varepsilon'$ is the dielectric loss tangent.

Examples of materials and their typical permittivity are shown in Table 3-1.

Table 3-1. Materials and Permittivity

| Material | Relative Permittivity |
|----------|----------------------|
| Alumina | 9.8 |
| Sapphire | 9.3 - 11.7 |
| Berrylia | 6.0 |
| Rutile | 100.0 |
| GaAs | 12.9 |

**Note** The values listed in the table are for illustrative purposes only; use values specifically measured for the materials that you are using.

## Dielectric Loss Tangent

The dielectric loss tangent associated with a material is a function of frequency. Examples of dielectric loss tangents for 10 GHz fields are shown in.

Table 3-2. Materials and Dielectric Loss Tangent

| Material | Dielectric Loss Tangent |
|----------|------------------------|
| Alumina | .0002 |
| Glass | .002 |
| Berrylia | .0001 |
| Rutile | .0004 |
| GaAs | .0016 |

**Note** These values are listed for illustrative purposes only. Use values specifically measured for the materials that you are using.

## Dielectric Conductivity

For some materials (for example, Silicon), the substrate loss effects are better described using the combination ($\varepsilon'$, $\sigma$ = conductivity) instead of ($\varepsilon'$, loss tangent). The complex dielectric constant is related to ($\varepsilon'$, conductivity) using the following formula:

$\varepsilon = \varepsilon' - ((j\sigma)/(\omega\varepsilon 0))$

where:

$\omega = 2\pi$ frequency

$\varepsilon 0 = 8.85\text{e-}12$ F/m (absolute dielectric constant free space)

Often, resistivity ($\rho$) is specified instead of conductivity. The relationship between resistivity and conductivity is

$\rho = 1/\sigma$

Resistivety is usually specified in $\Omega$ cm, conductivity is specified in S/m.

Example:

A typical value for resistivity of a Silicon material is

$\rho = 10 \ \Omega$ cm

which corresponds to a conductivity value of

$\sigma = 10$ S/m.

## Dielectric Relative Permeability

The relative permeability of all dielectrics is assumed to be complex:

$\mu = \mu' + j\mu''$

which can also be expressed as:

$$\mu = \mu'\left(1 + j\frac{\mu''}{\mu'}\right)$$

where $\mu'$ is the real portion of $\mu$ and $\mu''/\mu'$ is the magnetic loss tangent.

Examples of relative permeabilities are shown in Table 3-3.

Table 3-3. Materials and Relative Permeability

| Material | Relative Permeability |
|----------|-----------------------|
| Gold | .99996 |
| Air | 1 |
| Aluminum | 1.00002 |
| Nickel | 250 |
| Iron | 4000 |

**Note**   The values listed above are for illustrative purposes only. Use values specifically measured for the materials that you are using.

## Dielectric Magnetic Loss Tangent

The magnetic loss tangent associated with a material is a function of frequency. An example of a material with a magnetic loss tangent is polyiron which, at 30 GHz, has a loss tangent of 0.0208.

# Substrate Definition Summary

The entire substrate definition includes:

- Vias, in the same manner that they are part of the fabrication process.

- Air bridges, including their height and their associated vias.

- Holes in ground planes, if they are part of a slot signal path (slot or coplanar), or if they are relief holes for vias passing through a ground plane are also part of the media.

- Dielectric layer properties (Er, Mu, and loss).

- Ground plane losses and cover properties (conductivity or impedance).

Not included as part of the media or substrate definition are metal loss of strip and via layers. The numerical computation process, used by the substrate calculator, can take as little as one minute for stripline circuits. It can take several minutes for microstrip or even a few hours for an arbitrary multi-layered substrate with several vias.

Every change to a substrate definition requires a complete re-computation. The system checks for duplicate definitions, regardless of the name, and uses previously computed data if it exists.

# Substrate Examples

Use the following examples to help you set up your own circuits:

- "Substrate for Radiating Antennas" on page 3-27
- "377 Ohm Terminations and Radiation Patterns" on page 3-27
- "Substrate for Designs with Air Bridges" on page 3-27

# Substrate for Radiating Antennas

To define a substrate for an antenna that radiates into air, you can set up the following substrate definition:

**Substrate Layers**

Free_Space1
Free_Space2
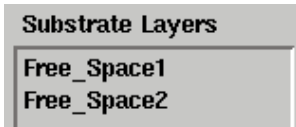
Both layers are defined as open boundaries with the characteristics of air (for permittivity and permeability, Real = 1, Loss Tangent = 0). The antenna design is positioned on the metallization layer in between the two layers of air.

# 377 Ohm Terminations and Radiation Patterns

It is possible to define the top and bottom layers of a design with 377 ohm terminations, then calculate radiation patterns for these structures after simulation. For such applications, note the following considerations:

- The actual value of the termination may be between 376 and 378 ohms.

- When replacing top or bottom ground layer with a 377 ohm termination, a thick layer of air in between the circuit and the 377 ohm termination is required in order to get agreement (both for S-parameter results and radiation patterns) with the simulation results with infinite top or bottom layer. An air layer of 10-20 substrate thicknesses is generally thick enough.

- There is a test on the impedance value prior to calculating the radiation patterns. The impedance value must be in the interval (377 - 10) to (377 + 10) ohm.

# Substrate for Designs with Air Bridges

In order to designing a substrate definition for a design that contains an air bridge, you need to first identify components of the air bridge and their placement on layout layers. Refer to the illustration that follows.

A basic air bridge between two components consists of the follow items:

- A via that connects the bridge to the first component
- The bridge itself
- A second via to complete the path to the second component
- The layer of air underneath the bridge

The layout must be drawn on at least three independent layers:

- The vias must be on at least one layer
- The bridge must be on a different layer
- The components must be on yet another layer

No two items that comprise the air bridge (except for the vias) can be on the same layer. For this illustration, the components that are connected by the air bridge are on the same layer.

The substrate definition would consist of the following items:

- An open boundary of air
- An interface layer of air at a finite thickness. This represents the layer of air under the bridge.
- An interface layer of dielectric material

You map the layout layers to the substrate as follows:

- The bridge is mapped to the metallization layer that is in between the open air substrate layer and the finite layer of air
- The vias are mapped to cut through the finite substrate layer of air
- The components are mapped to the metallization layer in between the finite layer of air and the dielectric layer

It is not required for the components that the bridge connects to be on the same layout layer or mapped to the same metallization layer. Nor do the vias need to be on the same layout layer. Just be sure that the substrate definition has the correct

number of substrate layers and that the components, vias, and bridge are mapped to the appropriate metallization layers.

Substrate Layer Setup

**Substrate Layers**

Free_Space ———————— Air, boundary is Open
air ————————————— Air under bridge, thickness is specified
GaAs ———————————— Dielectric, thickness is specified
air2
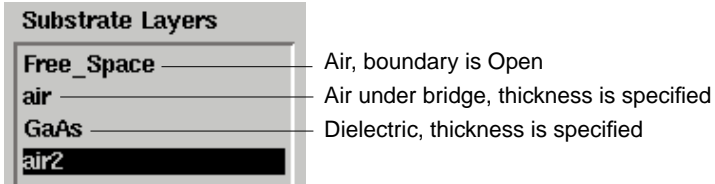
Metallization Layer Mapping

**Substrate Layers:**

Free_Space

Map bridge here ————— - - - - - - - **Strip bridge**
Map vias here ————— air ‖ via ———
Map strips here ————— - - - - - - - **Strip**

GaAs

- - - - - - - - -

air2

Physical Circuit

bridge
via          via
strip        strip
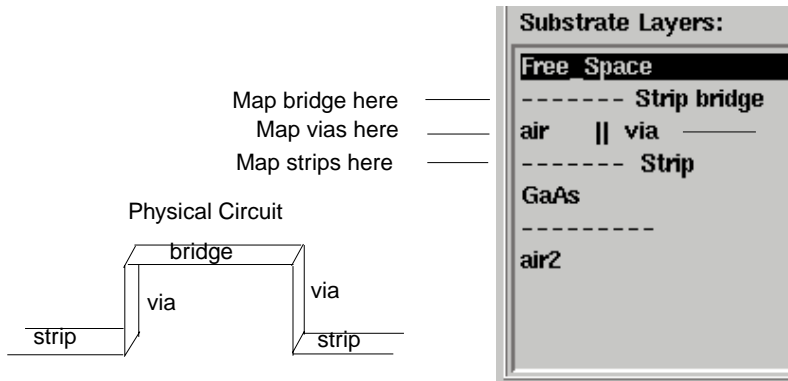
# Applying Vias

Momentum creates a via by extruding the object that is mapped as via through the substrate layer it is applied to. So, for example, for a sheet via, you draw a line, for a cylinder via you draw a circle. When the shape is mapped to a via in Momentum, another dimension is added to the object in order for the shape to cut through the substrate. Thus a line becomes a sheet, a circle becomes a cylinder.
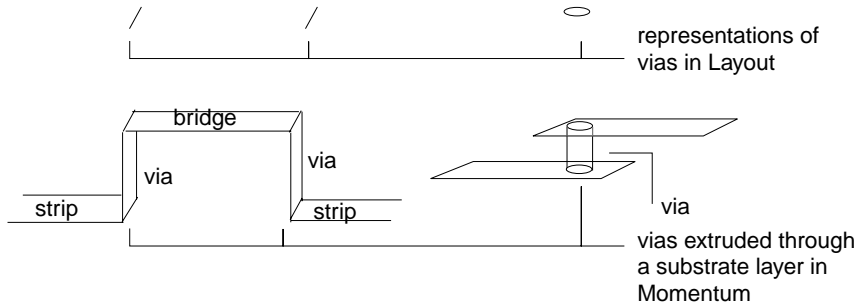


Regardless of how you draw vias, avoid having them extend over the sides of the objects that they connect to. Vias must be on the edge or inside the object. Any portion of the via outside of the object boundaries will not be taken into account during the simulation. The figure here illustrates vias connecting two strips.
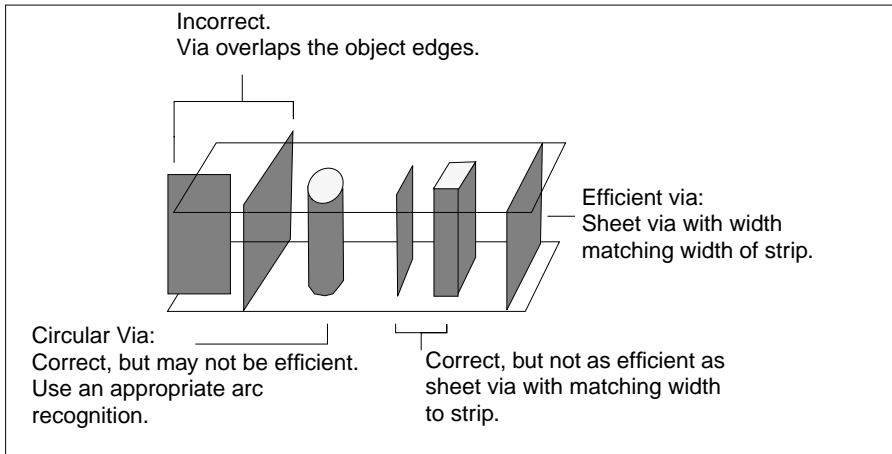


Vias can be applied to multiple layers so that they cut through one or more layers.

# Chapter 4: Ports

Ports enable energy to flow into and out of a circuit. Energy is applied to a circuit as part of the simulation process. A circuit solved using Momentum can have, at the minimum, one port.

Ports are defined in a two-step process. First, ports are added to a circuit when the circuit is drawn. Then, in Momentum, you specify the type of port in order to tailor the port to your circuit. This facilitates the simulation process.

This chapter begins with suggestions to keep in mind when adding ports to a circuit that will be simulated using Momentum. The remainder of this chapter describes the various port types in Momentum and gives instructions on how to specify a port type.

## Adding a Port to a Circuit

You can add a port to a circuit either from the Schematic window or a Layout window. For instructions on how to add a port to a circuit, refer to "Adding a Port to a Circuit" on page B-4. The procedures include considerations for adding ports to a circuit that will be simulated using Momentum.

### Considerations

Keep the following points in mind when adding ports to circuits to be simulated using Momentum:

- The components or shapes that ports are connected to must be on layout layers that are mapped to metallization layers that are defined as strips or slots. Ports cannot be directly connected to vias. For information on how to define strips and slots, refer to "Defining Metallization Layers" on page 3-12.

- Make sure that ports on edges are positioned so that the arrow is outside of the object, pointing inwards, and at a straight angle.

- Make sure that the port and the object you are connecting it to are on the same layout layer. For convenience, you can set the entry layer to this layer; the Entry Layer listbox is on the Layout tool bar.

- A port must be applied to an object. If a port is applied in open space so that is not connected to an object, Momentum will automatically snap the port to the edge of the closest object. This will not be apparent from the layout, however, because the position of the port will not change.

- If the Layout resolution is changed *after* adding ports that are snapped to edges, you must delete the ports and add them again. The resolution change makes it unclear to which edges the ports are snapped, causing errors in mesh calculations.

---

**Note** Do not use the ground port component (*Component > Ground*) in circuits that will be simulated using Momentum. Either add ground planes to the substrate or use the ground reference ports that are described later in this chapter.

Ground port component toolbar button:

---

# Determining the Port Type to Use

There are five port types in Momentum. The purpose of ports is to inject energy into a circuit and to allow energy to flow into and out of a circuit. The different port types enable you to tailor the ports in your circuit according to your type of circuit and its function in the circuit. In general, you should select the port type that best matches the intended application of your layout.

The table below gives a brief description of each port type. You can use a combination of port types in your circuit, although note that port types have limitations on where they can be applied. Only the Single port type can be applied to objects that are on either strip or slot metallization layers. Only the Internal and Ground Reference types can be applied to ports that are on the surface of an object, the remaining types can be applied to ports that are connected to an edge of an object.

---

**Note** Strips and slots refer to metallization layers. For more information on these layers, refer to "Defining Metallization Layers" on page 3-12.

---

If you elect not to assign port types, any port in the circuit will be treated as a Single port type during simulation.

**Note** All port types can be defined in both of Momentum's simulation modes (microwave and RF). However, de-embedding (reference offset) is not available in RF mode.

| Port Type | Description | Port Connected to | Object on |
|-----------|-------------|-------------------|-----------|
| Single (default) | The port is calibrated to remove any mode mismatch at the port boundary. Single ports on slot layers have polarity. Unless a port is given another type, it will be treated as a single port. | Edge of object | Strip or slot layer |
| Internal | The port is not calibrated. It is useful for making a connection with lumped elements or for representing other connections in the circuit. | Edge or surface of object | Strip layer |
| Differential | Two ports with opposite polarity. The port pair is simulated as a single port. | Edge of object | Strip layer |
| Coplanar | Two ports with opposite polarity. The port pair is simulated as a single port. | Edge of object | Slot layer |
| Common mode | Two or more ports excited with the same absolute potential and the same polarity. The ports are simulated as a single port. | Edge of object | Strip layer |
| Ground reference | An explicit ground for a single (strip), internal, or common mode port.The implicit ground is made available through the closest infinite metal. | Edge or surface of object | Strip layer |

Additional details about each port type and how to define them are given in the following sections.

# Defining a Single Port

Single is the default port type. It has the following properties:

- It is connected to an object that is on either a strip or slot metallization layer.

- It can be applied only to the edge of an object.

- The port is external and calibrated. The port is excited using a calibration process that removes any undesired reactive effects of the port excitations (mode mismatch) at the port boundary. This is performed by extending the port boundary with a half-wavelength calibration (transmission) line. The frequency wavelength selected during the mesh or simulation process is used to calculate the length of the calibration line. For more information about the calibration process, refer to "Calibration and De-embedding of the S-parameters" on page A-7.

- The port boundary can be moved into or away from the geometry by specifying a reference offset. S-parameters will be calculated as if the port were at this position. For more information, refer to "Applying Reference Offsets" on page 4-6.

- When two or more single ports are on the same reference plane, coupling effects caused by parasitics affects the S-parameters. The calibration process groups the ports so that any coupling in the calibration arms is included in the S-parameter solution. For more information, refer to "Allowing for Coupling Effects" on page 4-9.

- If the port is connected to an object on a strip layer, the substrate definition must include at least one infinite metal layer: a top cover, ground plane, or a slot layer, or a ground reference must be used in addition to the port. For more information on ground references, refer to "Defining a Ground Reference" on page 4-19.

- If the port is connected to an object that is on a slot layer, the port has polarity.
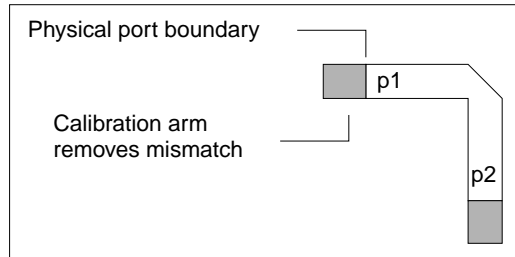
To define a single port type:

1. Choose **Momentum > Port Editor**.

2. Select the port that you want to assign this type to.

3. In the Port Editor dialog box, under Port Type, select Single.

4. Enter the components of the port impedance in the Real and Imaginary fields, and specify the units.

5. You can shift the port boundary, also referred to as the port reference plane. Shifting the boundary enables a type of de-embedding process that effectively adds or subtracts electrical length from the circuit, based on the characteristic impedance and propagation characteristic of the port. Enter the offset in the Reference Offset field, and select the units. A positive value moves the port boundary into the circuit, a negative value moves the port boundary away from the circuit.

6. Click **Apply** to add the definition to the port.

# Avoiding Overlap

Be aware that when using single ports, the calibration arm applied to a port may be long enough to overlap another element in the circuit. In this case, the port will be changed to an internal port type, and no calibration will be performed on it. If this occurs, a message will be displayed during simulation in the Status window indicating the change. For information about internal ports, refer to "Defining an Internal Port" on page 4-10.



The calibration arm overlaps the strip. An internal port will be used instead.
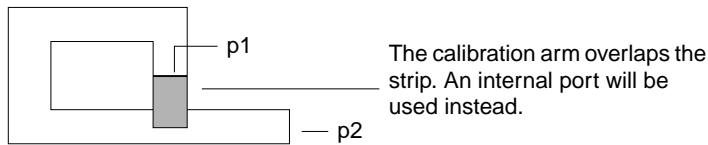
# Applying Reference Offsets

Reference offsets enable you to reposition single port types in a layout and thereby adjust electrical lengths in a layout, without changing the actual drawing. S-parameters are returned as if the ports were placed at the position of the reference offset.



S-parameters will be calculated based on the positions of p1 and p2

no reference offset
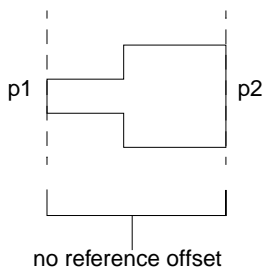
S-parameters will be calculated as if p1 and p2 were positioned at the reference offsets

reference offsets applied to p1 and p2
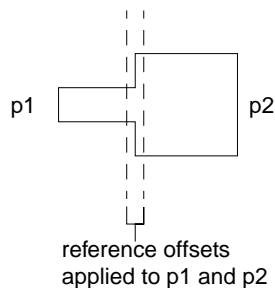
## Why Use Reference Offsets?

The need to adjust the position of ports in a layout is analogous to the need to eliminate the effect of probes when measuring hardware prototypes. When hardware prototypes are measured, probes are connected to the input and output leads of the Device Under Test (DUT). These probes feed energy to the DUT, and measure the response of the circuit. Unfortunately, the measured response characterizes the *entire* setup, that is, the DUT plus the probes. This is an unwanted effect. The final measurements should reflect the characteristics of the DUT alone. The characteristics of the probes are well known, so measurement labs can mathematically eliminate the effects of the probes, and present the correct measurements of the DUT.

There are significant resemblances between this hardware measurement process and the way Momentum operates. In the case of Momentum, the probes are replaced by ports, which, during simulation, will feed energy to the circuit and measure its response. The Momentum port feeding scheme also has its own, unwanted effect: low-order mode mismatch at the port's boundary, although this is eliminated by the calibration process. However, in order for this calibration process to work well, it is necessary that the fundamental mode is characterized accurately. This can only be accomplished when the distance between the port boundary and the first discontinuity is sufficiently large, that is, there exists a feedline that is long enough to provide this distance.

As a basic example, consider a linewidth that varies abruptly in some part of your circuit, as shown in the example below.
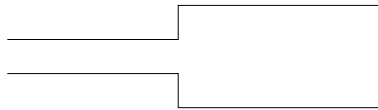
Strictly speaking, all you need is to characterize is the variation of the step-in-width itself, as shown below.

p1     p2
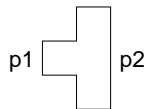
As mentioned previously, it takes a little distance for the fundamental mode to settle, which means that this "short" structure might not yield the accuracy that you expect from an Momentum simulation. In this case, allow for some feed line length:
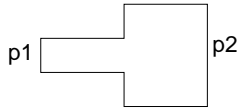


Now the simulation will yield accurate results, but the results will also contain the extra line lengths. To remedy this, use reference offsets. Although the circuit has been calculated with the long lines, reference offset shifting allows you to produce the S-parameters as if the short structure had been simulated instead:



apply reference offsets at
these locations
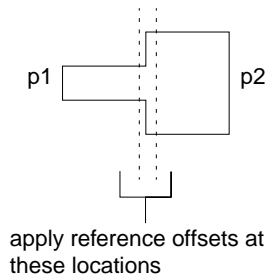
The effect of the extra feed lines is mathematically eliminated from the S-parameter solution.

This process of adding or subtracting line length is generally referred to as *de-embedding*. This is the basic process:

During the solution process, the impedance and propagation constant has been calculated for the ports, based on their physical location in the circuit. When you know the impedance, propagation constant, and the distance of de-embedding, you can cancel out the extra lengths of line from the S-parameter results, by compensating for the loss and phase shifts of those lines. The net result is a set of S-parameters, calculated as if the extra line lengths were not there.

## De-embedding Considerations

It is possible to de-embed right up to the discontinuity itself. However, make sure that you do not shift the reference offset beyond the first discontinuity. This would yield incorrect simulation results, as there is another linewidth beyond that

discontinuity, which means that there is another set of impedance and propagation values that applies there.

---

**Note**   You can de-embed away from the circuit, by placing reference offsets beyond the edges of the layout. This enables you to simulate the effect of a long feed line that was not drawn in the simulated structure.

---

Note that for calculation speed purposes, the impedance and propagation constant for grouped ports are approximated instead of explicitly calculated. The only way to have impedance and propagation constant explicitly calculated in all cases is to de-embed some small distance. This will cause some lengthy calculations, but the results should be more accurate for users who want to look at accurate impedance and propagation constants.

## Allowing for Coupling Effects

If you have two or more single ports that lie on the same reference plane, the calibration process will take into account the coupling caused by parasitics that naturally occurs between these ports. This yields simulation results that more accurately reflect the behavior of an actual circuit.

The figure below helps illustrate which ports will be grouped in order for the calibration process to account for coupling among the ports. In this setup, only the first two ports will be grouped, since the third port is an internal port type and the fourth port is on a different reference plane. Note that even though the second port has a reference offset assigned to it, for this process they are considered to be on the same plane and their reference offsets will be made equal.

If you do not want the ports to be grouped, you must add a small thickness of metal to the edge of the object that one of the ports is connected to. The ports will no longer be on the same plane, and will not be considered part of the same group.
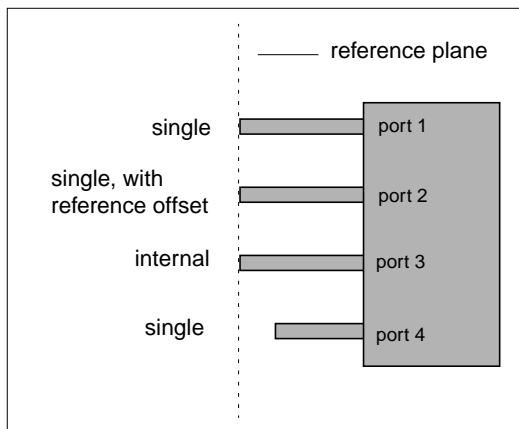


## Defining an Internal Port

Internal ports enable you to apply a port to the surface of an object in your design. By using internal ports, all of the physical connections in a circuit can be represented, so your simulation can take into account all of the EM coupling effects that will occur among ports in the circuit. These coupling effects caused by parasitics are included in your simulation results because internal ports are not calibrated. You should avoid geometries that allow coupling between single and internal ports to prevent incorrect S-parameters.

An example of where an internal port is useful is to simulate a bond wire on the surface on an object. Another example of where an internal port is necessary is a circuit that consists of transmission lines that connect to a device, such as a transistor or a chip capacitor, but this device is not part of the circuit that you are simulating. An internal port can be placed at the connection point, so even though the device is not part of the circuit you are simulating, the coupling effects that occur among the ports and around the device will be included in your simulation.

Internal ports are often used in conjunction with ground references. For more information, refer to "Defining a Ground Reference" on page 4-19 and to "Simulating with Internal Ports and Ground References" on page A-10.

An internal port type has the following properties:

- It can be applied to the interior of a circuit by applying it to the surface of an object.
- It can be applied to the edge of an object.
- It can be applied to objects that are on strip layers only.
- The orientation of the port is not considered if it is on the surface of an object. (For a description of port orientation, refer to "Adding a Port to a Layout" on page B-5.)
- No calibration is performed on the port.

Because no calibration is performed on the port, the results will not be as accurate as with a single port. However, the difference in accuracy is small.
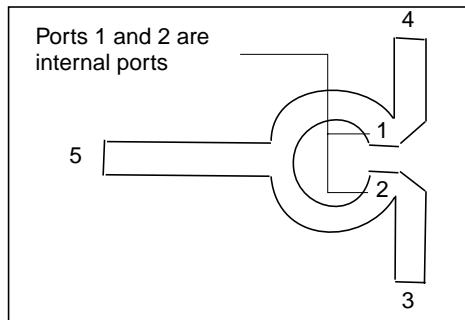


To define an internal port:

1. Choose **Momentum > Port Editor**.
2. Select the port that you want to assign this type to.
3. Click **Apply**.

# Defining a Differential Port

Differential ports should be used in situations where an electric field is likely to build up between two ports (odd modes propagate). This can occur when:

- The two ports are close together
- There is no ground plane in the circuit or the ground plane is relatively far away
- One port behaves (to a degree) like a ground to the other port, and polarity between the ports is developed.
- The ports are connected to objects that are on strip metallization layers.

The electric field that builds up between the two ports will have an effect on the circuit that should be taken into account during a simulation. To do this, use differential ports.

Differential ports have the following properties:

- They can be applied to objects on strip layers only.
- They are assigned in pairs, and each pair is assigned a single port number.
- Each of the two ports is excited with the same absolute potential, but with the opposite polarity. The voltages are opposite (180 degrees out of phase). The currents are equal but opposite in direction when the ports are on two symmetrical lines, and the current direction is approximated for other configurations.

• The two ports must be on the same reference plane.



An electric field will likely build up between these two lines. Assign the ports as differential ports. They will be treated as one port and assigned the same port number.

Assign as differential ports. They will be treated as one port and assigned the same port number.

---
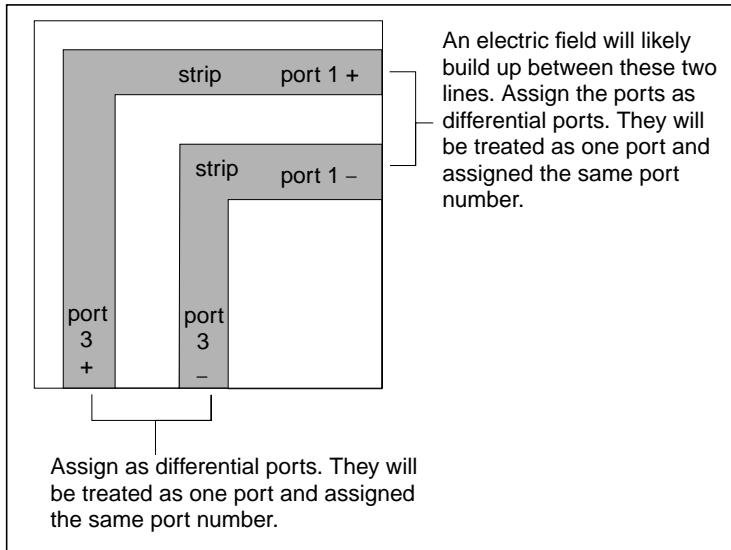
**Note**   Port numbers for differential ports are treated in the following manner: on the layout, you will continue to see the port numbers (instance names) that were assigned to each port when they were added to the layout. Use the Momentum Port Editor dialog box to identify which pair of ports will be treated as a differential port.

Momentum re-numbers differential ports to result in consecutive port numbers in the dataset.

Also, when you view results, you will see S-parameters for the differential port numbers. In the example above, the layout would show p1, p2, p3, p4. The S-parameter results will be for combinations of the original P1 and P3 only.

---

To define a differential port:

1. Choose **Momentum > Port Editor**.

2. Select the port that you want to assign this type to. Note the port number.

3. In the Port Editor dialog box, under Port Type, select **Differential**.

4. Under Polarity, make sure that **Normal** is selected.

5. Click **Apply**.

6. Select the second port.

7. In the Port Editor dialog box, under Port Type, select **Differential**.

8. Under Polarity, select **Reversed**.

9. Under Associate with port number, enter the number of the previously-selected port.

10. Click **Apply**.

11. Repeat these steps for other differential port pairs in the circuit.

12. Click **OK** to dismiss the dialog box.

# Defining a Coplanar Port

This type of port is used specifically for coplanar waveguide (CPW) circuits. It is similar to a differential port, but coplanar ports are applied to objects on slot layers (that is, where slots are used in the design). Coplanar ports should be used in situations where an electric field is likely to build up between two ports. This can occur when:

- The two ports are close together
- Polarity between the ports develops
- The ports are connected to objects that are on slot metallization layers

The electric field that builds up between the two ports will have an effect on the circuit that should be taken into account during a simulation. To do this, use coplanar ports.

Coplanar ports have the following properties:

- They can be applied to objects on slot layers only.
- They are assigned in pairs.
- Each of the two ports is excited with the same absolute potential, but with the opposite polarity. The voltages are opposite (180 degrees out of phase). The currents are equal but opposite in direction when the ports are on two

symmetrical lines, and the current direction is approximated for other configurations.

- The two ports must be on the same reference plane.

The illustration below shows ports pairs applied to a layout of coplanar waveguide slots. The coplanar port type is applied to each pair of ports in the design. The arrows in this illustration indicate the polarity assigned to each port and the direction of the voltages over the slots (arrows indicate the direction of the voltages).



Arrows indicate orientation (polarity) of the voltages over the slots.
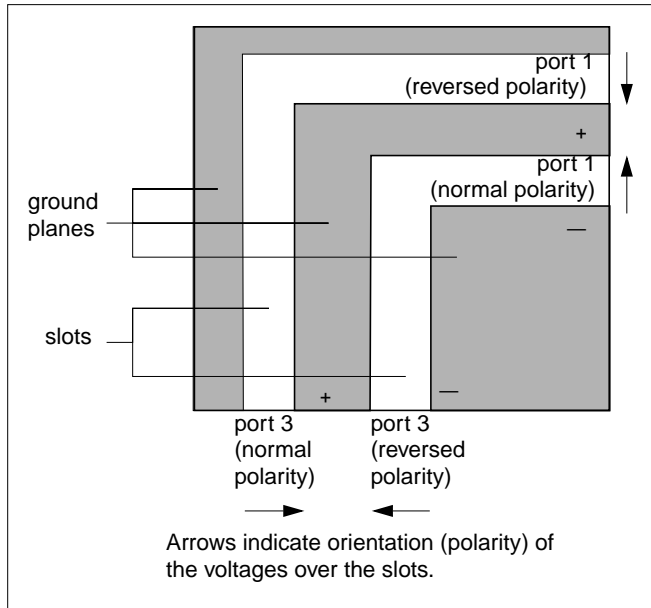
**Note** Port numbers for coplanar ports are treated in the following manner: on the layout, you will continue to see the port numbers (instance names) that were assigned to each port when they were added to the layout. Use the Momentum Port Editor dialog box to identify which pair of ports will be treated as a coplanar port.

Momentum re-numbers differential ports to result in consecutive port numbers in the dataset.

Also, when you view results, you will see S-parameters for the coplanar port numbers. In the example above, the layout would show p1, p2, p3, p4. The S-parameter results will be for combinations of the original P1 and P3 only.

Be careful when assigning polarity to coplanar ports. An incorrect choice of polarity can change the phase of transmission type S-parameters by 180 degrees. To verify polarity, zoom in on a coplanar port. You will notice two sets of arrows applied to the port. One appears when you add the port component to the circuit. The second will appear after the mesh is computed. It indicates the direction of the voltage over the slot.



This arrow appears after the mesh is computed and indicates the direction of the voltage over the slot.

This arrow is part of the port component
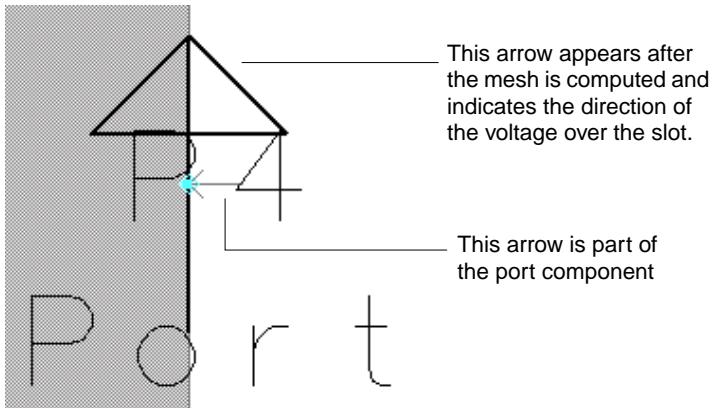
To define coplanar ports:

---

**Note**  Coplanar ports can be applied to objects on slot layers only.

---

1. Choose **Momentum > Port Editor**.

2. Select the port that you want to assign this type to. Note the port number.

3. In the Port Editor dialog box, under Port Type, select **Coplanar**.

4. Under Polarity, make sure that **Normal** is selected.

5. Click **Apply**.

6. Select the second port.

7. In the Port Editor dialog box, under Port Type, select **Coplanar**.

8. Under Polarity, select **Reversed**.

9. Under Associate with port number, enter the number of the previously-selected port.

10. Click **Apply**.

11. Repeat these steps for other differential port pairs in the circuit.

12. Click **OK** to dismiss the dialog box.

# Defining a Common Mode Port

Use common mode ports in designs where the polarity of fields is the same among two or more ports (even modes propagate). The associated ports are excited with the same absolute potential and are given the same port number.

Common mode ports have the following properties:

- They can be applied to objects on strip layers only

- A ground plane or other infinite metal (such as a cover) is required as part of the design

- Two or more ports can be associated

- Associated ports are excited with the same absolute potential (and same polarity)

- The ports must be on the same reference plane



Three ports are treated as one after they are
associated as common mode ports.
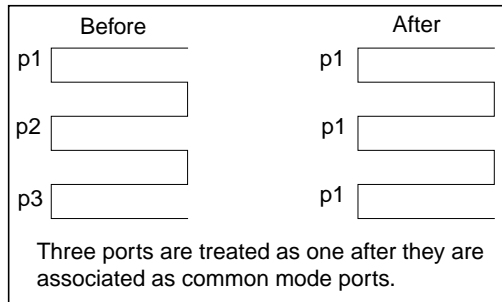
**Note**   Port numbers for common ports are treated in the following manner: on the
layout, you will continue to see the port numbers (instance names) that were
assigned to each port when they were added to the layout. Use the Momentum Port
Editor dialog box to identify which group of ports will be treated as a common port.

Also, when you view results, you will see S-parameters for the common port numbers.
In the example above, the layout would show p1, p2, p3. The S-parameter results will
be for combinations of P1 only.

To define a common port:

1. Choose **Momentum > Port Editor**.

2. Select the port that you want to assign this type to. Note the port number.

3. In the Port Editor dialog box, under Port Type, select **Common Mode**.

4. Click **Apply**.

5. Select the second port.

6. In the Port Editor dialog box, under Port Type, select **Common Mode**.

7. Under Associate with port number, enter the number of the port that you
   selected first.

Make sure that the value in the Associate with port number field is the same for
additional ports. For example, if you were associating three ports and the first port
was assigned as port 1, for the second and third port, the value entered into the

Associate with port number field would be 1. (For the first port you choose, no value is entered in this field.)

8. Click **Apply**.

9. Repeat these steps for other common mode ports in the circuit.

10. Click **OK** to dismiss the dialog box.

# Defining a Ground Reference

Ground references enable you to add explicit ground references to a circuit, which may be necessary if implicit grounds are in your design.

Implicit ground is the potential at infinity, and it is made available to the circuit through the closest infinite metal layer of the substrate. Implicit grounds are used with internal ports and with single ports that are connected to objects on strip metallization layers.

There are instances where the distance between a port and its implicit ground is too large electrically, or there are no infinite metal layers defined in the substrate. In these cases, you need to add explicit ground references to ensure accurate simulation results. For more information on using ground references, refer to "Simulating with Internal Ports and Ground References" on page A-10.

You can apply ground references to the surfaces of object. The object must be on strip metallization layers.

---

**Note** Multiple ground reference ports can be associated with the same port. To be associated with a single port, the ground reference port should be a port attached to an edge of an object in the same reference plane as the single port.

---

To add a ground reference:

1. Choose **Momentum > Port Editor**.

2. Select the port that you want to assign as the ground reference.

3. In the Port Editor dialog box, under Port Type, select **Ground Reference**.

4. Under **Associate with port number**, enter the number of the single or internal port that you want to associate with this ground reference. Make sure that the distance between the port and ground reference is electrically small.

5. Click **Apply**.

# Editing a Port

For best results, use the Port Editor (**Momentum > Port Editor...**) to change port properties. You edit the port in the same manner as when you entered the initial definition.

---

**Note**   Be aware that it is possible to edit port properties by selecting the port and choosing *Edit > Properties* from the Layout menu bar, however, *this method is not recommended*.

---

For more information on setting port definitions see the appropriate section:

"Defining a Single Port" on page 4-4

"Defining an Internal Port" on page 4-10

"Defining a Differential Port" on page 4-12

"Defining a Coplanar Port" on page 4-14

"Defining a Common Mode Port" on page 4-17

# Remapping Port Numbers

Some designs contain non-consecutive port numbers. This results in simulation data files that are difficult to use. When Momentum simulates designs containing non-consecutive port numbers, the ports are remapped to consecutive numbers in the resulting data file. The lowest port number is remapped to 1, and remaining numbers are remapped in consecutive order. *The port numbers are not changed in the design itself.* A message in the Status window announces the change, and lists the mappings.

For example, if you are simulating a design with ports numbered 37 and 101, the following status message informs you of the changes:

```
Layout has non-consecutive port numbers.
Output files will have consecutive port numbers.

   layout port -> output port
   37          -> 1
   101         -> 2
```

Port number remapping is done only for sampled and AFS CITIfiles and their corresponding S-parameter datasets. It is not done for Visualization and far field files. The remapping is done at the CITIfile level, and propagates to the dataset file. After remapping, all datasets are in sync with the new port numbering.

Ports

# Chapter 5: Boxes and Waveguides

When you specified the substrate definition of a circuit (from the *Momentum > Substrate* menu), you specified only the vertical dimension of the substrate and not the horizontal dimension. Given this definition, the substrate layers extend all the way to infinity in the horizontal direction. For many circuit designs this is not relevant and does not affect the simulation. However, there may be instances where you want to introduce horizontal boundaries. For these instances, you can use boxes or waveguides.

Boxes and waveguides enable you to specify substrate boundaries in the horizontal direction. A *box* enables you to set boundaries on four sides of the substrate. A *waveguide* enables you to set boundaries on two sides. These effects are available only when using the Momentum (microwave) simulation mode, and not with the Momentum RF mode.
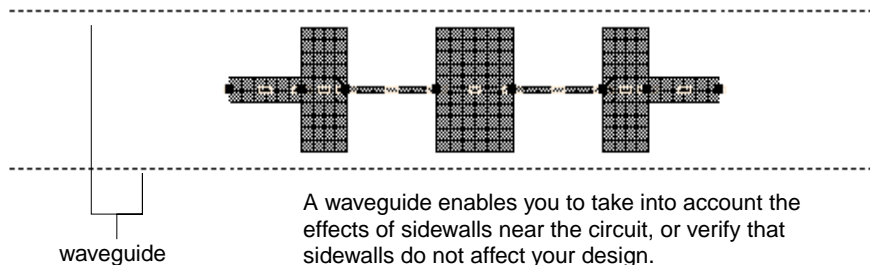
More specifically, for a box, you define four, perpendicular vertical planes of perfect metal as the horizontal boundaries of the substrate. These four vertical planes, or walls, form a rectangle, if you are looking at the circuit from the top, along the z-axis. A box can be used only where the top and bottom layers in the substrate definition are groundplanes or impedance termination. Thus, the four vertical metal walls, plus the top and bottom groundplanes result in a box, hence the name (the walls are the sides, and the top and bottom ground planes are lid and base of the box).



box
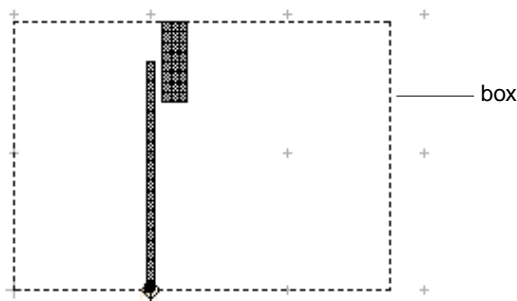
A box can represent a highly resonant
metallic enclosure. Its effect will be taken
into account during simulation.

A waveguide is similar, although for a waveguide you specify only two parallel walls. The substrate is therefore only bounded in the horizontal direction perpendicular to

these walls. In the horizontal direction parallel to these walls, the substrate still extends to infinity. The top and bottom layers of the substrate must also be defined as groundplanes. The two sidewalls in combination and the top and bottom covers form a waveguide.



waveguide

A waveguide enables you to take into account the effects of sidewalls near the circuit, or verify that sidewalls do not affect your design.

The next section describes how to apply a box or waveguide to a circuit. For more information about applications for these items, refer to "About Boxes and Waveguides" on page 5-5.

# Adding a Box

A box defines the boundaries on four sides of the circuit substrate. Either one box or one waveguide can be applied to a circuit at a time. A box can be applied to a circuit only if the top and bottom layers of the substrate definition are defined as groundplanes or impedance termination. The walls of the box are perfect metal. The ground planes can be defined either as perfect metals or a lossy metal.

Adding a box to the circuit enables you to analyze the effects of enclosing the circuit in metal, for example, to identify box resonance. Box resonance can have a significant effect on S-parameters in a small band centered around the box resonance frequency. When a simulation is performed, the resonance frequencies will be noted in the status window when the circuit is simulated, along with the frequency bands where no smooth S-parameters can be calculated.

During a simulation, all current directions on the sidewalls of the box are taken into account.

To add a box:

1. Choose **Momentum > Box-Waveguide > Add Box**.

2. Draw the box using one of the following two methods:

   • Position the mouse and click to define a corner of the box. Move the mouse to the diagonal corner and click.

   • From the Layout menu bar choose **Draw > Coordinate Entry** and use the *Coordinate Entry X* and *Coordinate Entry Y* fields to specify a corner of the box. Click **Apply**. Enter the coordinates of the diagonal corner and click **Apply**. Click **Cancel** to dismiss the dialog box.

The box is displayed in the layout.

## Editing a Box

Once the box is applied, you cannot change its dimensions. If you want to change the size, you must delete the current box and add a new one.

## Deleting a Box

To delete a box:

1. Choose **Momentum > Box-Waveguide > Delete Box**.

The box is removed from the layout.

## Viewing Layout Layer Settings of a Box

The box is defined as a layout layer named *momentum_box*. You may review the layout layer settings, but it is a protected layer, so you should not change the settings of this layer.

To view the box layer specifications:

1. Choose **Options > Layers**.

2. Select **momentum_box** from the *Layers* list. The layer settings are displayed. For more information on these parameters, refer to the *Layout* manual.

3. Click **Cancel** to dismiss the dialog box.

# Adding a Waveguide

A waveguide defines the boundaries on two, parallel sides of the circuit substrate. Either one box or one waveguide can be applied to a circuit at a time. A waveguide can be applied to a circuit only if the top and bottom layers of the substrate definition are defined as groundplanes or impedance termination. The walls of the waveguide are perfect metal. The ground planes can be defined either as perfect metals or metal with loss.

During a simulation, all current directions on the sidewalls of the waveguide. are taken into account.

To add a waveguide:

1. Choose **Momentum > Box-Waveguide > Add Waveguide**.

2. Select the direction of the waveguide. To draw the waveguide parallel to the x-axis, click **X-axis**. To draw the waveguide parallel to the y-axis, click **Y-axis**.

3. Draw the waveguide using one of the following two methods:

   • Position the mouse and click to define one wall of the waveguide. Move the mouse to the position of a point on the second wall and click.

   • From the Layout menu bar choose **Draw > Coordinate Entry** and use the *Coordinate Entry X* and *Coordinate Entry Y* fields to specify a point on the edge of the substrate. Click **Apply**. Enter the coordinates of a point on the second, parallel edge of the substrate and click **Apply**. Click **Cancel** to dismiss the dialog box.

These boundaries specify the edges of the substrate and the width of the waveguide.

## Editing a Waveguide

Once the waveguide is applied, you cannot change its dimensions. If you want to change the size, you must delete the current waveguide and add a new one.

## Deleting a Waveguide

To delete a waveguide:

1. Choose **Momentum > Box-Waveguide > Delete Waveguide**.

The waveguide is removed from the layout.

## Viewing Layout Layer Settings of a Waveguide

The waveguide is defined as a layout layer named *momentum_box*. You may review the layout layer settings, but it is a protected layer, so you should not change the settings of this layer.

To view the waveguide layer specifications:

1. Choose **Options > Layers**.

2. Select **momentum_box** from the *Layers* list. The layer settings are displayed. For more information on these parameters, refer to the *Layout* manual.

3. Click **Cancel** to dismiss the dialog box.

# About Boxes and Waveguides

There are a variety of reasons why you would want to simulate a circuit in a box or waveguide:

- The actual circuit is enclosed in a metal box
- Nearby metal sidewalls may affect circuit performance
- The box may resonate
- Propagating modes may be present

These situations are discussed next.

Circuits are often encased in metal enclosures. By adding a box to your design, the metal sidewalls that are present in the real structure may be included in the simulation. This is useful is you suspect that the presence of these sidewalls will have an immediate effect on the behavior of the circuit. For example, "broad-coupled filters" are placed in metal enclosures (a box) and the sidewalls can have an significant influence on the filter characteristics.

You may want to use a box or waveguide because metal sidewalls are present in the real structure and there may be an effect from these sidewalls on the characteristics of the circuit. This can be a parasitic, unwanted effect. If the effects of the sidewalls were not taken into account while designing the circuit, you can verify any effect that the sidewalls may have on your circuit. In most cases, when the sidewalls are not too close to the actual circuit, the effect of the sidewalls on the simulation results will be marginal. There is, however a specific, significant condition, which is unique for structures with sidewalls.

In the case of a box, this is the occurrence of one or more box resonances. A box resonance is a physical effect where, under the condition of certain frequency and box size combinations, the box actually starts resonating at a certain frequency. Because a box resonance has a significant effect on S-parameters in a (small) band centered around the box resonance frequency which cannot be represented by a smooth function, no smooth adaptive S-parameters will be available in this frequency band. When simulating such a circuit, Momentum will inform you of these box resonance frequencies, of the frequency bands where there will not be smooth S-parameters available and of the quality factor (a measure for the sharpness of the box resonance, depending on the losses).

In the case of waveguide, the effect is the excitation of a waveguide mode. If your circuit will be positioned near sidewalls, you may want to add a waveguide to determine whether they have an effect on the performance of the circuit.

You may have a situation where there are no metal sidewalls in the structure, but the substrate definition is bounded by other material (for example, you have a finite-size substrate where the dielectric material abruptly ends, so you have a dielectric-air transition). Although Momentum boxes and waveguides are defined as perfect metals and not dielectric material, you may decide that defining a metallic enclosure in the simulated circuit may be more representative of the real structure than using no enclosure.
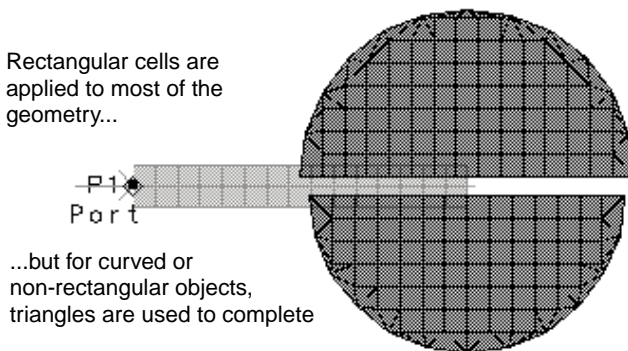
## Adding Absorbing Layers under a Cover

You may want to model your box or waveguide as having absorbing layers between the covers and the layout. You can use a substrate interface layer, define its thickness, and its absorbing properties using $\varepsilon$ and $\mu$ (make sure that $\varepsilon$ and $\mu$ are accurate specifications for this layer). Adding absorbing layers to a box would have an effect on any box resonances that would occur by producing a weaker resonance, that is, the quality factor of the box resonances would lower significantly.

## Boxes, Waveguides, and Radiation Patterns

If you have a structure enclosed in a box or waveguide and you want to calculate radiation patterns for it after the simulation, you need to set the top and bottom planes to values between 376 and 378. There are also other considerations. For more information, refer to "377 Ohm Terminations and Radiation Patterns" on page 3-27.

# Chapter 6: Mesh

A *mesh* is a grid-like pattern of triangles and rectangles, and each triangle or rectangle is a *cell*. This pattern of cells is based on the geometry of a circuit and, optionally, user-defined parameters, so each circuit will have a unique mesh calculated for it. The mesh is then applied to the circuit in order to compute the current within each cell and identify any coupling effects in the circuit during simulation. From these calculations, S-parameters are then calculated for the circuit. The mesh that was calculated for the double patch example is shown below.

Rectangular cells are applied to most of the geometry...

...but for curved or non-rectangular objects, triangles are used to complete

Creating a mesh consists of two parts:

- Defining mesh parameters
- Precomputing the mesh

It is not necessary to set up mesh parameters; default parameters will be used instead. You can precompute the mesh before simulating in order to view the mesh, otherwise the mesh will be computed as part of the simulation process. A mesh is required in order to simulate.

This chapter describes how to define mesh parameters and calculate the mesh. It also includes a description of how the mesh generator works and gives guidelines for setting mesh parameters for different applications.

# Defining a Mesh

Choose *Momentum > Mesh > Setup* to set up mesh parameters, which enable you to control the number of cells that are used to create the mesh. The more cells, the more accurate the simulation will be, but too many cells will slow down a simulation and provide little improvement in accuracy. You can choose not to set up mesh parameters, and default values will be used to create the mesh.

If you choose to define mesh parameters, you can set them for:

- The entire circuit
- The objects on a layout layer
- A single object

It is not necessary to specify parameters for all levels. You can, for example, specify mesh parameters for a single object, and use default values for the rest of the circuit.

For more information about how a mesh is generated, refer to "About the Mesh Generator" on page 6-13. For suggestions to consider when setting mesh parameters, refer to "Guidelines for Meshing" on page 6-23.

Procedures for setting up mesh parameters follow.

## Defining Mesh Parameters for the Entire Circuit

Global mesh parameters affect the entire circuit. To set up global parameters:

1. Choose **Momentum > Mesh > Setup**.

2. The Global parameters are displayed.

3. Enter the mesh frequency in the Mesh Frequency field and select the units. The wavelength of this frequency will be used to determine the density of the mesh. In general, set the value of the mesh frequency to the highest frequency that will be simulated. For more information, refer to "Adjusting Mesh Density" on page 6-14.

4. Enter the Number of Cells per Wavelength. This value will also be used to determine the density of the mesh. The relationship between wavelength and cells per wavelength can be described by example: if the circuit is 3 wavelengths long and the number of cells per wavelength is 20, the length of the circuit will be divided into 60 cells.For more information, refer to "Adjusting Mesh Density" on page 6-14.

5. Any curved areas in the circuit will be meshed using facets. In the Arc Facet Angle field, enter the number of degrees that will be included in a single facet. The maximum is 45 degrees per facet. The lower the value, the better the resolution and the denser the mesh will be. The minimum value is equal to the Arc/Circle Resolution which is set when the object is drawn. For more information, refer to "Using the Arc Facet Angle" on page 6-19.
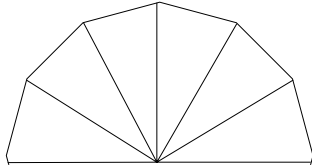


Arc Facet Angle is 30 degrees    Arc Facet Angle is 18 degrees
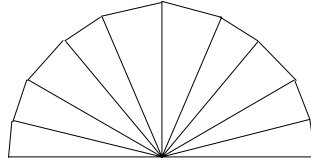
6. Enable **Edge Mesh** to add a relatively dense mesh along the edges of objects. Since most current flows along the edges of objects, the edge mesh can improve the accuracy and speed of a simulation.

If you want the edge mesh to be sized automatically, leave the Edge Width field blank. Otherwise, specify the edge width and select the units. For more information about the edge mesh, refer to "About the Edge Mesh" on page 6-15.

**Note**    An edge mesh that you specify will not be used if the edge width is larger than the cell size set by the *wavelength/number_of_cells_wavelength*. This is because such edge meshes would be very inefficient. If the edge mesh values must be used, decrease the number_of_cells_wavelength, which is specified in the *Number of Cells per Wavelength* field.

7. Enable **Transmission Line Mesh** to specify the number of cells along the width of a geometry. It is most useful for circuits with straight-line geometry.

Enter the number of cells that the width will be divided into in the Number of Cells Wide field. This will be the total number of cells along the width of the circuit. For more information about the transmission mesh, refer to "About the Transmission Line Mesh" on page 6-16.

8. Enable **Thin layer overlap extraction** in order to extract objects for the following situations:

   • Two objects on different layers overlap

   • The objects are separated with a thin substrate layer

If this is enabled, the geometry will be altered to produce a more accurate model for the overlap region. For more information, refer to "Processing Object Overlap" on page 6-21.

---

**Note**   This should always be enabled when modeling thin layer capacitors.

---

9. If you want to revert to default parameters, click **Reset**.

10. Click **OK** to accept the global mesh parameters.

## Defining Mesh Parameters for a Layout Layer

You can define mesh parameters that affect the objects on a single layout layer. If you have global parameters set, they will not be used where mesh parameters are defined for a layout layer.

To set up layout layer parameters:

1. Choose **Momentum > Mesh > Setup**.

2. Click the **Layer** tab.

3. From the Layout Layer list, select the metallization layer that you want to specify mesh parameters for.

4. Enter the Number of Cells per Wavelength. This value, along with the wavelength of the frequency specified in the Frequency field, will be used to determine the density of the mesh. (Recall that the frequency is a global parameter).

   The relationship between wavelength and cells per wavelength is best described by example. If the portion of the circuit on this layer is 2 wavelengths long and the number of cells per wavelength is 20, the length of the circuit on this layer will be divided into 40 cells. For more information, refer to "Adjusting Mesh Density" on page 6-14.

5. Enable **Edge Mesh** to add a relatively dense mesh along the edges of the objects on this layer. Since most current flows along the edges of objects, the edge mesh can improve the accuracy and speed of a simulation.

   If want the edge mesh to be sized automatically, leave the Edge Width field blank. Otherwise, specify the edge width and select the units. For more information about the edge mesh, refer to "About the Edge Mesh" on page 6-15.

6. Enable **Transmission Line Mesh** to specify the number of cells along the width of the geometry on this layer. It is most useful for circuits with straight-line geometry.

   Enter the number of cells that the width will be divided into in the Number of Cells Wide field. This will be the total number of cells along the width of the circuit on this layer. For more information about the transmission mesh, refer to "About the Transmission Line Mesh" on page 6-16.

7. If you want to change the settings for the selected layer, click **Reset** to return to default values or **Clear** to erase your settings.

8. Select another layer and repeat these steps for setting the mesh parameters.

9. Click **OK** to accept the mesh parameters for the layers.

## Defining Mesh Parameters for an Object

You can define mesh parameters that affect a single object. If you have global or layout layer parameters set, they will not be used where mesh parameters are defined for an object.

To set up object parameters:

1. Choose **Momentum > Mesh > Setup**.

2. Click the **Primitive** tab.

3. In the Layout window, select the object that you want to specify mesh parameters for.

4. Enter the Number of Cells per Wavelength. This value, along with the wavelength of the frequency specified in the Frequency field, will be used to determine the density of the mesh. (Recall that the frequency is a global parameter).

   The relationship between wavelength and cells per wavelength is best described by example. If the object is 1/2 wavelengths long and the number of cells per wavelength is 20, the length of the object will be divided into 10 cells. For more information, refer to "Adjusting Mesh Density" on page 6-14.

5. Enable **Edge Mesh** to add a relatively dense mesh along the edges of the object. Since most current flows along the edges of objects, the edge mesh can improve the accuracy and speed of a simulation.

   If want the edge mesh to be sized automatically, leave the Edge Width field blank. Otherwise, specify the edge width and select the units. For more information about the edge mesh, refer to "About the Edge Mesh" on page 6-15.

6. Enable **Transmission Line Mesh** to specify the number of cells along the width of the object. It is most useful for circuits with straight-line geometry.

   Enter the number of cells that the width will be divided into in the Number of Cells Wide field. This will be the total number of cells along the width of the object. For more information about the transmission mesh, refer to "About the Transmission Line Mesh" on page 6-16.

7. If you want to change the settings for the selected object, click **Reset** to return to default values or **Clear** to erase your settings.

8. Select another object and repeat these steps for setting the mesh parameters.

9. Click **OK** to accept the mesh parameters for the objects.

## Seeding an Object

For the majority of circuits, it is not necessary to seed an object, and it is not recommended because all internal mesh processes will be skipped. If an object is seeded incorrectly, an inefficient mesh will result, affecting simulation speed and accuracy. You should have a good understanding of electromagnetics and computational geometry to be able to use this meshing feature efficiently.

Primitive seeding enables you to specify the exact number of cells to be applied to a geometry:

- Horizontally, across the o,u-axis
- Vertically, from point V to the o,u-axis

The example below illustrates the position of the o,u-axis and the position of point V with respect to OU on a simple shape. Regardless of the object seeding is applied to, OV must be perpendicular to OU in order to create a correct mesh. This mesh was defined to be three cells wide along the o,u-axis, and five cells high.
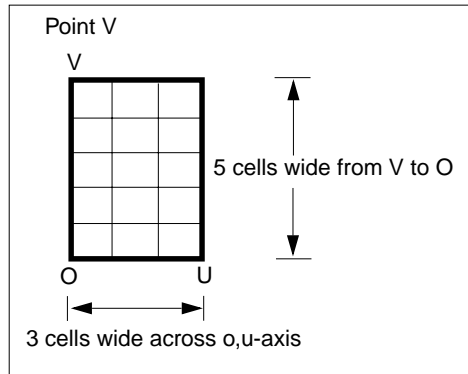


To seed a primitive:

1. Determine the method for entering mesh coordinates. You can:
   - Type coordinates directly into the Coordinate Entry dialog box
   - Select coordinates by clicking on objects in the Layout window

   If you choose select coordinates on the object, make sure that the vertex snap mode is on, and check your other snap modes to make sure that you can completely align the mesh with the structure. If snap modes are on when you select a coordinate, the system will snap to the nearest point that satisfies an enabled snap mode. This means that you may have difficulty aligning the mesh with the object if snap modes other than vertex snap are enabled. Snap modes can be found in the Layout window under the Options menu.

2. If a mesh is displayed on the layout, choose **Momentum > Mesh > Clear**.

3. Choose **Momentum > Mesh > Setup**.

4. Click the **Primitive Seed** tab.

5. In the Layout window, select the object that you want to seed. If necessary, zoom in.

6. Specify the o,u-axis. The axis determines the orientation of the mesh. Select the axis using one of the following two methods:

   • Position the mouse and click to define the origin of the o,u-axis. Move the mouse to the end of the axis and click.

   • From the Layout menu bar choose **Draw > Coordinate Entry** and use the Coordinate Entry X and Coordinate Entry Y fields to specify the origin of the o,u-axis. Click **Apply.** Enter the coordinates that define the end of the axis and click **Apply.**

7. The Primitive Mesh Control dialog box is displayed. Enter the number of cells that you want along the o,u-axis.

8. Specify point V. The point must be positioned so that OV is perpendicular to OU. Select the point using one of the following two methods:

   • Position the mouse and click to define the point and click.

   • From the Layout menu bar choose **Draw > Coordinate Entry** and use the Coordinate Entry X and Coordinate Entry Y fields to specify the coordinates of point V. Click **Apply.**

9. The Primitive Mesh Control dialog box is displayed. Enter the number of cells that you want along the ov-line.

10. The following information will appear for the seeded object:

   • The number and name of the layout layer that the object is on

   • Data describing the seeded object, including the type of object (for example, path or polygon) and its coordinates

   • The angle of the o,u-axis with respect to the x-axis

   • The length of each cell along the o,u-axis (the distance of OU divided by the number of cells)

   • The length of each cell along OV (the distance of OV divided by the number of cells)

An example is shown below:

```
layer data ── Layer 2 (bot_met):
object data ──Path  (-35.000000, -2.187000) (0.000000, 2.187000)
  OU angle ──── OU angle = 0.000000 deg.
  cell sizes ────── du = 3.483500 mm, dv = 17.632979 mm
```

11. If you want to change the settings for the selected object, click **Clear** to erase your settings.

12. Click **OK** to complete the command.

In the illustration below, the o,u-axis is positioned along the base of the polygon, and it is the length of the polygon. The number of cells along the axis is set to three. The number of cells from OU to V is set to five. The resulting mesh is displayed. Any other mesh parameters that had been specified were used in other parts of the circuit.



Remaining area of object is meshed by connecting vertices.

V

U

O

U to V is 5 cells wide.

O to U is 3 cells wide. Also, OU determines the orientation of the mesh.
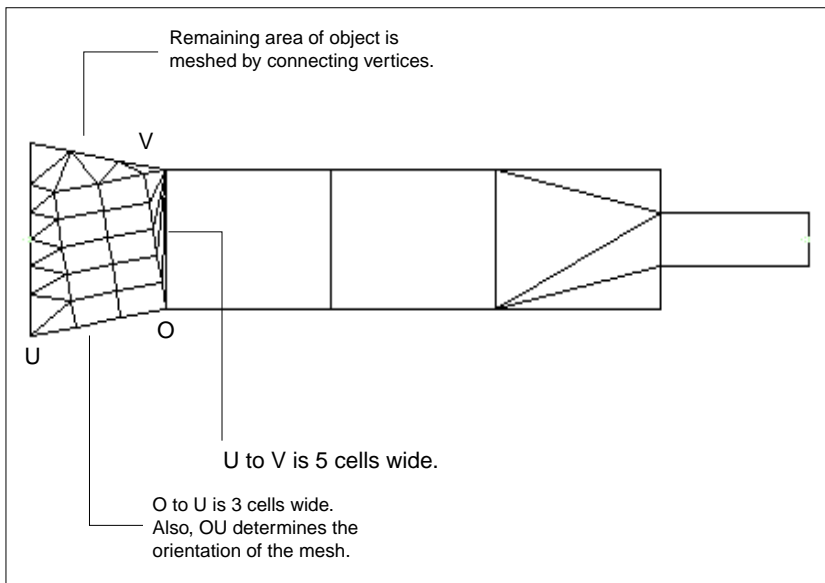
## Mesh Connectivity

If you choose to seed an object, and you have global, layout layer, or object mesh parameters set, these other parameters will not be used on the seeded object. In addition, the mesh cells must connect in an appropriate way throughout the circuit in

order to achieve accurate results. This is referred to as *mesh connectivity.* Be aware that in order to achieve mesh connectivity, the boundaries of seeded regions may be transformed. The only way to correct this is to modify the seeding of the neighboring objects. If you choose to seed one object, you should consider seeding the entire circuit.

Which cells are altered to assure connectivity is governed by creating a mesh that generates the fewest number of unknowns.

## Resetting and Clearing Mesh Parameters

If you want to change mesh parameter setting, you can:

- Change individual parameters
- Erase a set of mesh parameter values
- Reset mesh parameters to default values

To reset or clear mesh parameters:

1. Choose **Momentum > Mesh > Setup**.
2. Select the tab, Global, Layout, Primitive, or Primitive Seed, that contains the parameters you want to erase or reset.
3. Choose one of the following options:

   - For layout parameters, select the layout layer name.
   - For object mesh parameters or seeding, select the object from the Layout window.

   ---

   **Tip**   You may want to clear the mesh from the layout before attempting to select an object.

   ---

4. Click **Clear** to erase the settings, or click **Reset** to set them to default values.

---

**Note**   If you have trouble selecting an object, exit the dialog box, select the object, then reopen the dialog box.

---

# Precomputing a Mesh

**Note** In order to calculate a mesh, you must first precompute the substrate and apply ports to the circuit. For information on how to precompute the substrate, refer to Chapter 3, Substrates. For information on how to apply ports, refer to "Adding a Port to a Circuit" on page B-4, and to Chapter 4, Ports.

Choose *Momentum > Mesh > Precompute* to calculate the mesh. Using the specified frequency and any user-defined mesh parameters, a pattern of rectangles and triangles is computed and applied to the circuit. During a simulation, the surface current in each cell is calculated, and this information is then used to solve S-parameters for the structure.

To precompute the mesh:

1. Choose **Momentum > Mesh > Precompute**.

2. In the Precompute Mesh dialog box, enter the frequency you want to use for precomputing the mesh. The wavelength of this frequency will, in part, determine the number of cells in the mesh. For more information about mesh calculations, refer to "About the Mesh Generator" on page 6-13.

3. Click **OK** to start calculating the mesh.

When the computations are complete, the mesh will be displayed on the layout.

## Viewing Mesh Status

After mesh computations have started, any messages regarding the computations will appear in the Momentum Status window. Messages usually refer to any errors or warnings, or indicate when the mesh is complete.

If you close the status window and want to reopen it, from the Layout menu bar choose *Window > Restore Status*.

# Stopping Mesh Computations

To stop the mesh process:

1. From the Momentum Status window, choose **Simulation/Synthesis > Stop Simulation**.

The mesh computations will stop and no information will be saved.

# Viewing the Mesh Summary

If the mesh computation is successful, you can view mesh statistics. Some of the information returned includes:

- The number of rectangular cells in the structure
- The number of triangular cells
- The number of vertical rectangular cells used to model the vias
- The number of unknown currents
- Computer resources used and time to solve

The number of cells and the number of unknown currents will affect simulation time. The more cells, the denser the mesh, and the longer the simulation will be. Each unknown current is a variable in the matrix equation that is solved by the simulator. The greater the number of unknown currents, the larger the matrix, and the longer the simulation time will be. The memory required to solve the structure increases as well.

An example of mesh statistics follows.

```
===========================================================
                 --- MOMENTUM MESH STATISTICS ---
===========================================================

                     MOMENTUM 4.0 (*) 150.100
                     Thu Oct 12 15:50:17 2000

-----------------------------------------------------------

Mesh Info:
---------
Rectangular cells :        4
Triangular cells  :       51
Via cells         :        0
Unknown currents  :       60

Resources:
---------
Process size :      0.17 MB
User Time    :    0h 0m 0s
Elapsed Time :    0h 0m 5s

===========================================================
```

# Clearing a Mesh

You can delete the view of the mesh from the circuit. Clearing a mesh only deletes the view from the Layout window; it does not erase the mesh calculations.

To remove the mesh from the display:

1. Choose **Momentum > Mesh > Clear**.

2. If you want to redisplay the mesh, choose **Edit > Undo**. Otherwise, you must recompute the mesh in order to view it.

# About the Mesh Generator

The mesh generator provides the algorithm that divides the circuit geometry into cells (rectangles and triangles). In RF simulation mode, the mesh is automatically reduced to remove slivery cells of low quality and to remove electromagnetically redundant cells resulting in a mesh with polygonal cells. By discretizing the surface, it is possible to calculate a linear approximation of the arbitrary, varying surface current. The linear approximation involves applying a rooftop function to the cells. The better the linear approximation fits the actual current, the more accurate the results. Thus, a dense mesh generally provides better results at microwave frequencies.

The mesh generator attempts to apply an optimal pattern of cells so that an accurate simulation can be achieved with a minimal number of cells. In Momentum mode, the

minimal number of cells can be rather high, resulting in a dense mesh. For a given mesh density, the mesh reduction technology used in the RF mode achieves an optimal mesh with significantly fewer cells, hence improving the simulation performance with reduced memory usage and simulation time. Since the current in each cell is calculated in a simulation, a very dense mesh can increase simulation time.

The mesh parameters that you specify provide the mesh maker with the information required to divide your geometry into the various cell shapes and sizes. If you do not want to set mesh parameters, default ones will be used instead. In Momentum mode, the default mesh setting is 20 cells/wavelength using edge mesh. In RF mode, the default mesh setting is 20 cells/wavelength *not* using edge mesh.

## Adjusting Mesh Density

Two mesh parameters, *Frequency* and *Cells per Wavelength*, are used in combination in determining mesh density.

Using the wavelength of the frequency, a linear function is approximated, also referred to as a rooftop basis function. The higher the frequency, the more wavelengths fit across the structure. The Cells per Wavelength are the minimum number of cells that fit under each wavelength. The more cells, the better the sinusoid is represented, and the more accurate the simulation is. For example, if you use 30 cells per wavelength, the maximum deviation between the sinusoid and the linear approximation is about 1%. These parameters affect longitudinal current.

Higher frequencies will result in a greater number of cells (increased density) for a mesh. Similarly, increasing the minimum number of cells per wavelength will also increase the density. In general, for optimal density, it is better to increase the number of cells per wavelength rather than increasing the mesh frequency. The optimal value for the mesh frequency is the highest frequency that will be simulated. This may avoid having to recalculate the substrate frequency band if it is not sufficient.

If you enter a value for the mesh frequency, but you have already seeded the layout, the system will ignore the frequency you enter and the mesh frequency will only be used to compute the mesh for the remaining areas of the circuit.

You should be aware that the value entered for the number of cells per wavelength is only a lower limit and not the exact value used by the mesh generator. The following discussion explains how the actual number of cells used by the mesh generator may be greater. For example, suppose you enter 20 for *Cells per Wavelength*. In general,

the cell size used when the geometry is infinitely large corresponds with the number of cells per wavelength. But when other details, edges, and user-defined mesh settings are included, cells might appear that are smaller than $\lambda/20$ resulting in more cells per wavelength than the value of 20 that you entered. Here are two specific cases where the actual number of cells per wavelength is greater:

- When the layout has details that are smaller than $\lambda/20$, the mesh follows the shape of the details. Since the mesh consists of triangles and rectangles only, the cells will be less than $\lambda/20$.

- When you change the default settings in the Mesh Setup Controls dialog box, such as the number of cells per transmission line width, then you directly influence the number of cells over the width of the transmission lines. This might lead to cells that are smaller than $\lambda/20$.

## About the Edge Mesh

The edge mesh feature automatically creates a relatively dense mesh pattern of small cells along the edges of metal or slots, and a less dense mesh pattern of a few large cells in all other areas of the geometry. Because most of the current flow occurs along the edges of slots or metals, the edge mesh provides an efficient solution with greater accuracy.

Use the edge mesh to improve simulation accuracy when solving circuits where the modeling of current flow in any edge area is a critical part of the solution. This includes circuits where the characteristic impedance or the propagation constant are critical for determining the electrical model, circuits in which close proximity coupling occurs, or circuits where edge currents dominate the circuit behavior. Applications for using the edge mesh include:

- Tightly coupled lines

- Patch antennas

- Resonant circuits

- Delay lines

- Hairpin filters

Edge mesh is available for all levels of meshing: global, layout layer, and object meshing.

### Setting the Width of the Edge Mesh

When using an edge mesh, you can either specify the edge mesh width or leave it blank. If you leave it blank, an appropriate width will be determined and used to create the mesh. One way to get a visual approximation is to position the mouse on the primitive edge and click, then move the mouse inward and note the differential value., which is displayed at the bottom of the Layout window.

```
-85.000, -40.000        -20.000, -50.000      mm    A/RF
```

differential coordinates
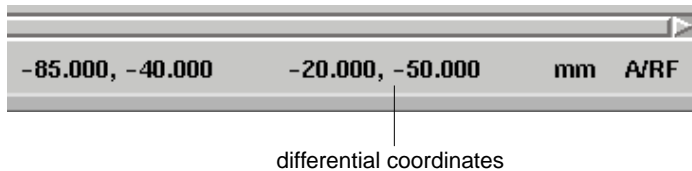
## About the Transmission Line Mesh

Use the transmission line mesh when you want to specify the number of cells between parallel lines in a layout. This feature can save computation time and memory because it will create a mesh that is appropriate for straight line geometry.

For example, the simulation results for a single transmission line with one or two cells across the width will be equal. If you have coupled lines, the results will differ.

## Combining the Edge and Transmission Line Meshes

Both transmission line mesh and edge mesh can be used together. The total number of cells wide will be the total number of transmission line cells plus edge cells. The minimum value permitted when using this combination is 3.

Basic Edge Mesh

cells/width = 3

Cell width along the length
will be equal for this
type of mesh.

Transmission Line Mesh

cells/width = 3

Edge Mesh and Transmission Line Mesh

cells/width = 5
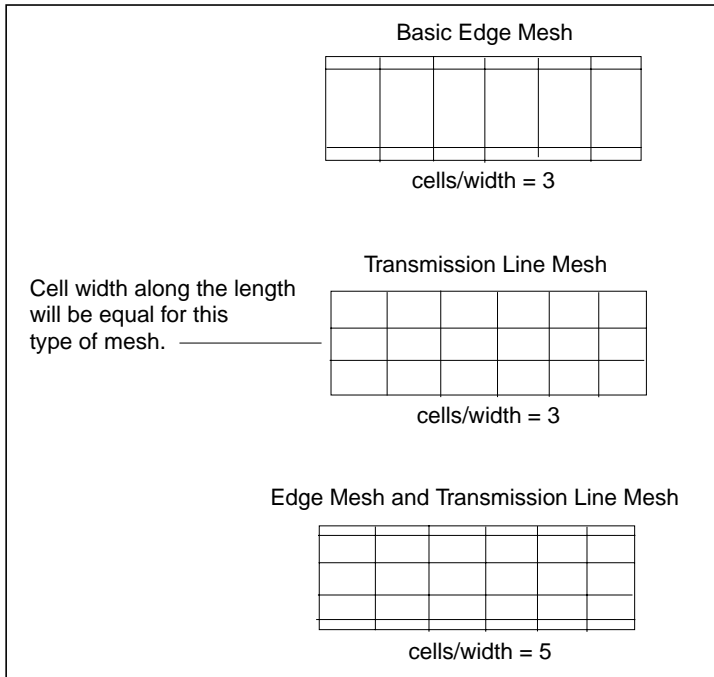
The edge and transmission line mesh affect the transverse current in the circuit. The
more cells specified using edge mesh and transmission line mesh, the more accurate
transverse current approximations will be.

The plot below illustrates the hairpin example simulated with and without and edge mesh. Although the simulation that includes the edge mesh takes more time, the results are closer to the actual measured results of the filter.
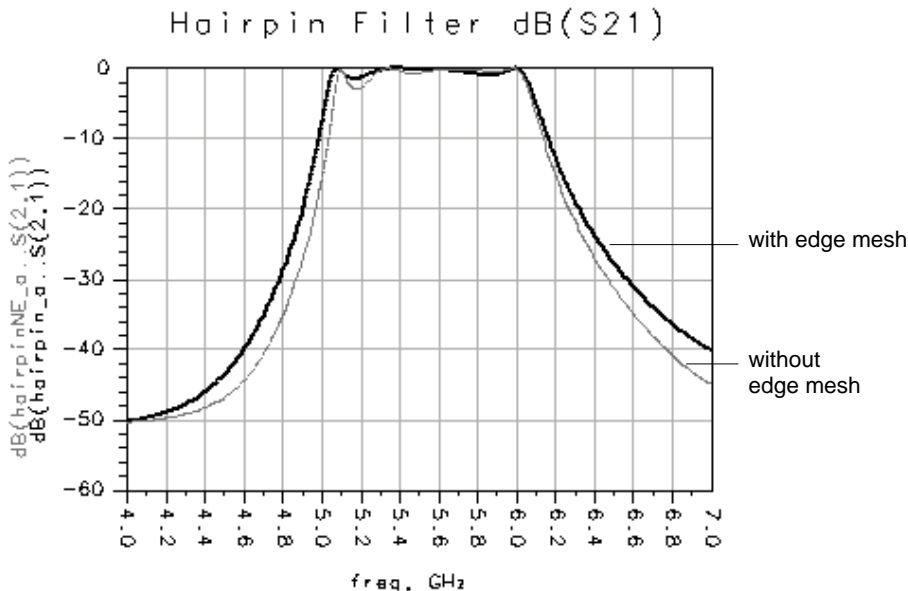


## Mesh Patterns and Simulation Time

The mesh generator is relatively fast, even for complex meshes. But be aware that the complexity of the mesh can have a significant on effect simulation time. In particular, the number of cells in the mesh (especially triangular cells) and the number of unknown currents, can affect simulation time.

Simulation times will be faster when the mesh consists mostly of rectangular cells. A simple mesh is one consisting of a few rectangles of similar size. A worst-case complex mesh would consist of many dissimilar triangles. You should try to minimize the number of triangular cells in the mesh.

The second variable, number of unknown currents, depends on the density, or total number of cells in the mesh. You should try to minimize the total number of cells in the mesh. This reduces the number of unknown currents, and thus reduces the size of the matrix to be solved during simulation. The amount of time required to solve the matrix is N*N*N, where N is the number of unknown currents.

For example, a thru-line at a global seeding of 30 might produce a 50 x 50 matrix to solve. Since the time required to solve the matrix is proportional to the matrix size, the same thru-line at a seeding of 60 might produce a 200 x 200 matrix because number of cells would be twice as dense in both X and Y dimensions. This means that it will take longer to solve.

Adjustments to your mesh parameters can be made to control complexity of the mesh and, in turn, the amount of time required to simulate.

For complex circuits, where simulation times may be long even with a simple mesh, reducing the complexity of a mesh can save significant amounts of simulation time.

## Mesh Patterns and Memory Requirements

The number of unknown currents, which is calculated when the mesh is precomputed, is based on the complexity of the mesh. The more unknowns, the larger the matrix that must be solved during simulation. The amount of memory required to solve is proportional to N*N, where N is the number of unknown currents.

## Using the Arc Facet Angle

The *Arc Facet Angle* parameter enables you to control the number of triangular cells that appear in the mesh. Triangular cells are used to generate a mesh on the curved area of an object. The larger the value in the Arc Facet Angle field, the fewer the number of triangular segments will be used on the curved object.

The maximum value for the Arc Facet Angle is 45 degrees. The minimum value depends on the original facetting used when the curved object was drawn. For example, the Double Patch Antenna is drawn with an Arc/Circle Resolution of 30 degrees, and the Arc Facet Angle is set to 45 degrees. The actual value used to refacet the object will be in between these values. The angle is chosen so that the facet length is smaller or equal to the cell size, so refacetting will not affect the cells per wavelength that is specified for computing mesh density.

---

**Note** In order for the mesh generator to produce an accurate mesh, the mesher must be able to recognize the polygon translation it receives as an arc. Therefore, a poorly defined layout resolution may prevent arc recognition and by that arc refacetting. For more information on layout resolution, refer to "Mesh Precision and Gap Resolution" on page 6-25.

---

The VIASTUB example is shown below, the Arc Facet Angle is set to 45 degrees, the coarsest setting, and 0.01degrees, which effectively turns off refacetting. The mesh for a stub based on each angle setting is displayed. The simulation for the denser mesh was longer, required more memory, and produced little improvement in the results.
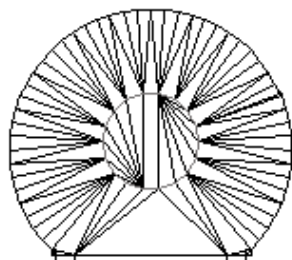


arc facet angle is 0.01          arc facet angle is 45



results are identical
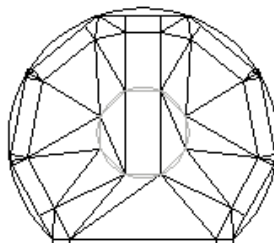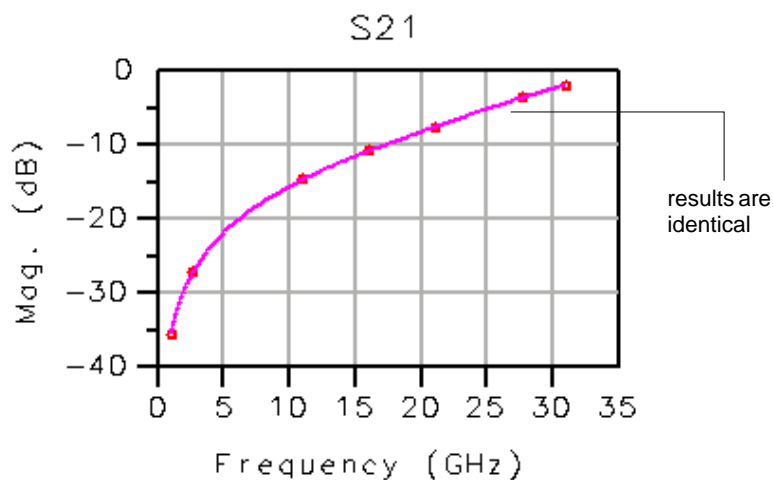
| | Arc Facet Angle = 45 | Arc Facet Angle = 0.01 |
|---|---|---|
| **Process size** | 6.18 MB | 12.791 MB |
| **User time** | 12 min | 20 min |
| **Elapsed time** | 15 min | 24 min |
| **Rectangular cells** | 96 | 80 |
| **Triangular cells** | 108 | 200 |
| **Via cells** | 16 | 36 |
| **Unknown currents** | 337 | 453 |

## Processing Object Overlap

The parameter *Thin layer overlap extraction* should be used in designs that include thin layers that are both close together and overlapping. It is possible that mesh cells can be generated that cross the overlap region, and this is not desirable.

As an example, consider two layers, one above the other, and close together. They carry a large charge density where they overlap, which increases proportionally to 1/distance. If there is no metal above the layers, the charge density is nearly zero where there is no overlap, and the change is very rapid—nearly a step—at the point of overlap.
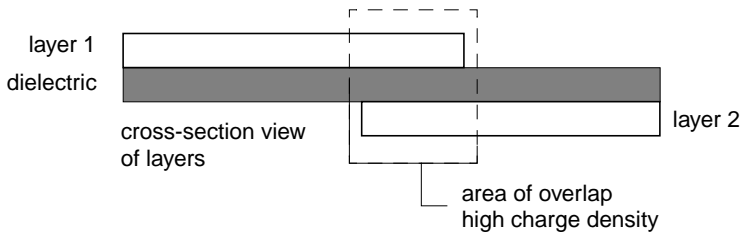


If no overlap extraction is performed, the mesher can create cells that will cross the border of the overlap. In the actual circuit, this cell would have a partially high charge density area where the overlap is present, and a partially low charge density area where there is no overlap. However, Momentum simulates cells with a *constant* charge density, and approximating the strongly-varying charge in the cell with a

constant is not an adequate representation. Thus, by enabling thin layer extraction, no cells will be created that cross an overlap region.



mesh cell crosses
into area of overlap

top views of layer 1

with overlap extraction,
cells are resized, there
is no crossover into
area of overlap
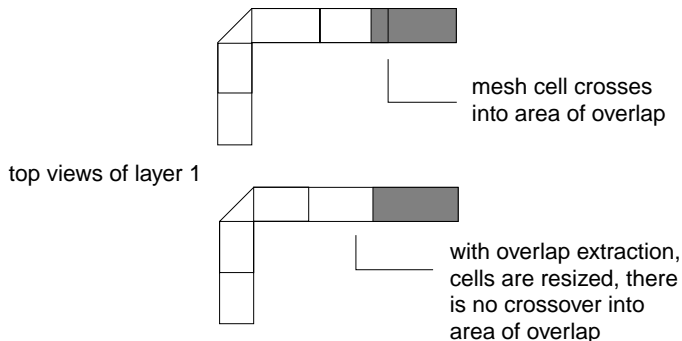
Because enabling this feature for all possible overlap in a circuit would create too dense a mesh, some empirical rules are implemented that switch the overlap extraction off if the layers are not close (h > 0.02 estimated cell size) or if the overlap area is small. To calculate the estimated cell size, the mesh frequency, the global number of cells per wavelength, and an estimation of the effective refractive index of the substrate are used.

If you are calculating your mesh at low frequencies, the estimated cell size is high, and you may not want thin layer overlap extraction. In this case, disable the Thin layer overlap function.

## Mesh Generator Messages

When a mesh is generated, you may see a message similar to this:

On some layers, the mesher used the maximum allowable cell size for the given substrate. You will not be able to generate a coarser mesh on these layers.

This means that the coarsest mesh has been achieved for the objects on certain layers. This is because the maximum cell size has been limited due to the accuracy of the Green's functions that were calculated for the substrate.

For example, consider a layout with 20-cell mesh when the mesh frequency is 10 GHz. Generally, you would get a 10-cell mesh if you set the mesh frequency to 5 GHz. But if, with the lower mesh frequency, the cell size would be larger that the maximum

allowed (the limit being based on the accuracy of the Greens' function of the substrate), the mesh would contain more than 10 cells.

# Guidelines for Meshing

The default mesh will provide an adequate and accurate answer for many circuit applications. For most other circuit applications, the using the global mesh parameters will be all that is needed to provide greater accuracy. In a few special cases, you may need to use the layer or primitive mesh controls.

For applications such as highly accurate discontinuity modeling or for geometries that have tightly coupled lines, the default mesh may not be dense enough in particular areas to provide enough accuracy. In such cases, edge mesh should be used or the mesh density must be increased. For example, noise floor and dynamic range are typically small numbers. In some cases, geometry solutions may show a low value of S21, like -60 dB, using a dense mesh. Such values are different for a default mesh, which may result in -40 dB for the same circuit.

In any one design, you can use any combination of the four types of mesh control. In general, greater mesh density can provide more accuracy. But greater density takes more computation time (more cells to solve) and, additionally, triangles always take longer to compute than rectangles.

## Meshing Thin Layers

Thin layers must be meshed so that the mesh cells are entirely within or entirely outside of the overlapping area. If the mesh cells and object boundaries are not aligned correctly, the simulation data may be less accurate.

For more information, refer to "Processing Object Overlap" on page 6-21.

## Meshing Thin Lines

When the geometry has narrow lines, like thin transmission paths in a spiral, it may be difficult to have a mesh that is more than one cell across the width if the default global mesh is used. If needed, use Edge Mesh or Transmission Line Mesh to capture the current distribution across the line.

## Meshing Slots

Slots should be meshed exactly the same as strips—there is no difference. For example, the edge mesh can be used for slots because the current distribution is basically the same, that is, it is concentrated on the edges of a slot.

## Adjusting the Mesh Density of Curved Objects

Vias or other curved objects, when drawn in a layout, have a default value for the number of facets used to draw the object, based on the command *Options > Preferences> Entry/Edit >Arc/Circle Radius (degrees)*. If the value is small, a relatively large number of facets are used to draw the circle, and this can result in more triangular cells created the curved areas during the mesh process. To change the number of facets for all arcs and circles, use the command shown above. To change them for a single object, select the object, then choose *Edit > Modify > Arc/Circle Resolution*. Then increase the number of degrees for the radius, as desired, to reduce the number of triangles. Be aware that this does alter the geometry of the circuit.

For information on how the mesh parameter Arc Facet Angle affects a mesh, refer to "Using the Arc Facet Angle" on page 6-19.

## Discontinuity Modeling

A microstrip transmission line with a bend, using the default mesh, may have a cell size equal to the width of the line (one cell per line-width). If it is long and the bend is not severe, then the default mesh may be adequate because the discontinuity is proportionally small compared to the line length. However, if the reference planes are moved inward or if the bend is more severe, the discontinuity and resulting parasitics are in greater proportion to the rest of the line. In that case, the default mesh may result in simulation data with an error.

To correct this inaccuracy, the mesh should be increased and edge mesh used. When the area near the discontinuity is meshed so that the cell size is equal to a third of the line-width (three cells per line-width) the resulting error is reduced. The denser mesh allows for current crowding (parasitic series inductance) at the interior corner of the bend and charge build-up (parasitic shunt capacitance) at the outer edge of the bend.

## Solving Tightly Coupled Lines

An example of a tightly coupled line would be a microstrip coupled line filter on a 25 mil Alumina substrate (Er = 9.8) where the line width is 25 mils and the separation between the lines is 2.5 mils. The filter response is sensitive to coupling between the lines and the default mesh is 1 cells per line-width. In this case, the reflection coefficient data does not match measurement data. To solve this design properly, edge mesh must be used.

## Editing Object Seeding

The recommended way to seed primitives is to use the o,u,v-coordinate system:

1. Select the object.
2. Choose **Momentum > Mesh > Setup**.
3. Click **Primitive Seed**.
4. Click **Clear**.
5. Select new seed parameters.

It is possible to modify these values by editing the properties of the object. This method is not supported or recommended. Select the object, then choose *Edit > Properties*. Seed values are displayed and can be changed.

## Mesh Precision and Gap Resolution

This section will apply mainly to designs that are imported into Advanced Design System.

Mesh resolution is directly related to the Layout Precision dbu (data base units) value. All gaps or other unresolved layout vertices will be resolved by the mesh generator if their distance is less than 2 dbu.

Use the Layout command *Options > Preferences > Layout Units* to change the units for your fabrication or layout process as needed.

Gap is 0.1 mil actual.

EXAMPLE: Precision dbu set to 0.01mil.

Mesh recognizes gap because precision is within the dbu units setting.

EXAMPLE: Precision dbu set to 1 mil.

Gap is collapsed in the mesh because the dbu precision value is greater than the gap.
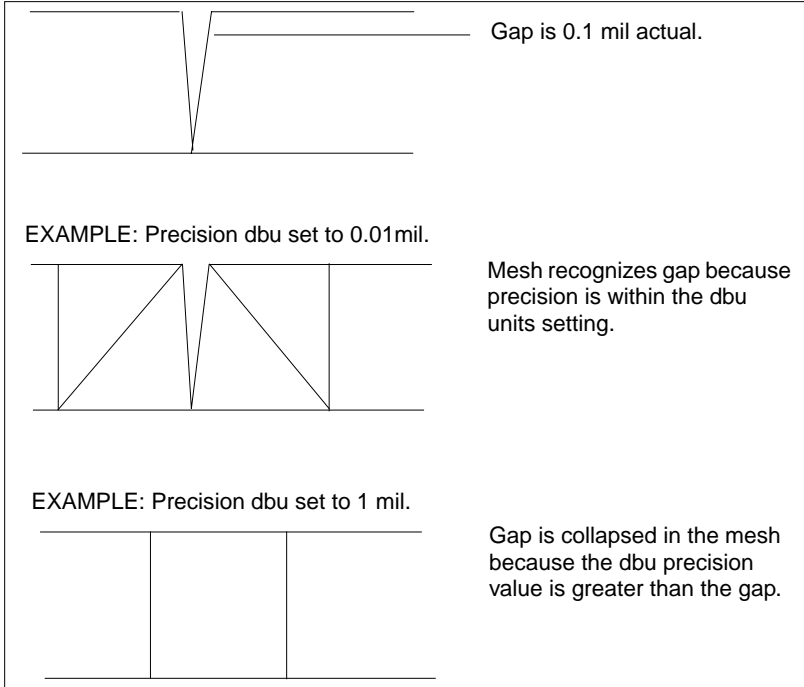
# Chapter 7: Simulation

The simulation process combines the Green's functions that were calculated for the substrate of a circuit, plus mesh information that was calculated for the circuit, and solves for currents in the circuit. Using these current calculations, S-parameters are then calculated for the circuit.

Prior to running a simulation, the following criteria must be met:

- A substrate definition must be specified for the circuit
- The circuit must include at least one port
- A mesh must be defined for the circuit
- A simulation frequency plan must be specified

If any one of the above criteria is not met, Momentum will report the error if you try to run a simulation.

The Momentum planar solver uses information from the substrate database and the mesh generator to perform the circuit simulation. The substrate calculations and mesh may be computed prior to running the simulation to ensure they are valid. This will reduce simulation time.

The steps for performing a simulation include:

- Specifying and editing frequency plans
- Selecting a process mode
- Specifying solution files
- Electing to view data
- Running the simulation

The steps for each part of this procedure follows.

---

**Note**   Momentum creates several files during simulation. One of these files, **../mom_dsn/proj_cvi** can be large. (This file contains the computed currents.) After you are finished with a particular project, you may want to delete this file to free disk space.

---

# Setting Up a Frequency Plan

You can set up multiple frequency plans for a simulation. For each plan, you can specify that a solution be found for a single frequency point or over a frequency range. You can also select one of several sweep types for a plan. This collection of frequency plans will be run as a single simulation.

To set up a frequency plan:

1. Choose **Momentum > Simulation > S-parameters**.

2. Select a sweep type. The choices are:

    • Adaptive

    • Logarithmic

    • Linear

    • Single point

   *Adaptive* is the preferred sweep type. It uses a fast, highly accurate method of comparing sampled S-parameter data points to a rational fitting model. The value entered in the Sample Points Limit field is the maximum number of samples used in an attempt to achieve convergence. The solutions from the final attempt will be saved. If convergence is achieved using fewer samples, the solutions are saved and the simulation will end. For more information about this sweep type, refer to "About Adaptive Frequency Sampling" on page 7-20.

   *Logarithmic* simulates over a frequency range, selecting the frequency points to be simulated in logarithmic increments. Type the start and the stop frequencies in the Start and Stop fields, and select frequency units for each. Enter the number of frequency points to be simulated per decade of frequency in the Points/Decade field, and select units.

   *Linear* simulates over a frequency range, selecting the frequency points to be simulated in linear increments based on the step size you specify. Type the start and the stop frequencies in the Start and Stop fields, and select frequency units for each. Enter the step size in the Step field, and select units.

   *Single* simulates at a single frequency point. Type the value in the Frequency field and select the units.

3. When you are finished setting the plan, click **Add to Frequency Plan List**.

> **Note**  In a simulation, frequency plans are executed in the order in which they appear in this list.

4. Repeat these steps to insert additional frequency plans.

5. When you are finished, click **Apply**.

## Considerations

In general, the Adaptive sweep type is the preferred sweep type. The exception is cases where a specific frequency point or points must be simulated, such as the need to view antenna patterns. Since radiation patterns can be calculated only at solved frequencies, you may want to specify a single specific frequency using the Single sweep type, or a range using Linear or Logarithmic sweep types.

If you use one or more sweep types, make sure to set up your frequency plan list so that plans with Adaptive sweep types are at the top of the list. Any additional plans that use other sweep types should follow these plans.

## Editing Frequency Plans

You can modify, delete, or move frequency plans in the Frequency Plans list.

To modify a frequency plan:

1. Choose **Momentum > Simulation > S-parameters**.

2. Select the frequency plan from the Frequency Plans list.

3. Edit the plan using the sweep type and frequency fields displayed under Edit/Define Frequency plan. These fields are described in the previous section.

4. When you are finished editing the plan, click **Update**.

5. Click **Apply** to save the changes.

To delete a frequency plan:

1. Choose **Momentum > Simulation > S-parameters**.

2. Select the frequency plan from the Frequency Plans list.

3. Click **Cut**.

4. Click **Apply** to save the changes.

To move a frequency plan:

    1. Choose **Momentum > Simulation > S-parameters**.

    2. Select frequency plan from the Frequency Plans list.

    3. Click **Cut**.

    4. Select the frequency plan that you want positioned below the plan that you cut.

    5. Click **Paste** to reenter the plan.

    6. Click **Apply** to save the changes.

---

**Tip**   You can perform several editing tasks, then click **Apply** once before you dismiss the dialog box.

---

# Selecting a Process Mode

You can select whether the simulation is to be performed in the foreground or in the background. The *Foreground* mode will pause a simulation if the substrate has not yet been precomputed. In this case, you can choose to stop the simulation or, if you choose to continue, the substrate will be precomputed and then the simulation will be performed. The *Background* mode automatically precomputes the substrate if necessary, without prompting you to stop or continue the simulation, then performs the simulation. In either mode, the Layout window is available to build or edit layouts. For more information on precomputing a substrate, refer to "Precomputing the Substrate Function" on page 3-19.

To set the process mode:

    1. Choose **Momentum > Simulation > S-parameters**.

    2. Select the process mode: choose either Foreground or Background.

    3. Click **Apply**.

# Reusing Simulation Data

To reduce the time required to complete the current simulation, you can use sampled data from the previous simulation. This function is useful if you extend or shift the frequency plan and resimulate, or if you want more samples near a resonance frequency. If you change the circuit geometry, substrate, ports, or mesh parameters, simulation data cannot be reused.

---

**Note** Momentum does not verify that the design has not changed. Make sure that only the frequency plan has changed before attempting to reuse data.

---

Enabling the function reuses previously calculated S-50 parameters.

To reuse data from the last simulation:

1. Choose **Momentum > Simulation > S-parameters**.

2. Enable **Reuse files from previous simulation** if you want to use data from the previous simulation. Sampled data from the previous simulation will be used in the current simulation, which can decrease simulation time.

3. Click **Apply**.

# Saving Simulation Data

A dataset stores the solutions for one simulation only. If you perform multiple simulations and want to save the data from previous simulations, you must specify a unique dataset name for each simulation.

One simulation can produce one or two datasets:

- Data at the frequency points computed by the simulator are stored in <project_name>.*ds*. It is the standard dataset created from any simulation.

- If the *Adaptive* sweep type is used to simulate, data computed by AFS at each sampled frequency point is stored in the dataset <project_name>_*a.ds*.

If you change the name in the Dataset field, the datasets will use the new name and not the default *project_name*.

To specify the dataset:

1. Choose **Momentum > Simulation > S-parameters**.

---

2. The name of the default dataset is displayed in the Dataset field. This is the file in which the simulation data will be stored. The default name is based on the name of the current project.

   If you want to use a different dataset, either type the name of a new dataset into the Dataset field, or click **Browse** to reuse a dataset. If you choose **Browse**, a list of the datasets in the *data* folder of the current project are displayed. Select file name and click **OK**.

Dataset files use the extension *.ds* and are stored in the *data* folder of the current project.

3. Click **Apply** to accept the changes.

## Saving Data for Export

Momentum simulation data can be saved to a file for use in IC-CAP. Momentum design geometry and setup information can be saved for import into Agilent HFSS *without* running a simulation first. For more information, refer to Appendix C, Exporting Momentum Data.

# Viewing Results Automatically

Simulation data is saved in datasets, so you can view simulation results at your convenience. If you choose to view results immediately after the simulation is complete, enable *Open data display when simulation completes*. A data display window containing default plot types or the data display template of your choice will be automatically opened when the simulation is finished. The defaults include magnitude and phase plots of S-parameters, and S-parameters plotted on Smith charts.

If you are using the Adaptive sweep type for your simulation, the data from both the standard and adaptive datasets are displayed for comparison. Data from the standard dataset will be displayed as discrete points, the adaptive sweep data will be displayed as a linear trace.

To open a data display window:

1. Choose **Momentum > Simulation > S-parameters**.

2. Enable **Open data display when simulation completes.**

3. If you want a template to be loaded into the window, type the name of the template in the Template field or click **Browse**. A list of templates in the *data* folder of the current project are displayed. Select a template and click **OK**.

---

**Note**   If you want to use a template from another project or location, copy the template into the *data* folder of the current project.

---

4. Click **Apply** to accept the changes.

# Starting a Simulation

The simulation process solves for currents by combining the Green's functions that were precomputed for the substrate and the mesh calculations. S-parameters are then calculated and saved to the dataset.

To start a simulation:

1. Choose **Momentum > Simulation > S-parameters**.
2. Verify that at least one frequency plan appears in the Frequency Plans list.
3. Check that the Process Mode, Solution Files, and Data Display options are set to your specifications.
4. Click **Simulate**.

Any messages or errors that occur are displayed in the Momentum Status window. To indicate progress, a line of dots appears across the Status window if the number of unknowns in the simulation is greater than 500; otherwise, only the frequencies are displayed.

If the simulation fails for a reason related to the setup, the partial data is saved and a message appears in the Status window.

## Viewing Simulation Status

After a simulation is started, any messages regarding the simulation will appear in the Momentum Status window. Messages usually refer to any errors found, the percent of completion, and simulation completion.

If you close the status window and want to reopen it, from the Layout menu bar choose *Window > Restore Status*.

### Understanding the "% Covered" Status Message

This message is generated when running Adaptive sweep types. It indicates the percentage of the total simulation frequency range that has been accurately modeled by rational functions. When an Adaptive sweep starts, the AFS process attempts to represent all S-parameters with a number of rational functions. New sample points are selected adaptively (by minimizing global error criteria). If the frequency range is too large, the frequency range will be split into sections. If the fitting model is accurate and stable within a subrange, the "% covered" message will be updated. Thus, the message indicates the percentage of the total simulation frequency range that has been accurately modeled by rational functions.

The AFS process always looks at the whole frequency range and adds samples where the fitting errors are maximal (a global error reduction approach). If the fitting converged in a section of the frequency range, it is likely to converge in the remaining sections, and it will not take many extra samples to achieve full convergence. This is why the percent completed message updates very slowly at first, but updates quickly when the process is nearly complete.

# Stopping a Simulation

To stop a simulation:

1. From the Momentum Status window, choose **Simulation/Synthesis > Stop Simulation**.

   The simulation will stop and the partial data will be saved. To reuse the saved data:

   • Choose **Momentum > Simulation > S-parameters**, enable **Reuse files from previous simulation**, click **Apply**, then click **Simulate**.

# Viewing the Simulation Summary

As the simulation progresses, you can view substrate, mesh, and solution statistics. Some of the information returned includes time to solve, the resources required, cell information, and any messages that were displayed in the status window. An example of solution statistics is given next.

Choose *Momentum > Simulation > Summary* to view the data. Click *Refresh* to update the statistics as the simulation progresses. The following example shows a (partial) summary displayed when using the Momentum microwave simulation mode.

```
========================================================================
                  --- MOMENTUM GENERATION STATISTICS ---
========================================================================
                        Momentum 4.0 (*)  150.999
                        Tue Jun 12 16:06:24 2001
------------------------------------------------------------------------
TOTALS
------------------------------------------------------------------------

Process size :     2.16 MB
User Time    :    0h01m11s
Elapsed Time :    0h01m13s
========================================================================
                     --- MOMENTUM MESH STATISTICS ---
========================================================================
                        Momentum 4.0 (*)  150.999
                        Tue Jun 12 16:06:24 2001

------------------------------------------------------------------------
Mesh Info:
---------
Rectangular cells :    148
Triangular cells  :      2
Via cells         :     16
Unknown currents  :    252

Resources:
---------
Process size :     1.41 MB
User Time    :    0h00m00s
Elapsed Time :    0h00m02s


========================================================================
                 ---   STATUS SERVER MESSAGES   ---
========================================================================
 Simulation / Synthesis Messages
Mesh will be generated @ 2.000000 GHz
Status / Summary
----------- Momentum Simulation --------------
Version: Momentum 4.0 (*)  150.999 Dec 13 2000
Simulation mode: microwave
Design: cir
start compute mesh on Tue Jun 12 16:06:16 2001
mesh computation successful on Tue Jun 12 16:06:24 2001
start simulate circuit on Tue Jun 12 16:06:24 2001
start calculate low frequency behavior.
simulation frequency = 0.000000 GHz
simulation frequency = 0.109000 GHz
simulation frequency = 0.218000 GHz
start adaptive frequency sweep.
simulation frequency = 2.00000 GHz
adaptive frequency sweep:   0.0 % of frequency range covered.
simulation frequency = 1.33333 GHz
simulation frequency = 0.666667 GHz
adaptive frequency sweep: 100.0 % of frequency range covered.
circuit simulation successful on Tue Jun 12 16:06:59 2001
Writing dataset files...
```

The next example shows a summary from another project using the Momentum RF mode. Notice that it contains information about the matrix size after mesh reduction and the statistics for the quasi-static calculations.

```
=======================================================================
                    ------ MOMENTUM SUMMARY ------
                      Momentum RF 4.0 (*) 150.400
                      Tue Sep 12 13:56:05 2000
-----------------------------------------------------------------------
MESH
-----------------------------------------------------------------------
STATISTICS
----------
 Rectangular cells :    295
 Triangular cells  :   2451
 Via cells         :    168
 Edge currents     :   3369

RESOURCES
---------
 Process size :    6.20 MB
 User Time    :   0h 5m49s
 Elapsed Time :   0h 5m52s
-----------------------------------------------------------------------
SOLVER               USER TIME          ELAPSED TIME      MEMORY
PROCESSES            (h:m:s)            (h:m:s)           (MB)
-----------------------------------------------------------------------
SIMULATION SET-UP
-----------------
 Initialization     |    0: 0: 0    |      0: 0: 2    |   2.93
 Ports analysis     |    0: 0: 0    |      0: 0: 0    |   0.53
 Mesh reduction     |    0: 0: 1    |      0: 0: 1    |   5.82
 Basis creation     |    0: 0: 0    |      0: 0: 1    |   6.64
 Basis unknowns : 783

QUASI-STATIC CALCULATIONS
-------------------------
 Green functions    |    0: 0: 0    |      0: 0: 0    |   0.20
 Matrix load        |    0:15:56    |      0:15:58    |   9.82

1. SIMULATION FREQUENCY = 0 Hz
-------------------------
 Matrix load        |    0: 0: 0    |      0: 0: 0    |   6.19
 Matrix solve       |    0: 0:13    |      0: 0: 0    |   0.03
 S parameters       |    0: 0: 8    |      0: 0: 8    |   7.35
```

For examples of substrate and mesh statistics, refer to "Viewing Substrate Computation Status" on page 3-20, and "Viewing the Mesh Summary" on page 6-12.

# Performing Remote Simulations

Momentum can require significant computer resources in order to perform a simulation. Depending on your needs, it might be advantageous to install a relatively powerful computer with a large amount of RAM, and dedicate it for running simulations only, while using less powerful workstations for other tasks. With such a setup, you can perform design and setup procedures from a local, less powerful workstation, while the simulations, which require the bulk of the computation-intensive operations, are sent off to the (remote) computer that you have dedicated for running simulations.

**Note**   Refer to the installation guides for instructions on performing remote simulations with other simulators.

Momentum is designed to support remote operation. You can perform remote simulations using these hardware configurations:

- UNIX® to UNIX
- PC to PC

The steps to perform remote simulations with these configurations are below.

# UNIX to UNIX

This section describes how to perform remote simulations from the Advanced Design System environment using UNIX workstations.

## Setting up the Remote Computer

To prepare your remote computer, perform the following steps:

1. Log in to the remote computer.

2. To launch the remote EMX daemon, set the HPEESOF_DIR variable, if this has not been done yet.

   Then, add $HPEESOF_DIR/bin to the PATH variable. Make sure that it appears at the very beginning of PATH.

   Finally, start the daemon using the command:

   ```
   hpremote
   ```

   or, to keep diagnostic messages in a file for verification, use the commands:

   ```
   bootscript.sh (sets up platform-specific environment variables)
   hpeesofemx -d <file_name> & (starts the EMX daemon)
   ```

---

**Note** If another user has already launched the eedaemon, then it must not be launched a second time. Subsequent remote users can connect to this daemon as well.

---

## Setting up the Local Computer

A configuration file must to be created or modified in order to specify that Momentum is to be run remotely, and to communicate the name of the remote server to the Advanced Design System environment.

To create or modify the file, perform the following steps:

1. Create or locate the configuration file named:

   ```
   hpeesof.cfg
   ```

2. Add the following line to the file:

   ```
   MOMENTUM_SIM_PATH=<remote_computer_name>
   ```

3. Place the file in one of four directories, which are searched in the following order:

- The current working directory, where Advanced Design System has been launched;

- $HOME/hpeesof/config (most used by individual users);

- $HPEESOF_DIR/custom/config (sets Momentum in remote mode for the whole site);

- $HPEESOF_DIR/config (the preinstalled configuration, valid for the whole site);

4. Start Advanced Design System by typing:

```
hpads
```

## Running the Simulation

Starting a remote Momentum simulation is no different than starting a local one. The same menu selections apply:

- Momentum > Substrate > Precompute

- Momentum > Mesh > Precompute

- Momentum > Simulation > S-parameters

- Momentum > Post-Processing > Radiation Pattern

There are some differences between local and remote simulations in the use of files, though. A local Momentum simulation is usually carried out in a temporary directory; remote Momentum simulations are calculated in another directory. The temporary directory is guaranteed to have a unique name by a call to a utility *tmpnam*, but is always placed under /usr/tmp or /var/tmp. Both of these directories can only be accessed by a remote server in very tightly coupled clusters. This is an unnecessary severe restriction. Therefore, the remote computations write their results under the local $HOME/mom_tmp/*<temporary_name>*.

## Ending Remote Operation

To end a remote simulation setup:

1. On the local machine, exit Advanced Design System.

2. Undo the modification made to the hpeesof.cfg file.

3. Kill the hpeesofemx daemon that is running on the remote server.

4. Log out from the remote server.

The next time Advanced Design System is launched, it will operate Momentum locally again.

## Restrictions

Be aware that the following restrictions apply to remote simulation:

The $HOME/mom_tmp/<*temporary_name*> directory has wide access permissions (0777).

The $HOME directory, which resides on the local file system, must be visible under the same path from the remote server (through NFS).

If a substrate computation is required, the user must set access permissions on $HOME/substrates to 0777, and the permissions of the files under it to 0666. However, if the user forgets to do so, the UI will test for it, and it will warn the user that this must be done.

Under ADS 1.3 and later, remote Momentum simulation behaves exactly the same way as under Series IV 6.1 release 305.

# PC to PC

This section describes how to perform remote simulations from the Advanced Design System environment using PCs with Windows NT® or Windows® 95/98/2000.

---

**Note**   If you are using Windows 2000 with Windows NT, refer to the additional instructions for proper set-up.

---

## Setting up the Remote Computer

To prepare your remote computer, you must start the remote EMX daemon by performing the following steps:

1. Set the HPEESOF_DIR variable, if this has not been done yet;

2. Add %HPEESOF_DIR%/bin to the PATH variable. Make sure that it appears at the very beginning of PATH.

3. Start the daemon with the command (either from Windows or from an MS-DOS® shell) by typing

   ```
   hpeesofemx
   ```

or, if diagnostic messages are to be kept in a file for verification:

   ```
   hpeesofemx -v -d <file_name>
   ```

---

   **Note**   Do not terminate the MS-DOS window that will pop up. Doing so will immediately kill the daemon as well.

---

## Setting up the Local Computer

A configuration file must to be created or modified in order to specify that Momentum is to be run remotely, and to communicate the name of the remote server to the Advanced Design System environment.

To create or modify the file, perform the following steps:

1. Create or locate the configuration file named:

   ```
   hpeesof.cfg
   ```

Add the following line to the file:

```
MOMENTUM_SIM_PATH=<remote_computer_name>
```

2. Place the file in one of four directories, which are searched in the following order:

- The current working directory, where Advanced Design System has been launched

- %HOME%\hpeesof\config (most used by individual users)

- %HPEESOF_DIR%\custom\config (sets Momentum in remote mode for the whole site)

- %HPEESOF_DIR%\config (the preinstalled configuration, valid for the whole site)

3. Start Advanced Design System by typing:

```
hpads
```

---

**Note** It is possible to perform a remote simulation on an NT machine from a Windows 95/98 computer and vice versa.

On NT and Windows 95/98 machines it *is* possible for two local users to link to the same remote server.

If using Windows NT machines with Windows 2000 machines, following the additional steps, "Additional Set-up for Using Windows NT with Windows 2000" on page 7-18.

---

## Additional Set-up for Using Windows NT with Windows 2000

If you are using Windows NT (local) with Windows 2000 (remote), perform these additional steps:

**Windows NT (Local)**

1. Set your HOME directory to a shared directory (e.g. x:\users\\*yourName*).

2. Make sure HOME is set in the control panel and in the registry (key HKEY_LOCAL_MACHINE\SOFTWARE\AGILENT\ADS\1.7\eeenv \HOME).

---

3. Start ADS to create $HOME/config and exit.

4. Set EEDAEMON_SOCKET=1801 in $HOME/hpeesof/config/hpeesof.cfg

5. Set MOMENTUM_SIM_PATH=*w2k_server_name* in
   $HOME/hpeesof/config/hpeesof.cfg

6. Make sure the directory **substrates** exists under $HOME

7. Restart ADS and check that EEDAEMON_SOCKET is set equal to 1801

**Windows 2000 (Remote)**

1. Login with your NT login name (same login on client and server)

2. Set HOME equal to the shared directory (e.g. x:\users\\*yourName*).

3. Make sure this directory is accessible.

4. Make sure HOME is set in the registry and in the control panel.

5. Make sure HOME and HPEESOF_DIR are defined as system variables, *not* as
   user variables.

6. Start hpeesofemx and check that EEDAEMON_SOCKET is set equal to 1801.

---

**Note** When launching Momentum, ignore the "Warning #21" messages. They
indicate that the remote process cannot chdir to the starting directory.

Your project can be located on the client side or on the shared drive (x).

---

## Running the Simulation

Launching Momentum simulations is no different than launching local ones. The
same menu commands apply:

- Momentum > Substrate > Precompute

- Momentum > Mesh > Precompute

- Momentum > Simulation > S-parameters

- Momentum > Post-Processing > Radiation Pattern

There are some differences between local and remote simulations in the use of files,
though. A local Momentum simulation is usually carried out in a temporary

directory; remote Momentum simulations are calculated in another directory. The temporary directory is guaranteed to have a unique name by a call to a utility *tmpnam*, but is always placed under \temp. As on UNIX, the remote computations write their results under the local %HOME%\mom_tmp\<*temporary_name*>.

### Ending Remote Operation

To end a remote simulation setup:

1. On the local machine, exit Advanced Design System.

2. Undo the modification made to the hpeesof.cfg file.

3. Kill the hpeesofemx daemon that is running on the remote server.

The next time Advanced Design System is launched, it will operate Momentum locally again.

### Restrictions

Be aware that the following restrictions apply to remote simulation:

- The $HOME directory, which resides on the local file system, must be visible under the same path from the remote server. The restriction here is that the whole drive must be shared among the PCs.

- If a substrate computation is required, the user must set the $HOME\substrates directory and the files under it accessible for reading and writing. However, if the user forgets to do so, the UI will test for it, and it will warn the user that this must be done.

# About Adaptive Frequency Sampling

Adaptive Frequency Sampling (AFS) is a method of comparing sampled S-parameter data points to a rational fitting model. In order to accurately represent the spectral response of the circuit, the AFS feature in Momentum takes a minimal number of frequency samples and then applies its algorithm to the sampled data. Wherever the S-parameters vary the most, more samples are taken. When the fitting model and the sampled data converge, the AFS algorithm is then complete and the S-parameter data is written into the dataset. In this way, the maximum amount of information can be obtained from the minimum amount of sampling.

By using the AFS feature, you can greatly reduce the amount of time required to simulate circuits that have resonances or other sharp responses that are difficult to detect. Data that would have remained hidden using other types of analyses can now be obtained.

Even in the case of a low-pass filter, AFS can solve for the S-parameters faster and more accurately than discrete data sampling. For example, if you were measuring a 2 GHz low-pass filter, you would enable the AFS feature and then enter a start frequency of 500 MHz and a stop frequency of 4 GHz. AFS would then sample the two end points, construct the fitting model, and then sample points in between as needed. As this occurs, the model is automatically refined and appropriate sample points are taken until the model and the sampled data converge. When convergence is reached, both sampled data and the AFS data are written into two separate datasets and can be presented as S-parameter traces on a plot. If the max sample size is reached before converging, the partial results will be saved and can be reused by simulating the circuit again.

---

**Note** AFS data must converge in order to be accurate. Be sure the data is converged by checking the message in the Momentum Status window. If data is not converged, you will see inaccurate results.

---

## AFS Convergence

Convergence is determined by how close the fitting model is to the sampled data. Convergence is achieved when the sampled points are accurate to -75 dB, and the overall accuracy is -60 dB.

When AFS is used, be sure that the sampled data has converged with the fitting model. This can be checked by the messages in the status window. If converged, the message will be that AFS is complete. If not converged, a warning will appear. Examples of such warning messages are:

```
Maximum number of adaptive frequency samples reached!

Fitting models not (completely) converged.

Consider:- increasing maximum number of sample points.
- activating the reuse option.
```

If you encounter errors, try decreasing the frequency range of the frequency plan and increasing the number of sample points. Theoretically, some circuits may require

several dozen points to converge if they are electrically long. This is because the AFS program samples the data at about each 60 degree rotation, in the Smith chart, of the S-parameter with the most phase variation.

---

**Note**   If you simulation does not converge, to help you assess the quality of the simulation, you can view convergence results in the Data Display. For more information, refer to "Viewing Convergence Data" on page 9-5.

---

## Setting Sample Points

If you are using the Adaptive sweep type in a simulation, you may want to select specific sample points to be used in the AFS process. In general, this is not necessary (and is in fact, discouraged), but there may be instances where it is beneficial. An application where this may be beneficial is simulating a structure that has a distinct variation in response at some point over a frequency range, such as a resonant structure. To set up the simulation:

1. Select the Single sweep type, enter either the value of the resonant frequency or a value near it in the Frequency field, and add this to the frequency plan list.

2. Select the Adaptive sweep type, set up the plan, then add it to the list of frequency plans.

For this situation, the sample point aids the AFS process by identifying an area where there is clearly a variation in the response of the circuit.

Applying extra sample points may be necessary for visualization or far-field calculations.

It is not beneficial to attempt to add sample points for other purposes, such as attempting to force smoothing to occur at specific points, without taking into consideration the response at these points. For example:

1. Selecting the Linear sweep type, adding sample points spread linearly across the frequency range, and adding the frequency plan to the list.

2. Adding an Adaptive sweep type, selecting the same frequency range, and adding it to the list.

For this situation, the sample points from the linear simulation will be included in the adaptive simulation. This impedes the AFS process of selecting the optimal

distribution of samples over the frequency range, and this can produce poor simulation results.

To remedy this situation, either run the Adaptive sweep plan first or make sure that the frequency ranges of the two plans do not overlap.

## Viewing AFS S-parameters

When viewing the S-parameters in an AFS dataset, if you zoom in you may see small, unexpected spurious ripples or oscillations, with an amplitude less than 0.0002. The amplitude of these oscillations is always less than the AFS accuracy level, which is approximately -60 dB. Generally, these ripples will only appear if the dynamic behavior of the S-parameters is limited, for example, an S-parameter that is nearly 1 over the frequency range simulated. This is likely due to the rational fitting model have too many degrees of freedom and being too complex for this situation.

# Momentum Components

The Momentum Components feature enables you to combine EM (Momentum) and circuit simulations from the schematic page.

From the layout page in ADS, you can create a Momentum component that can be inserted in a schematic page just like any other component. The symbol representing this component in schematic can either be a black box or a layout look-alike component.

The most relevant Momentum simulation parameters (e.g., mode, mesh density) can be set from the schematic page. When a circuit simulation is done in a schematic page that includes a Momentum component, the Momentum simulation engine is automatically called as part of the circuit simulation process.

The Momentum components have a built-in database mechanism that keeps track of previously Momentum simulation results, that is, once the Momentum simulation is done for a certain Momentum component, no new EM simulations are required, unless there is a change in the Momentum parameters for this component.

## Component Creation

Create or prepare the layout for Momentum simulation by performing the following steps:

1. Set up substrate and layer mapping.

2. Draw or import the layout.

3. Insert ports and set port types.

**Note**   Only single and internal ports are supported in the Momentum components. Other port types will be mapped to single or internal port types during component creation. More information on the port mapping is given in "Advanced Features" on page 7-32.
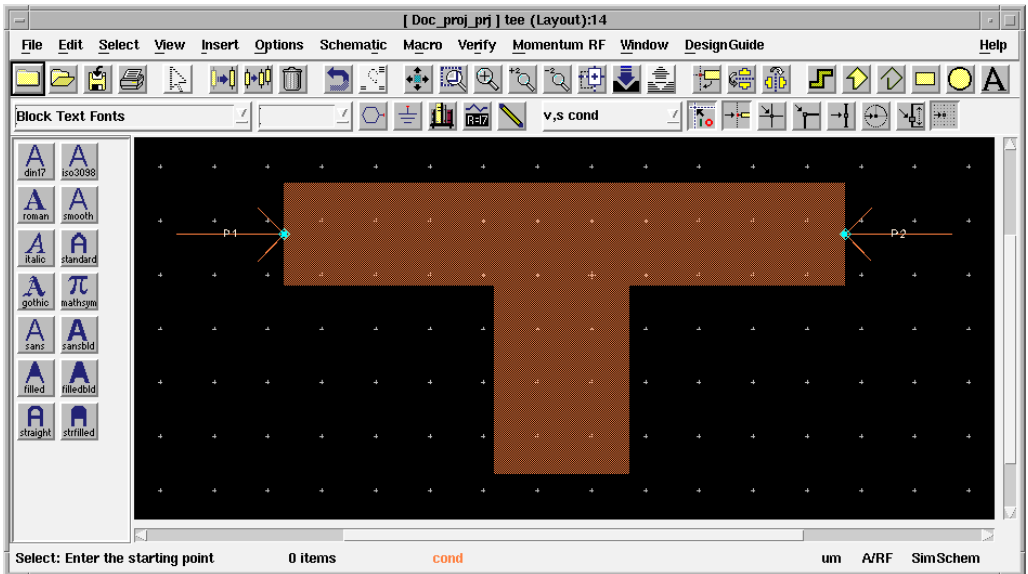


Figure 7-1. Creating a Layout

4. Open the Create Momentum Component dialog by choosing
   **Momentum** > **Component** > **Create/Update**.
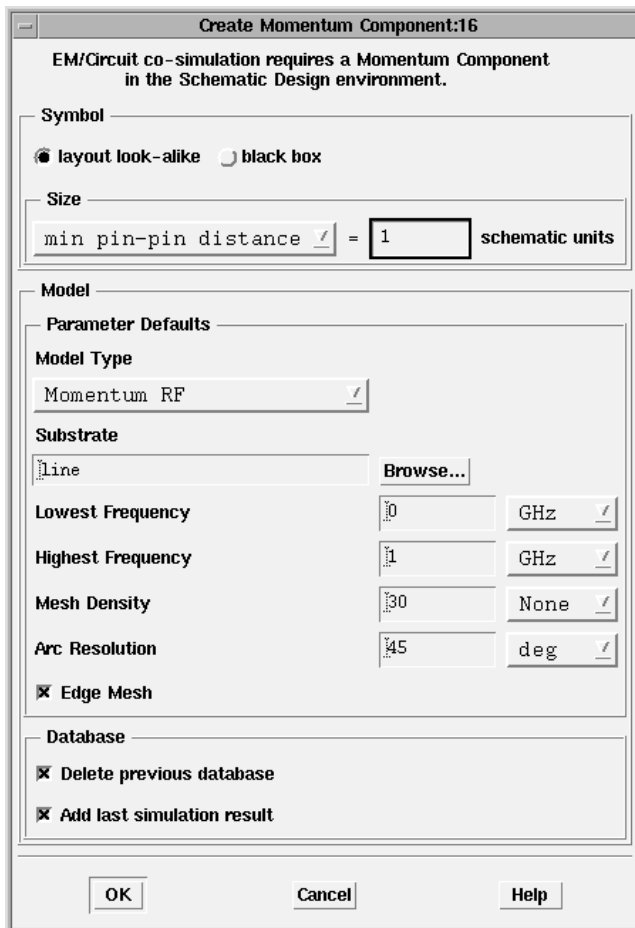


Figure 7-2. Create Momentum Component Dialog

5. Enter the following Symbol information:

   • Layout look-alike: creates a component for which the symbol shape is a
     scaled copy of the layout shape.
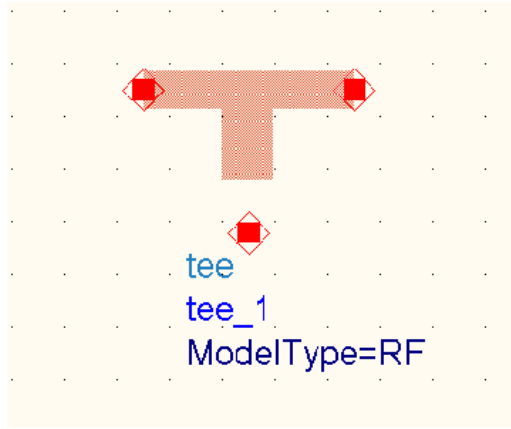


Figure 7-3. Scaled Copy of Layout Shape

   • Black box: creates a generic rectangular symbol model for the component.
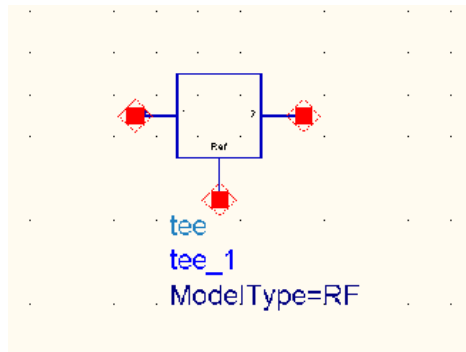


Figure 7-4. Generic Symbol Model

The scaling of the layout look-alike component can be set using different ways:

   • By setting the minimal pin-pin distance in schematic units.

   • By setting the maximum pin-pin distance in schematic units.

- By mapping the layout unit dimensions to schematic unit dimensions (recommended when combining different components in the same schematic page), the scaling is stored unless changes will be identical to the setting used in a previous component creation.

---

**Note**  All Momentum components have an explicit ground connection pin. This pin is the reference pin for all other pins.

---

6. Enter the following Parameter Defaults information:

   - Model Type: set the mode of Momentum to be used to Momentum MW or Momentum RF.
   - Substrate file (*.slm): specify the substrate description using browse button.
   - Lowest Frequency: lowest frequency to be used in the Momentum model generation for the adaptive frequency sweep algorithm.
   - Highest Frequency: highest frequency to be used in the Momentum model generation for the adaptive frequency sweep algorithm.

---

**Note**  If the circuit simulation requires frequency points beyond the specified range, this is done through extrapolation.

---

   - Mesh density: number of cells/wavelength
   - Arc resolution
   - Edge mesh on/off toggle

7. Enter the following Database information:

   - Delete previous database toggle: deletes all previous entries in the database for this component.
   - Add last simulation result: add simulation result initiated using Momentum simulation dialog in layout to the database.

8. Press **OK** to create the Momentum component, which has the same name as the layout design name.

> **Note** The parameter definitions for parameterized layouts need to be removed using the **File** > **Design Parameters** dialog before creating a Momentum component.

## Working with the Component Model Database

Open the Momentum Component Model Database dialog by choosing **Momentum** > **Component** > **Model Database**.
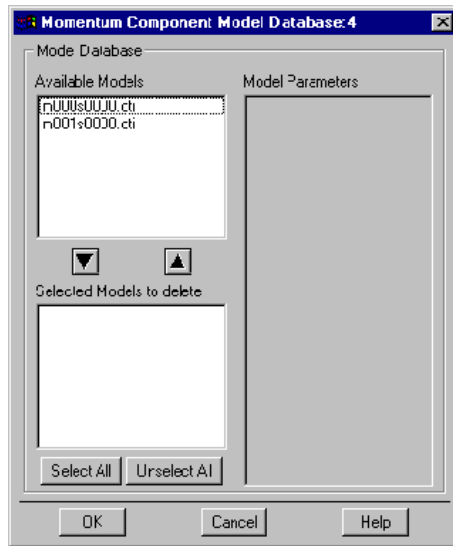


Figure 7-5. Momentum Component Model Database Dialog

Different simulation results can be associated with a Momentum component, e.g., to keep track of simulations done with different Momentum parameters, or Momentum modes (MW or RF). The models already available for a component can be viewed in the Momentum Component Model Database dialog. The parameter for every available model can be viewed. Models can be deleted. Selection and unselection of the models to be deleted can be done using the arrows or using the *Select All* or *Unselect All* buttons.

## Using Momentum Components in the Schematic Page

The Momentum components created in the layout window can be inserted in the schematic page, either by selecting the component by choosing **Insert** > **Component** > **Component Library...** dialog or by typing the name of the component in the component name selection tab. The ports added in the layout window are replaced with pins that can be connected to other components in the schematic.

### Momentum Components in the Schematic

By double clicking on a Momentum component in the schematic, the Momentum Component dialog is opened.
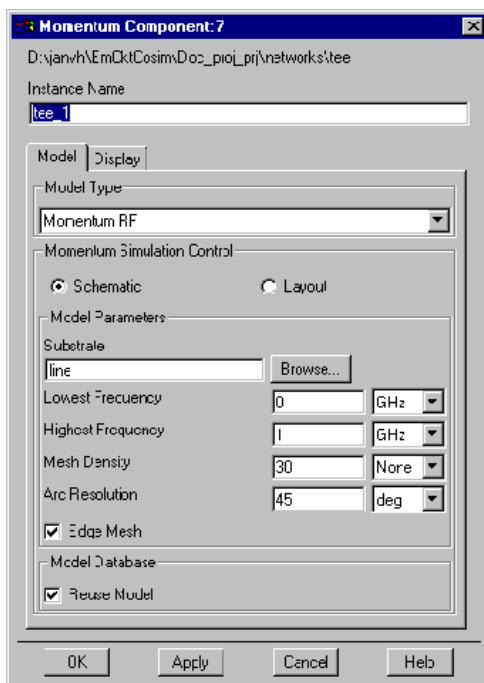


Figure 7-6. Momentum Component Dialog

The type selection box allows one of the following three choices:

• Momentum MW: the Momentum (MW) mode is used to generate the result, or when available the Momentum MW model will be retrieved from the database.

- Momentum RF: the Momentum RF mode is used to generate the result, or when available the Momentum RF model will be retrieved from the database.

- File Based

The first two options allow changing the setting of the most important Momentum parameters, like substrate name, lowest and highest frequency, mesh density, arc resolution and edge mesh. For this the schematic toggle needs to be set. By default, an adaptive frequency sweep simulation will be performed in the Momentum simulation when the schematic toggle is set.

If the Layout toggle is activated, the settings from the Momentum component parameter dialog are not taken into account and will be grayed-out. Instead, the Momentum simulation will use the parameters as they are specified in the corresponding layout design from where the component was created. This allows specifying Momentum parameter settings that cannot be specified in the Momentum Component dialog (e.g. the use of discrete frequency sweeps or more advanced mesh settings).

Previous simulation results are automatically stored in the Momentum component database. When a new simulation is requested, the database will be inspected and if the simulation result for the specified parameters is available, the corresponding simulation results will be reused (by checking the Momentum parameters specified above).

The "File Based" option allows selecting an S-parameter result available in a Dataset, CITIfile or Touchstone format as the model for the component. It also allows you to select a netlist to represent the model for the component.

**Using Momentum Components in Circuit Simulation**

Momentum components can be used just like any other components in schematic. When the circuit simulation is started from a schematic page with one or more Momentum components, the EM (Momentum) models for the components will be created automatically as part of the circuit simulation. Previously calculated Momentum results will be reused when possible (but can be deactivated by the user).

---

**Note** The use of Momentum components in a circuit simulation requires a Momentum license.

---

# Advanced Features

## Port Type Mapping (Layout and Schematic)

Only single and internal ports are supported in the Momentum components. When starting from a layout where ports of other port types are used (differential, common mode, ground reference or coplanar port types), these ports will be mapped automatically to either a single or internal port. The table below shows the mapping that is used.

Table 7-1.

|  | Layout Port Type | Schematic Port Type |
|---|---|---|
| Strip | Single | Single |
|  | Internal | Internal |
|  | Differential | Single |
|  | Common Mode | Single |
|  | Ground Reference | Internal |
| Slot | Single | Single |
|  | Coplanar | Single |

The following examples illustrate how simulations with differential, ground reference, and common mode port can be setup in schematic in a way equivalent to the simulations in layout.

### Differential Ports Example



Figure 7-7. Differential Ports Simulation Layout

Ports 1,3 and 2,4 are differential port combinations. A Momentum simulation in layout with this port configuration gives a two-by-two S-parameter result with differential excitation. In schematic, the differential ports are mapped to single ports,

meaning that Momentum will (internally) calculate a four-by-four S parameter result. This can be externally converted to a differential excitation using the setup illustrated in the schematic view (here illustrated for S-parameter ports).
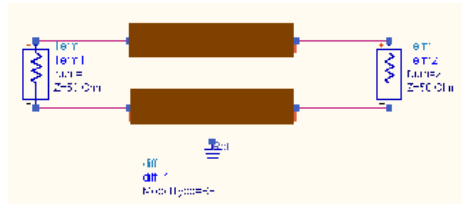


Figure 7-8. Differential Ports Simulation Schematic

**Ground Reference Ports Example**
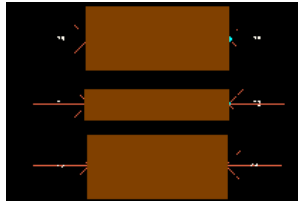


Figure 7-9. Ground Reference Ports Layout

Ports 3,5 are ground reference ports for port 1 and ports 2,4 are ground reference ports for port 2. In schematic, the ground reference ports are mapped to internal ports. To obtain the same results as with the ground reference ports, the setup of the schematic view can be used.
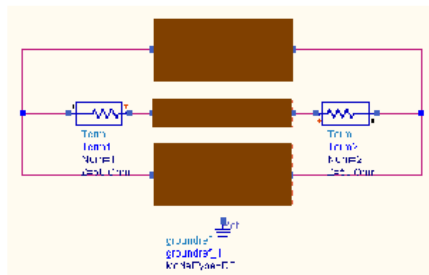


Figure 7-10. Ground Reference Ports Schematic

**Common Mode Ports Example**



Figure 7-11. Common Mode Ports Layout

Ports 1,3 and 2,4 are common port combinations. In the Momentum component, the common mode ports are mapped to single ports. To obtain the same results as with the ground reference ports, the setup of the schematic view can be used.
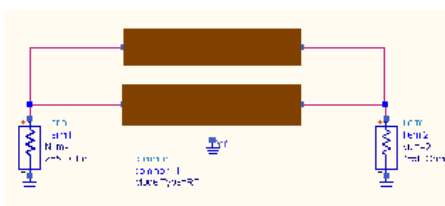


Figure 7-12. Common Ports Layout

## Hierarchical Momentum Components

The creation and usage illustrated using the following example schematic design called *bend* which consists of Microstrip or Multilayer components only (two MLIN components and a MBEND2 component in the example).
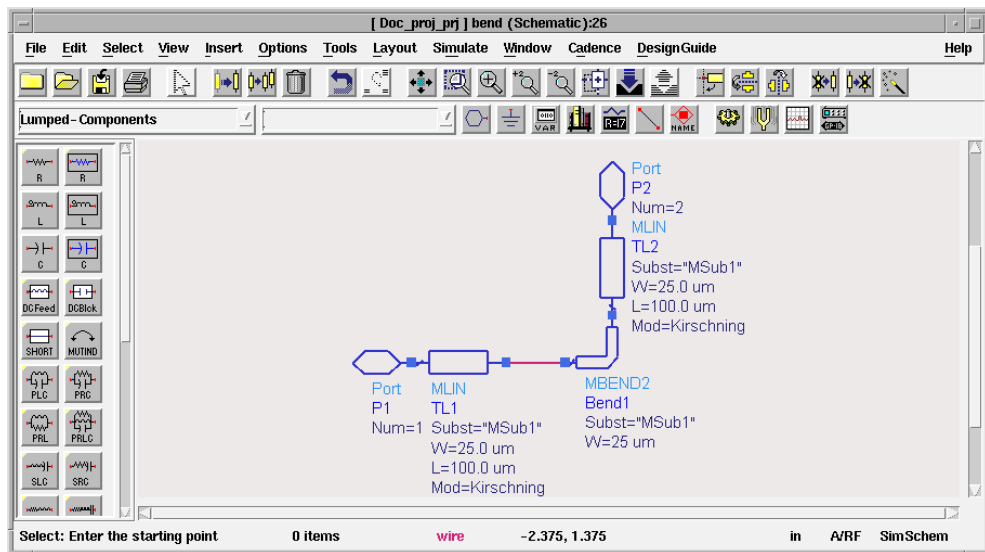


Figure 7-13. Example Microstrip Components

For this schematic, a layout can be created by choosing **Layout** > **Generate/Update Layout**, which creates a layout design called *bend*.
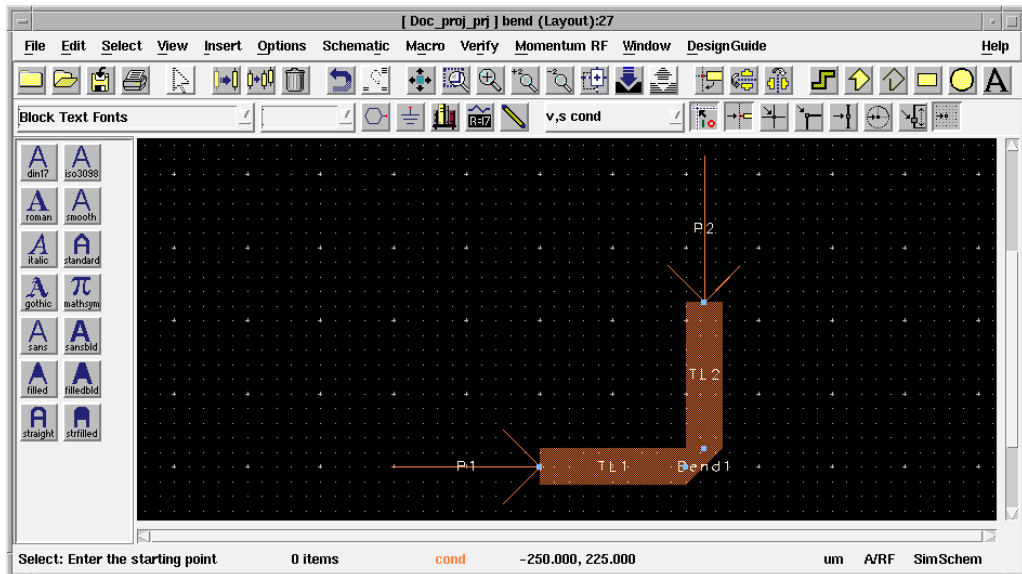


Figure 7-14. Example Layout

Choosing **Momentum** > **Component** > **Create/Update** which will make a Momentum component with the name *bend*. When using this component in a schematic design, it is possible to push into this component which will bring the user in the original *bend* schematic design. When double clicking on the *bend* component in schematic, it is possible to select *Subnetwork* as model type for this components, which means that instead of the EM or a file based model, the ADS models of the Microstrip and Multilayer components will be used.

# Chapter 8: Radiation Patterns and Antenna Characteristics

This chapter describes how to calculate the radiation fields. It also provides general information about the antenna characteristics that can be derived based on the radiation fields. In Momentum RF mode, radiation patterns and antenna characteristics are not available.

## About Radiation Patterns

Once the currents on the circuit are known, the electromagnetic fields can be computed. They can be expressed in the spherical coordinate system attached to your circuit as shown in Figure 8-1. The electric and magnetic fields contain terms that vary as $1/r$, $1/r^2$ etc. It can be shown that the terms that vary as $1/r^2$, $1/r^3$, ... are associated with the energy storage around the circuit. They are called the reactive field or near-field components. The terms having a $1/r$ dependence become dominant at large distances and represent the power radiated by the circuit. Those are called the far-field components ($\overline{E}_{ff}$, $\overline{H}_{ff}$).

$$\vec{E}(r \to \infty, \theta, \varphi) = \vec{E}_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$

$$\vec{H}(r \to \infty, \theta, \varphi) = \vec{H}_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$

**Note** In the direction parallel to the substrate (theta = 90 degrees), parallel plate modes or surface wave modes, that vary as 1/sqrt(r), may be present, too. Although they will dominate in this direction, and account for a part of the power emitted by the circuit, they are not considered to be part of the far-fields.

The radiated power is a function of the angular position and the radial distance from the circuit. The variation of power density with angular position is determined by the type and design of the circuit. It can be graphically represented as a radiation pattern.

The far-fields can only be computed at those frequencies that were calculated during a simulation. The far-fields will be computed for a specific frequency and for a specific excitation state. They will be computed in all directions (theta, phi) in the open half space above and/or below the circuit. Besides the far-fields, derived radiation pattern quantities such as gain, directivity, axial ratio, etc. are computed.

---

**Note** It is possible to calculate far-fields for some designs that have a top or bottom layer that is not defined as open space. For more information on these structures, refer to "377 Ohm Terminations and Radiation Patterns" on page 3-27.

---

# About Antenna Characteristics

Based on the radiation fields, polarization and other antenna characteristics such as gain, directivity, and radiated power can be derived.

## Polarization

The far-field can be decomposed in several ways. You can work with the basic decomposition in ($E_\theta$, $E_\phi$). However, with linear polarized antennas, it is sometimes more convenient to decompose the far-fields into ($E_{co}$, $E_{cross}$) which is a decomposition based on an antenna measurement set-up. For circular polarized antennas, a decomposition into left and right hand polarized field components ($E_{lhp}$, $E_{rhp}$) is most appropriate. Below you can find how the different components are related to each other.

$$\vec{E}_{ff}(\theta, \varphi) = E_\theta(\theta, \varphi)\vec{i}_\theta + E_\varphi(\theta, \varphi)\vec{i}_\varphi = E_{co}(\theta, \varphi)\vec{i}_{co} + E_{cross}(\theta, \varphi)\vec{i}_{cross} = E_{lhp}(\theta, \varphi)\vec{i}_{lhp} + E_{rhp}(\theta, \varphi)\vec{i}_{rhp}$$

$$H_\varphi = \frac{E_\theta}{Z_w}$$

$$H_\theta = -\frac{E_\varphi}{Z_w}$$

$$Z_\omega = \sqrt{\frac{\mu}{\varepsilon}}$$

$Z_\omega$ is the characteristic impedance of the open half sphere under consideration.

---

The fields can be normalized with respect to

$$max(\sqrt{|E_\theta|^2 + |E_\phi|^2})$$

## Circular Polarization

Below is shown how the left hand and right hand circular polarized field components are derived. From those, the circular polarization axial ratio ($AR_{cp}$) can be calculated. The axial ratio describes how well the antenna is circular polarized. If its amplitude equals one, the fields are perfectly circularly polarized. It becomes infinite when the fields are linearly polarized.

$$E_{lhp} = \frac{1}{\sqrt{2}}(E_\theta - jE_\phi)$$

$$E_{rhp} = \frac{1}{\sqrt{2}}(E_\theta + jE_\phi)$$

$$AR_{cp} = \frac{|E_{lhp}| + |E_{rhp}|}{|E_{lhp}| - |E_{rhp}|}$$

## Linear Polarization

Below, the equations to decompose the far-fields into a co and cross polarized field are given ($\alpha$ is the co polarization angle). From those, a "linear polarization axial ratio" ($AR_{lp}$) can be derived. This value illustrates how well the antenna is linearly polarized. It equals to one when perfect linear polarization is observed and becomes infinite for a perfect circular polarized antenna.

$$E_{co} = E_\theta \cos(\alpha - \phi) + E_\phi \sin(\alpha - \phi)$$

$$E_{cross} = (-E_\theta)\sin(\alpha - \phi) + E_\phi \cos(\alpha - \phi)$$

$$AR_{lp} = \frac{|E_{co}| + |E_{cross}|}{|E_{co}| - |E_{cross}|}$$
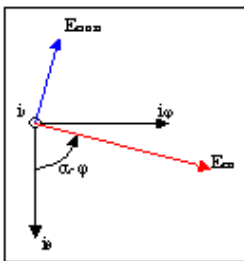
Figure 8-1. Co polarization angle

## Radiation Intensity

The radiation intensity in a certain direction, in watts per steradian, is given by:

$$U(\theta, \varphi) = \frac{1}{2}(\mathrm{E}_{ff}(\theta, \varphi) \times H_{ff}^*(\theta, \varphi)) = \frac{1}{2Z_w}(|E_\theta(\theta, \varphi)|^2 + |E_\varphi(\theta, \varphi)|^2)$$

For a certain direction, the radiation intensity will be maximal and equals:

$$U_{max} = \max_{\theta\phi}(U(\theta, \phi))$$

## Radiated Power

The total power radiated by the antenna, in Watts, is represented by:

$$P_{rad} = \int_\Omega U(\theta, \varphi) \cdot d\Omega = \frac{1}{2}\int_\Omega E_{ff} \times H_{ff}^* \cdot d\Omega$$

## Effective Angle

This parameter is the solid angle through which all power emanating from the antenna would flow if the maximum radiation intensity is constant for all angles over the beam area. It is measured in steradians and is represented by:

$$\Omega_A = \frac{P_{rad}}{U_{max}}$$

## Directivity

Directivity is dimensionless and is represented by:

$$D(\theta, \varphi) = 4\pi \frac{U(\theta, \varphi)}{P_{rad}}$$

The maximum directivity is given by:

$$D = 4\pi \frac{U_{max}}{P_{rad}} = \frac{4\pi}{\Omega_A}$$

## Gain

The gain of the antenna is represented by:

$$G(\theta, \varphi) = 4\pi \frac{U(\theta, \varphi)}{P_{inj}}$$

where $P_{inj}$ is the real power, in watts, injected into the circuit.

The maximum gain is given by:

$$G = 4\pi \frac{U_{max}}{P_{inj}}$$

## Efficiency

The efficiency is given by:

$$\eta = \frac{P_{rad}}{P_{inj}} = \frac{G}{D}$$

## Effective Area

The effective area, in square meters, of the antenna circuit is given by:

$$A_{eff}(\theta, \varphi) = \frac{\lambda^2}{4\pi} G(\theta, \varphi)$$

# Calculating Radiation Patterns

To calculate a radiation pattern:

1. Choose **Momentum > Post-Processing > Radiation Pattern**.

2. Select the **General** Tab.

3. Select a frequency from the Select Frequency listbox. The radiation pattern will be calculated at this frequency. These are frequencies that were calculated during the simulation.

4. Select a port from the Available Ports list to apply a steering circuit (based on the Thevenin equivalent) to the port and inject energy into your circuit. You can apply a steering circuit to any port.

5. In the Port Excitation fields, enter amplitude (in volts) and phase (in degrees) of the voltage of the steering circuit of that is applied to the port.

6. In the Port Termination Impedance fields, enter the real and imaginary components of the impedance of the steering circuit that is applied to the port.

---

**Note**   You may want to terminate all ports so that the ports that are not used to inject energy into the circuit do not cause reflections. For these ports, set the Port Excitation voltage to zero, and set the Port Impedance to the characteristic impedance of the port.

---

7. Select the display tool that you want to use. Enable either 2D Data Display or 3D Visualization. The 3D Momentum Visualization tool requires a separate codeword. Instructions for using the tool are in following chapters.

8. If you want the display tool to open and display results automatically, enable *Open display when computation completed*.

9. If you have selected to use the 3D tool, click **Compute**. More information on how to use the Momentum Visualization tool are in following chapters.

   If you are using the Data Display, click the **2D Settings** tab and continue with these steps.

10. Specify a dataset name where you want to store the far-field results.

11. Select the template that you want to use for formatting results, or accept the default.

12. Select either the planar or conical Cut Type Definition:

- A *planar* cut is perpendicular to the circuit layout plane.

- A *conical* cut is a cone around the z-axis.

Refer to the diagram in the Radiation Pattern Control dialog box. For more information, refer to "Planar (Vertical) Cut" on page 8-8 and "Conical Cut" on page 8-8.

13. Enter the value of the cutting angle, in degrees, in the Cutting Angle field.

- For a planar cut, this angle, phi, is relative to the x-axis. Phi will be kept constant while theta will be swept from -180 to 180 degrees.

- For a conical cut, the cutting angle, theta, is relative to the z-axis. Theta will be kept constant while phi will be swept from 0 to 360 degrees.

14. Enter the value of the co polarization angle in the Co Polarization Angle field. This angle will be used when decomposing the far-fields into a co and cross polar component (see "Linear Polarization" on page 8-3).

15. Enter the threshold for displaying data in the Lower Clipping Boundary field. Select units of dB or mag. Computed radiation pattern data that is below this value will be displayed as zero (when units are mag) or the lower clipping boundary (when units are dB).

16. Click **Compute** to compute the radiation pattern. After computation, the far-fields and derived antenna parameters will be stored in a dataset with the name you specified. An _f will be appended to the dataset name. The _f indicates that the dataset contains radiation pattern data. The dataset is stored in the /*data* directory of the current project.

After the radiation calculations are performed, data is saved depending on the display tool selected above:

- 2D Data Display was selected: all data is stored in a single dataset with the name you specified with *_f.ds appended*. The _f indicates that the dataset contains radiation pattern data. The dataset is stored in the */data* directory of the current project.

- 3D Visualization was selected: the far-fields and derived antenna parameters will be stored in respectively *proj.fff* and *proj.ant* in the *mom_dsn/*<design_name> directory of your project.

## Planar (Vertical) Cut

For the planar cut, the angle phi, which is relative to the x-axis, is kept constant. The angle theta, which is relative to the z-axis, is swept to create a planar cut. Theta is swept from 0 to 360 degrees. This produces a view that is perpendicular to the circuit layout plane. Figure 8-2 illustrates a planar cut.
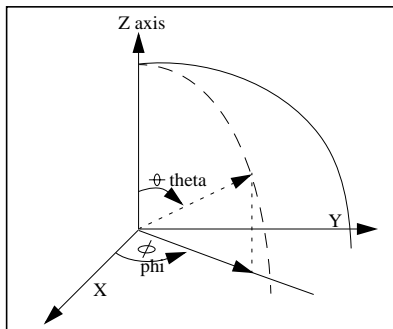


Figure 8-2. Planar (vertical) cut

## Conical Cut

For a conical cut, the angle theta, which is relative to the z-axis, is kept constant. Phi, which is relative to the x-axis, is swept to create a conical cut. Phi is swept from 0 to 360 degrees. This produces a view that is parallel to the circuit layout plane. Figure 8-3 illustrates a conical cut.
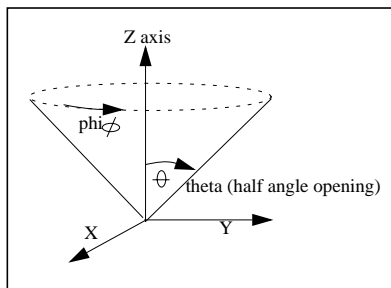


Figure 8-3. Conical cut

# Viewing Results Automatically in Data Display

If you choose to view results immediately after the far-field computation is complete, enable *Open display when computation completed*. When Data Display is used for viewing the far-field data, a data display window containing default plot types of the data display template of your choice will be automatically opened when the computation is finished. The default template, called *FarFields*, bundles four groups of plots:

- *Linear Polarization* with $E_{co}$, $E_{cross}$, $AR_{lp}$.

- *Circular Polarization* with $E_{lhp}$, $E_{rhp}$, $AR_{cp}$.

- *Absolute Fields* with $E_\theta, E_\varphi, H_\theta, H_\varphi$.

- *Power* with Gain, Directivity, Radiation Intensity, Efficiency.

For more information, please refer to "About Antenna Characteristics" on page 8-2.

# Exporting Far-Field Data

If 3D Visualization is selected in the *Radiation Pattern* dialog, the normalized electric far-field components for the complete hemisphere are saved in ASCII format in the file <project_dir>/*mom_dsn*/<design_name>/*proj.fff*. The data is saved in the following format:

```
#Frequency <f> GHz       /*  loop over  <f> */
  #Excitation #<i>        /*  loop over  <i> */
  #Begin cut              /*  loop over phi  */
<theta> <phi_0>  <real(E_theta)>  <imag(E_theta)> <real(E_phi)>
<imag(E_phi)>
                /*   loop over <theta>    */
  #End cut
  #Begin cut
  <theta> <phi_1>  <real(E_theta)>  <imag(E_theta)> <real(E_phi)>
<imag(E_phi)>
                /*   loop over <theta>    */
  #End cut
      :
      :
#Begin cut
  <theta> <phi_n>  <real(E_theta)>  <imag(E_theta)> <real(E_phi)>
<imag(E_phi)>
                /*   loop over <theta>    */
  #End cut
```

In the *proj.fff* file, E_theta and E_phi represent the theta and phi components, respectively, of the far-field values of the electric field. Note that the fields are described in the spherical co-ordinate system (r, theta, phi) and are normalized. The normalization constant for the fields can be derived from the values found in the proj.ant file and equals

$$\sqrt{\langle\text{E\_theta\_max}\rangle^2 + \langle\text{E\_phi\_max}\rangle^2}$$

The *proj.ant* file, stored in the same directory, contains the antenna characteristics. The data is saved in the following format:

```
Excitation <i> /* loop over  <i> */
Frequency <f> GHz /* loop over  <f> */
Maximum radiation intensity  <U> /* in Watts/steradian */
Angle of U_max   <theta> <phi> /* both in deg */
E_theta_max   <mag(E_theta_max)> ; E_phi_max   <mag(E_phi_max)>
E_theta_max   <real(E_theta_max)>   <imag(E_theta_max)>
E_phi_max   <real(E_phi_max)>   <imag(E_phi_max)>
Ex_max   <real(Ex_max)>   <imag(Ex_max)>
Ey_max   <real(Ey_max)>   <imag(Ey_max)>
Ez_max   <real(Ez_max)>   <imag(Ez_max)>
Power radiated   <excitation #i> <prad> /* in Watts */
Effective angle   <eff_angle_st> steradians <eff_angle_deg> degrees
Directivity   <dir> dB /* in dB */
Gain   <gain> dB /* in dB */
```

The maximum electric field components (E_theta_max, E_phi_max, etc.) are those found at the angular position where the radiation intensity is maximal. They are all in volts.

# Chapter 9: Viewing Results Using the Data Display

You have two tools available for displaying the results of an Momentum simulation: the Data Display, and Momentum Visualization. The visualization tool requires a separate codeword and is described in following chapters. This chapter describes how to use the Data Display.

You can display the following types of results from an Momentum simulation:

- S-parameters
- Radiation patterns (far-field plots) and derived antenna parameters

Additional data is calculated during a simulation and saved in the dataset, and can also be viewed or used in calculations:

- Propagation constant (gamma) for each port in the circuit
- Characteristic impedance of each port
- AFS convergence data

This chapter shows how to display Momentum results. For detailed information on how to use the Data Display, refer the *Data Display* manual.

## Opening a Data Display Window

You view simulation results using the Data Display. You can open a Data Display window using different methods:

- Open a Data Display window automatically when a simulation is complete. For steps on how to do this, refer to "Viewing Results Automatically" on page 7-6.
- Open a new Data Display by choosing *Window > New Data Display* from the Layout Window menu bar.
- Open a saved Data Display by choosing *Window > Open Data Display* and selecting a Data Display in the current project. Data Display files end in the extension *.dds*.

# Viewing Momentum Data

Simulated Momentum data is saved in one or more datasets:

- Data collected at the frequency points computed by the simulator are stored in <design_name>*.ds*. It is the standard dataset created from any simulation.

- If the *Adaptive* sweep type is used for simulation, densely resampled data, computed by AFS at each sampled frequency point, are stored in the dataset <design_name>*_a.ds*.

- If the radiation patterns are calculated and the visualization type selected is "2D Data Display", the results are stored in the dataset <design_name>*_f.ds*.

The datasets for the current project are displayed in the dataset listbox.



Datasets in the current project. The one in the menu bar is the default.
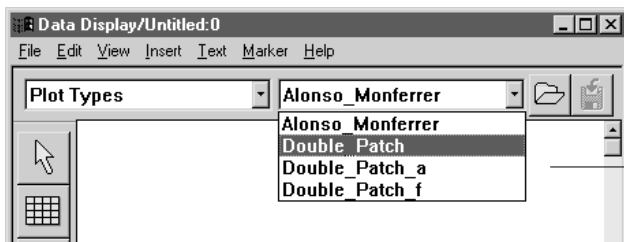
You can view Momentum data using any plot type:



Rectangular

Polar

Smith chart

Stacked

List
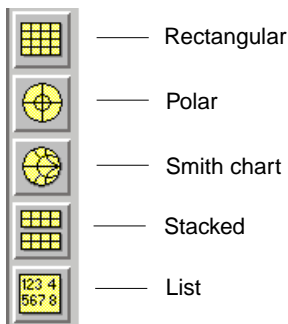
To view Momentum data:

1. Select the plot type by clicking the appropriate plot icon, dragging the mouse into the window, then clicking to place the plot.

2. The variables in the default dataset are displayed. Select a different dataset from the list, if desired.

3. Select the variables containing the data that you want to add to the plot, then click **Add**.

---

**Note** Most of the simulation data is in complex format. If you are adding data to a rectangular plot, you must choose to display the data in one of several scalar formats.

---

4. You can add data from different datasets to the same plot. Repeat steps two and three to add more data to the plot.

5. Click **OK** to view the data.
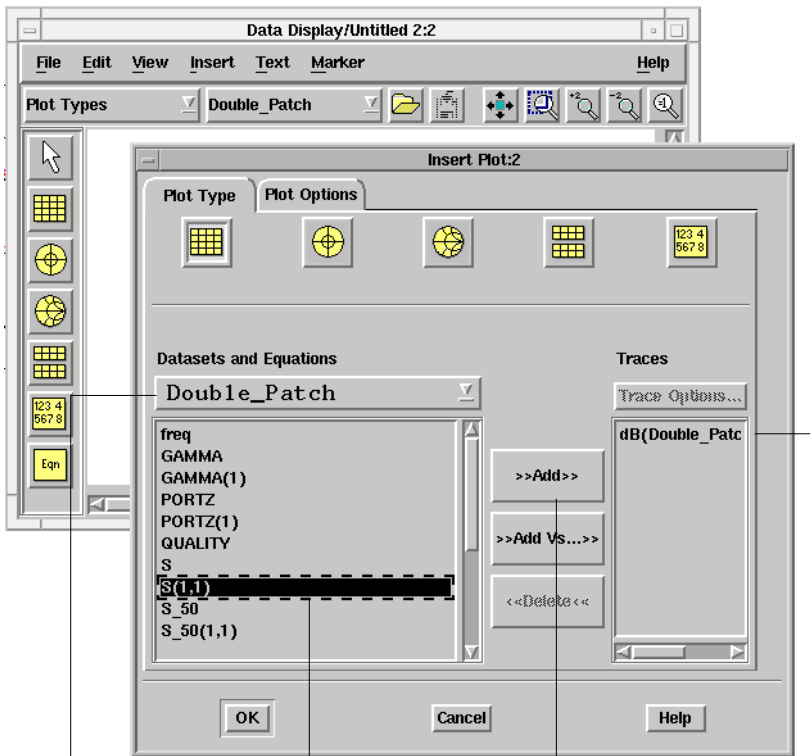


1. Select a dataset.  2. Select a variable.  3. Click Add.  4. Selected variables appear here.

---

**Note** Multiple graphs within the same view will overlay one another. To view multiple graphs simultaneously, assign each graph to a separate view.

---

# Viewing S-parameters

Specific information about the data available in the S-parameter datasets is given in the following sections.

## Variables in the Standard and AFS Dataset

A dataset generated from an Momentum simulation contains many variables. Below are the names of common variables that appear in a Momentum dataset and a description of the variable.

| Variable | Description |
|---|---|
| freq | The independent variable frequency, which was specified during the simulation setup. The number of data points is based on the sweep type and frequency plans specified during simulation. |
| GAMMA$n$ | The (modal) propagation constant of Port $n$. It is used in de-embedding, and is calculated for single, differential, and coplanar ports only. |
| PORTZ$n$ | The impedance of Port n as defined in the Impedance fields of the Port Editor dialog box. If an impedance is not specified, 50 ohms is assumed. |
| S | S matrix, normalized to PORTZ$n$. |
| S($i,j$) | S-parameters for each pairing of ports in the circuit, normalized to PORTZ$n$. |
| S_50 | S-matrix, normalized to 50 ohms. |
| S_50($i,j$) | S-parameters for each pairing of ports in the circuit, normalized to 50 ohms. |
| S_CONV and S_CONV($i,j$) | Boolean results indicating AFS convergence occurred (1) or failed (0) at a given frequency. Available in datasets from simulations using Adaptive sweep type. |

| S_ERROR and S_ERROR($i,j$) | Estimated error for a given frequency. For frequency points that converged, the value is below -60 dB. Points that fail to converge will be greater than -60 dB. Available in datasets from simulations using Adaptive sweep type. |
| S_Z0 | S matrix, normalized to Z0. |
| S_Z0($i,j$) | S-parameters for each pairing of ports in the circuit, normalized to Z0 of each port. |
| Z0$n$ | Characteristic impedance of Port $n$. It is not calculated for all port types, in which case 50 ohms is substituted. |

**Note**   Refer to Chapter 4, Ports, for an explanation of the S-parameters for layouts that use differential, coplanar, and common mode ports.

## Standard and AFS Datasets

If the *Adaptive* sweep type is used for a simulation, two datasets are created for storing data. The first one contains data computed by the simulator at the frequency points determined during the simulation. (This dataset is the standard one produced by any simulation.) The second contains data computed by AFS, resampled with a very dense frequency distribution. This "adaptive" dataset is denoted by _a appended to the dataset name. The adaptive dataset contains the same variables as the standard dataset, but data is calculated for a significantly greater number of frequency points.

Typically, when viewing data in the _a dataset, you should see very smooth results, which reflect the behavior of the circuit. Viewing the standard dataset on the same plot will show the frequency points for which the simulator was invoked, superimposed on the smooth curve. Those frequency points will be scattered over the whole range, with relatively more points grouped in areas where the S-parameters show more variation. Viewing both datasets can help you determine the quality of the AFS process on your simulation. For more information on AFS, refer to "About Adaptive Frequency Sampling" on page 7-20.

## Viewing Convergence Data

Viewing convergence data can be helpful if a simulation using the Adaptive sweep type has failed. You can view where convergence has been achieved, and evaluate the quality of unconverged fitting models.

For information on the AFS process, refer to "About Adaptive Frequency Sampling" on page 7-20.

The sections that follow describe how to view convergence data. This requires being familiar with using the Data Display and Equations. For more information on these topics, refer to the *Data Display* manual.

## Data Overview

There are two variables in the adaptive dataset (named *<project_name>_a*) that contain convergence data: S_CONV and S_ERROR.

- S_CONV contains boolean values indicating where convergence has been achieved (1) or has failed (0) for the entire S matrix. S_CONV($i,j$) contains boolean values indicating the convergence for S($i,j$).

- S_ERROR contains the estimated magnitude and phase fitting error for the entire S matrix. S_ERROR($i,j$) contains the fitting error for S($i,j$). The error will be less than -60 dB if convergence is achieved. You can view S_ERROR data in dB by adding the data to a rectangular plot.

## Viewing the Data

There are two suggested methods for viewing convergence. One is to simply plot S_ERROR(i,j) in dB on a rectangular plot and evaluate the quality of the AFS fit. In this example, the Low_pass_filter example is simulated with a Sample Points Limit of 19. The error for S2,1 is plotted. Convergence has been achieved over much of the frequency span, but not all of it, so the simulation has failed. This indicates the quality of the simulation over the entire frequency span that was simulated.
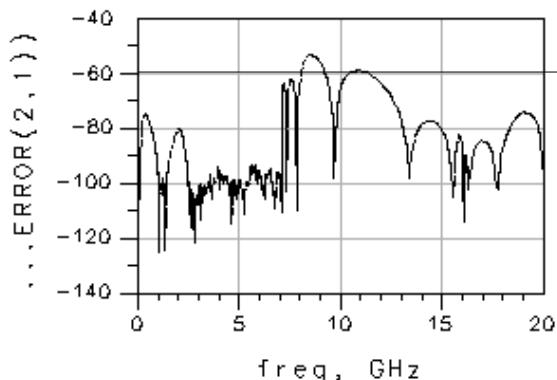


Several frequencies between 8 and 11 GHz fail to achieve convergence.

A second way to assess the simulation is to use S_CONV to separate converged and unconverged data for a set of S-parameters and view it on a plot.

To view convergence data:

1. Open a Data Display window and select the adaptive dataset <filename>_*a* containing the convergence data as the default dataset.

2. Choose **Insert > Equation**. Position the equation line in the display area and Click. Enter the following equation:

   `Converged_S`$ij$`=S(`$i,j$`)*S_CONV(`$i,j$`)`

   $i,j$ must refer to the same sets of ports throughout the equation. This equation will return the converged data, and zero otherwise.

3. Click **OK** to add the equation to the display area.

4. Choose **Insert > Equation** and add the following equation:

   `Not_Converged_S`$ij$`=S(`$i,j$`)*(1-S_CONV(`$i,j$`))`

   Use the same $i,j$ as in the previous equation. This equation returns the results where convergence was *not* achieved.

5. Click **OK** to add the equation to the display area.

6. Insert a rectangular plot in the display area.

7. The Insert Plot dialog box opens. Under the Datasets and Equations listbox, scroll to the bottom of the list and select Equation.

8. The variables *Converged_S(i,j)* and *Not_Converged_S(i,j)* appear in the variables list. Select one variable and add it to the plot, in dB.

9. Click **OK** to view the results on the plot.

10. Insert a second rectangular plot and add the second variable to it.

In the illustration below, the S2,1 convergence results for the low-pass filter, simulated with the samples limit set to 19, is displayed.

Note that because the AFS process breaks up the entire frequency span into smaller subsections, the variable S_CONV will be 1 for the points that achieve convergence over an entire subsection. So, even though the data in S_ERROR indicates almost the

entire simulation converged, the plots below indicate the frequency ranges that converged or failed.

Eqn Converged_S21=S(2,1)*S_CONV(2,1)
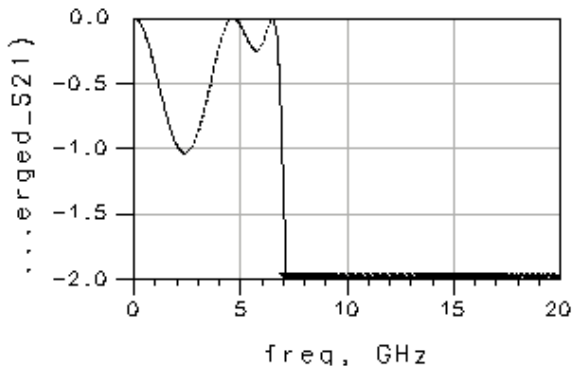


Eqn Not_Converged_S21=S(2,1)*(1−S_CONV(2,1))
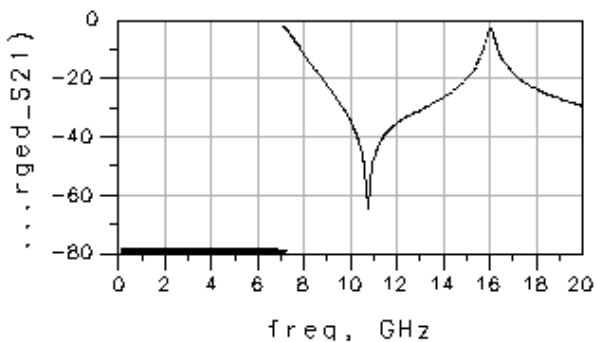


## Considerations

For efficiency purposes, the Adaptive Frequency Sampling process splits a simulation frequency range into smaller subranges that contain a maximum of 16 samples, and performs calculations on these smaller frequency ranges. The convergence factor (1 or 0) will be the same for all frequencies in a subrange. If convergence is achieved for

some subranges and not others, this will be apparent. Note that for simulations that have less than 16 samples, the entire frequency range is treated as one internal subrange, so the convergence data will be all zeros if convergence was not achieved.

# Viewing Radiation Patterns

Specific information about the data available in the far-field dataset is given in this section.

## Variables in the Far-field Dataset

When a radiation pattern is calculated, the following variables are generated for storing data:

| Variable | Description |
|---|---|
| THETA | The swept parameter of a planar cut. When THETA is swept, PHI is at a fixed angle specified in the Cut Angle field and is not returned to the dataset. |
| PHI | The swept parameter of a conical cut. When PHI is swept, THETA is at a fixed angle specified in the Cut Angle field and is not returned to the dataset. |
| Etheta and Ephi | Absolute field strength (in volts) of the theta and phi electric far-field components. |
| Htheta and Hphi | Absolute field strength (in amperes) of the theta and phi magnetic far-field components. |
| Elhp and Erhp | Normalized field strength of respectively left-hand and right-hand circular polarized electric far-field component. |
| ARcp | Axial ratio, derived from left-hand and right-hand circular polarized far-field components. |
| Eco and Ecross | Normalized field strength of co and cross polarized electric far-field components. |
| ARlp | Linear polarization axial ratio, derived from co and cross polarized far-field components. |
| Gain, Directivity, Efficiency, EffectiveArea | Gain, Directivity, Efficiency (in percent), Effective area (in $m^2$). |
| Power | Radiation intensity (in watts/steradian). |

Details about each of these parameters can be found in "About Antenna Characteristics" on page 8-2.

---

**Note**   The normalized far-field components (Elhp, Erhp, Eco, and Ecross) are normalized with respect to $max(\sqrt{|E_\theta(\theta, \varphi)|^2 + |E_\varphi(\theta, \varphi)|^2})$.

---

# Chapter 10: Momentum Visualization Basics

Momentum Visualization enables you to view and analyze these types of simulation data:

- S-parameters
- Currents
- far-fields
- Antenna parameters
- Transmission line data

Data can be analyzed in a variety of 2D and 3D plot formats. Some types of data are displayed in tabular form.

You can view the simulation results from any Momentum or Agilent HFSS project.

This chapter gives an overview on how to use the Momentum Visualization. Subsequent chapters describe in detail how to use Momentum Visualization for viewing specific types of data, such as S-parameters or currents.

## Starting Momentum Visualization

To start Momentum Visualization choose *Momentum > Post-Processing > Visualization*. The current project must run through a complete simulation use Visualization to view data. If simulation has been previously completed for a project, you can start Visualization directly to view the existing data.

# Working with Momentum Visualization Windows

The following figure highlights the basic elements of the Momentum Visualization window.

For an overview of Momentum, refer to "Momentum Overview" on page 1-2.
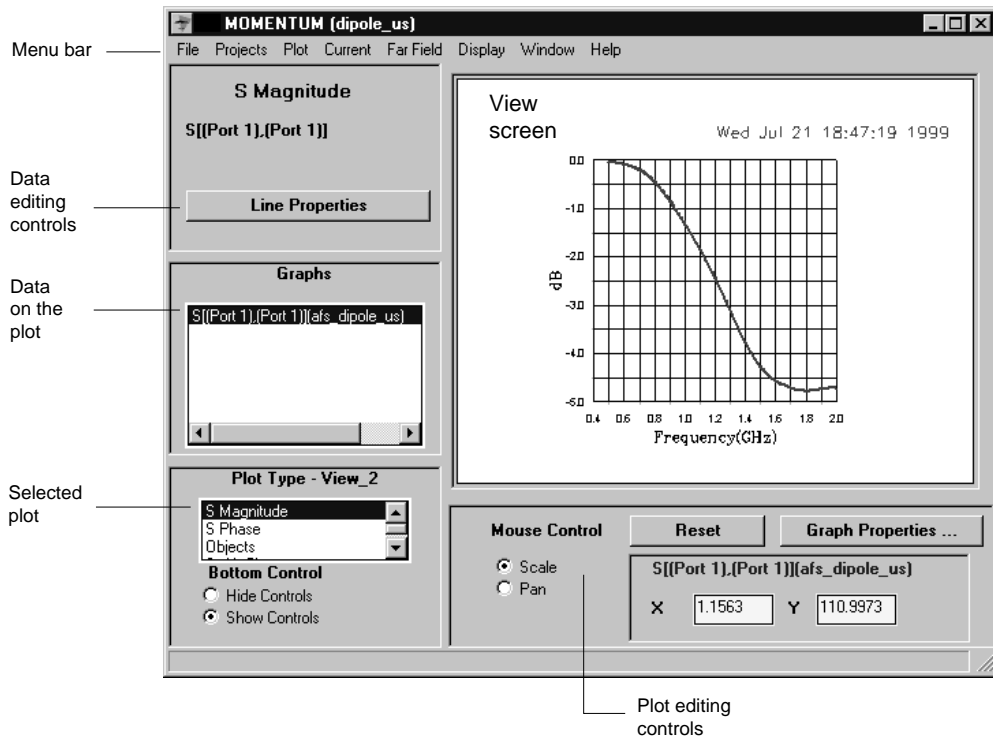
## Selecting the Number of Views

By default, the view screen displays a single view. You can optionally display four views. By using four views, you can display up to four different plots at one time.

To display four views:

- Choose **Window > Tile**.
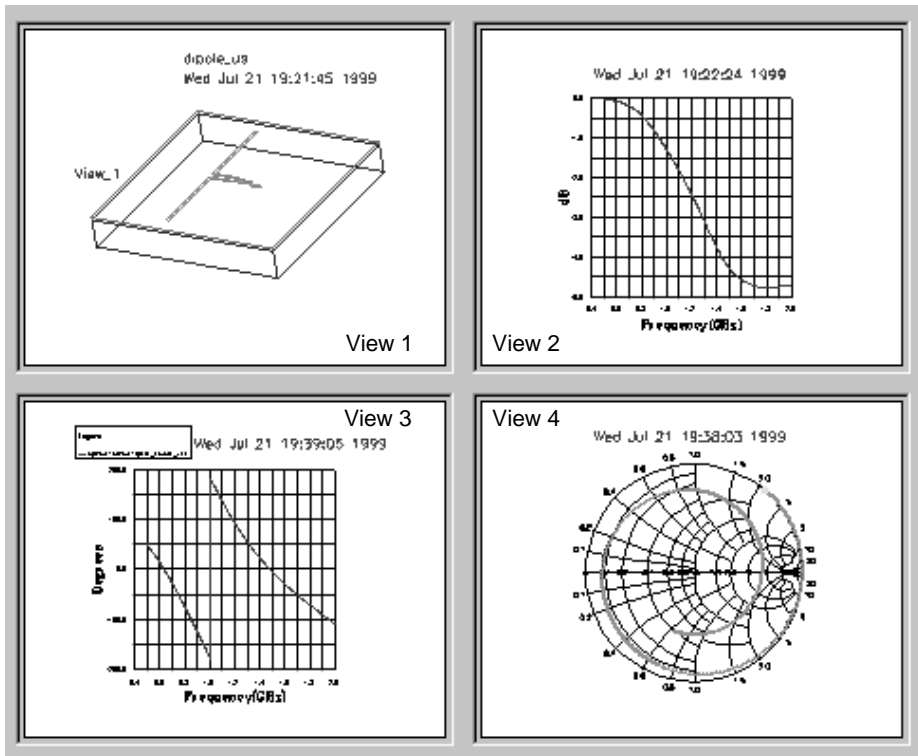


**Tip**    You can enlarge the viewing area by clicking **Hide Controls**.

To display a single view:

- Choose **Window > Full Window**. The currently-selected view fills the view screen.

## Selecting a View to Work in

You must choose from one of the four views to work in. You can change to a different view at any time.

To select a view:

  1. Position the mouse in the view and click.

To identify the selected view, note the title of the Plot Type - View list. This will change to identify the selected view. For example, if you choose View 3, the title will appear as Plot Type - View_3.

**Plot Type - View_3** ——— identifies the
selected view

| S Magnitude |
| S Phase |
| Objects |
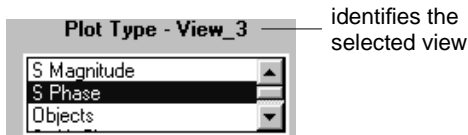
To select a view and enlarge it to a single view:

  1. Choose **Window > Select View**.

  2. Select a view from the View list.

  3. Click **OK**. The view screen displays the new view.

## Setting Preferences for a View

You can set the color preferences for a view, including the background, grid lines, and text along the plot axes.
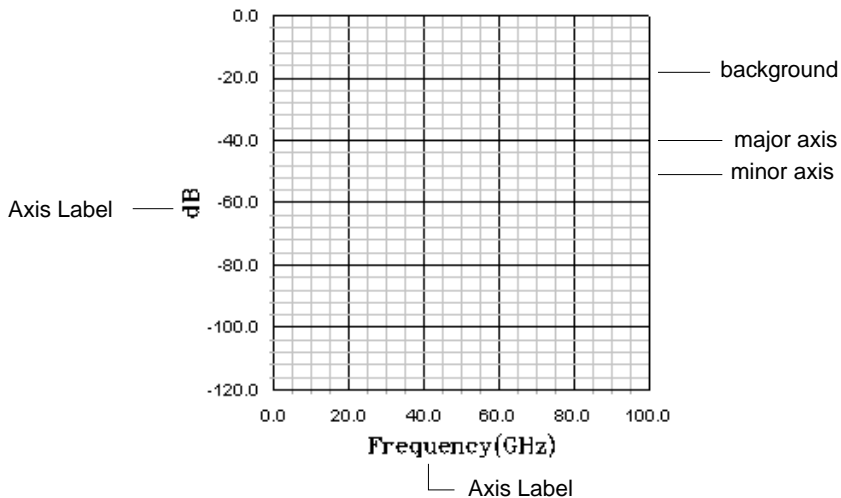


To set preferences:

1. Choose **Window > Preferences**.

2. Select a color for the background, the text that labels plot axes, and colors of the major and minor axes. The major axis includes the grid lines with numbers, the minor axis consists of grid lines that appear in between the numbered ones.
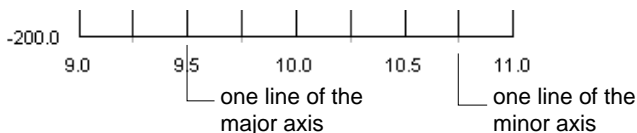


3. Click **Done** to dismiss the dialog box.

# Working with Annotation in a View

You can add text to a view, and edit the position and color of the text, by choosing *Display > Annotation*. You can save your settings so that they can be reused.
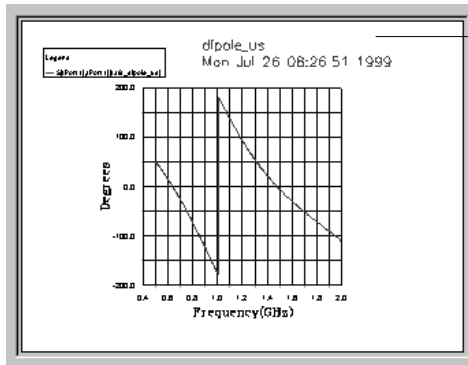


you can add text and position it anywhere in a view

## Adding Text

To add text to a view:

1. Select the view where you want to add text.

2. Choose **Display > Annotation**. Drag the Annotation dialog box so that any existing text is in view.

3. Click **New Annotation**.

4. Type your text into the Annotation Label field. If text already appears in this field, you can delete it and then type in your text. When you are finished, press the **Enter** key.

5. Use the X Location and Y Location scroll tools to position the text on the display.

6. Click **Done** to dismiss the dialog box and view the new text.

## Adding Variables

There are three variables that can be added to the annotation: %project%, %view%, and %date%. These variables can be displayed automatically whenever Momentum Visualization is in use.

## Editing Text

To edit text:

1. Select the view where you want to edit text.

2. Choose **Display > Annotation**. Drag the annotation dialog box so that any existing text is in view.

3. Select the text that you want to edit from the Annotations field.

4. To adjust the position of the text in the view, use the X Location and Y Location scroll tools.

5. To change the size of the text, use the Annotation Size scroll tool to increase or decrease the font size.

6. To change the thickness of the text characters, use the Annotation Thickness scroll tool to make fine, normal, or bold-faced characters.

7. To change the color of the text, select one of the colors listed under Annotation Color.

8. To change the orientation of the text, Horizontal or Vertical under Orientation.

9. To change the text, make your changes in the Annotation Label field.

10. To make changes to other lines of text, select the text from the Annotation field and edit as desired.

11. When you are finished, click **Done** to dismiss the dialog box.

## Deleting Text

To delete text:

1. Select the view that you want to add text to or edit text.

2. Choose **Display > Annotation**. Drag the annotation dialog box so that any existing text is in view.

3. Select the text that you want to delete from the Annotations field.

4. Click **Delete Annotation**.

5. Click **Done** to dismiss the dialog box.

## Retrieving Plots in a View

Multiple plot types can be stored under a single view. Only one plot at a time can be displayed in a view, but any other plots that you worked with in the view will be saved so that you can view them again.

To retrieve a plot from a view:

- Select a view and either:
  - Use the arrows keys on the keyboard to scroll through plots.
  - From the Plot Type - View list, select a plot type that you worked with. The plot and all graphical information that was added to the plot is displayed.
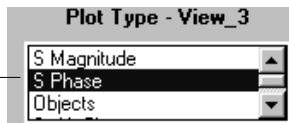


## Refreshing the Window

Choose *Window > Refresh* at any time to update the displayed views. Typically, the window refreshes automatically after a command is completed.

# Data Overview

You can display these types of data in Momentum Visualization:

- S-parameters
- Currents
- far-fields
- Antenna parameters
- Transmission line data (gamma and characteristic impedance)

The plots and tables that are available for viewing this data can be found under these menus.
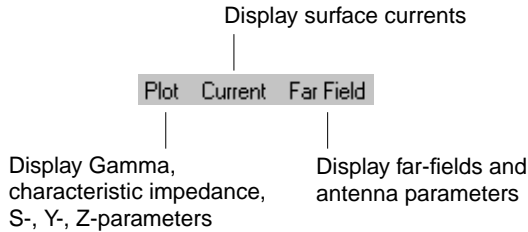


The chapters that appear later in this manual describe in detail how to work with these different types of data.

# Working with Plots and Data

There are many plot types available, plus many types of data to display. This section describes how to work with plots, such as adding a plot to a view, adding data to a plot, and editing a plot.

## Displaying a Plot

Momentum Visualization is designed so that for each basic type of data, such as S-parameters or currents, there is a set of plots available for displaying the data. The details for working with each basic data type and the available plots are in later chapters in this manual.
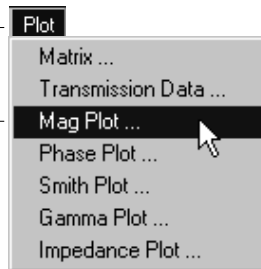
As an example, to display an S-parameter magnitude plot:

1. Choose **Plot > Mag Plot**.

2. Select a matrix and set of S-parameters from the Matrix and S-parameters list. Details about these lists appear in Chapter 10, Momentum Visualization Basics.

3. Select a view from the View list. The plot will appear in this view.

4. Click **Apply**.

## Adding Data to a Displayed Plot

Multiple sets of data can be added to a plot:

- For most rectangular plots, you simply repeat the steps that you took to add the plot to the view and choose a different set of data from the dialog box. If you want to use data from a different project, select the project from the *Projects* menu first, then repeat the steps.

- For current plots, you can add items that are listed under the **Display** menu.

The dialog box below illustrates how to add a second set of S-parameters to an S-parameter magnitude plot.
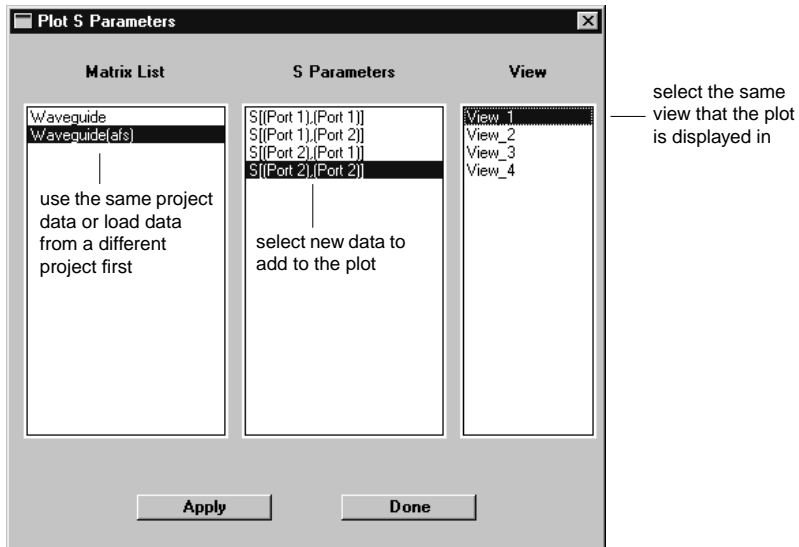
## Working with Data Controls

Data controls are available for editing the appearance of the following items:

- S-parameters
- The mesh
- Currents
- Far-fields

To access data controls, select the data from the Graphs list. Click the button that appears above the Graphs list. The illustration here is an example for S-parameter data or other data that appears on a rectangular plot.
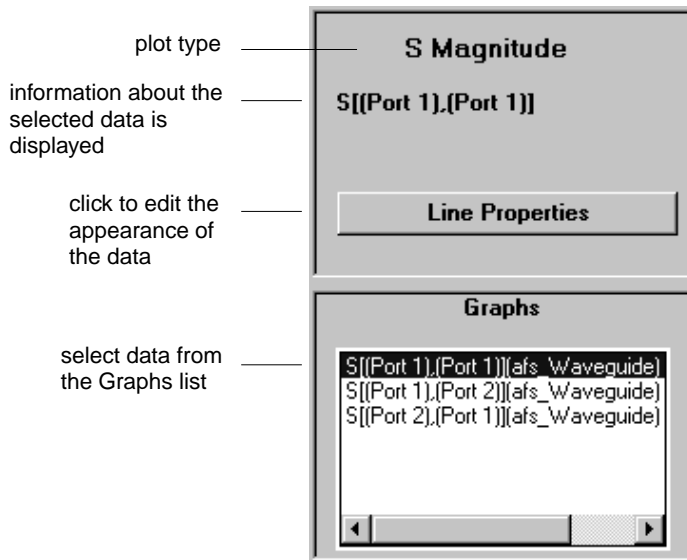


Data controls include:

- For rectangular plots, you can change the name of the data, the line color, and the line type used to display the data.
- For current plots, you can alter the appearance of the mesh, or animate currents.
- For far-field plots, you can set the translucency or constant Phi.

## Viewing Data from Another Project

The data from any Momentum or Agilent HFSS project can be added to a plot. After you load new projects, you can select among any of the projects for data to display.

To load an Momentum project:

1. Choose **Projects > Read Momentum Project**.

2. Navigate to the folder where the project is saved (project folder) and open the folder.

3. Under the project folder is another folder named *mom_dsn*. Open this folder.

4. Open the folder that contains your specific design.

5. Double-click on any file within this folder.

To load a new Agilent HFSS project:

1. Choose **Projects > Read Agilent HFSS Project**.

2. Select the project directory containing the project.

3. Select the project from the Projects list. Note the annotation at the bottom of the list, it will indicate if the selected project has saved solutions or not.

4. Click **OK**.

To change to a different project:

1. Choose **Projects > Select Project**.

2. Select a project from the list of Momentum or Agilent HFSS projects.

3. Click **Select Momentum** or **Select Agilent HFSS**.

---

**Note**   The title bar of the Momentum window will display the name of the currently selected project. Data will be retrieved from this project.

---

## Erasing Data from a Plot

Choose *Window > Erase* to delete part or all of the data from a plot. When the data is deleted, it is also removed from the Graph list.

1. Select the plot from which you want to erase the displayed data.

2. Choose **Window > Erase Plot**.

3. Select the data to be erased.

4. Click **Apply**.

5. Continue to select data and use **Apply** as needed.

6. Click **Done** to dismiss the dialog box.

## Reading Data Values from a Plot

Data for a single point can be retrieved from any plot. The values are displayed near the bottom of the Momentum window. The type of data that is returned is based on the plot type. Data is not retrieved from far-field plots.

To read the values of a single data point:

- Position the mouse on the data point of interest and click.

For rectangular plots and Smith charts, the returned values include the data name, and the values along the X and Y axes of the plot. In some cases, values may be rounded.



S-parameters displayed on a magnitude vs. frequency plot

S[[Port 1],[Port 1]](afs_dipole_us)

X  0.9277    Y  -138.7912

frequency, in GHz          magnitude, in dB
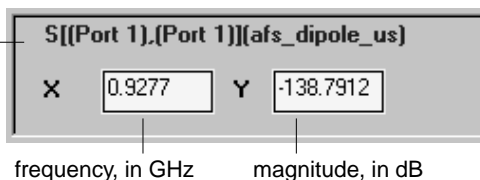
On current plots, you can return values selected on Objects added from the **Display** menu. These are the x, y, and z drawing coordinates of the layout.

The layout is displayed ——

Returns the x, y, and z drawing coordinates ——

| Objects(dipole_us) | | | | | |
|---|---|---|---|---|---|
| X | -0.0316 | Y | 0.0147 | Z | 0.0140 |
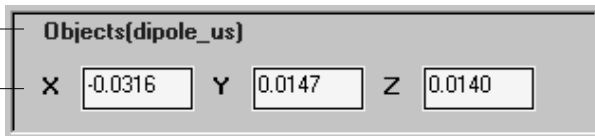
## Working with Plot Controls

You can use the plot editing controls at the bottom of the Momentum window to rotate, resize, and move plots to optimize the view of your data. When you are working with rectangular plots, you can adjust the scale of the plot.

### Rotating a Plot

The *Rotate* command moves a 3D plot in the X, Y, or Z direction. You can rotate Objects and 3D far-field plots only.

To rotate a plot:

1. Click **Rotate**.

2. Rotate the plot using one of the following methods:

   • Position the mouse anywhere on the plot, then drag the mouse up, down, or diagonally to rotate. To fix the plot position, release the mouse button.

   • Click **Rotation** to set X Rotation, Y Rotation, and Z Rotation values using scroll bars. Values range from -180 to 180 degrees.

   • Click **Views** to select a predefined view. Choices are **Front Angle**, **Back Angle**, **Top**, **Side**, or **Front**.

## Scaling a Plot

The *Scale* command enlarges or shrinks the size of a plot.

To scale a plot:

1. Click **Scale**.

2. To enlarge the plot, click anywhere on the plot and drag the mouse up or to the right.

3. To make the plot smaller, drag the mouse down or to the left.

## Moving the Centerpoint of a Plot

The *Pan* command moves a plot in the horizontal or vertical direction.

To move a plot:

1. Click **Pan**.

2. Click anywhere on the plot and drag the mouse to move the plot up, down, left, or right.

# Working with Rectangular-plot Editing Controls

Rectangular plots have additional plot controls for:

- Adjusting the x and y axes of the plot
- Adjusting the data legend on a plot

## Editing the Axes on a Plot

The options in the *Graph Properties* dialog box enable you to change the range of data that is displayed on a plot, to increase or decrease the number of lines in the plot grid, and to change the text that appears along each axis.

To change the data range on a plot:

1. Click **Graph Properties**.

2. The current boundaries are displayed in the X Axis and Y Axis Min and Max fields. To change the range of displayed data, type new values into the Min and Max fields for each axis.

3. Press **Enter** as you make each entry to view its effect on the plot.

To manually scale x, y axes:

1. Click **Graph Properties**.

2. Enter the number of units per division in the Div field. For example, if Div is set to 20 when Min is 0 and Max is 100, you would see 5 major divisions along the axis.
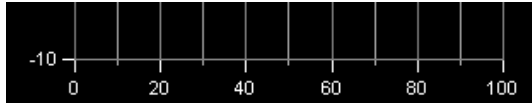


3. Enter the number of grid lines in each division in the Ticks field. In the figure above, Ticks is set to 1, so one line appears in each division.

4. Press **Enter** as you make each entry to view its effect on the plot.

To automatically scale the axes of a plot:

1. Click **Graph Properties**.

2. Click **Auto Scale** to automatically scale the x and y axes of a plot.

To change the text on each axis:

1. Click **Graph Properties**.

2. Type new text into the X Legend and Y Legend fields.

3. Press **Enter** as you make each entry to view its effect on the plot.

## Editing the Legend

When several sets of data are on a single plot, the legend can help to distinguish the data. The legend displays the colors used to display the data and the names of the data.

You can move and resize the legend, or erase it. You can also change the name of the data.
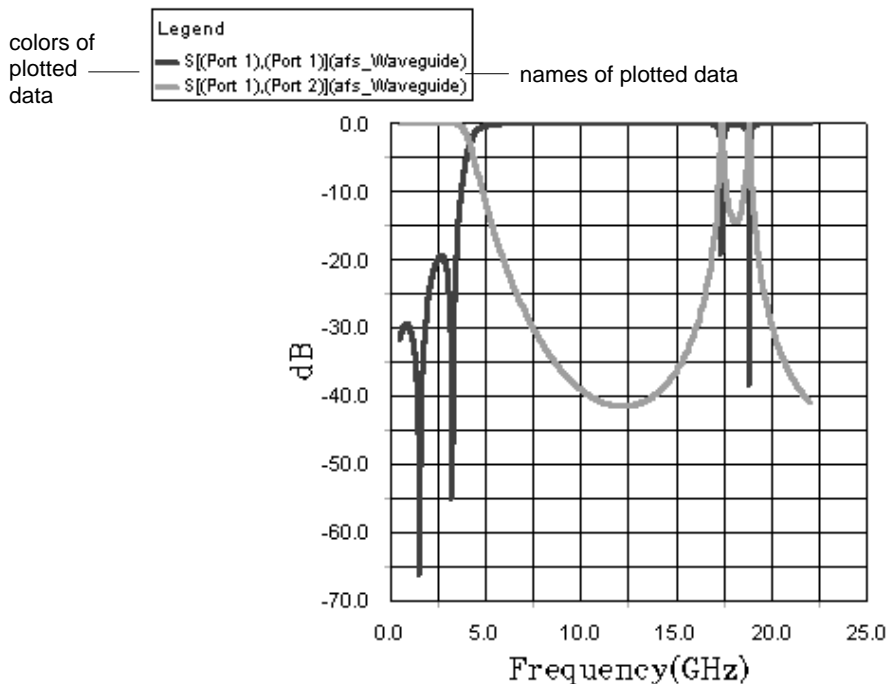
To edit the legend:

1. Click **Legend Properties**.

2. If the dialog box appears to be empty, enable Visible.

3. Use the X Location and Y Location scroll bars to adjust the position of the legend in the view.

4. Adjust the font size if the title using Legend Title Size.

5. Adjust the font size of the data names using Legend Entries Size.

6. To change the title above the data names, enter a different title in the Legend Title field.

7. Click **Done** to dismiss the dialog box.

To edit the name of the data:

1. Select the data from the Graphs list.

2. Click **Line Properties**.

3. Under Line Legend, enter a new name for the data.

4. Click **Done** to dismiss the dialog box.

# Saving a Plot

You can save plotted data and the plot type to a file. The data types include:
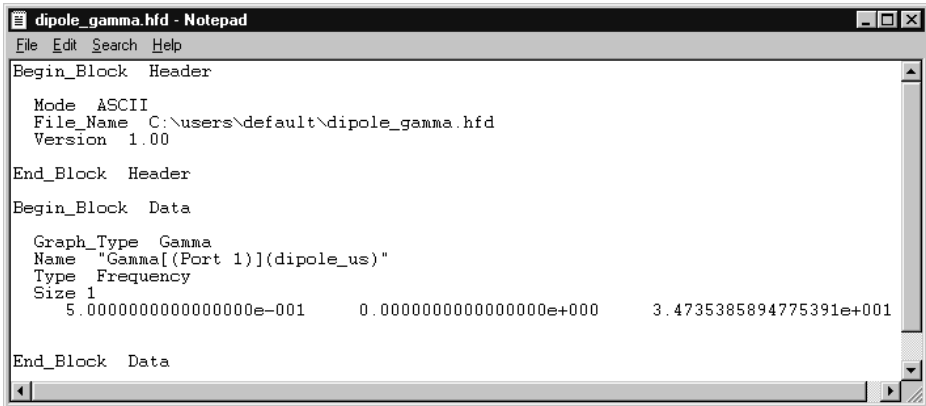
- S-parameter magnitude or phase
- Impedance
- Transmission data
- Far-field cuts

The file is in ASCII format, so you can use the data in other applications.

You can also import these files directly into Momentum Visualization. This eliminates the need to read in a project and then select the project in order to view the data.

To export plotted S-parameters:

1. Choose **File > Export Plot Data**.

2. Select the type of plot you want to export.

3. Under Plots, select the S-parameters that you want to apply to the plot.

4. Click **Browse**. Navigate to where you want to save the file and enter a file name. An extension (.*hfd*) will be automatically appended to the file name. Click **OK**.

5. If you are satisfied with your selections, click **Apply**. The calculated S-parameters and plot type are written to the file.

6. Continue to save other sets of S-parameters with other plot types, as desired. Be sure to click **Apply** so that the file is generated.

7. When you are finished, click **Done** to dismiss the dialog box.

8. The file is saved in ASCII format. You can view the file in any text editor.

```
dipole_gamma.hfd - Notepad
File  Edit  Search  Help

Begin_Block  Header

  Mode  ASCII
  File_Name  C:\users\default\dipole_gamma.hfd
  Version  1.00

End_Block  Header

Begin_Block  Data

  Graph_Type  Gamma
  Name  "Gamma[(Port 1)](dipole_us)"
  Type  Frequency
  Size 1
      5.0000000000000000e-001      0.0000000000000000e+000      3.4735385894775391e+001


End_Block  Data
```

# Importing a Plot

Plots that have been saved to a file in .hfd format can be imported. This eliminates the need to read in a project and then select the project in order to view the data.

To import a file:

1. Choose **File > Import Plot Data**.

2. Click **Browse** and select the file of interest.

3. The dialog box will display the following information:

   - *File Name*—the full path and name of the file

   - *Plot Type Label*—the type of plot that is saved in the file

   - *Plot Name Label*—the S-parameters that are saved in the file

   - *Import Name*—the name that is assigned by the post processor to the S-parameter plot. This is the name you will choose when you go to display the plot.

4. Click **Apply**. Continue importing S-parameter plots as desired.

5. When you are finished loading plot files, click **Done**.

To view an imported plot:

1. Choose **Plot > Plot Import Data**.

2. Select a plot type. A list of available plots is displayed.

3. Select the name of the plot that you want to view.

4. Select a view.

5. Click **Apply**. Continue adding plots to views as desired.

6. Click **Done** to dismiss the dialog box.

# Chapter 11: Displaying S-parameters

In Momentum Visualization, you can display S-parameters in tabular or plotted formats. This chapter describes how to view and work with S-parameters. It also gives general information about S-parameters.

## S-parameter Overview

When viewing and working with S-parameters, you should be aware how impedance is specified and what naming convention is used. These are described in the following sections.

### Normalization Impedance

The S-parameters that you can display are S-parameters with respect to a normalization impedance as defined in the Impedance fields of the Port Editor dialog box. If an impedance is not specified, 50Ω is assumed.

### Naming Convention

The S-parameter naming convention, which is based on the name of the project, indicates which set of S-parameters you are viewing or working with. There are only two data sets available for a project. One is named *design_name* which contains the discrete frequencies that were actually simulated. The second is *design_name(afs)* is available only when an adaptive frequency sweep is specified in the simulation setup. An adaptive frequency sweep data set contains a high resolution sampling based on the pole-zero model fitted on the simulated frequencies.

## Viewing S-parameters in Tabular Format

You can view S-parameters in tabular format. The table displays S-parameter values for each port in these formats:

- Magnitude
- Real and imaginary components
- Magnitude and phase

The table is for viewing data only, you cannot plot data from this table.

S-parameters are displayed as shown here.

Port number

S Matrix Display

**S Parameters**

dipole
dipole(afs)

(Port 1)

(Port 1)          (    0.9955      52.36)

Results in
magnitude
and phase

**Frequencies**

5.000000e+008
5.039063e+008
5.078125e+008
5.117188e+008
5.156250e+008
5.195313e+008
5.234375e+008
5.273438e+008
5.312500e+008

Results displayed
at selected
frequency

**Options**

○ Magnitude
○ Real/Imag
● Mag/Phase

Results displayed
in the selected
format

To view S-parameters:

1. Choose **Plot > Matrix**.

2. Select the data you want to view from the S Parameters list.

3. Select the frequency of interest from the Frequencies list. Some S matrices have data at only one frequency.

4. Select the data format:

   • **Magnitude**   Displays the magnitude of the S-parameters

   • **Real/Imag**   Displays the S-parameters as complex numbers

   • **Mag/Phase**   Displays the magnitude and phase of the S-parameters

5. Use the scroll bars to view large matrices.

6. Click **OK** to dismiss the dialog box.

# Plotting S-parameter Magnitude

The S-parameter magnitude plot displays magnitude in dB with respect to frequency.
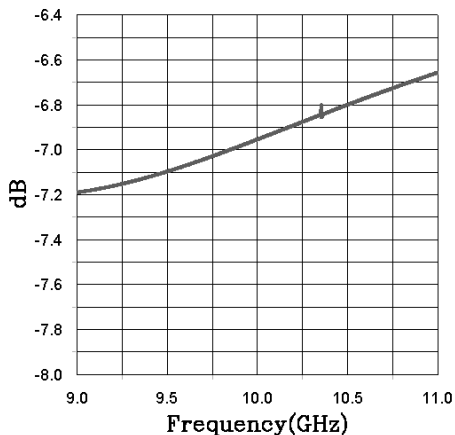


To display an S-parameter magnitude plot:

1. Choose **Plot > Mag Plot**.

2. Select an S matrix from the Matrix List.

3. Select the S-parameters. The combinations are based on ports.

4. Select the view where you want to display the plot.

5. Click **Apply**.

6. You can add other S-parameters to the plot, or click **Done** to dismiss the dialog box.

# Plotting S-parameter Phase

The S-parameter phase plot displays phase in degrees with respect to frequency. Note that when viewing S-parameter phase, in some cases the phase delay is plotted. For example, a layout of an electrical length with a phase of 60 degrees, $S_{21}$ represents a phase delay, and appears as -60 degrees. For phases greater than 180 degrees, it may

appear that the phase predicted from Gamma compared to $S_{21}$ is incorrect, but this is not so if delay is taken into account.
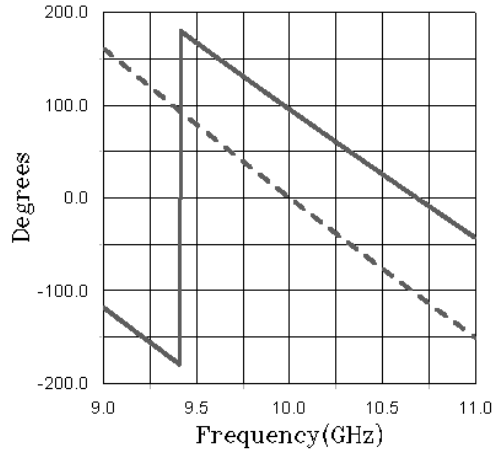


To display an S-parameter phase plot:

1. Choose **Plot > Phase Plot**.

2. Select an S matrix from the Matrix List.

3. Select the S-parameters. The combinations are based on ports.

4. Select the view where you want to display the plot.

5. Click **Apply**.

6. You can add other S-parameters to the plot, or click **Done** to dismiss the dialog box.

# Plotting S-parameters on a Smith Chart

The Smith chart displays real and imaginary components of S-parameters.
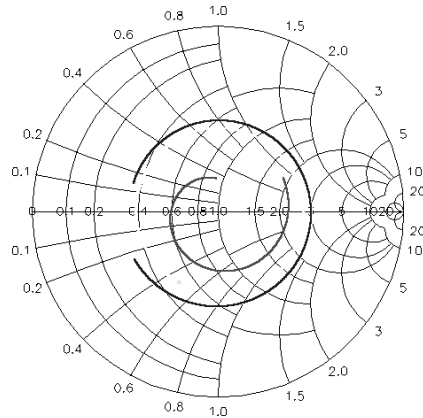


To display a Smith chart:

1. Choose **Plot > Smith Plot**.

2. Select an S matrix from the Matrix List.

3. Select the S-parameters.

4. Select the view where you want to display the plot.

5. Click **Apply**.

6. You can add other S-parameters to the Smith chart, or click **Done** to dismiss the dialog box.

# Exporting S-parameters

S-parameters can be exported to a file, in either Touchstone or CITIfile format. You can export S-parameters to use with other tools.

To export S-parameters:

1. Choose **File > Export S Matrix**.

2. Select the set of S-parameters to be exported from the Matrix list.

3. Select the file format, CITIfile or Touchstone.

4. Specify the file name either by typing the full file path into the File Name field or click **Browse**.

5. Click **OK** to complete the command.

# Chapter 12: Displaying Surface Currents

In Momentum Visualization, you can display electric surface current denisty (Amps/meter) on finite conducting objects, and magnetic surface currents on apertures in infinite ground planes. This chapter describes how to view currents.

## Setting Port Solution Weights

Before displaying a current plot, select a frequency and set the port solution weights for that frequency. Weighting port solutions enables you to specify the amount that any one port solution contributes to the solution at a given frequency. The weighting will be reflected in the current plots.

A Thevenin voltage source (voltage source + source impedance in series) is attached to each circuit port. The source impedance is either the characteristic impedance for single, differential, and coplanar ports; or 50 ohms for all other port types. The voltage source amplitude for each port is set to its corresponding solution weight (magnitude + phase). The displayed currents are those that correspond with this excitation state.

To set port solution weights:

1. Choose **Current > Set Port Solution Weights**.
2. Select a frequency.
3. Select a port, and enter the magnitude in the Solution Weight field, and enter the phase in the Solution Phase field.
4. Repeat the previous step for other ports, as necessary for the same frequency.
5. Click **OK** to complete the command.

## Displaying the Layout

An outline of the layout is loaded into a view (if it is missing) when the current plots are displayed. If a layout is not loaded or if it was deleted, choose *Display > Objects* to display the layout.

# Displaying a Current Plot

The current plot displays surface currents on all objects of the simulated layout. The plot can be animated to illustrate the currents propagating through the circuit.

---

**Note**    Make sure the layout is loaded into the view. Choose *Display > Objects* to load the layout if it is missing.

---

To display a current plot:

1. Choose **Current > Plot Currents**.

2. Select the graphical format:

   - **Arrow**   Plots a vector current quantity using arrows to indicate direction and magnitude.
   - **Contour**   Displays the "equal magnitude" contours of the vector current.
   - **Shaded**   Displays a shaded plot of a magnitude of the vector current. The magnitude being plotted is represented by a range of colors. A color scale showing the range of values also appears on the screen.

3. Select the view that you want to use to display the plot.

4. Click **OK**.

## Animating Plotted Currents

The currents have a sinusoidal, or harmonic, time dependence, with steady-state conditions assumed. There is an implied $e^{j\omega t}$ time dependence. Animation enables you to visualize the surface currents in the time domain, illustrating the propagation through a circuit. Changing time corresponds with changing the additional phase introduced by the $e^{j\omega t}$ factor. You can change this phase continuously or you can step through the phase changes manually.

To change the phase manually:

- Use the scroll bar. This enables you to freeze the phase in the layout.

To automate the animation:

1. Click **Display Properties**.

2. Enable **Animate**.

3. The currents are animated in time. The animation repeats automatically until Animate is disabled. The speed of the animation can be changed by editing the Increment field. A larger value speeds the animation, smaller values slow it down.

4. You can change the maximum value of the color scale. Depending on the selected graphical format, you can change other options:

   - For the shade and contour plots, you can set the translucency, and color of the current.

   - For the arrow plot, you can change the length of the arrow vector.

# Displaying the Mesh on a Current Plot

Choose *Display > Mesh* to add the mesh to a current plot.

To display a mesh:

1. Choose **Display > Mesh**.

2. Select the view displaying the plot to which you want to add the mesh. Meshes can be displayed in Objects only.

3. Click **OK** to complete the command.

---

**Tip** Depending on the drawing order, the mesh may be *under* the shaded current plot, and not visible. To see the mesh, lower the translucency of the shaded plot.

---

# Chapter 13: Displaying Radiation Results

In Momentum Visualization, you can view the following radiation data:

- Far-fields including E fields for different polarizations and axial ratio in 3D and 2D formats
- Antenna parameters such as gain, directivity, and direction of main radiation in tabular format

This chapter describes how to view the data. In Momentum RF mode, radiation results are not available for display. For general information about radiation patterns and antenna parameters, refer to Chapter 8, Radiation Patterns and Antenna Characteristics.

## Loading Radiation Results

In Momentum, computing the radiation results is included as a post processing step. The *Far Field* menu item appears in the main menu bar only if radiation results are available. If a radiation results file is available, it is loaded automatically. The radiation results file contains only the radiation results for a single frequency (the one that was last calculated). Since this file is loaded only when a new Momentum project is opened, you must reopen the project to load radiation results calculated at another frequency.

---

**Note**  The command *Set Port Solution Weights* (in the *Current* menu) has no effect on the radiation results. The excitation state for the far-fields is specified in the radiation pattern dialog box before computation.

---

You can also read in far-field data from other projects. First, select the project containing the far-field data that you want to view, then load the data:

1. Choose **Projects > Select Project**.
2. Select the name of the Momentum or Agilent HFSS project that you want to use.
3. Click **Select Agilent HFSS** or **Select Momentum**.
4. Choose **Projects > Read Field Solution**.

5. When the data is finished loading, it can be viewed in far-field plots and as antenna parameters.

# Displaying Far-fields in 3D

The 3D far-field plot displays far-field results in 3D.

To display a 3D far-field plot:

1. Choose **Far Field > Far Field Plot**.

2. Select the view in which you want to insert the plot.

3. Select the E Field format:

   • *E = sqrt(mag(E Theta)2 + mag(E Phi)2)*

   • *E Theta*

   • *E Phi*

   • *E Left*

   • *E Right*

   • Circular Axial Ratio

   • *E Co*

   • *E Cross*

   • Linear Axial Ratio

4. If you want the data normalized to a value of one, enable Normalize. For Circular and Linear Axial Ratio choices, set the Minimum dB. Also set the Polarization Angle for E Co, E Cross, and Linear Axial Ratio.

5. By default, a linear scale is used to display the plot. If you want to use a logarithmic scale, enable Log Scale. Set the minimum magnitude that you want to display, in dB.

6. Click **OK**.

## Selecting Far-field Display Options

You can change the translucency of the far-field and set a constant phi angle:

1. Click **Display Options**.

2. A white, dashed line appears lengthwise on the far-field. You can adjust the position of the line by setting the Constant Phi Value, in degrees, using the scroll bar.

3. Adjust the translucency of the far-field by using the scroll bar under Translucency.

4. Click **Done**.

# Defining a 2D Cross Section of a Far-field

You can take a 2D cross section of the far-field and display it on a polar or rectangular plot. The cut type can be either planar (phi is fixed, theta is swept) or conical (theta is fixed, phi is swept). The figure below illustrates a planar cut (or phi cut) and a conical cut (or theta cut), and the resulting 2D cross section as it would appear on a polar plot.

The procedure that follows describes how to define the 2D cross section.



planar cut at
phi = $0^0$

conical cut at
theta = $90^0$
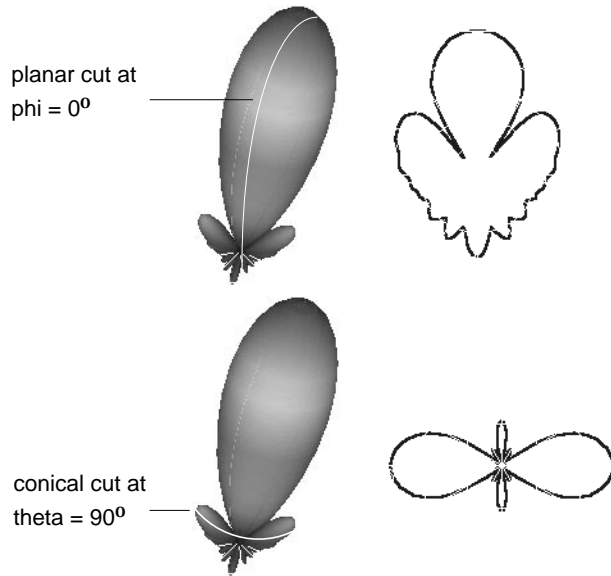
To define a cross section of the 3D far-field:

1. Choose **Far Field > Cut 3D Far Field**.

2. If you want a conical cut, choose Theta Cut. If you want a planar cut, choose Phi Cut.

3. Set the angle of the conical cut using the Constant Theta Value scroll bar or set the angle of the planar cut using the Constant Phi Value scroll bar.

4. Click **Apply** to accept the setting. The cross section is added to the Cut Plots list.

5. Repeat these steps to define any other cross sections.

6. Click **Done** to dismiss the dialog box

# Displaying Far-fields in 2D

Once you have defined a 2D cross section of the 3D far-field plot, you can display the cross section on one of these plot types:

- On a polar plot
- On a rectangular plot, in magnitude versus angle

In the figure below, a cross section is displayed on a polar and rectangular plot.
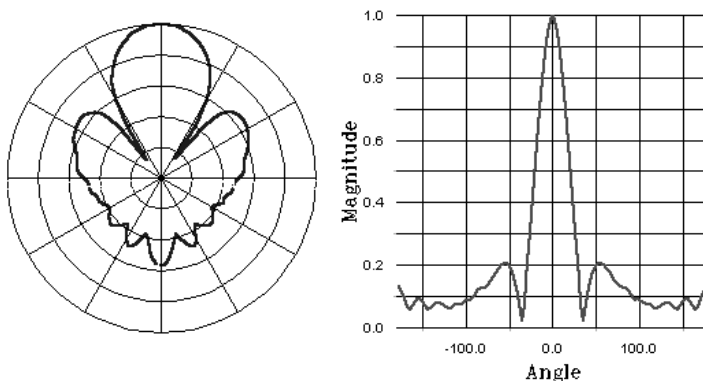


To display a 2D far-field plot:

1. Choose **Far Field > Plot Far Field Cut**.

2. Select a 2D cross section from the 2D Far Field Plots list. The type of cut (phi or theta) and the angle identifies each cross section.

3. Select the view that you want to use to display the plot.

4. Select the E-field format.

5. Select the plot type, either Cartesian or Polar.

6. If you want the data normalized to a value of one, enable Normalize.

7. By default, a linear scale is used to display the plot. If you want to use a logarithmic scale, enable Log Scale. If available, set the minimum magnitude that you want to display, in dB; also, set the polarization angle.

8. Click **OK**.

# Displaying Antenna Parameters

Choose *Far Field > Antenna Parameters* to view gain, directivity, radiated power, maximum E-field, and direction of maximum radiation. The data is based on the frequency and excitation state as specified in the radiation pattern dialog. The parameters include:

- Radiated power, in watts
- Effective angle, in degrees
- Directivity, in dB
- Gain, in dB
- Maximum radiation intensity, in watts per steradian
- Direction of maximum radiation intensity, theta and phi, both in degrees
- E_theta, in magnitude and phase, in this direction
- E_phi, in magnitude and phase, in this direction
- E_x, in magnitude and phase, in this direction
- E_y, in magnitude and phase, in this direction
- E_z, in magnitude and phase, in this direction

Details about each of these parameters can be found in "About Antenna Characteristics" on page 8-2.

---

**Note**   In the antenna parameters, the magnitude of the E-fields is in volts.

---

# Chapter 14: Displaying Transmission Line Data

In Momentum Visualization, you can display the following transmission line data for all calibrated ports, except coupled ports:

- Propagation constant
- Characteristic impedance

The following port types are calibrated:

- Single
- Differential
- Coupled
- Coplanar

For each calibrated port, an infinitely long (coupled) transmission line with the same cross section as the port is solved using a cross section solver. For single ports, the propagation constant and characteristic impedance is a complex but scalar quantity. For differential, coupled, and coplanar ports, those are both matrices with a dimension equal to the number of ports involved (nodal representation). For differential and coplanar ports, the modal gamma and characteristic impedance for the differential or coplanar waveguide mode are calculated, too. Those are scalar quantities again and can be visualized by Momentum Visualization.

The characteristic impedance, Z0, at each port is computed from power and current using the relationship $Z0 = 2 P / (I.I^*)$.

You can view the data either in a table or on plots. In Momentum RF mode, transmission line data are not available for display.

This chapter describes how to view the data. It also provides general information about gamma and characteristic impedance.

# Viewing Transmission Data

You can review gamma and characteristic impedance in a tabular format. The table displays gamma and the characteristic impedance for all single, differential, and coplanar ports present in the circuit. For all other ports, gamma is set to 0 [1/m], and the characteristic impedance is set to 50 [ohm].

The table is for viewing data only; you cannot plot data from this table.

To view transmission data:

1. Choose **Plot > Port Data**.

2. Select the matrix containing the data that you want to view.

3. Use the scroll bars to view large matrixes.

4. Click **OK** to dismiss the dialog box.

# Displaying Gamma on a Plot

The gamma plot displays the magnitude, real or imaginary components of the propagation constant gamma. Real values indicate the decay of a wave; imaginary values indicate the phase velocity through the media.

To display a gamma plot:

1. Choose **Plot > Gamma Plot**.

2. Select a matrix from the matrix list.

3. Select the port for which you want to view gamma.

4. Select the view where you want to display the plot.

5. Click **Apply**.

To select the component that you want to display:

1. Select the data that you just plotted from the Graphs list.

2. Click **Line Properties**.

3. Under Plot Type, select the component you want to display: magnitude, real, or imaginary.

4. Click **Done**.

# Displaying an Impedance Plot

The impedance plot displays the characteristic impedance of a port. The value represents the characteristic impedance of a transmission line extending from the port and having the same cross section as the port.

To display an impedance plot:

1. Choose **Plot > Impedance Plot**.

2. Select a matrix from the matrix list.

3. Select the port from the Impedance list.

4. Select the view where you want to display the plot.

5. Click **Apply**.

# Chapter 15: Momentum Optimization

Momentum Optimization, in combination with Momentum, is an effective software tool for automated electromagnetic (EM) design optimization of planar structures. Momentum Optimization automatically adjusts the layout in order to improve the circuit performance according to your design goals, and supports designs developed using the Momentum microwave or Momentum RF mode.

Momentum is invoked directly by the optimizer for accurate EM simulations of various layout instances. This provides the optimizer with information with which it makes decisions on how to modify the layout.

Momentum is an analysis tool that can be used to accurately account for various EM effects such as couplings or radiation. It is extremely valuable for design verification, but cannot, by itself, determine the layout that meets your design goals. Physical design is usually a trial and error process in which the designer uses Momentum to achieve the goals. By automating this process, Momentum Optimization saves valuable engineering time. It can significantly shorten the design cycle, facilitating faster time-to-market, and for more complex structures, it can usually find better designs than those found manually.

---

**Note**    You may want to refer to the examples at the end of this chapter to become familiar with terminology and basic procedures in using Momentum Optimization.

---

# What can be optimized by Momentum Optimization?

Momentum Optimization enables you to optimize geometrical dimensions of 2D layout objects (shape optimization). This includes optimizing:

- Typical dimensions such as lengths, widths, gaps, spacing, or diameters
- Interdependent layout modifications such as length and width varying simultaneously
- Many unconventional layout modifications

The objects can be metal strips, slots or vias. The underlining technology of object parameterization provides for significant flexibility in layout modification. The parameterization process is intuitive and is carried out within the ADS layout drawing environment.

## What cannot be optimized by Momentum Optimization?

This version of Momentum Optimization does not accommodate optimization of

- Material parameters such as the dielectric constant
- Layout cross-section parameters such as the substrate thickness
- Discrete parameters such as the number of turns of a spiral inductor
- Parameters changing layout topology (including changing the number of object vertices)

## What responses can be optimized by Momentum Optimization?

Momentum Optimization allows you to set design goals for the S-parameters in any of the following forms:

- Magnitude in dB
- Magnitude
- Phase (in degrees)
- Real part
- Imaginary part

Any combination of the above forms for any selection of port indices can be optimized simultaneously. The S-parameters available for optimization are with respect to 50Ω reference impedance.

# Typical steps in using Momentum Optimization

In general, a typical Momentum Optimization session would take place as follows:

1. Create an ADS layout to the best of your design experience. Draw it as close as possible to what you expect the final design to be. You do not need, however, to

spend a lot of time selecting exact dimensions. Rather, take advantage of drawing convenience to save time.

2. Setup an Momentum simulation: substrate, metallization, port, meshing, etc.

3. Simulate your design using the *Momentum > Simulation* menu option. View the responses of interest. If you are satisfied with the circuit performances you do not need to run Momentum Optimization.

4. Make as good a guess as possible whether some adjustments to layout dimensions could improve your circuit performance. Consider how the layout should be modified to achieve the desired improvement. If a fundamental change to the layout topology is needed, do it now and repeat the simulation.

5. Define conceptually as few independent parameters as possible that could facilitate the desired layout modifications (see Appendix F, Momentum Optimization - Layout Parameterization for more details and hints).

6. Decide on how the circuit performance can be formally expressed by design goals in terms of frequency points, S-parameters, and upper, lower, equality or window specifications.

7. Save your design and invoke *Momentum > Optimization* menu option.

## The Optimization Process

After choosing *Momentum > Optimization* you need to sequentially step through the three submenu options:

**1. Parameters**

**2. Goals**

**3. Run**

It is recommended that you initially accept as many default settings as possible. The necessary actions for each step are outlined as follows.

- **Layout Parameterization**  You need to define at least one parameter to be able to perform an optimization. Momentum Optimization allows you to define geometrical parameters in an intuitive manner within the layout drawing environment of ADS.

- To define a parameter you need to provide a separate design to show the program how an incremental change in that parameter value affects your original layout. The original layout is referred to as the "nominal design" and

the new one is called a "perturbed design". The program will conveniently guide you through this process. When you add a new parameter a separate design containing a copy of the nominal design is created. You should make your changes in this design. The modified copy is then saved as the perturbed design.

- You need to define a separate perturbed design for each parameter.

- **Design Goals**   You need to define at least one design goal to be able to perform an optimization. A single design goal consists of a frequency plan (a single point or a linear sweep), a response of interest (e.g., the magnitude of S21), the specification type, and a numeric value that the response should either exceed or achieve.

- For multiple design goals you can emphasize/de-emphasize their relative importance using weights associated with each goal.

- **Optimization Run**   In most situations you should be able to accept all default settings in the corresponding dialog box (Optimization Control window).

- Once at least one parameter and one goal are defined you can click **Start** to invoke the optimizer. The optimization progress is displayed in the Momentum Optimization window.

The log file of the optimization progress is available for viewing and/or printing at any time at the menu option *Momentum > Optimization > Summary.*

---

**Note** A working design is created during the optimization process with the name *designName_***work** (the original design file name is appended with "_work").

---

## When Optimization is Finished

Once the optimization process is finished the best (optimal) layout is shown in a separate window and saved as a separate design. Furthermore, the results of all Momentum simulations carried out during the optimization are saved in a database. This can significantly save time in future optimization runs.

In order to assess the circuit performance after optimization is finished you need to view the responses corresponding to the best layout found by the optimizer. You can do it in one of the following two ways.

- Use the Display menu in the Momentum Optimization window, or

- Invoke the *Momentum > Simulation* menu option from within the optimal design layout window, perform simulation and use the Data Display Server.

## Restarting Optimization

You can always go back to *Momentum > Optimization > Run* and restart optimization. If you do it keeping the same starting point (the *Starting Value* field) as in the previous run, the Momentum results will be quickly retrieved from the database, and the optimization will run very quickly. If you start from a different point, some additional Momentum simulations may be performed. The results of those simulations will be added to the database.

Caution needs to be exercised when you want to restart optimization after making modifications to the optimization setup (see "Removing Obsolete Databases" on page E-3).

# Working with Optimization Parameters

The geometric parameters that can vary during optimization are defined in the Optimization Parameters Editor. Choose *Momentum > Optimization > Parameters* to open the dialog box:

## Parameters (list)

This box contains the list of parameters for optimization. Three parameter attributes are displayed in the list:

Starting point for the optimization

Optimization result

Indicates if the parameter will be considered as variable by the optimizer (yes), or its value is to remain fixed (no) during optimization.

| Parameters |  |  |  |
|---|---|---|---|
| Name | Starting value | Optimal value | Allowed to vary |
| length | 100 |  | yes |
| width | 25 |  | yes |

## Parameter Definition

This box contains edit fields and buttons to define a parameter:

An ASCII string to identify the parameter. Only alphanumeric characters and the underscore are allowed. The first character must be alphabetic.

| Parameter Definition |  |  |
|---|---|---|
| Name | width |  |
| Nominal Value | 10 | mil |
| Perturbed Value | 20 | mil |
| Starting Value | 15 | mil |

The value of the parameter corresponding to the nominal project

Changing the unit will not rescale the layout

The value of the parameter corresponding to the perturbed design

The current value of the parameter. In optimization, this value is used as the starting point. By default, the starting values are set to the nominal parameter values. You may also enter any meaningful value that is different from both the nominal and perturbed values.

The values do not have to be true geometrical dimensions. But it is recommended that you use meaningful values.

**Note** On Windows platforms (Win95, Win98, NT), parameter names are case insensitive. They must be distinguished by some means other than case (for example, *width* and *Width* are considered as the same name).

In general, all the defined parameters are listed in the window as candidates for optimization variables. You can select an individual parameter as a variable by clicking the check mark box associated with that parameter. To undo the selection, click the check mark box again. In this way, you can select any subset of the parameters defined as variables while the others are kept fixed at the values specified as Starting Values. Two buttons are available to access more settings, and to view a perturbed design. If you click *More*, see "Using Advanced Parameter Settings" on page 15-8 for more details.

| More... | Click to access a dialog for more settings. |
|---|---|
| View/Modify Perturbed Design | Click this button to display the perturbed design associated with the selected parameter |
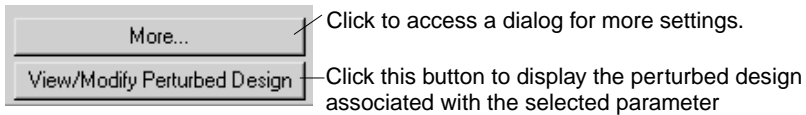
Two buttons are provided to view and control the start design:

| View Start Design | Back Annotate Optimal Values |
|---|---|
| Click this button to display the starting design for optimization | Click this button after optimization is finished to update the starting values with the optimal values |
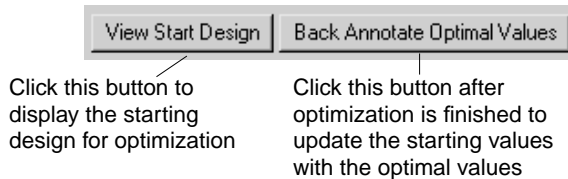
# Using Advanced Parameter Settings

Clicking *More* in the Optimization Parameters Editor enables to access advanced settings. These settings apply individually to the parameter currently highlighted in the Parameters Editor.
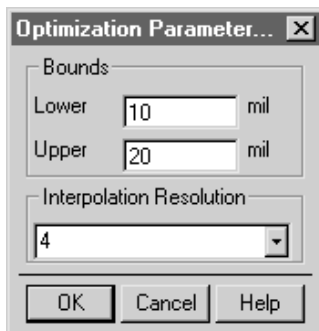


**Lower and Upper Bounds** - You may enter a lower and an upper bound for the parameter. This forces the parameter value to stay within these bounds during optimization.

Although the bounds are optional, they are useful for preventing the optimizer from producing invalid layouts (for example, due to overlapping objects). If you choose not to specify explicit bounds, the default bounds are defined to ensure that the parameter values do not change signs during optimization:

- If both the nominal and perturbed parameter values are greater than or equal to zero, then the default lower bound is zero and the upper bound is the layout limit.

- If both the nominal and perturbed parameter values are lower than or equal to zero, then the default lower bound is the layout limit and the upper bound is zero.

- If the nominal and perturbed values have opposite signs, then the default lower and upper bounds are the layout limit.

---

**Note**   The layout limit is controlled by the design precision. You can increase the limit by reducing the design precision in the Layout/Units tab of the Option/Preferences dialog.

---

**Interpolation Resolution** - The number of interpolation intervals between the nominal and the perturbed values. Momentum Optimization employs interpolation to significantly enhance its efficiency. The optimizer invokes Momentum for EM simulation only if the parameter value is moved across an interpolation interval. For small changes in parameter values, interpolation is applied instead of EM analysis to obtain the S-parameters. This can substantially reduce the number of Momentum calls required for optimization. See Appendix E, Momentum Optimization - Response Interpolation for further discussion on interpolation.

# Adding Parameters to Designs

The basic steps required for adding a parameter through the Optimization Parameter Editor are depicted in the following figure:

6. New parameter appears here and a new layout window opens

1. Enter a name

2. Specify the nominal and perturbed values

3. Optionally, specify a starting value, bounds and the interpolation resolution

5. Click Add

4. Make sure this box is checked

7. An information dialog explains how you can modify the design to describe the direction of change for the parameter.



9. Draw the changes resulting from a variation in the parameter value

8. Check this box if you do not want to see this dialog again

A copy of the original (nominal) design is shown in the new layout window. This copy is named *<dsnName>_<paramName>.dsn*, where *<dsnName>* is the name of your

nominal Momentum design and *<paramName>* is the name of the parameter. This layout is referred to as the "perturbed design" for the parameter *<paramName>.*

Edit the perturbed design to reflect the changes to the structure resulting from a variation in the parameter value. By comparing the nominal and perturbed designs, the optimizer will automatically extract the information needed for translating a parameter value into the corresponding geometrical coordinates.

Repeat this stepped procedure to define more parameters. When you have finished, click **OK** to close the dialog and the perturbed design window. The perturbed designs are saved automatically at that time.

---

**Note**    Once you have created the perturbed designs, you should not alter the original design.

---

## Editing Parameters

To edit one or more parameters:



1. Choose a parameter

2. The parameter definition is displayed here.

Edit name and values

4. Click to close dialog and save changes.

3. Click Update or press Return to apply changes.

---

**Important**    If you have changed the parameter's name, the perturbed design is renamed automatically when you click Update. This operation cannot be undone.

---

To delete a parameter:

1. Highlight the parameter in the Parameters list box.

2. Click **Delete**. The entry for the parameter is deleted from the list.

3. Use **Undelete** to add the parameter back if you change your mind.

---

**Note**    The perturbed design associated to the parameter removed from the list is not deleted by the program. Therefore, it can be reused if you decide to add the parameter back after closing the dialog.

---

## Verifying Your Changes

To verify that the incremental changes are correctly interpreted, you can have Momentum Optimization generate the starting design, and then visually inspect the results.

To generate the starting design:

1. Click **View Start Design** in the Optimization Parameters Editor.

2. A Simulation/Synthesis Message window appears. The relevant differences between the nominal and perturbed designs are extracted to establish the dependency of the design layout on the parameters.

3. When the comparison process is finished, the starting design for optimization is displayed in a new layout window. It is named *<dsnName>_work* where *<dsnName>* is the name of the nominal Momentum design.

If parameterization fails, watch for a message describing the problem in the Simulation/Synthesis Message window:
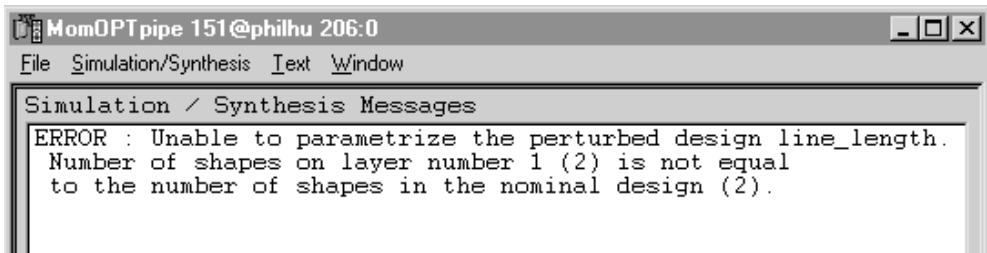
If an incremental change cannot be parameterized, Momentum Optimization might open two error layout windows. The nominal design is displayed in the window *<dsnName>_errNom*, while the perturbed design is displayed in the window *<dsnName>_errPer*. The name of the perturbed design displayed in *<dsnName>_errPer* is displayed in the Simulation/Synthesis Message window. Shapes for which the mapping cannot be found are highlighted in these windows. Their number can be different if connectivity problems occur or if the number of vertices differs. For instance, if the nominal design has two rectangles and the perturbed design one triangle and one rectangle, then only the triangle in the perturbed design will be highlighted. After all, the one rectangle in the perturbed design might be a valid mapping for each rectangle in the nominal design.

## Viewing and Editing Your Design

You can view and edit a perturbed design by clicking **View/Modify Perturbed Design:**



After viewing or editing a perturbed design, you can select another parameter. The corresponding perturbed design is then shown automatically.

Clicking **OK** in the dialog will close and save all perturbed designs opened for editing.

# Working with Optimization Goals

Optimization goals express the objectives you wish to achieve through optimization. They are defined in the Design Goals Editor. Select **Momentum > Optimization > Goals** to open the dialog:

Values of selected goal.

Displayed for editing

List of goals

Buttons to add or edit goals

Close dialog and save changes

Save changes without closing dialog

Close dialog without saving changes

## Edit/Define Goal box

The first line in the Edit/Define box enables you to specify the frequency range:

A change in one of these fields causes the remaining field to be calculated automatically

The second line enables you to express the goal:



Recognized S-parameter responses include the magnitude, the magnitude evaluated in dB, the phase evaluated in degrees, the real part, and the imaginary part

Optional weighting factor. The default value is 1. If given, the weighting factor must be a positive number.

The **Activated** check box controls whether or not the selected goal is considered during optimization. By default this option is on. If you turn it off, the frequency plan will be calculated and saved in the database, but a weighting factor of zero will be assigned to the goal. Thus, deactivating does not obsolete the database.

☑ Activated

See "Design Goals/Specifications" on page D-2 for further information on design goals.

## Defining a Goal

In general, the definition of a goal involves the following steps:

1. Define the frequency range: enter the start and end frequencies. Enter also a frequency step or a number of points.

2. Define a goal for an S-parameter response

3. Optionally, enter a numerical value for the weighting factor

4. Make sure this box is checked

5. Click Add

6. New goal appears here

Repeat this stepped procedure for all goals you want to add. When you are through adding goals, click **OK** to dismiss the dialog box.

To edit one or more parameters for a goal:

1. Highlight the goal to edit in the Goal list box.

2. The dialog box displays the parameters of the selected goal.

3. Type the new values in the parameter value editing field.

4. Click **Update**. The Goal list box is updated to reflect the new values.

To delete a goal:

1. Highlight the goal to delete in the Goal list box.

2. Click **Delete**. The entry for the goal is deleted from the list.

3. Use **Undelete** to add the goal back if you change your mind.

# Performing an Optimization

Before performing an optimization, you have the opportunity to set several options that control the optimization. After setting the options, you can start the optimization and monitor its progress. The following sections describe these topics in detail.

## Setting Optimization Controls

A variety of optimization options are available from the Optimization Control Editor. Choose **Run** in the **Momentum > Optimization** menu to bring up this dialog:
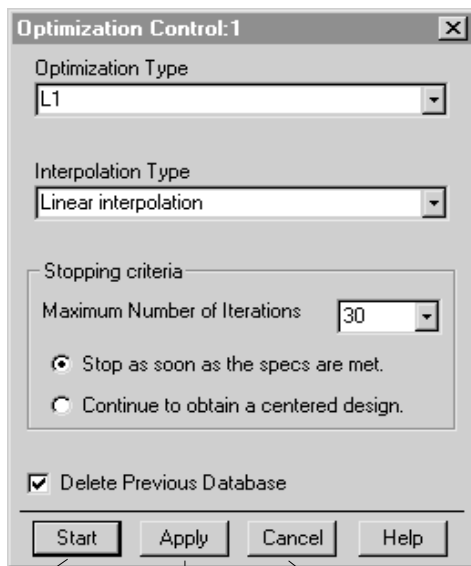


Toggle button to Start and Stop optimization

Save changes without closing dialog

Close dialog without saving changes

The Optimization Control Editor provides the following options:

**Optimization Type** Choice of the optimization type (algorithm and/or objective function). You can select a specific optimization type from this list: Minimax, L1, L2, Quasi-Newton, Random, and Simulated Annealing. See "Optimization Algorithms" on page D-6 for further discussion on optimization types.

**Interpolation Type** Choice of the interpolation type. Momentum Optimization utilizes response interpolation in order to significantly speed up the optimization process. Circuit responses corresponding to parameter values that do not fall on a user-defined grid are approximated by interpolating Momentum responses calculated at the neighboring on-grid values. See Appendix E, Momentum Optimization - Response Interpolation for further discussion on interpolation.

## Stopping Criteria

- **Maximum Number of Iterations**   This option enables you to limit the number of iterations. If the optimization does not converge to an optimal solution before reaching this limit, it will stop and present, as the solution, the best result up to this limit.

- **Stop / Continue**   The *Stop* option enables you to interrupt the optimization process as soon as all the specifications are satisfied. If you prefer a well-centered design where the responses exceed the specifications as much as possible, select the *Continue* option.

See "Stopping Criteria for the Optimization Process" on page D-7 for further discussion on the stopping criteria.

**Delete Previous Database**   Momentum Optimization employs a database system to avoid duplicate EM analysis. Every time Momentum Optimization invokes Momentum, the simulation results are stored in a database together with the corresponding parameter values. Subsequently, when Momentum analysis results are needed, Momentum Optimization checks this database first.

You may choose to have Momentum Optimization delete the database before starting an optimization; to do this turn on this option.

This option must be selected whenever the existing database becomes no longer compatible with the current setup. Any re-definition made within the Parameters or Specifications editors is likely to necessitate deletion of a previously stored database. See "Actual EM Simulations and Database" on page E-3 for further discussion on the database.

## Starting an Optimization

To start optimization, click the button *Start*. A Simulation/Synthesis Message window appears.

The relevant differences between the nominal and perturbed designs are first extracted to establish the dependency of the design layout on the parameters. When the comparison process is finished, the Advanced Design System invokes Momentum Optimization which then launches its own optimization monitor and display window. A new layout window also opens in which the starting design is shown. During optimization, you can follow the evolution of your design in this window.

**Tip** As the layout windows are made active each time a new layout is shown, you may find helpful to minimize the windows during optimization. Also, you can configure ADS to leave the Simulation/Synthesis Message window in the background. To do this add the following line to the configuration file hpeesofsess.cfg:

```
SIMEXEC_MAP_TO_FRONT = off
```
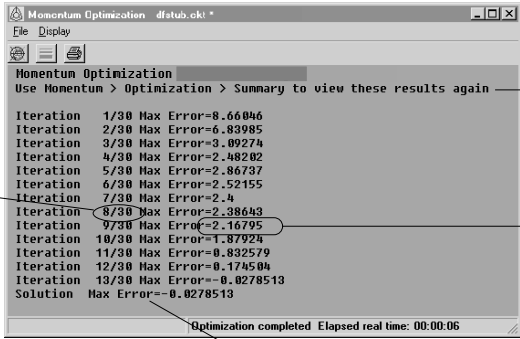
You can interrupt an optimization in progress.

To stop an optimization:

1. Open the Optimization Control Editor.

2. The *Start* button label is changed to *Stop.* Click *Stop.*

# Viewing Optimization Status

The progress of optimization is reported in the optimization window:



You can view this information any time after optimization is finished using the summary menu option.

Iteration count

Value of the objective function

Optimizer stops when all design goals are met

Optimization techniques employ iterative algorithms. An iteration leads to performing an optimization step, i.e., changing the optimization variables. As a result, the objective function is normally decreased.

It is advisable to monitor how the optimization process proceeds. Occasionally, it is possible that the optimizer makes a step in the up-hill direction. This is not an error.

It is inherent in the optimization algorithms as various search directions and steps are being explored. The best result will be retained as the optimized solution.

---

**Note** The optimization window has a different appearance on Windows and Unix.

---

# Viewing Optimization Results

When the optimization is finished, the optimized design is created and displayed in a separate layout window. It is named *<dsnName>_opt* where *<dsnName>* is the name of the nominal Momentum design. The value of the parameters can be read in the Parameter dialog which opens automatically:

Optimization results

Optimized design



Nominal design
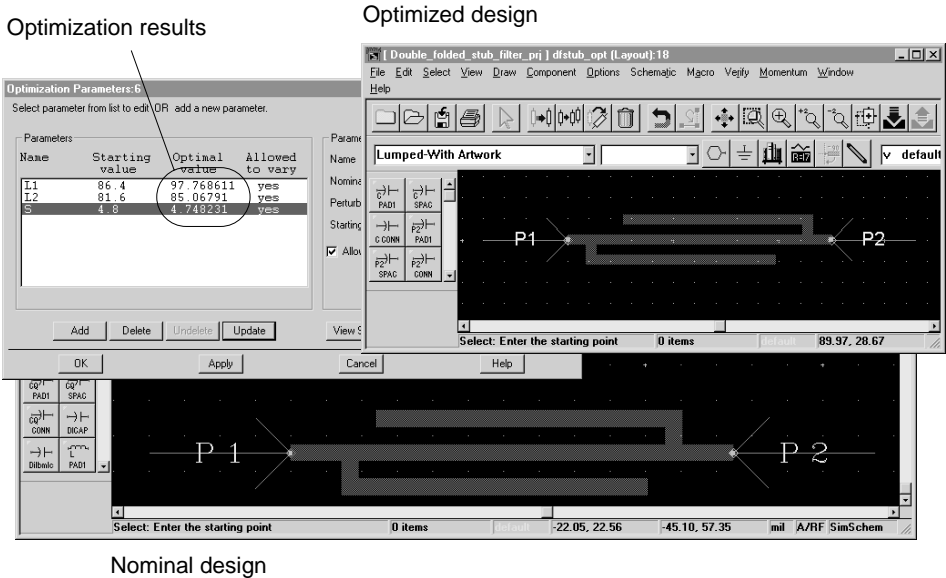
You can check if all optimization goals are met in the optimization data display:

# On a PC

1. Choose Rectangular to display responses versus frequency or Smith/Polar to display complex responses
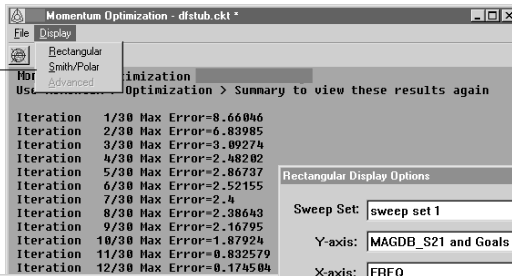
2. Select the response to display and, optionally, change the display options.
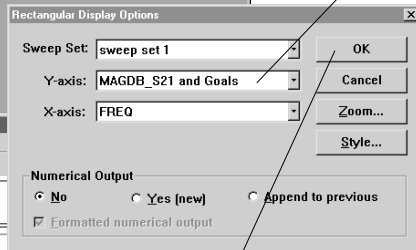
3. Click OK
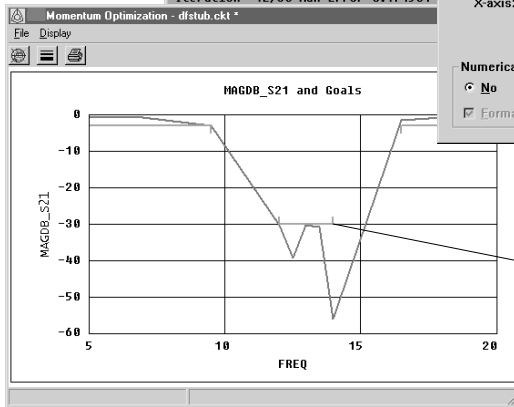
4. The response is displayed in the optimization window. Design goals are superimposed.

# On a Unix System



1. Click display

2. The MomOpt.Display menu appears. Click Rectangular to display responses versus frequency or Smith/Polar to display complex responses

3. A pop-up window appears. Select the response to display on the "Y-axis" line. Optionally, change the default display settings

4. Click Go
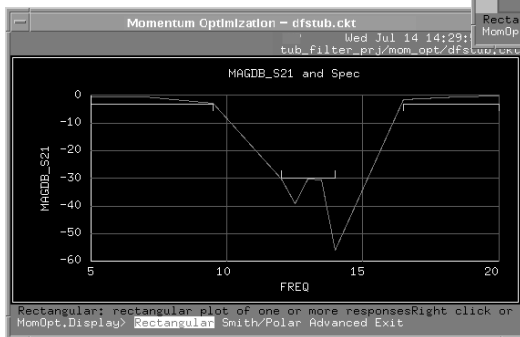
5. The response is displayed in the optimization window. Design goals are superimposed.

# Example - Optimizing a Microstrip Line

This exercise shows how to perform an optimization on a microstrip line. The microstrip is initially designed as a simple 100 mil L x 25 mil W line. The microstrip substrate consists of layers of air, alumina, and a ground plane, where the microstrip rests on the 25-mil alumina layer. The alumina layer dielectric constant is 9.8. The metallization of the microstrip is assumed to be lossless.

The goal is to optimize the length L of the line to satisfy:

Phase S21 = -120º at f = 10 GHz

The sections that follow describe how to optimize the microstrip design to achieve this goal.

## Step 1: Creating the Project and Layout

This section describes how to create a project and draw the microstrip.

If you want to skip this and other entries, an example project containing the complete microstrip design is in the *Examples* directory. Examples are in read-only format, so to work with an example you need to make a copy of it.

The microstrip in the example project has already been simulated and optimized. Since Momentum simulations and optimizations can reuse data, the optimization will run rapidly.

### Creating the Project

1. From the Main window, choose **Options > Preferences**. Ensure that **Create Initial Layout Window** is enabled. Click **OK**.

2. From the Main window, choose **File** > **New Project**. The New Project dialog box appears.

3. In the *Name* field, type **mstrip**.

4. Click **OK**.

## Creating the Initial Layout

This section describes how to draw the microstrip layout.

1. In the Layout window, ensure that the current layer is **cond**. You can change the current layer using from the layout list on the toolbar.

2. From the Layout Window, choose **Draw > Rectangle**. Position the cursor at 0,0 (coordinates appear at the bottom of the Layout window) and click. Move the cursor to the position 100, 25 and click. This creates a microstrip with the dimensions L = 100 mils, W = 25 mils.

3. Under the Options menu, enable midpoint snap and vertex snap by clicking each name. They are located near the bottom of the menu, and a check next to the mode indicates that it is enabled. Using these modes will assist with placing ports in the correct positions on the microstrip.

4. Click the port button and insert a port at each end of the microstrip. Rotate the port before applying if necessary. End the command.

   In this example, it will not be necessary to use the Momentum Port Editor since the default Momentum port type, internal, will be used and this port type is automatically applied to any port that is not assigned a type. Your layout should resemble the figure here.



5. From the Layout window, chose **Momentum > Substrate > Create/Modify**. Select the Alumina layer. Set the Thickness to **25** mils and Er to **9.8**.

6. Click **Metallization layers** and verify that the cond layer is positioned between the Alumina and Free_Space layers. Click **OK**.

7. From the Layout window, choose **File > Save Design As** and save the design as **mstrip**. This will also save the substrate definition as *mstrip_lay.slm* in the *networks* folder of the project.

This layout that you have just created will be referred to as the *nominal design*. It will serve as the baseline or reference for the optimization process.

# Step 2: Defining the Optimized Parameters

As mentioned earlier, the goals that are specified for this microstrip
(phase S21 = -120º at f = 10 GHz) will be achieved by changing the length of the microstrip. For this exercise, there is only the one geometrical parameter, length, that will be optimized.

The optimization process requires that for each parameter you choose to optimize, you must create a new design and alter one parameter in the design. In this exercise, only one parameter will be optimized, so only one new design is required. If you chose to optimize the width as well, you would need two new designs, one with a change to the length, and the other with a change to the width. By comparing a new project with the initial project, Momentum Optimization will identify the parameter that you want to optimize and proceed to optimize it.

The original project that you drew is referred to as the *nominal design*, the new design containing the change to the length is referred to as a *perturbed design*.

This section describes how to identify the parameter that you want to optimize, including creating a new design.

1. From the Layout window, choose **Momentum > Optimization > Parameters**. The Optimization Parameters dialog box appears.

2. To define the length as an optimizable parameter, enter the following information into the Parameter Definition fields:

| Field | Enter | Explanation |
|---|---|---|
| Name | length | The name that you give to an optimized parameter. Use a descriptive name. |
| Nominal Value | 100 mil | This is the value of the parameter in the nominal project. The microstrip that you drew has a length of 100 mils. |
| Perturbed value | 110 | When you draw the perturbed project, you will draw it so that the length of the microstrip is 110 mil. |
| Starting value | | Leave this blank. |

3. Click **Add**. The parameter definition is added to the list, as shown in the figure here.
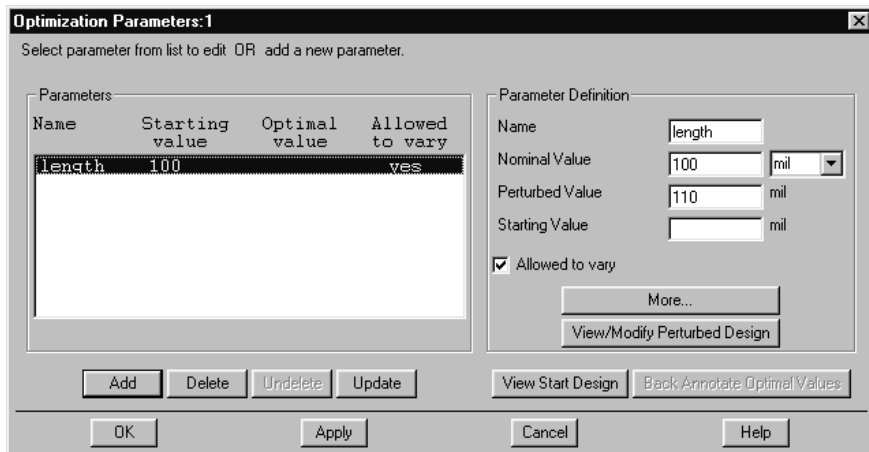
4. When you click Add, a new Layout window appears. This is the perturbed design, and it contains a *copy* of the nominal design. The perturbed design is automatically named *mstrip_length*. An information dialog is also displayed, and it explains how you can modify the perturbed design to change the parameter length. Further detail is presented in the next steps of this exercise. Drag the dialog box off of the Layout window. Until you become familiar with this process, you may want to keep this dialog box open. Otherwise, click **OK** to dismiss the box.
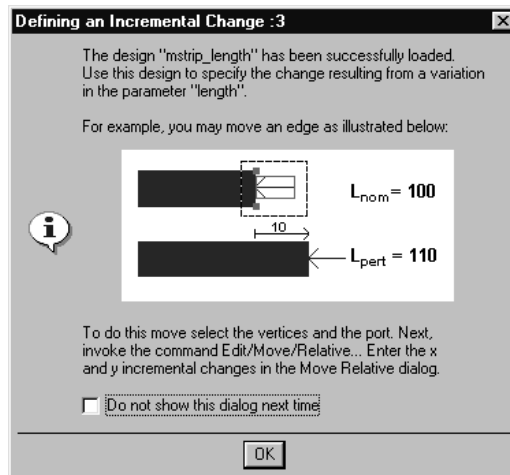
5. To edit the layout, select the right edge of the rectangle and port P2. One way to do this is to click and drag an area that contains a small portion of the right side of the rectangle and the entire port. The selected area should resemble the figure here.



6. Choose **Edit > Move > Move Relative**. A dialog box for changing x, y, coordinates appears. To alter the length of the microstrip, type **10** in the X field and click **Apply**. Click **Cancel** to dismiss the dialog box.

7. Locate the Optimization Parameters dialog box. In the Parameter Definition area of the dialog, click **More** to examine additional settings. One setting is interpolation resolution, it is set to 4 by default. This sets the size of the interpolation interval. In this case, since the incremental change in length is 10 mils, the interpolation interval is 10 mils /4 = 2.5 mils. Suppose, for example, that in some steps during optimization the parameter length varies within the interval between 110 and 112.5. Momentum Optimization will invoke EM simulation for the two end points: L = 110 and L = 112.5, and then use interpolation to obtain S parameters for all other values of L between 110 and 112.5.

8. Click **OK**. The dialog box closes, the Layout window containing the perturbed design is also closed, and the perturbed design is automatically saved in the *networks* folder of the project. The nominal project remains open.

## Step 3: Defining Optimization Goals

This section describes how to set up the optimization goals.

1. From the Layout window, choose **Momentum > Optimization > Goals**. The Optimization Goals dialog box appears.

2. Recall that the goal is Phase S21 = -120º at f = 10 GHz. The goal is at a single frequency, so in the From and To fields, enter **10** and set the units to GHz. Skip the Step Size field, but enter **10** in the Number of Points field. In the Response list, select **Phase S (degree)**. In the Ports list, select **21**. Select **=** and then in the Goal field, enter **-120**. In the Weight field, enter **1**.

3. Click **Add**. The dialog box should resemble the figure here.
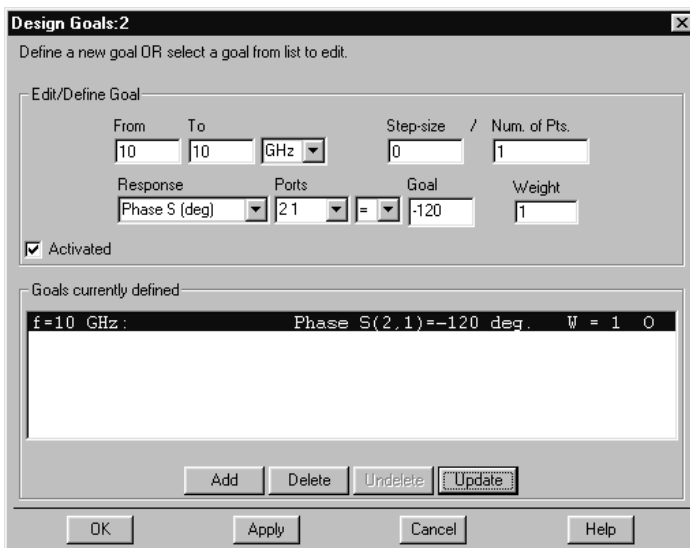


4. Click **OK** to accept the goal and close the dialog box.

## Step 4: Running the Optimization

You are ready to run the optimization. This section describes how to run an optimization.

1. From the Layout window, chose **Momentum > Optimization > Run**. The Optimization Control Editor appears. It is not necessary to alter any settings. Click **Start**. The optimization process begins:

   • The optimization is started and a Status window opens.

   • The nominal project is compared to the perturbed project. The relevant differences are extracted to establish the dependency of the design layout on the parameter *length*. This information enables Momentum Optimization to translate any given value of L into the corresponding design.

   • When the comparison process is finished, the starting design for optimization is created in a new layout window. It is named *mstrip_opt*.

   • The Momentum Optimization window opens and the progress of the optimization is reported. On the UNIX platform, the window may appear empty until the optimization is complete.

The numerical values you actually see may be different from those shown here, due to differences in the computer hardware and/or software versions. Sometimes you may also see the objective function value increasing instead of decreasing from the previous iteration. This is not unusual. It is inherent in the optimization algorithm.

- During optimization, Momentum is invoked in the background whenever a new EM analysis is required. For example, since the nominal project had not been simulated, any necessary computations, such as computing the mesh and running the simulation, will be performed and reported in the Status window.

- You can follow the evolution of the line in the window *mstrip_opt*.

# Step 5: Viewing Optimization Results

When optimization is finished, the optimized design, *mstrip_opt*, is created and displayed in a new Layout window. The Optimization Parameters dialog box reopens and you can read the optimal value for the length under Parameters.

You can also verify graphically that the goals were achieved. In this case, you can view the phase noise goal on a rectangular plot. The graphical tool is accessed from the menu bar of the Momentum Optimization window (this is the window where the optimization progress, in iterations, was reported). The procedure for viewing goals is different, depending on your platform.

## Unix

1. On the Momentum Optimization window, from the menu bar at the bottom of the window, you will see **MomOpt >** and two choices, **Display** and **Exit**. Use the mouse to highlight **Display** and click.

2. At the bottom of the window, the **MomOpt.Display >** menu appears. Click **Rectangular**. A pop up window appears.

3. The default settings in the window are for displaying S21 phase and the goal. Click **GO** to accept the default settings.

## PC

1. From the Momentum Optimization window, choose **Display > Rectangular**.

2. The default settings are for displaying S21 phase and the goal. Click **OK** to accept the default settings.

The goal and the actual response of the microstrip line is displayed. Since the optimization goal was set at a single frequency (10 GHz), both the goal and the response are displayed as single points at 10 Ghz. It appears as if there is only one point marked on the graph, but both the goal and response are displayed, since the goal was achieved, the response (circle) overlaps with the specification (cross).

This concludes the first optimization exercise. To save your work, from the Main window choose **File > Save All Designs**, then **File > Close All Designs**, answering **Yes to All** if any message windows appear.

# Example - Optimizing a Filter

This exercise describes how to optimize a double folded-stub microstrip bandstop filter, shown in the figure below. For this exercise, three parameters will be optimized:

- The distance between a folded stub and the main line. This spacing will be referred to as *S.*

- The length of the spacing between the main line and folded segment, referred to as *L1.*

- The length of each folded segment, referred to as *L2.*

Note that each when each of these physical dimensions are altered, it will be done so that the symmetry of the layout is retained.
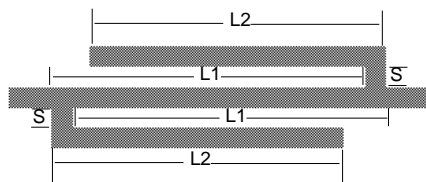


The optimization goal is to achieve the stopband in the frequency range of 12-14 GHz. The goal can be defined as:

$|S21| < $ -30 dB   for 12 < f < 14 GHz

$|S21| > $ -3 dB   for f < 9.5 GHz or f > 16.5 GHz

# Step 1: Creating the Initial Layout

This section describes how to create the project and draw the initial layout. Refer to the previous exercises if you need additional details on how to draw in Layout, define substrates, or apply ports.

Use the following steps to create an initial layout.

1. Create a new project and name it **dfstub**. Open a Layout window.

2. Draw the layout on the **cond** layer with the following parameter values: L1 = 86.4 mils, L2 = 81.6 mils, S = 4.8 mils. You can do this using the **Draw > Rectangle** and **Draw > Coordinate Entry** commands and the following coordinates for lower-left and upper-right corners of rectangles. You may need to zoom in, and be sure to end the commands when needed.
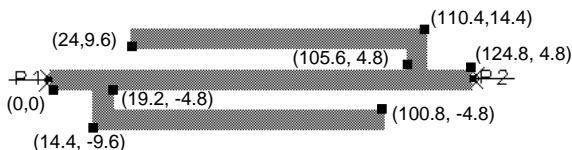


3. Attach a port at both ends. The default Momentum port type, internal, will be used, so it is not necessary to access the Momentum Port Editor. You may want to change the port component and text size, which is available under the **Options > Preferences** menu.

4. Choose **Momentum > Substrate > Create/Modify** to define a substrate of air, alumina, and a ground plane. Define the alumina layer as 5 mils thick with a relative dielectric constant of 9.9. Be sure to save the substrate.

5. Save the design as **dfstub**.

## Step 2: Defining the Optimized Parameters

There are three physical dimensions that are candidates for optimization:

- The spacing between each of the folded stubs and the main line, referred to as *S*.

- The length of the spacing between the main line and the folded segment, referred to as *L1*.

- The length of each folded segment, referred to as *L2*.

As part of defining the parameters, you must to create three perturbed designs, with each design containing a change to only one parameter. This section describes how to define the parameters and draw the perturbed designs.

1. From the Layout window, choose **Momentum > Optimization > Parameters**.

2. Define the first parameter by entering the listed values in these fields:

| Field | Enter |
|---|---|
| Name | L1 |
| Nominal Value | 86.4 mils |
| Perturbed Value | 76.4 mils |

   Leave the Starting Value field empty. Click **Add**.

3. The parameter is added to the list and a new Layout window appears. A copy of the original layout (nominal design) is shown and named **dfstub_L1**. An information dialog is also displayed that describes how to make a change to this layout. If you do not want to see this dialog again, check the box at the bottom of the dialog and click **OK**.

4. You will modify the physical dimension that you defined as L1 by reducing it from 86.4 mils to 76.4 mils. You will do this by shifting the position of the folded segments along the main line. Use the mouse to select the 6 vertices that define the top stub. Choose **Edit > Move > Move Relative**. In the dialog that appears, type **-5** in the X field and click **Apply**. This shifts the folded stub to the left so that the length of the spacing between the main line and the folded segment is reduced. Select the 6 vertices of the bottom stub, type **5** in the X field and click **Apply**. Keep this dialog box open.
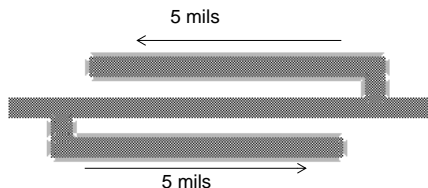


5. Define the next parameter, L2. Return to the Optimization Parameters dialog box and enter these values in the fields:

| Field | Enter |
|---|---|
| Name | L2 |
| Nominal Value | 81.6 mils |
| Perturbed Value | 91.6 mils |

6. Click **Add**. Another copy of the original layout is displayed and named **dstub_L2**.

7. You will modify the physical dimension defined as L2 by moving the edges of the folded segments by 10 mils. Select the edge of the top folded segment, and use the X field in the Move Relative dialog box to shift the edge of the segment 10 mils. Then move the edge of the bottom segment 10 mils.

8. Return to the Optimization Parameters dialog box and enter the third parameter, S. Change the perturbed layout that appears as shown here.

| Field | Enter |
|---|---|
| Name | S |
| Nominal Value | 4.8mils |
| Perturbed Value | 6.8 mils |

2 mils



2 mils
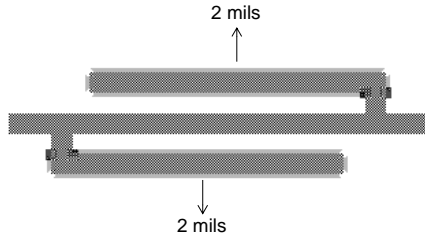
9. The definitions in the Optimization Parameters dialog box are shown here. Click **OK**. The three perturbed designs and the parameter definitions are saved automatically when you click **OK**.
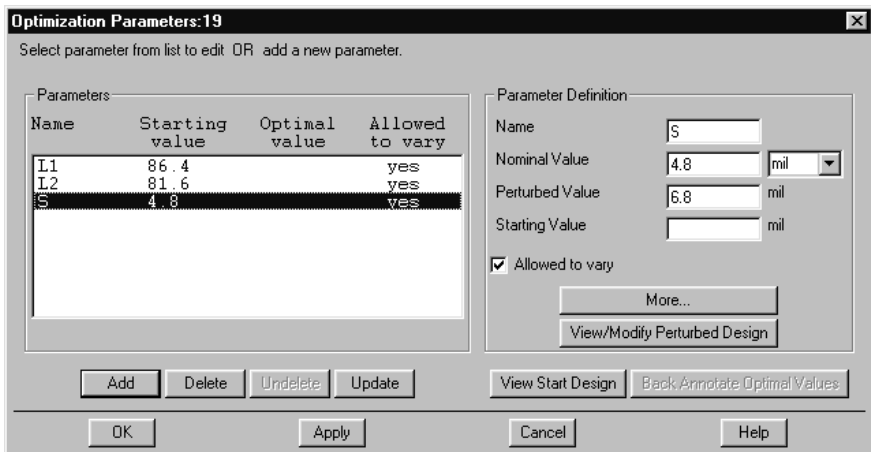
# Step 3: Defining Optimization Goals

As mentioned earlier, the goal for this design is to achieve the stopband in the frequency range of 12-14 GHz. The goal can be defined as:

$|S21| < -30$ dB  for $12 < f < 14$ GHz

$|S21| > -3$ dB  for $f < 9.5$ GHz or $f > 16.5$ GHz

This section describes how to define the goals.

1. From the Layout window, chose **Momentum > Optimization > Goals**.

2. Add the following three goals to the list, clicking **Add** after each complete goal entry:

|  | From | To | Step Size | Response | Ports | Goal | Weight |
|---|---|---|---|---|---|---|---|
| **Entries for goal 1** | 12 | 14 GHz | 0.5 | Mag S(dB) | S21 | < -30 dB | 1 |
| **Entries for goal 2** | 5 | 9.5 GHz | 2 | Mag S (dB) | S21 | > -3 dB | 1 |
| **Entries for goal 3** | 16.5 | 20 GHz | 2 | Mag S (dB) | S21 | > -3 dB | 1 |

The completed dialog box should resemble the figure here. If you make a mistake, select the goal from the list, modify the necessary fields, and click **Update**. Click **OK**.
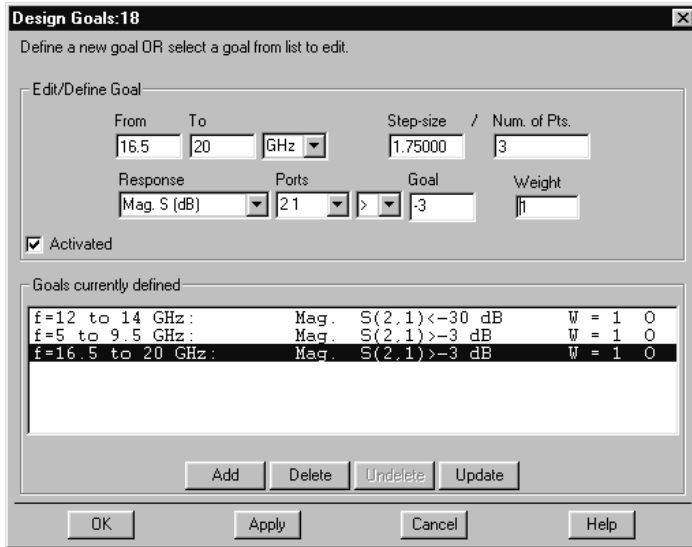
# Step 4: Running the Optimization

This section describes how to set up and run the optimization.

1. From the Layout window, choose **Momentum> Optimization > Run**. Under Optimization Type, select **Minimax**. Use the default Interpolation Type, linear interpolation. Set the Maximum Number Of Iterations to **30**. Enable **Stop as soon as specs are met** to halt the optimization when the goals are achieved. If you enable **Delete Previous Database**, any existing data will be deleted and EM simulations will be invoked as required.

2. Click **Start**. The Momentum Optimization is started:

   • A status window opens.

   • The nominal project is first compared to the perturbed projects. The relevant differences are extracted to establish the dependency of the design layout on the parameters. This information enables Momentum Optimization to translate any given set of parameter values into the appropriate design.

   • When the comparison process is finished, the starting design for optimization is created in a new layout window. Is it named *dfstub_opt*.

   • The Momentum Optimization window opens. On the screen, the progress of optimization is reported. The numerical values you actually see may be slightly different from those shown here, due to differences in the computer hardware and/or software versions.

   • If you have chosen to delete the database, Momentum is invoked in the background whenever a new EM analysis is required. Follow the evolution of the structure in the window displaying *dfstub_opt*.

## Step 5: Viewing Optimization Results

When the optimization is finished, the optimized design, *dfstub_opt*, is created. It is displayed is a new layout window. The Optimization Parameters Dialog is opened and you can read the optimal values of all three parameters in the list:
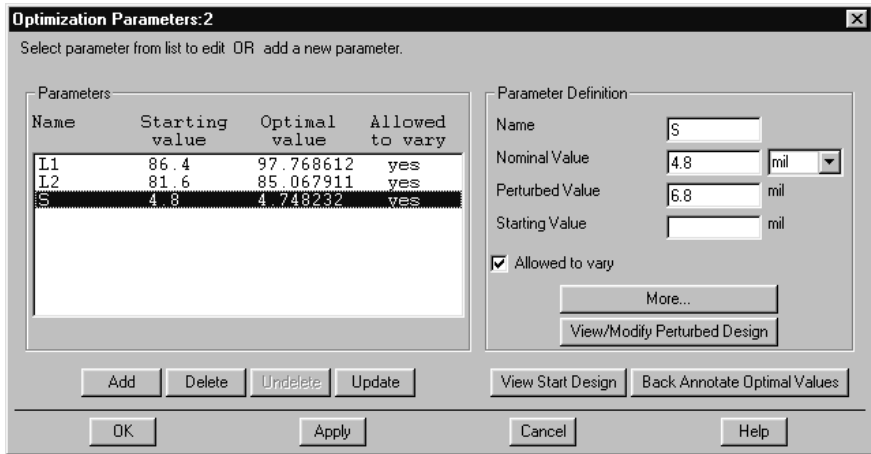


You can check that the goals are met. The procedure for viewing goals is different, depending on your platform.

### Unix

1. On the menu bar at the bottom of the Momentum Optimization window, you will see **MomOpt >** and two choices, **Display** and **Exit**. Use the mouse to highlight **Display** and click.

2. At the bottom of the window, the **MomOpt.Display >** menu appears. Click **Rectangular**. A pop up window appears.

3. The default settings in the window are for displaying S21 phase and the goal. Click **GO** to accept the default settings.

## PC

1. From the Momentum Optimization window, choose **Display > Rectangular**.

2. The default settings are for displaying S21 phase and the goal. Click **OK** to accept the default settings.

   The goals and the actual response of the filter is displayed.
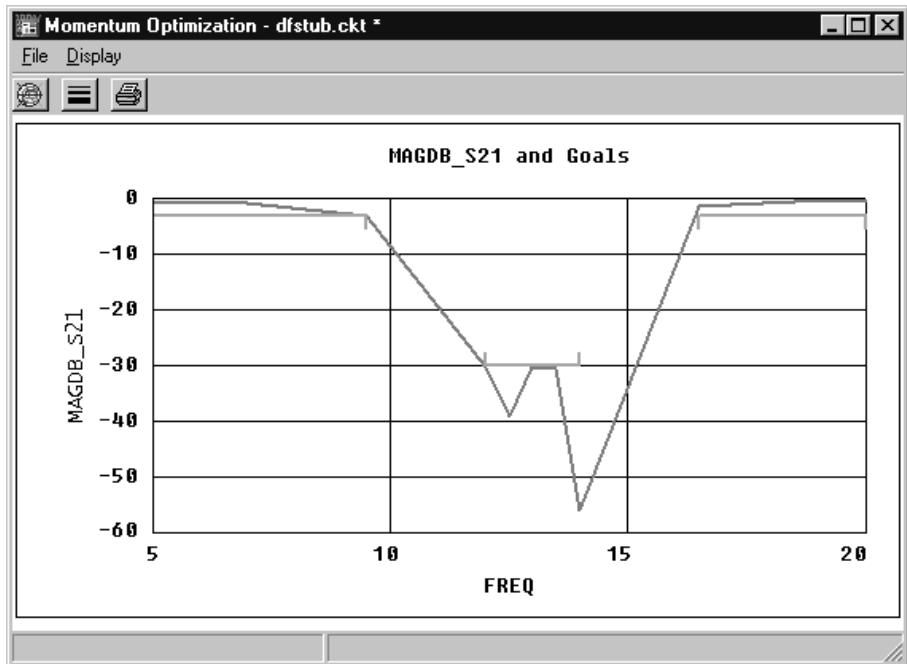
# Appendix A: Theory of Operation

Momentum is based on a numerical discretization technique called the *method of moments*. This technique is used to solve Maxwell's electromagnetic equations for planar structures embedded in a multilayered dielectric substrate. The simulation modes available in Momentum (microwave and RF) are both based on this technique, but use different technologies to achieve their results.

Momentum uses specific steps to achieve solutions. The technology used by each simulation mode varies slightly in each step. In Momentum's microwave mode, the planar metallization (strip, via) and aperture (slot) patterns in the signal layers are meshed using rectangular and triangular cells. Momentum RF mode generates a mesh of polygonal cells. The unknown electric (strip, via) and magnetic (slot) surface currents are discretized using rooftop expansion functions defined over the cells in the mesh. In Momentum RF, rooftop functions are regrouped in star and loop functions to eliminate low-frequency breakdown. This discretization process transforms the electromagnetic equations into an equivalent network model of self and mutual inductive and capacitive elements. The microwave mode uses fullwave Green's functions resulting in L and C elements that are complex and frequency dependent. The RF mode uses quasi-static functions resulting in L and C elements that are real and frequency independent.

The sources applied at the ports of the circuit yield the excitations in the equivalent network model. The currents in the equivalent network are the unknown amplitudes of the rooftop expansion functions. Solving the equivalent network for a number of independent excitation states yields the unknown current amplitudes. A port calibration process is used to calculate the S-parameter data of the circuit from the current solution.

The following sections in this chapter contain more information about:

- The method of moments technology
- The Momentum solution process
- Special simulation topics
- Considerations and software limitations

# The Method of Moments Technology

The method of moments (MoM) technique is based upon the work of R.F. Harrington, an electrical engineer who worked extensively on the method an applied it to electromagnetic field problems, in the beginning of the1960's. It is based on older theory which uses weighted residuals and variational calculus.

In the method of moments, prior to the discretization, Maxwell's electromagnetic equations are transformed into integral equations. These follow from the definition of suitable electric and magnetic Green's functions in the multilayered substrate. In Momentum, a mixed potential integral equation (MPIE) formulation is used. This formulation expresses the electric and magnetic field as a combination of a vector and a scalar potential. The unknowns are the electric and magnetic surface currents flowing in the planar circuit.

Using notations from linear algebra, we can write the mixed potential integral equation in very general form as a linear integral operator equation:

$$\iint dS \overline{\overline{G}}(r, r') \cdot J(\mathbf{r}) = E(\mathbf{r}) \tag{1}$$

Here, $\mathbf{J}(\mathbf{r})$ represents the unknown surface currents and $\mathbf{E}(\mathbf{r})$ the known excitation of the problem. The Green's dyadic of the layered medium acts as the integral kernel. The unknown surface currents are discretized by meshing the planar metallization patterns and applying an expansion in a finite number of subsectional basis functions $\mathbf{B}_1(\mathbf{r})$, ..., $\mathbf{B}_N(\mathbf{r})$:

$$J(\mathbf{r}) \approx \sum_{j=1}^{N} I_j \mathbf{B}_j(\mathbf{r}) \tag{2}$$

The standard basis functions used in planar EM simulators are the subsectional rooftop functions defined over the rectangular and triangular cells in the mesh. Each rooftop is associated with one edge of the mesh and represents a current with constant density flowing through that edge

(Figure A-1). The unknown amplitudes Ij, j=1,..,N of the basis function expansion determine the currents flowing through all edges of the mesh.
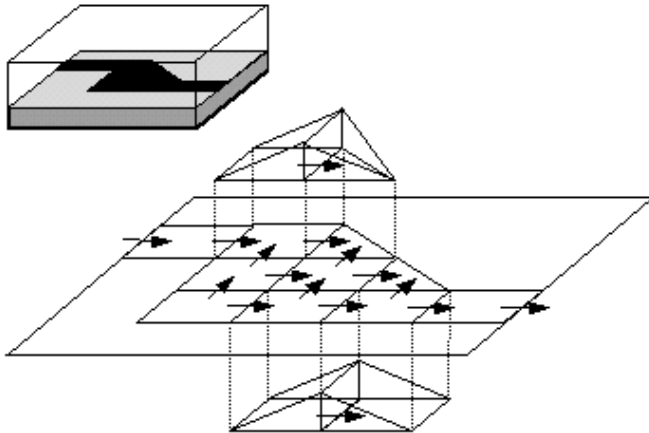


Figure A-1. Discretization of the surface
currents using rooftop basis functions.

The integral equation (1) is discretized by inserting the rooftop expansion (2) of the currents. By applying the Galerkin testing procedure, that is, by testing the integral equation using test functions identical to the basis functions, the continuous integral equation (1) is transformed into a discrete matrix equation:

**for i=1,...,N**

$$\sum_{j=1}^{N} Z_{i,j} I_j = V_i \quad \text{or} \quad [Z] \cdot [I] = [V] \tag{3}$$

**with**

$$Z_{i,j} = \iint_S \mathrm{d}S \boldsymbol{B}_i(\boldsymbol{r}) \cdot \iint_S \mathrm{d}S \overline{\overline{\boldsymbol{G}}}(\boldsymbol{r}, \boldsymbol{r'}) \cdot \boldsymbol{B}_j(\boldsymbol{r}) \tag{4}$$

$$V_i = \iint_S \mathrm{d}S \boldsymbol{B}_i(\boldsymbol{r}) \cdot \boldsymbol{E}(\boldsymbol{r}) \tag{5}$$

**The left hand side matrix [Z] is called the interaction matrix, as each element in this matrix describes the electromagnetic interaction between two rooftop basis functions. The dimension N of [Z] is equal to the number of basis functions. The right-hand side vector [V] represents the discretized contribution of the excitations applied at the ports of the circuit.**

The surface currents contribute to the electromagnetic field in the circuit by means of the Green's dyadic of the layer stack. In the MPIE formulation, this Green's dyadic is decomposed into a contribution from the vector potential **A**(r) and a contribution from the scalar potential V(**r**):

$$\overline{\overline{G}}(\boldsymbol{r}, \boldsymbol{r}') \; = \; j\omega G^A(\boldsymbol{r}, \boldsymbol{r}')\overline{\overline{I}} - \frac{1}{j\omega}\nabla[\,G^V(\boldsymbol{r}, \boldsymbol{r}')\nabla']\tag{6}$$

The scalar potential originates from the dynamic surface charge distribution derived from the surface currents and is related to the vector potential through the Lorentz gauge.

By substituting the expression (**6**) for the Green's dyadic in the expression (**4**) for the interaction matrix elements, yields the following form:

$$Z_{i,\, j} \; = \; j\omega L_{i,\, j} + \frac{1}{j\omega C_{i,\, j}}\tag{7}$$

with

$$L_{i,\, j} \; = \; \iint_S \mathrm{d}S\, \boldsymbol{B}_i(\boldsymbol{r}) \cdot \iint_S \mathrm{d}S\, G^A(\boldsymbol{r}, \boldsymbol{r}')\boldsymbol{B}_j(\boldsymbol{r}')\tag{8}$$

$$\frac{1}{C_{i,\, j}} \; = \; \iint_S \mathrm{d}S\nabla \cdot \boldsymbol{B}_i(\boldsymbol{r}) \iint_S \mathrm{d}S\, G^V(\boldsymbol{r}, \boldsymbol{r}')\nabla \cdot \boldsymbol{B}_j(\boldsymbol{r}')\tag{9}$$

This allows the interaction matrix equation to be given a physical interpretation by constructing an equivalent network model (**Figure A-2**). In this network, the nodes correspond to the cells in the mesh and hold the cell charges. Each cell corresponds to a capacitor to the ground. All nodes are connected with branches which carry the current flowing through the edges of the cells. Each branch has in inductor representing the magnetic self coupling of the associated current basis function. All capacitors and inductors in the network are complex, frequency dependent and mutually coupled, as all basis functions interact electrically and magnetically (**Figure A-3**). The ground in this equivalent network corresponds with the potential at the infinite metallization layers taken up in the layer stack. In the absence of infinite metallization layers, the ground corresponds with the sphere at infinity. The method of moments interaction matrix equation follows from

applying the Kirchoff voltage laws in the equivalent network. The currents in the network follow from the solution of the matrix equation and represent the amplitudes of the basis functions.
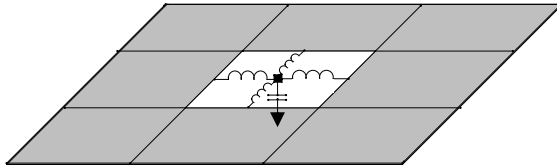


**Figure A-2. The equivalent circuit is built by replacing each cell in the mesh with a capacitor to the ground reference and inductors to the neighboring cells.**
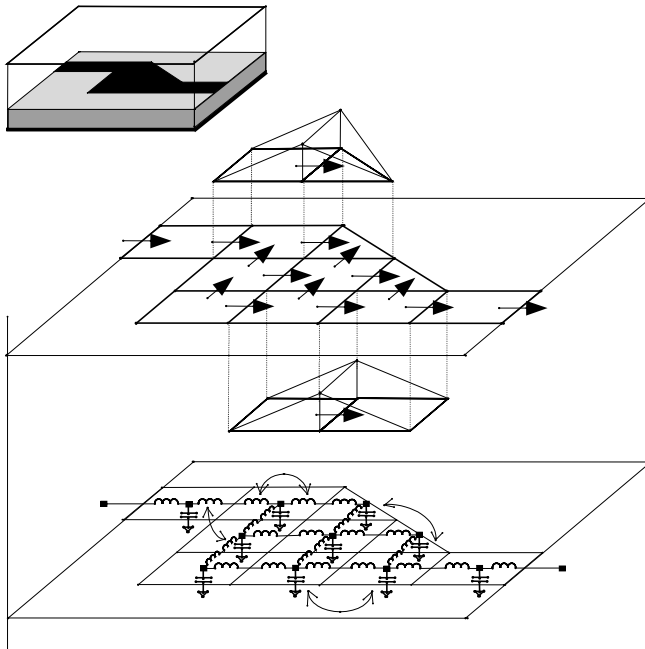


**Figure A-3. Equivalent network representation of the discretized MoM problem.**

# The Momentum Solution Process

Different steps and technologies enable the Momentum solution process:

- Calculation of the substrate Green's functions
- Meshing of the planar signal layer patterns
- Loading and solving of the MoM interaction matrix equation
- Calibration and de-embedding of the S-parameters
- Reduced Order Modeling by Adaptive Frequency Sampling

## Calculation of the Substrate Green's Functions

The substrate Green's functions are the spatial impulse responses of the substrate to Dirac type excitations. They are calculated for each pair of signal (strip, slot and/or via) layers mapped to a substrate level. Although it is necessary to know which signal layers are mapped to a substrate level, since only impulse responses are being calculated, it is not necessary to know the patterns on these signal layers. This implies that the Green's functions can be pre-calculated and stored in a substrate database. This allows the substrate Green's functions to be reused for other circuits defined on the same substrate.

The high frequency electromagnetic Green's functions depend upon the radial distance and the frequency. The computations are performed up to very large radial distances over the entire frequency band specified by the user. The frequency points are selected adaptively to ensure an accurate interpolation with respect to frequency. Computations performed over very wide frequency ranges can consume more CPU time and disk space to store the results. To increase speed, the RF mode uses quasi-static electromagnetic functions based on low-frequency approximation.

## Meshing of the Planar Signal Layer Patterns

The planar metallization (strip, via) and aperture (slot) patterns defined on the signal layers are meshed with rectangular and triangular cells in the microwave simulation mode. As translational invariance can be used to speed up the interaction matrix load process, the meshing algorithm will maximize the number if uniform rectangular cells created in the mesh. The meshing algorithm is very flexible as different parameters can be set by the user (number of cells/wavelength, number of cells/width, edge meshing, and mesh seeding), resulting in a mesh with different

density. It is clear that the mesh density has a high impact on both the efficiency and accuracy of the simulation results. Default mesh parameters are provided which give the best accuracy/efficiency trade-off. The RF mode uses mesh reduction technology to combine rectangular and triangular cells to produce a mesh of polygonal cells, thus reducing demand for computer resources.

## Loading and Solving of the MoM Interaction Matrix Equation

The loading step of the solution process consists of the computation of all the electromagnetic interactions between the basis functions and the filling of the interaction matrix and the excitation vector. The interaction matrix as defined in the rooftop basis is a dense matrix, that is, each rooftop function interacts with every other rooftop function. This electromagnetic interaction between two basis functions can either be strong or weak, depending on their relative position and their length scale. The matrix filling process is essentially a process of order $(N^2)$, i.e. the computation time goes up with the square of the number of unknowns.

In the solving step, the interaction matrix equation is solved for the unknown current expansion coefficients. The solution yields the amplitudes of the rooftop basis functions which span the surface current in the planar circuit. Once the currents are known, the field problem is solved because all physical quantities can be expressed in terms of the currents. In Momentum, the matrix equation is solved with a standard matrix factorization technique. This implies that the matrix solve process is essentially a process of order $(N^3)$.

## Calibration and De-embedding of the S-parameters

Momentum performs a calibration process on the single type port, the same as any accurate measurement system, to eliminate the effect of the sources connected to the transmission line ports in the S-parameter results. Feedlines of finite length are added to the transmission line ports of the circuit. Lumped sources are connected to the far end of the feedlines. These sources excite the eigenmodi of the transmission lines without interfering with the circuit. The effect of the feedlines is computed by the simulation of a calibration standard and subsequently removed from the S-parameter data. A built-in cross section solver calculates the characteristic impedance and propagation constant of the transmission lines. This allows to shift the phase reference planes of the S-parameters, a process called de-embedding. Results of the calibration process includes the elimination of low-order mode mismatches at the port boundary, elimination of high-order modes, and removal of all port excitation parasitics.

Besides transmission line ports, Momentum offers the user the ability to define direct excitation or internal ports. These ports can be specified at any location on the planar metallization patterns as either a point or a line feed. They allow to connect both passive and active lumped components to the distributed model of the planar circuits. The S-parameters associated with these ports are calculated from the excitation consisting of a lumped source connected to the equivalent network model at the locations of the internal ports. The parasitic effects of these lumped sources are *not* calibrated out of the S-parameters results.

## Reduced Order Modeling by Adaptive Frequency Sampling

A key element to providing fast, highly accurate solutions using a minimum of computer resources is the Adaptive Frequency Sampling (AFS) technology. When simulating over a large frequency range, oversampling and straight line interpolation can be used to obtain smooth curves for the S-parameters. Oversampling however implies a huge amount of wasted resources. Momentum allows the user to benefit from a smart interpolation scheme based on reduced order modeling techniques to generate a rational pole/zero model for the S-parameter data. The Adaptive Frequency Sampling algorithm selects the frequency samples automatically and interpolates the data using the adaptively constructed rational pole/zero model. This feature allows important details to be modeled by sampling the response of the structure more densely where the S-parameters are changing significantly. It minimizes the total number of samples needed and maximizes the information provided by each new sample. Typical types of structures where the AFS module is very helpful include:

- Unmatched transmission line structures

- Resonant structures, such as filters

- Structures where unexpected coupling occurs

In fact, nearly any kind of structure can take advantage of the AFS module. The Adaptive Frequency Sampling technology reduces the computation time needed for simulating large frequency ranges with Momentum significantly.

# Special Simulation Topics

Some special simulation topics are discussed in this section:

- Simulating slots in ground planes
- Simulating metallization loss
- Simulating internal ports and ground references

## Simulating Slots in Ground Planes

Slots in ground planes are treated in a special manner by Momentum. An electromagnetic theorem called the equivalence principle is applied. Instead of attempting to simulate the flow of electric current in the wide extent of the ground plane, only the electric field in the slot is considered. This electric field is modeled as an equivalent magnetic current that flows in the slot.

Momentum does not model finite ground plane metallization thickness. Ground planes and their losses are part of the substrate definition.

By using slot metallization definitions, entire structures, such as slot lines and coplanar waveguide circuits, can be built. Slots in ground planes can also be used to simulate aperture coupling through ground planes for multi-level circuits. Structure components on opposite sides of a ground plane are isolated from each other, except for intermediate coupling that occurs through the slots. This treatment of slots allows Momentum to simulate slot-based circuits and aperture coupling very efficiently.

## Simulating Metallization Loss

When using Momentum, losses in the metallization patterns can be included in the simulation. Momentum treats all conductors as having zero thickness. However, the conductivity and thickness can be specified to approximate frequency dependent losses in the metallization patterns.

Momentum uses a complex surface impedance for all metals that is a function of conductor thickness, conductivity, and frequency. At low frequencies, current flow will be approximately uniformly distributed across the thickness of the metal. Momentum uses this minimum resistance and an appropriate internal inductance to form the complex surface impedance. At high frequencies, the current flow is dominantly on the outside of the conductor and Momentum uses a complex surface impedance that closely approximates this skin effect. At intermediate frequencies, where metal

thickness is between approximately two and ten skin depths, the surface impedance transitions between those two limiting behaviors. This treatment of metal loss is a good approximation for many cases but can be less accurate for very narrow conductors.

The meshing density can affect the simulated behavior of a structure. A more dense mesh allows current flow to be better represented and can slightly increase the loss. This is because a more uniform distribution of current for a low density mesh corresponds to a lower resistance.

# Simulating with Internal Ports and Ground References

Momentum offers the ability to use internal ports within a structure. Internal ports can be specified at any location on the planar metallization patterns, and they make possible a connection for both passive and active lumped components to the distributed model of the planar circuits. Refer to Figure A-4.

The S-parameters associated with these ports are calculated from the excitation consisting of a lumped voltage source connected to the equivalent network model, as shown in Figure A-3. The ground reference for these ports in the resulting S-parameter model is the ground of the equivalent network, and this ground corresponds *physically* to the infinite metallization layers taken up in the layer stack. In the absence of infinite metallization layers, the ground no longer has a physical meaning and corresponds *mathematically* with the sphere at infinity.

It is important to mention that in this case, the associated S-parameters also lose their physical meaning, as the applied voltage source is assumed to be lumped, that is electrically small, since it sustains a current flow from the ground to the circuit without phase delay. To overcome this problem, a ground reference must be specified

**at a distant electrically small from the internal port. Failure to do so may yield erroneous simulation results.**



direct excitation point feed
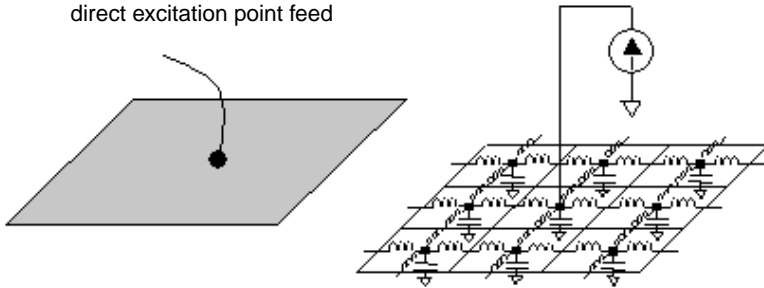
Figure A-4. Internal port and equivalent network model.

**The following sections illustrate the use of internal ports with ground planes and with ground references, and the results.**

## Internal Ports and Ground Planes in a PCB Structure

Figure A-5 shows the layout for a PCB island structure with two internal ports. The infinite ground plane is taken up in the substrate layer stack and provides the ground reference for the internal ports. The magnitude and phase of the S21-parameter calculated with Momentum are

shown in Figure A-6. The simulation results are validated by comparing them with the measured data for the magnitude and the phase of S21.

substrate layer stack

metallization layer

L = 3 inches

air: $\varepsilon_r = 1.0$

W = 2 inches

FR4: H = 59 mil, $\varepsilon_r = 4.4$

infinite ground plane taken
up in the layer stack

two internal ports
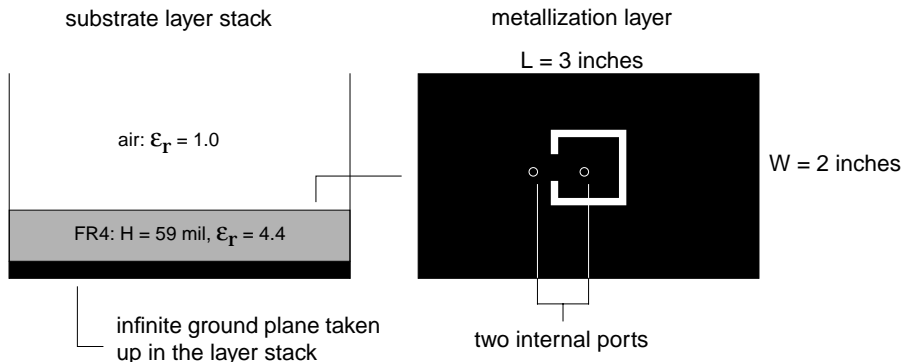
Figure A-5. PCB Substrate layer stack and metallization layer.
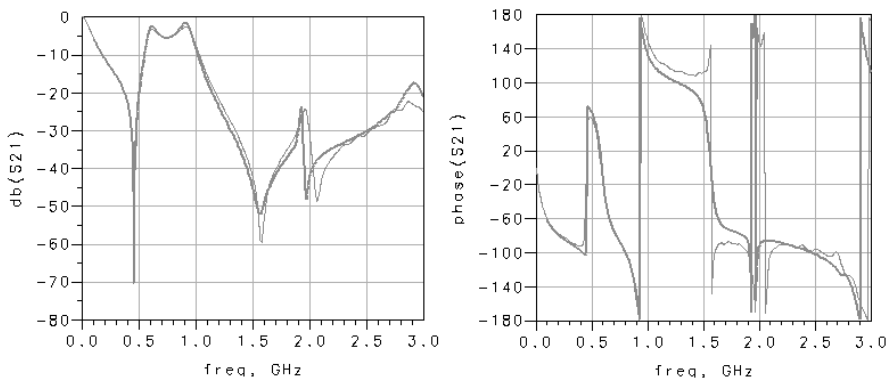
Figure A-6. Magnitude and phase of $S_{21}$.
The thicker line is Momentum results, the thinner line is the measurement.

## Finite Ground Plane, no Ground Ports

The same structure was resimulated with a finite groundplane (Figure A-7). The substrate layer stack contains no infinite metallization layers. No ground reference was specified for the ports, so

the sphere at infinity acts as the ground reference for the internal ports. The S-parameters obtained from a simulation for these ports no longer have a physical meaning. Using such ports in the simulation yields incorrect simulation results as shown in **Figure A-8**.
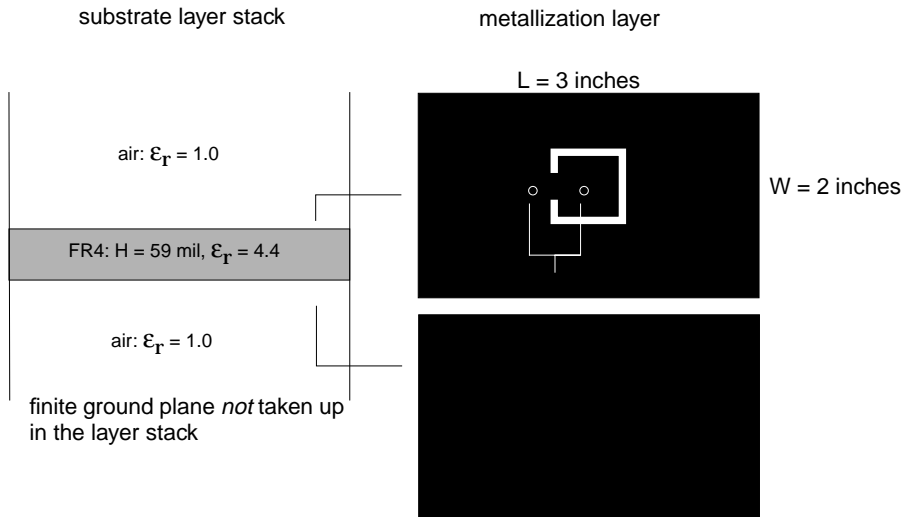


**Figure A-7. PCB substrate layer stack and metallization layers.**
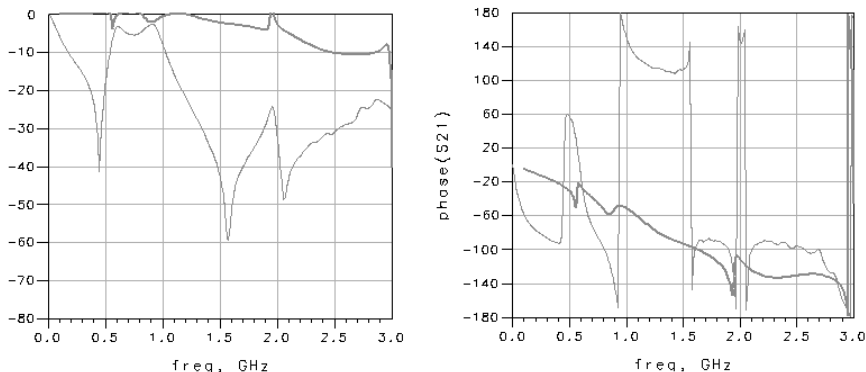


**Figure A-8. Magnitude and phase of S21.**
**The thicker line is Momentum results, the thinner line is the measurement.**

## Finite Ground Plane, Internal Ports in the Ground Plane

One way to define the proper grounding for the internal ports is to add two extra internal ports in the ground plane (Figure A-9). The resulting four-port structure is simulated with Momentum. Note that because no ground reference is specified for the internal ports, the resulting S-parameters for the four port structure have no physical meaning. However, by properly recombining the four ports, a two-port structure is obtained with the correct ground references for each of the ports. Care should be taken with this recombination, as the ground reference for the four individual ports does not act as the ground reference for the two recombined ports. Therefore, the top recombination scheme shown in Figure A-10 is incorrect. The correct recombination scheme is the bottom one shown in Figure A-10.
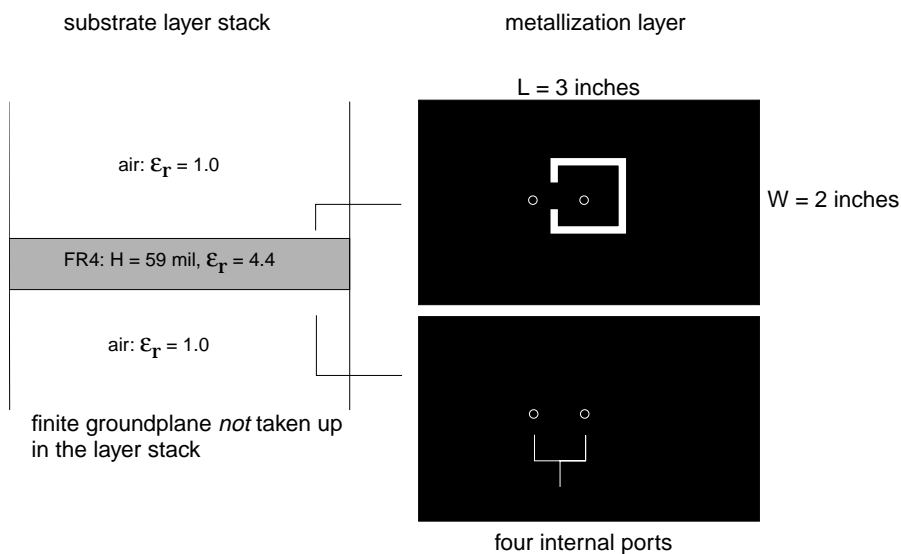


**Figure A-9. PCB substrate layer stack and metallization layers.**
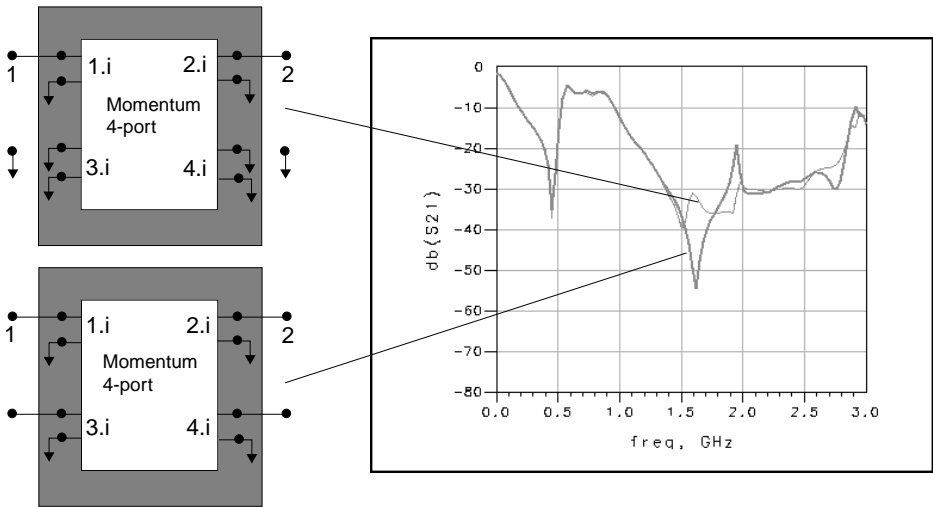
**Figure A-10. Simulation results.**
**The bottom illustration shows the correct recombination of ports.**

## Finite Ground Plane, Ground References

The process of adding two extra internal ports in the ground plane and recombining the ports in the correct way can be automated in Momentum by defining two ground reference ports

(Figure A-11). The S-parameters obtained with Momentum are identical to the S-parameters of the two-port after the correct recombination in the section above, also shown in Figure A-12.

substrate layer stack                    metallization layer

L = 3 inches

air: $\varepsilon_r$ = 1.0

W = 2 inches

FR4: H = 59 mil, $\varepsilon_r$ = 4.4

air: $\varepsilon_r$ = 1.0

finite groundplane *not* taken up
in the layer stack

four internal ports with associated
ground reference ports
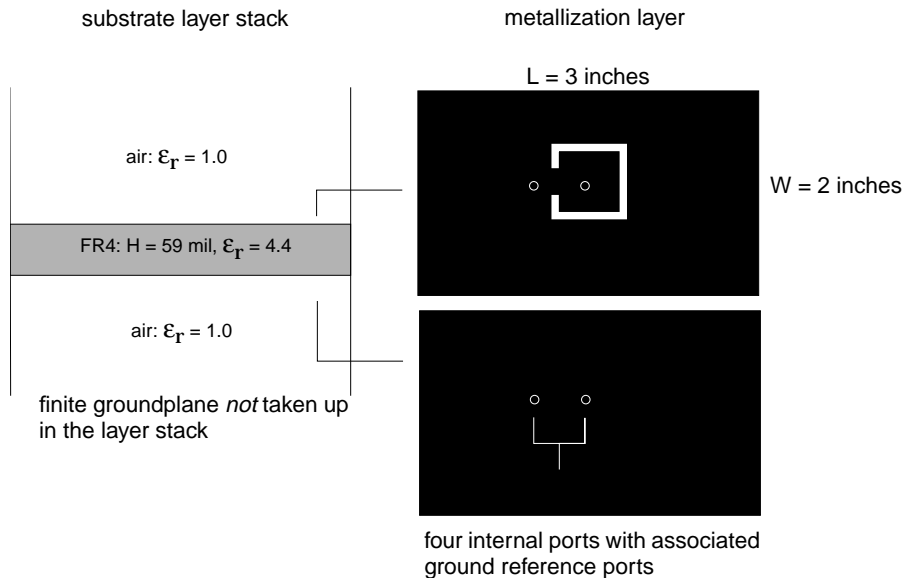
**Figure A-11. Substrate layer stack and metallization layers.**
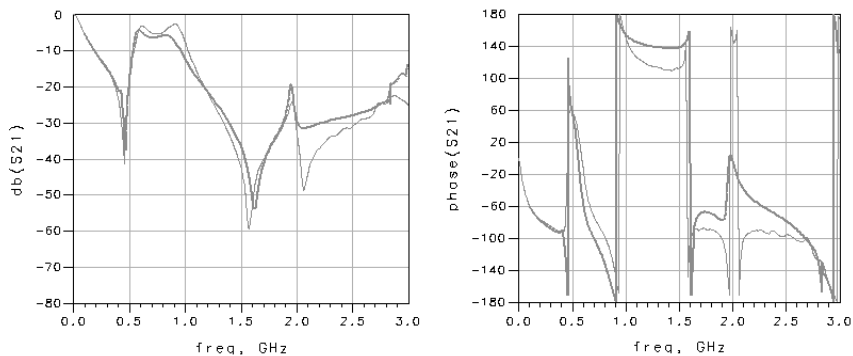


**Figure A-12. Magnitude and phase of S21.**
**The thicker line is Momentum results, the thinner line is the measurement.**

## Internal Ports with a CPW Structure

Figure A-13 and Figure A-14 show the layout for a CPW step in width structure with an integrated sheet resistor (45 µm x 50 µm, 50Ω). Six internal ports are added to the metallization layers. The six-port S-parameter model is recombined into a two-port model to describe the CPW-mode S-parameters. Different recombination schemes are possible, yielding different simulation results. Scheme (c) is the correct one, as the ground reference for the internal ports is the sphere at infinity and this cannot act as the ground reference for the CPW ports.
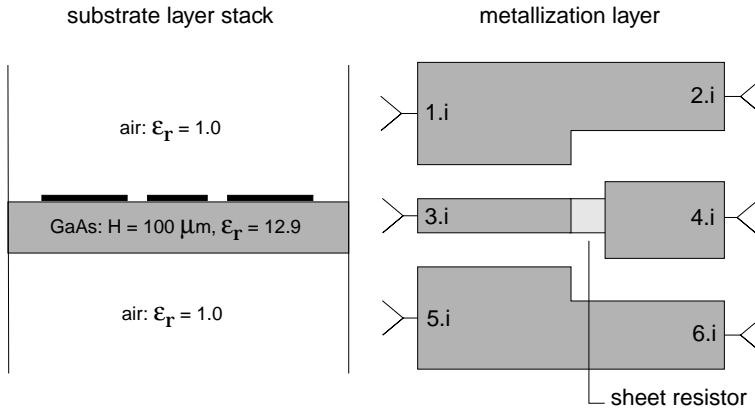


**Figure A-13. CPW step-in-width with integrated sheet resistor**

.
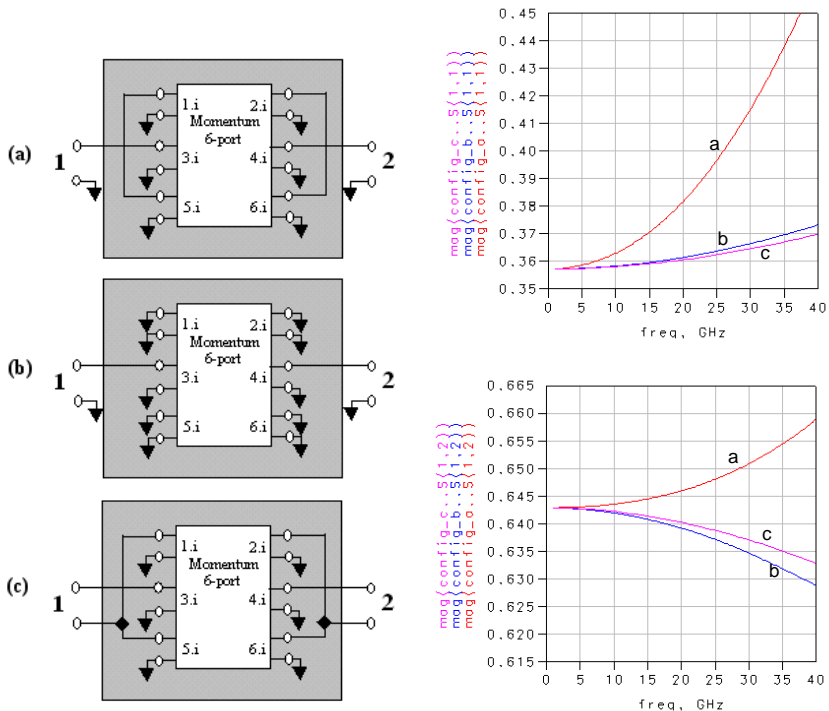


**Figure A-14. CPW Step-in-width structure with integrated
sheet resistor, port configurations and results.**

The process of adding extra internal ports to the ground metallization patterns and recombining the
ports in the correct way can be automated in Momentum by defining ground reference ports

([Figure A-15](#)). The S-parameters obtained with Momentum are identical to the S-parameters of the two-port for the correct recombination scheme ([Figure A-14](#) (c)).
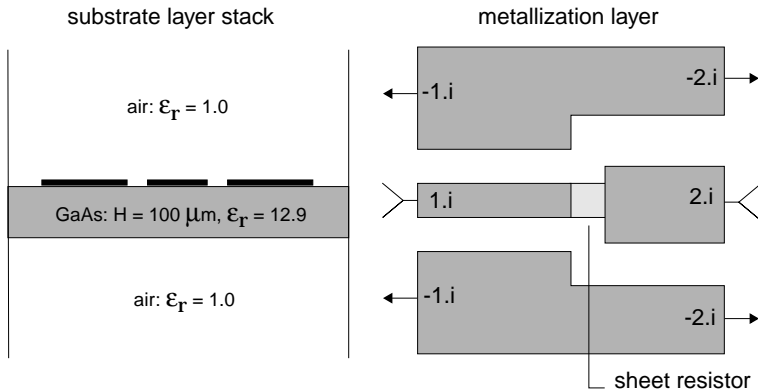


Figure A-15. CPW step-in-width structure with ground
references for the internal ports.

# Limitations and Considerations

This section describes some software limitations and physical considerations which need to be taken into account when using Momentum:

- Comparing the microwave and RF simulation modes
- Matching the simulation mode to circuit characteristics
- Higher-order modes and high frequency limitation
- Interpolating Results below the low frequency limitation
- Substrate waves and substrate thickness limitation
- Parallel-plate modes and high frequency limitation
- Slotline structures and high frequency limitation
- Via structures and metallization thickness limitation
- Via structures and substrate thickness limitation
- CPU time and memory requirements

# Comparing the Microwave and RF Simulation Modes

The Momentum (microwave) and Momentum RF modes use different Method of Moments technologies to produce S-parameter models for layout-based, physical designs. Momentum uses fullwave electromagnetic functions based on Maxwell's equations which includes substrate and space wave radiation effects. Momentum RF uses quasi-static electromagnetic functions based on low-frequency approximation which excludes substrate and space wave radiation effects. Though Momentum RF excludes radiation effects from its resulting models, both modes produce models that include these physical effects:

- Quasi-static inductance
- Quasi-static capacitance
- DC conductor loss
- DC substrate loss
- Dielectric loss
- Skin effect loss

Both modes generate a mesh in preparation for simulating the circuit. The Momentum mode uses all of the resulting rectangular and triangular cells in the mesh to arrive at a set of S-parameters for a circuit. Calculating currents in each cell can require significant computer memory and computation time. However, Momentum RF uses a mesh-reduction technology to combine rectangular and triangular cells to produce a mesh of polygonal cells. The reduction eliminates low-quality slivery cells and electromagnetically redundant interactions. The resulting mesh contains far fewer cells which requires much less computer memory and computation time for a highly accurate set of S-parameters at RF frequencies.

Another advantage of Momentum RF over the Momentum mode is its use of star-loop technology to eliminate low-frequency breakdown of numerical solutions, and give stable solutions down to DC. Momentum mode simulations use rooftop basis functions defined over the rectangular and triangular cells in a mesh. While a formulation based on rooftop functions produces accurate results at microwave frequencies, it becomes unstable at lower, RF frequencies and a break-down of the solution occurs. To eliminate the instability, Momentum RF uses a formulation based on star-loop technology, which regroups the rooftop functions into star and loop functions. The loop functions model the solenoidal part of the current solution, while the star functions model the non-rotational part. This allows the elimination of the low-frequency instability in the numerical solution.

# Matching the Simulation Mode to Circuit Characteristics

The Momentum RF mode can be used to simulate RF and microwave circuits, depending on your requirements. However, Momentum RF is usually the more efficient mode when a circuit is electrically small, geometrically complex, and does not radiate. This section describes these characteristics.

## Radiation

**Momentum RF provides accurate electromagnetic simulation performance at RF frequencies. This is typically up to about 10 GHz. However, this upper limit depends on the size of your physical design. At higher frequencies, as radiation effects increase, the accuracy of the Momentum RF models declines smoothly with increased frequency.**

## Electrically Small Circuits

Momentum RF works best for electrically small circuits as its accuracy smoothly decreases with increasing electrical size relative to a given frequency. A circuit is considered electrically small relative to a given frequency if its physical dimension is smaller than half the wavelength of the frequency. Depending on which value you know, maximum circuit dimension or maximum simulation frequency, you can determine a qualitative approximation of the circuit's electrical size. Suppose you have a circuit with dimension $D$ as shown in the following figure:
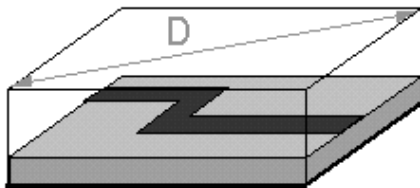


You can use one of the following two expressions strictly as a guideline to have an awareness about the circuit's electrical size relative to the maximum frequency you plan to run the simulation. When you know the value of $D$, use the first expression to approximate the maximum frequency up to which the circuit is electrically small. When you know the maximum simulation frequency, use the second expression to approximate the maximum allowable dimension:

$$F < \frac{150}{D}$$

or

$$D < \frac{150}{F}$$

where:

$D$ = the maximum length in *mm* diagonally across the circuit.

$F$ = the maximum frequency in *GHz*.

During a simulation, Momentum RF calculates the maximum frequency up to which the circuit is considered electrically small, and displays that value in the status display. This is similar to using the first expression above since the dimension of a layout is typically fixed, and it is the simulation frequency that is swept.

## Geometrically Complex Circuits

The mesh generated for a simulation establishes a geometric complexity for a circuit. A circuit is considered geometrically complex if its shape does not fit into a uniform, rectangular mesh, and the mesh generation produces a lot of triangles. Layouts containing shapes such as circles, arcs, and non-rectangular polygons usually result in meshes with many triangles. A measure of increasing geometric complexity is when the ratio of triangular cells to rectangular cells grows larger:

$$\text{Complexity} = \frac{\text{Triangles}}{\text{Rectangles}}$$

The increased simulation speed offered by Momentum RF over Momentum is the direct result of its mesh-reduction technology. While Momentum uses the entire mesh to complete a simulation, Momentum RF eliminates the electromagnetically redundant rectangular and triangular cells. This reduces the time required to complete the simulation. As a circuit's geometric complexity increases, Momentum RF's efficiency increases over Momentum.

# Higher-order Modes and High Frequency Limitation

Since Momentum does not account for higher-order modes in the calibration and de-embedding process, the highest frequency for which the calibrated and de-embedded S-parameters are valid is determined by the cutoff frequency of the port transmission line higher-order modes. As a rule of thumb for microstrip transmission lines, the cutoff frequency (in GHz) for the first higher-order mode is approximately calculated by:

Cutoff frequency $f_c = 0.4\ Z_0$ / height

where $Z_0$ is the characteristic impedance of the transmission line. For a 10 mil alumina substrate with 50 ohm microstrip transmission line, we obtain a high frequency limit of approximately $f_c = 80$ GHz.

## Interpolating Results below the Low Frequency Limitation

Simulations in Momentum at low frequencies are less accurate than simulations at high frequencies. To compensate, Momentum uses an interpolation scheme similar to Adaptive Frequency Sampling (AFS). However, to get low-frequency results by interpolation that are equivalent to high-frequency simulation results, three additional simulations are required. The additional simulations are performed at DC (0 Hz), at the low frequency limit ($f_0$), and at $2f_0$. These three samples are used to generate the interpolated results between 0 Hz and $f_0$.

The low frequency limit $f_0$ is a calculated value that is above the range where inaccurate low frequency simulations occur. The limit is chosen in a rather conservative, empirical way. The limit is a function of the cell edge lengths and of the substrate height. It is inversely proportional to the edge lengths; as the cells become smaller, resulting in shorter edges, the low frequency limit is set higher.

The accuracy of the interpolation results for the low frequencies below $f_0$ is the same as the simulation results for high frequencies. Momentum uses a similar interpolation method it uses in AFS, which is a highly accurate, rational-function interpolation with poles and zeros. For information about AFS, see .

## Parallel Plate Modes

In the region between two infinite parallel plates or ground planes, parallel plate modes exist. Any current flowing in the circuit will excite all of these modes. How strong a mode will be excited depends on how well the field generated by the current source matches with the field distribution of the parallel plate mode.

A distinction can be made between the fundamental mode and the higher order modes. The fundamental mode, which has no cut-off frequency, propagates at any frequency. The higher order modes do have a cut-off frequency. Below this frequency, they decay exponentially and only influence the local (reactive) field around the source. Above this frequency, they propagate as well and may take real energy away from the source.

Momentum takes the effect of the fundamental mode into account. The higher order modes are taken into account as long as they are well below cut-off. If this is not the case, a warning is issued saying that higher-order parallel plate modes were detected close to their cut-off frequency. Simulation results will start to degrade from then on.

## The Effect of Parallel Plate Modes

Let's concentrate on the fundamental mode since this one is always present. The fundamental mode has its electric field predominantly aligned along the Z-axis from the top plate to the bottom plate. This means that this mode creates a difference in the potential between both plates. This mode can be excited by any current in your circuit: electric currents on a strip or via, magnetic currents on a slot. It behaves as a cylindrical wave that propagates to infinity where it feels a short circuit (since both plates are connected there). However, since infinity is very far away, no reflected wave ever comes back.

In a symmetric strip line structure, the current on the strip won't excite the fundamental mode due to this symmetry. However, the presence of a feature such as a slot in one of the plates creates an asymmetry. The slot will excite the fundamental mode!

The accuracy of the Momentum results for calibrated ports degrades when the parallel plate modes become important. For calibrated ports is assumed that they excite the circuit via the fundamental transmission line mode. However, this excitation is not pure. The source will excite the circuit via the parallel plate modes, too. This effect cannot be calibrated out, since those contributions are not orthogonal. The consequence is that the excitation doesn't correspond with a pure fundamental transmission line mode excitation.

## Avoiding Parallel Plate Modes

Since we know the field orientation of the parallel plate modes, it is easy to understand that you can short circuit them by adding vias to your structure at those places where parallel plate modes will be excited.

Similar problems will exist with the measurements. Both plates must be connected explicitly with each other! This won't take place at infinity but somewhere at the side of the board. If no vias are used in the real structure, the fundamental mode may be excited. This mode will propagate to the borders and reflect. Thus, the real structure differs in that from the simulations where no reflection will be seen since a reflection only happens at infinity.

One way to counter the difference between measurements in reality and simulation is to take into account the package (box) around the structure. Another possibility is adding vias to your structure, both in real life and in the simulations. Try to make the physical representation of your circuit as close as possible to reality. However, the actual shape of the via should not be that important. Small sheets can be used to represent the small circular vias that are realized in the real structure.

## Surface Wave Modes

If the substrate is not closed (open or half open), and not homogeneous, surface wave modes exist. The parallel plate mode can be seen as a special case of a surface wave mode. Their behavior is identical. Both are cylindrical waves that propagate radially away from the source. They are *guided* by the substrate. Both fundamental and higher order surface wave modes exist. Similar conclusions can be drawn with respect to limitations such as the effects of the modes.

## Slotline Structures and High Frequency Limitation

The surface wave accuracy deterioration of the calibration process is somewhat more prominent for slotline transmission lines than for microstrip transmission lines. Therefore, slotline structures simulated with Momentum will exhibit a somewhat higher noise floor (10-20 dB higher) than microstrip structures. For best results, the highest simulation frequency for slotline structures should satisfy:

Substrate thickness < 0.15 * effective wavelength

Simulation of narrow slot lines are accurate at frequencies somewhat higher than the high frequency limit determined by this inequality, but wide slotlines will show deteriorated accuracy at this frequency limit.

## Via Structures and Metallization Thickness Limitation

In Momentum, the current flow on via structures is only allowed in the vertical direction; horizontal or circulating currents are not modeled. As via structures are metallization patterns, the modeling of metallization losses in via's is identical to the modeling of metallization loss in microstrip and slotline structures. Simulation of finite metallization thickness can be approximated in Momentum by defining two horizontal metallization patterns and joining the edges with via structures. This however will give only first-order accurate simulation results, as the horizontal via currents are not modeled.

# Via Structures and Substrate Thickness Limitation

The vertical electrical currents on via structures are modeled with rooftop basis functions. In this modeling, the vertical via structure is treated as one cell. This places an upper limit to the substrate layer thickness as the cell dimensions should not exceed 1/20 of a wavelength for accurate simulation results. Momentum simulations with via structures passing through electrically thick substrate layers will become less accurate at higher frequencies. By splitting the thick substrate layer into more than one layer, more via-cells are created and a more accurate solution is obtained.

# CPU Time and Memory Requirements

Both CPU time and memory needed for a Momentum simulation increase with the complexity of the circuit and the density of the mesh. The size N of the interaction matrix equation is equal to the number of edges in the mesh. For calibrated ports, the number of unknowns is increased with the edges in the feedlines added to the transmission line ports.

## CPU Time

The CPU time requirements for a Momentum simulation can be expressed as:

$$\text{CPU time} = A + B\,N + C\,N^2 + D\,N^3$$

where

$N$ = number of unknowns

$A, B, C, D$ = constants independent of N

The constant term A accounts for the simulation set up time. The meshing of the structure is responsible for the linear term, BN. The loading of the interaction matrix is responsible for the quadratic term and the solving of the matrix equation accounts for the cubic term. It is difficult to predict the value of the constants A, B, C and D, as they depend on the problem at hand.

The quadratic and the cubic terms are dominating the overall CPU-time requirements. For small to medium size problems, as the constant C is much larger than D, the solution time is dominated by the loading of the matrix. For larger problems the matrix solve time with its cubic term will eventually dominate the CPU-time requirements.

## Memory Usage

The memory requirement for a Momentum simulation can be expressed as:

$$\text{Memory} = X + Y\,N + Z\,N^2$$

where

N = number of unknowns

X, Y, Z constants independent of N

Like with the CPU time expression, the constants X, Y and Z are difficult to predict for any given structure.

For medium to large size problems, the quadratic term, which accounts for storing of the interaction matrix, always dominates the overall memory requirement. For small structures, memory usage can also be dependent upon the substrate. The substrate database must be read and interpolated, which requires a certain amount of memory. Algorithms to make trade-off between time and memory resources are implemented in the simulator. These algorithms result in additional usage above that required to solve the matrix. Total memory consumption is typically less than 1.5 times what is required to store the matrix, for large matrices.

# Appendix B: Drawing Tips

This chapter provides suggestions and examples when drawing layouts to be simulated using Momentum. These examples take into consideration things that are necessary during the drawing phase to ensure that requirements for Momentum are met.

## Using Grid Snap Modes

Working in Layout enables you to control all geometry precisely. The variety of available snap modes can help you draw and position shapes. As an example, if you are connecting shapes, you should have the vertex snapping mode turned on. You will also want to review snap mode settings prior to adding ports to a layout.

Snap grid spacing also helps control the positioning shapes. In general, snap grid spacing should be about 1/2 of the minor grid point value. For example, if the minor grid points are 1.0, then the snap grid spacing should be 0.5. This makes it easy to know how your geometry will snap into place.

For more information on snap modes and snap grid spacing, refer to Chapter 2 of *Layout*.

## Choosing Layout Layers

The standard set of layout layers begins with the layer named *default*. Do not use this layer for drawing your circuit. The first valid layer for Momentum is *cond*. For more information on layout layers, refer to the *Layout* manual.

## Keeping Shapes Simple

In general, when drawing shapes, you should use a minimal number of vertices per shape, since this will make the mesh easier to compute. When drawing curved objects, consider using a relatively large value for Arc/Circle Radius (under *Options > Preferences > Entry/Edit*). This will minimize the number of vertices, and facets used to represent the shape.

# Merging Shapes

Merging shapes is often useful for eliminating small geometry overlaps and can also, in some cases, result in simpler mesh patterns. For example, if you draw a layout using multiple polygons and you suspect there is some overlap, merging the polygons will prevent the system from returning an overlap error. To merge the shapes, select each shape that you want to merge, then choose *Edit > Merge*.

# Viewing Port and Object Properties

Any item selected in a layout, including ports, shapes, or other components, has associated properties. One way to view the properties is to select the object of interest, then choose *Edit > Properties.* This method also enables you to change object properties. If you want to change properties that have been attributed to an object for Momentum, you should change these properties only through the Momentum Menu selections.

To view the properties of the entire layout, choose *Options > Info.*

# Using Layout Components

There are some limitations on the components that are available from the layout palettes:

- Do not use substrate components that might appear on layout component palettes. You must use the dialog box under *Momentum > Substrates > Create/Modify* to create or edit substrate definitions.

- Lumped elements cannot be included in a layout that is to be simulated by Momentum.

- Components from the Microstrip palette can be used, with limitations:

  - The substrate name (for example, Subst = MSub1) is not used in Momentum. Substrate definitions must be loaded from the *Momentum > Substrates* menu.

  - Any electrical properties, if defined, are not used. For example, the parameters CPUA, RsT, and RsB in MFTC are ignored.

- For most of the components in Layout, only the geometry is transferred to Momentum. Any electrical properties must be defined from the Momentum menus.

- Components from the Printed Circuit Board or Waveguide palette cannot be used in a layout that will be simulated using Momentum.

- Stripline and suspended substrates components can be used, but the substrate name parameters for these components will be ignored.

# Drawing Coplanar Waveguide in Layout

Coplanar waveguide is, generally, slots filled with a dielectric or air, surrounded by metal, as shown in the illustration below.
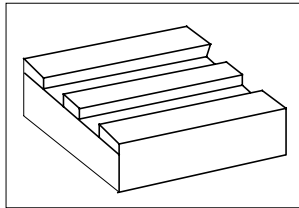


There are two basic methods for constructing coplanar waveguide in layout:

- The objects that you draw are the slots on a groundplane. In the substrate definition, the layout layer that the slots are drawn on is mapped to a slot metallization layer. Additionally, the top plane of the substrate definition is defined as a ground plane. This method gives you infinite grounds on the coplanar plane. An example is the coplanar waveguide bend, shown below.
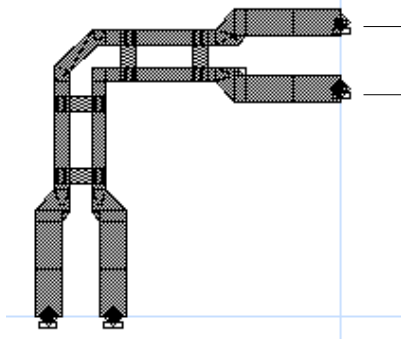


You draw the slots. By choosing a slot metallization layer, the area around the slots is treated as a ground plane, the slots are not metal.

- The objects that you draw represent ground. For example, you can draw strips with gaps in between. The gaps are treated as slots, the strips are treated as ground. This layer is mapped to a metallization layer defined as a strip. Additionally, the top plane of the substrate definition is defined as a ground plane. This method gives you infinite grounds on the coplanar plane. Typically, this method is used when coupled microstrip lines are created initially as schematic elements.



Drawing slots on a ground plane the preferred method because a mesh with fewer edges or unknowns will result, compared to meshing the strips. If a coplanar waveguide design is drawn with strips, it can be converted to a slot pattern with a boolean operation if a bounding box is provided.

# Adding a Port to a Circuit

You can add a port to a circuit either from the Schematic window or a Layout window. The procedures below include considerations for adding ports to a circuit that will be simulated using Momentum.

## Adding a Port to a Schematic

1. Select a port using by choosing the menu item **Component > Port**, or the port

   icon from the tool bar:

2. Position the mouse where you want the port and click. Verify the connection has been made.
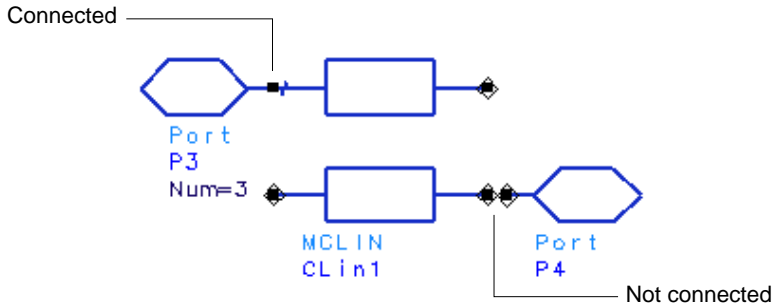


3. Select the port and choose **Edit > Component > Edit Component Parameters**.

4. The parameter *layer=* indicates the layer that the port is applied to. When you convert your schematic to a layout, all components will be assigned to layout layers. You can assign the port to a specific layer at this time. To change the layer, select **layer=** from the *Select Parameter* list, and choose a new layer from the *Parameter Entry Mode* listbox. Make sure that the layer you choose will also be mapped as a strip or slot metallization layer. For more information about metallization layers, refer to Chapter 3, Substrates.

5. Click **OK** to accept the new layer specification.

When you generate a layout from the schematic, you should verify that the port is positioned correctly. For more information, refer to the procedure for adding a port to a layout and to "Considerations" on page B-7.

## Adding a Port to a Layout

To add a port:

1. Determine where you want to position the port. A port can be applied to:

   • The pin of a component

   • The edge of a component or object (such as a rectangle or polygon), usually at the midpoint

• The surface of an object



Port connected to a pin       Port connected to an edge       Port connected to a surface

---
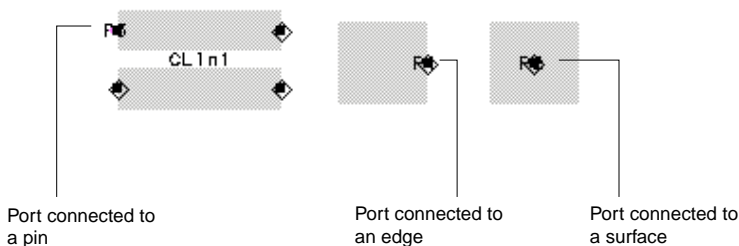
**Note** Do not place a port on the corner of an object. When you add a port to an edge, make sure the appropriate snap modes are enabled. It is a good practice to enable the modes *Midpoint Snap* and *Edge Snap* (under the *Options* menu) to ensure accurate simulation results.

---

2. If you want to add a port to the surface of a polygon, you may need to disable the following or other snap modes (under the *Options* menu):

   • Pin snap

   • Edge snap

   • Midpoint snap

3. Identify the name of the layer on which the component or shape is entered. If you don't know the name of the layout layer, choose **Options > Layers**. Select each layer, noting the color in the dialog box. When the color matches that of the one in the Layout window, note the name of the layout layer. Dismiss the dialog box.

4. Select a port using by choosing the menu item **Component > Port**, or the port

   icon from the tool bar:

5. The Port parameters dialog box is displayed. The parameter layer indicates the layer that the port will be applied to. The parameter defaults to the currently active drawing layer.

6. This should be the same as the layer you noted in step 3. To change the layer, select **layer=** from the Select Parameter list, and choose a new layer from the Parameter Entry Mode listbox. Make sure that the layer you choose is also a

strip or slot metallization layer. Do *not* apply ports to shapes that are on layers that are mapped to via metallization layers. For more information about metallization layers, refer to Chapter 3, Substrates.

---

**Note**   If you cannot find the layer name, choose *Options > Layers*, then select the layer name of interest from the Layers list. Note the layer number in the Number field. In the Port dialog box, select the *layer=* parameter and, under Parameter Entry Mode, select *Integer Value*. Enter the layer number, then click *Apply*.

---

7. Click **Apply** to accept the new layer specification.

8. Position the mouse where you want to place the port and click. The port is added to the circuit.

---

**Note**   If you are applying a port to an edge, the port must be positioned so that the arrow is outside of the object, pointing inwards, and at a straight angle. Generally, this happens automatically when you add a port to an edge. You may need to zoom in to verify this.

---



9. The Instance Name in the Port dialog box is incremented if you want to add another port. Verify the **layer=** parameter, then use the mouse to add the next port.

10. When you are finished adding ports, click **OK** to dismiss the dialog box.

## Considerations

Keep the following points in mind when adding ports to circuits to be simulated using Momentum:

• The components or shapes that ports are connected to must be on layout layers that are mapped to metallization layers that are defined as strips or slots. Ports cannot be directly connected to vias. For information on how to define strips and slots, refer to "Defining Metallization Layers" on page 3-12.

- Make sure that ports on edges are positioned so that the arrow is outside of the object, pointing inwards, and at a straight angle.

- Make sure that the port and the object you are connecting it to are on the same layout layer. For convenience, you can set the entry layer to this layer; the Entry Layer listbox is on the Layout tool bar.

- A port must be applied to an object. If a port is applied in open space so that is not connected to an object, Momentum will automatically snap the port to the edge of the closest object. This will not be apparent from the layout, however, because the position of the port will not change.

- If the Layout resolution is changed *after* adding ports that are snapped to edges, you must delete the ports and add them again. The resolution change makes it unclear to which edges the ports are snapped, causing errors in mesh calculations.

---

**Note**   Do not use the ground port component in circuits that will be simulated using

Momentum (*Component > Ground* or the toolbar button  ⊥̄  ).

Either add ground planes to the substrate or use the ground reference ports that are described later in this chapter.

---

# Drawing Vias in Layout

Vias are drawn as lines or closed polygons. A simple line segment via is the simplest and most practical way of drawing a via. Momentum creates a via by extruding the object that is mapped as via through the substrate layer it is applied to. Vias drawn as lines are often called sheet vias because, when the line is extruded through the substrate, it is treated as a horizontal metal sheet. For vias of other shapes, you draw a closed polygon. So, for example, for a cylinder via you draw a circle. When the shape is mapped to a via metallization layer in Momentum, another dimension is added to

the object in order for the shape to cut through the substrate. Thus a line becomes a sheet, a circle becomes a cylinder.



representations
of vias in Layout

bridge

via

via

strip

strip

via

vias extruded
through a
substrate layer
in Momentum
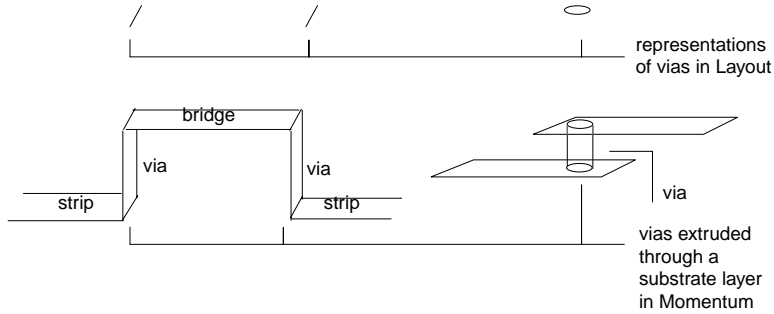
Regardless of how you draw vias, avoid having them extend over the sides of the objects that they connect to. Vias must be on the edge or inside the object. Any portion of the via outside of the object boundaries will not be taken into account during the simulation. The figure below illustrates various vias connecting two strips.



Incorrect:
vias overlap the
object edges.

Efficient via:
sheet via with
width matching
width of strip.

Circular Via:
correct, but
may not be efficient.
Use an appropriate
arc recognition.

Correct, but
not as efficient
as sheet via
with matching
width to strip.
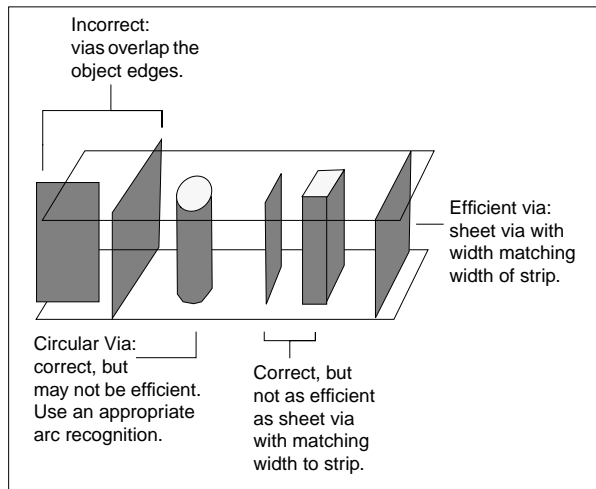
Vias are treated in the following manner:

- Vias are represented as infinitely thin vertical sheets of metal. If a cylinder is drawn as a via, then it is also treated as separate sheets of metal that comprise the cylinder walls.

- Vias are assumed to be opened, not covered, with current traveling on the sheet. Vias can be drawn as covered, using a cap (cover) of the same size as the via end, which is a terminating metal strip layout layer.

- A via cannot be coincident with another via if they cut through the same substrate layer and they are on different layout layers. But two vias can be coincident on the same layout layer.

- Vias should be wholly contained within any primitives that are to be connected.

- Vias can be specified with loss but are considered to have zero thickness, as are all metals in Momentum.

## Vias through Multiple Substrate Layers

When drawing vias that cut through multiple substrate layers, vias must be copied exactly (both dimensions and location) onto each layout layer. One via must be drawn or copied for each substrate layer that it cuts through. In Momentum, this will result in multiple vias, having a different names, and being represented as multiple process layers. The following are two methods for drawing the vias:

- Draw vias on multiple layers by setting the command *Draw > Entry Layer* to the target layer. Select the polygon or polyline. Then use *Edit > Copy/Paste > Copy to Layer* to create an exact replica of the via on the target layer.

- Draw a via for one layer only. Use the **Metallization** tab to assign the via layer to all substrates.
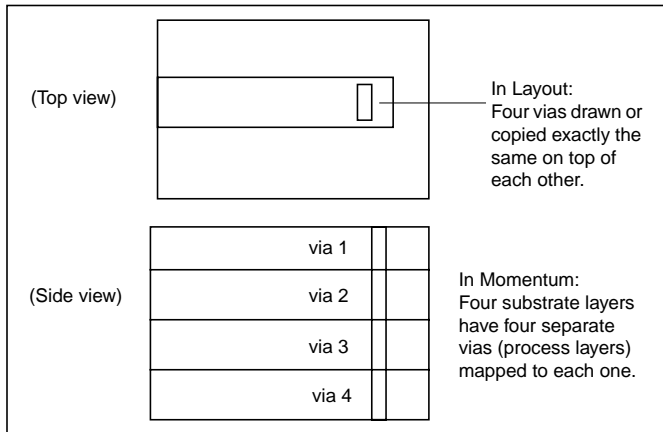


## Using Vias to Model Finite Thickness

You can use vias to model the finite thickness of a metal structure in the following way:

- Draw the top surface of the structure.

- Copy the shape onto another layout layer, positioning the shape exactly over the first one. The shapes must match exactly.

- Draw a sheet via on the perimeter of one shape, but make sure the via is on a third layout layer.

- As part of the substrate definition, make one layer an interface layer, with a finite thickness and characteristics of the metal that you want to model.

When you apply this layout to a substrate definition, you will map the via to the interface layer described above, then map the shapes to the metallization layers that appear above and below this substrate layer. Recall that metallization layers will be present only between two substrate layers, so you may need to add layers, such as air,

above and below the interface layer. The following drawing illustrates how to draw the metal structure.
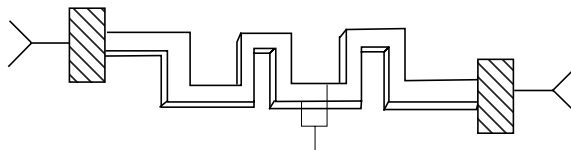


Draw the via along the perimeter of one of the strips.

# Drawing Air Bridges in Layout

If you are simulating an air bridge, you will have to create a layout and a substrate definition that will support the geometry. This means adding an air layer (height equal to the air bridge) which the vias will cut through. For example, you would have some geometry (like a spiral) on a dielectric layer. Then a via would rise vertically, cutting through the air layer, where it would connect to a metal bridge. Then another via would connect from the bridge, down to the previous layer, and onto another path. For further information refer to "Substrate for Designs with Air Bridges" on page 3-27.
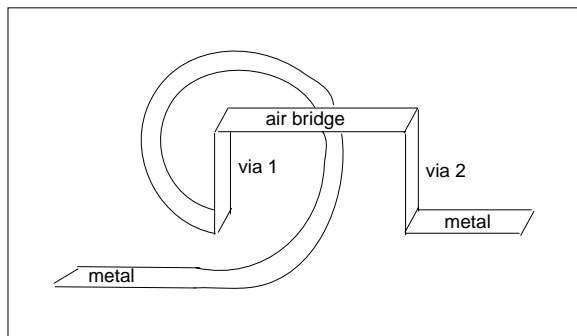


# Ports and Air Bridges in Coplanar Designs

When you draw coplanar waveguide as slots on a ground plane, it is treated, by default, as a four-port device composed of two, coupled slot lines. To make sure that it will be simulated as coplanar waveguide, be sure to apply the Coplanar port type to the ports on the circuit. This port type combines the two ports on the same reference

plane, and the two ports are treated as one. For instructions on how to apply the Coplanar port type, refer to "Defining a Coplanar Port" on page 4-14.

If you do not use the Coplanar port type, a moding problem can result when a coplanar waveguide transmission line also propagates a slot mode. If Coplanar ports are used, the slot modes are short-circuited at the port interfaces. However, if the slot modes are excited somewhere in the circuit, they can cause resonances at certain frequencies. This will appear as non-physical results because the external boundary conditions are critical to the electrical behavior in such cases. By adding air bridges (straps) near the discontinuities the voltage level of the two grounds can be equalized to eliminate the slot mode. An example of air bridges can be found on the coplanar waveguide bend example, *CPW_bend_prj*. For information on how to include air bridges in a circuit, refer to "Substrate for Designs with Air Bridges" on page 3-27.

Drawing Tips

# Appendix C: Exporting Momentum Data

Momentum data can be exported to Agilent HFSS and to IC-CAP for additional analysis. The method you use to export data depends on the target application.

## Exporting Momentum Data to IC-CAP

Momentum can write S-parameter output files in the IC-CAP Measured Data Management (MDM) File Format. These ASCII data files can easily be read and used for modeling purposes in IC-CAP 5.x.

### Enabling the IC-CAP MDM File Output

The IC-CAP MDM files are not created by default. You must create or edit the Momentum configuration file:

`($HOME)/hpeesof/config/momentum.cfg`

and add the line:

`MOM_MDM_OUTPUT = TRUE`

If this variable is set, you will find the IC-CAP MDM file once the Momentum simulation is complete:

`(current_ADS_project_name)/mom_dsn/(current_layout_name)/proj.mdm`

This file will only be available for 1-port and 2-port structures.

The data is densely sampled (as a function of frequency) if the *adaptive* simulation mode was used during the Momentum simulation, otherwise, only the requested sample points appear in the *proj.mdm* file.

### Disabling the IC-CAP MDM File Output

If you do not want output in an *.mdm* file, you must disable the *MOM_MDM_OUTPUT* variable.

You must set:

`MOM_MDM_OUTPUT = FALSE`

or remove that line in the Momentum configuration file:

`($HOME)/hpeesof/config/momentum.cfg`

## Example

This is an example of the contents of an *.mdm* file that was created from a Momentum simulation.

```
BEGIN_HEADER

ICCAP_INPUTS

freq F LIST 16    0.000000e+00 1.000000e+07 5.008333e+09 8.965347e+09
    1.000667e+10 1.667111e+10 2.666778e+10 3.000000e+10

ICCAP_OUTPUTS
s S B C GROUND
END_HEADER

BEGIN_DB
#freq R:s(1,1) I:s(1,1) R:s(1,2) I:s(1,2) R:s(2,1) I:s(2,1) R:s(2,2) I:s(2,2)

0.000000e+00  0.00000  0.00000  1.00000  0.00000  1.00000  0.00000  0.00000
 0.00000
1.000000e+07  0.00000 -0.00139  0.99999 -0.00238  0.99999 -0.00238  0.00000
-0.00139
5.008333e+09 -0.57309 -0.24306  0.30188 -0.71430  0.30188 -0.71430 -0.57309
-0.24306
8.965347e+09 -0.59290 -0.28893 -0.38554  0.39683 -0.38554  0.39683 -0.59291
-0.28893
1.000667e+10 -0.85726  0.47812 -0.00500 -0.00983 -0.00500 -0.00983 -0.85726
 0.47812
1.667111e+10  0.38444  0.32013  0.52995 -0.65935  0.52995 -0.65935  0.38444
 0.32013
2.666778e+10 -0.44899 -0.19974 -0.39885  0.68719 -0.39885  0.68719 -0.44899
-0.19974
3.000000e+10  0.03191  0.96691 -0.01394 -0.00395 -0.01394 -0.00395  0.03191
 0.96691


END_DB
```

# Exporting Momentum Data to Agilent HFSS

You can export Momentum data to Agilent HFSS without running a Momentum simulation first. The files that Agilent HFSS requires to reconstruct the geometry are generated by using a menu command in the ADS Layout window. Using the menu command saves time since you do not need to complete a simulation.

---

**Note**   Agilent HFSS 5.5 (or later) can import the files that are generated when you use the menu command in the Layout window, or files from a successful Momentum simulation. Agilent HFSS 5.4 is only able to import projects that have been successfully simulated by Momentum.

---

To generate files and import them into Agilent HFSS:

1. In the ADS Layout window, choose **Momentum** > **HFSS** > **Save Files for HFSS**.

   The files are generated and saved in the current project under <project_name>/*mom_dsn*/<design_name>/*hphfss*. They contain the design geometry and Momentum setup.

2. In Agilent HFSS, choose **Project** > **Translate** > **ADS/Momentum Layout**.

   This converts the files into an Agilent HFSS project.

Exporting Momentum Data

# Appendix D: Momentum Optimization - Background Information

Optimization is an area of applied mathematics dealing with minimization (a maximization problem can be formulated as minimization) of an objective function. Many optimization algorithms have been developed over the years. Some of the best are made available here for the purpose of layout design. Momentum Optimization employs a comprehensive set of optimizers including L1, L2 (least squares), Minimax, quasi-Newton, random and simulated annealing algorithms. All of these have been adapted specifically to handle engineering design problems.

## Optimization Variables

In the process of minimizing an objective function the optimizer makes changes to optimization variables. In Momentum Optimization some or all of the parameters defined in layout parameterization can be designated as optimization variables. You can designate parameters as variables by clicking on the *Allowed to vary* box in the Optimization Parameters Editor so a check mark appears. At least one parameter must be selected as an optimization variable in order to perform an optimization.

There are several reasons why you may want to fix a parameter value and not allow the optimizer to vary it. For example, you may want to parameterize one layout for different applications where different constraints may be required. Then you use different subsets of all parameters as optimization variables in separate optimizations of the layout. A typical situation, however, is when you may want to carry out the optimization process in a few phases. You start with one or two optimization variables that may influence the responses most significantly. Once that phase is finished you may release more variables and restart optimization to further refine the responses.

The values relevant to optimization are:

- The Starting value is the value at which the optimizer begins its search
- The Optimal value is the value declared by the optimizer as the best when the search is finished

By default (if the Starting Value entry is not filled in) the nominal value is taken as the starting value. The assumption is that the nominal design is not far from the final design. However, you are free to enter a different value as the starting value. This is useful if you want to restart optimization from a different starting point or if the

nominal design is drawn far away from the expected solution simply for drawing convenience.

Whenever possible you should run optimization with a good selection of the starting values. The closer the starting point is to the solution, the faster the optimization process might be, requiring fewer Momentum simulations. There is also a better chance of avoiding traps of local minima.

If a parameter is not allowed to vary, its value is fixed at the Starting Value, as entered in the Optimization Parameters Editor. If no entry is made, its value is fixed at the nominal value (the default Starting Value).

The optimal values are reported in the "Parameters" list in the Optimization Parameters Editor after optimization is finished. They are stored with your nominal design and can only be changed by the optimizer after another optimization run leading to a different solution.

A mechanism is provided to update the starting values with the optimal values. By clicking the Back Annotate Optimal Values button, you can replace the current starting values with the optimal values. This might be useful if you want to restart optimization under different conditions and you believe this would be a good selection of the starting values. The starting values can still be modified manually.

Two additional values are relevant to optimization:

- The Lower bound is the lowest value the optimizer is allowed to generate
- The Upper bound is the highest value the optimizer is allowed to generate

The bounds establish the limits of the exploration region for the optimizer. It is highly recommended to limit the exploration region if you can. The importance of reasonably setting the lower and upper bounds on the optimization variables is particularly evident in the case of random optimizers. By exploring vast areas far away from the solution, a significant computational effort may be wasted.

# Design Goals/Specifications

A single design goal consists of a frequency plan, a response of interest, the specification type, and a numeric value that the response should either exceed or achieve. The latter is referred to as the goal value. A common term for a design goal is "design specification". It will be used here as well. Design goals are defined in the Optimization Goals Editor (*Momentum > Optimization > Goals*).

A frequency plan can consist of either a single point or a linear sweep. As a response of interest you can select an S parameter in any of the following forms:

- Magnitude in dB

- Magnitude

- Phase (in degrees)

- Real part

- Imaginary part

The S-parameters available for optimization are with respect to the reference impedance of $50\Omega$. Any combination of the above forms for any selection of port indices can be optimized simultaneously. Two different forms for the magnitude are provided intentionally. It makes a difference for the optimizer what form is used, and thus the optimization process may proceed and converge differently. The first form describes relative variations, while the other describes the absolute variations.

The specification type is defined using one of the following symbols:

- <

- =

- >

They are called an upper specification, an equality specification and a lower specification, respectively. The first and the third type together are referred to as inequality specifications. For window specifications you need to define two design goals for the same frequency plan: one using an upper specification and the other one using a lower specification.

You can define multiple design goals for different frequency plans, and using different specification types. You can emphasize/de-emphasize their relative importance using weights associated with each goal.

You can run optimization using a subset of all design goals that have been defined. Each design goal can be activated (this is the default setting) or deactivated. At least one design goal needs to be defined and activated to be able to run optimization.

The Momentum simulation results, subsequently stored in the database (see Actual EM Simulations and Database), are generated for all frequency points defined in the Optimization Goals Editor. Any frequency plan that may have been defined using *Momentum > Simulation* is not used. All the design goals, whether activated or not, are used to determine the frequency points for EM simulations. Thus, activation or

deactivation of any of the goals does not obsolete the database. This is very useful if you want to refine your design by restarting optimization with a different subset of design goals. In other words, you may simply define additional "design goals" that are deactivated in initial optimization and this will allow you to monitor the circuit behavior at the corresponding frequency points.

# How Design Goals are used by the Optimizers

Momentum Optimization creates a single function to quantify the deviation of the circuit responses from the design goals. It is called an objective function. A minimum of the objective function corresponds to the optimal solution. There are many ways to define an objective function. One important feature of various formulations is that the objective function is

- **Positive** if at least one design goal is not met (one design specification is violated)

- **Zero** if all design goals are met but at least one of them is just met (the response becomes equal to goal value)

- **Negative or zero** if all design goals are met with some margin

It is worth noticing that design goals may be defined too tight so that no solution satisfying all goals exists. Thus, negative values of the objective function may not be attainable.

An error function for every goal and at every frequency point is formulated based on the response value, goal value, the type of specification and the weight. The formulation follows the same rule that a positive value means violation of the specification. In the following discussion, we use the notation

$F_j$   j=1, 2, ...., m

to denote all individual error function values. Momentum optimization uses the following objective functions formulated from individual error function values:

- **Max Error** (minimax objective function) defined as

$$U = max\{F_j\}$$

- **L1 Error** (least-one objective function) defined as

$$U = \sum_{j=1}^{m} |F_j|$$

- **L2 Error** (the classical least-squares objective function) defined as

$$U = \sum_{j=1}^{m} F_j{}^2$$

- **GL2 Error** (the generalized least-squares objective function) defined as

$$U = \begin{cases} \sum F_j{}^2 \Big| F_j \geq 0 & \text{if at least one } F_j \geq 0 \\[2ex] \dfrac{-1}{\displaystyle\sum_{j=1}^{m} \dfrac{1}{F_j{}^2}} & \text{if } F_j < 0 \text{ for all } j \end{cases}$$

The Max Error and the GL2 Error objective functions are more suitable for inequality and window specifications, as they become negative when all goals are exceeded by some margin. The L1 Error and L2 Error objective functions do not account for design goals that have been exceeded. The individual error functions corresponding to inequality specifications are set to zero when they become negative.

Each objective function has specific advantages over other formulations. They are briefly discussed in "Optimization Algorithms" on page D-6. The optimal solution in terms of one objective function may be different (and it usually is) from the optimal solution in terms of another objective function.

The user's choice of the objective function is made indirectly by selecting the Optimization Type in the Optimization Control Editor.

# Optimization Algorithms

Many different methods exist for solving minimization problems. None of them is universally applicable. Momentum Optimization employs several state-of-the-art optimization types (algorithms/objective function formulations) to help you achieve your design goals. An algorithm together with its implementation in the program is referred to as an optimizer. Each optimizer constructs an objective function from the design goals and then employs a unique algorithm to minimize the object function.

- The Minimax optimizer uses the Max Error objective function. It concentrates its effort on minimizing the largest violation of specifications. Once all the specifications are satisfied it continues its effort on maximizing the smallest design margin (minimizing the negative objective function). Thus, it is most suitable for optimization with upper and lower specifications. The Minimax optimizer tends to achieve an equi-ripple solution: while attempting to minimize the largest individual error another "opposing" error may be increased. The optimizer will stop when such "opposing" errors become equal.

- The **L1** optimizer uses the **L1 Error** objective function. It tries to drive as many individual errors as possible to exactly zero. Thus, it is most suitable for optimization involving equality specifications. In the case of inequality specifications this optimizer will stop once all design goals are met.

- The **L2** optimizer uses the **L2 Error** classical least-squares objective function. The smoothness of the objective function makes it a very robust and reliable technique. In the case of inequality specifications this optimizer will stop once all design goals are met.

- The **Quasi-Newton** optimizer uses the **GL2 Error** objective function. It implements the Fletcher-Powell-Davidon algorithm. The objective function formulated for this optimizer, similarly to Minimax, facilitates improving the design margin when all specifications are satisfied (design centering).

- The **Random** optimizer also uses the **GL2 Error** objective function. The Random optimizer uses a random search algorithm and does not require a gradient. It is less susceptible to potential traps of local minima when compared with gradient-based optimizers, but it also converges less efficiently. This optimizer searches the entire exploration region determined by the lower and upper bounds on the optimization variables. It is thus important to reasonably set those bounds.

- The **Simulated Annealing** optimizer uses the **GL2 Error** objective function. It employs a global optimization strategy, but requires a large number of simulations to reach a solution.

The first four optimizers are so-called gradient-based. They need first-order partial derivatives in addition to the value of the objective function to make a decision on the subsequent steps. They usually move fairly quickly towards a minimum particularly if one exists close to the starting point. If possible, they tend to search fairly limited areas of available exploration regions and are susceptible to traps of local minima. The last two optimizers do not require gradient information. Since they normally use a large number of steps, a caution is recommended when trying to use them in conjunction with more time consuming EM simulations.

The first three optimizers have been specifically developed for minimization of their corresponding objective functions. Therefore, their performance in handling their objective functions is likely to be far superior over any other optimizer. The last three optimizers are general purpose techniques.

Momentum Optimization provides the **default** Optimization Type such that it is best suited to the design goals provided.

- If there are inequality design goals the default optimizer is Minimax.
- If there are only equality design goals the default optimizer is L1.

It is recommended to accept the default setting unless you have a good reason to change it.

# Stopping Criteria for the Optimization Process

The optimizer stops when it cannot minimize the objective function anymore. Except for local minima, improperly set design goals, or rare numerical deficiencies, this should be the desired solution. Momentum Optimization provides two additional stopping criteria for the optimization process. The optimizer will stop even if it would otherwise continue if:

- The *Maximum Number of Iterations* specified in the Optimization Control Editor is reached; if the optimization does not converge to an optimal solution before reaching this limit, it will stop and present the best result up to that point as the solution

- The *Stop as soon as the specs are met* radio button in the Optimization Control Editor is on (the default setting); this instructs the optimizer to stop immediately when all design goals are met for the first time

The first stopping criterion is essential for making sure the optimizer stops in the rare cases of numerical instability, slow convergence, or excessive iterations making insignificant refinements to find the solution. This limit is fairly predictable for typical optimization problems. For the L1, L2 and Minimax optimizers it usually takes between 10 and 30 iterations to reach the solution. This is fairly independent of the number of optimization variables. The limit on the number of iterations is not a limit on the number of actual Momentum simulations. The number of Momentum simulations needed to reach a solution is far less predictable. It may vary significantly depending on the number of variables, whether interpolation is used or not, the size of the interpolation interval, and the size of the exploration region.

The second criterion is intended for use in conjunction with inequality specifications. It makes no difference for the L1 and L2 optimizers since their objective functions do not account for exceeding the specifications. In the case of other optimizers this criterion does not require the goals to be met exactly. Depending on the optimization trajectory the solution may already feature some goal margin when the process is stopped. If you want to achieve a design goal exactly you should use an equality specification instead.

You can override the second stopping criterion by selecting the *Continue to obtain a centered design* option. This is a preferred choice to achieve good designs for circuits such as filters, particularly if manufacturing tolerances are of concern. However, if there are no "opposing" individual errors, this option may lead to excessive iterations and undesired solutions. For example, if you want to minimize coupling between two strips by adjusting the spacing, the optimizer will attempt to drive the strips as far apart as allowed by the upper bound imposed on the spacing.

You do not need to be afraid of wasting valuable computer resources and time if you want to interrupt or stop optimization. If you restart optimization under identical conditions, the optimization process will repeat itself using data from the database and it will arrive almost instantly at the same stage where the interrupt occurred. Similarly, if optimization is stopped because either the maximum number of iterations has been reached or all the goals have been met you can restart optimization after changing the corresponding stopping criterion. Again, the optimizer will arrive almost instantly at the stage where the stop occurred.

# Appendix E: Momentum Optimization - Response Interpolation

In order to improve the overall efficiency of the optimization process, Momentum Optimization employs parameter discretization and response (S parameters) interpolation. It also maintains a database that contains the results of every Momentum simulation performed by the optimizer.

It is under the user's control how each of the parameters is discretized (see "Interpolation Grid and Resolution" on page E-1) and what interpolation scheme is applied (see "Types of Interpolation" on page E-8). The interpolation feature can be turned off (see ""No interpolation" Option" on page E-7).

The database is created and maintained by the program. The user can delete an obsolete database (see "Removing Obsolete Databases" on page E-3). The database is specific to Momentum Optimization. Any results (datasets) obtained using the menu option *Momentum > Simulation* are not stored in the database because they do not contain any information on layout parameterization.

## Interpolation Grid and Resolution

If the interpolation feature is used (default setting) then the parameter values are discretized. The resulting discrete parameter values establish a rectangular, multidimensional interpolation grid. The size of the grid can be controlled by the user by means of the *Interpolation Resolution* defined in the *More* dialog in the parameter definition area of the Optimization Parameters Editor. Momentum Optimization associates each parameter with an interpolation interval whose size (length) is defined as:

$$\text{Interpolation Interval} = \frac{|\text{Perturbed Value} - \text{Nominal Value}|}{\text{Interpolation Resolution}}$$

Two consecutive grid points in the respective direction are Interpolation Interval apart. Coarser grids correspond to smaller interpolation resolutions while higher interpolation resolutions mean finer grids. The interpolation resolution can only be a positive integer. The coarsest grid available corresponds to an interpolation resolution of 1. Then, Interpolation Interval is equal to the distance between the nominal and perturbed parameter values.

Since a separate interpolation resolution is defined for each parameter, a grid can be coarser for one parameter and finer for another. This provides the flexibility in accommodating different sensitivities of responses with respect to different parameters.

By definition, the parameter values corresponding to the nominal and perturbed designs are on the grid. The grid thus expands from those two points in both positive and negative directions of each parameter. The grid is not bounded by the nominal and perturbed values, but may be limited for other reasons (see "Zero Parameter Values" on page E-5 and "The Impact of Parameter Bounds on Interpolation" on page E-6).

# How Does the Interpolation Feature Work?

If the interpolation feature is enabled, the only points (sets of parameter values) that can be submitted for Momentum simulation are on-grid points.

A point is off the grid if at least one of the parameter values falls in the interior of its interpolation interval. If a point of interest (requested by the optimizer) is off the grid then a number of neighboring on-grid points, called the base points, are selected by the program. Momentum simulation is requested at all of those base points. The responses at the point of interest are then calculated by interpolation. This interpolation is performed with respect to the parameter values.

The aforementioned rule applies to all situations, including a single point simulation such as for the optimal parameter values. Therefore, the optimal results (displayed after optimization is finished) are normally obtained by interpolation. Similarly, if you define two parameters, allow one to vary by the optimizer, and keep the other one fixed at an off-grid value, it may look like the optimizer "changes" the "fixed" parameter for EM simulations. This is correct. Indeed, the neighboring on-grid values are used in order to interpolate the responses at the requested fixed value.

# Actual EM Simulations and Database

Before calling Momentum to actually perform a simulation at a base point, the database is checked to see whether the requested dataset is already there. If it is, the results from that dataset are reused instead of performing an actual EM simulation. If not, Momentum is invoked to simulate the design corresponding to that base point. After simulation is completed, the database is augmented by the new results. Once in the database, these results can be reused whenever the responses at the same base point are needed.

During optimization, Momentum is invoked for an EM simulation only if at least one parameter has moved beyond its interpolation interval from the previous iteration. For small changes in parameter values, interpolation is used to obtain the responses instead of calling for additional Momentum simulations. This can substantially reduce the total number of EM simulations needed for optimization.

# Removing Obsolete Databases

A *Delete Previous Database* check box is available in the Optimization Control Editor. The default setting is "unchecked" so that the database, if one exists, will be reused.

The database is a very valuable asset since it may contain the results of hours or days of EM simulation time. Therefore, care should be exercised while asking the program to remove an existing database.

There are several reasons why an existing database may become obsolete. All of them are related to the information contained in the database being no longer compatible with the current setup for optimization.

The database contains the information on:

- The number of parameters
- The number of frequencies
- The number of ports

Each dataset in the database contains information on:

- Parameter values
- Frequency point values
- S-parameters for each frequency point

The database has no knowledge of the layout information, including the nominal and perturbed designs, or the actual parameter values associated with those designs. Meshing setup, units, etc., are also not known to the database.

Therefore, in order to preserve the validity of an existing database, you should not change:

- The nominal and perturbed design layouts, including material properties and port definitions
- The number and definition of parameters, except for allowing/disallowing them to vary
- The sequence in which the parameters are listed in the Optimization Parameters Editor
- Meshing parameters
- The nominal and perturbed parameter values
- Frequency plans, except for activation/deactivation of individual goals or changing the selection of individual S-parameter responses

Whenever possible, the program will detect potential incompatibility with the existing database and issue a warning. Certain changes such as layout modifications cannot be detected. If the existing database is not deleted, additional results of Momentum simulations will be added to it. This may lead to a database containing, for example, some datasets simulated by Momentum with an edge mesh, and others without it.

Before deleting a database, it is advisable to consider other options. The simplest is to rename the current design using "Save As" or "Copy Design" and institute desired changes within the new design.

# Hints for Selecting the Interpolation Resolution

A good choice for the interpolation interval is obviously problem dependent. If the interval is too large, then the interpolated responses may not be accurate. On the other hand, a very small interpolation interval increases the number of EM simulations and ultimately defeats the purpose of using interpolation.

A practical approach is to start with a coarse resolution (for example, 4) and then increase the resolution if the optimized solution is not sufficiently accurate. The

choice of the resolution also depends on how large an increment in parameter value has been established by the nominal and perturbed designs.

For small perturbations, say less than 10%, a value lower than the default setting for the interpolation resolution might be acceptable. If a large perturbation is used (for example one value is twice as large in magnitude as the other) the default resolution is unlikely to be adequate. The sensitivity of the response with respect to the parameter value is an important factor. In many designs, fortunately, designers may have a good insight into their design sensitivity before setting up the optimization task.

If an increasing sequence of interpolation resolution values is used, it is advisable to use an exact multiple of the previous value to set the subsequent value. This will guarantee that all previous grid points will be on the new grid, thus making the previously simulated points available for reuse if needed.

# Zero Parameter Values

As already mentioned, the parameter values corresponding to the nominal and perturbed designs are on the grid. Depending on your definition of the nominal and perturbed layouts, as well as of the interpolation resolution, the zero value of a parameter may or may not be on the grid.

A zero value of a parameter may require special handling if it falls on the grid. If the parameter represents a geometrical dimension such as length, the value of zero may lead to a degenerate geometry. In order to avoid Momentum simulation of a degenerate geometry, the selection of base points is modified in cases when a zero value would normally be needed. When the point of interest falls into the first interpolation interval (with zero as one of its endpoints) the next two grid points are selected as the base points and the results are determined by extrapolation.

As an exception, a zero value grid point is treated like any other grid point in the following three situations.

- Either the nominal or perturbed value is exactly zero
- The nominal and perturbed values are of the opposite signs
- At least one of the parameter bounds is of the opposite sign than the nominal and perturbed values which are of the same signs

In any of the above cases it is assumed that the zero value corresponds to a legitimate geometry, and thus the process of selecting the base points is not modified.

A typical situation when both the nominal and perturbed values are positive and the lower bound is zero is treated by the modified approach. Thus, for any parameter value within the first interval including zero the responses are obtained by extrapolation from the results at the first two nonzero grid points.

# The Impact of Parameter Bounds on Interpolation

The *Lower* and *Upper* parameter *Bounds* in the *More* dialog box in the parameter definition area of the Optimization Parameters Editor serve a dual purpose.

In optimization, the bounds establish the limits of the exploration region. It is advantageous to limit the exploration region if one has the knowledge to do it. This is similar to a good selection of the starting point. The closer we are to the solution and the smaller the exploration region the faster the optimization process might be. This translates directly to the time required for finding the optimal solution. With time consuming EM simulations in the background, these advantages cannot be underestimated. Fortunately, a designer's own experience may help in setting these quantities. The importance of reasonably setting the lower and upper bounds on the optimization variables is particularly evident in the case of random optimizers. By exploring vast areas far away from the solution, a significant computational effort may be wasted.

In the case of EM optimization where the variables are geometrical dimensions the bounds play another very important role. They are effective in preserving the physical meaning of the layout. By the term "preserving the physical meaning" we mean primarily that the topology of the layout remains unchanged within the parameter bounds. Varying the parameter values should not create gaps or overlaps. Any such creation would inevitably lead to discontinuities in the responses. Then, interpolation would not only be misleading to the optimizer. It may also be highly inaccurate.

The term *preserving the physical meaning* also means that generation of extreme layouts can be avoided. As an example, consider varying the length of a rectangle to the extent that the length becomes several orders of magnitude larger than the width.

If the interpolation feature is enabled, none of the base points is generated outside the parameter bounds. The largest base point can coincide with the upper bound, if the upper bound is on the grid. Similarly, the smallest base point can coincide with lower bound, if the lower bound is on the grid. The only exception to these rules is if

the bound is exactly zero. The special handling of zero parameter values, if applicable, takes precedence (see ).

The optimizer will not generate parameter values outside the bounds. However, it may generate points very close to the bounds. If the bound is not on the grid then for any value between the last grid point and the bound (inclusive) the results are generated by extrapolation. Similarly, extrapolation applies to any parameter value outside the bounds at which the user may choose to perform simulation.

# "No interpolation" Option

You can choose *No interpolation* under *Interpolation Type* in the Optimization Control Editor. Selecting No interpolation disables the feature and every single point (set of parameter values) requested by the optimizer will be submitted for Momentum simulation. The results are stored in the database regardless of whether the point is on or off the grid.

It is important to remember that, in order to estimate response gradients, Momentum will additionally be invoked for a number of points at every iteration of optimization.

# Interpolation in Gradient Evaluation

Momentum Optimization estimates the gradients by extracting the difference in a response at slightly different parameter values. If the field solver is invoked to simulate two structures with slightly different dimensions, the difference in the response may not be reliably extracted due to inherent numeric noise.

Numerical stability in gradient evaluation comes as an important benefit of interpolation. The interpolating function is well defined since it is based on more significant differences in parameter values. The gradient is calculated from the interpolating function. And, it is a true gradient of the function that is actually seen by the optimizer.

# Types of Interpolation

The *Interpolation Type* entry in the Optimization Control Editor allows you to choose the interpolation algorithm. This, in turn, has an impact on how many and which neighboring on-grid points are selected as the base points.

## Linear Interpolation

Linear interpolation at an off-grid point requires EM analyses at $n + 1$ base points, where n is the number of parameters with off-grid values. The placement of the grid points for $n = 2$ is illustrated in Figure E-1.
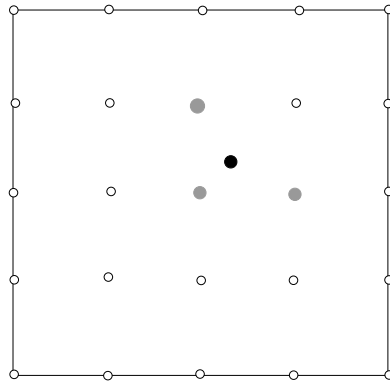


Figure E-1. The Base Points (Shaded) Required for Linear Interpolation at an Off-Grid Point (Solid)

Linear interpolation is adequate when the grid size is sufficiently small such that a linear function can approximate the response of interest within the interpolation interval with a reasonable accuracy.

## Quadratic Interpolation

Quadratic interpolation generally provides more accurate results than linear interpolation, at the expense of increased computational effort. Quadratic interpolation at an off-grid point requires EM analyses at $2n + 1$ base points, where n is the number of parameters with off-grid values. The placement of the base points for quadratic interpolation for $n = 2$ is illustrated in Figure E-2.
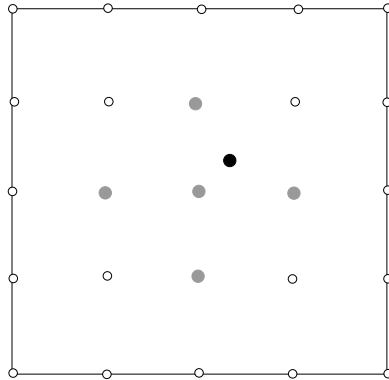


Figure E-2. The Base Points (Shaded) Required for Quadratic Interpolation at an Off-Grid Point (Solid)

# Appendix F: Momentum Optimization - Layout Parameterization

Momentum Optimization captures the user-defined design parameters movements by comparing each perturbed design with the nominal design. The layout parameterization is accomplished by determining a mapping, which links the parameter values with the coordinate of all the vertices. It is normally assumed that if one (any one) parameter is varied then each vertex moves along a straight line, or it does not move. For simultaneous changes of more than one parameter the resulting movements of the vertices are defined by superposition.

---

**Note** Rotation parameters define inherent non-linear vertex trajectories. Therefore such parameters must be carefully defined to avoid inconsistencies. For instance, it is advised to force the rotation parameters to stay within the [nominal, perturbed] value range during optimization.

---

One should be careful when defining the optimization parameters, the nominal and perturbed designs and their bounds. The optimizers within Momentum Optimization may fail to converge when they encounter large discontinuities, such as connectivity's being broken or newly created. The layout parameterization verifies that this does not happen for the parameter values defined by the nominal and perturbed design, but it does not do that for all the other values of the parameters.

# Guidelines for Preparing Perturbed Designs

The following are the general guidelines for setting up Momentum designs for layout parameterization.

- The number of components and artshapes must remain the same between the nominal and the perturbed designs.

- The number of ports must remain the same between the nominal and the perturbed designs.

- The number of vertices of each object must remain the same. In other words, the basic topology of the structure must not change. For instance, you can change the size of a rectangle, but you cannot change a rectangle into a triangle. Be careful when changing arc lengths, if by any operation they are translated to polygons, the number of vertices may differ. Use the command *Edit > Modify > Arc/circle Resolution* to assure the same number of vertices along the outline.

- The connectivity must remain the same between the nominal and the perturbed designs. For instance, the following incremental change cannot be parameterized:
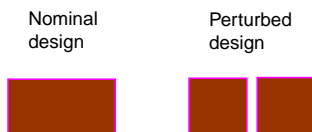


- Components and ports mapped to a metallization layer must have the same name in the nominal and perturbed designs.

---

**Note**   The perturbed designs do not have to include the setup of substrate, metallization and mesh.

---

# Vias Parameterization

Sometimes it might help to redefine some parameters. For instance, the figure below shows a non optimal solution for optimizing the width of a via and the width of the hole the via passes through:
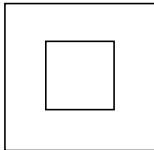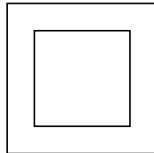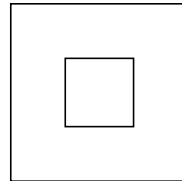


| Nominal | Perturbed_via | Perturbed_viahole |

Because for certain parameter values the via might be bigger than the hole, which would drastically change the connectivity of the circuit. For that reason a preferred parameterization is:
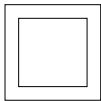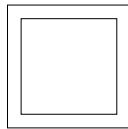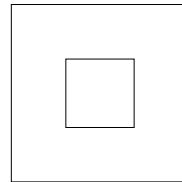


| Nominal | Perturbed_via | Perturbed_viahole |

This parameterization defines the width of the via and the distance between the via and the hole as the parameters. Constraining this distance to be bigger than the nominal value will eliminate connectivity changes.

# Appendix G: Command Reference

## Momentum Menu

Layout Window

_____

## Simulation Mode

### Enable RF Mode / Disable RF Mode

Toggles Momentum's simulation mode setting between RF mode and microwave mode. In each case, the Layout window's Momentum menu label indicates the current mode.

## Substrate Menu

### Open...

Provides you with a choice to open a predefined, supplied substrate (*Yes*) or a project specific user defined substrate (*No*).

### Save

Saves the substrate and defined metallization for the open design.

### Save As...

Saves the substrate and defined metallization by a name of your choice.

### Delete...

Deletes the substrates chosen.

### Create/Modify...

Creates a substrate/metallization definition or modifies an existing one.

### Update From Schematic

If the substrate used in a Momentum layout is defined in a schematic (for example, with the MSUB component) and the substrate on the schematic is modified, you must update the substrate information in the layout by choosing *Momentum > Substrate > Update From Schematic*.

### Precompute...

- *Substrate > Precompute* - Computes the substrate function over a designated range.

- *Mesh > Precompute* - Specifies the frequency for which the mesh will be precomputed. The default value in the Global Mesh Setup Panel or the Precompute Substrate dialog box is used if no value is specified here. You can set the frequency to any value within the range of the substrate definition.

**Summary...**

Displays the summary of a Momentum process:

- *Substrate > Summary* - Displays a summary of the substrate processes which have occurred.

- *Mesh > Summary* - Provides a count of the cell types. A count of the resources used to compute the mesh is also provided. Every mesh computation completed is reported.

- *Simulation > Summary* - Provides a summary of simulation data. Details about mesh reduction and quasi-static calculations are included when using the RF mode.

- *Optimization > Summary* - Displays a copy of the Optimization window data.

## Port Editor...

Defines the Momentum specific characteristics of the ports in the layout. The port editor cannot be activated until after the substrate dielectric layers and the circuit metallization have been defined.

## Box-Waveguide Menu

### Add Box...

A box can be added onto a substrate that is between an upper and a lower ground plane.

### Delete Box

Click and the previously defined box will be deleted.

### Add Waveguide...

A waveguide can be added onto a substrate that is between an upper and a lower ground plane. A waveguide can be defined parallel to either the X-axis or the Y-axis.

### Delete Waveguide

Click and the previously defined waveguide will be deleted.

## Mesh Menu

### Setup...

Defines the parameters to be used in computing a mesh.

### Precompute...

Specifies the frequency for which the mesh will be precomputed. The default value in the Global Mesh Setup Panel or the Precompute Substrate dialog box is used if no value is specified here. You can set the frequency to any value within the range of the substrate definition.

### Summary...

Provides a count of the cell types. A count of the resources used to compute the mesh is also provided. Every mesh computation completed is reported.

### Clear

Erases the mesh that is displayed on the structure after the mesh is computed.

## Simulation Menu

### S-parameters...

Defines the parameters and performs a simulation.

### Summary...

Provides a summary of simulation data. Details about mesh reduction and quasi-static calculations are included when using the RF mode.

## Optimization Menu

### Parameters...

Specifies the layout parameters to be optimized. Opens any designs related to optimization. Displays the optimal values of specified parameters.

### Goals...

Specifies the goals of the optimization.

### Run...

Runs the optimization. Specifies the type of optimization method to be used.

### Summary...

Displays a copy of the Optimization window data.

## Post-Processing Menu

### Radiation Pattern...

Defines the properties for calculating a radiation pattern and performs the calculations.

### Visualization...

Opens the Momentum Visualization display tool for viewing simulation results in 3D.

## HFSS Menu

### Save Files for HFSS

Writes Momentum data to Agilent HFSS project files for import into Agilent HFSS. The files are saved in <project_name>/*mom_dsn*/<design_name>/*hphfss*, and contain the design geometry and the Momentum setup.

# Index