Getting Started with

Q3D Extractor®

A 2D Grounded Coplanar Waveguide Model

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
USA
Tel: (+1) 724-746-3304
Fax: (+1) 724-514-9494
General Information: AnsoftInfo@ansys.com
Technical Support: AnsoftTechSupport@ansys.com

November 2011
Inventory 0000000330

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

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

Edition	Date	Software Version
1	February 2008	8
2	April 2010	9
3	October 2010	10
4	November 2011	11

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this guide.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:

  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.

  - On-screen prompts and messages, names of options and text boxes, and menu commands.

  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Menu commands are often separated by the ">" symbol. For example, "Click **Draw>Cylinder**".

Alternate methods or tips are listed in the left margin in blue italic text.

- Italic type is used for the following:

  - Emphasis.

  - The titles of publications.

  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press **Shift+F1**" means to press the **Shift** key and the **F1** key at the same time.

Getting Help

ANSYS Technical Support

To contact ANSYS technical support staff in your geographical area, please log on to the ANSYS corporate website, https://www1.ansys.com. You can also contact your ANSYS account manager in order to obtain this information.

All ANSYS software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

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

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

Edition	Date	Software Version
1	February 2008	8
2	April 2010	9
3	October 2010	10
4	November 2011	11

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this guide.

Alternate methods or tips are listed in the left margin in blue italic text.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:

  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.

  - On-screen prompts and messages, names of options and text boxes, and menu commands.

  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Menu commands are often separated by the ">" symbol. For example, "Click **Draw>Cylinder**".
- Italic type is used for the following:

  - Emphasis.

  - The titles of publications.

  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press **Shift+F1**" means to press the **Shift** key and the **F1** key at the same time.

Getting Help

ANSYS Technical Support

To contact ANSYS technical support staff in your geographical area, please log on to the ANSYS corporate website, https://www1.ansys.com. You can also contact your ANSYS account manager in order to obtain this information.

All ANSYS software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Table of Contents

4. Running the Analysis

5. Index

1 Introduction

This *Getting Started Guide* leads you step-by-step through creating, solving, and analyzing the results of a parameterized 2D model.

By following the steps in this guide, you will learn how to perform the following tasks in 2D Extractor:

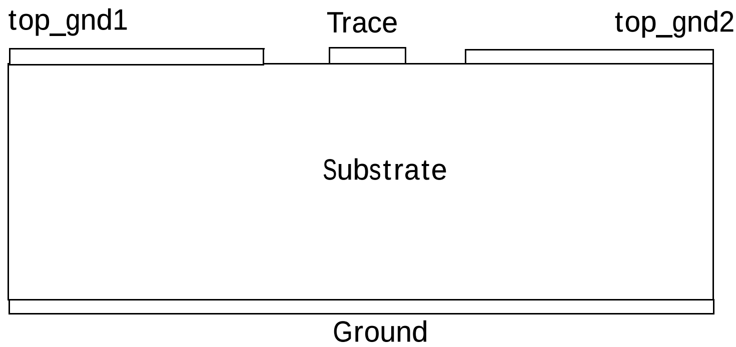
- Draw a geometric model.
- Modify a model's design parameters.
- Assign conductors.
- Reduce Matrices.
- Specify solution settings for a design.
- Validate a design's setup.
- Run a 2D Extractor simulation.
- Create a plot of results.

Estimated time to complete this guide: 60 minutes.

The Model

The model consists of a central rectangle and three thin rectangular copper conductors on top, and one long conductor on the bottom.

The model is shown below.



You will perform a reduce matrix operation, run a frequency sweep, parametric sweep, and plot the results.

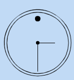
2

Creating the Model

In this chapter you will complete the following tasks:

- Save a new project.
- Set the drawing units for the design.
- Draw the model.
- Assign conductors.

Estimated time to complete this chapter: 20 minutes.



Open Q3D Extractor and Save a New Project

A project is a collection of one or more designs that is saved in a single file. A new project is automatically created when Q3D Extractor is launched. Open Q3D Extractor and save the default project under a new name.

If Q3D Extractor was already open and a default project is not listed in the project tree, add a new project:
Click File>New.

1 Double-click the **Q3D Extractor** icon on your desktop to launch Q3D Extractor.

A new project is listed in the project tree in the **Project Manager** window and is named **Project1** by default. Project definitions, such as material assignments, are stored under the project name.

2 Click **File**>**Save As**.

The **Save As** dialog box appears.

3 Locate and double-click the folder in which you want to save the project, such as C:\Ansoft\2D\Projects.

4 Type **cpw_gnd_gsg** in the **File name** box, and then click **Save**.

The project is saved as *cpw_gnd_gsg.q3dx* in the folder you selected.

5 Click **Project>Insert 2D Extractor Design** to add a design.

A new design called **2DExtractorDesign1** is added to the project tree.

Set the Drawing Units

Set the units of measurement for drawing the geometric model.

1 Click **Modeler>Units**.

The **Set Model Units** dialog box appears.

2 Verify that **mm** is selected in the **Select units** pull-down list, and click **OK**.

Draw the Substrate

1 Click **Draw>Rectangle**.

2 Specify the corner of the rectangle as (-0.5, 0):

a. Press **Tab** to move to the **X** box in the status bar.

b. Type **-0.5** in the **X** box, and then press **Tab** to move to the **Y** box.

c. Type **0.0** in the **Y** box, and then press **Enter**.

3 Specify the dimensions of the rectangle: Type (1, 0.2) in the **dX** and **dY** boxes, and press **Enter**.

A new object called **Rectangle1** is created.

In the **Properties** window, click the **Attribute** tab.

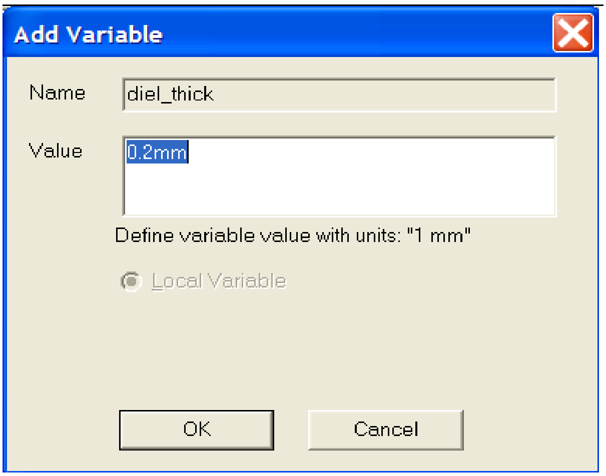
4 Rename **Rectangle1** to **substrate**:

a. Click on the **Attribute** tab to see all the properties associated with the box.

b. Type **Substrate** in the **Value** box in the **Name** row, and press **Enter**.

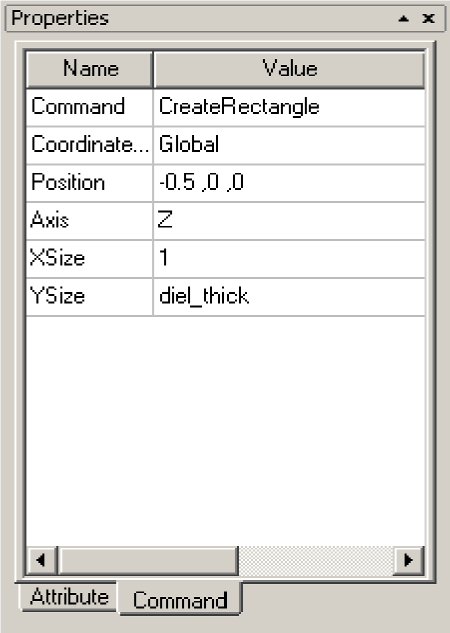
5 Click the **Command** tab. Define a variable for the y distance: Type **diel_thick** in the **Ysize** box, and press **Enter**.

The **Add Variable** dialog box appears.



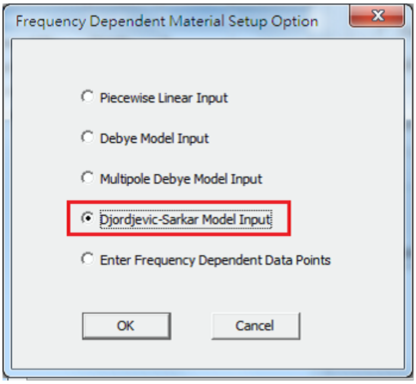
6 Click **OK**.

YSize is now defined by the variable **diel_thick**, as shown in the **Properties** window.



7 Right click **substrate** and select **Assign Material**. The **Select Definition** dialog box appears.

8 Add **silicon_lossy** to the material list.

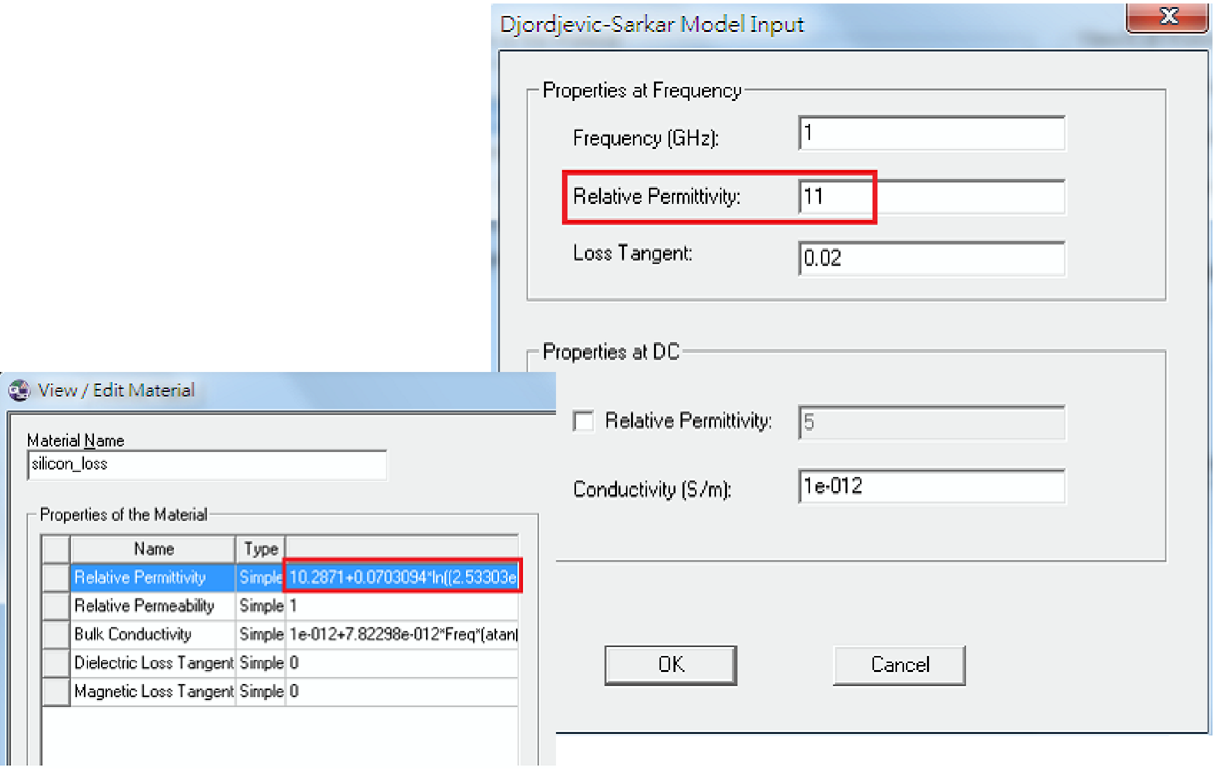


a. Click **Add Material** to open the **View/Edit Material** dialog box.

b. Enter **silicon_lossy** in the **Material Name** text box.

c. Click **Set Frequency Dependency** to open the **Frequency Dependent Material Setup Option** dialog box and choose **Djordjevic-Sarkar Model Input**, and click **OK**. Djordjevic-Sarkar is the recommended frequency dependent material model as it ensures causality for subsequent time-domain simulations.

The **Djordjevic-Sarkar Model Input** dialog box appears.



d. Enter **11** in the **Relative Permittivity** box, and **0.02** for **Loss Tangent**.

9 Click **OK** repeatedly to close all the open dialog boxes.

Draw the Ground

Now you will draw another rectangle at the base of **Substrate**.

1 Click **Draw>Rectangle**.

2 Specify the corner of the rectangle as (-0.5, 0):

a. Press **Tab** to move to the **X** box in the status bar.

b. Type **-0.5** in the **X** box, and then press **Tab** to move to the **Y** box.

c. Type **0** in the **Y** box, and then press **Enter**.

3 Specify the dimensions of the rectangle: Type (1,-0.01) in the **dX** and **dY** boxes, and press **Enter**.

A new object called **Rectangle1** is created.

In the **Properties** window, click the **Attribute** tab.

4 Rename **Rectangle1** to **Ground**:

a. Click on the **Attribute** tab to see all the properties associated with the box.

b. Type **Ground** in the **Value** box in the **Name** row, and press **Enter**.

5 If needed, view the entire model: Click **View>Fit All>Active Views**.

Create the Top Grounds

Now you will draw 2 ground planes:

1 Click **Draw>Rectangle**.

2 Specify the dimensions:

Position	**-0.5mm, 0.2mm**
XSize	**0.374**
YSize	**0.01**

3 Rename **Rectangle1** to **top_gnd1**:

a. Click on the **Attribute** tab to see all the properties associated with the box.

b. Type **top_gnd1** in the **Value** box in the **Name** row, and press **Enter**.

4 Select **top_gnd1**.

5 In the **Properties** window, click the **Command** tab.

6 Enter **diel_thick** for the y-coordinate (0.2) in the **Position** text box.

The starting Y point needs to be parameterized so that **top_gnd1** stays connected to the **Substrate** as its thickness changes.

7 Select **top_gnd1** and choose **Edit>Duplicate>Mirror**.

8 Click the origin, and then click any point on the x-axis

top_gnd1 is duplicated on the opposite side of the y-axis, and is placed in the Drawing area and is called **top_gnd1_1**.

9 Select **top_gnd1_1** and rename it to **top_gnd2**.

10 Click **OK.**

Create the Trace

Now you will draw a trace:

1 Click **Draw>Rectangle**.

2 Specify the dimensions:

Position	**-0.05mm, 0.2mm**
XSize	**0.1**
YSize	**0.01**

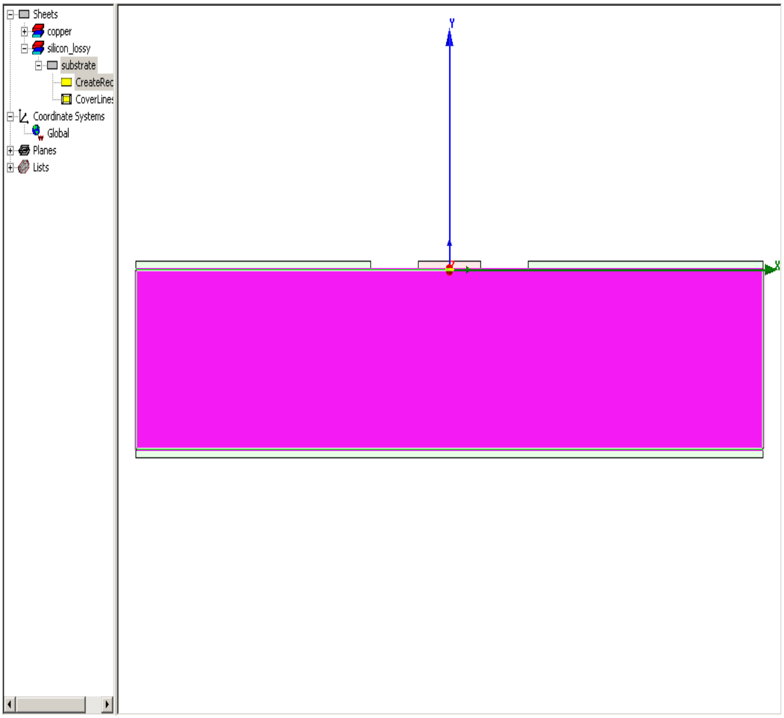
3 Rename **Rectangle1** to **trace**:

a. Click on the **Attribute** tab to see all the properties associated with the box.

b. Type **trace** in the **Value** box in the **Name** row, and press **Enter**.

You now have a new object called **trace**.

The project is shown below.



4 Select **trace**.

5 In the **Properties** window, click the **Command** tab.

6 Enter **diel_thick** for the y-coordinate (0.2) in the **Position** text box.

   The starting Y point needs to be parameterized so that **trace** stays connected to the **Substrate** as its thickness changes.

Assign Conductors

1 Define **trace** as a signal line:

a. Select **trace** and click **2D Extractor>Conductor>Assign>Signal Line**.

The **Signal Line** dialog box appears.

b. Leave the default value in all the fields, and click **OK**.

2 Define **Ground** as **Reference Ground**:

a. Select **Ground** and click **2D Extractor>Conductor>Assign>Reference Ground**.

The **Reference Ground** dialog box appears.

b. Leave the default value in all the fields, and click **OK**.

3 Define **top_gnd1** and **top_gnd2** as **Non Ideal Ground**:

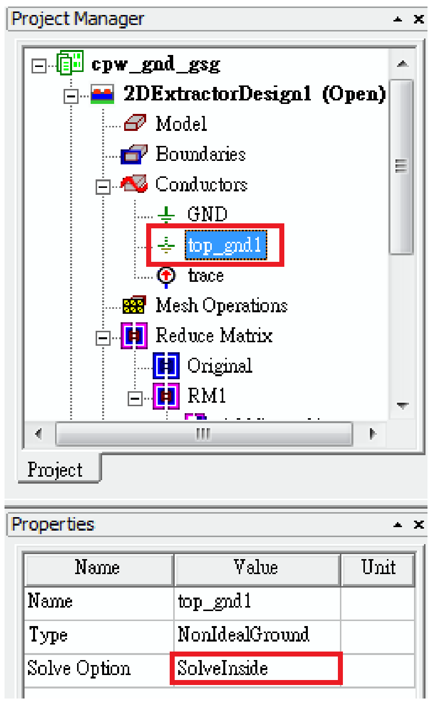
a. Select **top_gnd1** and press **CTRL** and select **top_gnd2**, and and click **2D Extractor>Conductor>Assign>Non Ideal Ground** to define the combined conductor as **Non Ideal Ground**.

The **Non Ideal Ground** dialog box appears.

b. Click the **Create Single Conductor** radio button from the **Choose an option for multiple selected objects** section. This joins the two coplanar grounds in parallel.

c. Leave the default value in all the other fields, and click **OK**.

You have completed the conductor assignment.



3

Setting Up an Analysis

In this chapter you will complete the following tasks:

- Add a solution setup.
- Add a frequency sweep.
- Perform matrix reduction.
- Validate the setup.

Estimated time to complete this chapter: 20 minutes.

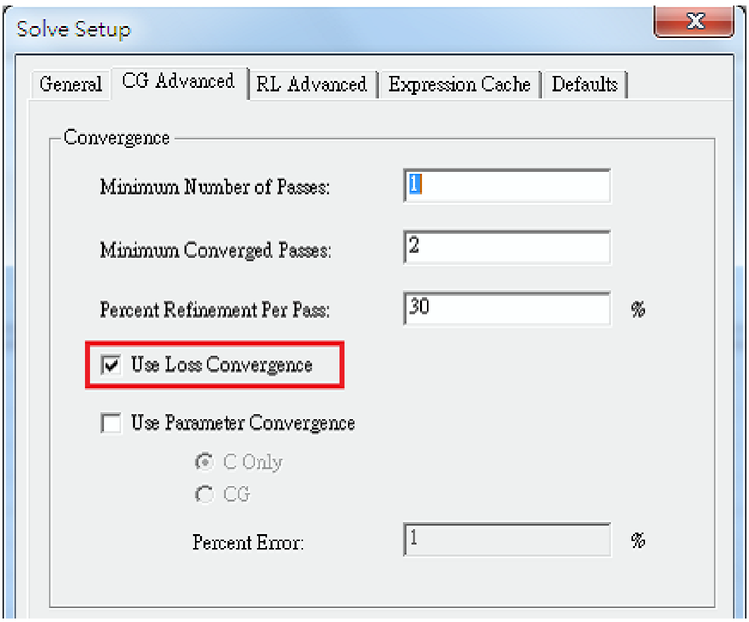
Add a Solution Setup

1. Click **2D Extractor**>**Analysis Setup>Add Solution Setup**.

   The **Solve Setup** dialog box appears with the **General** tab selected.
2. Verify that the **Solution Frequency** is set to **3 GHz**.
3. Verify that the **Admittance (Capacitance/Conductance)** box is selected.
4. Verify that the **Impedance (Resistance/Inductance)** box is selected.
5. Enter **20** in the **Maximum Number of Passes** text box.
6. Click the **CG Advanced** tab.
7. Type **2** in the **Minimum Converged Passes** box.
8. Click the **Use Loss Convergence** check box.
9. Click the **RL Advanced** tab, and repeat steps 7 to 8.
10. Click **OK**.

Using loss convergence provides a more accurate solution by converging based on Power Loss and Energy error.

The setup is listed in the project tree under **Analysis** as **Setup1**.



Add a Frequency Sweep

1 Click **2D Extractor**>**Analysis Setup>Add Frequency Sweep**.

The **Select** dialog box appears. Select **Setup1** and click **OK**.

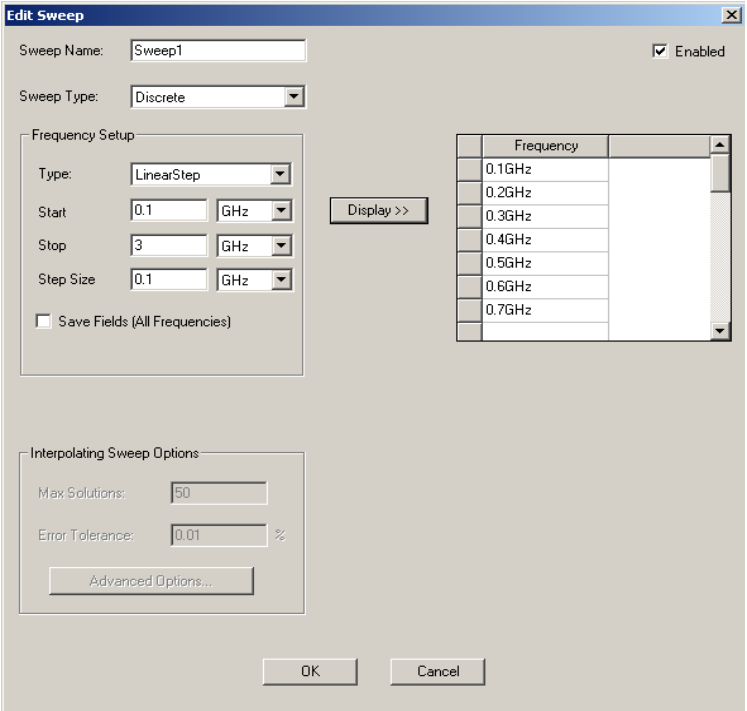
The **Edit Sweep** dialog box appears.

2 Type **0.1** in the **Start** box.

3 Type **3** in the **Stop** box.

4 Type **0.1** in the **Step Size** box.

5 Verify that **GHz** is selected as the units for all the frequencies.



6 Click **OK**.

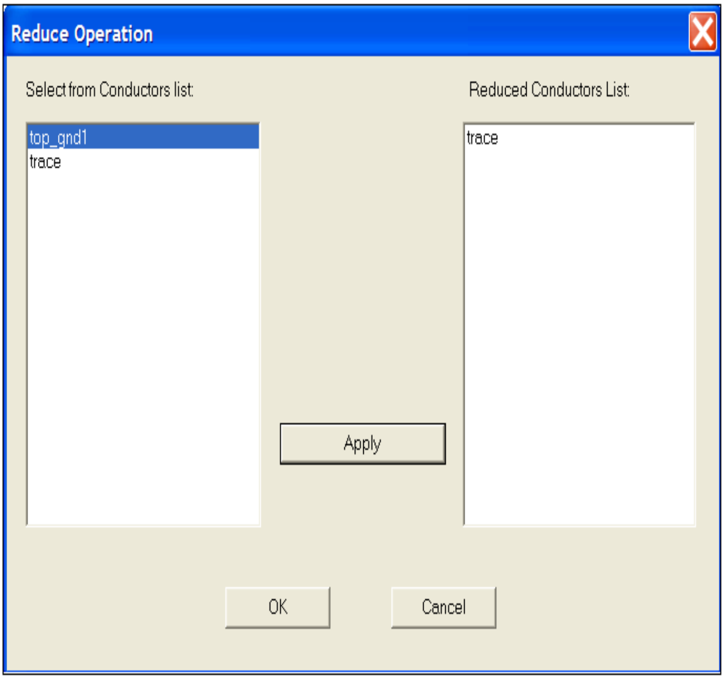
The frequency sweep is listed in the project tree, under **Setup1** as **Sweep1**.

Perform a Matrix Reduction

1 Click **2D Extractor>Reduce Matrix>Add Ground**.

The **Reduce Operation** dialog box appears.

2 Select **top_gnd1** from the **Select from conductors list** table, and click **Apply**.



3 Click **OK**.

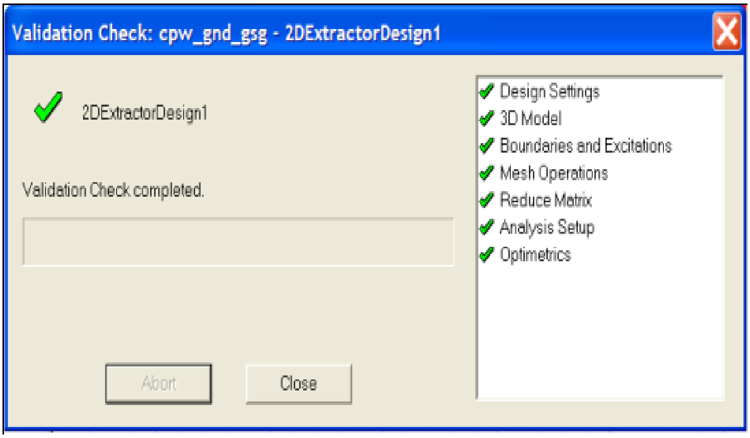
This ties the objects **top_gnd1**, **top_gnd2** and **Ground** in parallel and treats them as the reference ground.

Validate the Setup

You must verify that all the steps have been properly completed before you launch the field solvers.

1 Click **2D Extractor**>**Validation Check**.

2D Extractor checks the project setup, and the **Validation Check** dialog box appears.



2 Verify that you receive a green check mark for every operation. If something is wrong, you will receive a red X mark or a yellow warning. You must fix any error conditions before you proceed with a solution.

4

Running the Analysis

In this chapter you will complete the following tasks:

- Solve the Problem.
- View convergence and matrix data.
- Generate the Reporter Plots.
- Generate a Field Plots.
- Add a Parametric Sweep .
- Run the Parametric Analysis.

Estimated time to complete this chapter: 20 minutes.

Solve the Problem

If you have no errors from the validation check, you are ready to launch the field solvers.

You can run the analysis that you set up earlier.

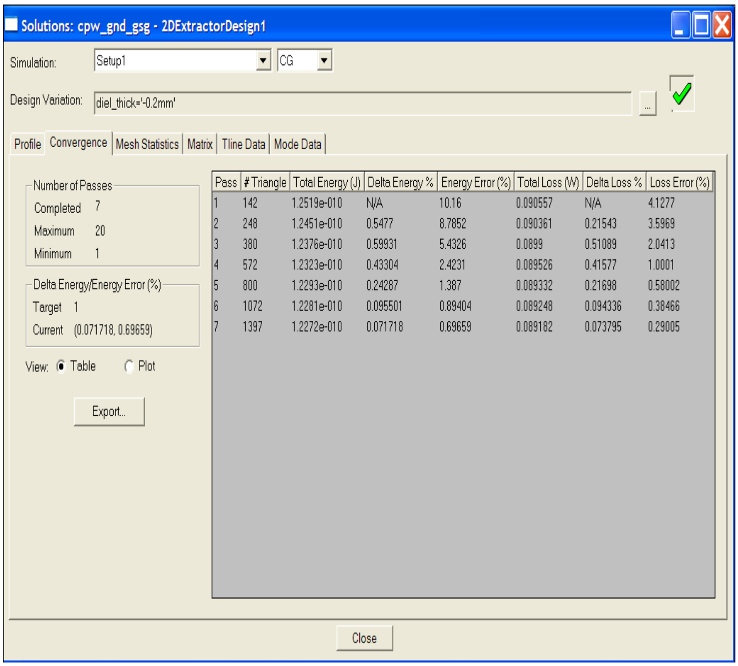
1 Right-click **Setup1**, and click **Analyze** from the shortcut menu.

   The **Progress** dialog box appears and displays the analysis.

2 View details about the ongoing solution: Right-click **Setup1**, and click **Convergence** from the shortcut menu.

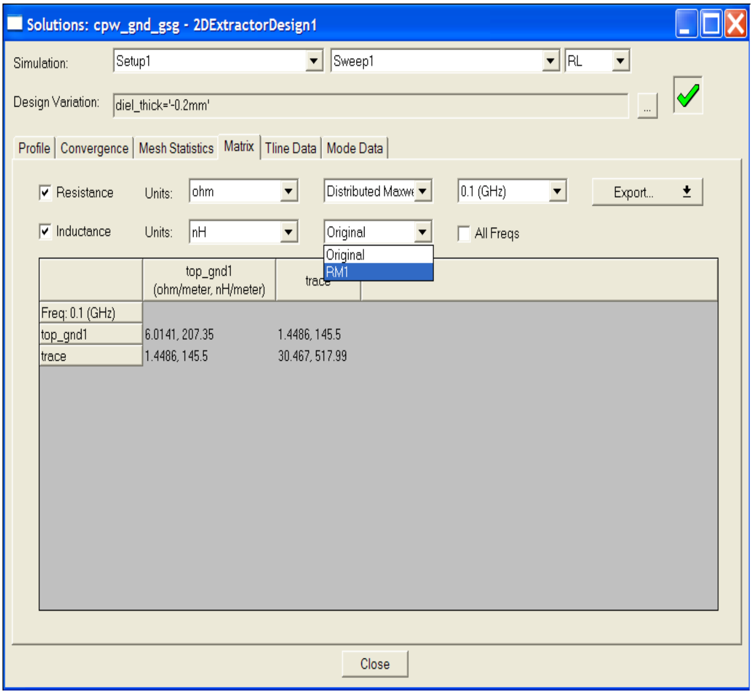
   The **Solutions** dialog box appears.

   This window shows how the mesh grows from one adaptive solution pass to the next and how much the solution changes (delta%) between passes.

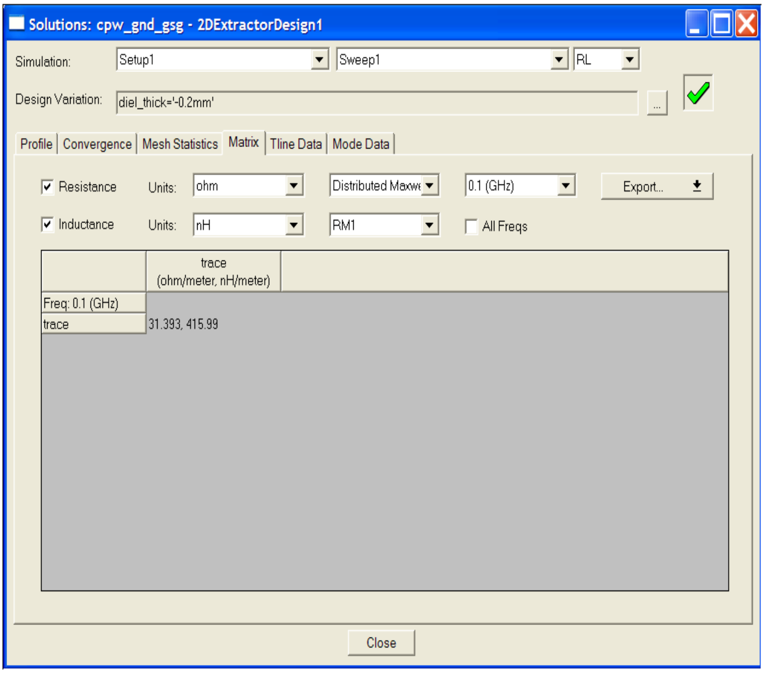


3 View convergence data for the impedance solution: select **RL** instead **CG**.

4 Click the **Matrix** tab, and then select the **RM1** instead of **Original** from the Matrix pull-down list.



The matrix data is displayed.



5 Click the **Profile** tab to see run-time profile information, such as the amount of CPU time or memory used in the solution.

6 Click the **Tline Data** tab to see data such as characteristic impedance and cross-talk coefficients.

7 Click the **Mode Data** to see propagation data - like velocity and attenuation.

8 Click **Close**.

Generate the Reporter Plots

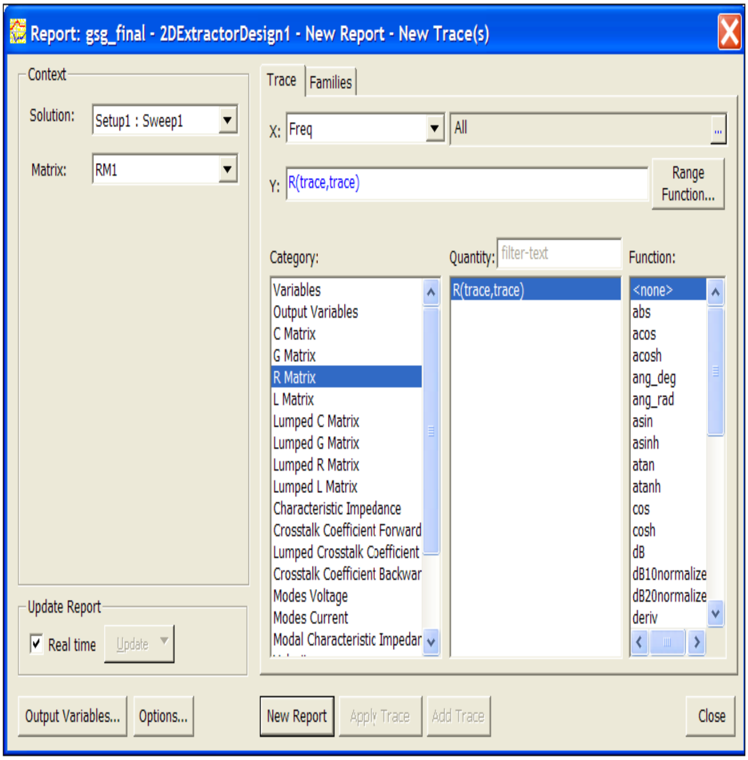
After running the simulation, you can view various plots.

We will first view the resistance vs. frequency plot.

1. Click **2D Extractor**>**Results>Create Matrix Report>Rectangular Plot**.

   The **Report** dialog box appears.

2. Verify that **Setup1:Sweep1** is selected in the **Solution** pull-down list.
3. Select **RM1** instead of **Original** from the Matrix pull-down list.
4. Select **R Matrix** from the **Category** list.
5. Select **R(trace,trace)** from the **Quantity** list.
6. Leave the default values for all the other fields, and click **New Report.**



A plot of R(trace, trace) vs. frequency is generated.

7 Click **Close** to close the **Report** dialog box.

You can change the X axis to a log scale.

8 Double-click on the X axis. The **Properties** dialog box appears.

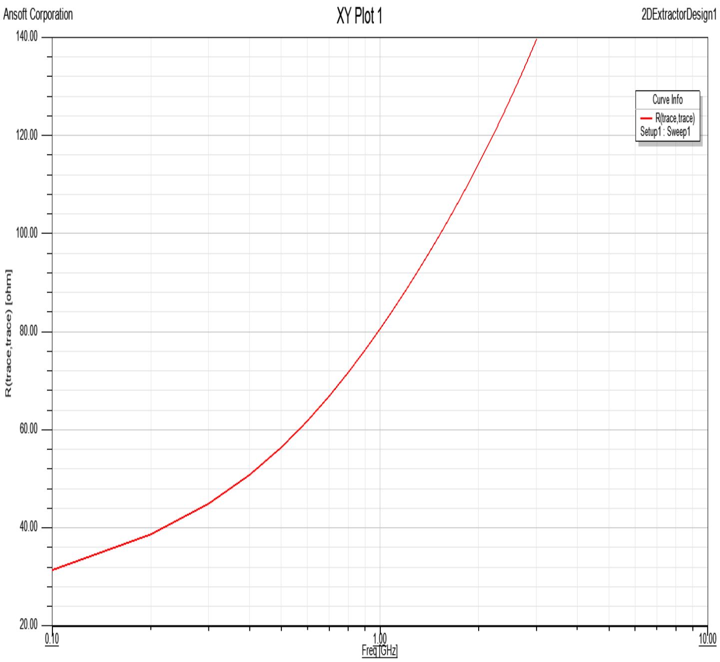
a. Click the **Scaling** tab.

b. In the **Value** text box in the **Axis Scaling** row, click on **Linear**.

A pop-up pull-down list appears.

c. Select **log**.

The Report gets modified.



9 Click **OK** to close the **Properties** dialog box.

Create Another Plot

Now you will generate a plot of conductance vs. the frequency.

1 Click **2D Extractor**>**Results>Create Matrix Report>Rectangular Plot**.

   The **Report** dialog box appears.

2 Verify that **Setup1:Sweep1** is selected in the **Solution** pull-down list.

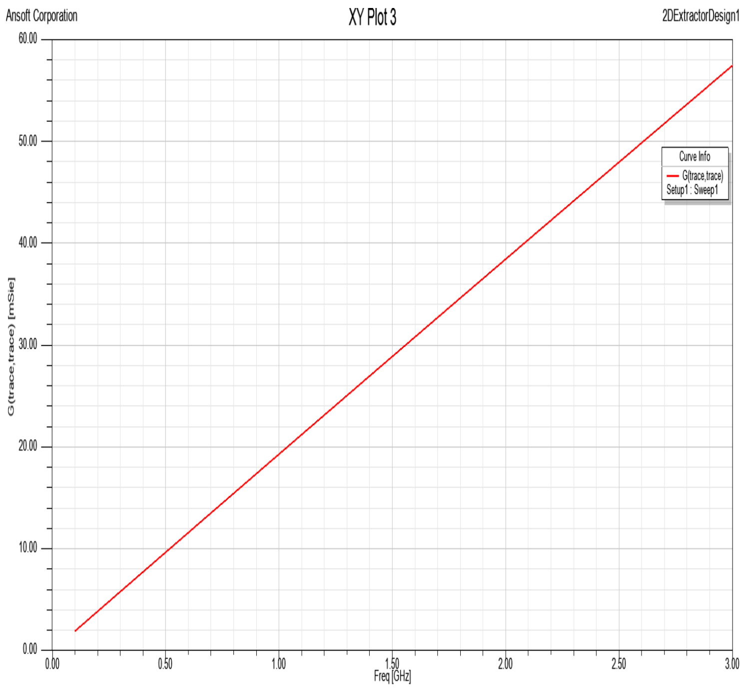
3 Select **RM1** instead of **Original** from the Matrix pull-down list.

4 Select **G Matrix** from the **Category** list.

5 Select **G(trace,trace)** from the **Quantity** list.

6 Leave the default values for all the other fields, and click **New Report.**

   A plot of G(trace, trace) vs. frequency is generated.



Generate a Field Plot

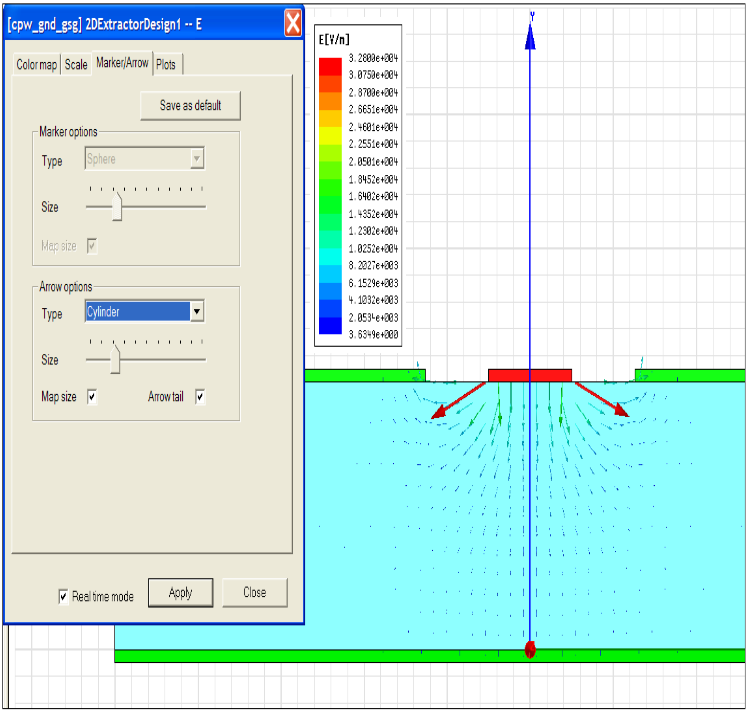
Field plots represent basic or derived quantities on surfaces of objects. You will plot Vector E for CG.

1. Click **Substrate** in the **3D Modeler** window.
2. Click **2D Extractor**>**Fields>CG Fields>E>VectorE**.

   The **Create Field Plot** dialog box appears.
3. Leave the default values unchanged, and click **Done.**

   The fields plot appears.

   You can double-click on the legend to make modifications. For example, to modify the arrow size in the vector plot: click on the **Marker/Arrow** tab and then drag the **Size** slider under **Arrow Options**.



Generate Another Field Plot

Now you will plot the current density for RL solution.

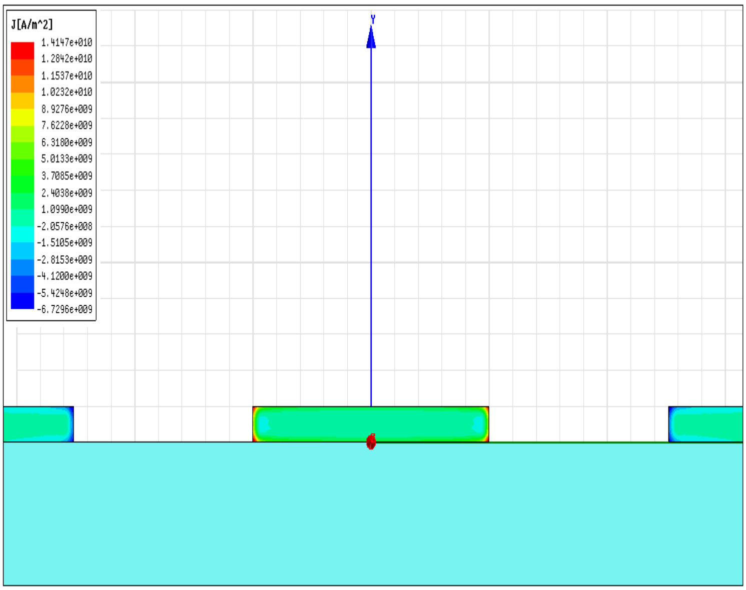
1 Click **Ground, top_gnd1, top_gnd2** and **Trace** in the **3D Modeler** window.

2 Click **2D Extractor>Fields>RL Fields>J>JrL**.

The **Create Field Plot** dialog box appears.

3 Leave the default values unchanged, and click **Done.**

The fields plot appears.

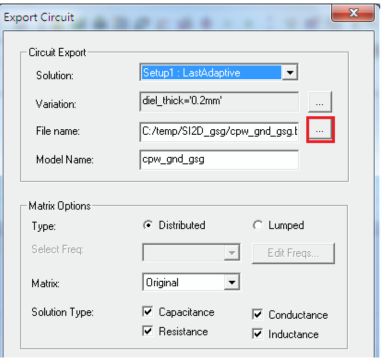


You can see that a large amount of current is returning through the coplanar grounds.

You can change the amount of current (RL) or voltage (CG) that is applied to a particular conductor, by clicking **2D Extractor>Fields>Edit Sources**. The **Edit Sources** dialog box appears. You can modify the values in the **CG** and **RL** tabs.

Export a Circuit Model

Now that you have the solution, you will export a SPICE model to simulate the effects of the trace on a signal that passes through it.



1 Right-click **Setup1** in the project tree, and click **Export Circuit** from the shortcut menu.

The **Export Circuit** dialog box appears.

2 Select **Setup1:Sweep1** in the **Solution** pull-down list.

3 Specify the circuit type for export:

a. Click the **... button** beside the **File name** box.

The **Choose Files** dialog box appears.

b. Select **Nexxim/HSPICE W Element (*.sp)** from the **Files of Type** pull-down list.

c. Leave the default values unchanged, and click **Open**.

Since we chose to export based on the sweep solution, a tabular W element table model (RLGC data at each frequency point) will be created.

4 Click **Export Circuit**.

By default, the file is exported as *cpw_gnd_gsg.sp.* A message window confirms the location and filename.

5 Click **OK**.

Currently, the model has only a single variation available for the nominal value diel_thick equals 0.2mm.

Next, you will set up a parametric analysis to sweep the variable over a range of values. Then, you can export different equivalent circuit models corresponding to the different values of these variables.

Add a Parametric Sweep

A parametric setup is made up of one or more variable sweep definitions. A variable sweep definition is a set of variable values within a range that Optimetrics drives 2D to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

1 Click **2D Extractor**>**Optimetrics Analysis>Add Parametric**.

The **Setup Sweep Analysis** dialog box appears.

a. Under the **Sweep Definitions** tab, click **Add**.

The **Add/Edit Sweep** dialog box appears.

b. Click **diel_thick** in the **Variable** pull-down list.

c. Verify that **Linear Step** is selected.

d. Specify the following values:

Parameter	Value
Start	0.2mm
Stop	0.5mm
Step Size	0.05 mm

e. Click **Add.**

f. Click **OK** to exit the **Add/Edit Sweep** dialog box and return to the **Setup Sweep Analysis** dialog box.

2 Click the **Table** tab to see all the values of **diel_thick** that will be simulated.

3 Click **OK**.

The model is simulated with various values in the specified range, including the start and stop values.

The sweep is listed in the project tree under **Optimetrics** as **ParametricSetup1**.

Now you will add an output calculation of the resistance matrix.

4 Right-click **ParametricSetup1**, and then click **Properties** from the shortcut menu.

The **Setup Sweep Analysis** dialog box re-appears.

5 Click the **Calculations** tab, and then click **Setup Calculations**.

The **Add/Edit Calculations** dialog box appears.

a. Verify that **Matrix** is selected in the **Report Type** pull-down list.

b. Verify that **Setup1:Last Adaptive** is selected in the **Solution** pull-down list.

c. Select **RM1** instead of **Original** from the Matrix pull-down list.

d. Click **R Matrix** in the **Category** table.

e. Select **R(trace,trace)** from the **Quantity** list.

f. Click **Add Calculation**.

g. Click **Done** to exit the **Add/Edit Calculations** dialog box and return to the **Setup Sweep Analysis** dialog box.

6 Click **OK**.

Disable the Frequency Sweep

For this project, you want to see only the nominal (3GHz) resistance at each parametric variation, therefore, you can disable the frequency sweep before beginning the parametric solution. This will also significantly decrease the parametric solution time.

Right-click **Sweep1**, and then click **Disable Sweep** from the shortcut menu.

Run the Parametric Analysis

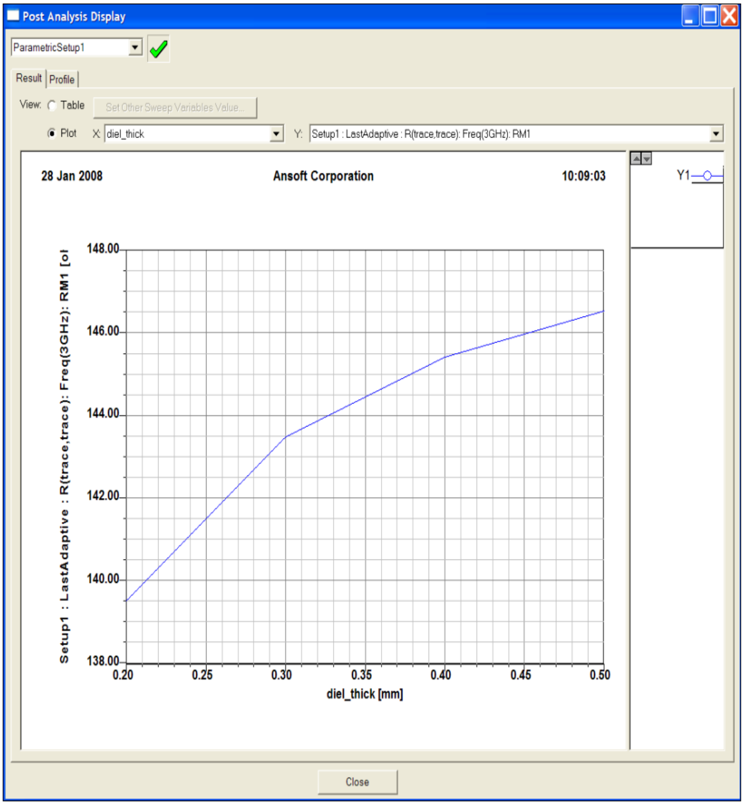
Now you can run the parametric analysis that was set up in the last step.

1 Right-click **ParametricSetup1**, and click **Analyze** from the shortcut menu.

  The **Progress** dialog box appears and displays the analysis.

2 Right-click again on **ParametricSetup1,** and click **View Analysis Result** from the shortcut menu.

  The **Post Analysis Display** dialog box appears, showing a table listing the values of **diel_thick** that have actually been solved.



3 Click the **Profile** tab to see how long it takes to solve each variation.

The parametric result shows how the resistance gets larger as the ground plane moves farther away from the trace. When the ground plane is farther away, more current returns in the nearby coplanar grounds (top_gnd1 and top_gnd2), which have a smaller effective cross-section.

Close the Project and Exit Q3D Extractor

Congratulations! You have successfully completed the *Getting Started with 2D Extractor* guide! You may close the project and exit the software.

1. Click **File>Save**.
2. Click **File>Close**.
3. Click **File>Exit**.

Index

R

S

T

U

V