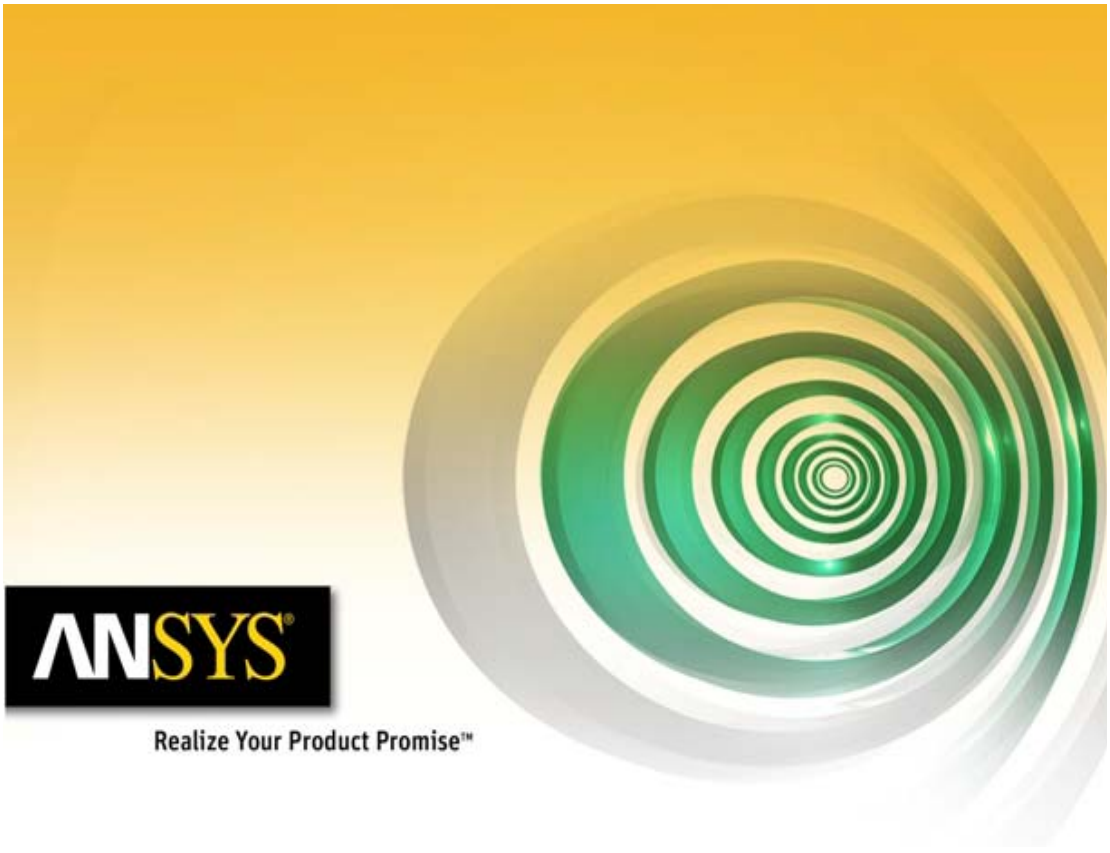


Getting Started with
Q3D Extractor®
A 3D PCB Via Model

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 0000000329

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

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.

Table of Contents

1 Introduction

This *Getting Started Guide* leads you step-by-step through creating, solving, and analyzing the results of a parameterized 3D model representing a via on a Printed Circuit Board.

By following the steps in this guide, you will learn how to perform the following tasks in Q3D:

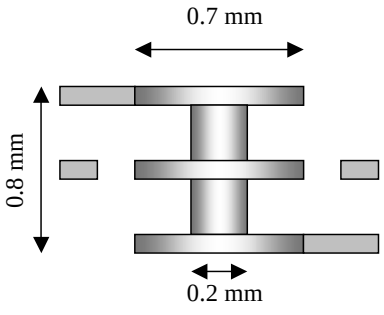
- Draw a geometric model.
- Set up vias and traces.
- Modify a model's design parameters.
- Assign variables to a model's design parameters.
- Specify solution settings for a design.
- Validate a design's setup.
- Run a simulation.
- Create a plot of results.

Estimated time to complete this guide: 60 minutes.

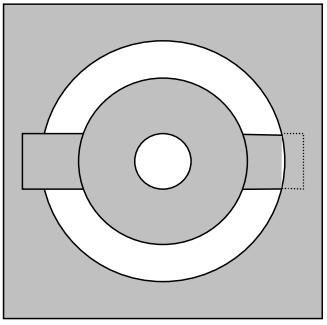
The Via Model

The via consists of a central cylinder and three thin cylindrical copper pads. It passes through a clearance hole (antipad) in a large copper ground plane. The entire structure is embedded in an FR-4 dielectric material.

The via model is shown below.



Side view



Top-down view

You will extract the parasitic inductance, resistance, and capacitance of this via. The drill hole radius and the pad radius should be made variable quantities so that you can conduct a parametric study of their effects.

2 Creating the Via Model

In this chapter you will complete the following tasks:

- Save a new project.
- Set the drawing units for the design.
- Draw the via.
- Draw the model.
- Create and assign variables.

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.

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.

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

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\Q3D\Projects.
4. Type **via_gsg.q3dx** in the **File name** box, and then click **Save**.

The project is saved in the folder you selected by the file name *via_gsg.q3dx*.

5. Rename the default design:
   a. Right-click **Q3DDesign1** in the project tree, and then click **Rename** on the shortcut menu.
   b. Type **ViaModel**, and then press **Enter**.

Set the Drawing Units

Set the units of measurement for drawing the geometric model.

1. Click **3D Modeler>Units**.

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

2. Verify that **mm** is selected in the **Select units** pull-down list.
3. Click **OK**.

Create the Via's Central Barrel

Create the first cylinder.

1. Click **Draw>Cylinder**.
2. Specify the radius:
   a. Click at the origin of the XYZ coordinate system.
   b. Press **Tab** to move to the **dX** box.
   c. Type **0.2** in the **dX** box, and then press **Tab** to move to the **dY** box.
   d. Type **0.0** in the **dY** box, and then press **Tab** to move to the **dZ** box.
   e. Type **0.0** in the **dZ** box and press **Enter**.

If you move the mouse now, you will see a circle in the xy plane that dynamically grows and shrinks as the cursor moves.

The radius of the cylinder becomes fixed, and the height of the cylinder changes dynamically as you move the cursor.

3. Specify the height of the cylinder:
   a. Move the mouse over the z-axis.
   b. Click when **dZ** displays **0.8**.

   The **Properties** window appears showing the **CreateCylinder** command. You can modify the cylinder's dimensions in this window.
4. Click on the **Attribute** tab to see all the properties associated with the cylinder.
5. Change the name of the cylinder to **Via:**
   a. Click the **Value** text box in the **Name** row.
   b. Type **Via**, and press **Enter**.
6. In the **Value** text box in the **Material** row, verify that the material type is **copper**.
7. Click **OK**.

Create the First Via Pad

Now you will create another cylinder to represent the via pad.

1. Click **Draw**>**Cylinder**.
2. Specify the radius as **0.7mm**, and the height as **0.025mm**. The start point will be at the origin.

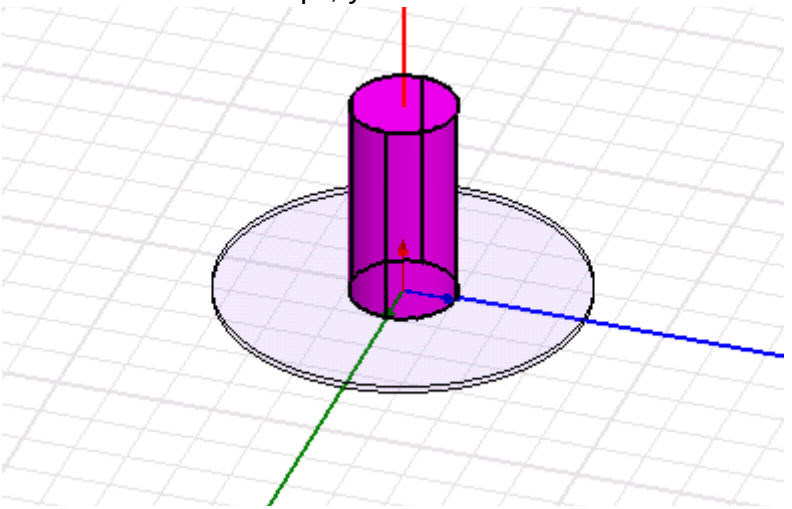
See Steps 2 through 3 above if you need help.

3. Click on the **Attribute** tab to see all the properties associated with the cylinder.
4. Change the name of the cylinder to **Pad1**.
5. In the **Value** text box in the **Material** row, verify that the

material type is **copper**.

6 Click **OK**.

After these steps, your model should look similar to:



Duplicate the First Via Pad

Now you will create two more via pads from the first one.

1 Click **Pad1** in the **3D Modeler** window.

2 Click **Edit**>**Duplicate>Along Line**.

3 Specify the distance between pads as **0.4mm**:

a. Click at the origin of the XYZ coordinate system.

b. Press **Tab** to move to the **dX** box.

c. Type **0** in the **dX** box, and then press **Tab** to move to the **dY** box.

d. Type **0** in the **dY** box, and then press **Tab** to move to the **dZ** box.

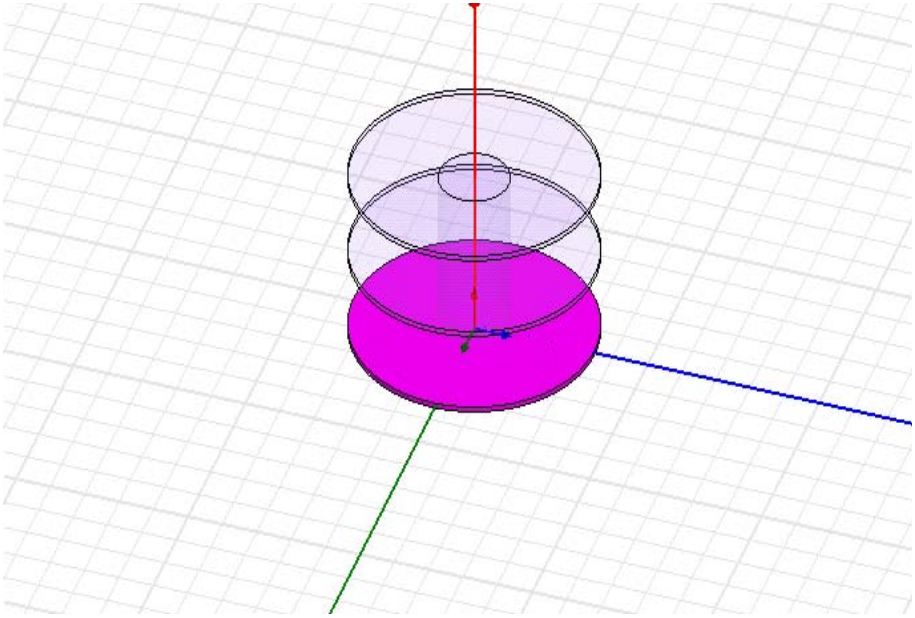
e. Type **0.4** in the **dZ** box, and press **Enter**.

The **Duplicate along line** dialog box appears.

4 Type **3** in the **Total number** box, and click **OK**.

Pad1 is duplicated 2 times. By default, the new objects are called **Pad1_1** and **Pad1_2**.

The **3D Modeler** shows the via model.



Create Geometric Variables

You will now define variables for the radius of the center barrel and the pads. Later, you will modify their values.

1. Click **Q3D Extractor>Design Properties**.

   The **Properties** dialog box appears, with no data listed.
2. Click **Add**.

   The **Add Property** dialog box appears.
3. Type **viarad** in the **Name** box.
4. Type **0.2mm** in the **Value** box.
5. Click **OK**.

   The **Properties** dialog box is updated.
6. Click **Add** again.
7. Do the following in the **Add Property** dialog box:
   a. Type **padrad** in the **Name** box.
   b. Type **0.7mm** in the **Value** box.
   c. Click **OK**.
8. Click **OK** to return to the model.

Assign a Variable to the Center Barrel

After you have defined the variables, you will need to assign them. You previously specified the via's radius to be 0.2mm; now replace that value by the **viarad** variable.

1. Click **Via** in the **3D Modeler** window.
2. In the **Properties** window, click the **Command** tab.
3. Assign the variable **viarad** to the radius:
   a. Click the **Value** box in the **Radius** row.
   b. Type **viarad**, and press **Enter**.

The **Properties** dialog box is updated, but the **3D Modeler** window remains unchanged.

Assign a Variable to the Pads

You previously specified the pad's radius to be 0.7mm; now replace that value by the **padrad** variable.

1. Click **Pad1** in the project tree**.**
2. Click the + icon in front of **Pad1** to expand it.
3. Click **CreateCylinder**.
4. In the **Properties** window, click the **Command** tab.
5. Assign the variable **padrad** to the radius:
   a. Click the **Value** box in the **Radius** row.
   b. Type **padrad**, and press **Enter**.

   The **Properties** dialog box is updated, but the **3D Modeler** window remains unchanged. The **3D Modeler** window is updated only when the value of the variable changes.
6. Click **Q3D Extractor>Design Properties**.

   The **Properties** dialog box appears.
7. Type **0.5 mm** in the **Value** box, in the **padrad** row.
8. Click **OK**.

   The solid model is updated. The radius changes for all the pads, since **Pad1_1** and **Pad1_2** inherit the changes from **Pad1**.
9. Return **padrad** to **0.7mm**, and verify that **viarad** is set to **0.2mm.**

Draw the Trace Stubs

You will now add some trace stubs to the via model. Draw stubs to represent traces connecting to the top and bottom of the via.

1. Click **Draw>Box**.
2. Specify the base corner of the box as (-0.25, 0.4, 0):
   a. Press **Tab** to move to the **X** box in the status bar.
   b. Type **-0.25** in the **X** box, and then press **Tab** to move to the **Y** box.
   c. Type **0.4** in the **Y** box, and then press **Tab**.
   d. Type **0** in the **Z** box**,** and then press **Enter**.
3. Specify the dimensions of the box: Type (**0.5, 1.2, 0.025**) in the **dX**, **dY**, and **dZ** boxes, and then press **Enter**.

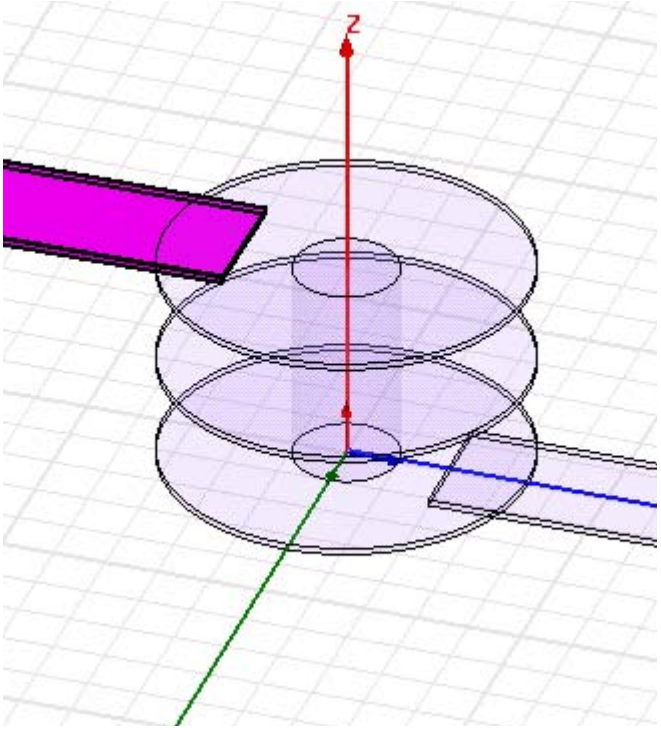
   A new object called **Box1** is created.

   The **Properties** dialog box appears. Click the **Attribute** tab and verify that the material type is **copper**. Click **OK**.
4. Repeat steps 1 through 3 with the following values to create another box, **Box2**:

   X= -0.25, Y=-0.4, Z=0.8

   dX= 0.5, dY=-1.2, dZ= 0.025.

The **3D Modeler** window is shown below.



The trace stubs created here partially overlap the pads. The via's center barrel also overlaps the pad objects.

Partially overlapping objects can create ambiguity about the type of material to be used in the overlap region. In the special case of one object being completely contained within another, it is assumed that the material of the smaller object applies within the overlap region. This is the only case where overlapping objects are allowed.

Unite Via Objects

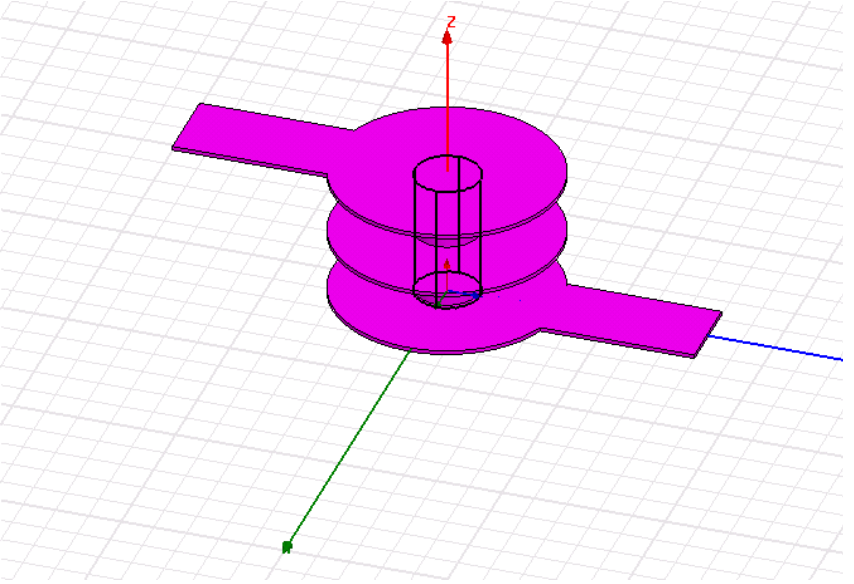
You need to eliminate the partial overlaps between the various objects in this model. Since all the objects are of the same material (copper), you can unite them into a single solid object.

1. Click **Edit**>**Select All**.
2. Click **3D Modeler**>**Boolean>Unite**.

   The list of objects is replaced by a single object named **Via**

that occupies the same volume of space as the original set of objects. The material type of **Via** is **copper**.

The **3D Modeler** window is shown below.



Create the Ground Plane

You will create the ground plane as a box.

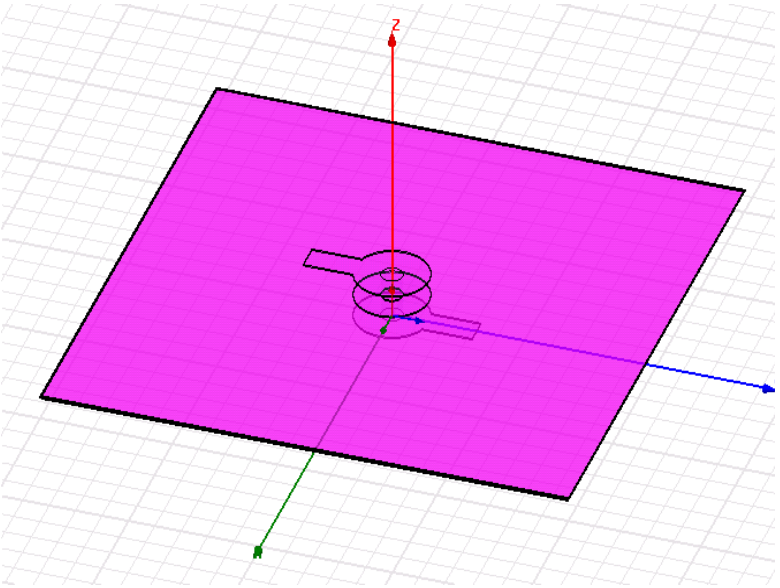
1. Click **Draw>Box**.
2. Specify the base corner of the box as (-5, -5, 0.4):
   a. Press **Tab** to move to the **X** box in the status bar.
   b. Type **-5** in the **X** box, and then press **Tab** to move to the **Y** box.
   c. Type **-5** in the **Y** box, and then press **Tab**.
   d. Type **0.4** in the **Z** box**,** and then press **Enter**.
3. Specify the dimensions of the box: Type (**10, 10, 0.025**) in the **dX**, **dY**, and **dZ** boxes, and then press **Enter**.

   A new object called **Box3** is created. Its material type is **Copper**.
4. Rename **Box3** to **GroundPlane**:
   a. Click on the **Attribute** tab to see all the properties associated with the box.
   b. Type **GroundPlane** in the **Value** box in the **Name** row, and press **Enter**.

5. View the entire model: Click **View>Fit All>Active Views**.
6. Make **GroundPlane** transparent to view the via:
   a. In the **Properties** window, click the **Value** box in the **Transparent** row.

      The **Set Transparency** dialog box appears.
   b. Type **0.5,** and press **OK**.

The **3D Modeler** window shows **GroundPlane** overlapping **Via**.



This will be corrected in the next step.

Create the Antipad

To prevent the via from shorting to the ground plane, create a circular hole (or antipad) in the plane.

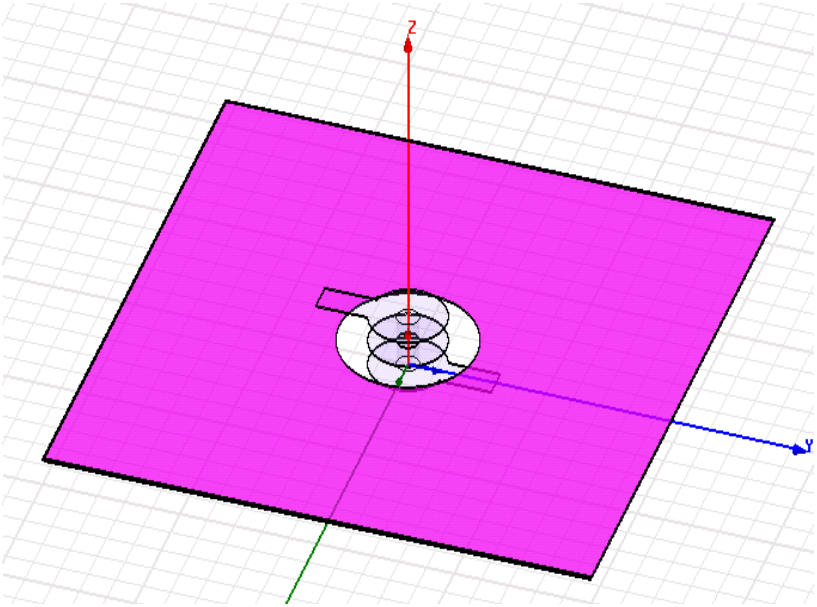
1. Click **Draw>Cylinder.**
2. Specify the base of the cylinder:
   a. Press **Tab** to move to the **X** box.
   b. Type **0.0** in the **X** box, and then press **Tab** to move to the **Y** box.
   c. Type **0.0** in the **Y** box, and then press **Tab** to move to the **Z** box.
   d. Type **0.4** in the **Z** box and press **Enter**.
3. Specify the dimensions: Type (**1.25, 0, 0.025**) in the **dX**, **dY**, and **dZ** boxes, and then press **Enter**.

The **Properties** window appears.

4. Click on the **Attribute** tab to see all the properties associated with the cylinder.
5. Change the name of the cylinder to **Hole:**
   a. Click the **Value** box in the **Name** row.
   b. Type **Hole**, and press **Enter**.
   c. Click **OK**.
6. Click both objects **GroundPlane** and **Hole**, to select them.
7. Click **3D Modeler**>**Boolean>Subtract**.

   The **Subtract** dialog box appears.
8. Verify that **GroundPlane** appears under **Blank Parts** and that **Hole** appears under **Tool Parts**.
9. Click **OK**.

A circular hole is created in **GroundPlane**.



Define the Background Material

You have now completed all of the 3D drawing operations for the metal objects in the problem. By default, Q3D Extractor uses “vacuum” as the background material. However, for this problem, you have to model the via in an FR-4 substrate, so the surrounding material needs to be changed.

There are two possible ways to change the background material:

- By explicitly drawing an object surrounding the conductors and assigning a new material to it. This option permits you to create several regions with different dielectric constants.
- By changing the background material. This option is less flexible but still adequate in this problem, so we will use this method.

1. Click **Q3D Extractor>Set Background Material.**

   The **Select Definition** dialog box appears.

2. Select **FR4_epoxy** from the list of materials**,** and click **OK.**

3 Setting Up the Simulation

In this chapter you will complete the following tasks:

- Set up sources and sinks for inductance and resistance calculations.
- Request the parameters (R, L, or C) for which to solve.
- Launch the field solver.
- View the results.

Estimated time to complete this chapter: 15 minutes.

Set Up Sources and Sinks

To set up the solution for resistance and inductance, you will need to first define source and sink terminals. Attached to each source terminal is an independent current source. Assume that current enters and leaves at the ends of the two trace stubs you created. You need to put a source at the end of the top stub, and a sink at the end of the bottom stub.

Define the Source

1. Click **View**> **Visibility.**

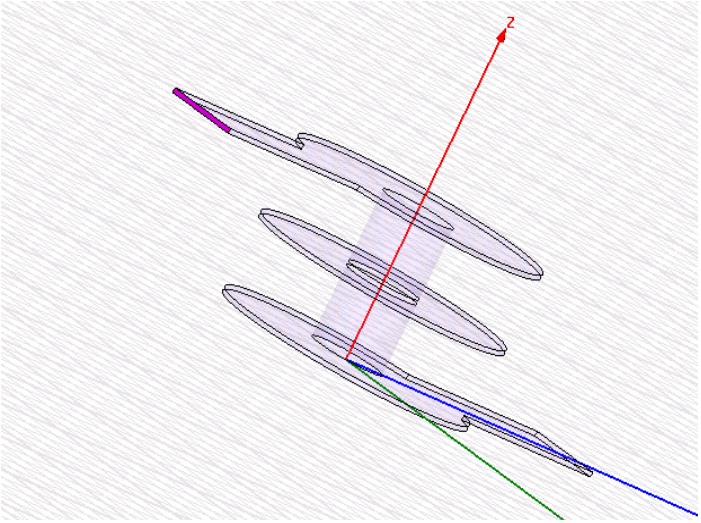
   The **Visibility** dialog box appears.
2. Clear the **Visibility** check box for **GroundPlane**, and click **Done**.
3. Click **View**>**Zoom In.**
4. Click **Via** in the **3D Modeler** window.
5. Click **View**>**Fit Selections>All Views.**

*You can skip intervening faces by pressing **B**.*

6. Rotate the model so that you are looking at the end of the top trace stub: Click **View**>**Rotate**, and then click-and-drag in the **3D Modeler** to spin the model around.
7. Right-click and then choose **Select Objects.** Click to select the top stub face.
8. Right-click in the **3D Modeler** window, and then click **Assign Excitation>Source** from the shortcut menu.

   The **Source** dialog box appears.

9 Leave the default name **Source1** unchanged, and click **OK**.



Assign the Sink

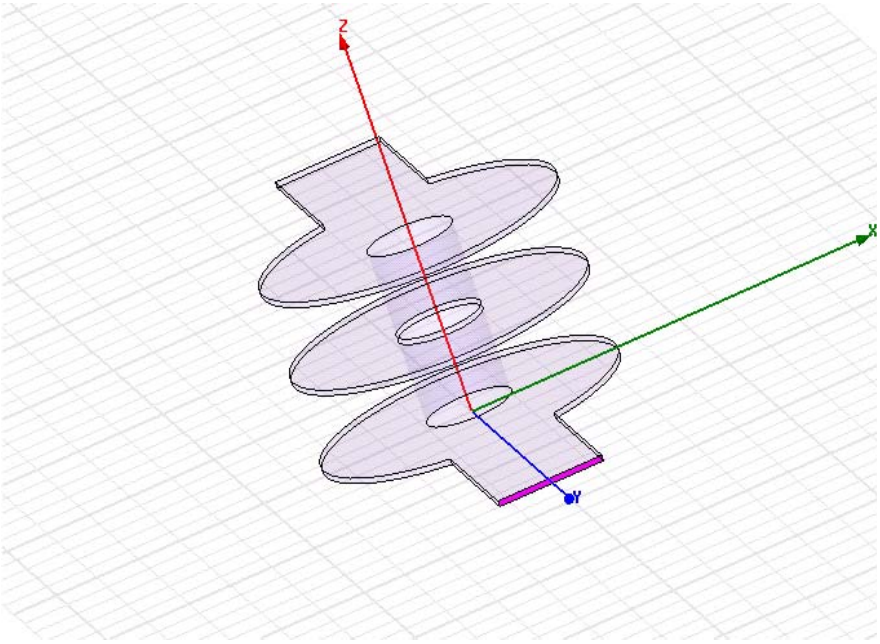
The sink terminal collects all of the current injected at the source terminals and allows it to flow out of the conductor back into the independent sources, completing the electrical circuit.

1 Select the face at the bottom stub. You may rotate the model again to view this stub.

2 Right-click in the **3D Modeler**, and then click **Assign Excitation>Sink** from the shortcut menu.

The **Sink** dialog box appears.

3 Leave the default name **Sink1** unchanged, and click **OK**.



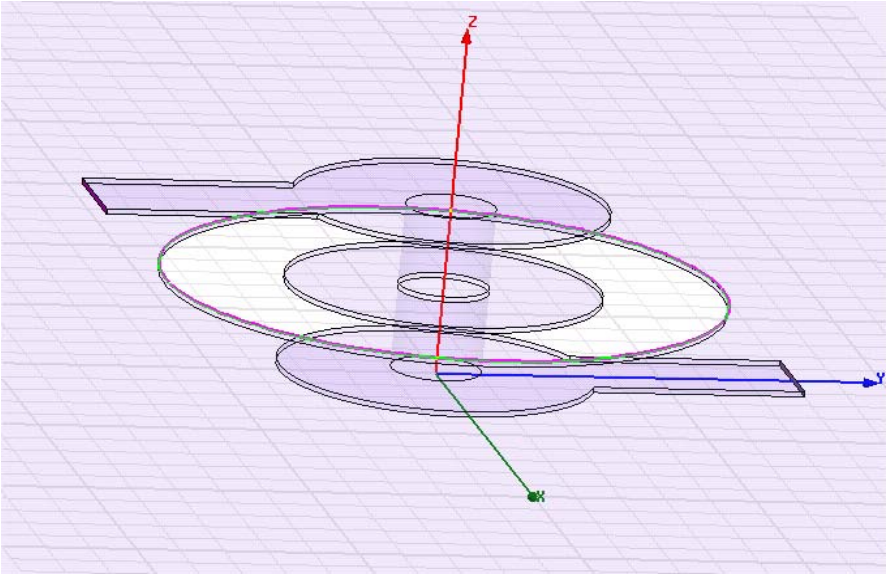
4 Restore the visibility of **GroundPlane:**

a. Click **View**>**Visibility.**

The **Visibility** dialog box appears.

b. Select the **Visibility** check box for **GroundPlane**, and click **Done**.

After assigning the source and sink, the **3D Modeler** looks as below:



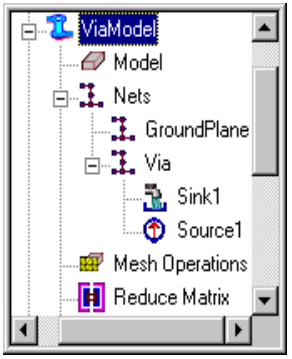
Identify the Nets

The model is almost ready for a field solution. The final step is to define the distinct nets in the problem.

To automatically create nets:
Right-click in the **3D Modeler**, and then click **Auto Identify Nets** from the shortcut menu.

Q3D identifies two nets - **Ground-Plane** and **Via**.

It also identifies that **Source1** and **Sink1** lie on the net named **Via,** and places them accordingly in the project tree.



Add a Solution Setup

Now start solving for the electrical parasitics of the via.

1. Right-click **Analysis** in the project tree, and then click **Add Solution Setup** from the shortcut menu.

   The **Solve Setup** dialog box appears.

2. Under the **General** tab, verify that **Capacitance/Conductance**, **DC Resistance/Inductance,** and **AC Resistance/Inductance** are checked.

3. Click **OK.**

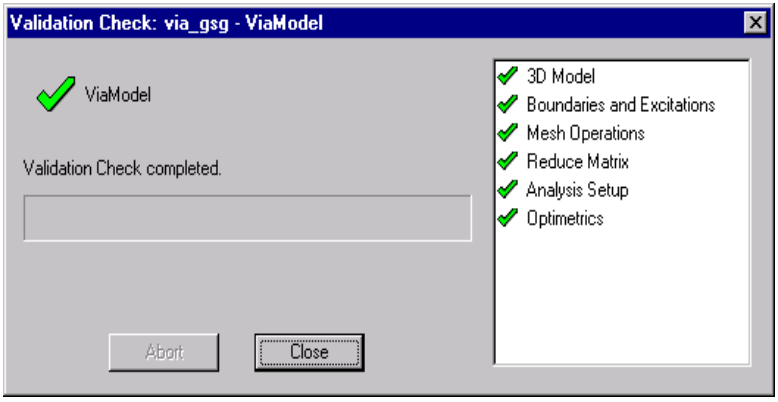
   Setup1 is added to the project tree under **Analysis**.

Validate the Setup

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

1. Click **Q3D Extractor**>**Validation Check**.

   Q3D 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.

Solve the Problem

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

1. In the project tree, click **Analysis** to expand it.

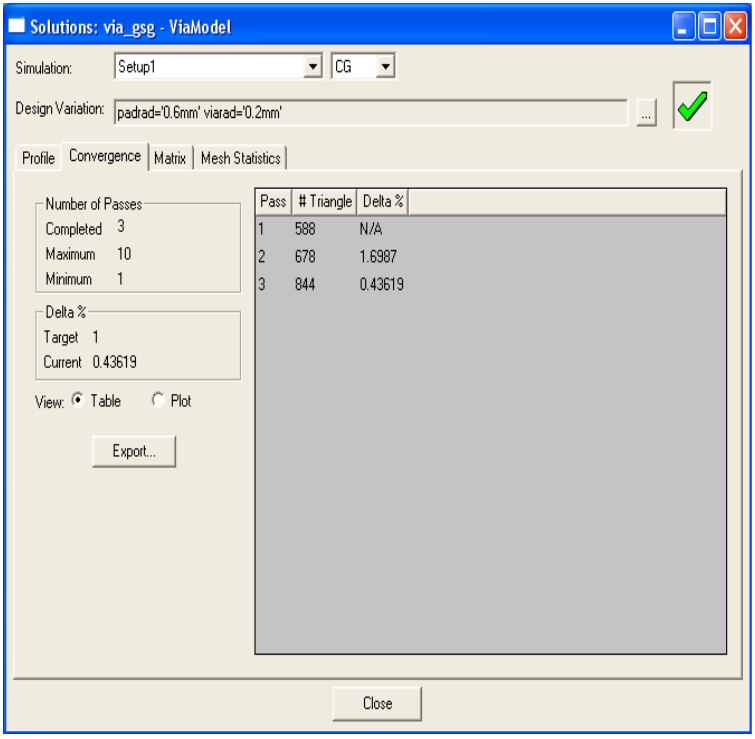
   Setup1 is listed.

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

   Q3D begins to mesh and solve the problem.

3. 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.

4. Click the **Matrix** tab to see the actual capacitance solution data.

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 **Close**.

Generate a Field Plot

Field plots represent basic or derived quantities on surfaces or objects. You will now generate a field plot on **Via.**

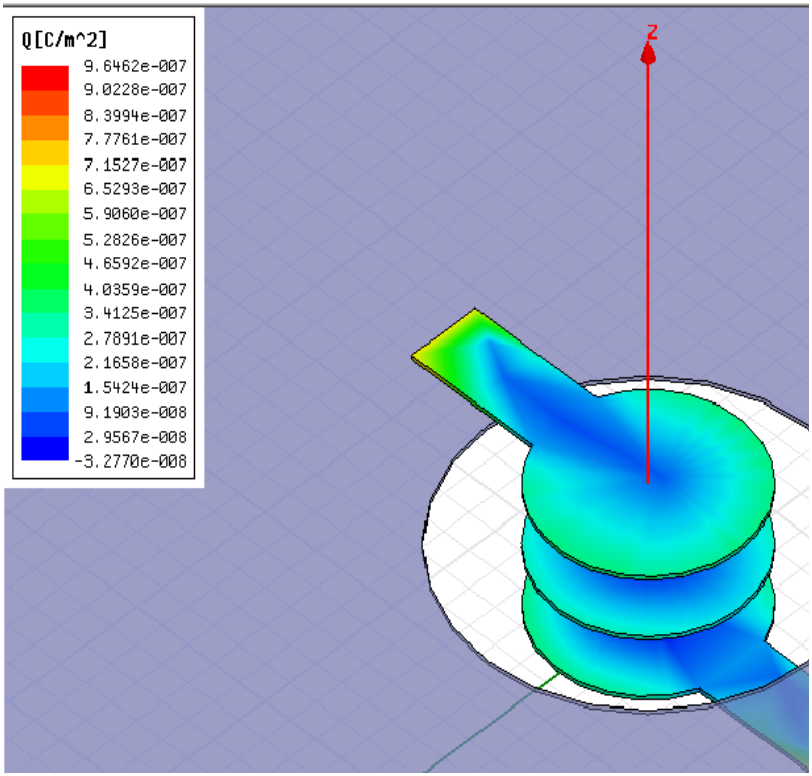
1 Click **Via** in the **3D Modeler** window.

2 Click **Q3D Extractor>Fields>C Fields>SmoothQ**.

The **Create Field Plot** window appears.

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

The resulting plot shows the charge when 1V is applied to **Via** and 0V is applied to **GroundPlane**.

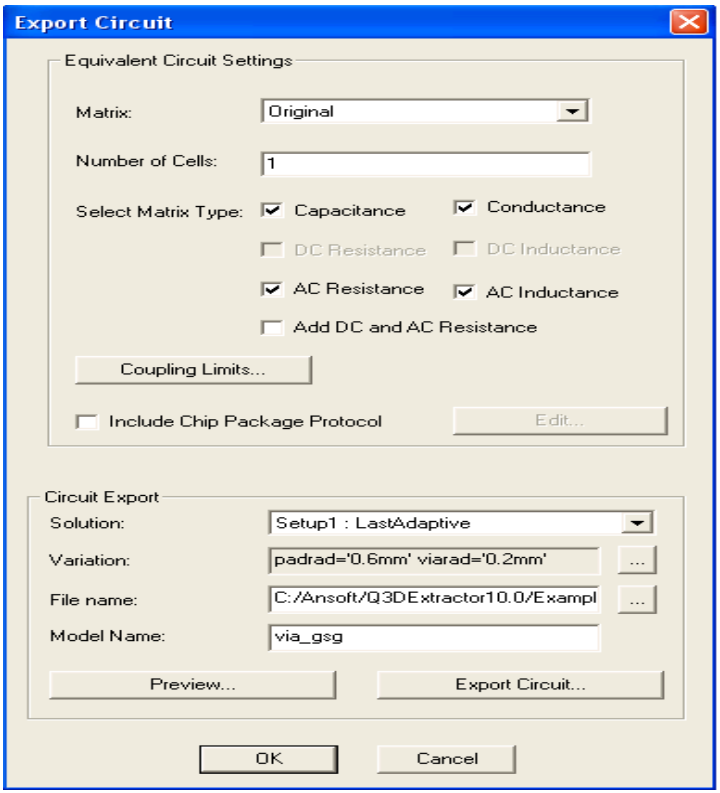


Export a Circuit Model

Now that you have the solution, you will export a SPICE model to simulate the effects of the via 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. Accept the default settings.
3. Click **Export Circuit**.

   By default, the file is exported as *via_gsg.cir.* A message window
   confirms the location and filename.
4. Click **OK**.

Currently, the model has only a single variation available for the nominal values padrad equals 0.7mm and viarad equals 0.2mm.

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

4 Setting Up a Parametric Analysis

In this chapter you will complete the following tasks:

- Add a parametric sweep.
- Run the parametric analysis.

Estimated time to complete this chapter: 10 minutes.

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 Q3D to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

1. Click **Q3D Extractor**>**Optimetrics Analysis>Add Parametric**.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.
3. Click **viarad** in the **Variable** pull-down list.
4. Verify that **Linear Step** is selected.
5. Specify the following values:

Start	**0.2mm**
Stop	**0.5mm**
Step Size	**0.05 mm**

6. Click **Add.**
7. Click **OK** to exit the **Add/Edit Sweep** dialog box and return to the **Setup Sweep Analysis** dialog box.
8. Click the **Table** tab to see all the values of **viarad** that will be simulated.
9. Click **OK**.

   Q3D Extractor simulates the model with various values in the specified range, including the start and stop values.

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

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 **viarad** that have actually been solved.

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

5 Comparing the Solutions

In this chapter you will complete the following tasks:

- Create a plot of the parametric sweep.
- Plot the results.
- Vary the pad radius to see its effect on capacitance.
- Close the project and exit Q3D Extractor.

Estimated time to complete this chapter: 15 minutes.

Plot the Results

1. Right-click **Results** in the project tree, and then click **Create Report**.

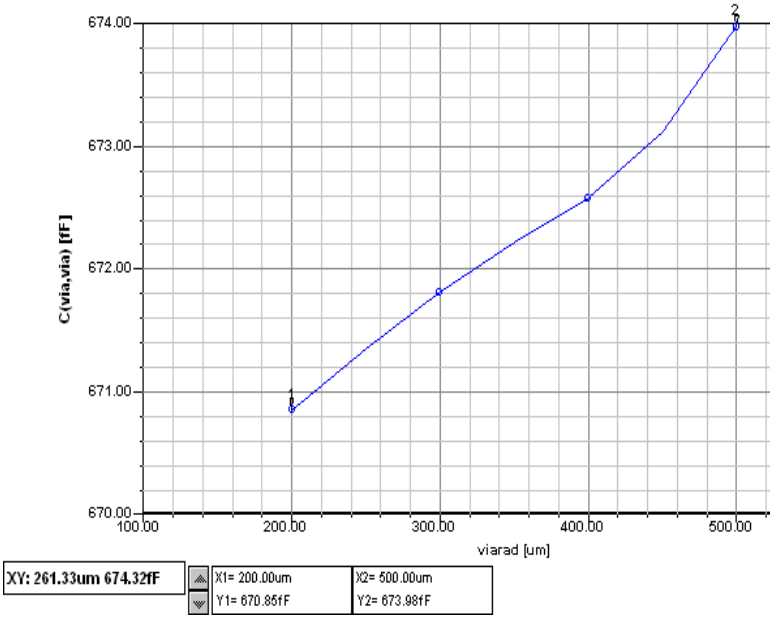
   The **Create Report** dialog box appears.
2. Verify that **Matrix** is selected in the **Report Type** list.
3. Verify that **Rectangular Plot** is selected in the **Display Type** list, and then click **OK**.

   The **Traces** dialog box appears, with the **Y** tab selected.
4. In the **Quantity** list, click **C(Via, Via)**.
5. Click **Add Trace**.

   A trace represents a line connecting data points on the plot. The
   X column reads **viarad**, and the Y column shows **C(Via, Via)**.
6. Click the **Sweeps** tab, and then select the **Sweep Design and Project variable values** radio button.
7. Click **Done**.

A graph showing the variation of **C(Via, Via)** versus **viarad** is generated. The plot is listed as **XY Plot1** under **Results** in the project tree.

The line styles in the plot were modified in the Trace Properties *dialog box for better visualization. To add data markers to all lines on the plot as shown: Double-click a line. In the* Trace Properties *dialog box, click the* Line Style *tab, select* Show Symbols On All Traces, *and then type 2 in the text box. The symbols associated with each line, shown in the legend to the right of the plot, will be added to the lines at every other data point. To change a line's color: Under the* Color *tab, modify the selected line's color by specifying new RGB values.*



The graph shows that the via capacitance is only slightly affected by the radius of the center barrel: the change in capacitance is from
0.670 pF to 0.674 pF. There is about 0.5% variation in capacitance when via radius changes by a factor of 2.5 times.

Plot the Results After Changing Capacitance

Next, generate another parametric sweep by varying the pad radius to see to its effect on the capacitance.

Modify the Pad Radius

1. Click **Q3D Extractor**>**Optimetrics Analysis>Add Parametric**.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.
3. Click **padrad** from the **Variable** pull-down list.
4. Verify that **Linear Step** is selected in the **Type** list.
5. Specify the following values:

Start	**0.5mm**
Stop	**1.1mm**
Step Size	**0.1mm**

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

   Q3D simulates the model with various values in the specified range, including the start and stop values.

   The frequency sweep is listed in the project tree under **Optimetrics** as **ParametricSetup2**.

Run Parametric Analysis

Now you can run the parametric analysis.

1. Right-click **ParametricSetup2**, and click **Analyze** from the shortcut menu.

   The **Progress** dialog box appears.
2. Right-click again on **ParametricSetup2,** and click **View Analysis Result** from the shortcut menu.

The **Post Analysis Display** dialog box appears, listing the values of **padrad** that have actually been solved.

Plot Results

1. Right-click **Result** in the project tree, and then click **Create Report**.

   The **Create Report** dialog box appears.
2. Verify that **Matrix** is selected in the **Report Type** list and that **Rectangular Plot** is selected in the **Display Type** list.
3. Click **OK**.

   The **Traces** dialog box appears.
4. Click the **Sweeps** tab, and then select the **Sweep Design and Project variable values** radio button.

   The available sweep variables are **viarad** and **padrad**; **viarad** being the **Primary Sweep** variable.
5. Specify sweep values for **viarad**:
   a. Click **viarad**.

      A small select menu appears, listing **viarad** and **padrad**.
   b. Select **padrad** from the list. **padrad** moves to the first row, becoming the primary sweep variable.

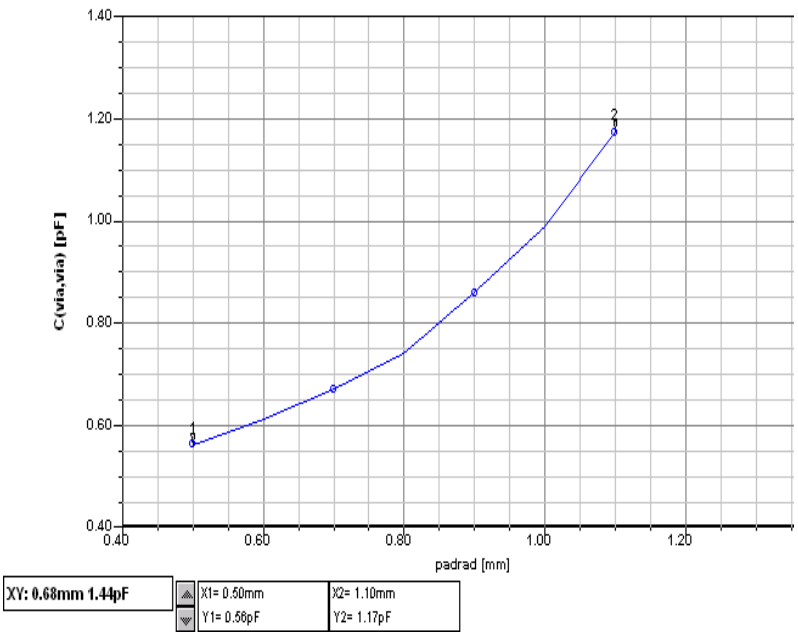
      The **Description** column reads **All Values**. But, you only want to see the variation of capacitance versus **padrad** from the second parametric sweep. During this sweep, **viarad** remains constant at **0.2mm**.
   c. Click the box next to **viarad** to highlight its row.
   d. Click **All Values**.
   e. Click **0.2mm** from the list.

   The **Description** column for **viarad** changes from **All Values** to **0.2mm**.
6. Click the **Y** tab.
7. In the **Quantity** list, click **C(Via, Via)**.
8. Click **Add Trace**.

   The X column displays **padrad**.
9. Click **Done.**

A graph appears, displaying the variation of the via capacitance versus the pad radius.

The line styles in the plot were modified in the Trace Properties *dialog box for better visualization.*
To add data markers to all lines on the plot as shown: Double-click a line. In the Trace Properties *dialog box, click the* Line Style *tab, select* Show Symbols On All Traces, *and then type 2 in the text box. The symbols associated with each line, shown in the legend to the right of the plot, will be added to the lines at every other data point.*
To change a line's color: Under the Color *tab, modify the selected line's color by specifying new RGB values.*



The graph shows a much stronger effect from the pad radius than the center barrel radius - the capacitance changes from 0.56 pF to 1.17 pF (100% variation) over the range of the sweep.

Close the Project and Exit Q3D Extractor

Congratulations! You have successfully completed the *Getting Started with Q3D Extractor: A PCB Via Model* guide! You may close the project and exit the software.

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

Index