



February 2011
EM Simulation

© **Agilent Technologies, Inc. 2000-2011**

5301 Stevens Creek Blvd., Santa Clara, CA 95052 USA

No part of this documentation may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Mentor products and processes are registered trademarks of Mentor Graphics Corporation. * Calibre is a trademark of Mentor Graphics Corporation in the US and other countries. "Microsoft®, Windows®, MS Windows®, Windows NT®, Windows 2000® and Windows Internet Explorer® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Oracle and Java and registered trademarks of Oracle and/or its affiliates. Other names may be trademarks of their respective owners. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC. FLEXIm is a trademark of Globetrotter Software, Incorporated. Layout Boolean Engine by Klaas Holwerda, v1.7 <http://www.xs4all.nl/~kholwerd/bool.html> . FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg. QuestAgent search engine (c) 2000-2002, JObjects. Motif is a trademark of the Open Software Foundation. Netscape is a trademark of Netscape Communications Corporation. Netscape Portable Runtime (NSPR), Copyright (c) 1998-2003 The Mozilla Organization. A copy of the Mozilla Public License is at <http://www.mozilla.org/MPL/> . FFTW, The Fastest Fourier Transform in the West, Copyright (c) 1997-1999 Massachusetts Institute of Technology. All rights reserved.

The following third-party libraries are used by the NlogN Momentum solver:

"This program includes Metis 4.0, Copyright © 1998, Regents of the University of Minnesota", <http://www.cs.umn.edu/~metis> , METIS was written by George Karypis (karypis@cs.umn.edu).

Intel® Math Kernel Library, <http://www.intel.com/software/products/mkl>

SuperLU_MT version 2.0 - Copyright © 2003, The Regents of the University of California, through Lawrence Berkeley National Laboratory (subject to receipt of any required approvals from U.S. Dept. of Energy). All rights reserved. SuperLU Disclaimer: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

7-zip - 7-Zip Copyright: Copyright (C) 1999-2009 Igor Pavlov. Licenses for files are: 7z.dll: GNU LGPL + unRAR restriction, All other files: GNU LGPL. 7-zip License: This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA.

unRAR copyright: The decompression engine for RAR archives was developed using source code of unRAR program. All copyrights to original unRAR code are owned by Alexander Roshal. unRAR License: The unRAR sources cannot be used to re-create the RAR compression algorithm, which is proprietary. Distribution of modified unRAR sources in separate form or as a part of other software is permitted, provided that it is clearly stated in the documentation and source comments that the code may not be used to develop a RAR (WinRAR) compatible archiver. 7-zip Availability: <http://www.7-zip.org/>

AMD Version 2.2 - AMD Notice: The AMD code was modified. Used by permission. AMD copyright: AMD Version 2.2, Copyright © 2007 by Timothy A. Davis, Patrick R. Amestoy, and Iain S. Duff. All Rights Reserved. AMD License: Your use or distribution of AMD or any modified version of AMD implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. AMD Availability: <http://www.cise.ufl.edu/research/sparse/amd>

UMFPACK 5.0.2 - UMFPACK Notice: The UMFPACK code was modified. Used by permission. UMFPACK Copyright: UMFPACK Copyright © 1995-2006 by Timothy A. Davis. All Rights Reserved. UMFPACK License: Your use or distribution of UMFPACK or any modified version of UMFPACK implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. UMFPACK Availability: <http://www.cise.ufl.edu/research/sparse/umfpack> UMFPACK (including versions 2.2.1 and earlier, in FORTRAN) is available at <http://www.cise.ufl.edu/research/sparse> . MA38 is available in the Harwell Subroutine Library. This version of UMFPACK includes a modified form of COLAMD Version 2.0, originally released on Jan. 31, 2000, also available at <http://www.cise.ufl.edu/research/sparse> . COLAMD V2.0 is also incorporated as a built-in function in MATLAB version 6.1, by The MathWorks, Inc. <http://www.mathworks.com> . COLAMD V1.0 appears as a column-preordering in SuperLU (SuperLU is available at <http://www.netlib.org>). UMFPACK v4.0 is a built-in routine in MATLAB 6.5. UMFPACK v4.3 is a built-in routine in MATLAB 7.1.

Qt Version 4.6.3 - Qt Notice: The Qt code was modified. Used by permission. Qt copyright: Qt Version 4.6.3, Copyright (c) 2010 by Nokia Corporation. All Rights Reserved. Qt License: Your use or distribution of Qt or any modified version of Qt implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This

library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. Qt Availability: <http://www.qtsoftware.com/downloads> Patches Applied to Qt can be found in the installation at: \$HPEESOF_DIR/prod/licenses/thirdparty/qt/patches. You may also contact Brian Buchanan at Agilent Inc. at brian_buchanan@agilent.com for more information.

The HiSIM_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or STARC.

Errata The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Warranty The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this documentation and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

Technology Licenses The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license. Portions of this product include the SystemC software licensed under Open Source terms, which are available for download at <http://systemc.org/>. This software is redistributed by Agilent. The Contributors of the SystemC software provide this software "as is" and offer no warranty of any kind, express or implied, including without limitation warranties or conditions or title and non-infringement, and implied warranties or conditions merchantability and fitness for a particular purpose. Contributors shall not be liable for any damages of any kind including without limitation direct, indirect, special, incidental and consequential damages, such as lost profits. Any provisions that differ from this disclaimer are offered by Agilent only.

Restricted Rights Legend U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

EM Simulation Overview	7
Getting Started with an EM Simulation	7
Process for Simulating a Design	8
Setting up EM Simulations	10
Overview	10
Using the EM Setup Window	10
Selecting an EM Simulator	13
Using the EM Toolbar	15
Viewing Layout Information	16
Assigning Substrate	17
Generating Ports	19
Assigning a Layout Pin to the S-parameter Port	19
Creating and Deleting Ports	21
Viewing S-parameter Ports	22
Viewing Layout Pins	28
Defining the Frequency and Output Plan	29
Selecting Output Dataset	32
Defining Simulation Options	32
Defining Expert Options for Momentum	54
Specifying Simulation Resources	58
Generating EM Model and Symbol	63
Running EM Simulations	70
Starting an EM Simulation	70
Viewing Simulation Status	70
Stopping an EM Simulation	71
Viewing the Simulation Summary	71
Viewing Simulation Summary	73
Viewing Momentum Mesh	73
Viewing S-param Simulation Summary	73
Viewing Mesh Generation Summary	74
Viewing Geometry Preproc Summary	75
Viewing Substrate Generation Summary	76
Viewing 3D Simulations	77
Starting an EM Visualization	77
Visualizing 3D View before EM Simulation	77
Validating Your Geometry Visually	78
Navigating in the 3D Environment	79
Specifying Material and Object Settings	79
Customizing the Simulation View	82
Setting Up the Viewer on External X Window Displays	84
Visualizing Momentum Simulations	84
Starting Momentum Visualization	85
Controlling Visualization Excitations	86
Plotting Properties	87
Animating Momentum Fields	89
Plotting Momentum Mesh	90
Visualizing FEM Simulations	90
Starting FEM Visualization	90
Controlling Visualization Excitations	92
Setting up Plots	92
Computing Radiation Patterns	100
About Radiation Patterns	100
About Antenna Characteristics	101
Calculating Far Fields	104
Viewing Far Fields	105
Exporting Far Field Data	107
Adding Boxes, Waveguides, and Symmetry Planes	109
Adding a Box	109
Adding a Waveguide	110
About Boxes and Waveguides	112
Adding a Symmetry Plane	113
Defining Component Parameters	114
Using Nominal/Perturbed Designs	114
Using Subnetwork Design	116
Using an EM Model	117

Advanced Design System 2011.01 - EM Simulation

Creating an EM Model	117
Using an EM Model Window	118
Updating an EM Model	121
Using an EM Model for Circuit Simulation	122
Specifying EM Model Options	123
Using Advanced Model Composer	126
Creating AMC Models	126
AMC Model Generation Technique	129
Using the Job Manager	131
Starting the Job Manager	131
Job Progression	132
Managing Jobs	133
Managing Hosts	134
Setting Preferences	134
Using EMPro 3D Components in ADS	142
Using EMPro Default 3D Components in ADS	142
Using Parametrized EMPro 3D Components in ADS	149
Performing SI/PI Analysis	151
Overview	151
Setting up SI/PI Analyzer	151
Using the SI/PI Analyzer Setup	151
Flattening the Design Hierarchy	154
Selecting Nets	155
Analyzing Pins	157
Inspecting Components	158
Specifying EM Simulation Setup	159
Using SI and PI Analyzer	161
Opening SI/PI Analyzer	161
Defining Power Distribution Network Analysis	162
Defining Signal Path Analysis	164
Performing Analysis	166
Viewing results	167
Example	169
Performing Momentum Circuit Excitation	171
Enabling Momentum Circuit Excitation AEL	171
Creating EM Models	171
Creating Circuits for AC/HB	172
Designing a Microstrip Line	175
Defining the EM Setup	179

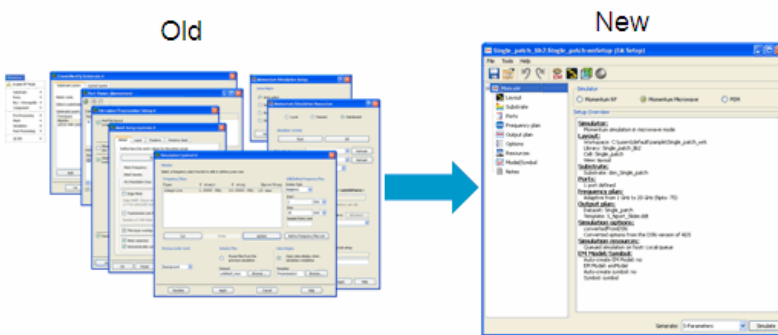
EM Simulation Overview

ADS 2011.01 provides the EM simulation tools that you need for designing and evaluating modern communications systems products. It provides a unified interface for Momentum and FEM simulators. These are electromagnetic simulators that compute S-parameters for general planar circuits, including microstrip, slotline, stripline, coplanar waveguide, and other topologies.

The new use model for EM simulations provides the following advantages:

- Single EM Setup window that replaces the tasks performed by multiple dialog boxes.
- Unified use model for Momentum, Momentum RF and FEM.
- Job manager provides improved ease-of-use for monitoring multiple local, remote, queued or distributed simulations.
- EM models are fully compatible with the new dynamic model selection.
- EM Simulation toolbar provides shortcuts to frequently used menus.
- Generates reusable EM setups that offer the following benefits:
 - Creates easy to save and restore EM setups.
 - Reduces the efforts to define EM settings.
 - Simplifies the simulation process that can be easily implemented by all types of users.
 - Ensures easier, faster, and accessible EM simulations.

In the earlier versions, you had to open multiple dialog boxes to configure settings for an EM simulation. However, with ADS 2011.01, you can use the EM Setup window to specify multiple settings by using a single interface. The following figure highlights this difference.



Getting Started with an EM Simulation

ADS 2011.01 includes a new EM menu that enables you to perform various tasks such as simulating a circuit, configuring a simulation, and adding a box or waveguide. Using the EM menu, you can perform the following tasks:

- **Performing simulations:** Choose **EM> Simulate** to perform an EM simulation. You can also press **F7** to start the simulation process.
- **Configuring a Simulation Setup:** Choose **EM> Simulation Setup...** to open the **EM Setup** window. You can also press **F6** to open the **EM Setup** window. The EM Setup window enables you define frequency and output plan, configure engine options, generate ports and EM model. For more information, refer to *Setting up EM Simulations* (em).
- **Modifying the Output Dataset name:** Choose **EM> Choose Output Dataset...** to change the name of the dataset that is used to store the EM S-Parameter results for viewing with data display.
- **Stopping a simulation:** To end a simulation before it is finished, choose **EM> Stop and Release Simulator**. This will release your simulation license.

- **Clearing the Momentum Mesh:** Choose **EM> Clear the Momentum Mesh** to remove the momentum mesh.
- **Viewing the recent simulations:** You can view the latest EM simulations from the current EM Setup by selecting the **EM> Show Most Recent** option.
- **Creating/editing Layout components:** You can create, edit, or select a parameter by using the **Layout Component Parameters** dialog box. Select **EM> Components> Parameters** to open this dialog box.
- **Viewing the results:** You can view and analyze, S-parameters, currents, far-fields, antenna parameters, and transmission line data by choosing **EM> Post-Processing > Visualization**. For more information, refer to *Visualizing 3D View before EM Simulations (em)*.
- **Adding or deleting a box or waveguide:** You can specify a box or waveguide around the substrate by selecting **EM> Box - Waveguide**. You can select the required options for adding or deleting a box or waveguide. For more information, refer to *Adding Boxes, Waveguides, and Symmetry Planes (em)*.
- **Defining radiation patterns:** You can compute the electromagnetic fields by choosing **EM> Post-Processing> Far Field**. For more information, refer to *Computing Radiation Patterns (em)*.
- **Adding and deleting FEM symmetry planes:** You can add an FEM symmetry plane by selecting **EM> FEM Symmetry Plane> Add Symmetry Plane**. For more information, refer to *Adding Boxes, Waveguides, and Symmetry Planes (em)*.


Process for Simulating a Design

Perform the following process for creating and simulating a design:

- **Creating a physical design:** You can 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 the Schematic Capture and Layout documentation. For information on importing designs, refer to the Importing and Exporting Designs documentation.
- **Selecting an EM simulator:** From the EM Setup dialog box, you can select the required simulator: Momentum Microwave, Momentum RF, and FEM. For more information, refer to *Overview#Selecting an EM Simulator (em)*.
- **Defining the substrate:** A substrate is the media where a circuit resides. Using the EM Setup dialog box, you can assign predefined substrates. For more information, refer to *Assigning Substrate (em)*.
- **Assigning 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. There are several different types of ports that you can use in your circuit, depending on your application. For more information, refer to *Generating Ports (em)*.
- **Defining the Frequency and Output Plan:** You can set up multiple frequency plans for a simulation. For each plan, a solution can be found for a single frequency point or over a frequency range. You can define the output plans. For more information, refer to *Defining the Frequency and Output Plan (em)*.
- **Setting Simulation Options:** You set up a simulation by specifying the parameters of preprocessor, substrates, and mesh. You can specify a variety of mesh parameters

to customize the mesh to your design. For more information, refer to *Defining Simulation Options* (em).

- **Simulating the circuit:** After completing the setup, you can run an EM simulation. Choose the **Simulate** option to perform an EM simulation. You can also press F7 to start the simulation process.

 **Note**
You need to specify only valid data in the EM Setup window. If you copy EM_setup file from one library to another library, all the information is not copied. For example, you may need to define the substrate in the new library. Similarly, if a layout does not have the pins referenced in the original layout, the pins will not be visible and all the ports are invalid.

Setting up EM Simulations

ADS 2011.01 simplifies the overall process of setting up EM simulations. The new EM usability provides a user-friendly EM Setup window for defining Momentum and FEM simulations. Using the EM Setup window, you can specify the settings for your EM simulation from a single interface. You can assign substrates, configure ports, define frequency and output plans, specify local, remote or distributed setup options, and customize the EM data name generated during a simulation.

This section provides the following information about setting up EM simulations:

- *Overview (em)*
- *Viewing Layout Information (em)*
- *Assigning Substrate (em)*
- *Generating Ports (em)*
- *Defining the Frequency and Output Plan (em)*
- *Defining Simulation Options (em)*
 - *Overriding Substrate Definition for Momentum (em)*
 - *Defining Preprocessor Settings (em)*
 - *Defining Mesh Settings for Momentum (em)*
 - *Defining Mesh Settings for FEM (em)*
 - *Defining Expert Options for Momentum (em)*
 - *Defining Simulation Settings (em)*
- *Specifying Simulation Resources (em)*
- *Generating EM Model and Symbol (em)*

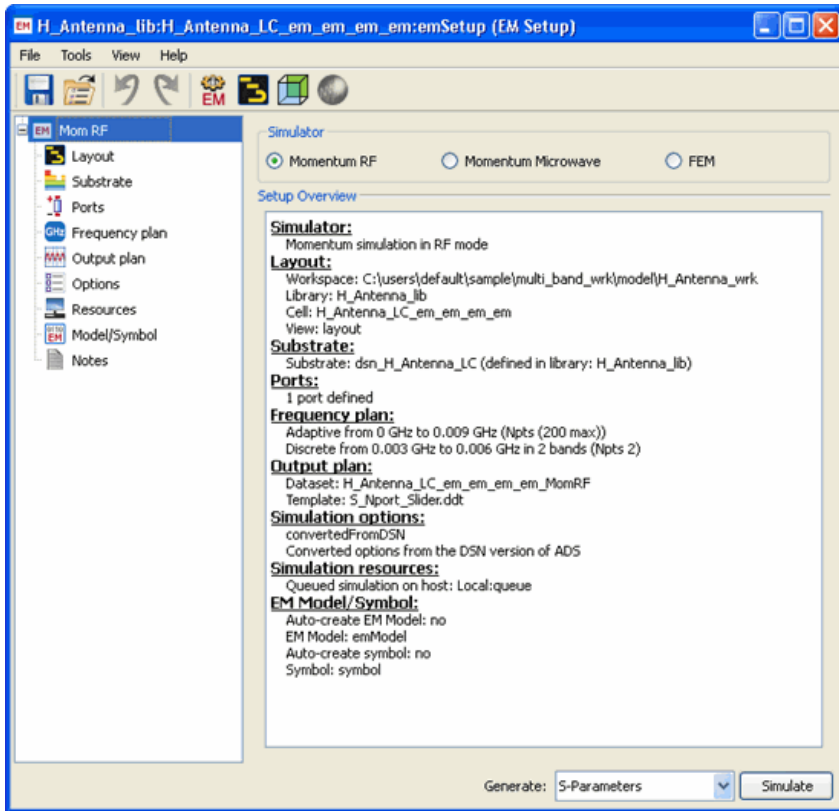
Overview

ADS 2011.01 provides an improved EM Simulation usability. It is now easier to create and use EM models for arbitrary parts of a layout by designers who are not EM experts. ADS 2011.01 provides a new EM GUI, EM Setup window, which provides a single interface to control all EM setup related features. It provides a:

- Unified EM interface to reduce maintenance efforts.
- User-friendly interface for defining Momentum and FEM simulations.

Using the EM Setup Window

To open the EM Setup window, choose **EM > Simulation Setup**. The EM Setup window enables you to define simulator options such as substrate, ports, and frequency plan. The following figure displays an EM Setup window:



The following sections describe the EM Setup window:

- [Specifying Simulation Settings](#)
- [Selecting a Simulator](#)
- [Viewing Setup Overview](#)
- [Performing Tasks](#)
- [Simulating a Design](#)

Specifying Simulation Settings

In the EM Setup window, options required for specifying simulation settings are listed in the left pane. You can select the required option to open the settings in the right pane of the EM Setup window. You can specify settings for the following options:

- **Layout:** You can view information about the workspace, library, cell, and view by selecting **Layout**. For more information, refer *Viewing Layout Information (em)*.
- **Substrate:** You can open a predefined substrate file from ADS by selecting **Substrate**. For more information, refer *Assigning Substrate (em)*.
- **Ports:** You can refresh layout pins information, create, delete, and resequence ports, and search the required S-parameter ports or layout pins by selecting **Ports**. For more information, refer to *Generating Ports (em)*.
- **Frequency Plan:** You can add or remove frequency plans for your EM simulations by selecting **Frequency Plan**. For more information, refer to *Defining the Frequency and Output Plan (em)*.
- **Output Plan:** You can specify the data display settings for your EM simulations by selecting **Output Plan**. For more information, refer to *Defining the Frequency and Output Plan (em)*.
- **Options:** You can define the preprocessor, mesh, simulation, and expert settings by selecting **Options**. For more information, refer to *Defining Simulation Options (em)*.
- **Resources:** You can specify the local, remote, and third party settings by selecting

Resources. For more information, refer to *Specifying Simulation Resources (em)*

- **Model/Symbol:** You can generate an EM model and symbol by selecting **Model/Symbol**. For more information, refer to *Generating EM Model and Symbol (em)*.
- **Notes:** You can add comments to your EM Setup window by selecting **Notes**.

Selecting a Simulator

In the EM setup window, you can select one of the following types of EM simulators:

- Momentum Microwave
- Momentum RF
- FEM


For more information, refer to the [Selecting an EM Simulator](#) section.

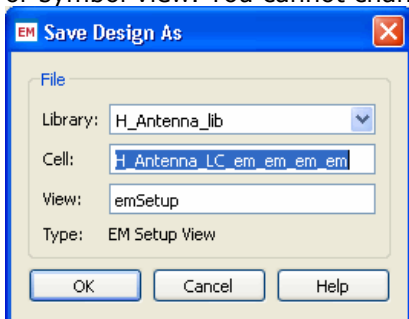
Viewing Setup Overview

You can view the summary of the EM simulation options in the Setup Overview section of the EM Setup window.

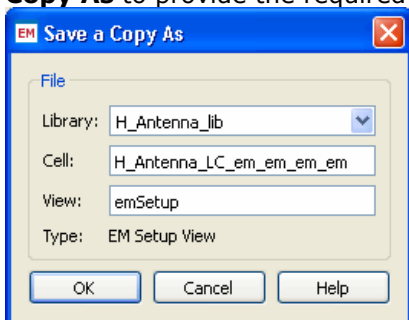
Performing Tasks

You can perform the following tasks by using the menus and toolbar options present in the EM Setup window:

- **Saving the simulation settings:** Click **Save** () in the EM Setup window to save the settings specified for your EM simulation. You can also use the following commands from the File menu for saving an EM setup:
 - The Save command enables you to save changes to an existing design. If you choose Save in an untitled window, the Save Design As dialog box appears.
 - The Save As command enables you to save an existing EM setup with a new name. You can change the Library name, cell name, and view name. Type represents the type of cell to identify whether it is a EM setup, schematic, layout or symbol view. You cannot change the cell Type.



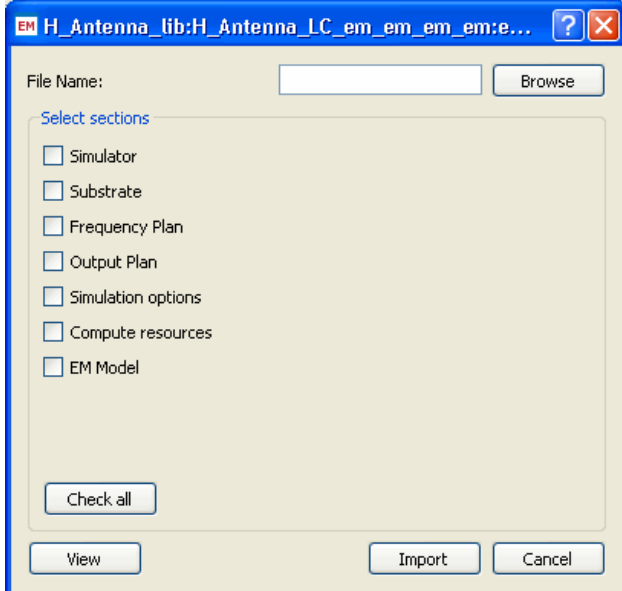
- The Save a Copy As command enables you to save a copy of an existing EM setup with a new cell name in new library and in new view. Select **File > Save a Copy As** to provide the required library name, cell name, and view.





Note
To reuse the saved EM setup, select **File > Open > EM Setup View** in the ADS Main window. Select the required EM Setup view in the Open Cell View dialog box. You can also open an existing EM setup by using the EM toolbar.

• **Importing files:** To import another file in the EM Setup window:

1. Choose **File > Import** or click the **Import** icon () to open the **Import EM Setup** dialog box, as shown in the following figure:




2. Specify the file name or click **Browse** to open the required file.
3. Select the required sections from the **Select sections** list. You can also select all the sections by clicking **Check all**.
4. Click **Import**.

- **Opening the 3D Preview window:** Choose **Tools > 3D EM Preview** or click the **3D EM Preview** icon () to preview your simulation.
- **Opening the Layout window:** Choose **Tools > Open Layout Window** or click the **Open Layout Window** icon () to display a layout window.

Simulating a Design

You can save the current settings and start the simulation process by clicking the

Simulate button or click () to simulate an EM simulation. You can also press **F7** to start the EM simulation. If you have specified invalid EM settings, an error message is displayed.

Selecting an EM Simulator

You can use the following types of EM simulators:

- Momentum Microwave
- Momentum RF
- FEM

The following sections describe these simulators.

Momentum Simulator Basics

Momentum provides 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.

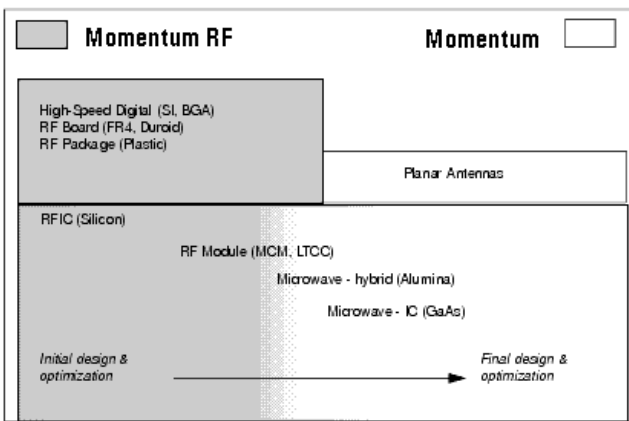
Momentum provides two types of simulation modes: Momentum Microwave and Momentum RF. The Microwave mode uses full-wave formulation, while the RF mode uses a quasi-static formulation. In the quasi-static formulation, Momentum RF provides accurate electromagnetic simulation performance at RF frequencies. 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.

Momentum enables you to:

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

Selecting the Momentum Simulation Mode

From the EM Setup dialog box, you can choose the required simulation mode. In each case, the menu label in the EM Setup dialog box changes to the current mode. You can choose a simulation mode based 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.



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 Microwave to include radiation effects for final design and optimization. Momentum RF is an efficient mode for a circuit that is electrically small, geometrically complex, and does not radiate.

Locating Examples

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

To open an example:

1. From the ADS Main window, choose **File > Open > Example**.
2. In the **Choose an Archived Example to open** dialog box, select **Momentum**.
3. Select the required example directory.
4. Click **OK**.

FEM Simulator Basics

The FEM simulator provides a complete solution for electromagnetic simulation of arbitrarily-shaped, passive three-dimensional structures. An FEM simulator provides the best price/performance, 3D EM simulator on the market, with a full 3D electromagnetic field solver, and fully automated meshing and convergence capabilities for modeling arbitrary 3D shapes such as bond wires and finite dielectric substrates. Along with Momentum, FEM provides RF and microwave engineers access to some of the most comprehensive EM simulation tools in the industry.

By combining fast solution times, efficient memory usage, and powerful display capabilities, FEM Simulator for ADS delivers leading price/performance solution to complex high-frequency problems. FEM Simulator for ADS users need very little background in electromagnetic field theory in order to operate and achieve accurate, and meaningful solutions.

Major Features and Benefits

The FEM simulator provides:

- Conductors, resistors, isotropic dielectrics, isotropic linear magnetic material modeling that allow a wide range of application coverage.
- An unlimited number of ports, which enables simulating multi-I/O design applications such as packages.
- Electric and magnetic fields modeling, enabling visualization of EM fields in a design.
- Absorbing boundary condition (free space), enabling antenna modeling.
- Full-wave, EM-accuracy for first-pass design success.
- Antenna parameters (gain, directivity, polarization, and so on), to enable better insight into antenna design.
- FEM Simulator/ADS integration providing an integrated approach to EM/Circuit design.

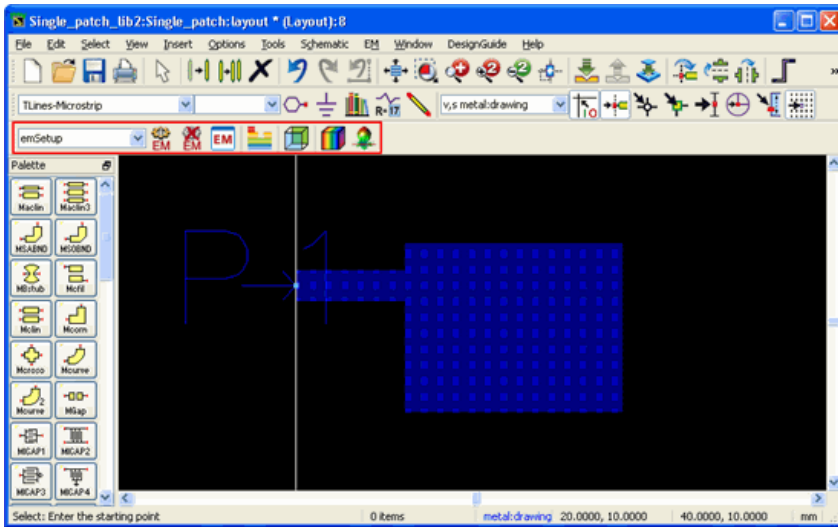
Application Areas

The FEM simulator supports the following applications:








- Microstrip, stripline, CPW elements such as filters, couplers, and spiral inductors
- Multilayer structures
- Ceramic filters
- Adapters/transitions
- Antennas
- Couplers
- Power splitters/combiners

Using the EM Toolbar

You can also use the EM toolbar to perform EM simulations. The EM toolbar provides shortcuts to the frequently-used menus. It is present in the Layout window, as shown highlighted in the following figure:



You can perform the following tasks by using the EM toolbar:

- **Starting an EM simulation:** Click **Start EM Simulation** () to initiate the simulation process.
- **Stopping an EM simulation:** Click **Stop EM Simulation** () to stop an EM simulation and release the license.
- **Opening the EM Setup window:** Click **EM Simulation Setup** () to display the EM Setup window.
- **Opening the substrate editor window:** Click **Substrate Editor** () to open the substrate editor window.
- **Opening the 3D Preview window:** Click **3D EM Preview** () to preview an EM simulation.
- **Opening the 3D Viewer window:** Click **Visualization** () to open the 3D Viewer and load the simulation results of the current EM setup.
- **Calculating far fields:** Click **Far Field** () to calculate far fields from the simulation results of the current EM setup.

Viewing Layout Information

You can view information about the current layout in the EM Setup window. This window provides the following information about the location of a workspace, library, cell, and view of the current layout:

- **Workspace:** Specifies the complete path to the location where you have saved the workspace file.
- **Library:** Specifies the name of the library.
- **Cell:** Specifies the name of the cell.
- **View:** Specifies the view that is opened for setting up the EM simulation.

Note
For detailed information about workspace, library, cell, and view, refer to [Advanced Design System Quick Start](#) .

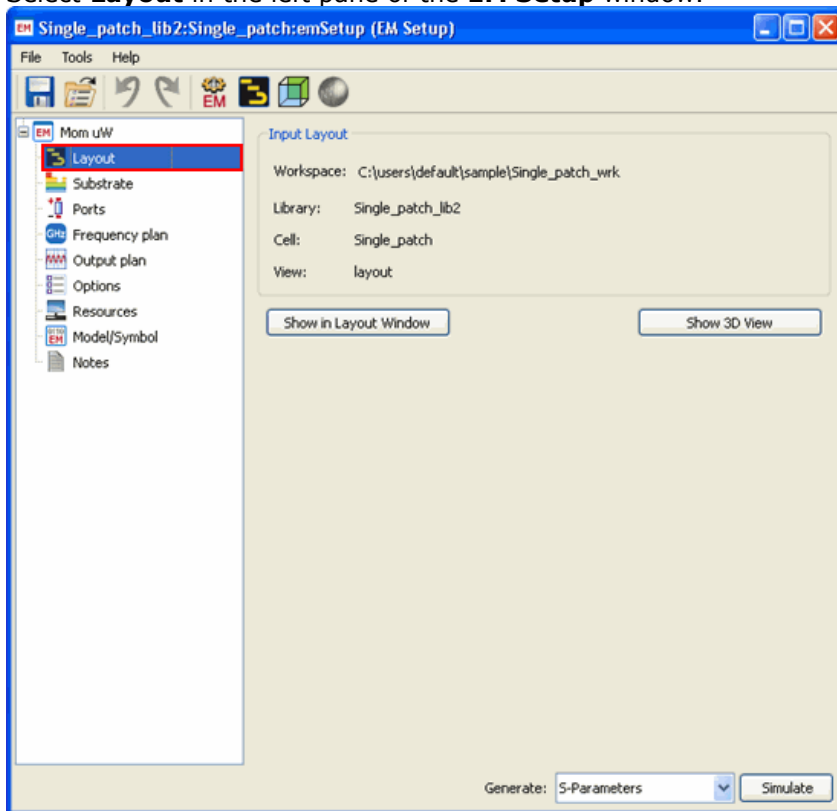
From the EM Setup window, you can also open the structure in the following window:

- 3D view
- Layout window

Displaying Layout Information

To display the layout information for your EM simulation:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Layout** in the left pane of the **EM Setup** window.



3. View the **Workspace, Library, Cell, and View** information in **Input Layout**.
4. Click **Show in Layout Window** to open the layout window.
5. Click **Show in 3D View** to open a 3D view for your EM simulation.

Assigning Substrate

You can define a substrate in EM simulations to describe the media where a circuit exists. 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. For example, the substrate of a multilayer circuit board consists of layers of metal traces, insulating material, ground planes, and vias that connect traces and the air that surrounds the board. In addition, 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. You can save the substrate definitions and use with other circuits.

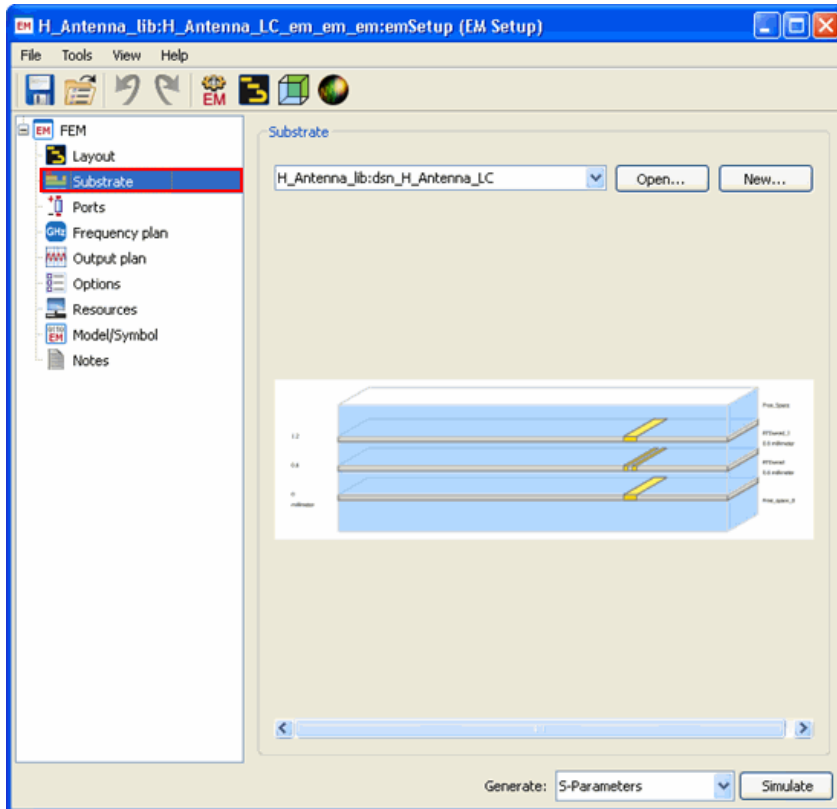
A substrate consists of following two layers:

- **Substrate Layer:** This layer defines the dielectric media, ground planes, covers, air, or other layered material.
- **Metallization Layers:** This is the conductive layers in between the substrate layers, which 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.

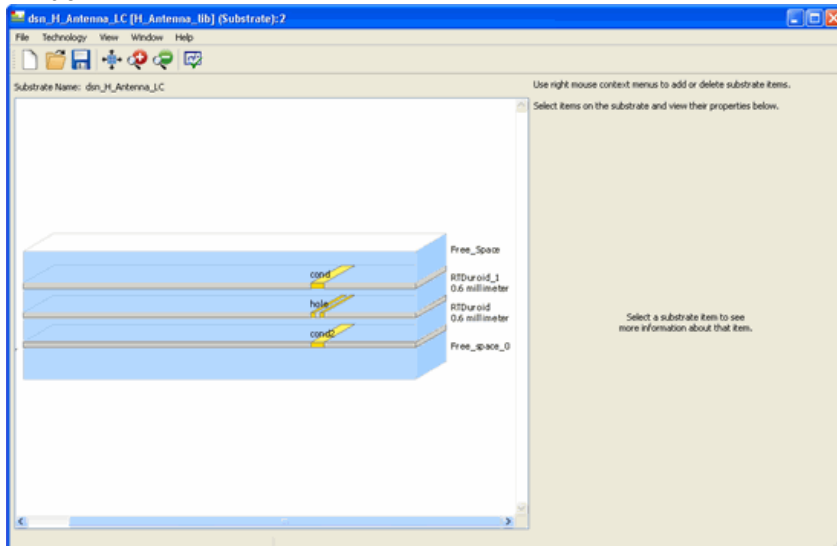
Selecting a Predefined Substrate

To open a predefined substrate:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Substrate** in the left pane of the EM Setup window.



3. Select an existing ADS substrate file from the drop-down list.
4. To open the **Substrate** window, click **Open**. The following figure displays a Substrate window.



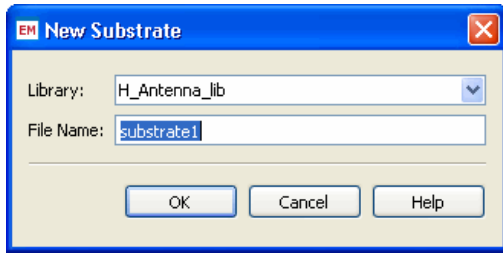
Note

For more information about substrates, refer *Substrates in EM Simulation* (adstour)

Creating a Substrate

To create a new substrate:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Substrate** in the left pane of the EM Setup window.
3. Click **New** to display the **New Substrate** dialog box, as shown below:



4. Select the required library from the **Library** drop-down list.
5. Type a name in the **Substrate** text box.
6. Click **OK**.

Generating Ports

To determine how an EM model can interconnect with the other components in a schematic, you need to configure ports for the EM model generation. Using the EM Setup window, you can improve the use model of the port setup process. You can refresh layout pins information, create, delete, resequence ports, and search the required S-parameter ports or layout pins. It provides the following benefits:

- Provides easier editing or viewing capability.
- Provides flexible S-parameter port setup capability.
- Assists in setting up S-parameter ports.

Note
The port definitions are stored with the layout. Therefore, it is recommended to enable the **Save all designs when simulation starts** preference from the main window of PDE.

Assigning a Layout Pin to the S-parameter Port

Momentum and FEM generate N-port S-parameter models for your circuit. The S-parameter Ports define the input/output locations where current can be injected and/or measured and/or exchanged. They are also required to connect the physical structure with other electrical circuit components in an EM/circuit co-simulation.

Each port consists of two terminals, **+** terminal (signal) and **-** terminal (reference). The port current flows from the **+** terminal into the circuit and leaves the circuit via the **-** terminal. The port voltage is defined as the difference of the electric scalar potential between the two terminals:

$$V_p = V(r+) - V(r-)$$

Note
The scalar potential $V(r)$ at infinity is zero.

While the **+** terminal is always explicitly defined, the **-** terminal can be defined in the following ways:

- When there is an infinite ground plane, the **-** terminal is implicitly located in the infinite ground plane closest to the **+** terminal.
- In the absence of an infinite ground plane, the **-** terminal needs to be explicitly defined somewhere close to the **+** terminal.
- For Momentum, when there is no **-** terminal and no infinite ground plane, the port voltage is defined as the value of the electric scalar potential at the **+** terminal: $V_p = V(r+)$. You can rewrite this as follows:

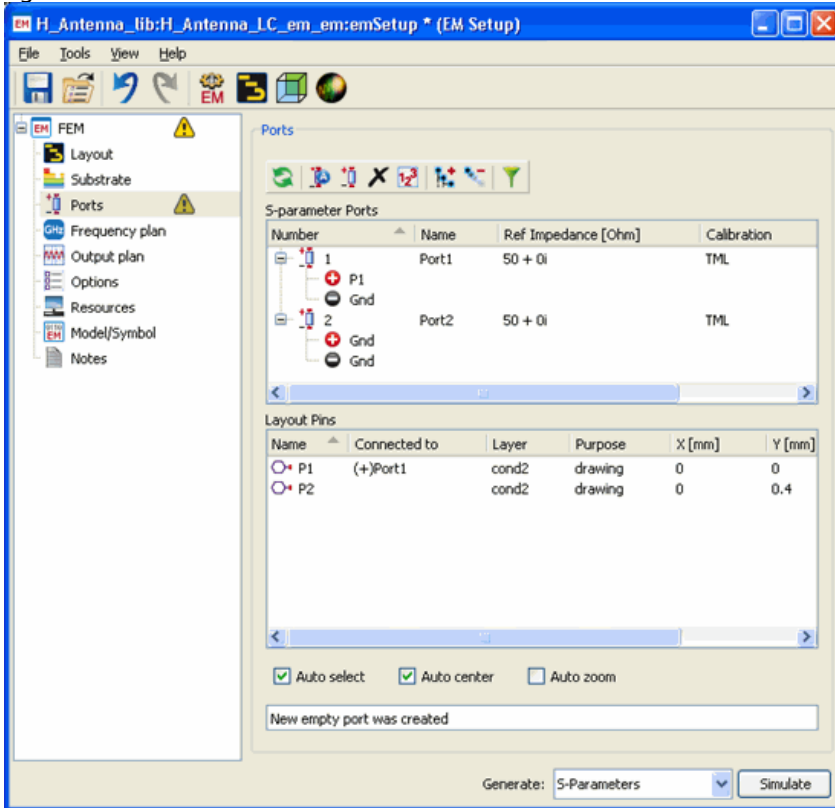
$$V_p = V(r+) - V(\text{at infinity})$$

Therefore, you can consider infinity as the location of the implicit **-** terminal for this case. For FEM, **-** terminals need to be defined in all cases in absence of an infinite ground. Ports are defined in a two-step process. First, layout pins are added to the circuit layout. These pins define the places where current can enter or leave the circuit. Then, using the EM Port Editor, you specify how the layout pins are used to define the S-parameter ports of

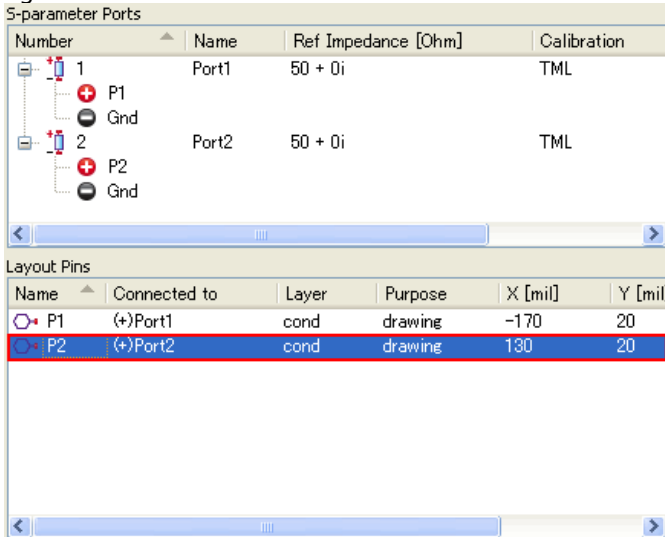
the EM simulation. The EM Port Editor is a separate tab in the EM setup window.

To assign a layout pin to the S-parameter port:

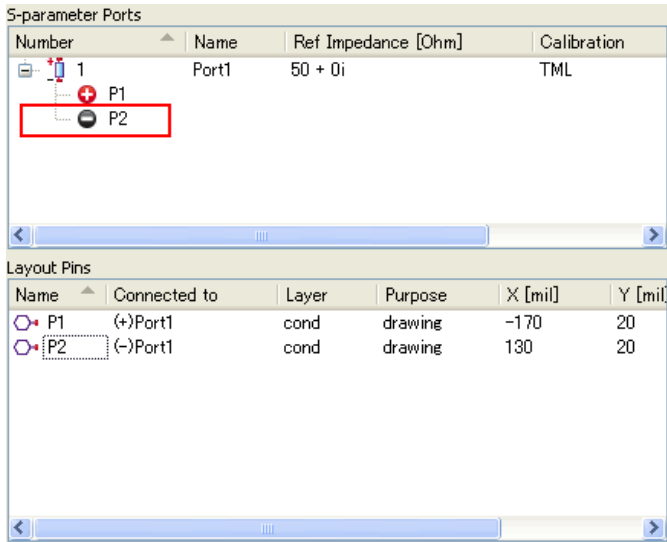
1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Ports** in the left pane of the EM Setup window, as shown in the following figure:



3. Select a pin row in **Layout Pins**. For example, **P2** row is selected in the following figure:



4. Drag the pin from Layout Pins and drop it on the **+** or **-** terminal of the required port in **S-parameter Ports**. For example, P2 is dragged and dropped on the - terminal of **Port1** in the following figure:




5. A message box is displayed if your target pin is already used by other ports.
6. Click the **Delete Ports and Continue** button in the message box.


Note
You can drag and drop multiple layout pins and connect with a S-parameter port.

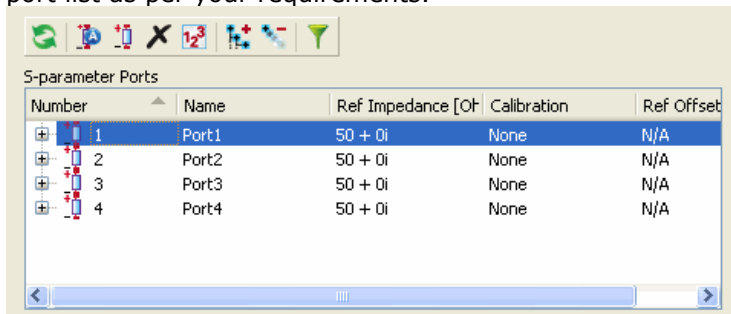
Creating and Deleting Ports


Using the toolbar options, you can create new ports, refresh layout pins, or delete selected ports. The following section describes the tasks that you can perform using the options available on the toolbar:

- **Refreshing Layout Pins and S-parameter S-parameter Ports information:**

Click **Refresh pins and ports** () to update the list of pins and ports with the latest information in the Layout window.


- **Automatically creating New Ports:** Click **Auto-create new ports** () to automatically generate one port per pin in the S-parameter Ports. You can edit the port list as per your requirements.



- **Creating New Ports:** To manually create a new port, click **Create New Port** () in the Ports toolbar. Then, select the required layout pins in Layout Pins and drag and drop them on the positive or negative terminal of the port to make connection.

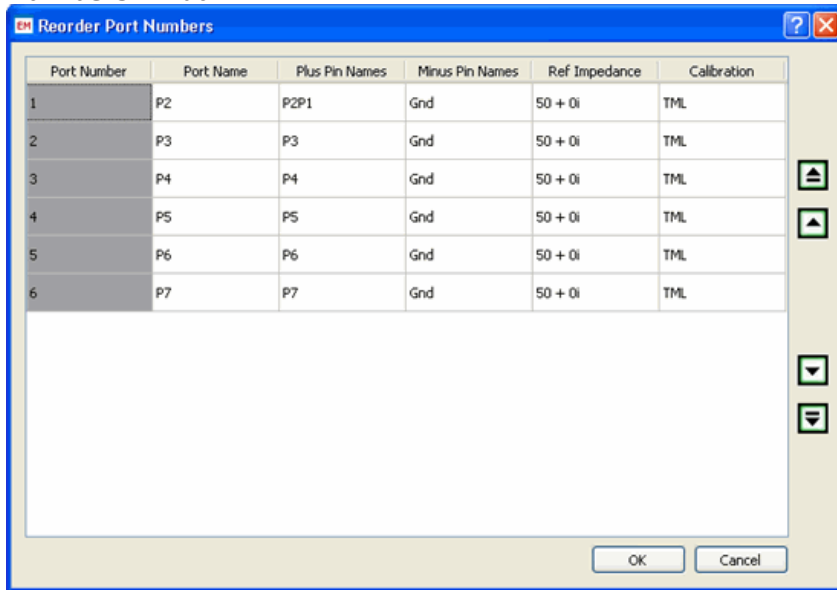
Note
There must be an unconnected pin in the layout pins to create a new port.

Note
You can also select and drag the required pin and connect it with a port in the S-parameter Port View.



- **Deleting selected items:** To delete the selected ports in S-Parameter Ports, click **Delete selected items** () in the Ports toolbar.

Note
The port number will always be retained in sequence.

- **Resequencing Ports:** Click **Resequence Ports** () to open the **Reorder Port Numbers** window.



From the Reorder Port Numbers window, select the required port and click the **Up** and **Down** keys to change the ports sequence according to your requirements.

- **Expanding S-parameter Ports tree:** To display the detailed information about each port in S-Parameter Ports, click **Expand S-parameter Ports tree** ().
- **Collapsing S-parameter Ports tree:** To hide the details of ports in S-Parameter Ports, click **Collapse S-parameter Ports tree** ().
- **Searching the required information:** For easy filtering and sorting commands, click **Toggle Search Filter On/Off** (filter.jpg!). Type the required string in the **Search** text box.

Viewing S-parameter Ports

You can view the list of S-parameter port numbers and terminals, as shown in the following figure:

Number	Name	Ref Impedance [Ohm]	Calibration	Ref Offset [mm]
1	Port1	50 + 0i	None	N/A
	P1			
	Gnd			
2	Port2	50 + 0i	None	N/A
	P2			
	Gnd			
3	Port3	50 + 0i	None	N/A
	P3			
	Gnd			

The following table describes the S-parameter Ports fields:

Field	Description
Name	Specifies the name of the first positive pin added. You cannot edit a port name.
Number	Specifies the s-parameter port number.
Ref Impedance [Ohm]	Specifies the reference impedance value in ohms.
Calibration	Allows you to set the following types of calibration: <ul style="list-style-type: none"> • TML • TML (zero length) • SMD • DeltaGap • None
Ref Offset [mil]	Specifies the Ref Offset value (shown in layout units).

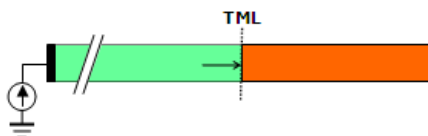
Selecting the Calibration Type

You should select the calibration type based on the type of circuit component you want to connect to the port in the schematic. You can select TML and TML (zero length) calibration only for the S-parameter ports that have associated layout pins located at an edge of the structure. You can select SMD and Delta gap only for S-parameters ports that have two associated layout pins located at the edges of a gap in the structure. For S-parameter ports which have at least one associated layout pin inside the structure, None is the only option that can be selected as calibration type. Depending on your circuit component, you can select the following types of calibration:

- **TML**: Connects a transmission line component.
- **TML(zero length)**: Connects a lumped component.
- **SMD**: Connects a 2-pin SMD component (model includes parasitics).
- **Delta gap**: Connects a 2-pin SMD component (model does not include parasitics).
- **None**: If none of the available calibration types is valid for the connecting component, specify None.

Selecting TML Calibration

If you select the TML calibration, Momentum automatically adds a transmission line feed line to the port. A source is added at the end of the feed line to excite the structure. The parasitic effect of the feed line and the open-end at the end of the feed line are automatically removed from the simulated S-parameters.



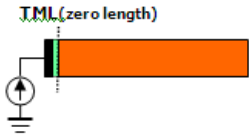
Within an FEM simulation, the TML calibration results in a waveguide port definition. FEM will only interpret TML ports when they are at the boundary of the simulation domain, if not the FEM simulator will automatically translate the TML port to a non-calibrated port and issue a warning.

Avoiding Overlap

While using TML 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.

Selecting TML (Zero Length) Calibration

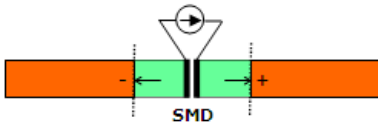
If you select the TML (zero length) calibration, Momentum directly adds a source at the port to excite the structure. The parasitic effect of the open-end at the port is automatically removed from the simulated S-parameters.



Note
For FEM, a TML (Zero Length) calibration is the same as a TML calibration line.

Selecting SMD Calibration

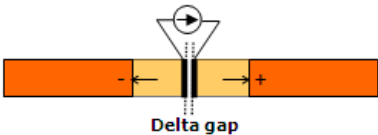
If you select the SMD calibration, Momentum automatically adds two feed lines to the two layout pins of the port. A source is added over the delta-gap connecting the two feed lines. The parasitic effect of the feed lines is automatically removed from the simulated S-parameters. SMD calibration should be used for connecting with a 2-pin SMD component whose model includes parasitics.



Note
While performing an FEM simulation, the SMD ports are treated as uncalibrated sheet ports and the reference offset specification is ignored. This means that in the FEM simulation model, the gap between the pins of the SMD port will contribute to an inductive effect in the resulting S-parameter model, unlike Momentum.

Selecting Delta Gap Calibration

if you select the Delta gap calibration, Momentum automatically adds two feed lines to the two layout pins of the port. A source is added over the delta-gap connecting the two feed lines. The parasitic effect of the feed lines is not removed from the simulated S-parameters. You should use Delta gap calibration for connecting with a 2-pin SMD component whose model does not include parasitics.

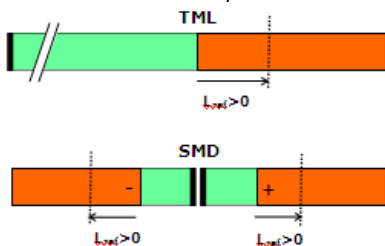


Note
While performing an FEM simulation, the Delta Gap ports are treated as uncalibrated sheet ports. The inductive effect of the gap between the pins of the Delta Gap port is included in the resulting S-parameter model, and this setup therefore behaves the same way as the Momentum Delta Gap source.

Selecting the Reference Offset

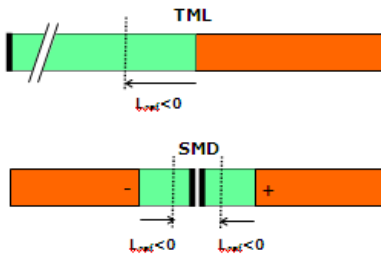
The calibrated ports enable you to shift the location of the reference lines used for the S-parameters. You can select the following types of reference offsets:

- Specifying a positive reference offset: Shifts the reference planes in the direction towards the circuit, as shown in the following figure:



- Specifying a negative reference offset: Shifts the reference planes in the direction

away from the circuit, as shown in the following figure:



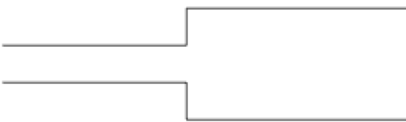
Why Use Reference Offsets?

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. However, 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.

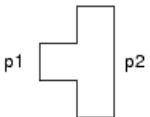
There are significant resemblances between this hardware measurement process and the way Momentum and FEM operate. In the case of Momentum and FEM, the probes are replaced by ports, which, during simulation, will feed energy to the circuit and measure its response. The port feeding scheme also has its own, unwanted effect: low-order mode mismatch at the port boundary, although this is eliminated by the calibration process.

However, for a calibration process, 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 feed line that is long enough to provide this distance.

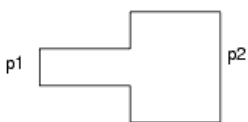
As a basic example, consider a line width that varies abruptly in some part of your circuit, as shown in the following figure:



You need characterize only the variation of the step-in-width itself, as shown in the following figure:

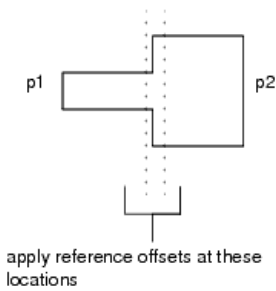


As mentioned previously, it takes a little distance for the fundamental mode to settle, which means that this *short* structure might not yield the accuracy that you expect from the simulation. In this case, allow for some feed line length, as shown in the following figure:



Now, the simulation will yield accurate results, but the results will also contain the extra line lengths. To resolved this issue, 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, as shown in the following

figure:



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. 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

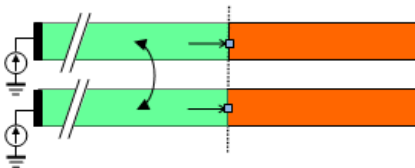
It is possible to de-embed up to the discontinuity itself. However, make sure that you do not shift the reference offset beyond the first discontinuity. This would yield incorrect simulation results, as there is another line width beyond that discontinuity, which means that there is another set of impedance 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.

S-parameter Port Grouping

The calibration process automatically groups S-parameter ports with terminals attached to layout pins that share a common reference plane. The following figure displays a grouped port:



S-parameter Port Terminals

In S-parameter Ports, you can view the  and  pins, which represent the plus and minus terminals of a port. The name besides the plus and minus pins specifies the pin name that is connecting to these terminals. For example, if you have assigned P3 for plus and P4 for minus, it is equivalent to the following figure:



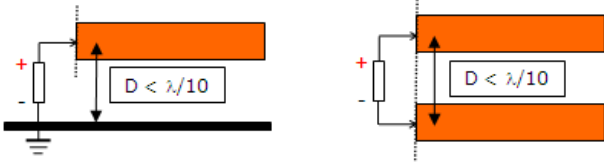
Note

You can assign any number of pins to the plus and minus terminal. However, if you do not assign any pin, it is shown as **Gnd**, which means that the implicit value is ground.

Physical Port Setup

You can connect the ports of an EM model to other circuit components in a schematic. To make a relevant connection physical, it is important that the physical dimensions of the S-parameter ports and the layout pins to which the ports are connected are **electrically small**.

For **S-parameter ports**, the physical distance between the plus and minus pins must be smaller than 1/10 of the wavelength at the highest frequency.



For **layout pins**, the length of the edge (layout pins at an edge of the structure) or the diameter of the area (layout pins inside the structure) must be smaller than 1/10 of the wavelength at the highest frequency.



The physical port setup restrictions stem from circuit theory that stipulates that all physical dimensions are electrically small so that no phase delays are present in the circuit. Failing to comply with the physical port setup restrictions, generates unphysical circuit S-parameters.

Note
To comply with the physical port setup, S-parameter ports for circuits with no infinite ground plane in the substrate should always be defined between two explicit layout pins.

Editing the S-parameter Ports Properties

You can modify the value of S-parameter Ports fields:

1. Click the required field in S-parameter Ports.
2. Specify the required values in the editable field.

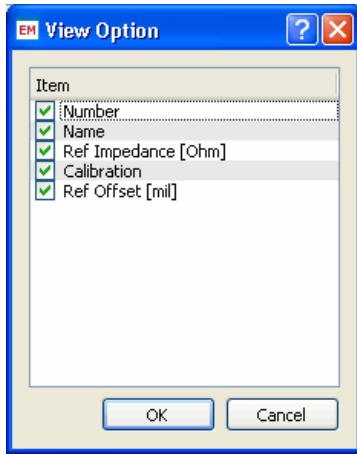
To modify multiple values at once in a single column

1. Edit one field
2. Right-click and copy the value
3. Select the fields that should get the new value
4. Right-click paste the value

Selecting the S-Parameter Ports Title Fields

You can select the required heading fields in S-parameter Ports by performing the following steps:

1. Right-click the title bar and then select **Header Visibility** option. This opens the **View Option** dialog box, as shown in the following figure:



2. Select the required fields.
3. Click **Ok**.

Viewing Layout Pins

You can view the list of layout pins and terminals in the EM Setup window, as shown highlighted in the following figure:

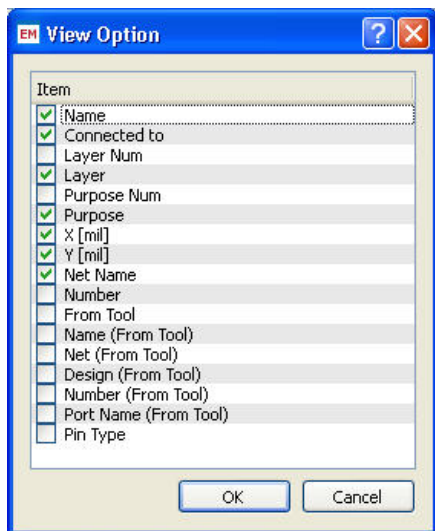
Name	Connected to	Layer	Purpose	X [mil]
P1	(+)Port1	cond	drawing	0
P2	(+)Port2	cond	drawing	200

Field	Description
Name	Specifies the layout pin name.
Connected To	Specifies the connecting terminal name.
Layer	Specifies the layer name.
Purpose	Specifies the purpose.
X [mm]	Specifies the X axis value (shown in layout units).
Y [mm]	Specifies the Y axis value (shown in layout units).
Net Name	Specifies the Net name.

Selecting the Layout Pins Title Fields

You can select the required heading fields in Layout Pins by performing the following steps:


1. Right-click the title bar and then select **Header Visibility** option. This opens the **View Option** dialog box, as shown in the following figure:



Note

The Tool-related parameters such as Name (from Tool) are available only when the pins are imported through ADFI. Otherwise, these parameters are shown as "N/A", if you try to select in the View Option.

2. Select the required fields.
3. Click **OK**.

 You cannot edit the the layout pin properties, as these are read-only fields.

Selecting Layout Pins

You can specify how pins are selected in a layout. You can choose one of the following options:

- **Auto select:** Selects pin(s) automatically.
- **Auto center:** Selects pin(s) from the center.
- **Auto zoom:** Increases the layout window zoom to the selected pin(s).

The following figure displays these options:

Auto select Auto center Auto zoom

Defining the Frequency and Output 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 a sweep type for your frequency plan. However, the collection of frequency plans will run as a single simulation. You can also specify the data display settings for your EM simulations by using the EM Setup Window.

Setting up Frequency Plans

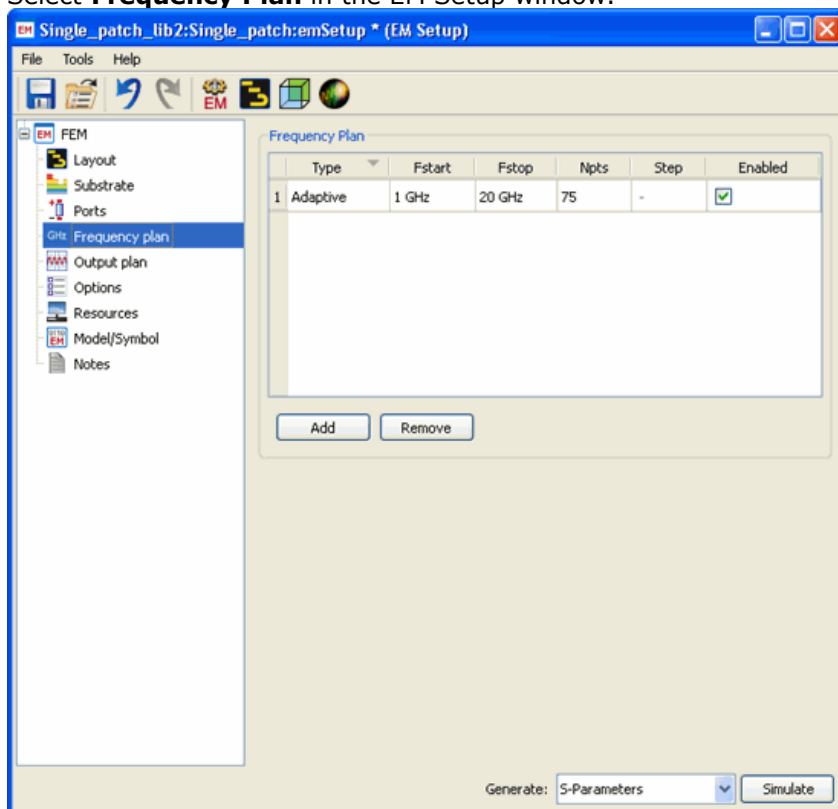
To add a new frequency plan, you need to specify the following parameters in the EM Setup Window:

Column	Description
Type	Specifies the type of frequency: <ul style="list-style-type: none"> • Adaptive: It is the preferred sweep type. It uses a fast and 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. • Linear: It 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. • Log: It 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. • Single: It simulates at a single frequency point. Type the value in the Frequency field and select the units.
Fstart	Specifies the frequency start value. You can specify the value in Hz, KHz, MHz, GHz, or THz units.
Fstop	Specifies the frequency stop value. You can specify the value in Hz, KHz, MHz, GHz, or THz units.
Npts	Specifies the number of frequencies to simulate. For Adaptive this is the maximum number of samples to use.
Step	Specifies the step value. You can specify the value in Hz, KHz, MHz, GHz, or THz units.
Enabled	Allows you to enable the frequency plan.

Adding a Frequency Plan

To add a frequency plan:

1. Choose **EM > Simulation Setup** to display the **EM Setup** window.
2. Select **Frequency Plan** in the EM Setup window.



3. To add a new frequency plan, click **Add**.
4. Select a sweep from the **Type** column.
5. Specify the frequency start value in the **Fstart** column.
6. Specify the frequency stop value in the **Fstop** column.
7. Specify the value of N points in the **Npts** column.
8. Specify the step value in the **Step** column.
9. Select the check box in the **Enabled** column to enable the frequency plan.
10. Repeat the preceding steps to insert additional frequency plans.

**Note**

In a simulation, the frequency plans are executed in the order they appear in this list with the exception that the Adaptive sweep only starts if all others have been executed

Removing a Frequency Plan

To delete a frequency plan:

1. Choose **EM > Simulation Setup**.
2. Select **Frequency Plan** in the **EM Setup** window.
3. Select the required frequency plan.
4. Click **Remove**.

Defining the Output Plan

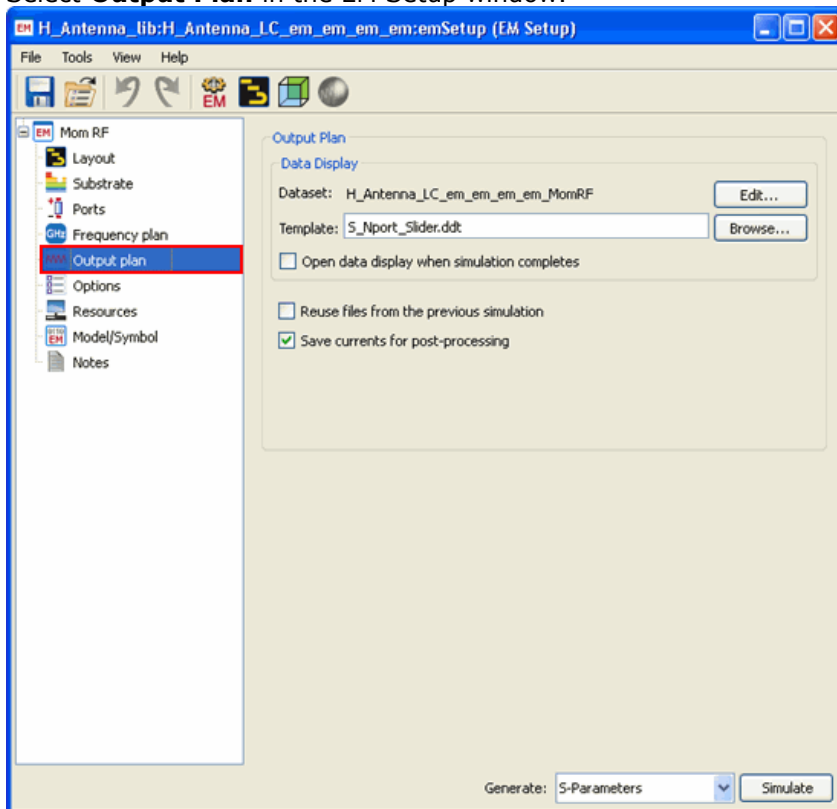
Using the **EM Setup** window, you can specify the following data display settings for your EM simulations.

Field	Description
Dataset	Specifies the dataset used for storing results. You can also click Edit to select the required dataset.
Template	Specifies the template to be used for displaying the data. You can also click Browse to select the required template.
Open data display when simulation completes	Automatically opens the data display window on the completion of a simulation.
Reuse files from previous simulation	Allows you to use the files that were used during previous simulations.
Save currents post-processing	Allows you to save the current settings for post processing.

Creating an Output Plan

To create an output plan:

1. Choose **EM > Simulation Setup** to display the **EM Setup** window.
2. Select **Output Plan** in the EM Setup window.



3. The default dataset name is displayed. To specify a new dataset, click **Edit**. For more information, see [Selecting Output Dataset](#).

- Specify the template to be used for displaying the data in the **Template** text box. You can also click **Browse** to select the required template.
- Select **Open data display when simulation completes**, if you want to open the data display window automatically.
- Select **Reuse files from previous simulation** if you want to use the previous simulation files.
- Select **Save currents for post-processing**.

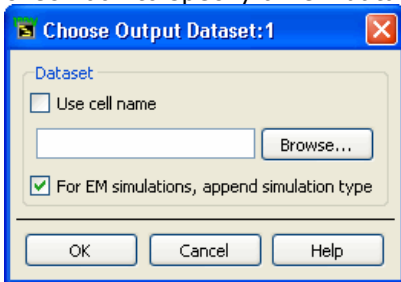
Selecting Output Dataset

When you run an EM simulation, the results are stored in a dataset. This dataset is then used by the Data Display for viewing results. You can select the name and location of the dataset you want to use for a simulation. The default value is the cell name. Datasets are stored in a workspace or data subdirectory. You can also select a dataset name from the existing dataset file names. When using an existing name, new results overwrite existing contents. This means that you should provide unique names for the datasets that will be generated from different cells in the same workspace directory.

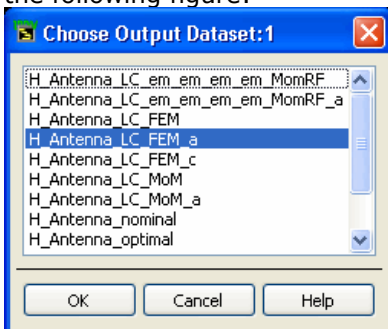
Specifying a Dataset

To select an output dataset:

- Choose **EM > Choose Output Dataset**.
- By default, the **Use cell name** check box is selected. Remove selection from this check box to specify a new dataset.



- Type a name for the dataset where you want simulation data to be saved. Click **Browse** to view the existing dataset names in the current workspace, as shown in the following figure:



- Select the required dataset and click **OK**.
- The **For EM simulations, append simulation type** check box is selected by default.
- Click **OK**.

Defining Simulation Options

You can specify engine-specific options such as, substrate overrides, preprocessor, and mesh for EM simulations. In this section, you will learn:

- [Creating Simulation Options Set](#)
- [Specifying Simulation Options](#)

Creating Simulation Options Set

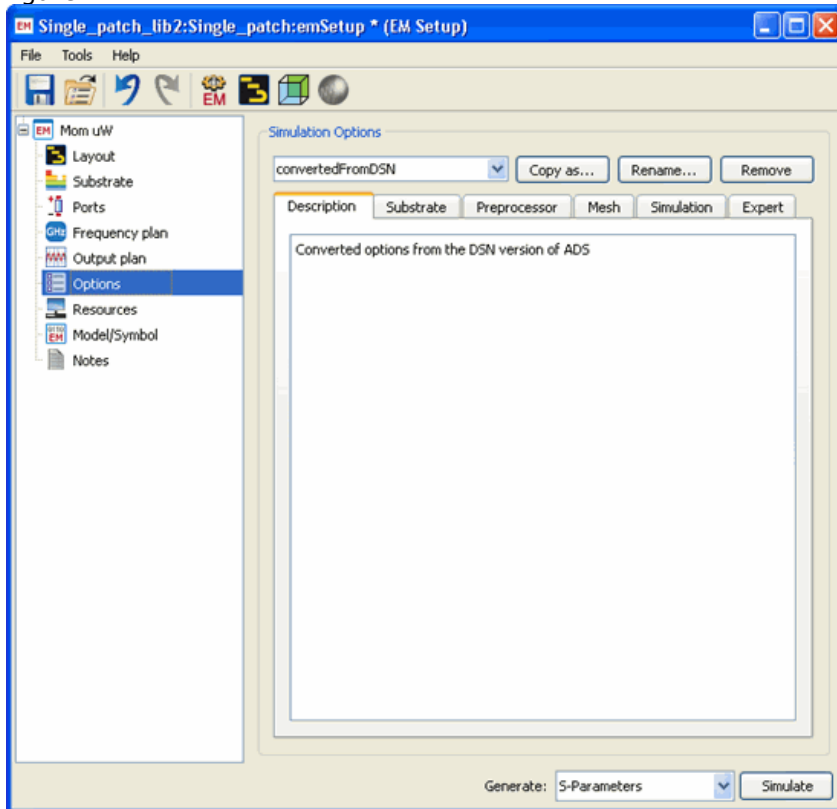
You can either use a predefined set of simulation options or create a new set in

Simulation Options of the EM Setup window. You can perform the following operations to edit the list of simulation sets:

- Creating a copy of the option set: You can create a copy of the predefined simulation set by clicking **Copy as**. Specify a name for the copied option. You can now define new simulation options in this set.
- Renaming an option set: You can change the name of your simulation set by clicking **Rename**.
- Removing an option set: You can delete a simulation set by clicking **Remove**. After you remove a simulation set, the simulation set value changes to **Default** in the drop-down list.

To create a new set of simulation options:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Options** in the left pane of the EM Setup window, as shown in the following figure:



3. Click **Create a copy**.
4. Specify a name for the copied option.
5. Define new simulation options in the EM Setup window.

Specifying Simulation Options

You can specify preprocessor, mesh, and simulation settings by using the Simulation Options dialog box. It consists of the following tabs:

- Description
- Substrate
- Preprocessor
- Mesh
- Simulation
- Expert

In the Description tab, you can add information about the simulation options. The following sections provide information about the other tabs available in the EM Setup

window:

- *Overriding Substrate Definition for Momentum* (em)
- *Overriding Substrate Definition for FEM* (em)
- *Defining Preprocessor Settings* (em)
- *Defining Mesh Settings for Momentum* (em)
- *Defining Mesh Settings for FEM* (em)
- *Defining Simulation Settings* (em)
- *Defining Expert Options for Momentum* (em)

Overriding Substrate Definition for Momentum

You can either choose the options specified in the substrate or override the substrate definition. You can override the substrate definition specified earlier by applying the substrate overrides either globally or on selected layers.

In this section, you will learn about:

- [Specifying Global Substrate Options](#)
- [Specifying Layer-specific Substrate Options](#)

Specifying Global Substrate Options

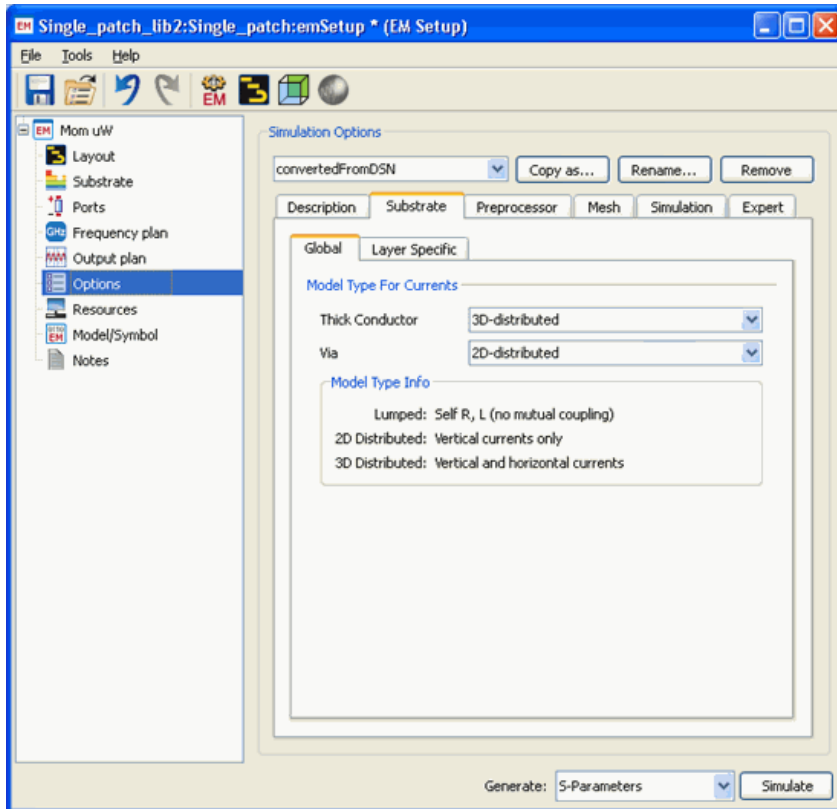
Using the Global tab, you can override the substrate definition on all the layers. This tab enable you to specify the modeling type for thick conductors and vias. To identify the conductive areas of a layout layer, you can define the model type for current. The current flow at the surface is modeled depending on the simulation model chosen. You can choose one of the following options:

- **Lumped**: A lumped element network represents the via connection. The actual via mesh is ignored and a single lumped branch, connecting the largest bottom and top cell covering the via object (or touching in case of an open polyline via), models the via connection.
- **2D Distributed**: You can specify this value for both thick conductor and vias simulation models. For a 2D distributed value, a distributed model for a object is created based on the surface mesh. At the vertical side walls, the vertical component of the current is modeled only, using vertical oriented rooftop functions. Hence, the modeled current distribution of the vertical component varies both in the cross-section and in the vertical direction (2D current distribution).
- **3D Distributed**: You can specify this value for both thick conductor and vias simulation models. For a 3D distributed value, a distributed model for a object is created based on its surface mesh. At the vertical side walls, both the horizontal and vertical components of the current are modeled, using horizontal and vertical oriented rooftop functions. This yields a complete 3D current modeling of the objects.

Defining Engine-specific Substrate Options

To override the substrate definition globally:

1. Choose **EM> Simulation Setup** to open EM Setup window
2. Select **Options** in the left pane of the EM Setup window.
3. Select the *Substrates* tab.
4. Ensure that the *Global* tab is displayed, as shown in the following figure:

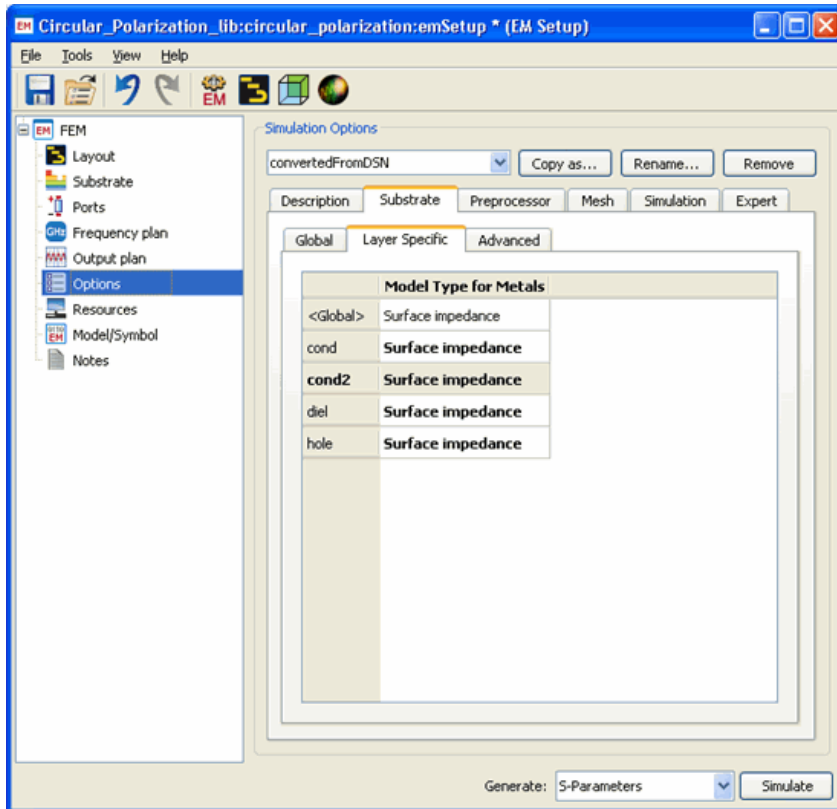


5. Select a value from the **Thick Conductor** drop-down list.
6. Select a value from the **Via** drop-down list.

Specifying Layer-specific Substrate Options

To override substrate definition for a specific layer:

1. Select **EM> Simulation Setup** to open EM Setup window.
2. Select **Options** in the left pane of the EM Setup window.
3. Select the *Substrates* tab.
4. Select the *Layer Specific* tab, as shown in the following figure.



5. Select a value from the **Model Type for Currents Thick Conductor** drop-down list in the required layer.
6. Select a value from the **Model Type for Via** drop-down list in the required layer.

Defining Preprocessor Settings

The EM Preprocessor prepares a layout for electromagnetic simulation. It converts a given layout into a new layout by applying the specified transformations. In this section, you will learn about:

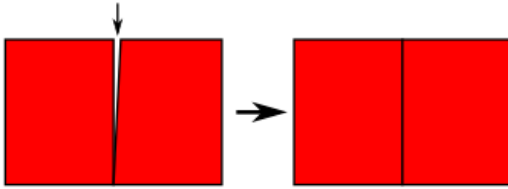
- [EM Simulation Transformations Overview](#)
- [Describing the Preprocessor Parameters](#)
- [Guidelines for Defining Preprocessor Settings](#)
- [Specifying the Preprocessor Settings](#)

EM Simulation Transformations Overview

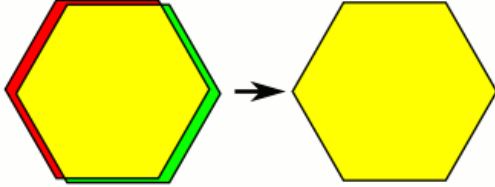
The duration of an EM simulation with a conformal mesh is dependent on the number of edges in the mesh. This relation is superlinear. If the number of edges increases, the simulation time increases more than proportionally. Reducing the complexity of the mesh improves performance, especially for larger designs. Besides the complexity of the mesh, the quality of the mesh is also important. A mesh containing triangles with acute angles can lead to a reduced accuracy of the solution or even cause numerical issues while solving.

Using the preprocessor settings, you can define the following optional transformations that can alter the input layout geometry to improve the quality and complexity reduction of a generated mesh:

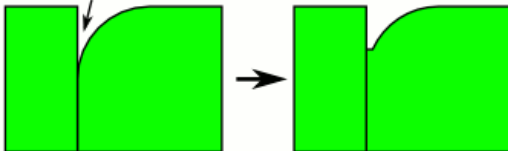
- **Layout healing:** Layout healing merges vertices and edges in close proximity. It corrects numerical issues in the layout that obstruct the creation of a high-quality conformal mesh. Layout healing:
 - Closes unintentional gaps between adjacent shapes that are frequently created by layout generators and import or export processes, as shown in the following figure:



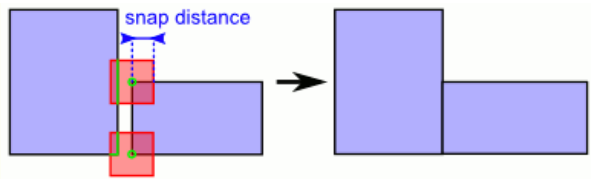
- Ensures that stacked vias have only one joint boundary instead of two nearly identical boundaries, as shown in the following figure:



- Eliminates acute angles, as shown in the following figure:

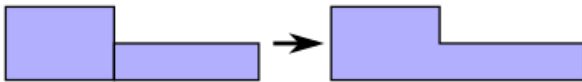


- Defining snap distance: Within the same substrate layer, the vertices and edges that are closer together than the snap distance are merged. The distances for layout healing are calculated using the L_∞ metric. In this metric, the distance between two points is the largest absolute difference between both horizontal and between both vertical components. Any set of points equidistant to a given point is an axis-aligned square. Distances calculated using the L_∞ metric always amount to a fraction between $1/\sqrt{2}$ and 1 of the Euclidean distance.

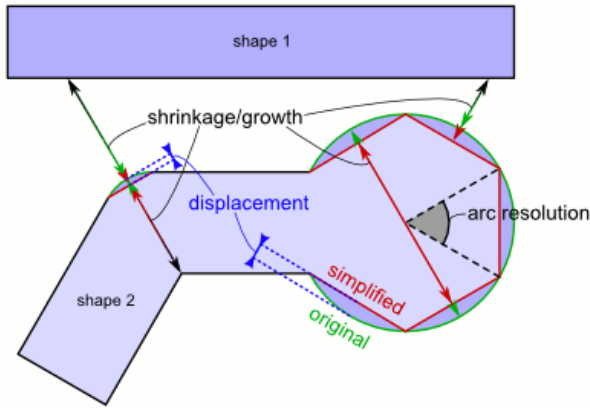


The layout healing algorithm starts from an empty layout and inserts all edges of all polygons and polylines one by one. The stack of substrate layers is processed from top to bottom. Within one substrate layer, longer edges are inserted before shorter edges. Newly inserted edges merge with already present features on the same substrate layer within the snap distance. You can define automatic snap distance that enables the preprocessor to automatically provide a snap distance. Layout healing is enabled in this mode by default.

- **Merge shapes touching each other where possible:** This EM transformation merge the shapes that are touching each other in the design. In this type of transformation, shared edges in the are not preserved. The following figure displays an example of such transformation:



- **Layout simplification:** This EM transformation replaces polygons by nearly identical polygons with less vertices. The main purpose is to generate a conformal mesh containing minimum number of edges. Layout simplification also creates higher quality meshes as it removes many small segments.



Note
Layout healing and simplification can be applied in isolation or together.

Describing the Preprocessor Parameters

The following table describes the preprocessor parameters:

Option	Description
Heal the layout	This option enables the layout healing feature. It merges the vertices and edges closer than an automatically determined or a user-specified snap distance. Detailed information about the algorithm is in the section dedicated to layout healing. It is recommended to specify automatic snap distance setting healing. If an aggressive healing approach is required, choose a value between 1-10% of the width of the smallest trace or the tightest clearance.
Merge shapes touching each other where possible	This option controls whether edges shared between adjacent shapes need to be preserved in the result.
Simplify the Layout	This option toggles the usage of the layout simplification feature. In this section, you can specify the following parameters: <ul style="list-style-type: none"> Constraints (upper limits): <ul style="list-style-type: none"> Upper displacement limit (layout simplification): This value is around one third of the width of the smallest trace in the layout. Upper shrinkage/growth limit (layout simplification): The reasonable values are in the range of 5 to 15%. For specific requirements, such as via simplification, the values up to 30-50% can be acceptable (this reduces circles to squares or triangles). Use different constraints for vias: The vias may need to be simplified more than the rest of the layout. You can specify a different set of constraints for vias: Displacement and Shrinkage/growth limit
Save Preprocessor Messages as a DRC Result	This option enables you to display the location-bound information and warning messages in the layout using the DRC Results Viewer. The name of the DRC Result and the ADS layer where errors are shown can be configured.

Guidelines for Defining Preprocessor Settings

While defining the preprocessor parameters, you can refer the following guidelines:

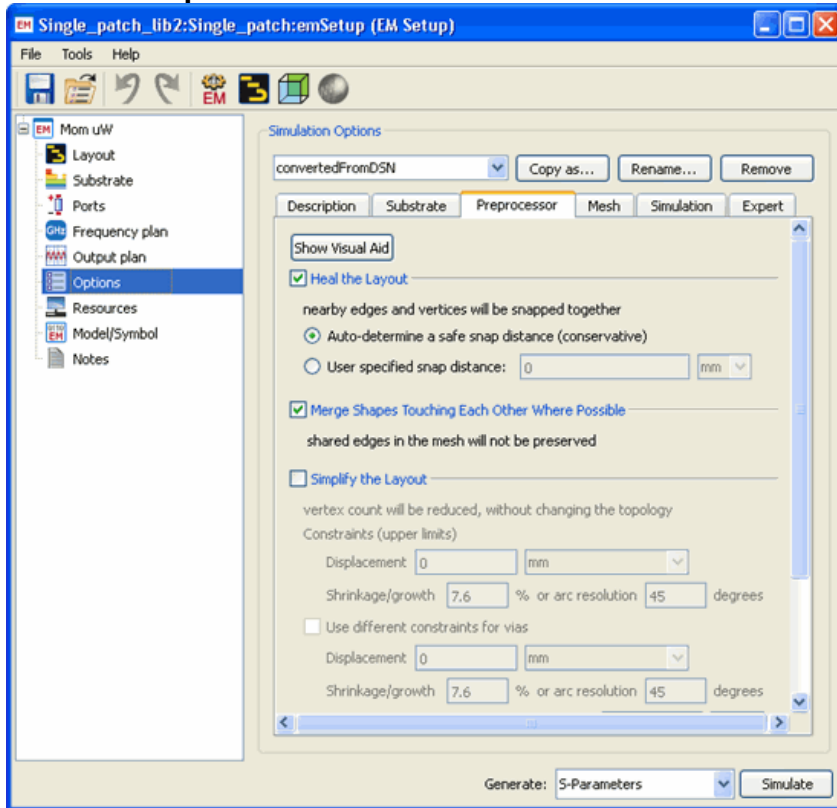
- Layout healing can change the layout topology. Therefore, the healing algorithm is designed to make small changes to the layout topology. It is important to keep the snap distance below the minimal clearance between different layout nets, otherwise short circuits might appear. The snap distance must also be smaller than the width of thin layout traces, otherwise these traces will disappear.
- Specify a snap distance at least 10 times smaller than the minimal clearance and trace width.
- Whenever layout healing performs an actual layout change, a warning is displayed in the status window. The details of the change are written to the DRC Result or the layout processing report (dependent on the setup option Save preprocessor messages as a DRC Result). Consult the DRC result or the layout processing report to verify these changes.

- Do not use layout healing as a polygon simplification tool: It recommended to avoid choosing a large snap distance. For this purpose, the layout simplification feature provides a far better and safe alternative.

Specifying the Preprocessor Settings

To specify preprocessor settings:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Options** in the EM Setup window.
3. Select the **Preprocessor** tab.



Note

You can view a graphical representation of the available optional transformations by clicking **Visual Aid**.

4. Enable the **Heal the layout** option and specify the required settings.
5. Enable the **Merge shapes touching each other where possible** option to preserve the edges shared between adjacent shapes.
6. Enable the **Simplify the layout** option to generate a conformal mesh containing minimum number of edges. Specify the required settings in this section.
7. Enable the **Save Preprocessor Messages as a DRC Result** to display the location-bound information and warning messages. Specify the required settings in this section.

Defining Mesh Settings for Momentum

In the Momentum solution approach, any electromagnetic configuration of a 3D design can be represented without loss of generality as a voltage/current distribution on its surfaces. This continuous distribution is approximated by a finite number of unknowns using a mesh. This mesh is a grid-like pattern of triangular and/or quadrilateral cells tessellating all surfaces.

In an S-parameter simulation, coupling effects between cells are being calculated using the Green function database. The currents passing through mesh edges are then derived from mesh cell voltages to match the calculated coupling effects. From these results, S-parameters are calculated for the circuit.

The quality of the mesh determines the quality of a solution. The cells need to be small enough in order to capture real world spatial variations of the current/voltage distribution. However, covering low-variability regions with larger cells is necessary to keep the problem size within reasonable limits.

The default configuration of the mesh generator produces suitable meshes in the vast majority of situations. When desired, this configuration can be adjusted through a number of user-adjustable parameters. The default parameters are used automatically. The mesh is normally computed as part of the simulation process; no specific user intervention is required. It is also possible to explicitly preview the mesh without simulating in order to assess its quality.

In this section, you will learn about:

- [Defining a Mesh](#)
- [Defining Global Mesh Parameters](#)
- [Specifying Layer-specific Mesh Options](#)
- [Specifying Shape-specific Mesh Options](#)
- [Adjusting Mesh Density](#)
- [Effect of Mesh Reduction on Simulation Accuracy](#)
- [Setting the Width of the Edge Mesh](#)
- [About the Transmission Line Mesh](#)
- [Processing Object Overlap](#)
- [Guidelines for Meshing](#)

Defining a Mesh

You can define mesh parameters for the following areas:

- Entire circuit
- Objects on a layout layer
- Shape of an object

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

Defining Global Mesh Parameters

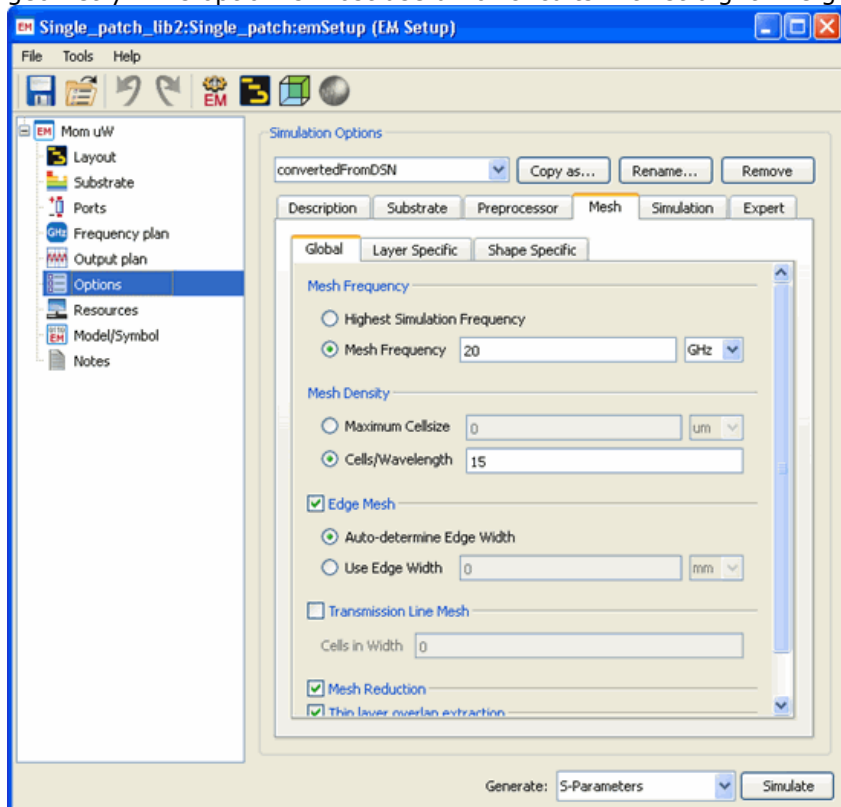
Global mesh parameters affect the entire circuit. To set up global parameters:

1. Select **EM> Simulation Setup** to open the **EM Setup** window.
2. Select **Momentum** as the EM simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Mesh** tab. By default, the Global parameters are displayed.
5. Specify the mesh frequency. You can select one of the following options:
 - Select **Highest Simulation Frequency**. Set the value of the mesh frequency to the highest frequency that will be simulated. For more information, refer to [Adjusting Mesh Density](#).
 - Type the mesh frequency in the **Mesh Frequency** field and select the units. The wavelength of this frequency is used to determine the density of the mesh.
6. Specify the mesh density. You can select one of the following options:
 - Specify the **Maximum Cellsize** value and select the units.
 - Enter **Number of Cells per Wavelength**. This value will also be used to determine the density of the mesh. For more information, refer to [Adjusting Mesh Density](#).
7. Enable **Edge Mesh** to add a relatively dense mesh along the edges of objects. Due to the flow of current along the edges of objects, the edge mesh can improve the accuracy and speed of a simulation. An edge mesh will similarly be created on thick conductor side walls. You can select one of the following options:
 - If you want the edge mesh to be sized automatically, select **Auto-determine Edge Width**.
 - To specify the edge width, select **Edge Width** and select the units. For more information about the edge mesh, refer to [About the Edge Mesh](#).

**Note**

An edge mesh that is specified with a width larger than the cell size set by the *wavelength/number_of_cells_wavelength*, will be ignored. This is because such edge meshes would be very inefficient. However, if these edge mesh values must be used, you can decrease the *number_of_cells_wavelength*, which is specified in the *Number of Cells per Wavelength* field.

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



9. 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](#).
10. Enable **Mesh reduction** to obtain an optimal mesh with fewer small cells and an improved memory usage and simulation time. For more information on Mesh Reduction, refer to *Effect of mesh reduction on simulation accuracy*.
11. Enable **Thin layer overlap extraction** 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](#).

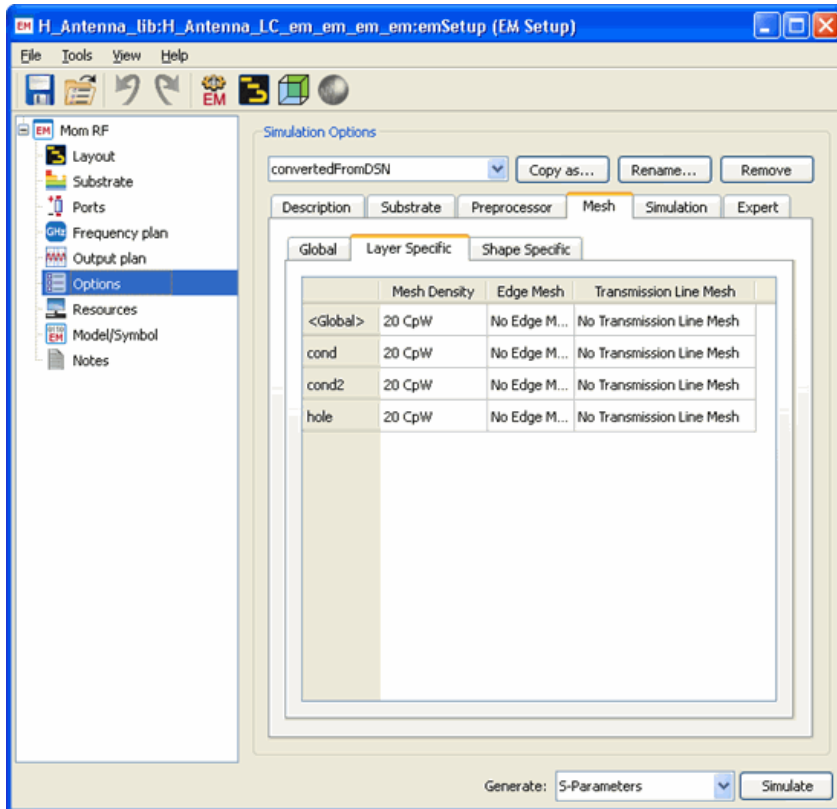
**Note**

This should always be enabled for modeling thin layer capacitors.

Specifying Layer-specific Mesh Options

You can use the Layer Specific tab to define the mesh options for a specific layer. Perform the following steps:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Momentum** as the EM simulator.
3. Select **Options** in the EM Setup window.
4. Select the **Mesh** tab to display the mesh options for Momentum.
5. Select the **Layer Specific** subtab to specify settings at the layer level, as shown in the following figure:

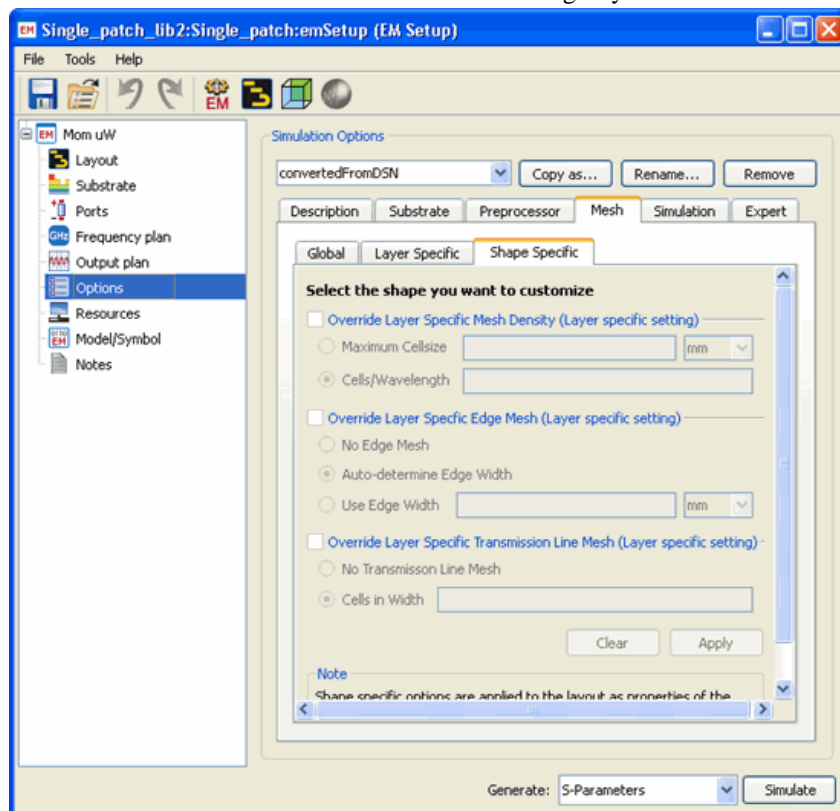


6. Select a value from the **Mesh Density** drop-down list in the required layer.
7. Select a value from the **Edge Mesh Width** drop-down list in the required layer.
8. Select a value from the **Transmission Line Mesh Width** drop-down list in the required layer.

Specifying Shape-specific Mesh Options

You can customize the selected shape by using the Shape Specific tab. Perform the following steps:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Momentum** as the EM simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Mesh** tab to display the mesh options for Momentum.
5. Select the **Shape Specific** subtab to specify settings at the shape level, as shown below.



Note

To enable the shape-specific options, you need to select a shape in the layout before specifying the options.

6. Enable the **Override Layer Specific Mesh Density** section. You can select one of the following options:
 - Select **Maximum Cellsize**. Set the maximum value of the mesh density that will be simulated. For more information, refer to [Adjusting Mesh Density](#).
 - Type the number of cells per wavelength in the **Cells/Wavelength** text box.
7. Enable **Override Layer Specific Edge Mesh**. You can select one of the following options:
 - No Edge Mesh
 - Auto-determine Edge Width
 - Use Edge Width
8. Enable the **Override Layer Specific Transmission Line Mesh** section. You can select one of the following options:
 - No Transmission Line Mesh
 - Cells in Width

Adjusting Mesh Density

A minimal mesh density is guaranteed by enforcing all cells to be smaller than a certain maximum cell size limit. When this maximum cell size is not stated explicitly, it is calculated from the mesh frequency and the number of cells per wavelength. The cells need to be small enough in order to capture real world spatial variations of the longitudinal current distribution. The spatial extent of waves is shortest at the highest frequency. Consider wave phenomena to be described by a sinusoid of a certain wavelength. Each mesh cell is able to approximate only the portion of the sinusoid it covers with a linear function. The accuracy of this piecewise linear fit improves as the number of cells per wavelength increases. For example, if you use 30 cells per wavelength, the maximum deviation between the sinusoid and the linear approximation is about 1 percent.

Increasing either the mesh frequency or the number of cells per wavelength makes the mesh denser. When a denser mesh is desired, it is generally better to increase the number of cells per wavelength than the mesh frequency. The optimal value for the mesh frequency is the highest simulation frequency that will be eventually used. In fact, an

increase of the mesh frequency may trigger a recalculation or extension of the Green's functions database.

Note

Only the *maximal* cell size is enforced and not the *minimal* cell size. A polygon with N sides can be tessellated with no less than $N-2$ triangles (or half as many quadrilaterals). This imposes a minimal cell size dependent on the level of detail of the geometry.

Sometimes the geometric detail is much smaller than a fraction of the wavelength (typically when including integrated circuits in the simulation). In this case it is advised to choose an explicit maximal cell size not larger than four times the regular trace width. Otherwise the mesh may contain cells with a very large aspect ratio that cannot properly model the actual current flow.

Effect of Mesh Reduction on Simulation Accuracy

Mesh reduction is a technology that aims at removing mesh complexity originating from the meshing of geometrically complex shapes. In a normal situation, the user specifies the mesh density (the number of cells per wavelength) that needs to be used in the simulation in order to obtain a specific accuracy. However, due to geometrical constraints, the mesher can be forced to use more cells than strictly needed by the wavelength criterion. In this case, the mesh introduces "redundant" degrees of freedom in the solution process ("redundant" with respect to the electromagnetic behavior). Mesh reduction is a technology that automatically removes these redundant degrees of freedom, prior to the solution of the problem. Hence, it should have a negligible impact on the accuracy of the results.

Warning

Mesh reduction currently ignores layer or shape specific mesh density settings. Disable mesh reduction if you have other than global mesh settings.

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. The following applications use the edge mesh:

- Tightly coupled lines
- Patch antennas
- Resonant circuits
- Delay lines
- Hairpin filters

Note

Edge mesh is available at all levels of meshing: global, layout layer, and object meshing.

Setting the Width of the Edge Mesh

When using an edge mesh, choose **Auto-determine Edge Width** to let Momentum determine an appropriate width. Alternatively, enter an explicit width. To this end you can use **Insert > Measure...** to measure distances on the layout.

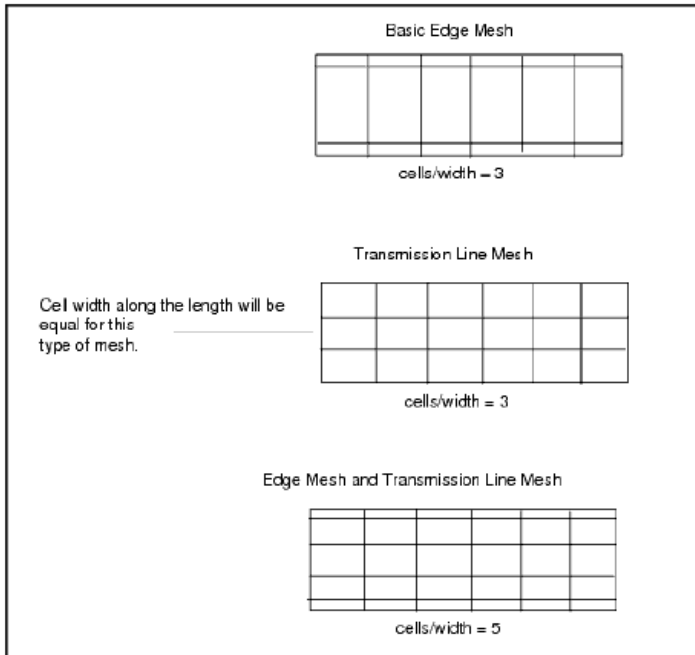
About the Transmission Line Mesh

Use the transmission line mesh when you want to explicitly specify the number of cells in the direction perpendicular to the current flow. This has no effect on the mesh density in the direction of the current flow.

This setting can be used to improve the accuracy of a simulation with a coarse mesh density. For example, the simulation results for a single transmission line with one or two cells across its width will be equal. If you have coupled lines, the results will differ.

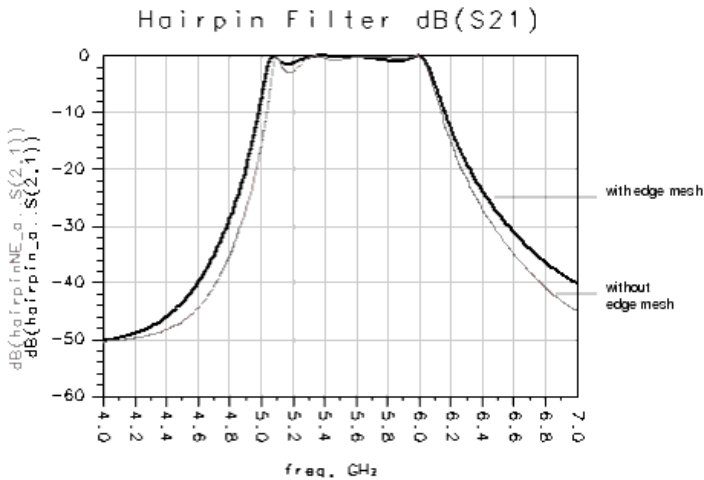
Combining the Edge and Transmission Line Meshes

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.



The edge and transmission line mesh settings affect the transverse current distribution. The accuracy of transverse current approximations improves as the number of cells specified using edge mesh and transmission line mesh increases.

The plot below illustrates the hairpin example simulated with and without an 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.

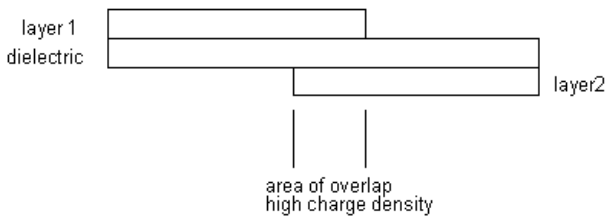


Processing Object Overlap

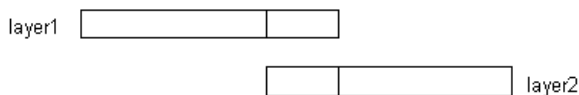
The parameter *Thin layer overlap extraction* should be used in designs that have thin layers in-between overlapping objects. It is possible that mesh cells are generated that cross the overlap region and this is not desirable. Typical situations where this is important are:

- Parallel plate capacitor: The charge will be accumulated where the plates overlap. Just next to that overlap area, charge will be much lower. As charge is assumed to be uniformly distributed on a mesh cell, when such cell crosses the plate overlap area, the charge will be spread out too much, resulting in a less accurate capacitance extraction.
- Transmission line above a finite ground plane: The return current in the ground plane will concentrate just under the transmission line at higher frequencies. Large mesh cells under the transmission line do not allow to describe that return current flow accurately. The overlap extraction provides a way to force a mesh that follows the shape of the transmission line allowing a more accurate approximation of the return current flow.

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.

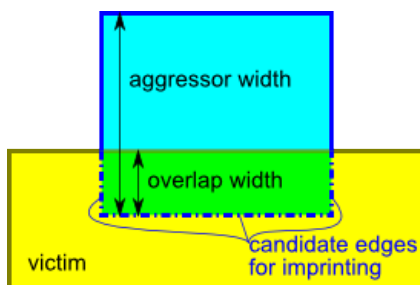


mesh cells with thin layer overlap extraction ON



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.

Imprinting all possible overlaps can lead to a very dense mesh. Therefore, some rules are in place that determine whether any particular overlap instance is sufficiently important to warrant imprinting. To this end, every pair of overlapping shapes is analyzed. Only direct overlap (not shielded by a third shape) is being considered. Both shapes take turns being the aggressor and the victim. However, it needs to be decided whether or not to imprint the boundary of the aggressor onto the victim, as shown in the following figure:



The following analysis determines whether the overlap should be imprinted:

- **Capacitive overlap**

- **Relevance:** To determine whether this is a relevant capacitive overlap region, the width of the overlap must be large in comparison to the distance between the planes containing both shapes. The parameter `oe_capacitor_width_threshold` determines the threshold for the ratio between

the overlap width and the effective z-plane distance. The effective z-plane distance is the z-plane distance divided by the relative static permittivity.

Above this threshold, capacitive overlap is considered to be relevant.

- **Accuracy:** Edges of a relevant capacitive overlap region are imprinted according to an accuracy specification. The parameter `oe_capacitor_accuracy` determines the threshold for the ratio between the effective maximal cell size of the victim and the effective overlap width. Above this threshold, edges of relevant capacitive overlap regions are imprinted.

The *effective* maximal cell size is the maximal cell size decreased with the z-plane distance times `oe_maximal_cell_size_height_shrink_factor`.

Likewise, the *effective* overlap width is the overlap width increased with the z-plane distance times `oe_overlap_width_height_grow_factor`.

- **Inductive overlap**

- **Relevance:** Only sufficiently long boundaries are considered for inductive overlap. The length of the boundary must exceed `inductor_length_threshold` times the minimal wavelength.

In addition, the ratio of the overlap width to the aggressor width must exceed `oe_inductor_width_threshold`.

- **Accuracy:** Edges of a relevant inductive overlap region are imprinted according to an accuracy specification. The parameter `oe_inductor_accuracy` determines the threshold for the ratio between the effective maximal cell size of the victim and the effective overlap width. Above this threshold, edges of relevant inductive overlap regions are imprinted.

When an edge is being imprinted, it is allowed to snap to existing geometry if the snap distance is small enough to uphold the specified accuracy. This snap distance is additionally limited to `oe_maximal_overlap_width_change` times the overlap width.

The final Boolean parameter `oe_pedantic` can be used to impose the most strict overlap extraction rules. When enabled, it forces all other parameter values to zero.

You can set all parameters by selecting **Options > Expert** in the EM Setup window. When unset, corresponding `momentum.cfg` configuration variables are consulted for default values. Finally, built-in default values are being used. These values guarantee proper overlap extraction across a wide range of designs.

The following table summarizes all overlap extraction related parameters. Note that all numerical parameters are dimensionless quantities.

Options > Expert key in EM Setup (highest priority)	\$HOME/hpeesof/config/momentum.cfg variable name	permissible values	built-in default value
<code>oe_capacitor_width_threshold</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_CAPACITOR_WIDTH_THRESHOLD</code>	<i>not negative</i>	15
<code>oe_capacitor_accuracy</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_CAPACITOR_ACCURACY</code>	<i>not negative</i>	0.05
<code>oe_inductor_length_threshold</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_INDUCTOR_LENGTH_THRESHOLD</code>	<i>not negative</i>	0.1
<code>oe_inductor_width_threshold</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_INDUCTOR_WIDTH_THRESHOLD</code>	<i>not negative</i>	0.5
<code>oe_inductor_accuracy</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_INDUCTOR_ACCURACY</code>	<i>not negative</i>	0.5
<code>oe_maximal_cell_size_height_shrink_factor</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_HEIGHT_SHRINK_FACTOR</code>	<i>not negative</i>	0.88
<code>oe_overlap_width_height_grow_factor</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_HEIGHT_GROW_FACTOR</code>	<i>not negative</i>	0.88
<code>oe_maximal_overlap_width_change</code>	<code>MOM_MESHER_OVERLAP_EXTRACTION_DEFAULT_MAXIMAL_OVERLAP_WIDTH_CHANGE</code>	<i>not negative</i>	0.5
<code>oe_pedantic</code>	<code>EMPREP_PEDANTIC_OVERLAP_EXTRACTION</code>	ON or OFF	OFF

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 design, you can use any combination of the four types of mesh control. In general, greater mesh density provides better accuracy. However, a greater density increases the computation time (more cells to solve). You can refer to the following guidelines:

- **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.
- **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 Resolution (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 required, to reduce the number of triangles. However, this alters the geometry of a circuit.

Defining Mesh Settings for FEM

An FEM simulation mesh is a sub-division of the entire 3D problem domain into a set of tetrahedra (or cells). 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 electric within each cell and identify any coupling effects in the circuit during simulation. From these calculations, S-parameters are then calculated for the circuit.

FEM Simulator implements an adaptive mesh algorithm, where an initial mesh is generated and the electric fields (and S-parameters) are computed on that initial mesh for a single frequency. An error estimate is generated for each tetrahedron. The tetrahedra with the largest estimated error are refined to create a new mesh on which the electric fields (and S-parameters) are computed. The S-parameters from consecutive meshes are compared. If the S-parameters do not change significantly, then electric fields (and S-parameters) are computed for all the requested frequencies. If the S-parameters do change significantly, then new error estimates are computed, a new mesh is generated and new electric fields (and S-parameters) are computed.

In this section, you will learn about:

- [Defining FEM Mesh Options](#)
- [Selecting Refinement Frequency](#)
- [Defining Advanced Mesh Options](#)
- [Specifying FEM Mesh Parameters](#)

Defining FEM Mesh Options

The following table describes the mesh options available for FEM Simulations:

Option	Description
Delta Error	Global Delta S-parameter sets a value that is applied to all S-parameters in the solution. Enter a value in the Delta Error field. This is the allowable change in the magnitude of the vector difference for all S-parameters for at least two consecutive refinement passes.
Minimum Number of Adaptive Passes	Specifies the number of passes that will be solved before testing for the delta error. This can be particularly useful when running a parameter sweep to ensure that the meshes generated for all parameter values have a comparable accuracy. Typically, a value between 1 and 5 is recommended.
Maximum Number of Adaptive Passes	Specifies the maximum number of passes to be attempted. If the number of refinement passes entered is reached before the delta error criteria is met, the refinement process will end, based upon this limit. The number of passes from all prior simulations is also displayed. Typically, a value between 10 and 20 is recommended.
Refine at a Specific Frequency	Enables you to change the frequency at which the mesh refinement is performed. By default, the mesh refinement is performed at the highest frequency specified in the simulation.
Automatically open mesh convergence plot	Opens the mesh convergence plot automatically.
Number of bondwire segments	Enables you to specify the number of bondwire segments.

Selecting Refinement Frequency

The following table describes the advanced mesh options available for FEM Simulations:

- Multiple fixed refinement frequencies: This strategy computes the fewest frequency points during the adaptive refinement stage and therefore will usually lead to the fastest simulations. This strategy is recommended for:
 - Electrically short transition structures. These can include matching networks, packages, impedance transformations, etc. If the transmission parameters accumulate less than 180 degrees of phase, then it is sufficient to refine only at the highest frequency.
 - Designs where you know the frequencies of the critical features. These can include dual (or multi-) band antennas, band pass filters etc. For these structures set the refinement frequencies to the frequencies of most interest (i.e. in each antenna frequency band, at the low end and high end of the band pass filter). In many cases it is advisable to also include the maximum frequency of the simulation as a refinement frequency.
 - Initial analysis of a structure where speed is preferred over accuracy.
- Refinement frequencies chosen automatically after the initial pass: This strategy is a compromise between manually selecting the refinement frequencies and letting the FEM simulator automatically select the refinement frequencies at each adaptive pass. This strategy is recommended when you do not have a good idea a priori where the important characteristics will occur in your frequency but you know the frequency transitions are not extremely sharp (i.e. are not high Q resonances).
- Refinement frequencies chosen automatically after each pass: This is the automated strategy and tends to generate accurate results, but it also tends to result in the longest simulation times. This strategy is recommended:
 - For structures with high Q resonances.
 - During final analysis of a structure where accuracy is preferred over fast simulations.

Defining Advanced Mesh Options

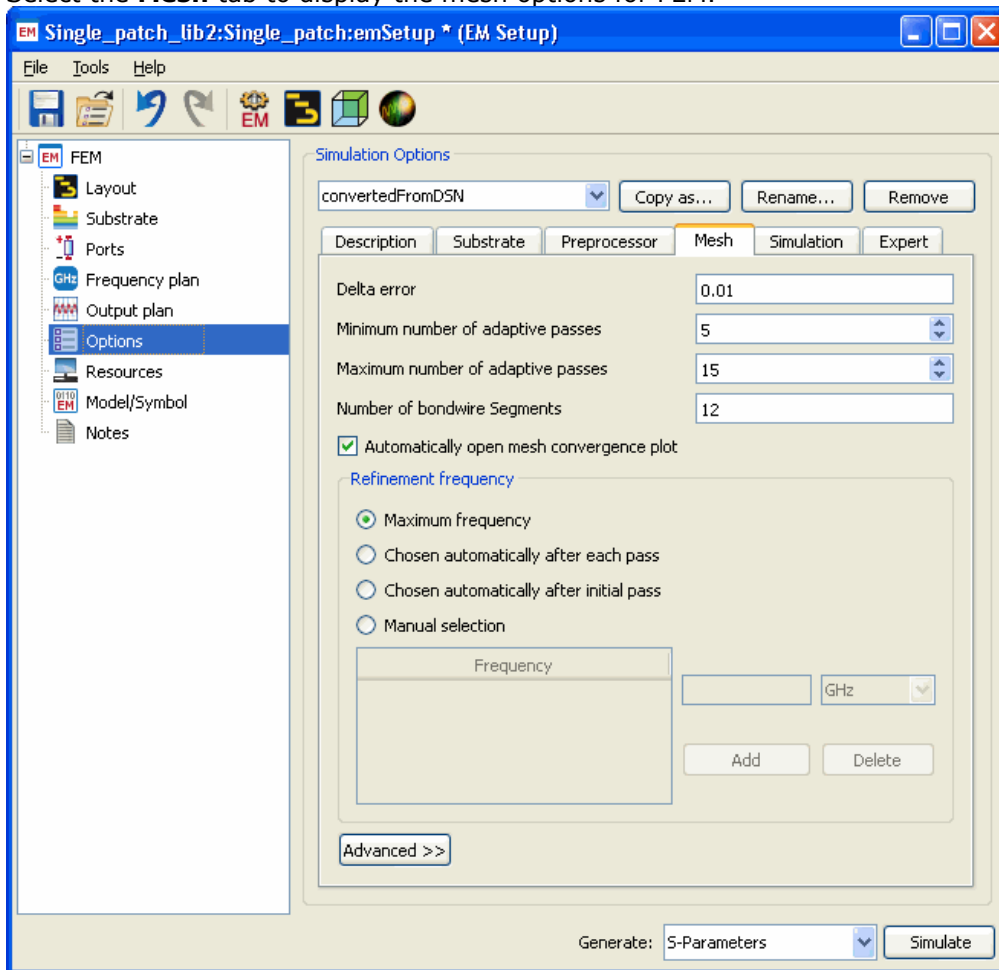
The following table describes the advanced mesh options available for FEM Simulations:

Option	Description
Consecutive passes of Delta Error Required	Specifies the number of consecutive passes that must meet or exceed the delta error. If you have concerns about convergence, you can increase this value, otherwise, use the default value of 1.
Target mesh growth	Specifies the required mesh growth
Use initial target mesh size	Specifies the initial mesh size and units. You can also determine the initial mesh size automatically.
Use initial minimum mesh size	Specifies the initial minimum mesh size and units.
Conductor edge mesh length	Specifies the edge mesh length of a conductor.
Conductor vertex mesh length	Specifies the vertex mesh length of a conductor.
Order of basis functions	Specifies the polynomial order of the FEM basis functions. With a setting of 2, the electric fields will be approximated with quadratic variations within each tetrahedron. A setting of 1 will limit the approximation to linear within each tetrahedron. This will require less memory but will typically be less accurate. A value of 1 is more appropriate if the geometry is very small electrically.

Specifying FEM Mesh Parameters

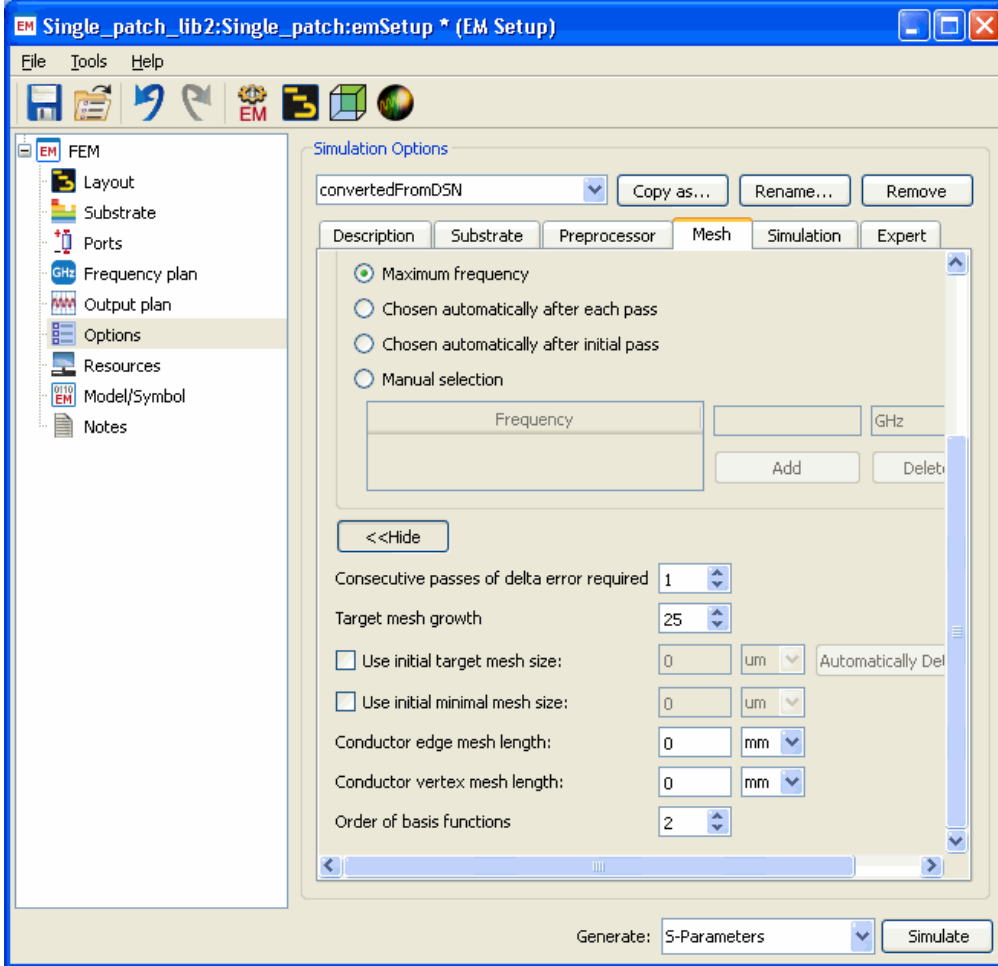
To set up mesh parameters for FEM simulations:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **FEM** as the EM simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Mesh** tab to display the mesh options for FEM.



5. Type a value for the delta error.
6. Type the minimum number of adaptive passes.
7. Type the maximum number of adaptive passes.
8. Specify the number of bondwire segments.

9. Enable the **Automatically open mesh convergence plot** option.
10. Select a refining frequency.
11. Select the **Advanced** tab to specify the advanced mesh options for FEM simulations.



12. Specify the consecutive passes of delta error required.
13. Select the **Use initial target mesh size** option.
14. Specify a value and select units for the initial target mesh size.

Note
To determine the initial mesh size automatically, click **Automatically Determine**.

15. Select the **Use initial minimal mesh size** option.
16. Specify a value and select units for the edge mesh length of a conductor.
17. Specify a value and select units for the vertex mesh length of a conductor.
18. Specify a value and select units for the order of basis functions.

Defining Simulation Settings

You can define the simulation settings for both Momentum and FEM. The simulation process combines the Green functions that were calculated for the substrate of a circuit and 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.

In this section, you will learn about:

- [Defining Simulation Settings for Momentum](#)
- [Defining Simulation Settings for FEM](#)

Defining Simulation Settings for Momentum

You can select the matrix solver used by Momentum during a simulation. Momentum provides the following matrix solvers:

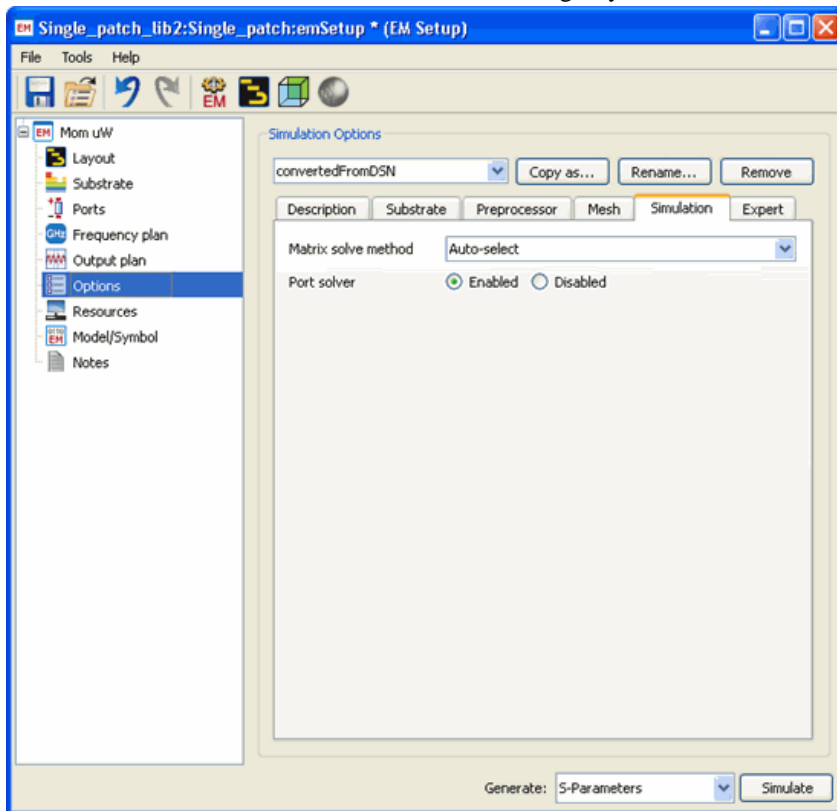
- **Auto-select:** When selecting **Auto-select**, Momentum automatically selects the most appropriate matrix solver. This is the default matrix solver selection option.

- **Direct Dense:** When selecting **Direct dense**, Momentum selects the direct dense matrix solver to be used in the simulation. This implies that the matrix is stored in a dense matrix format (requiring order N^2 memory) and solved using a direct matrix factorization technique (requiring order N^3 computer time). The direct dense matrix solver guarantees to always provide a solution using a predetermined number of operations. The main disadvantage is that the computer time it requires scales cubic with the matrix size N , yielding larger simulation times for large problem sizes.
- **Iterative Dense:** When selecting **Iterative dense**, Momentum selects the iterative dense matrix solver to be used in the simulation. This implies that the matrix is stored in a dense matrix format (requiring order N^2 memory) and solved using an iterative matrix solve technology (requiring order N^2 computer time). The computer time scales quadratic with the matrix size N , yielding faster simulation times for large problem sizes when compared to the direct dense matrix solver. The main disadvantage is that the iterative solver does not guarantee to converge fast. The iterative solver will monitor its convergence rate and will automatically switch to the **Direct dense** matrix solver when it observes that the convergence stagnates or the convergence rate is too slow.
- **Direct Compressed matrix solver:** When selecting **Direct compressed**, Momentum selects the direct compressed matrix solver to be used in the simulation. This implies that the matrix is stored in a compressed matrix format (requiring order $N \log N$ memory) and solved using a direct compressed matrix factorization technique (requiring order $(N \log N)^{1.5}$ computer time). The direct compressed matrix solver guarantees to always provide a solution using a predetermined number of operations. Moreover, the computer time it requires scales linear-logarithmic with the matrix size N , making this the preferred matrix solver for very large problem sizes.

Specifying Simulation Settings for Momentum

To set the simulation options for Momentum:

1. Select **EM > Simulation Setup** to open the EM Setup window.
2. Select **Momentum RF** or **Momentum Microwave** as the EM simulator.
3. Choose **Options** in the left pane of the EM Setup window.
4. Select the **Simulation** tab, as shown in the following figure.



5. Select the matrix solver from the **Matrix solver method** drop-down list.
6. Enable or disable the **Port Solver** option as per your requirements.

Defining Simulation Settings for FEM

You can select the following types of matrix solver methods for FEM:

- **Direct:** When selecting Direct, the FEM Simulator will choose the direct dense matrix solver to be used in the simulation. This implies that the matrix is stored in a dense matrix format (requiring order N^2 memory) and solved using a direct matrix factorization technique (requiring order N^3 computer time). The direct dense matrix solver guarantees to always provide a solution using a predetermined number of operations. The main disadvantage is that the computer time it requires scales cubic with the matrix size N , yielding larger simulation times for large problem sizes.
- **Iterative:** When selecting Iterative, FEM Simulator will choose the iterative dense matrix solver to be used in the simulation. This implies that the matrix is stored in a dense matrix format (requiring order N^2 memory) and solved using an iterative matrix solve technology (requiring order N^2 computer time). The computer time scales quadratic with the matrix size N , yielding faster simulation times for large problem sizes when compared to the direct dense matrix solver. The main disadvantage is that the iterative solver does not guarantee to converge fast. The iterative solver will monitor its convergence rate and will automatically switch to the Direct matrix solver when it observes that the convergence stagnates or the convergence rate is too slow. This is the default matrix solver selection option.

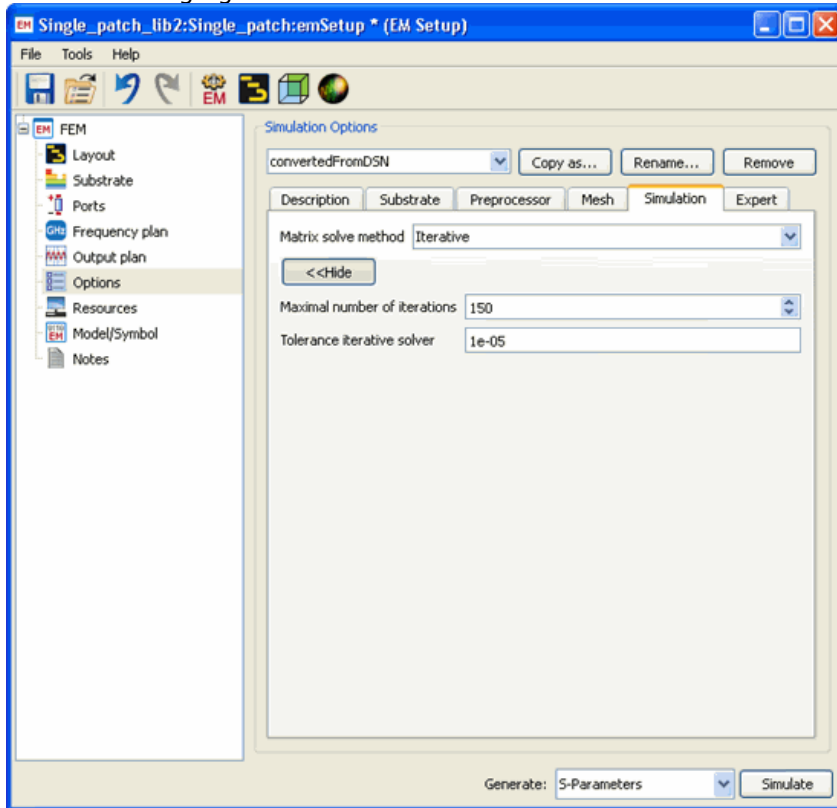
You can also specify the following advanced simulation settings:

- **Maximum Number of Iterations:** Specify the maximum number of iterations allowed during the iterative solve process. This value might need to be increased to achieve a lower absolute tolerance. Typically a value of 100 to 150 is sufficient.
- **Tolerance iterative solver:** Specify the absolute error tolerance that the solver process must reach to achieve convergence. A typical value would be $1.0e-4$ to $1.0e-5$. You may need to increase the Maximum Number of Iterations if you significantly decrease the tolerance.

Specifying Simulation Settings for FEM

To set the simulation options for FEM:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **FEM** as the EM simulator.
3. Choose **Options** in the left pane of the EM Setup window.
4. Select the **Simulation** tab.
5. Select a matrix solver from the **Matrix solver method** drop-down list.
6. Select the **Advanced** tab to specify advanced simulation settings for FEM, as shown in the following figure:



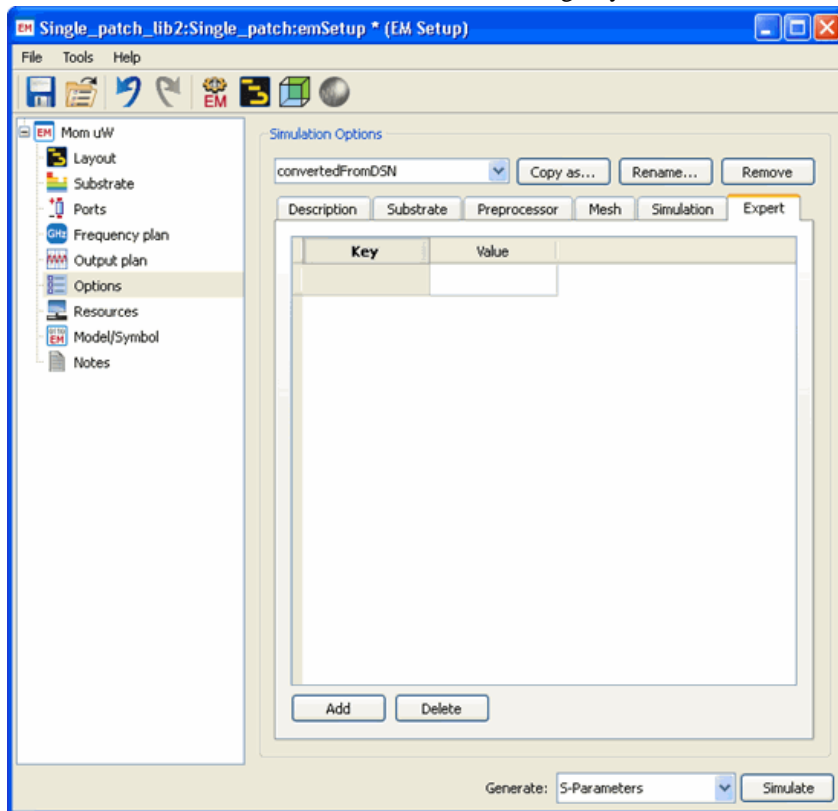
7. Specify the **Maximum Number of Iterations** value.
8. Specify the **Tolerance iterative solver** value.

Defining Expert Options for Momentum

You can define advanced settings for your Momentum simulation by using the Expert tab.

To specify the expert options:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Momentum** as the EM simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Expert** tab.
5. Click **Add** to insert a new row.



6. Specify a value in the **Key** field.
7. Specify a value in the **Value** field.

Overriding Substrate Definition for FEM

You can override the substrate definition at the global or layer-specific level. You can also specify advanced substrate options for FEM simulations.

In this section, you will learn about:

- [Specifying Global Substrate Options](#)
- [Specifying Layer-specific Substrate Options](#)
- [Specifying Advanced Substrate Options](#)

Specifying Global Substrate Options

Using the Global tab, you can specify the substrate overrides that are applied to all the layers. It provides the following options:

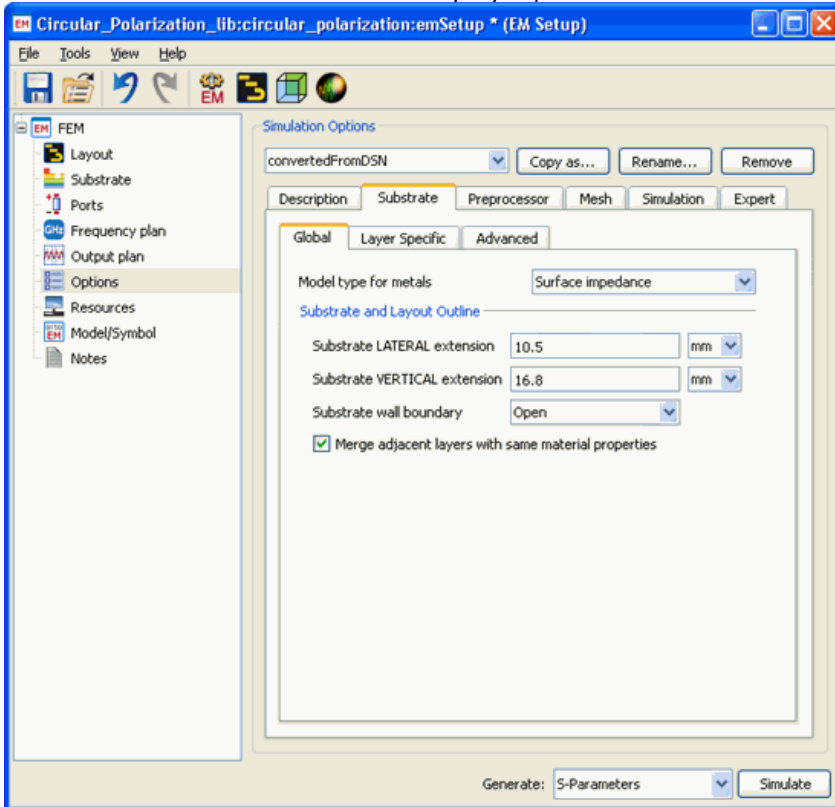
- **Specifying 3D Metal Expansion for Thick Conductors:** Conductors with finite thickness can be modeled using the 3D metal expansion feature. This feature will automatically expand the mask of a conductor with finite thickness in the direction orthogonal to the layered medium, using the specified thickness of this conductor. You can specify the following settings for 3D Metal Expansion feature:
 - As in substrate: Enables you to use the settings specified for substrate.
 - On: Allows you to override the substrate file definition by enabling the 3D Metal Expansion feature.
 - Off: Prevents overriding the substrate definition.
- **Modeling Currents:** To identify the conductive areas of a layout layer, you can define the model type for current. The current flow at the surface is modeled depending on the simulation model chosen.

Defining Global Substrate Options

To override the substrate definition globally:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.

2. Select **FEM** as the EM simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Substrates** tab.
5. Ensure that the **Global** subtab is displayed, as shown below:

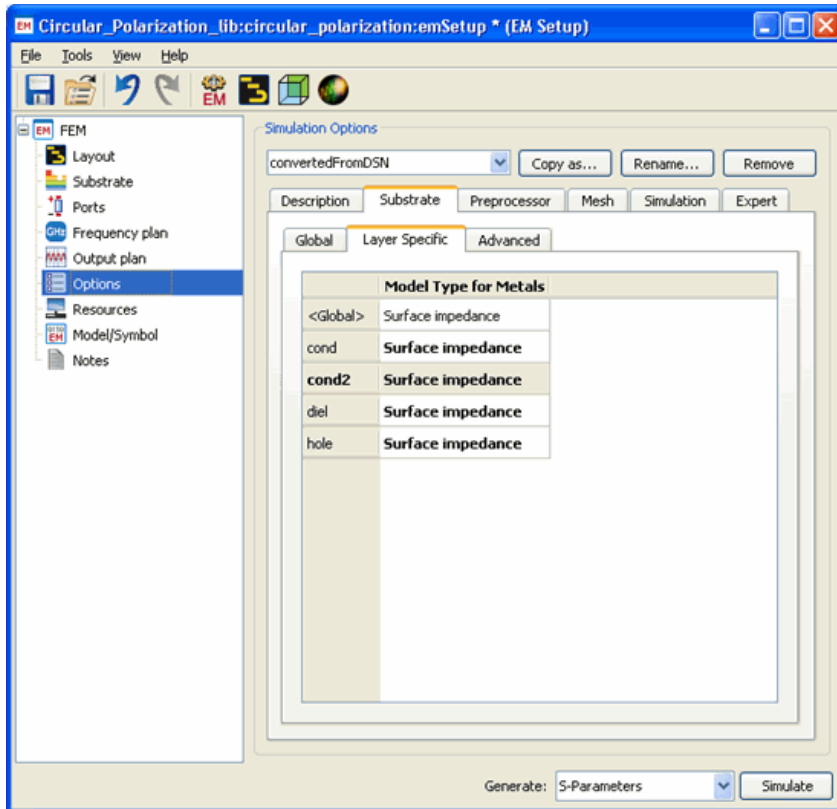


6. Select a value from the **Model Type for Metals** drop-down list.
7. Type a value in the **Substrate Lateral Extension** text box and select a unit.
8. Type a value in the **Substrate Vertical Extension** text box and select a unit.
9. Select a value from the **Substrate wall boundary** drop-down list.
10. Select **Merge adjacent layers with same material properties**.

Specifying Layer-specific Substrate Options

To define substrate overrides for a specific layer:

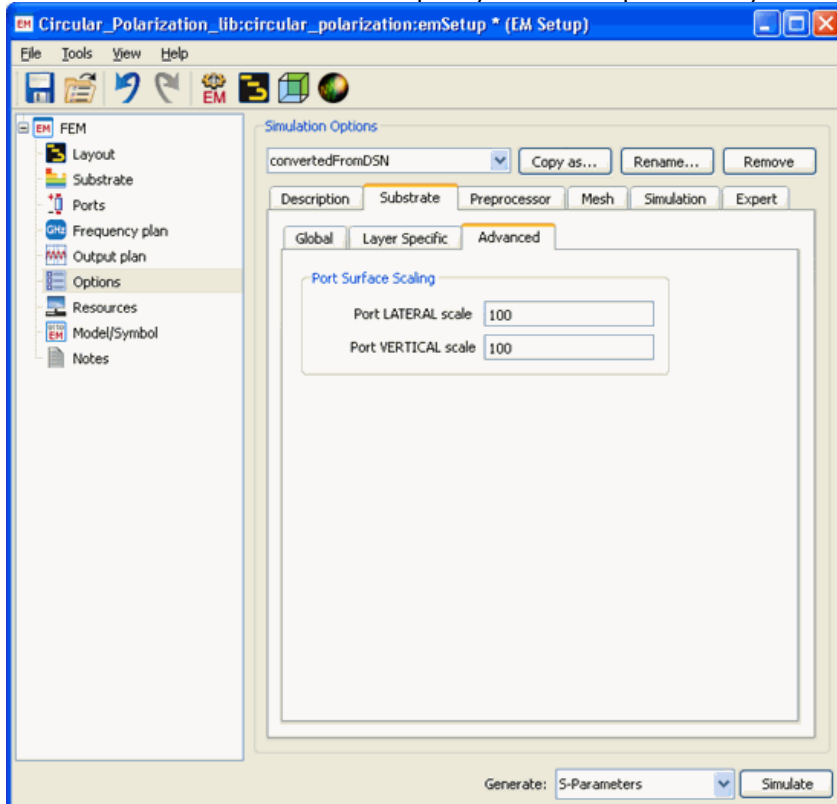
1. Choose **EM> Simulation Setup** to display the **EM Setup** window.
2. Select **FEM** as the simulator.
3. Select **Options** in the left pane of the EM Setup window.
4. Select the **Substrates** tab to display the substrate options that you can specify for FEM.
5. Select the **Layer Specific** subtab to override substrate at the layer level.



6. Select a value from the **Model Type for Metals** drop-down list in the required layer.

Specifying Advanced Substrate Options

1. Choose **EM> Simulation Setup**. The **EM Setup** window is displayed.
2. Select **FEM** in the **Simulator** section.
3. Select **Options** in the EM Setup window.
4. Select the **Substrates** tab.
5. Select the **Advanced** subtab to specify advance options for your FEM simulation.



6. Type a value in the **Port Lateral Scale** text box.
7. Type a value in the **Port Vertical Scale** text box.

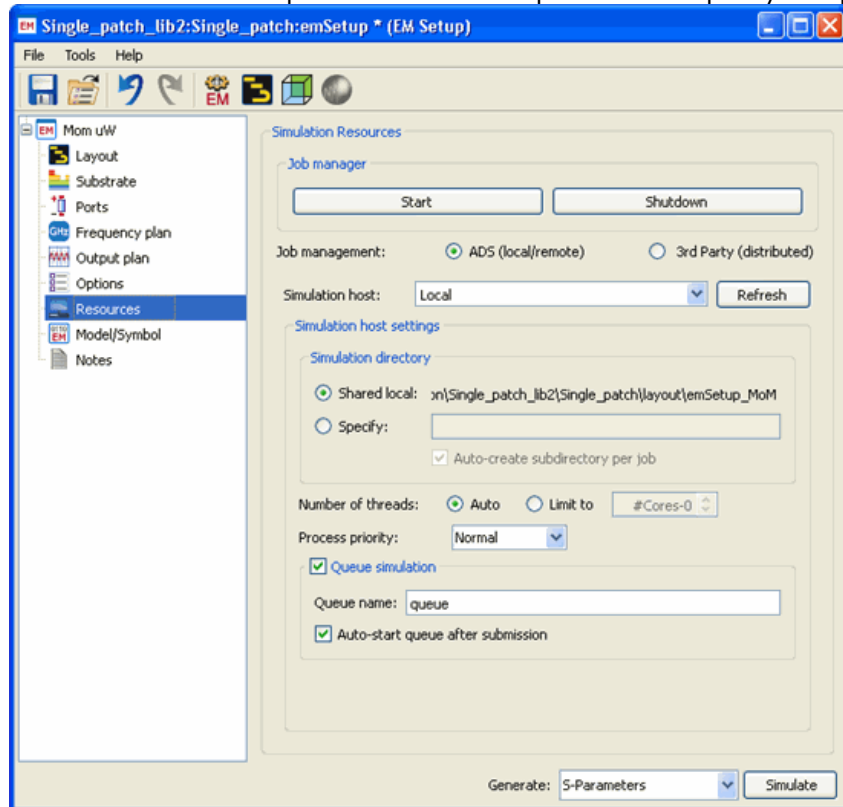
Specifying Simulation Resources

You can run EM simulations on a local or remote machine, or take advantage of a third party load balancing and queuing system such as LSF, Sun Grid Engine, and PBS Professional. You can choose one of the following simulation modes:

- [Setting up Local and Remote Simulation](#)
- [Setting up Third Party \(Distributed\) Simulation](#)

Setting up Local and Remote Simulation

Using the EM Setup window, you can specify the setup options for local or remote simulations. Choose **EM > Simulation Setup** to open the **EM Setup** window. Select **Resources** in the left pane of the EM Setup window to specify setup options:



In the EM Setup window, you can specify the following setup options for local or remote simulations:

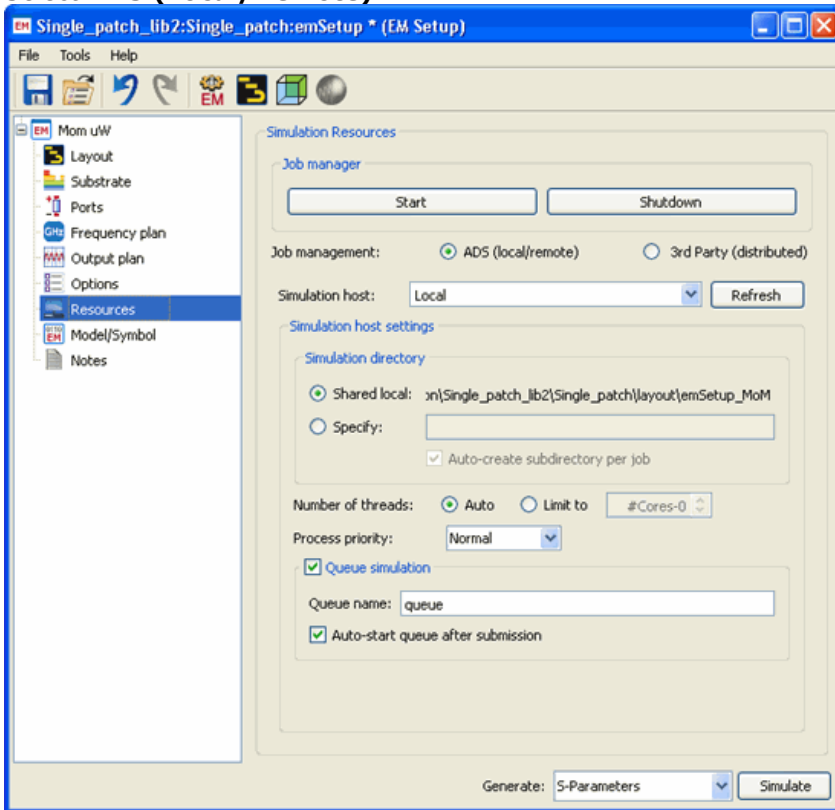
- The capability to start and stop the Job Manager: Click **Start** to initiate the Job Manager window and click **Stop** to close the Job Manager process. For more information, see *Using the Job Manager (em)*.
- Specifying a simulation directory: You can either specify a directory or use the shared directory for storing simulation data:
 - Selecting **Shared local** displays the path of the local directory.
 - Selecting **Specify** allows you to type a new directory. If you select **Auto-create subdirectory per job**, sub-directories are created automatically.
- Specifying the number of threads: You can specify the number of threads used in the local simulation:
 - Auto: The number of threads are chosen depending upon the number of cores of the machine.
 - Limit to: Allows you to specify the number of threads. This number can be:
 - N: This would limit the threads to the N or #Cores of machine whichever is smaller.

- #Cores - N: This would free N Cores of the machine. The threads would limit to #Cores - N. In the auto mode, the value of n is equal to 0.
- Specifying the process priority: You can choose the Normal, Below Normal, or Idle types of process priority. This maps to the nice number under Linux/Unix and process priority under windows.
- Specifying queue simulation: If you select **Queue simulation**, your simulation will be put on a queue and must be started from the Queue Manager.
 - Queue name: Specify a queue name.
 - Auto-start queue after submission: Selecting this option starts the simulation automatically you submit the changes.

Specifying the Local/Remote Setup Options

To specify the host settings for a local or remote simulation:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Resources** in the left pane of the EM Setup window.
3. Select **ADS (Local/Remote)**.



4. Specify the simulation directory.
5. Specify the number of threads.
6. Choose the process priority from the drop-down list.
7. Select **Queue simulation** to queue your simulation.
8. Provide a queue name.
9. If you want the queue to start automatically, select **Auto-start queue after submission**.

Setting up Third Party (Distributed) Simulation

If you want to use a third party workload management system, such as LSF, PBS Professional, and Sun Grid Engine, you can use the Distributed mode. In the distributed mode, you have access to the third party workload/queue management system to submit an EM simulation to a cluster of machines.

Before using the distributed mode, you need to set up a connection between your personal machine and the target machine that you want to use as submit host to the workload

management system. The submit host is the one machine that will add and monitor simulation jobs given to the workload management system on behalf of your ADS session.

The submit host can be your local machine or can be a remote machine. To create this connection with the submit host you must first make the Job Manager aware of the workload management system on the submit host by adding the necessary configuration settings of the workload management system to the startup environment of the PVM daemon on the submit host.

On a Linux or a Solaris submit host add the necessary environment variables or source the required configuration script for your system into the *PVM boot script* (em) `~/eesofpvmprofile`. On a Windows based submit host make sure the workload management system can be used by all users from a Windows command window. Refer to the documentation of your workload management environment for details on these configuration requirements. Next, ensure that the connection settings for the submit host are properly configured in the *Job Manager Preferences* (em).

Note
For the distributed simulation with a third party workload management system where the local machine is a valid submit host, you do not need to define any formal PVM cluster configuration. A local PVM daemon will start and provide the submit and monitor capabilities with the workload management system.

To run simulations, the distributed mode assumes that the third party workload management system has been preconfigured to run ADS related tasks from the user command line environment on the submit host and on the execution hosts (the machines that will run the EM simulation) of the workload management system.

The distributed mode assumes that the third party workload management system has been preconfigured to run ADS related tasks from the command line environment. For setting up the third party workload management system refer to the documentation for your particular environment. You can find information for the supported workload management systems at the following location:

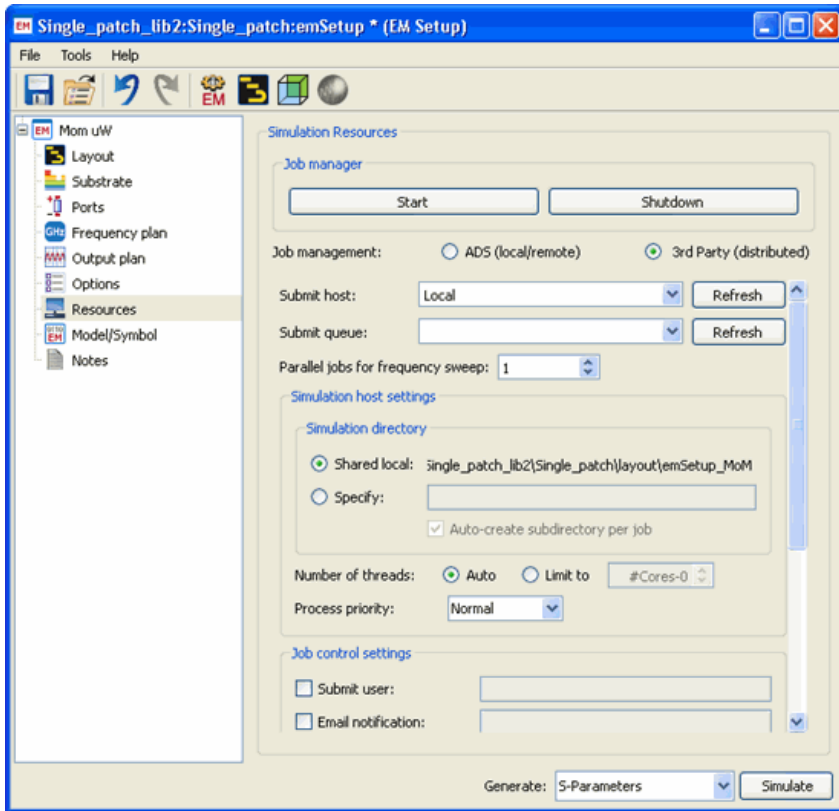
- LSF: <http://www.platform.com>
- PBS Professional: <http://www.pbsgridworks.com/Default.aspx>
- Sun Gridengine: <http://gridengine.sunsource.net/>

Make sure that the execution environment has the essential variables such as *HPPEESOF_DIR*, *PATH* and ADS license settings are correctly defined on the hosts in the workload management system. For further details about the ADS configuration, refer to the ADS installation guide.

On Linux or Solaris machines, you can also add a ksh script `.adsrc` with the appropriate ADS environment settings inside your home directory on the cluster machines. The `~/adsrc` file is picked up by the simulation scripts just before a simulation starts.

Third Party (Distributed) Setup Options

In the EM Setup window, you can specify setup options for the distributed mode, as shown in the following figure:



In the EM Setup window, you can specify the following setup options for the distributed mode:

- The capability to start and stop the Job Manager: Click **Start** to initiate the Job Manager window and click **Stop** to close the Job Manager process. For more information, see *Using the Job Manager (em)*.
- Specifying a submit host: Choose the required submit host (this can also be the local machine, as the submission host) for distributed simulation. Click **Refresh** to retrieve the list again.
- Specifying a submit queue: Choose the required submit queue on the selected submit host for distributed simulation. Click **Refresh** to retrieve the list again.
- Specifying parallel jobs for frequency sweep: Allows to specify the number of frequency points that will be calculated in parallel. It controls how the frequency sweep is distributed over the cluster. If it is equal to 1, it specifies a normal momentum simulation. If the value is greater than 1, you need a momentum turbo license and this is only supported between a cluster of Linux machines.
- Specifying a simulation directory: You need to define the remote simulation directory on the submit host for your distributed simulations. It sets the directory on the remote computer holding the EM files while the simulation is running. If the host mode is remote but the host is set to the local machine, the remote directory should be set to **Shared local**.

Note

For the distributed simulation with a third party workload management system it is very important that this directory is visible to all cluster hosts in exactly the same way. Make sure mount points are identical.

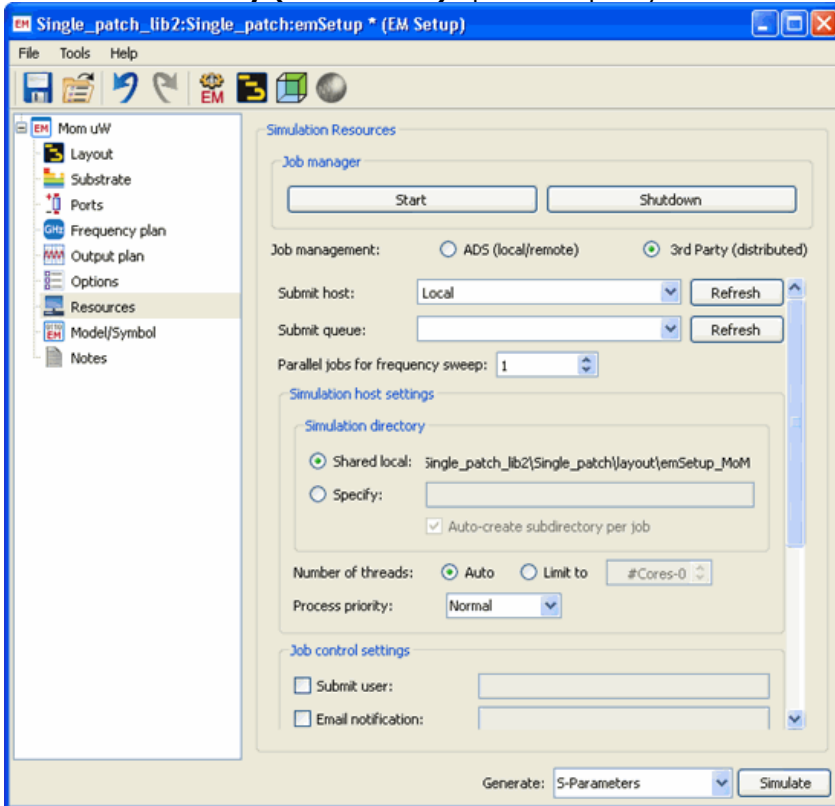
- Specifying the number of threads: You can specify the number of threads used in the distributed simulation. Refer to *Number of Threads* for details.
- Specifying process priority: You can choose the process priority among the **Normal**, **Below Normal** and **Idle**. This maps to the nice number on the Linux/Unix platform and process priority on the Windows platform.

- Specifying job control settings: You can specify the following job control settings:
 - Submit user: Enables you to provide a user name if the local user name differs from that on the cluster.
 - Email notification: Enables you to specify this option for notification purposes.
 - Job name: Enables you to specify a name.
 - Shell command at finish: Allows you to use shell commands.
 - Resource String: Allows you to specify a resource string.
 - Export env. Variables: Enables you to export environment variables.
- Specifying the start time: You can configure the simulation start time by selecting the required option from **Start**. You can choose either **Now**, **At time**, or **After job(s)** option.
- Specifying the termination time: You can configure the simulation expire time by selecting the required option from **Expire**. You can choose either **Never** or type the required time in the **At/After** text box.

Specifying Third Party (Distributed) Setup Options

To specify the distributed mode setup options:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Resources** in the left pane of the EM Setup window.
3. Select the **3rd Party (distributed)** option to specify the distributed setup options.

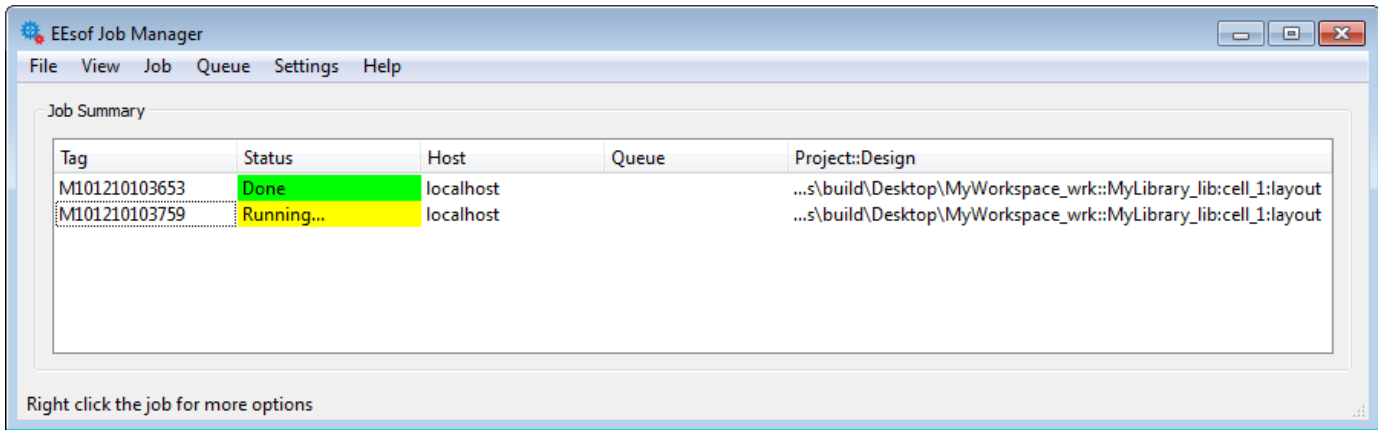


4. Specify the simulation directory.
5. Specify the number of threads.
6. Choose the process priority from the drop-down list.
7. Specify the job control settings.
8. Select the **Start** option.
9. Select the **Expire** option.

Using the Job Manager

The EM Setup window provides the capability to start and stop the Job Manager. For more information, refer *Using the Job Manager (em)*. The Job Manager is automatically started in minimized mode when an EM Setup is being opened. Click the **Start** button to restart a closed Job Manager or to restore its main window, as shown in the following

figure:



After the successful start of the Job Manager, the Simulation host list is populated with the active hosts and those defined in the *Job Manager Preferences* (em). Select the desired host where you want to launch the simulation.

To close the Job Manager, close its main window or click the **Shutdown** button in the EM Setup window. Click **Shutdown** again if you need to bring the underlying PVM infrastructure to a halt (a confirmation dialog box is displayed).

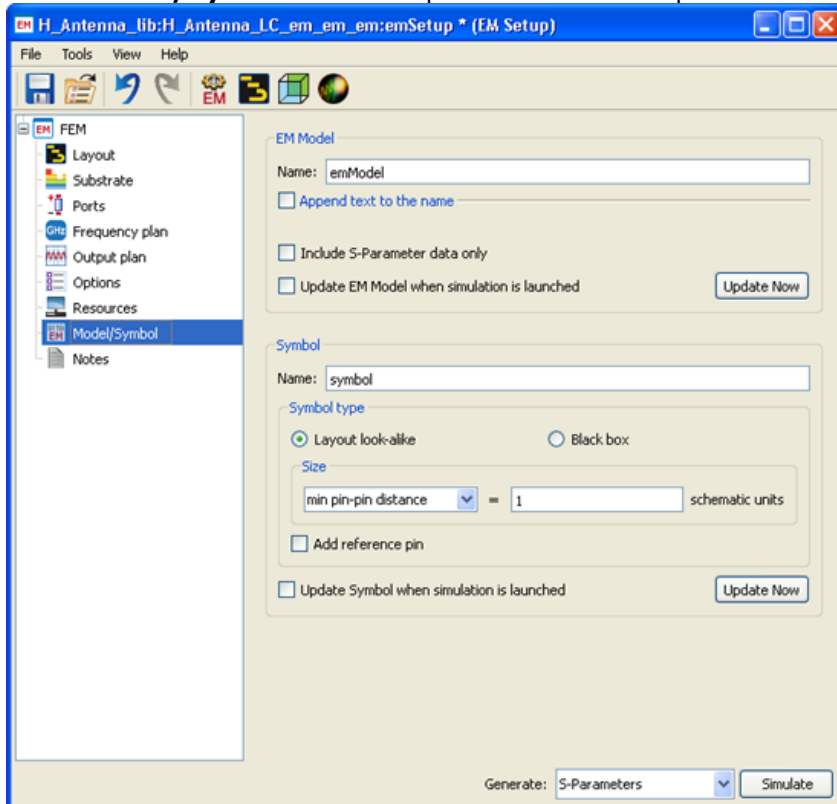
Generating EM Model and Symbol

The EM Setup window enables you to generate an EM model and symbols. For more information about how to use an EM model, refer *Using an EM Model* (em).

Generating an EM Model

To create an EM model:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Model/Symbol** in the left pane of the EM Setup window.



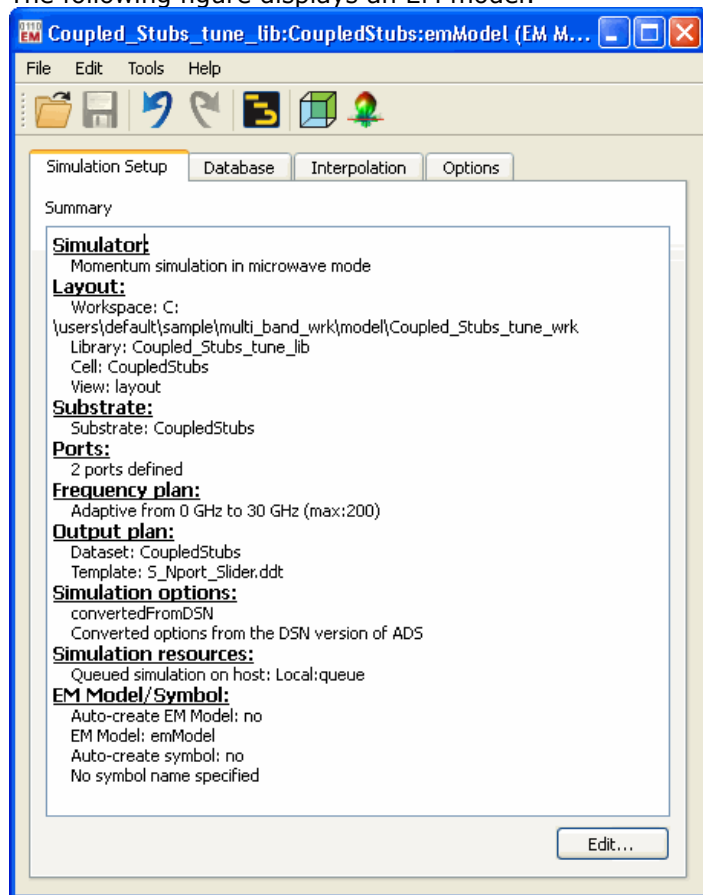
3. Type a name for the EM model.

Note The default name for an EM Model is *emModel*. It is recommended to use the default name unless you are using model polymorphism.

4. Do not select **Include S-Parameter data only**, if you want to store all EM simulation files with the EM model. For example, all log files are stored in the EM model directory. It will also allow you to perform post-processing operations on the EM Model in case you specified to save the currents (Momentum) or fields (FEM) in the **Output plan** tab.
5. Select **Create EM model when simulation is launched** to create an EM model after the simulation process.
6. To generate an EM model immediately, click **Create Now**.

Note If the **Create Now** button is labeled **Update Now**, then the EM Model already exists for this cell.

The following figure displays an EM model:



You can also generate an EM model by selecting **EM > Component > Create EM Model and Symbol**. To know more about an EM Model, refer to *Using an EM Model (em)*.

Customizing the EM Model Naming Convention

You can customize the naming convention of the EM model generated during a simulation by adding a prefix or suffix to the data name. To generate the EM model with the required naming convention, you need to specify the following parameters in the EM Setup dialog box:

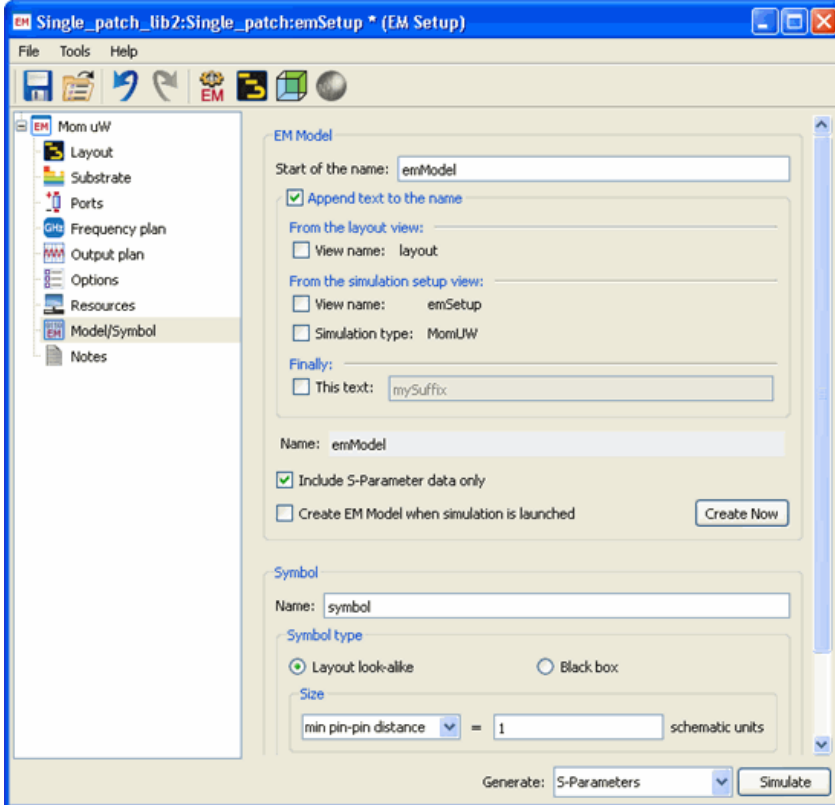
- Specifying the EM model name: You can specify the EM model name in the **Name** field.
- Appending customized text to the EM model name: If you want to include customized text with the EM model name, select **Append text to the name**. You can customize the data name as follows:
 - Adding text from the layout view: Select the **View name** option in the **From the layout view** section to add text from the layout view.
 - Adding text from the simulation setup view: Select **View name** in the **From the simulation setup view** section to add text from the simulation setup view. You

can also include the type of simulator in the EM model name by selecting **Simulation Type**.

- Adding user-defined text: You can add a suffix to the EM model name in the **This text** field.

To customize the EM model name:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Model/Symbol** in the left pane of the EM Setup window.
3. Specify the EM model name in the **Name** field.
4. Select **Append text to the name**, as shown in the following figure:



5. Select **View name** in the **From the layout view** section.
6. Select **View name** in the **From the simulation setup view** section.
7. Select **Simulation Type**.
8. Add a suffix to the EM model name in the **This text** field.

Generating Symbols

Layout symbols are user-defined components that you can create from a layout window in ADS. You can insert symbols in a schematic window just like any other component, and represents the (parametric) layout from the layout page. The symbol representing this component in schematic can either be a black box or a layout look-alike symbol.

After creating a symbol, you can use it in one of the following ways:

- Include the symbols directly in the schematic. When doing a circuit simulation from the schematic environment, the planar EM solver is called automatically during the circuit simulation (EM/Ckt co-simulation) to generate an EM model. The user-defined layout parameters and the most relevant Momentum simulation parameters (e.g., model type, mesh density) can be set from the schematic page. With the co - optimization feature you can also determine an optimized value for your Layout Component without returning to layout. The layout components have a built-in database mechanism that keeps track of previously calculated EM simulation results, that is, once the EM simulation is complete for a symbol, no new EM simulations are required, unless there is a change in the parameters for this component.

- You can use the symbols in Advanced Model Composer (AMC), for more details refer to *Using Advanced Model Composer (em)*. AMC enables you to generate a parameterized electrical model for a symbol from within the layout window. After defining and creating the symbol, you can specify the ranges for the parameters (with continuous and discrete parameter range options) and launch the AMC model generation tool from within the layout environment. In contrast to the EM/Ckt co-simulation feature, AMC is an up-front model generation tool. This means that the parameterized electrical model is calculated before the component is used in a schematic design, based on a number of EM simulations, which are run in the background. No EM simulations need to be performed when the component is used in schematic.

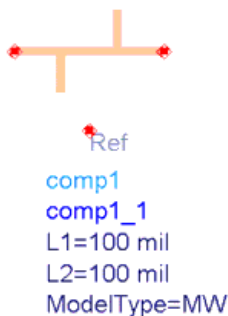
Symbols and EM/Circuit Co-simulation

The Electromagnetic-Circuit co-simulation feature enables you to combine EM and circuit simulations from the schematic. From a layout in Advanced Design System, you can create and insert a (parametric) layout symbol in a schematic. Symbols have a built-in database mechanism that keeps track of previously calculated simulation results. Once the EM simulation is completed for a certain layout component, no new EM simulations are required unless there is a change in the component parameters.

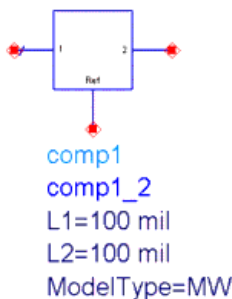
Selecting a Symbol

The symbol representing this component in a schematic can either be a black box or a layout look-alike symbol:

- A layout look-alike symbol for which the symbol shape is a scaled copy of the layout shape.



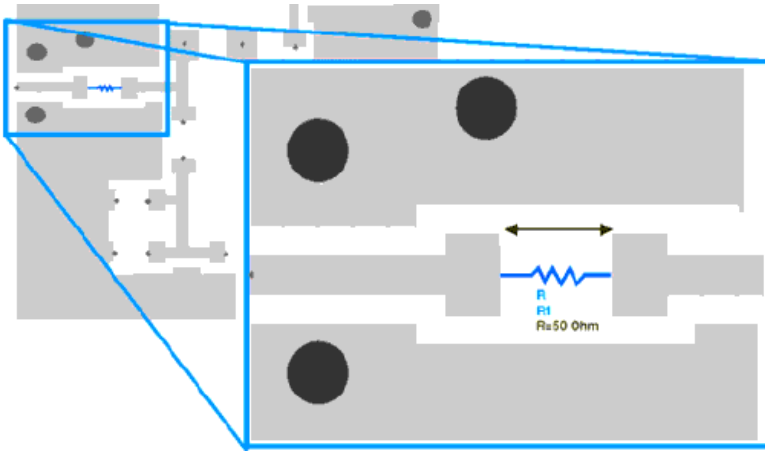
- A black box symbol with a generic rectangular shape.



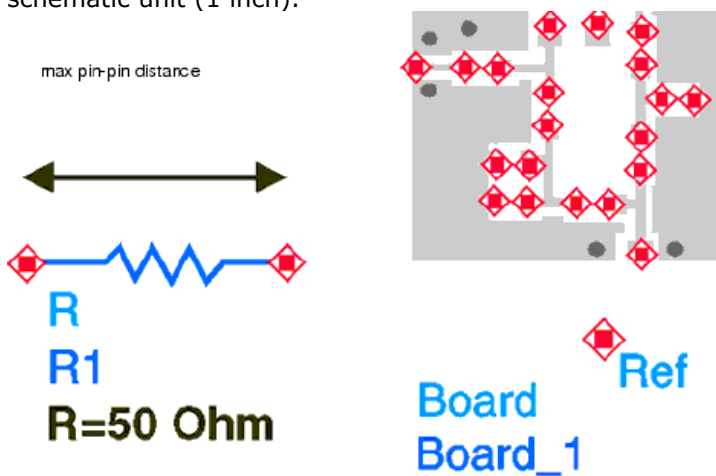
Scaling the Look-alike Symbol

You can scale the look-alike symbol in the following ways:

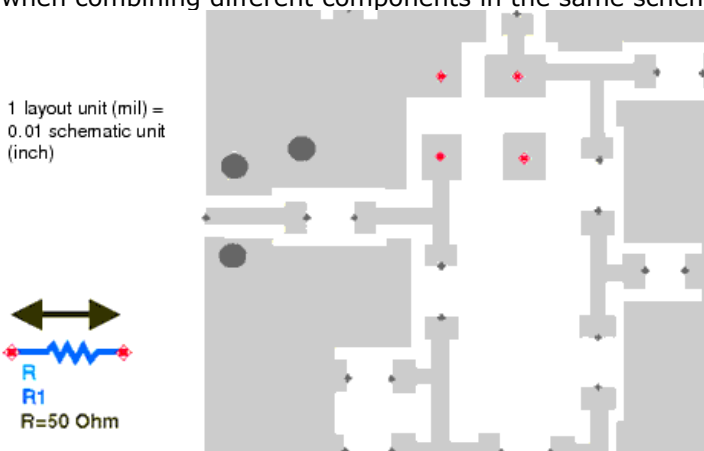
- Setting the minimal pin-pin distance in schematic units: One schematic unit corresponds with the size of most standard two port components such as resistors, capacitors, and inductors. Setting the minimal pin-pin distance to one schematic unit (1 inch) ensures that a component with a given symbol size will fit between two pins.



- Setting the maximum pin-pin distance in schematic units: The following figure illustrates the scaling when the maximal pin-pin distance is set to be equal to one schematic unit (1 inch).



- Mapping the layout unit dimensions to the schematic unit dimensions (recommended when combining different components in the same schematic page).



When you combine different layout components on a single schematic, a fixed, absolute scaling factor is the most appropriate choice to ensure that the relative sizes of the layout components are preserved.

Adding a Reference Pin

Using the Add Reference Pin check box, you can enable or disable the addition of an extra reference pin during layout component creation. This "extra" pin is used by EM to provide a reference voltage for all other pins in the layout and enables you to model ground effects (e.g., ground bounce, imperfect ground connections, etc.). In the absence of an

explicit Reference Pin, this reference voltage is zero what corresponds with connecting the Reference Pin to the absolute ground (GROUND component) which gets netlisted as node 0.

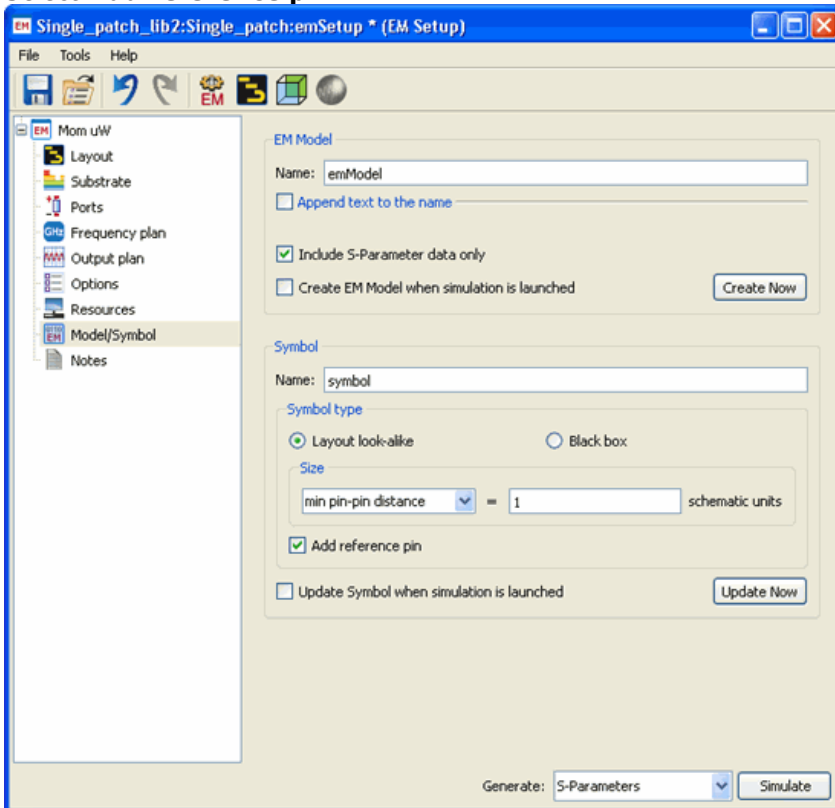
The Add Reference Pin selection is only available for the layout look-alike symbol selection. When selecting black box this option gets enabled (grayed out). Because black box uses standard symbols, a reference pin will appear in all schematic black box representations. This can cause problems if you are creating a structure with finite ground planes and wish to use them in the schematic. In this case, the layout look-alike symbol with Add Reference Pin disabled is a better choice.

This option should also be disabled and the layout look-alike symbol be used for components intended for use with the Schematic > Generate/Update command. This extra pin will cause a number of pins mismatch error during design synchronization.

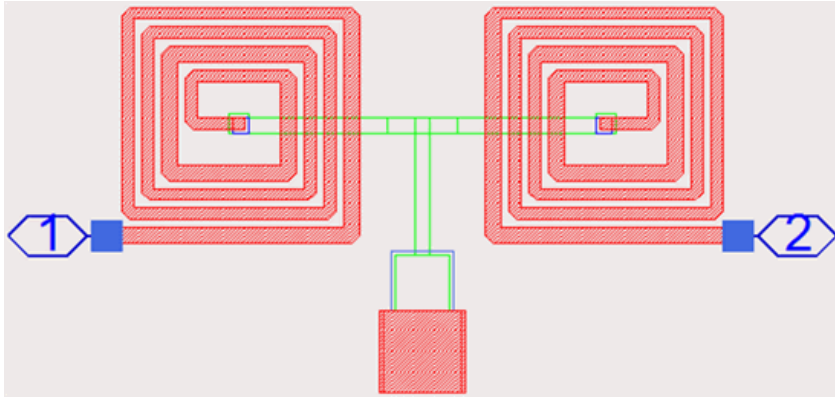
Creating a Symbol

To create a symbol:

1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Model/Symbol** in the in the left pane of the EM Setup window.
3. In the **Symbol** section, specify a name for the symbol.
4. Select the symbol type.
5. Specify the symbol size.
6. Select **Add reference pin**.



7. Click **Create Symbol when simulation is launched** to create a symbol after the simulation process.
8. To generate a symbol immediately, click **Create Now**. The following figure displays a layout look-alike symbol:



Note
If the **Create Now** button is labeled as **Update Now**, then a symbol with the given name already exists. Clicking **Update Now** updates that symbol from the layout.

There are two other ways to create or update a layout look-alike symbol:

- In the layout window, choose **EM > Component > Create EM Model and Symbol**.
- In the EM Setup window, on the **Model/Symbol** screen, select **Update Symbol when simulation is launched**. When this option is selected, every time you run an EM simulation using this EM Setup, the layout look-alike symbol will be created (or updated if it already exists).

There might be some visual differences in between the look-alike symbol you create and the output displayed in the Layout. These differences are described below:

- Look-alike symbol generation in ADS 2011.01 does not translate the EM simulation specific layout objects: Box, Waveguide and FEM Symmetry Plane. They are special layout objects with no equivalent for the schematic environment. You can manually add a graphical representation to the symbol with the symbol editor if you need to see these objects in your schematics.
- In multi-technology structures, the mapped layers in the top level technology are used to display features of instances that use a nested technology. Mapped layers do not automatically take the display properties of the underlying technologies and can give a very different visual impression. Access the Layer Display Properties of the top level technology and update the colors of the mapped layers.
- The symbol generation flattens the hierarchy and scales shapes. This can distort or drop small features.
- The background color of Layout and Schematic windows are usually different. This can change the visual impression of objects on layers that use a fill pattern and/or transparency.
- Layout pins on different layers on the same position are shifted away from each-other in the symbol because pins are not layer based in the schematic environment.

Running EM Simulations

The simulation process generates a mesh for the circuit and solves E-fields in the circuit. Using these electric field calculations, S-parameters are calculated for the circuit. The mesh electric field is then refined and new electric fields and S-parameters are computed. Before running a simulation, the following criteria must be met:

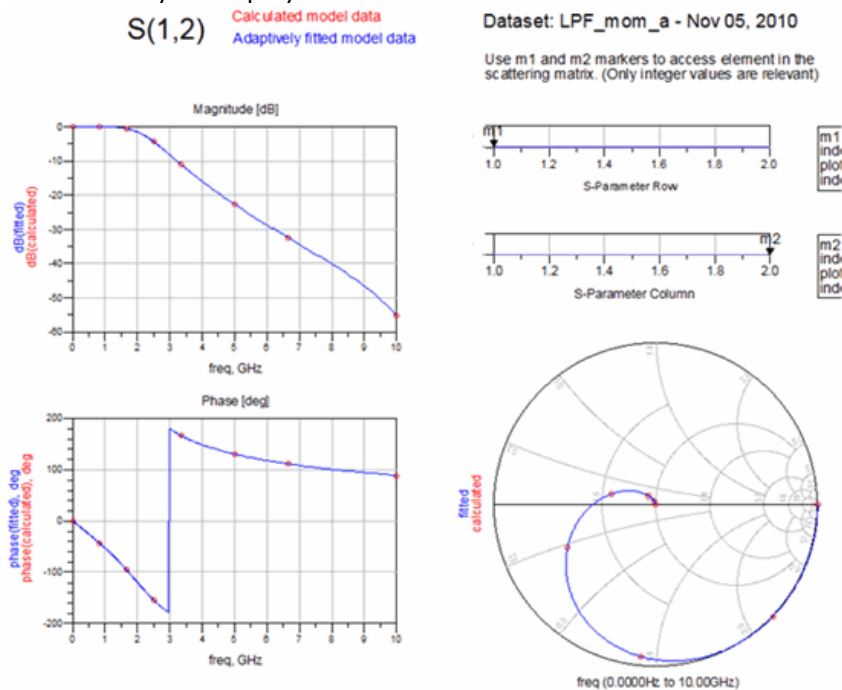
- A substrate definition must be specified for the circuit.
- The circuit must include at least one port.
- The mesh is automatically calculated when you simulate.
- A simulation frequency plan must be specified.

If any one of the above criteria is not met, an error is generated if you try to run a simulation.

Starting an EM Simulation

To start a simulation:

1. Choose **EM > Simulation Setup**.
2. In the EM Setup window:
 - Verify that at least one frequency plan appears in the Frequency Plans list.
 - Check that the Process Mode, Solution Files, and Data Display options are set to your specifications.
 - Check whether the simulation mode is set to local machine.
3. Click **Simulate** in the EM Setup window or press **F7**. A data display opens automatically to display the results.



Viewing Simulation Status

When you start a simulation, new job entry is created in the Job manager. For more information, refer to the *Using the Job Manager (em)* document. If the simulation has to wait in a Job Manager queue, the new entry in main Job Manager window is displayed. You can double-click this entry to open a detailed log window. If the simulation does not need to wait, the log window is immediately displayed, which displays the simulation status and simulation messages. Messages usually refer to any errors found, the percent of completion (refer to Understanding the "% Covered" Status Message) and simulation completion. If you close the log window and want to reopen it, double-click the job entry in the main Job Manager window. To open this window from a layout window, choose **Window > Simulation Status**.

To indicate progress, a progress bar appears at the bottom of the log window, if the number of unknowns in the simulation is greater than 500; otherwise, only the frequencies are displayed. If a simulation fails due to setup related issues, partial data is saved and a message appears in the log window.

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.

Stopping an EM Simulation

To stop an EM simulation, choose **EM > Stop and Release Simulator**. Alternatively, the simulation can be stopped using **Job > Stop/Cancel Job** in the *Using the Job Manager* (em).

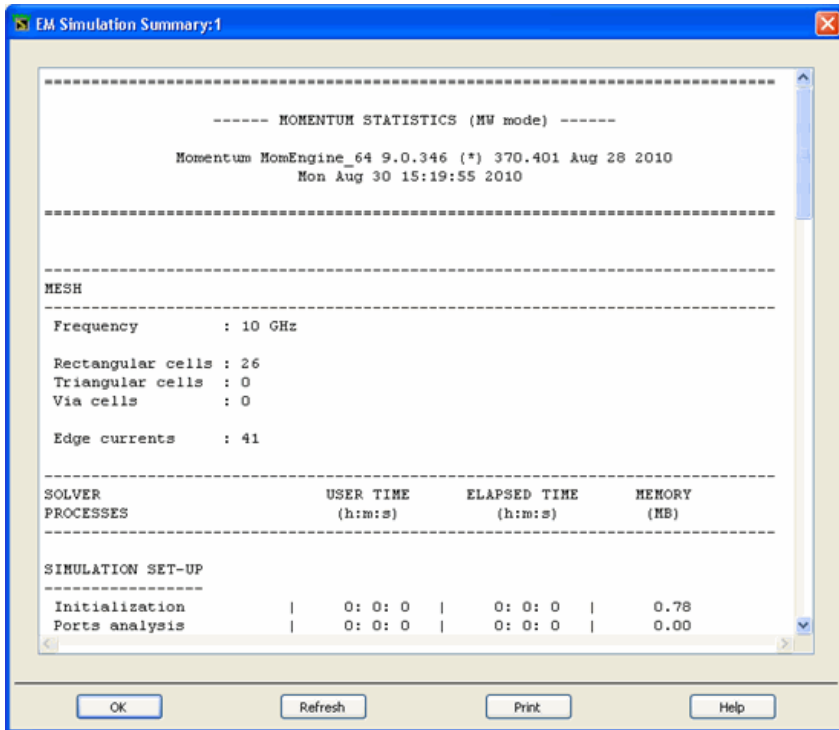
Viewing the Simulation Summary

As a simulation progresses, you can view solution statistics. Some of the information returned includes time to solve, the resources required, mesh information, and any messages that were displayed in the status window.

Choose **EM > Show Most Recent** and select one of the following option to view the required data:

- Momentum Mesh
- S-Param Simulation Summary
- Mesh Generation Summary
- Geometry Preproc Report
- Substrate Generation Summary

The following example shows a (partial) summary displayed when using the Momentum Microwave simulation mode.



Clicking **Refresh** updates the statistics for a simulation. However, this option is applicable only for local and remote EMX simulations.

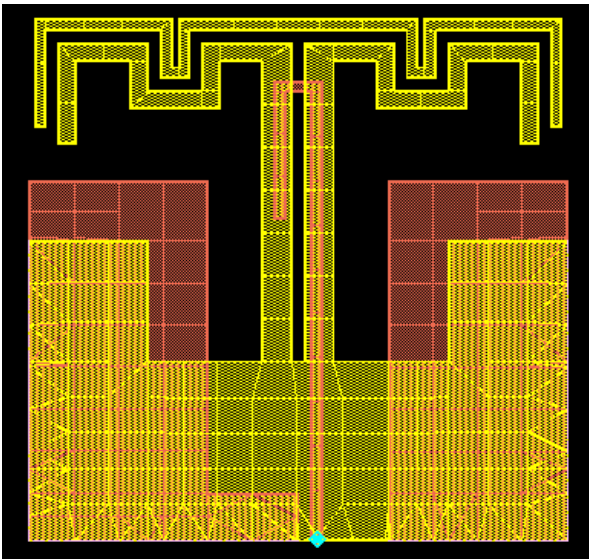
Viewing Simulation Summary

During a simulation process, you can view the statistics of momentum mesh, S parameter, substrate, mesh, and geometry statistics. The simulation summary includes time to solve, the resources required, cell information, and any messages that were displayed in the status window. You can view the following types of summary for an EM simulation:

- Momentum mesh
- S-param simulation summary
- Mesh generation summary
- Geometry preproc summary
- Substrate generation summary

Viewing Momentum Mesh

You can view the momentum mesh by selecting **EM > Show Most Recent > Momentum Mesh**. This selects the mesh view in a circuit, as shown in the following figure:



Clearing a Mesh

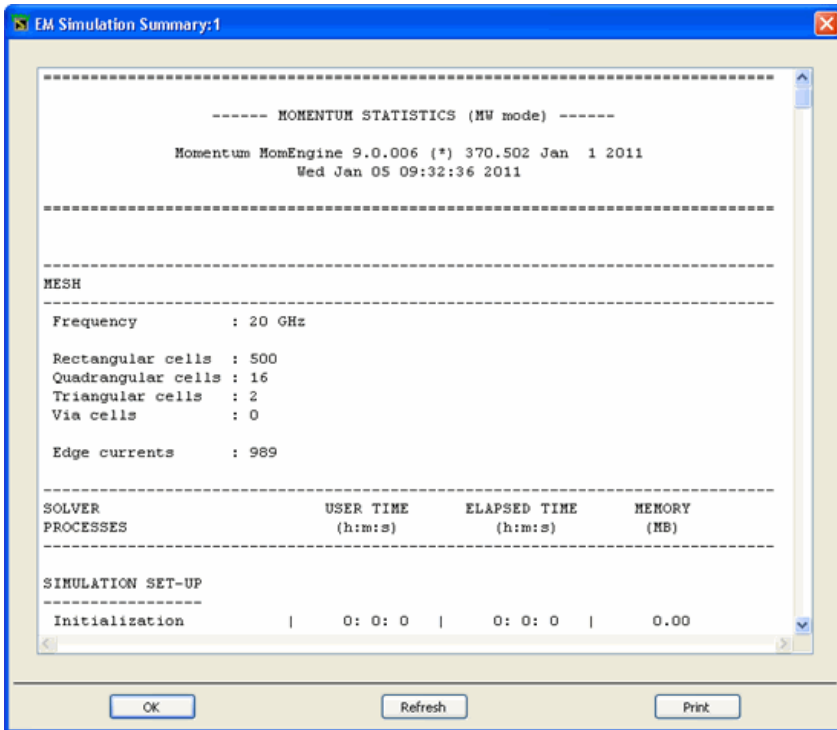
You can delete the view of the mesh from the circuit. Clearing a mesh only deletes the view from the Layout window; it does not erase the mesh calculations.

To remove the mesh from the display:

1. Choose **EM > Clear Momentum Mesh**.
2. If you want to redisplay the mesh, choose **Edit > Undo**. Otherwise, you must recompute the mesh in order to view it.

Viewing S-param Simulation Summary

You can view the S parameter simulation summary by selecting **EM > Show Most Recent > S-Param Simulation Summary**. This displays the **EM Simulation Summary** window. The following example displays a summary window for the Momentum microwave simulation mode.



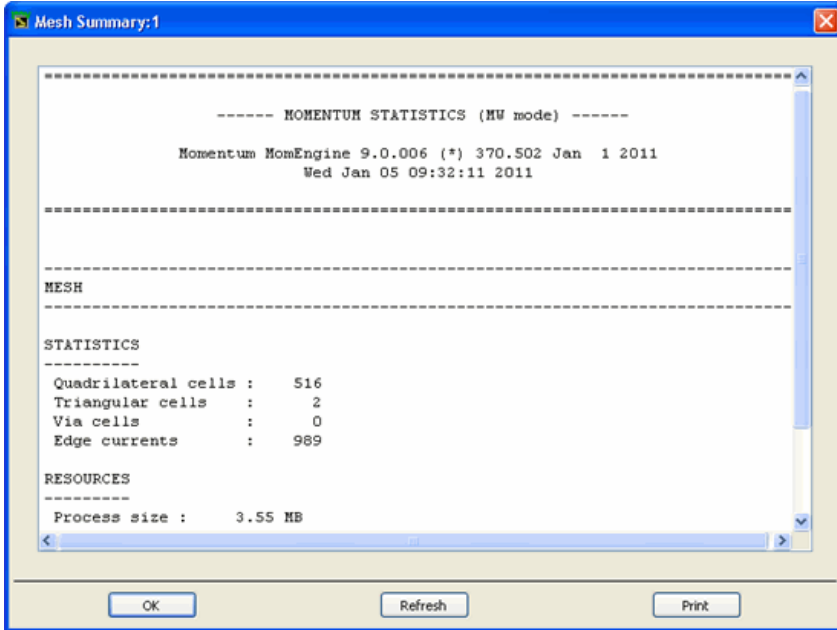
In the EM Simulation Summary window, you can view information about the matrix size after mesh reduction and the statistics for the quasi-static calculations. The User Time and Elapsed Time are listed as separate entries in the Summary window. The difference between the two is that elapsed time represents the overall time it took to complete the simulation, while user time only presents the time it took the CPU to complete the calculations. In other words, if the CPU is able to work exclusively on the simulation and is not occupied by another process, the user time and elapsed time will be equal.

If the mesh computation process is successful, you can view the following mesh statistics:

- The number of rectangular cells in the structure
- The number of quadrangular cells in the structure
- The number of triangular cells
- The number of via cells
- The number of edge currents
- Simulation setup information

Viewing Mesh Generation Summary

You can view the generated mesh statistics by selecting **EM > Show Most Recent > Mesh Generation Summary**. This displays the **Mesh Summary** window. The following example displays the summary window for the Momentum microwave simulation mode.

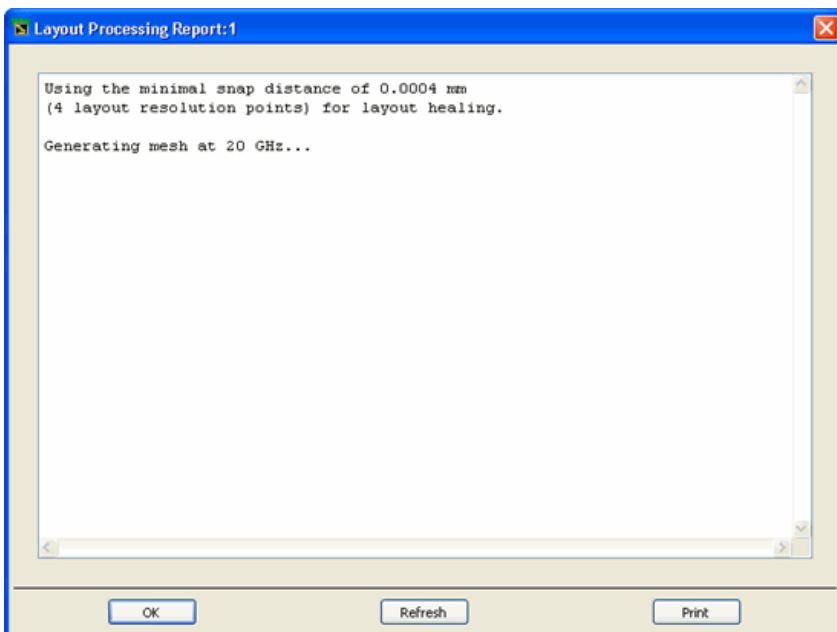


In the Mesh Summary window, you can view the following mesh statistics:

- Number of Quadrilateral cells
- Number of triangular cells
- Number of via cells
- Number of edge currents
- Computer resources used and time to solve
- User Time and Elapsed Time

Viewing Geometry Preproc Summary

You can view the layout processing report by selecting **EM > Show Most Recent > Geometry Preproc Summary**. This displays the **Layout Processing Report** window. The following example displays a summary window for the Momentum microwave simulation mode:

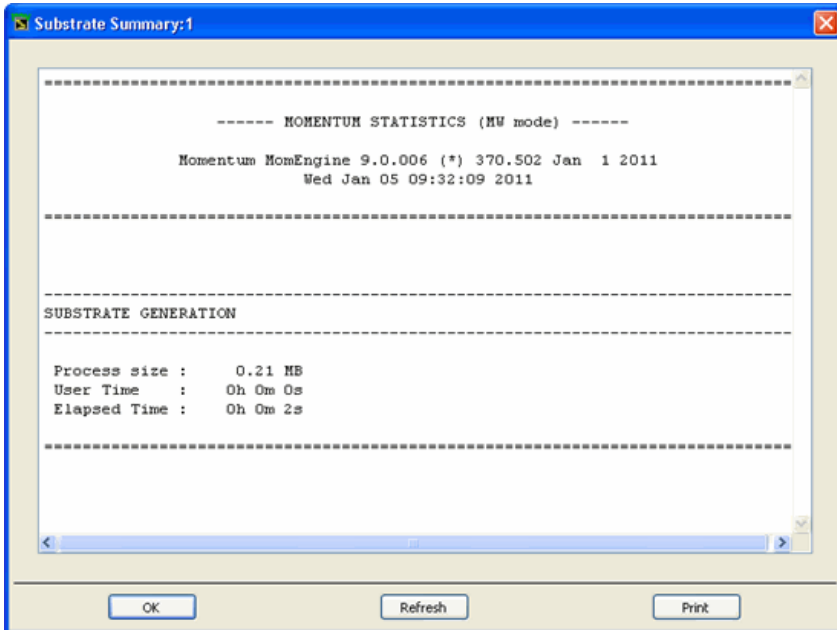


This summary window provides information about the minimum snap distance used for

layout healing and frequency of mesh generation.

Viewing Substrate Generation Summary

You can view the substrate summary by selecting **EM > Show Most Recent > Substrate Generation Summary**. This displays the **Substrate Summary** window. The following example displays a summary window for the Momentum microwave simulation mode:



This summary window provides information about the process size, user time, and elapsed time.


Viewing 3D Simulations

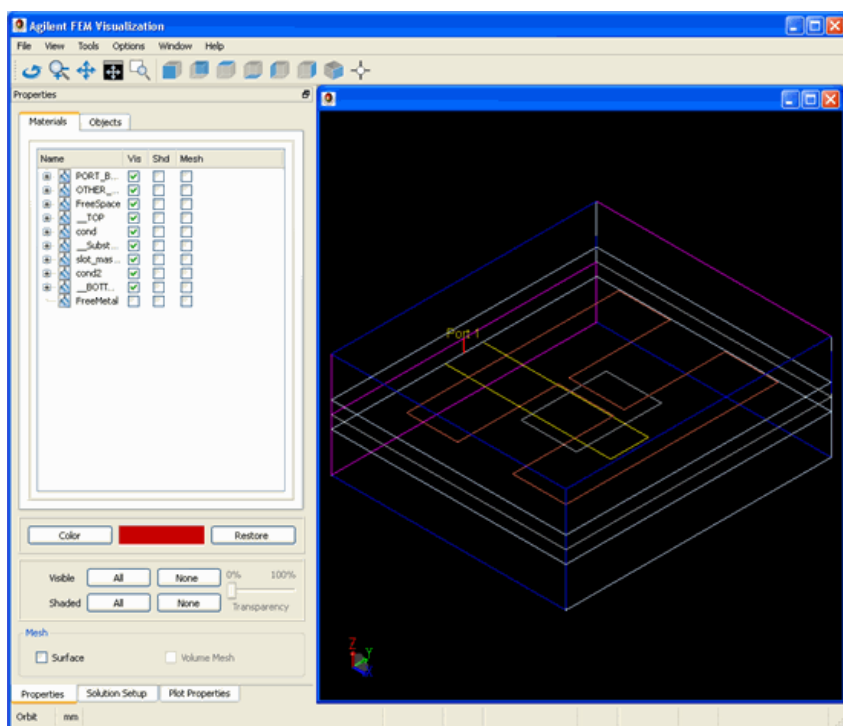
Using the EM visualization feature, you can view and analyze the following types of simulation data:

- S-parameters
- Currents
- far-fields
- Antenna parameters
- Transmission line data

Starting an EM Visualization

Before using the visualization feature, you need to complete the simulation process. After simulating your design, choose **EM > Post-Processing > Visualization** to open the

Visualization window. You can also open this window by clicking **Visualization**  in the EM toolbar. The following figure displays a Visualization window:

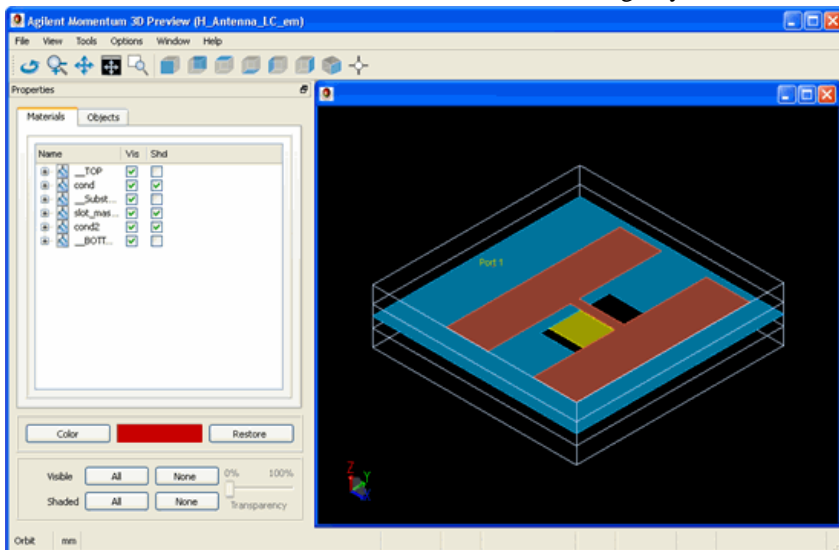


The Visualization window controls the display of the individual mask layers, substrates, and port boundaries. In the Visualization window, the left pane contains the basic controls for the view in the docking widget. In the following sections, you will learn how to preview a simulation and visualize Momentum and FEM fields:

- *Visualizing 3D View before EM Simulations (em)*
- *Visualizing Momentum Simulations (em)*
- *Visualizing FEM Simulations (em)*

Visualizing 3D View before EM Simulation



Before simulating a design, you can validate your three dimensional design in the 3D Preview window. The representation seen in the 3D Preview window consists of the same definition that will be used in the EM simulator. If a view is incorrect in the Preview window, it is essential to redesign before attempting a simulation. The following figure displays a 3D Preview window:



The key benefits of this visual confirmation are:

- **Correct Substrate Set Up:** The substrate information within ADS is defined by using the Substrate menu options, which help you to define the height of each substrate and assign the corresponding layers. The Previewer window enables you to validate that correct height and mappings are specified for your design.
- **Correct Bondwire and Dielectric Brick Placement:** Bondwires, which are described in the schematic window of ADS, are mapped to the ADS layout and incorporated into the overall FEM design for simulation. Dielectric bricks are defined as part of the substrate definition. Since these portions of the design are difficult to see in layout, you can use the Previewer window to validate the location and design of bondwires.

You can create the following types of 3D views:

- **Layout View:** Select **View > 3D View > All** to generate a 3D view of the layout without any EM pre-processing. This is the fastest way to generate a 3D view. You do not create an EM Setup view for generating the 3D view from layout. The first substrate found in the library will be used.
- **Momentum Simulator View:** Click the **3D EM Preview** icon  from the EM toolbar, EM menu, or EM Setup window with Momentum specified as simulator type to generate a 3D view that displays the design in 3D as seen by Momentum, including any specified pre-processing operations.
- **FEM Simulator View:** Click the **3D EM Preview** icon  from the EM toolbar, EM menu, or EM Setup window with FEM specified as simulator type to generate a 3D view that displays the design in 3D as seen by FEM, including any specified pre-processing operations.

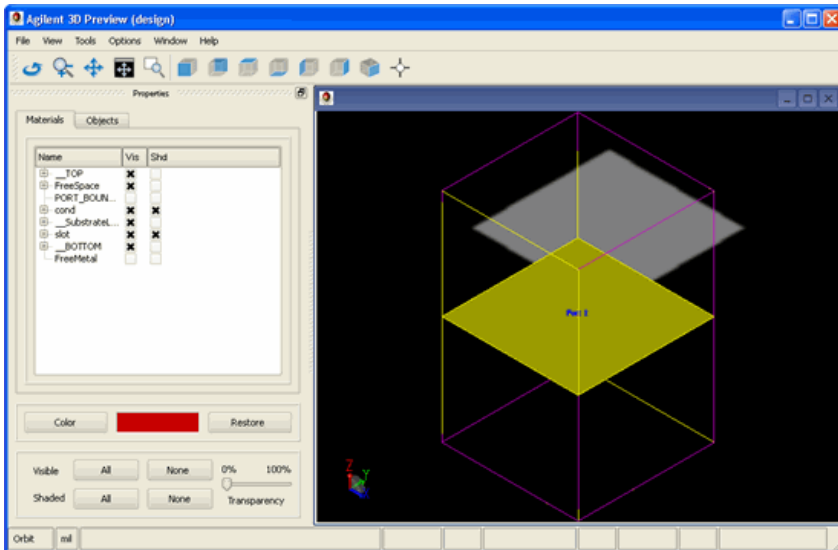
Validating Your Geometry Visually

You can validate a geometry by opening the 3D Preview window in a layout. Before reviewing your design, you need to start the previewer regularly, as it does not synchronize with the current ADS layout.

Note














It is recommended that you do not keep more than one instance of the previewer open at a time. It will not interfere with the data, however, it may be confusing and cause an unnecessary error.

The previewer window contains a dockable widget that controls the display of the individual mask layers, substrates, and port boundaries. Upon startup, a dockable widget is docked on the left-hand side of the previewer, as shown in the following figure:




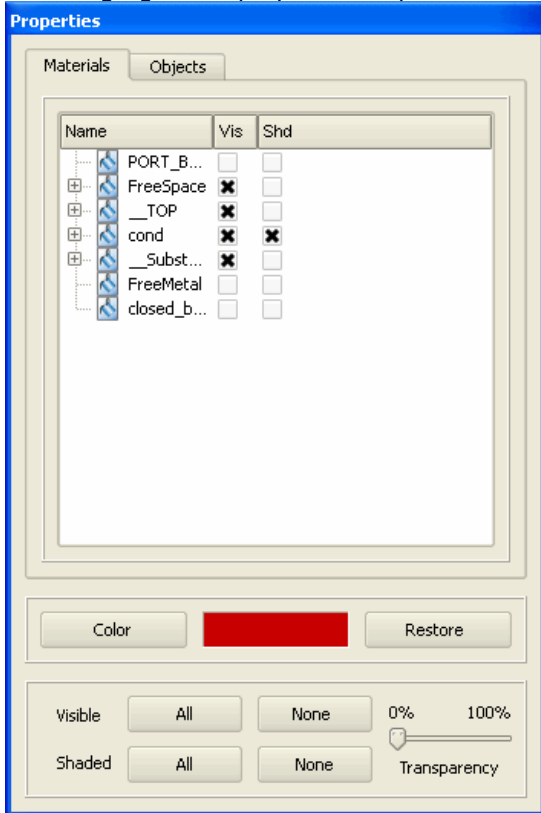
Navigating in the 3D Environment

You can perform the following tasks in the 3D Preview window:

- **Rotating the view:** You can rotate a design around its current origin by holding down the main mouse button and moving it around the screen. Click **Orbit** () on the toolbar or choose **Window > Orbit** to rotate a view.
- **Modifying the zoom settings:** You can increase or decrease the zoom settings on a design by moving the mouse up or down on the image respectively. Click **Zoom** () on the toolbar or select **Window > Zoom** to modify zoom settings.
- **Moving the design:** You can move the design around on the screen by holding down the main mouse button and moving it around the screen. Click **Pan** () on the toolbar or select **Window > Pan** to move a design.
- **Opening the standard view:** You can change the view of a design to the standard view settings. Click **Zoom to Extents** () on the toolbar or select **Window > Zoom to Extents** to open the standard view for your design.
- **Modifying the zoom settings of a specific area:** You can zoom to a specific area of your design by placing a box around the desired view. Click **Zoom to Window** () from the toolbar or select **Window > Zoom to Window** to change the zoom settings.
- **Changing Views:** You can open your design in various types of standard views by clicking **Front** (), **Back** (), **Top** (), **Bottom** (), **Left-side** (), **Right-side** (), and **Isometric** (). You can also access these views from the **View** menu.
- **Querying the design:** You can click an object edge or vertex to display the location and object name in the status bar, which is located in the lower right hand section of the window. In addition, the object is highlighted and the object is automatically selected in the Object and Material tabs in the Docking widget. Visually, a solid dot is placed if the selected location is on a vertex and a hollow dot is placed if the selected location is on an edge. Click **Query** () or select **Tools > Query** to run a query.

Specifying Material and Object Settings

In the 3D Preview window, you can control the material and object settings by using the **Properties** window. Click  to separate the Properties window from the 3D Preview window. To include the Properties dialog box, double-click the Properties title bar. The following figure displays the Properties window:



In the Properties window, you can:

- Select the required material and object.
- Control the visibility and shading settings for all the materials and objects.
- Modify the color settings for a material or object.
- Restore the color settings for a material or object.
- Control the transparency settings.

The following sections provide information about how to select and highlight materials and objects.

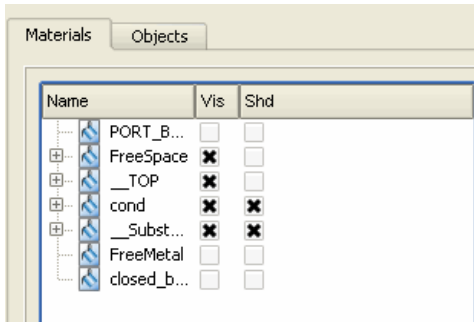
Selecting Materials and Objects

You can select and highlight individual objects by using one of the following methods:

- Selection on the screen: Objects can be selected graphically using the mouse. If the previewer is in Query mode, an object can be selected by clicking on any line or vertex of the object. Once selected, the object lines are highlighted. The object is also selected in the material and object list and the coordinates of the selection point are displayed in the lower right area of the status bar.
- Selection from the material or object list: Objects can be selected from either the material or object list box. Once selected, the object lines are highlighted.

Controlling the Visibility and Shading of Selected Objects and Materials

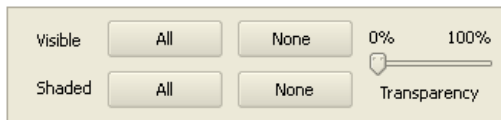
After selecting an object or material, you can control the visibility and shading by using the **Materials** and **Objects** tab, respectively. You can select the **Visibility** and **Shading** fields associated with a material and object to control the visibility and shading, as shown in the following figure:



It is also possible to control the visibility and shading for the substrate and mask layers. Within the Materials portion, each material and object has separate toggles for visibility and shading. By setting these controls appropriately, you can control the visibility and shading for all the objects that share this substrate or mask.

Controlling the Visibility and Shading of All Objects and Materials

You can control the visibility and shading of all the objects and materials by using the All and None buttons, as shown below:



In the Visible area:

- Click **All** to apply visibility to all the objects and materials.
- Click **None** to make all the objects and materials invisible.

In the Shaded area:

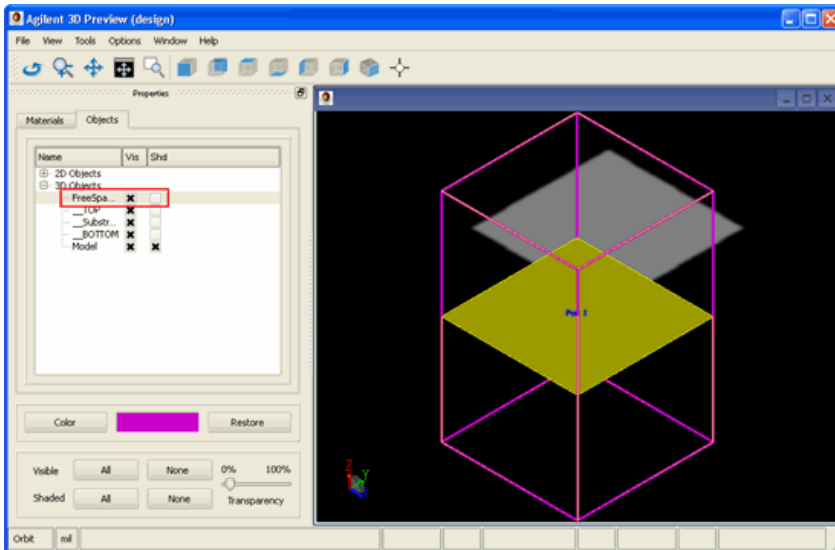
- Click **All** to apply shading to all the objects and materials.
- Click **None** to remove shading from all the objects and materials.

To set the transparency level for objects and materials, specify the transparency level on the range of 0 to 100 percentage.

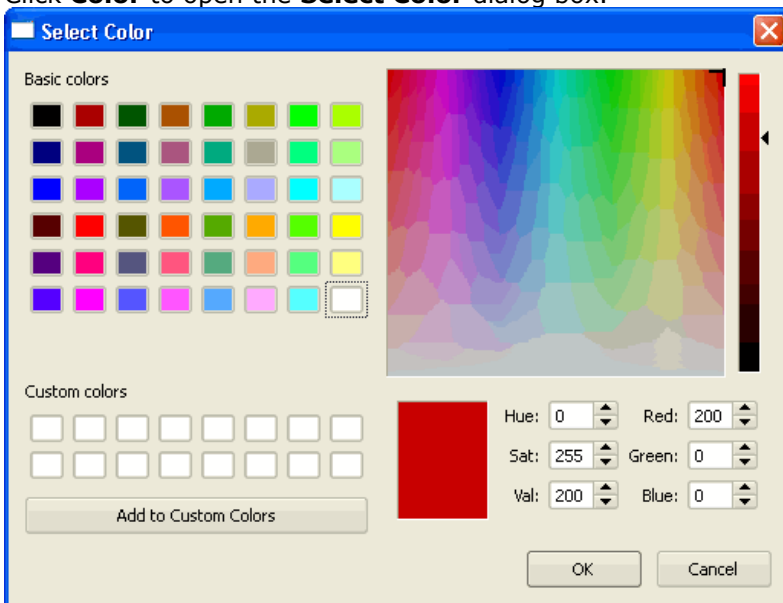
Selecting Color

You can select, change colors, and highlight specific areas in your design. For example, to highlight the free space:

1. Click the check boxes associated with "Free_Space" from the 3D object tree.



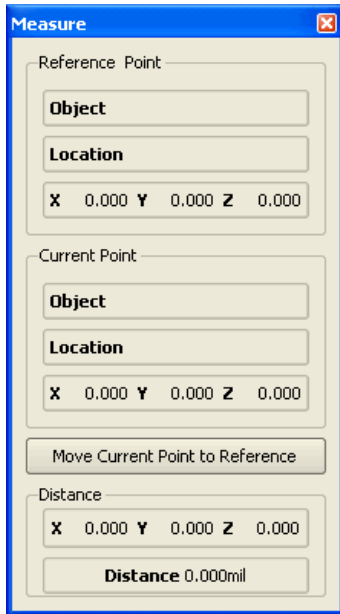
2. Click **Color** to open the **Select Color** dialog box.



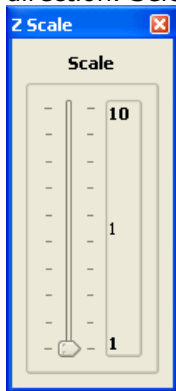
3. Select a color and click **OK**.
4. You can revert the original color by clicking **Restore**.

Customizing the Simulation View

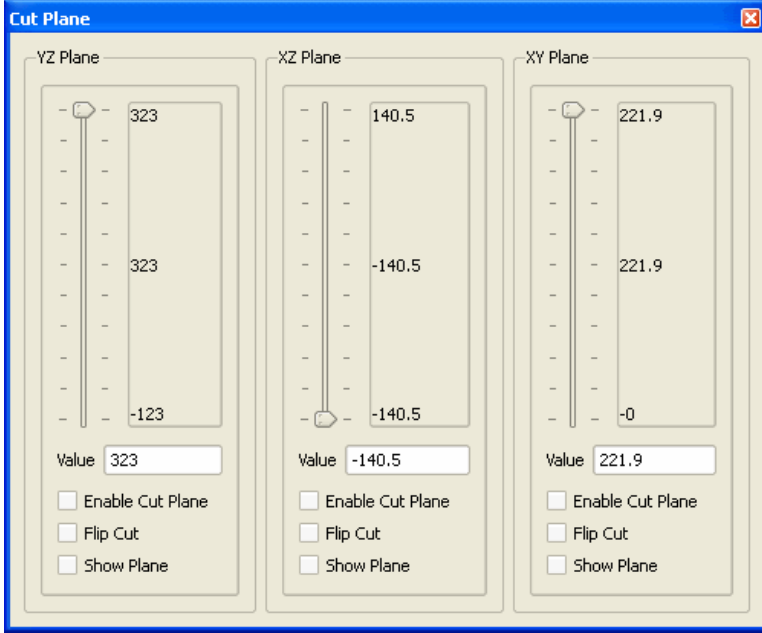
- **Measuring distances:** You can measure the distance between a *reference point* and the current *query point*. The query point is updated after every mouse selection using the query command. The reference point remains fixed until it is explicitly updated using the *Move Current Point to Reference*. Select **Tools > Measure** to open the **Measure** dialog box:



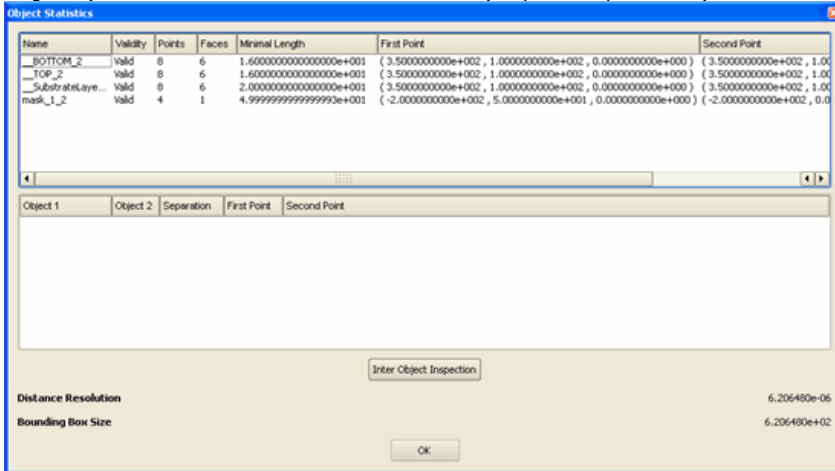
- **Z-Scaling:** You can change the geometry dimension of a model in the z-direction using a sliding scale between 1 and 10. If you move the dimension value of the slide bar up, the model is expanded in the z-direction. Select **Tools > Z Scale** to open the **z-scale** dialog box.



- **Cutting Plane:** This feature enables you to slice through your design in the YZ, XZ, and XY planes. The check boxes associated with each slide bar activate the cut. It allows you to flip the cut and to show the plane as it moves through the design. Selecting **Tools > Cut Plane** opens the **Cut Plane** window, as shown below:



- **Viewing 3D Connectivity:** You can select the required object and choose **Tools > View 3D Connectivity**.
- **Viewing Object Statistics:** You can view detailed information about the 2D and 3D object parameters such as substrate layer, mask, and top statistics.



Setting Up the Viewer on External X Window Displays

On Unix/Linux Systems, it is common practice to use a local machine as a display and do the actual processing on another machine. The display from the original machine is typically mapped back to the local display using the following command:

```
export DISPLAY=machine:0.0
```

In the 3D Previewer window, you need to set an additional environment variable, *HOOPS_PICTURE*. This environment variable is set to the same value as the display variable with the addition of X11/ prior to the machine name. Using the above example, the *HOOPS_PICTURE* variable is set to the following values:

```
export HOOPS_PICTURE=X11/machine:0.0
```

Visualizing Momentum Simulations

You can view and analyze the following type of data for Momentum simulations:

- S-parameters
- Currents
- far-fields
- Antenna parameters
- Transmission line data

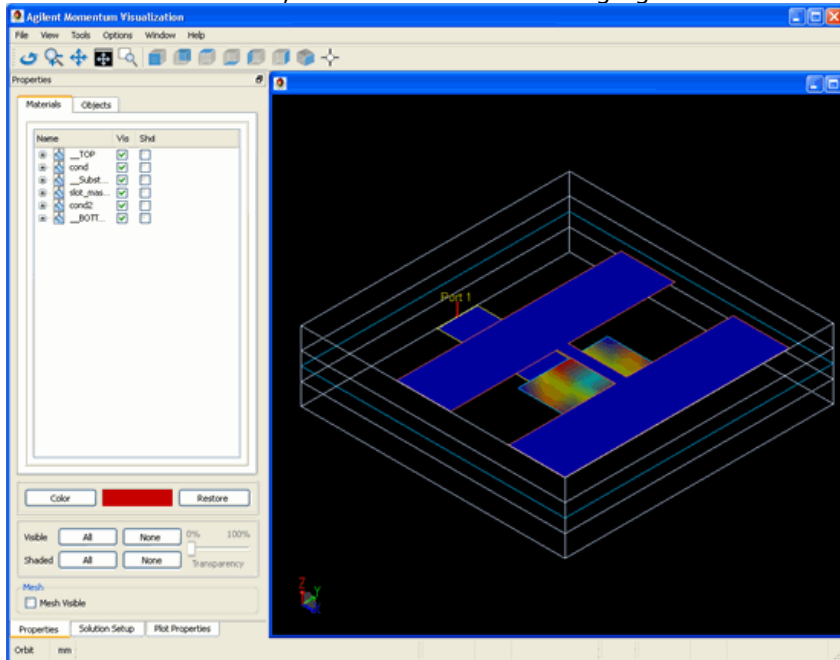
Starting Momentum Visualization

You must complete the simulation process to view data for a Momentum design. If you have already simulated a design, you can start the Visualization feature directly to view the existing data. Select **EM > Post-Processing > Visualization** to open the **Agilent Momentum Visualization** window. You can also open this window by clicking

Visualization () in the EM toolbar.

To visualize Momentum simulations:

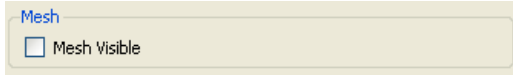
1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Momentum RF** or **Momentum Microwave** in the EM Setup window.
3. Specify the required settings in the EM Setup window for a layout, substrate, port, frequency plan, output plan, options, resources and model or symbol.
4. Click **Simulate** in the EM Setup window to simulate your design.
5. Select **EM > Post-Processing > Visualization** to open the **Agilent Momentum Visualization** window, as shown in the following figure:



The Agilent Momentum Visualization window appears similar to the EM Previewer window. The initial window layout is similar to that of EM Previewer. The left side contains the basic controls for the view in the docking widget. However, the Agilent Momentum Visualization window includes the following modifications:

- EM Port surfaces are no longer displayed.
- Object shading is removed.
- A shaded current plot for Port 1 at the lowest frequency is displayed.
- In addition to the Properties tab, Solution Setup and Plot Properties tabs are added for controlling the view.

In the Agilent Momentum Visualization window, the Properties tab provides the same functionality as the EM Previewer properties tab. However, in the Agilent Momentum Visualization window, you can also display mesh by selecting the **Mesh Visible** check box in the Properties tab, as shown in the following figure:



Note
For more information about the Properties tab, refer to *Visualizing 3D View before EM Simulations (em)*.

You can plot the field quantities by using the **Plot Properties** and **Solution Setup** tabs. The Solution Setup tab controls the excitations for the visualization, while the Plot Properties tab controls the visual display of the excitation.

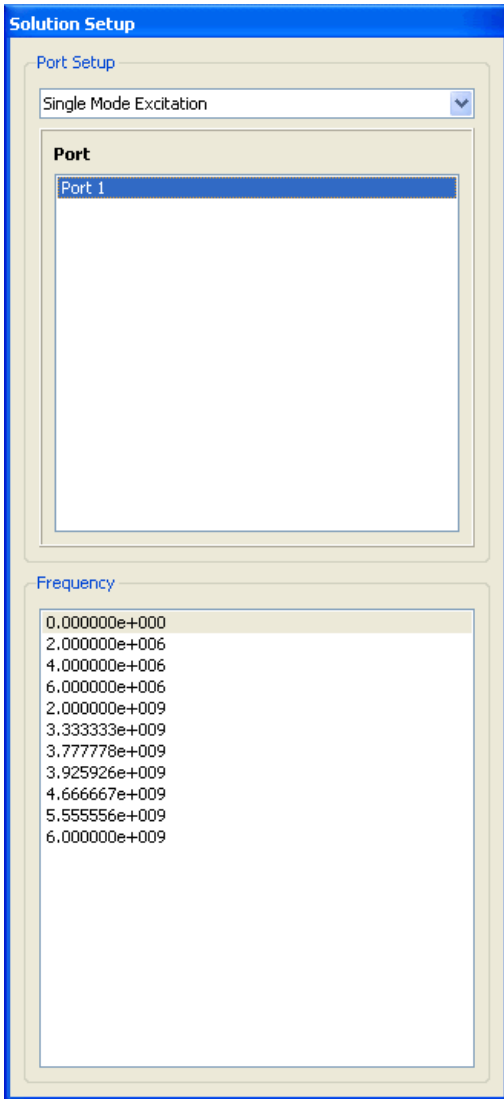
Controlling Visualization Excitations

The Solution Setup tab is used to select the current excitation for the plots. All the plots automatically reflect the current solution configuration once it is selected. By selecting either a port or frequency, the excitation is changed and the plots are automatically updated.

In the **Agilent Momentum Visualization** window, select the **Solution Setup** tab present at the bottom of the window. In this tab:

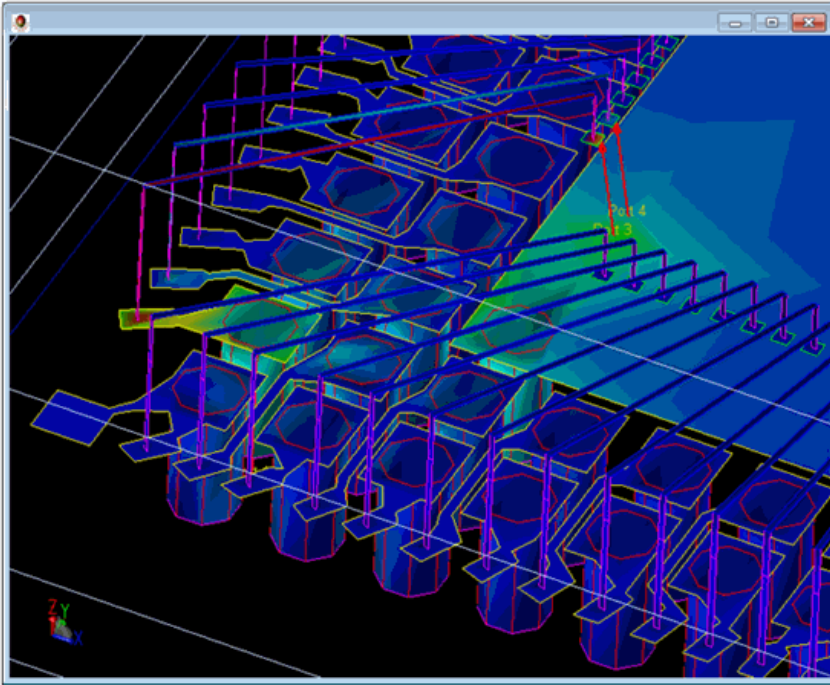
- Define the port excitation value from the drop-down list in the **Port Setup** region. You can either select **Single Mode Excitation** for ports to be excited individually or **Multiple Mode Excitation**.
- View the frequency changes in the **Frequency** region. After the port or frequency selection is changed, the plots are automatically updated using the new configuration.

The following figure displays a Solution Setup tab:



Plotting Properties

The Plot Properties tab enables you to control the three basic plots, as well as, the animation settings. This tab is used to set up plots within the visualization tool. The visualization tool plots J, which is the current calculated within the Momentum solvers, as shown in the following figure:



Using the Plot Properties tab, you can create the following types of plots:

- Shaded Plot
- Arrow Plot
- Contour Plot

Displaying a Shaded Plot

Displaying the shaded current plot is controlled by using the check box next to the plot name. When it is selected, the plot is visible. Within the plot there are some basic controls:

- Log Scale - This controls whether the scaling and color representation uses a logarithmic scale or a linear one.
- Transparency - This controls the transparency of the shaded plot.

Displaying an Arrow Plot

After selecting the **Arrow** tab, select **Enable** to display the arrow plot. You can control the following properties of an arrow plot:

- Scaling arrows: You can select the **Scale** check box to control whether the arrows are scaled, based on the relative magnitude of the current density through out the design. When it is selected, the arrows will be scaled, making the lower current density areas have smaller arrows. If it is not selected, all the arrows are displayed with the same size. However, their size can still be changed by changing the arrow size.
- Using a logarithmic scale: You can select the **Log Scale** check box to control whether the scaling and color representation use a logarithmic scale or a linear one. If scaling is not enabled, only the color weighting is affected.
- Specifying arrow size: You can specify the relative size of the arrow in **Arrow Size**. Remember that if the arrows are not scaled, the default size of the arrows appear to be larger than when the arrows are scaled.

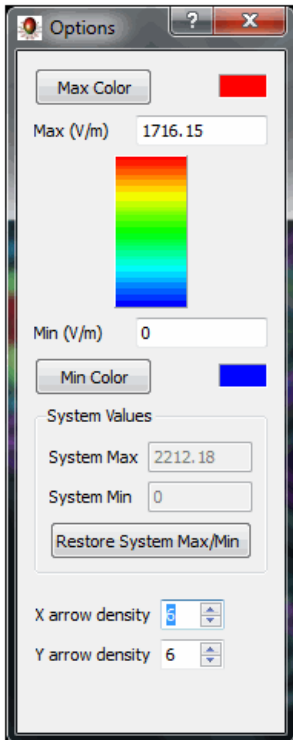
Displaying a Contour Plot

After selecting the **Contour** tab, select **Enable** to display the contour plot. You can control the following properties of a contour plot:

- Using a logarithmic scale: You can select the **Log Scale** check box to control whether the scaling and color representation use a logarithmic scale or a linear one. If scaling is not enabled, only the color weighting is affected.
- Specifying the number of divisions: You can specify the number of divisions in the **Divisions** combo box.

Modifying Plot Options

You can modify the plots by using the options menu in the Plot Properties tab. The options menu displays the color key for the plot and can also be used to change the Minimum and Maximum values that are being plotted. The color used for the plot can also be specified from the Options dialog box.



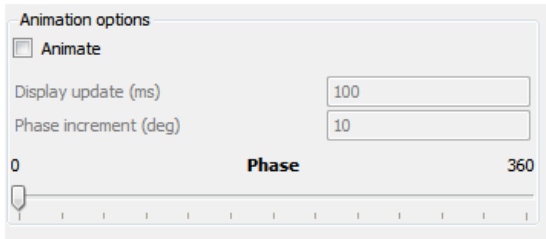
System Min and System Max: The system minimum and maximum values represent the minimum values for all the plots that are currently being drawn. However, in order to keep scale consistently, maximum and minimum values are not changed when the Solution Setup is modified. The maximum and minimum values are displayed as System Max and System Min. If the user wishes to use these values instead, he can use them by selecting the Restore Max and Min button.

Similarly, the user can modify the maximum and minimum values that are used for displaying the data by typing in new values. These will not be changed as the Solution Setup is modified.

X and Y Arrow Density control the density of arrows within a sensor. They have no effect on Object Surface Sensors.

Animating Momentum Fields

You can animate Momentum fields by selecting **Animate** at the bottom of the Plot Properties tab. You can also change the phase by sliding the Phase Bar, as shown in the following figure:

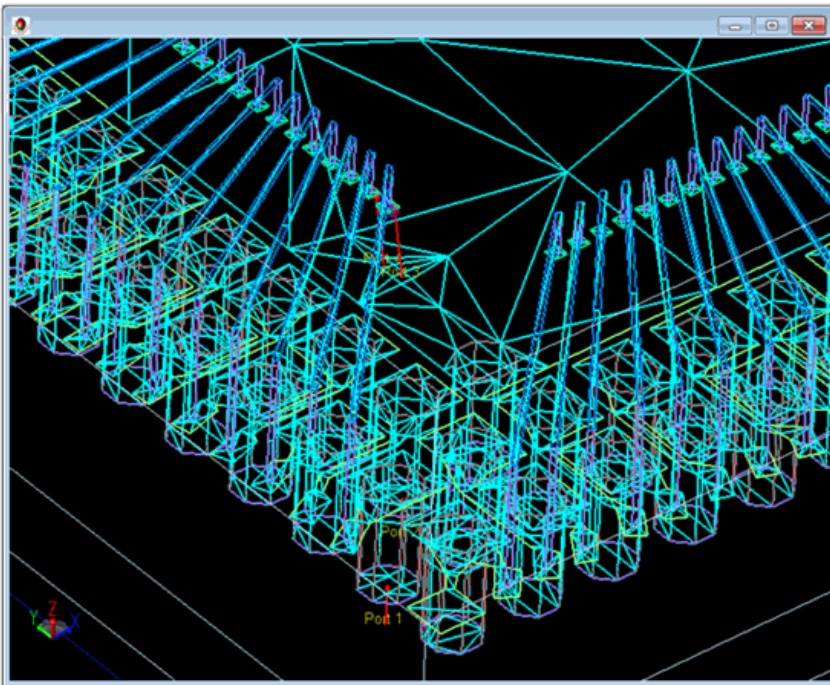


You can control the following animation options of arrow, shaded, and contour plots:

- Determine the display update time: Specify a value in the **Display Update** text box to determine the minimal time required between display updates in milliseconds. Since some updates may take longer than this setting, this value is only a minimum number and not an absolute one.
- Determine the Phase Increment value: Specify a value in the Phase Increment text box to control the number of degrees added to the current phase when an update occurs.

Plotting Momentum Mesh

When a solution has been loaded, the momentum mesh can be displayed by checking the Mesh Visible box on the Properties Tab. The displayed mesh is the one which is used to display the field. This will most likely be more detailed than the one seen in the Layout View, which is the reduced mesh that is used in the Momentum solution computation.



Visualizing FEM Simulations


You can view and analyze the following type of data for FEM simulations:

- E Fields
- Far-fields
- Antenna parameters
- Transmission line data

Starting FEM Visualization

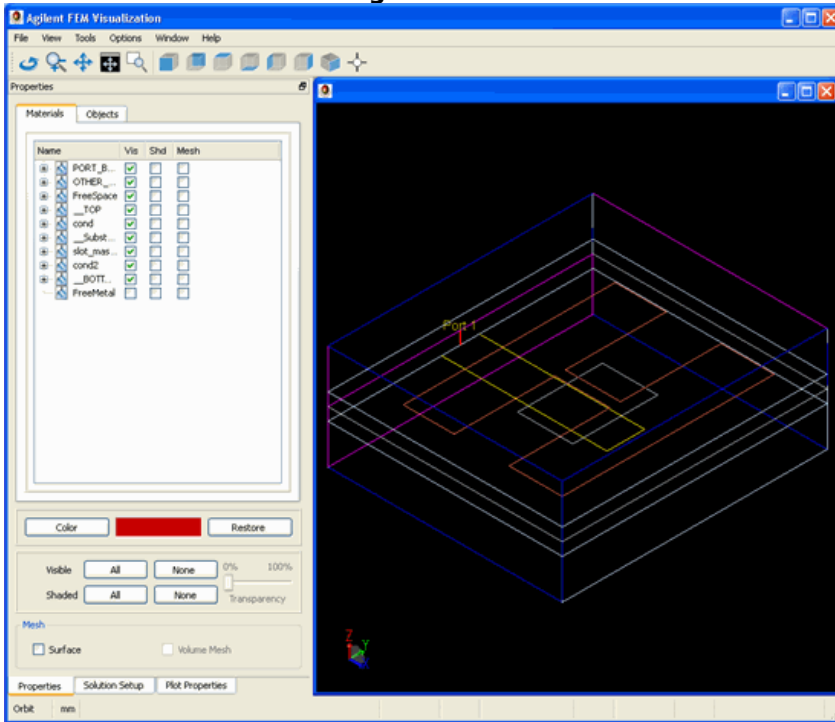
You must complete the simulation process to view data for your FEM design. If you have already simulated a design, start the Visualization feature directly to view the existing

data. Select **EM > Post-Processing > Visualization** to open the **Agilent FEM**

Visualization window. You can also open this window by clicking **Visualization** () in the EM toolbar.

To visualize FEM simulations:

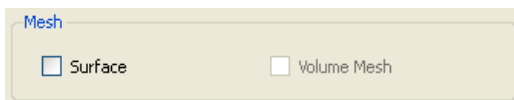
1. Choose **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **FEM** in the EM Setup window.
3. Specify the required settings in the EM Setup window for a layout, substrate, port, frequency plan, output plan, options, resources and model or symbol.
4. Click **Simulate** in the EM Setup window to simulate your design.
5. Select **EM > Post-Processing > Visualization**.



The Agilent FEM Visualization window appears similar to the EM Previewer window. The initial window layout is similar to that of an EM Previewer. The left side contains the basic controls for the view in the docking widget. However, the Agilent FEM Visualization window includes the following modifications:

- EM Port surfaces are no longer displayed.
- Object shading is removed.
- A shaded current plot for Port 1 at the lowest frequency is displayed.
- In addition to the Properties tab, Solution Setup and Plot Properties tabs are added for controlling the view.

The Properties tab provides the same functionality as the Previewer properties tab. However, in the **Agilent FEM Visualization** window, you can specify the type of mesh: **Surface** or **Volume Mesh**, as shown in the following figure:



Note
For more information about the Properties tab, refer *Visualizing 3D View before EM Simulations (em)*.

You can display the field quantities by using the Plot Properties and Solution Setup tabs. The Solution Setup tab controls the excitations for the visualization, while the Plot Properties tab controls the visual display of the excitation.

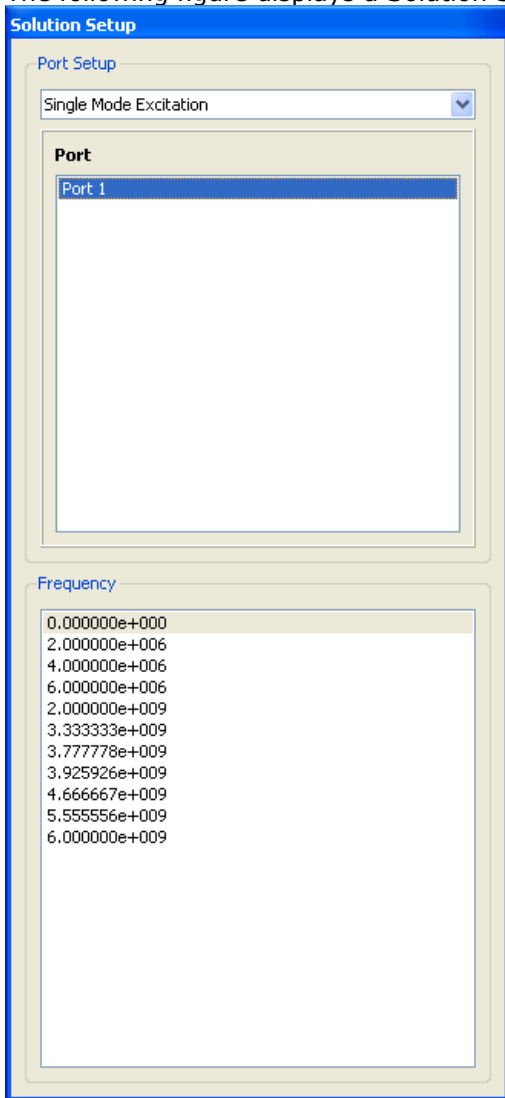
Controlling Visualization Excitations

The Solution Setup tab is used to select the current excitation for the plots. All the plots automatically reflect the current solution configuration once it is selected. By selecting either a port or frequency, the excitation is changed and the plots are automatically updated.

In the **Agilent FEM Visualization** window, select the **Solution Setup** tab present at the bottom of the window. In this tab:

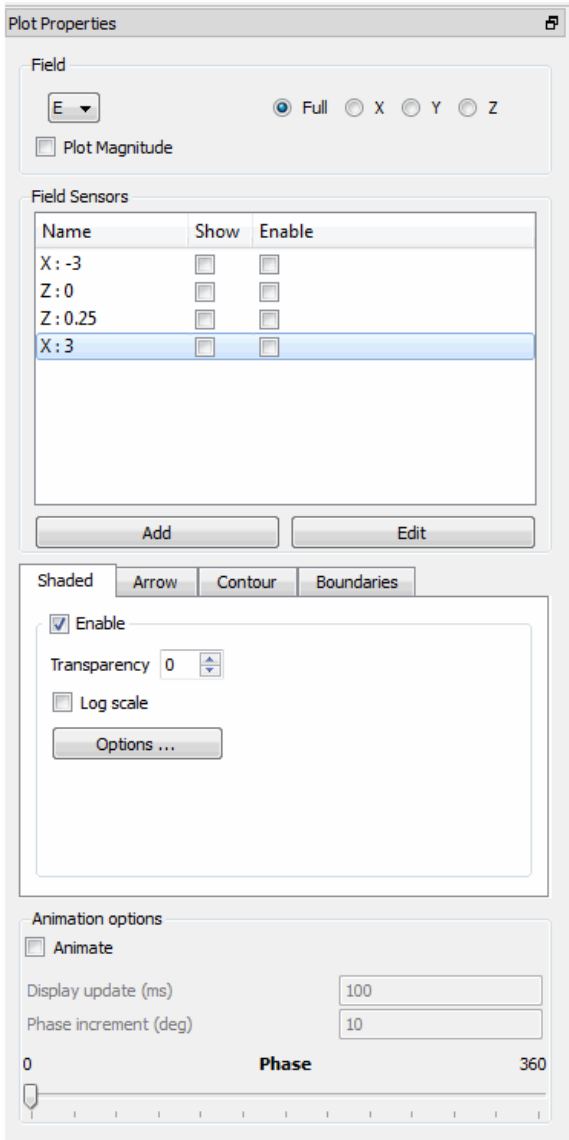
- Define the port excitation value from the drop-down list in the **Port Setup** region. You can either select **Single Mode Excitation** for ports to be excited individually or **Multiple Mode Excitation**.
- View the frequency changes in the **Frequency** region. After the port or frequency selection is changed, the plots are automatically updated using the new configuration.

The following figure displays a Solution Setup tab:



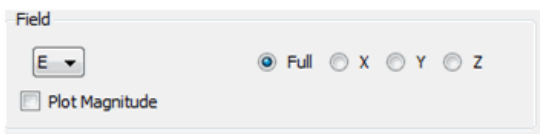
Setting up Plots

The Plot Properties tab enables you to select fields, field sensors, basic plots, and animation settings. The following figure displays the Plot Properties tab:



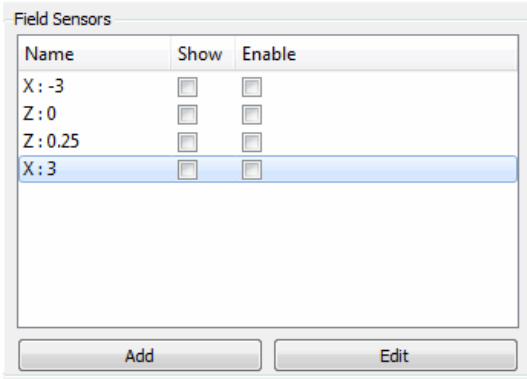
Selecting Field Types

You can select the required field quantity (**E, H, or J**) to plot as well as the Vector component. All the field quantities are represented as steady state sinusoidal waves, so the field plots will be done at a specified phase. If you want to include the total field magnitude, select the **Plot Magnitude** check box.



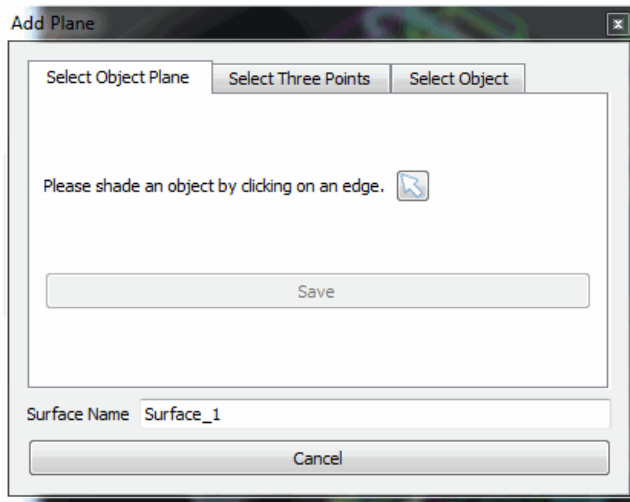
Enabling Field Sensors

All the plots are displayed on surfaces. By default, surfaces that are connected to Ports are automatically created and listed in the Sensor Frame.

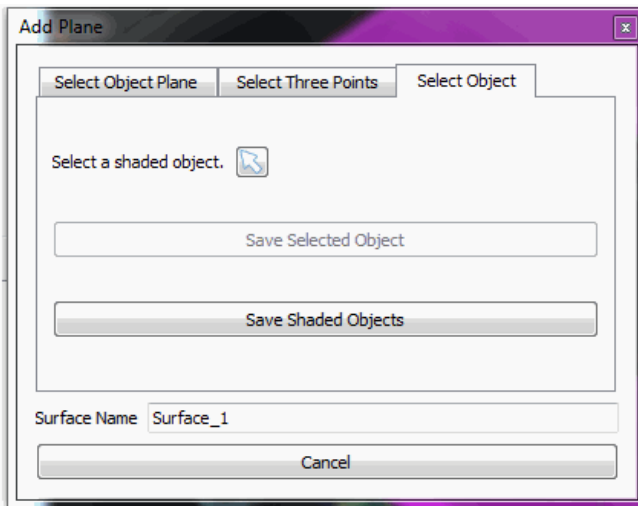


The Field Sensors region consists of two columns, **Show** and **Enable**. The Show check box allows you to display the triangular regions where the field quantities are plotted. The Enable check box allows you to plot the field quantities on that Sensor. The fields that are plotted are determined by the field plotting choices, Shaded, Arrow or Contour.

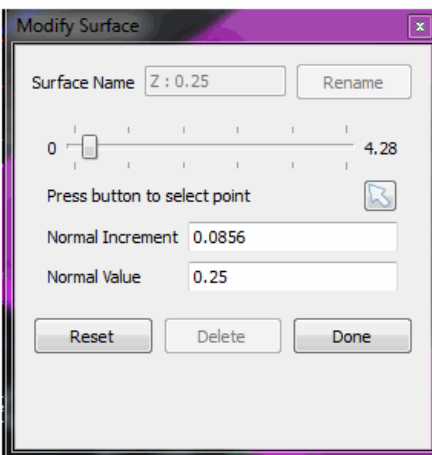
You can add new sensors by clicking the **Add** button. There are three different options for adding new sensors:



- **Object Plane:** You can select a shaded object by clicking on a face. If no object is currently shaded, you should first select an object edge to shade the surface. When an object plane is selected, a plane is defined through the entire design region. In this case, the plane will extend beyond the object definition.
- **Three Point Plane:** You can select three points that determine plane.
add_plane2.gif
- **Object Surface:** You can select a shaded object by clicking on a face. If no object is currently shaded, you must first select an object edge to shade the surface. Only the surface of the object is used. Unlike the object plane mode, the surface does not have to be planar. A second option also allows you to select all shaded objects.



Using the Edit button, you can rename a plane or move it within the design area. Object Surfaces cannot be moved since they are assigned to a specific object and not a location. You cannot rename or delete predefined planes.



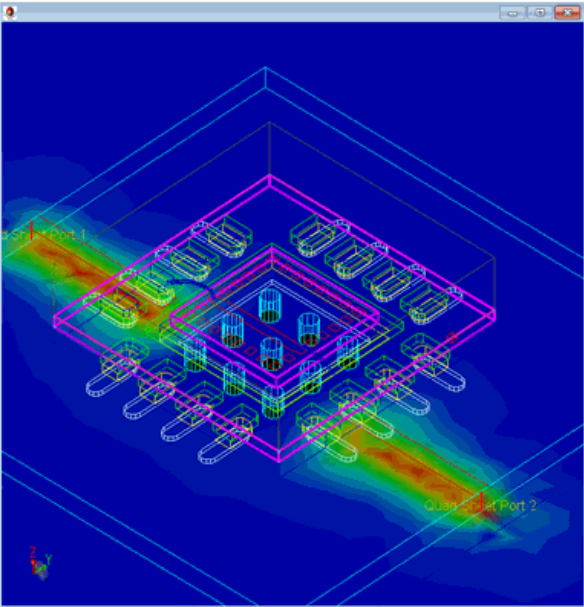
Plotting Properties

Using the Plot Properties tab, you can create the following types of plots:

- Shaded Plot
- Arrow Plot
- Contour Plot

Displaying a Shaded Plot

The shaded plot allows you to plot the magnitude of the field quantity on the sensor surface, as shown in the following figure:

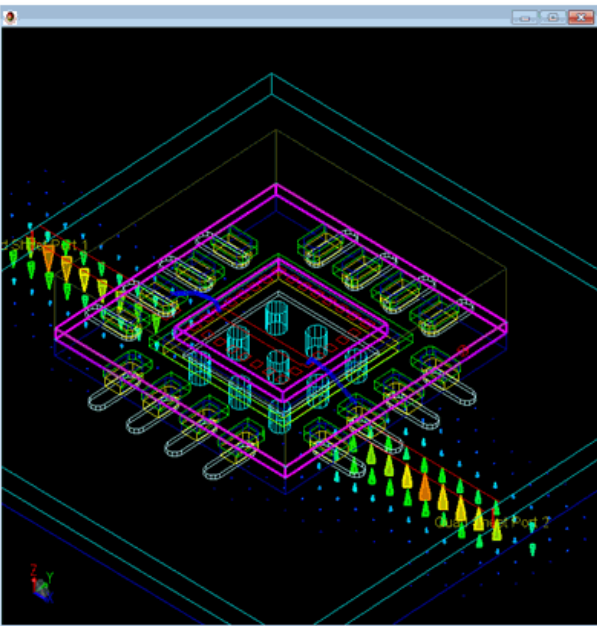


Displaying the shaded current plot is controlled by using the check box next to the plot name. When it is selected, the plot is visible. Within the plot there are some basic controls:

- Log Scale: This controls whether the scaling and color representation uses a logarithmic scale or a linear one.
- Transparency: This controls the transparency of the shaded plot.

Displaying an Arrow Plot

The arrow plot allows you to plot the quantity on the sensor surface, as shown in the following figure:



After selecting the **Arrow** tab, select **Enable** to display the arrow plot. You can control the following properties of an arrow plot:

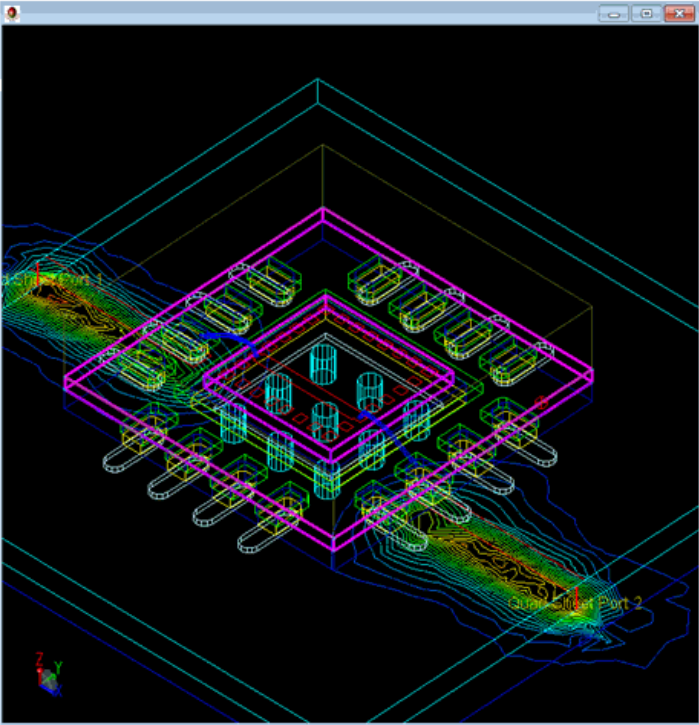
- Scaling arrows: You can select the **Scale** check box to control whether the arrows are scaled, based on the relative magnitude of the current density through out the design. When it is selected, the arrows will be scaled, making the lower current density areas have smaller arrows. If it is not selected, all the arrows are displayed

with the same size. However, their size can still be changed by changing the arrow size.

- Using a logarithmic scale: You can select the **Log Scale** check box to control whether the scaling and color representation use a logarithmic scale or a linear one. If scaling is not enabled, only the color weighting is affected.
- Specifying arrow size: You can specify the relative size of the arrow in **Arrow Size**. Remember that if the arrows are not scaled, the default size of the arrows appear to be larger than when the arrows are scaled.

Displaying a Contour Plot

The contour plot allows you to plot the magnitude of the field quantity on the sensor surface, as shown in the following figure:

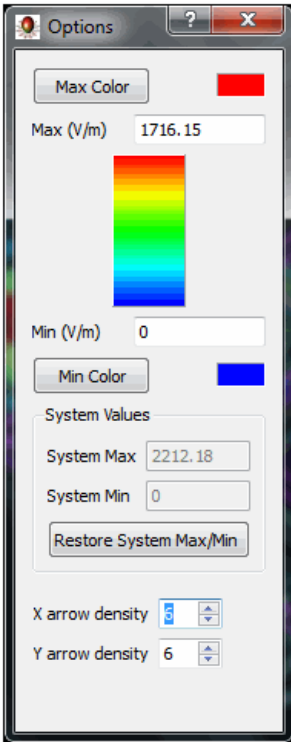


After selecting the **Contour** tab, select **Enable** to display the contour plot. You can control the following properties of a contour plot:

- Using a logarithmic scale: You can select the **Log Scale** check box to control whether the scaling and color representation use a logarithmic scale or a linear one. If scaling is not enabled, only the color weighting is affected.
- Specifying the number of divisions: You can specify the number of divisions in the **Divisions** combo box.

Specifying Plot Options

You can modify the plots by using the options menu. The options menu displays the color key for the plot and can also be used to change the Minimum and Maximum values that are being plotted. The color used for the plot can also be specified from the Options dialog box.

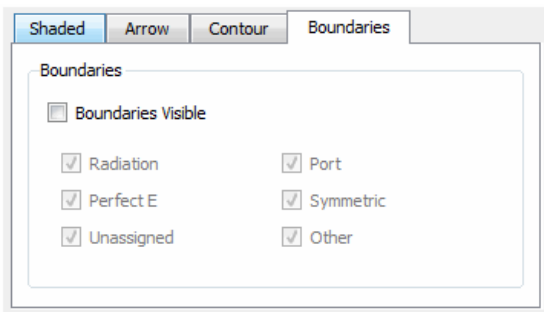


System Min and System Max: The system minimum and maximum values represent the minimum values for all the plots that are currently being drawn. However, in order to keep scale consistently, the maximum and minimum values are not changed when the Solution Setup is modified. The maximum and minimum values are displayed as System Max and System Min. If you want to use these values instead, click the Restore Max and Min button.

Similarly, you can modify the maximum and minimum values that are used for displaying the data by typing in new values. These will not be changed as the Solution Setup is modified.

X and Y Arrow Density control the density of arrows within a sensor. They have no effect on Object Surface Sensors.

Plotting Boundary Conditions



Use the Boundary Conditions tab in the plotting regions to plot the boundary surfaces. By selecting the Boundaries Visible box, you can pick the boundary surfaces that are visible on the screen. The unassigned surfaces are those which are on the surface of objects, but do not have any boundary conditions assigned to them.

Plotting 3D Mesh

When a solution is loaded, a third column, Mesh, becomes available in the Properties tab. By selecting the Mesh column check box, the mesh inside the material or individual object can now be seen. In some cases, there may not be any mesh inside the object if the

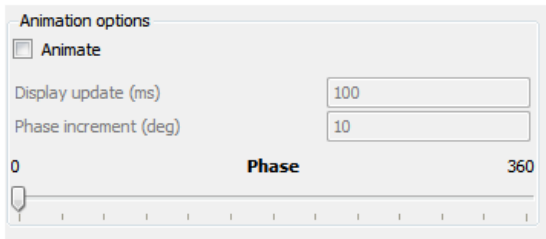
object was not assigned any tetrahedral. Flat objects, by definition, do not have any tetrahedral assigned to them.

Shaded Mesh

The surface mesh of the objects can be drawn by selecting the Surface Check box in the Mesh box at the bottom of the Properties Tab. Once selected, the volume mesh can also be selected.

Animating FEM Fields

You can animate the FEM fields by selecting the **Animate** box at the bottom of the Plot Properties Tab. If the Plot Magnitude button is selected, the animation option is not available. X and Y Arrow Density control the density of arrows within a sensor. They have no effect on Object Surface Sensors. You can also change the phase by sliding the Phase Bar, as shown in the following figure:



You can control the following animation options of arrow, shaded, and contour plots:

- Determine the display update time: Specify a value in the **Display Update** text box to determine the minimal time required between display updates in milliseconds. Since some updates may take longer than this setting, this value is only a minimum number and not an absolute one.
- Determine the Phase Increment value: Specify a value in the Phase Increment text box to control the number of degrees added to the current phase when an update occurs.

Computing Radiation Patterns

This section 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.

Note

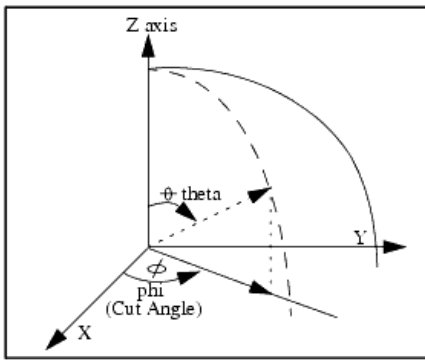
In the Momentum RF mode, radiation patterns and antenna characteristics are not available.

About Radiation Patterns

After calculating currents on the circuit, you can compute the electromagnetic fields. They can be expressed in the spherical coordinate system attached to your circuit. The electric and magnetic fields contain terms that vary as $1/r$, and $1/r^2$. It can be shown that the terms that vary as $1/r^2$, and $1/r^3$ 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 (E_{ff} , H_{ff}).

$$\vec{E}(r \rightarrow \infty, \theta, \varphi) = \vec{E}_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$

$$\vec{H}(r \rightarrow \infty, \theta, \varphi) = \vec{H}_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$



In a layout, there is a fixed coordinate system such that the monitor screen lies in the XY-plane. The X-axis is horizontal, the Y-axis is vertical, and the Z-axis is normal to the screen.

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 (θ , φ) 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.

About Antenna Characteristics

Based on the far 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_θ, E_φ) . 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.

$$\vec{E}_{ff}(\theta, \varphi) = E_\theta(\theta, \varphi)\hat{i}_\theta + E_\varphi(\theta, \varphi)\hat{i}_\varphi = E_{co}(\theta, \varphi)\hat{i}_{co} + E_{cross}(\theta, \varphi)\hat{i}_{cross} = E_{lhp}(\theta, \varphi)\hat{i}_{lhp} + E_{rhp}(\theta, \varphi)\hat{i}_{rhp}$$

$$H_\varphi = \frac{E_\theta}{Z_\omega}$$

$$H_\theta = -\frac{E_\varphi}{Z_\omega}$$

$$Z_\omega = \sqrt{\frac{\mu}{\epsilon}}$$

Z_ω is the characteristic impedance of the open half sphere under consideration.

The fields can be normalized with respect to:

$$\max(\sqrt{|E_\theta|^2 + |E_\varphi|^2})$$

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.

$$E_{lhp} = \frac{1}{\sqrt{2}}(E_\theta - jE_\varphi)$$

$$E_{rhp} = \frac{1}{\sqrt{2}}(E_\theta + jE_\varphi)$$

$$AR_{cp} = \frac{|E_{lhp}| + |E_{rhp}|}{|E_{lhp}| - |E_{rhp}|}$$

Linear Polarization

Below, the equations to decompose the far-fields into a co and cross polarized field are given (α 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.

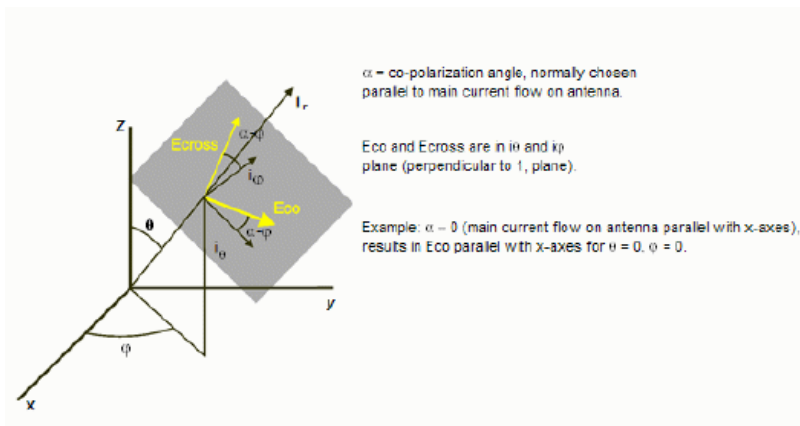
$$E_{co} = E_{\theta} \cos(\alpha - \varphi) + E_{\varphi} \sin(\alpha - \varphi)$$

$$E_{cross} = (-E_{\theta}) \sin(\alpha - \varphi) + E_{\varphi} \cos(\alpha - \varphi)$$

$$AR_{lp} = \frac{|E_{co}| + |E_{cross}|}{|E_{co}| - |E_{cross}|}$$

Note

E_{co} is defined as colinear and E_{cross} implies a component orthogonal to E_{co} . For a perfect linear polarized antenna, E_{cross} is zero and the axial ratio $AR=1$. If $E_{cross} = E_{co}$ you no longer have linear polarization but circular polarization, resulting in $AR = \text{infinity}$.



Radiation Intensity

The radiation intensity in a certain direction, in watts per steradian, is given by:

$$U(\theta, \varphi) = \frac{1}{2} (E_{ff}(\theta, \varphi) \times H_{ff}^*(\theta, \varphi)) = \frac{1}{2Z_w} (|E_{\theta}(\theta, \varphi)|^2 + |E_{\varphi}(\theta, \varphi)|^2)$$

For a certain direction, the radiation intensity will be maximal and equals:

$$U_{max} = \max_{\theta\phi} (U(\theta, \varphi))$$

Radiated Power

The total power radiated by the antenna, in Watts, is calculated from the free space far-field E and H components (E_{ff} , H_{ff}) integrating the radiation intensity over the open half space(s):

$$P_{rad} = \int_{\Omega} U(\theta, \varphi) \cdot d\Omega = \frac{1}{2} \int_{\Omega} E_{ff} \times H_{ff}^* \cdot d\Omega$$

Note

The power going into the surface wave is not taken into account in this calculation.

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 looks at radiation intensity in a certain direction versus the radiated power. Directivity is dimensionless and is calculated as:

$$D(\theta, \varphi) = 4\pi \frac{U(\theta, \varphi)}{P_{rad}}$$

Directivity indicates the best you could get out of the antenna. In reality, there will be loss mechanisms in the antenna (see Gain) and the feed network is never perfectly matched. The maximum directivity is given by:

$$D = 4\pi \frac{U_{max}}{P_{rad}} = \frac{4\pi}{\Omega_A}$$

Gain

Gain looks at radiation intensity in a certain direction versus the power injected into the antenna. The gain of the antenna is calculated as:

$$G(\theta, \varphi) = 4\pi \frac{U(\theta, \varphi)}{P_{inj}}$$

where P_{inj} is the real power, in watts, injected into the circuit. The injected power, or net input power, equals to the (incident - reflected) power at the port(s). Consequently, the impact of mismatch is not included in this definition of Gain. For a good antenna, a large part of the injected power is radiated, the rest will be lost. The loss is due to the presence of lossy materials (either the conductors or the dielectrics used) or gets trapped into the surface wave that propagates cylindrically along the substrate plane.

The injected power is calculated from the S-parameters and the excitation sources (Thevenin sources) at the ports as specified by the user. In case of a 1-port antenna, the excitation source specified by the user doesn't really matter since both Directivity and Gain are normalized values. For a multi-port antenna, the exact source excitation setup at each port is really important. That is where the far field excitation utility available from the knowledge center (search for Far Field Computations from AC/HB Analysis of Antennas Plus Circuitry) is very relevant.

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

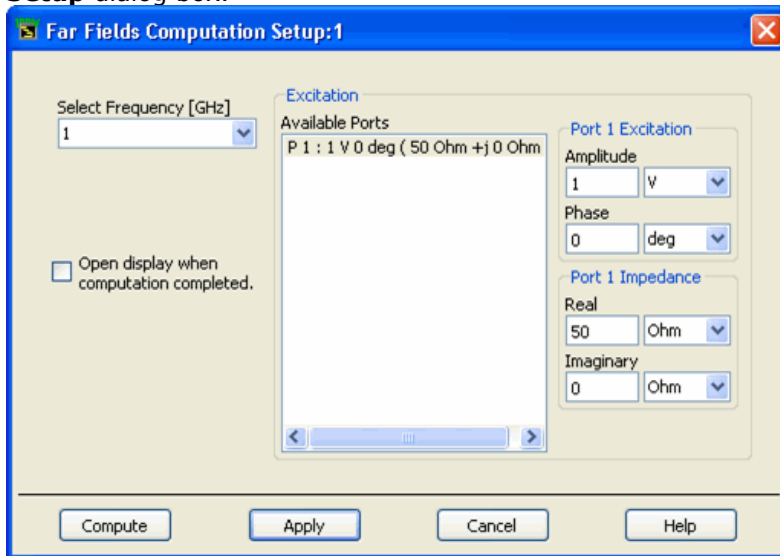
The effective area, in square meters, of the antenna circuit is given by:

$$A_{eff}(\theta, \varphi) = \frac{\lambda^2}{4\pi} G(\theta, \varphi)$$

Calculating Far Fields

To calculate far fields:

1. Choose **EM > Post-Processing > Far Field** to open the **Far Field Computation Setup** dialog box.



2. Select a frequency from the **Select Frequency** list. The list contains all the frequencies that were calculated during the ongoing simulation. Far fields will be calculated at the selected frequency.
3. 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.
 - 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.
 - In the **Port Impedance** fields, enter the real and imaginary components of the impedance of the steering circuit that is applied to the port.

Note

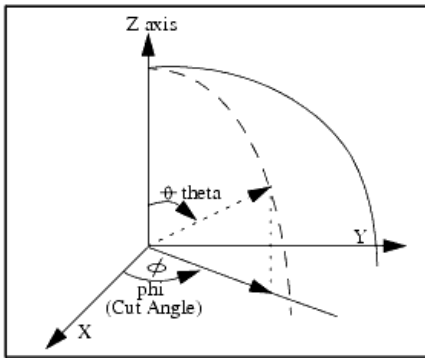
You may want to terminate all ports so that the ports that are not used to inject energy into the circuit do not cause reflections. For these ports, set the Port Excitation voltage to zero, and set the Port Impedance to the characteristic impedance of the port.

4. If you want the display tool to open and display results automatically, enable **Open display when computation completed**.
5. 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 workspace.

Planar (Vertical) Cut

For a planar cut, the angle phi (*Cut Angle*), 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.

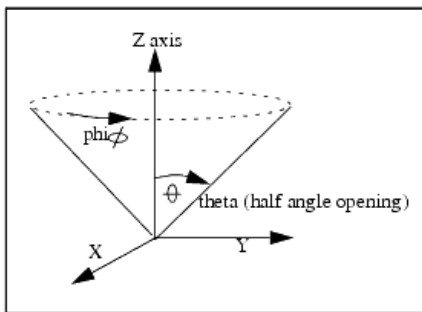


Note

In layout, there is a fixed coordinate system such that the monitor screen lies in the XYplane. The X-axis is horizontal, the Y-axis is vertical, and the Z-axis is normal to the screen. To choose which plane is probed for a radiation pattern, the *cut angle* must be specified. For example, if the circuit is rotated by 90 degrees, the cut angle must also be changed by 90 degrees if you wish to obtain the same radiation pattern from one orientation to the next.

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.



Viewing Far Fields

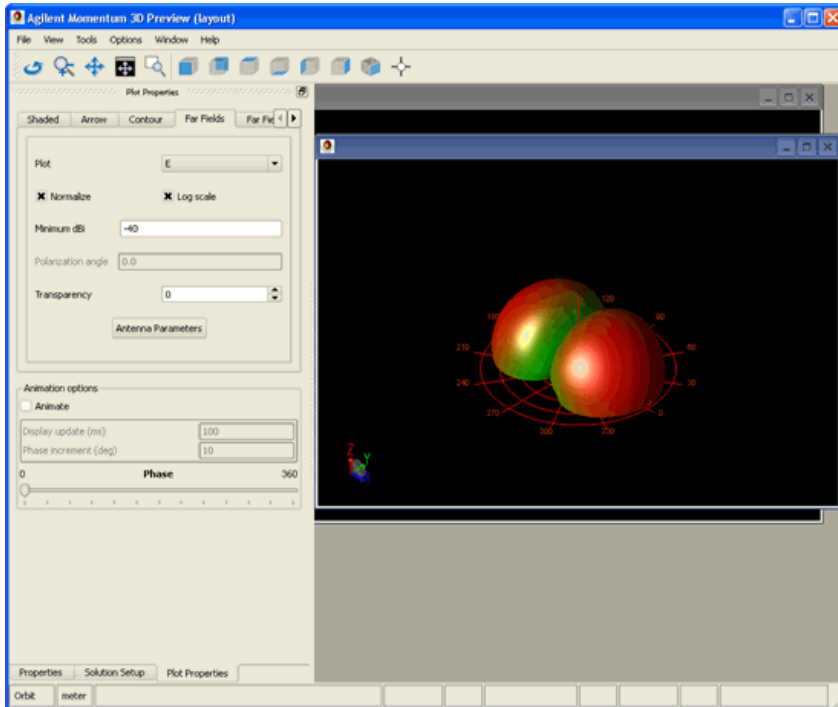
After the far fields computations are performed, data is saved depending on the display tool. The following types of displays are used:

- 2D Data Display: 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 workspace.
- 3D Visualization: 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 workspace.

Viewing Far Fields in 3D Visualization

When **3D Visualization** is selected as Visualization Type for viewing the far-field results, the 3D Visualization Tool will be automatically opened when the computation is finished. Two windows will be opened, a Geometry Window and a Far Field Window. The Far Field Window and associated properties tab is only shown if radiation results are available. The radiation results are for a single frequency, shown on the properties tab, and using the excitation state as specified in the Radiation Pattern Control dialog before computation. A new Visualization Window can be opened when far fields are calculated at another frequency.

Note
The **Solution Setup** and **Plot Properties** tabs have no association with the Far Field Window. They apply to the surface current visualization on the Geometry Window.



Select the **Plot** type:

- $E = \sqrt{\text{mag}(E \text{ Theta})^2 + \text{mag}(E \text{ Phi})^2}$
- $E \text{ Theta}$
- $E \text{ Phi}$
- $E \text{ Left}$
- $E \text{ Right}$
- $E \text{ Co}$
- $E \text{ Cross}$
- Circular Axial Ratio
- Linear Axial Ratio

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 $E \text{ Co}$, $E \text{ Cross}$, and Linear Axial Ratio. By default, a logarithmic scale is used to display the plot. If you want to use a linear scale, disable **Log scale**. Set the minimum magnitude that you want to display, in dB.

Click **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 Control 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, magnitude in Volt and phase in degrees in direction of maximum radiation
- E_phi, magnitude in Volt and phase in degrees in direction of maximum radiation
- E_x, magnitude in Volt and phase in degrees in direction of maximum radiation
- E_y, magnitude in Volt and phase in degrees in direction of maximum radiation
- E_z, magnitude in Volt and phase in degrees in direction of maximum radiation

Exporting Far Field Data

ASCII file with 3D 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 `<workspace_dir>/mom_dsn/<design_name>/proj.fff`. The data is saved in the following format:

```
#Frequency <f> GHz           // loop over frequencies
#Excitation # <i>           // loop over excitation states
#-----
#Begin cut                   // loop over phi
<theta_0> <phi_0> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)> // loop over
<theta> for <phi_0>
<theta_1> <phi_0> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
:
<theta_n> <phi_0> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
#End cut
#Begin cut
<theta_0> <phi_1> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)> // loop over
<theta> for <phi_1>
<theta_1> <phi_1> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
:
<theta_n> <phi_1> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
#End cut
:
:
#Begin cut
<theta_0> <phi_n> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)> // loop over
<theta> for <phi_n>
<theta_1> <phi_n> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
:
<theta_n> <phi_n> <real(E_theta)> <imag(E_theta)> <real(E_phi)> <imag(E_phi)>
#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:

$$\sqrt{\langle E_{\theta_{\max}} \rangle^2 + \langle E_{\phi_{\max}} \rangle^2}$$

ASCII file with Antenna Characteristics

The proj.ant file, stored in the same directory, contains the antenna characteristics. The data is saved in the following format:

```
Excitation <i>                // For Momentum, a single excitation state is specified in the
Radiation Pattern Control dialog, consequently <i> is always 1.
Frequency <f> GHz            // For Momentum, a single frequency is selected in the Radiation
Pattern Control dialog
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 <i> <prad> // in Watts
Effective angle <eff_angle_st> steradians <eff_angle_deg> degrees
Directivity <dir> dB // in dB (10*log10)
Gain <gain> dB // in dB (10*log10)
```

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.

Dataset file with 2D Cut Far Field Data

Specific information about the data available in the far-field dataset is given in this section.

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).

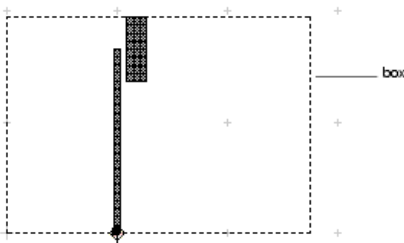
Note
 The normalized far-field components (Elhp, Erhp, Eco, and Ecross) are normalized with respect to $\max(\sqrt{|E_{\theta}(\theta, \varphi)|^2 + |E_{\varphi}(\theta, \varphi)|^2})$.

Adding Boxes, Waveguides, and Symmetry Planes

While specifying the substrate definition of a circuit, only the vertical dimension of the substrate and not the horizontal dimension are defined. This extends the substrate layers 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.

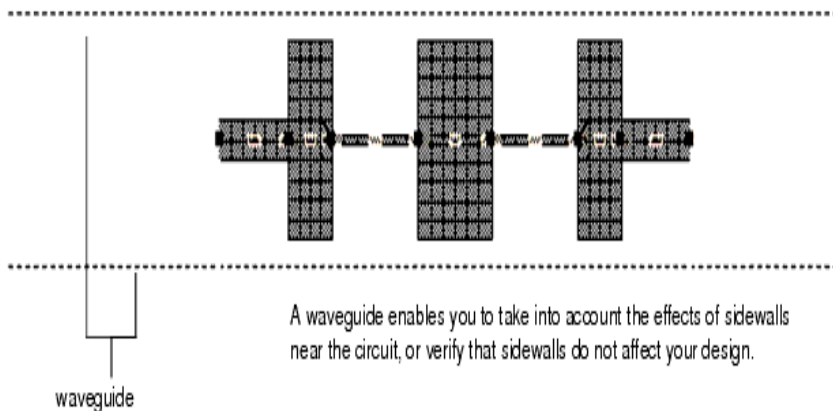
Boxes and waveguides enable you to specify substrate boundaries in the horizontal direction. A box enables you to set boundaries on four sides of the substrate. A waveguide enables you to set boundaries on two sides.

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).



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.



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

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 ground planes or impedance termination. The walls of the box are perfect metal. The ground planes can be defined either as perfect metals or a loose 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.

Note
Calibrated ports in the circuit (i.e., Single, Differential, Coplanar and Common mode ports) must be located on and perpendicular to the box edge, or an error will occur. Internal (uncalibrated) ports may be placed anywhere in the circuit and point in any direction.

To add a box:

1. Choose **EM > Box-Waveguide > Add Box**.
2. Position the mouse and click to define a corner of the box. Move the mouse to the diagonal corner and click once in the layout window.

You can also add a box by performing the following steps:

1. Choose **EM > Box-Waveguide > Add Box**.
2. From the Layout menu bar choose **Insert > Coordinate Entry** to open the Coordinate Entry dialog box.
3. Specify a value in the **Coordinate Entry X** and **Coordinate Entry Y** fields to specify a corner of the box.
4. Click **Apply**.

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, choose **EM > Box-Waveguide > Delete Box**.

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 *Schematic Capture and Layout* (usrguide).

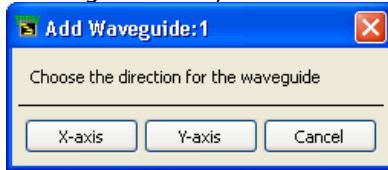
Adding a Waveguide

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 ground planes 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 **EM > Box-Waveguide > Add Waveguide**. This opens the Add Waveguide message window, as shown in the following figure:



2. Select the direction of the waveguide. To insert the waveguide parallel to the x-axis, click **X-axis**. To insert the waveguide parallel to the y-axis, click **Y-axis**.
3. 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 once in the layout window.

You can also add a waveguide by performing the following steps:

1. Choose **EM > Box-Waveguide > Add Waveguide**.
2. Select the direction of the waveguide. To insert the waveguide parallel to the x-axis, click **X-axis**. To insert the waveguide parallel to the y-axis, click **Y-axis**.
3. Choose **Insert > Coordinate Entry** in the layout.
4. Specify a value for the **Coordinate Entry X** and **Coordinate Entry Y** fields to specify a point on the edge of the substrate.
5. Click **Apply**.
6. Enter the coordinates of a point on the second, parallel edge of the substrate and click **Apply**.

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. Select **EM > Box-Waveguide > Delete Waveguide**.
2. Click **Yes** in the message window to remove the waveguide 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**.
Select **momentum_box** from the *Layers* list. The layer settings are displayed. For

2. more information on these parameters, refer to *Schematic Capture and Layout* (usrguide).

About Boxes and Waveguides

You may need to simulate a circuit in a box or waveguide in the following situations:

- The actual circuit is enclosed in a metal box: 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 if 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 a significant influence on the filter characteristics.
- The nearby metal sidewalls may affect circuit performance: 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.
- The box may resonate: A resonating box is a physical effect where, under the condition of certain frequency and box size combination, 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.
- The propagating modes may be present: 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 ϵ and μ (make sure that ϵ and μ 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.

Adding a Symmetry Plane

A symmetry plane defines the boundary on one side of the circuit substrate. Only one box, or one waveguide, or one symmetry plane can be applied to a circuit at a time. When a symmetry plane is defined, the simulation results will be equivalent to the results of a larger circuit that would be created by mirroring the circuit about the symmetry plane. For symmetric circuits, this enables faster simulations that require less memory because only half the actual structure needs to be simulated.

To add a symmetry plane:

1. Choose **EM > FEM Symmetry Plane > Add Symmetry Plane**.
2. Select the direction of the symmetry plane. To insert the symmetry plane parallel to the x-axis, click X-axis. To insert the symmetry plane parallel to the y-axis, click Y-axis.
3. Insert the symmetry plane using one of the following two methods:
 - Position the mouse and click to define the location of the symmetry plane.
 - From the Layout menu bar choose **Insert > Coordinate Entry** and use the Coordinate Entry X and Coordinate Entry Y fields to specify a point on the edge of the substrate.
4. Click **Apply**.

This boundary specifies the edge of the substrate where the plane of symmetry will be applied.

Editing a Symmetry Plane

Once the symmetry plane is applied, you cannot change its location. If you want to change the location or orientation, you must delete the current symmetry plane and add a new one.

Deleting a Symmetry Plane

To delete a symmetry plane:

1. Choose **EM > FEM Symmetry Plane > Delete Symmetry Plane**. The symmetry plane is removed from the layout.

Defining Component Parameters

You can define parameters for a component in a layout that enable you to sweep, tune, or optimize geometrical (shape) variations of planar layout objects. You can specify the following parameters for a component:

- Dimensions such as lengths, widths, gaps, spacing, or diameters
- Interdependent layout modifications such as length and width varying simultaneously
- Unconventional layout modifications such as varying port locations

Note
There is a limit of 100 layout parameters per component.

There are two ways to create a parameterized layout:

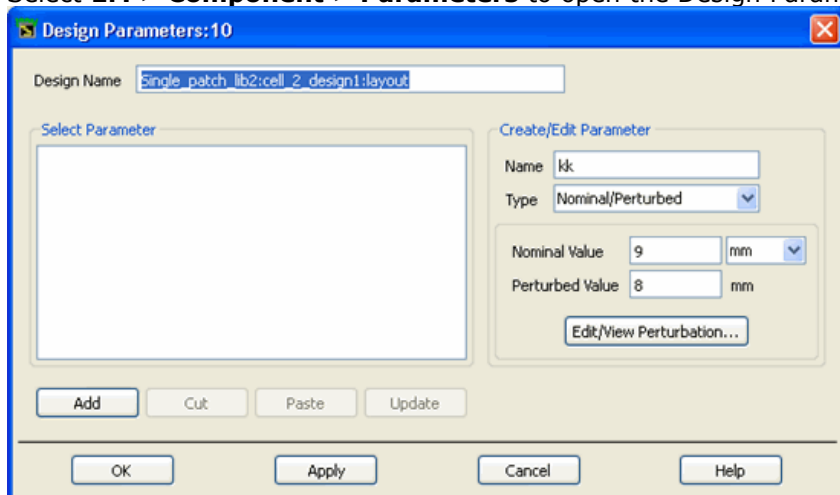
- Using nominal/perturbed designs
- Using subnetwork parameters

These approaches are mutually exclusive and you cannot combine them when defining parameters for a single component. Nominal/perturbed parameters can be defined when the layout is flattened and contains only ports and primitive artwork shapes (such as polygons, rectangles, circles, etc.). When a combination of existing (built-in or custom defined) layout components and any other type of artwork primitives are used, the layout is hierarchical and only subnetwork parameters can be used.

Using Nominal/Perturbed Designs

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 layout is called a *perturbed design*. The program will 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 by applying perturbations as specified in the Edit/View perturbation dialog. The modified copy is then saved as the perturbed design. You need to define a separate perturbed design for each parameter.

1. Select **EM > Component > Parameters** to open the Design Parameters dialog box.

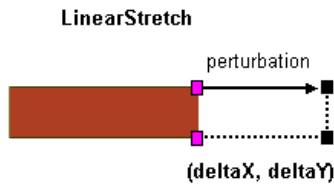
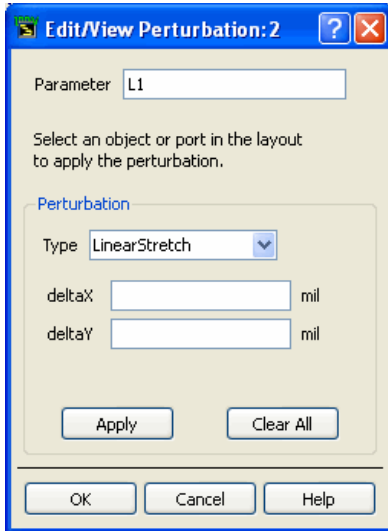


2. Type a parameter name in the Name text box.
3. Ensure **Nominal/Perturbed** is selected in the **Type** drop-down list.
4. Type a nominal value and select a unit from the drop-down list.
5. Type a perturbed value.
6. Click **Add** to add the parameter or **Edit/View Perturbation** to display the Edit/View Perturbation dialog box where you can specify a **Linear Stretch**, **Radial Stretch**, or **Rotation** perturbation for the specified parameter.

The following section describes these values.

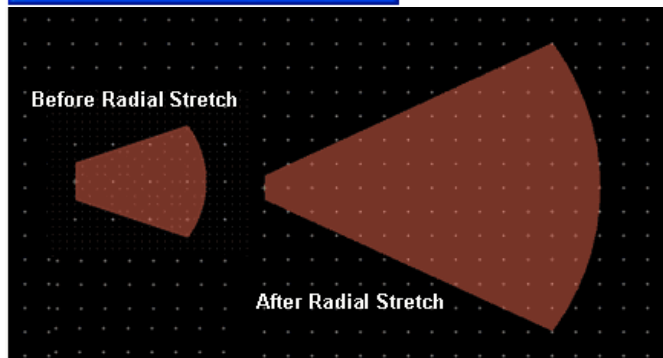
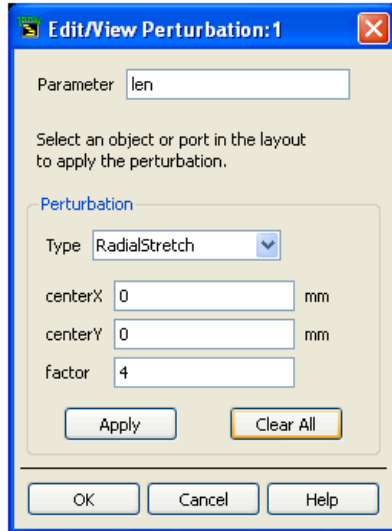
Linear Stretch

Specifying a linear stretch for a parameter enables you to move a single selected vertex point or a group of selected vertex points linearly using a deltaX, deltaY specification.



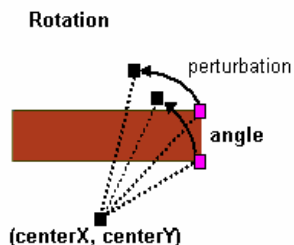
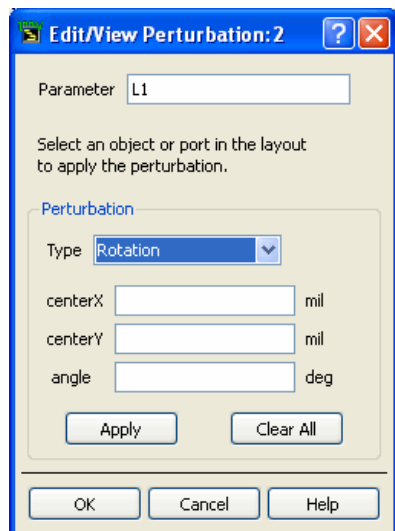
Radial Stretch

Specifying a radial stretch for a parameter enables you to move a single selected vertex point or a group of selected vertex points radially by specifying the centerX, centerY coordinates of the center point and a factor. The following illustration shows the effect of the factor on the selected vertex points.



Rotation

Specifying a rotation for a parameter enables you to move a single selected vertex point or a group of selected vertex points by rotating over a specified angle around a center point specified by the coordinates centerX, centerY.

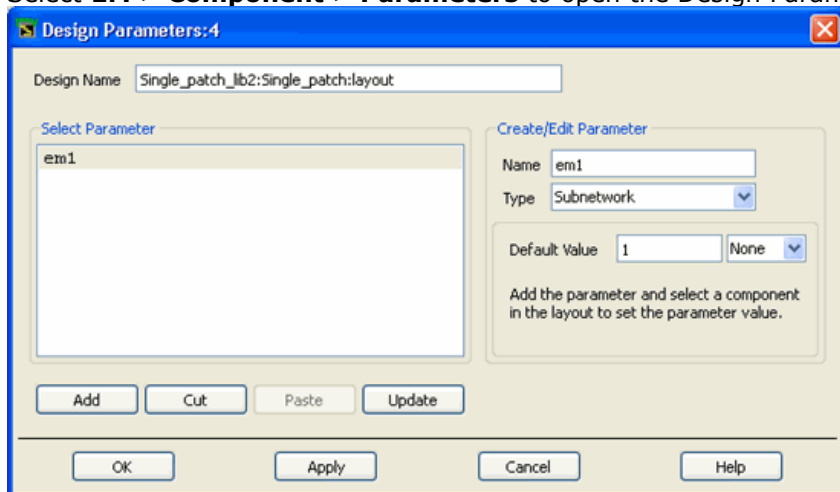


Using Subnetwork Design

A parameterized layout can be set up using a combination of several built-in components. You can define a subnetwork for hierarchical components only.

To specify subnetwork parameters for the layout component:

1. Select **EM > Component > Parameters** to open the Design Parameters dialog box.



2. Type a parameter name in the Name text box.
3. Select **Subnetwork** in the **Type** drop-down list.
4. Type a default value and select a unit from the drop-down list.
5. Click **Add** to add the parameter.

Once a subnetwork parameter is defined, it can be used to set the parameters values of one or more component instances used in the layout.

Caution
Only the top-level design parameters can be used to set the lower level instance parameters. The use of any other variable is prohibited.

Using an EM Model

An EM Model is used by a circuit simulator to model a component by using the S-parameters generated by an EM simulation. The EM-based S-parameters can be precomputed or you can generate them while running the circuit simulation. An EM Model caches the S-parameters to improve performance. EM Models are compatible with Dynamic Model Selection, which enables you to switch between two different models for the same cell. For example, a faster, schematic-based representation, and an accurate EM-based representation.

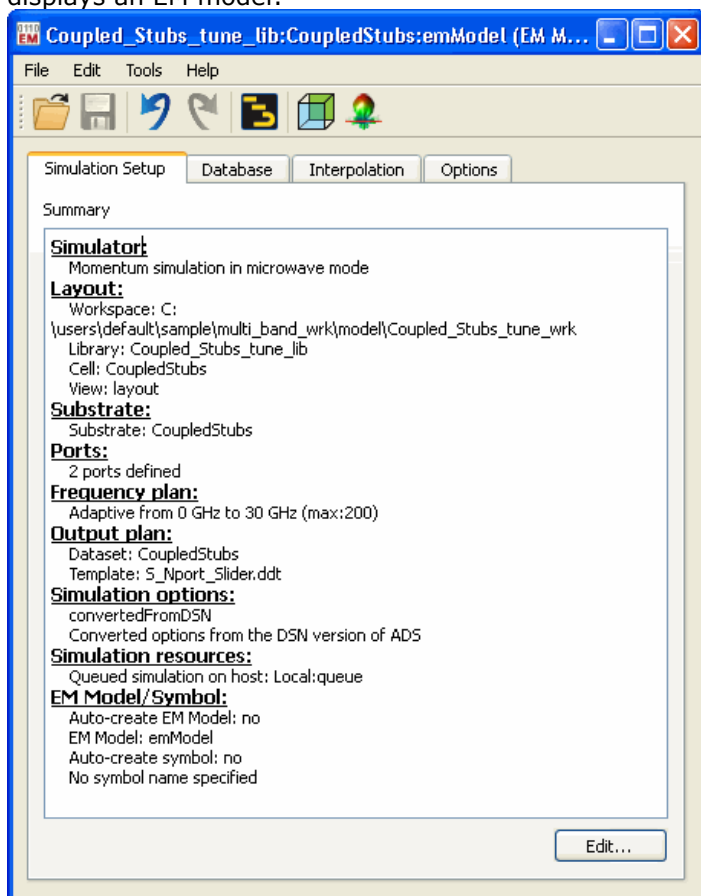
Note
In ADS 2009 Update 1 and earlier, a Layout Component is used instead of an EM Model. For more information, refer [Difference between Layout Components and EM Models](#).

Important when working with multiple libraries
EM Model generation while running the circuit simulation requires that the library of the testbench schematic references the technology from the library where the cell to be modeled by EM is defined. For example, a cell to be modeled by EM is defined in library1_lib, and this cell is instantiated in a schematic testbench defined in library2_lib. Then it is required that library2_lib references the technology definition of library1_lib. Otherwise, the EM model generation will fail because the substrate is not known in library2_lib.

Creating an EM Model

To create an EM model:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Model/Symbol** in the left pane of the EM Setup window.
3. Click **Create EM model when simulation is launched** to create an EM model after the simulation process.
4. To generate an EM model immediately, click **Create Now**. The following figure displays an EM model:



**Note**

If this button is labeled **Update Now**, the EM Model already exists for this cell.

The default name for an EM Model is *emModel*. It is recommended to use the default name. You can also create an EM Model by using the following methods:

- In the layout window, choose **EM > Component > Create EM Model and Symbol**.
- In the EM Setup window, select **Model/Symbol** in the left pane. If you select **Update EM Model when simulation is launched**, the EM Model will be created (or updated if it already exists) whenever you run an EM simulation.

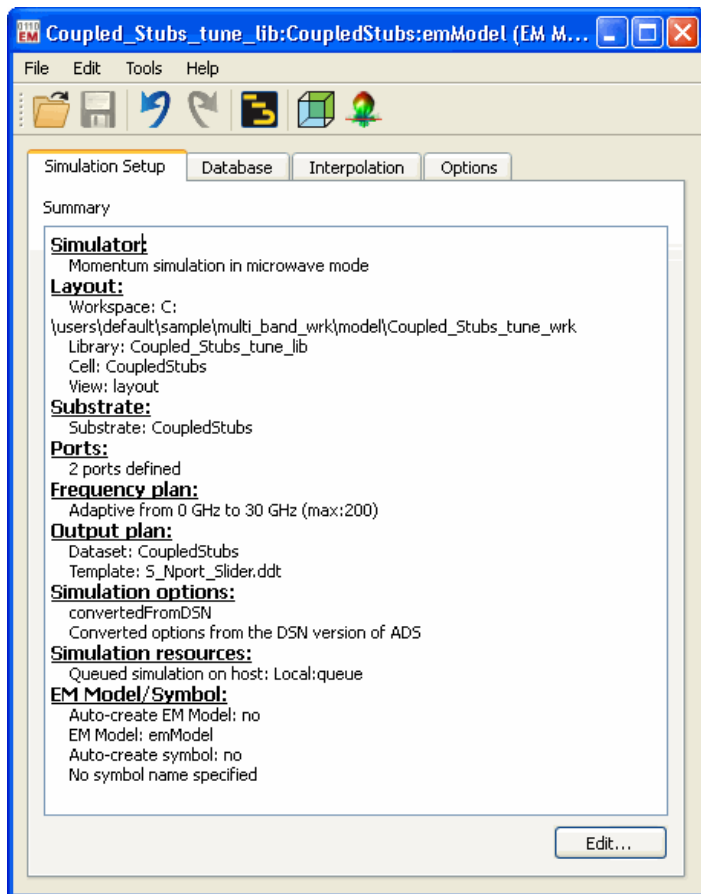
Using an EM Model Window

When an EM Model is created from an EM Setup window, all the settings specified in the EM setup window are copied to the EM Model. Later, during a circuit simulation, if the EM Model needs to run an EM simulation to generate on-demand S-parameters, this copy of the settings is used to launch the EM simulation.

The EM Model window consists of four tabs: Simulation Setup, Database, Interpolation, and Options. The following sections describe the tasks that you can perform using these tabs.

Viewing Simulation Setup Summary

You can view a summary of your EM simulation setup in the Simulation Setup tab. This tab provides information about the type of simulator, layout, substrate, ports defined for your simulation, frequency and output plans, simulation options and resources, and EM model and symbol information.

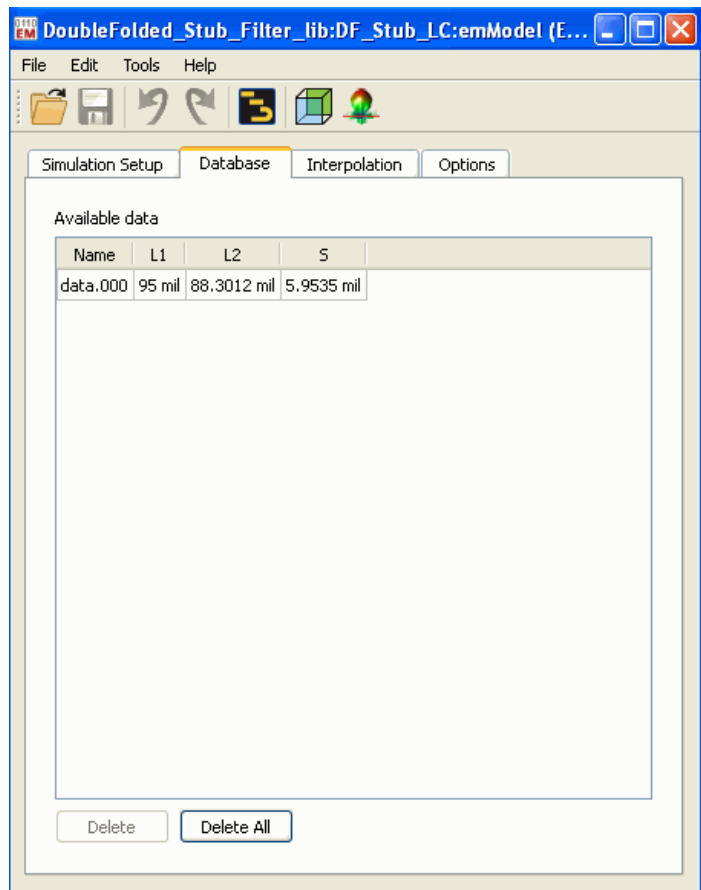


You can view the EM setup details and modify the setup by clicking **Edit**.

Note
When you change the value of a simulation option that affects the EM simulation results, the system deletes all the EM results cached in the EM Model. For more information, refer [Using the EM Model Database](#).

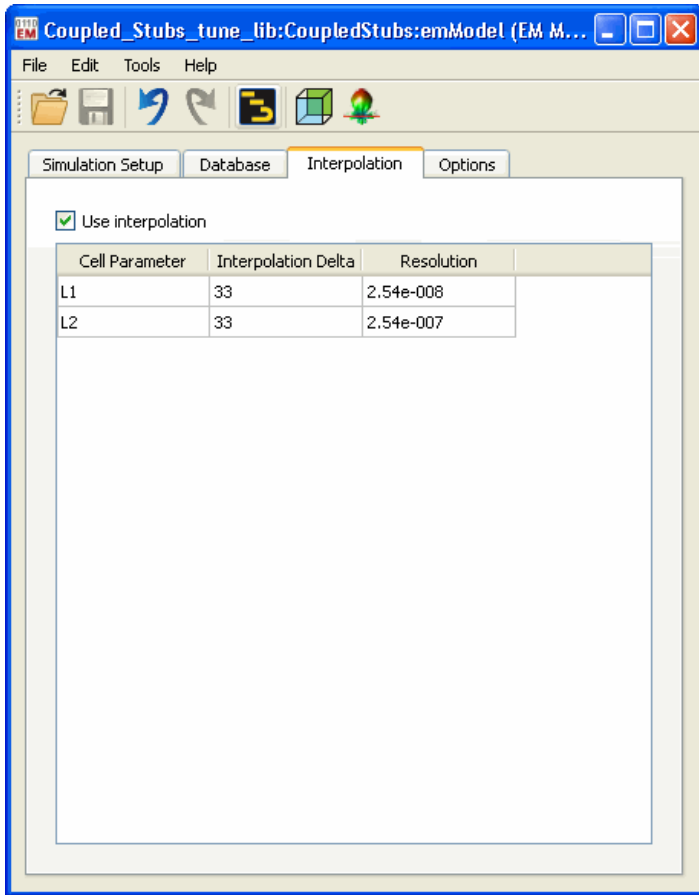
Viewing Database

You can view the available data by using the Database tab. This tab provides information about the database name, L1, L2, and S values. You can also remove databases from the list by clicking **Delete** or **Delete All**. The following figure displays the Database tab:



Specifying Interpolation Options

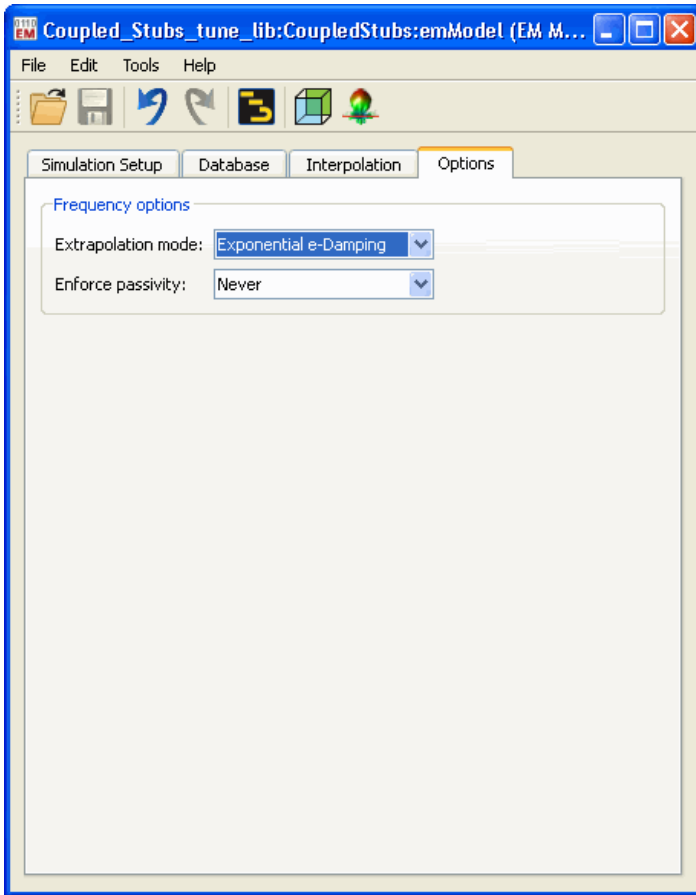
You can specify the interpolation parameters in the Interpolation tab. This tab provides information about the cell parameters, interpolation delta, and resolution values, as shown in the following figure:



Specify appropriate units. In the absence of a unit name, MKS units are assumed.

Specifying Frequency Options

You can specify the frequency options in the Options tab. You can select values for the extrapolation mode and enforce passivity, as shown in the following figure:



You can specify the following values for the extrapolation mode:

- Linear (default): A linear extrapolation scheme is used for both magnitude and phase. By default, passivity of the S-parameter model is enforced during extrapolation.
- Constant: The S-parameters at the lowest or highest available frequency are used.
- Exponential e-damping: A mixed extrapolation scheme is used. S-parameters are kept constant during extrapolation towards low frequencies. Towards high frequencies, S-parameters are e-damped in magnitude and linearly extrapolated versus phase.

The passivity of the S-parameter model can be enforced. You can enable or disable passivity enforcement through the Enforce Passivity parameter. You can specify the following values for enforce passivity:

- Only when extrapolating: S-parameter model passivity is enforced only during frequency domain extrapolation.
- Always: S-parameter model passivity is enforced at all frequencies.
- Never: S-parameter model passivity is not enforced.

Updating an EM Model

If you create an EM Model from an EM Setup and then change some settings in the EM Setup, the changes are not updated in the EM Model because it contains a copy of the EM setup. To copy the modified EM Setup to the EM Model:

1. Choose **EM > Simulation Setup** to open the EM Setup window.
2. Select **Model/Symbol** in the left pane of the EM Setup window.
3. Click **Update Now** in the EM Model panel.

Note
If this button is labeled as **Create Now**, then the EM Model does not exist for that cell.

When you update an EM Model and this causes the EM Model setup to change in a way

that affects the EM simulation results, the system deletes all the cached EM results. See The EM Model database for more information.

You can also update an EM Model by using the following methods:

- In the layout window, choose **EM > Component > Create EM Model and Symbol**.
- In the EM Setup window, on the **Model/Symbol** page, check **Update EM Model when simulation is launched**. With this checked, every time you run an EM simulation using this EM Setup, the EM Model will be updated (or created if it doesn't exist).

Using an EM Model for circuit simulation.

Using an EM Model for Circuit Simulation

Default View used by an EM Model

By default, the *Schematic* view is used by ADS circuit simulation before a view named *emModel*. Therefore, if you insert an instance of a cell that has an EM Model view named *emModel*, there are two possibilities:

- If the cell has no view named *schematic*, then the EM Model view named *emModel* is used for circuit simulation.
- If the cell has a view named *schematic*, the EM Model view is not used for circuit simulation. Instead, the view named *schematic* is used.

You can change the default behavior by using Dynamic Model Selection. For more information, refer *Dynamic Model Selection* (cktsim).

Default View used by a Circuit Simulation

To find the view that is used for a particular instance during circuit simulation, open the top-level schematic and choose **Simulate > Hierarchy Explorer**. In the **Hierarchy Explorer** dialog box, find the instance name in the **Instance** column. Then search in the **View for Simulation** column to find the view that will be used for circuit simulation. If the view for simulation is not *emModel*, you need to specify the system to use *emModel*.

Specifying an EM Model for Circuit Simulation

To specify the view to be used for a circuit simulation:

1. Select the instance on the schematic.
2. Click the **Choose View for Simulation** toolbar button, immediately to the left of the Simulate toolbar button. This dialog box lists all the views available for circuit-simulation for the cell associated with this instance.
3. Select the view named *emModel*.
4. Click **OK**. The circuit simulator will now use the view named *emModel* for this instance.

Note that the view name *emModel* is now displayed on the instance. This is a quick way to tell whether a specific view has been chosen for an instance. You can use the Hierarchy Explorer (choose **Simulate > Hierarchy Explorer** in the top-level schematic) to verify that this change has taken effect.

If you specify *emModel* in the **Choose View for Simulation** dialog box, but the Hierarchy Explorer shows that *emModel* is not the view that is used for circuit simulation, the top-level schematic is using a hierarchy policy that has the **Honor instance specializations** option selected.

Using the EM Model Database

To improve circuit simulation performance, each EM Model maintains a database of the EM simulation results. When a circuit simulator requests S-parameters from the EM Model, an EM simulation is performed only when the S-parameters are not available in the database. If an EM simulation is performed, the results are added to the database.

Any change that affects the EM simulation results causes an EM Model database to become out of date. Examples of this type of changes are:

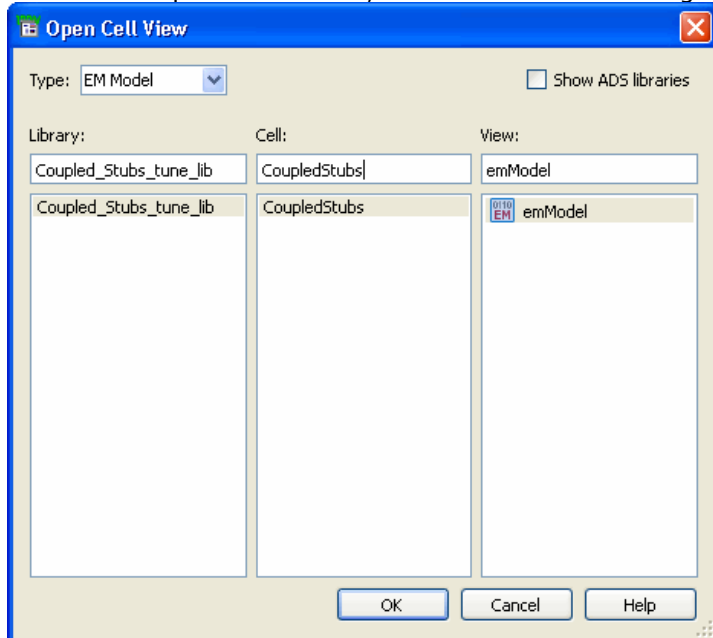
- EM Model simulation setup changes: For example, changing the mesh density.
- Changes to the cell parameter set: Adding a parameter to the cell or deleting a parameter from the cell.
- Substrate changes: For example, changing the thickness of a metal layer in the substrate.
- Layout changes: For example, adding shapes or components to the layout.

The EM model automatically detects changes to the EM Model and changes to the cell parameter set. When such a change is detected, the EM Model automatically deletes its database. However, other changes are not automatically detected. If you make such a change, you should open the EM Model, access the Database tab and click **Delete All**.

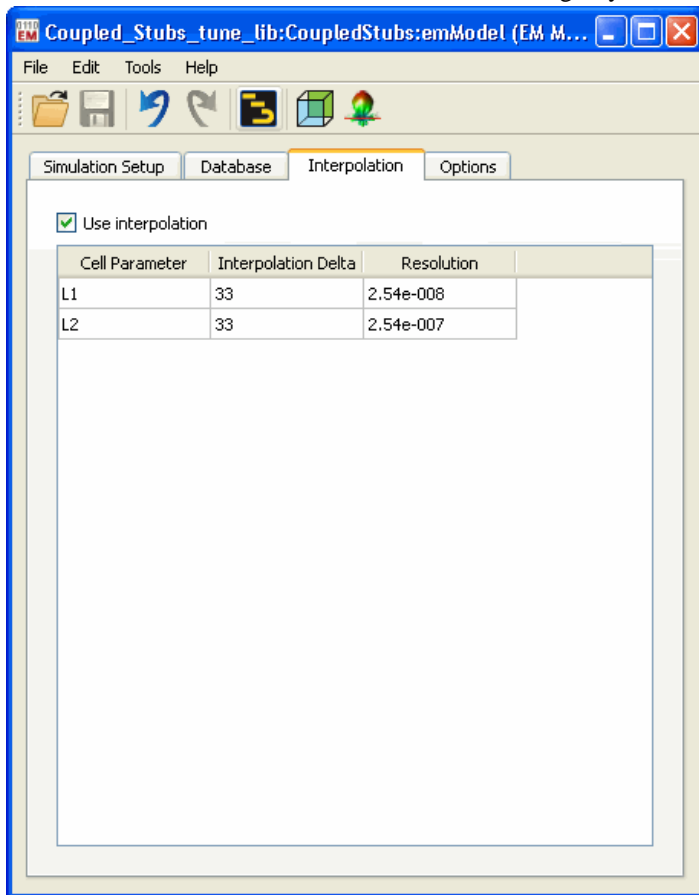
Specifying EM Model Options

To specify the EM Model options:

1. Choose **File > Open > EM Model**.
2. Select the required EM Model, as shown in the following figure:



3. Click **OK**. The selected EM Model is displayed.
4. Select the **Interpolation** tab in the EM Model window.
5. Select **Use Interpolation**.
6. Specify a value for the **Interpolation Delta** field for the required cell parameter.
7. Specify a value for the **Resolution** field.



8. Select the **Options** tab in the EM Model window.
9. Select a value from the **Extrapolation Mode** drop-down list.
10. Select a value from the **Enforce Passivity** drop-down list.

Difference between Layout Components and EM Models

In ADS 2009 Update 1 and earlier versions, you used a special type of component called a layout component. A layout component consists of the following features:

- Symbol: A symbol is a layout look-alike symbol.
- Model: In a circuit simulation, you can model a layout component by:
 - A schematic
 - EM-based S-parameters
 - File-based S-parameters

In ADS 2011, these features are still available, but due to the cell and view structure of libraries, they are separated into two distinct features:

- Symbol views: The system provides a command to generate a layout look-alike symbol.
- Dynamic Model Selection: You can specify which model to use for a circuit simulation:
 - A schematic view
 - A layout view
 - An EM Model view provides a model that uses EM-based S-parameters
 - File-based models can be implemented by creating a schematic view and inserting one or more Data components on it

EM Model Options

In ADS 2009 Update 1 and earlier, many EM simulation options (such as substrate name, frequency plan, and mesh density) are specified as parameters of each instance of the layout component. In ADS 2011.01, these parameters are no longer instance parameters. Instead, they are options of the EM Model. This leads to some differences in behavior.

To illustrate the differences, assume that you want to change the mesh density from 20 cells per wavelength to 30 cells per wavelength. First, assume that you want to make this change globally, that is, you want all instances of this component to use 30 cells per wavelength. In ADS 2009 Update 1 and earlier, you need to find every instance of the component, double-click it, and change the mesh density to 30. If you miss an instance, that instance will continue to use 20 cells per wavelength while all the other instances use 30. In ADS 2011.01, you simply go to the EM Model view and change the mesh density to 30. All instances that use this EM Model for circuit simulation will use the new value of 30.

As a second example, assume you want to change the mesh density from 20 to 30, but you want that change to apply only to instance A of the component. You want instance B to continue to use the old value of 20. In ADS 2009 Update 1 and earlier, you simply go to instance A, double-click it, and change the mesh density to 30. In ADS 2011.01, an EM Model has only one value for the mesh density. To use two different values, you need two separate EM Models. To do this, open the EM Model and choose **File > Save As** to create a second EM Model named *emModel2*. In this second version of the EM Model, which is a copy of the first, change the mesh density to 30. Then, back on the schematic, select instance A, click the *Choose View for Simulation* toolbar button, and set the view to *emModel2*.

Using Advanced Model Composer

You can use the Advanced Model Composer (AMC) feature to create multidimensional, parametrized, passive planar models for arbitrary-shaped, and user-defined parametrized layout components on custom substrates. You can add these models to a schematic or to a design kit. AMC combines the capabilities of the Model Composer tool with the generality of layout components. Also, Advanced Model Composer can now generate a scalable EM model based on FEM simulations.

Note
Advanced Model Composer and Momentum licenses are required to generate parametrized models for user-defined layout components. Once a model is generated, you do not need any license for using the model in a schematic environment.

You can also use the Model Composer feature to create multidimensional, parametrized, and passive planar models for standard interconnect components such as opens, stubs, bends, and tees on custom substrates. For more information, refer to the *Model Composer* (modcomp) documentation. Advanced Model Composer and Model Composer rely on a unique EM-based modeling method to provide EM accuracy at traditional circuit simulation speed. Models are generated in a model generation step and added to the model database associated with the layout component. Models generated by AMC can be recognized by their *.pml* extension. AMC provides the following features:

- AMC provides an option for populating the model database of a Layout component, up front, prior to using the Layout component in a schematic. Typically, the model database of a Layout component is updated with Momentum simulation data on-the-fly, while using the Layout component in a schematic.
- AMC uses an adaptive rational/polynomial curve fitting algorithm, which is a multidimensional version of Momentum's Adaptive Frequency Sampling (AFS) algorithm. The *.rat* (=RATional) files generated by Momentum AFS algorithm contain information about the rational fitting model of single Momentum simulations. There can be multiple *.rat* files in the model database for a single layout component.
- The AMC algorithms combine multiple *.rat* files into one global *.pml* (Passive Model Library) file. This file contains the multidimensional rational/polynomial model that is used to represent the S-data in the user-defined parameter/design space.

Note
Multiple *.rat* files that build up a *.pml* can be deleted. However, this is not necessary and may not be advisable as these *.rat* files can be re-used in a future AMC model generation.

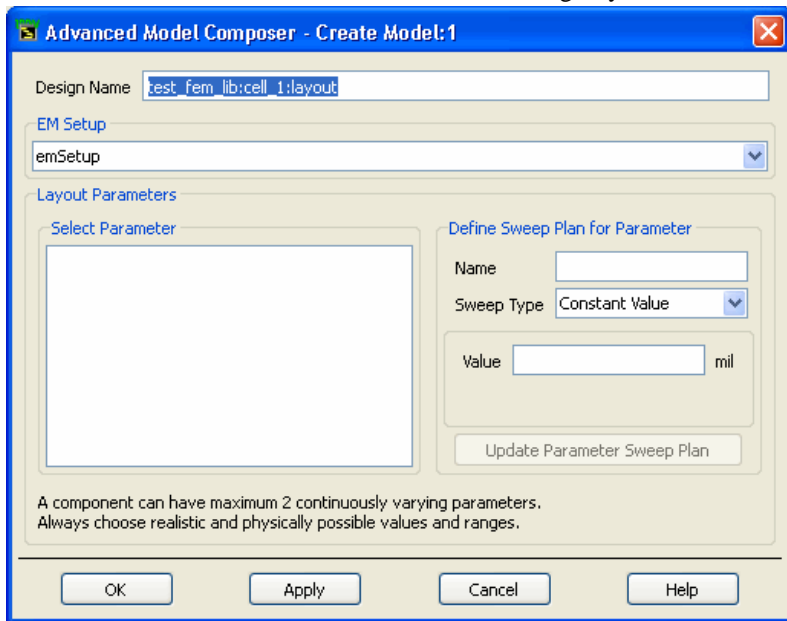
Creating AMC Models

The process of using an AMC model is as follows:

1. Create a model and monitor the model generation progress.
2. Use the model in the schematic environment.
3. Add the model to a standard ADS design kit (optional).

To access AMC:

1. From an ADS Layout window, choose **EM > Component > Advanced Model Composer > Create Model**. This opens the main **Advanced Model Composer** dialog box, as shown below:



2. Specify a sweep type for each layout parameter. You can choose one of the following options:
 - Constant Value: This value is used to specify that the layout parameter does not have to be swept. A specific sweep type must be specified for each layout parameter. The *Constant Value* option can be used for fixed values.
 - Discrete List: This value contains a set of discrete values. When you use the component on a schematic, you can set the parameter to any value in this list.
 - Continuous Range: This value is used to create a model covering a specific range with minimum and maximum values. When the component is used in a schematic, you can then select any value in this range. While there is no hard-coded limit to the number of continuous parameters, we recommend not using more than 2 continuous parameters. For more information, refer to [Troubleshooting](#).
3. Click **OK** to open an information window where you can view the *Parameter Sweep* plan and general information about the model generation process.
 - Click **OK** in this information window to start the model generation process.
 - A new (non-visual) ADS session will be launched in the background to build the model.

Note
You can continue to use this ADS session for other activity (depending on memory availability). Or, you can exit this ADS session without affecting the model generation process.

4. You can view the model generation process as described in the next section [Viewing and Controlling the Model Generation Progress](#).
5. Once model generation is complete, the parametrized model will automatically be stored in the model database associated with this Layout component. The associated Layout component with its parametrized electrical model can be added to a schematic like any other ADS component.

Optionally, the model can be added to a design kit as described in *Adding AMC Models to a Design Kit*.

Note
There is an important distinction between the use of Layout components and components generated with AMC.

When using Layout components directly in the schematic environment, the Momentum engine is called on-the-fly to generate an EM model as part of the circuit simulation (EM/circuit co-simulation).

When using AMC components, the parametrized EM model is generated prior to using the components in the schematic environment, therefore running Momentum is not required during the circuit simulation.

Viewing and Controlling the Model Generation Progress

The model generation progress can be viewed and controlled from any ADS session that is running on the same computer and started by the same user. Type in the *Component* name or click *Browse* to locate the component of interest. The default location is under the $\$HOME/hpeesof/amc/components$ directory, the name of the model is the same as the name of the layout component.

To view the model generation progress, choose **EM > Component > Advanced Model Composer > Status/Control** to open the **Advanced Model Composer Status/Control** dialog box. This dialog box:

- Describes the status of the model generation.
- Displays any error messages or warnings.
- Enables you to stop the model generation.

Troubleshooting

You may face problems while trying to generate an AMC model. This can be due to the following non-physical combination of:

- Substrate
- Maximum frequency
- Layout parameters

Due to these factors, the higher order modes are excited, the Momentum simulator produces non-physical results, and the AMC modeling process cannot converge.

In such cases, you can refer the following guidelines:

- Limit the frequency range.
- Limit parameter ranges (for example, limit width).
- Check units (for example, m instead of μm).
- Limit the number of continuously varying parameters. The maximum is two per component, for practical reasons. Consider that the modeler uses about 5 to 10 Momentum simulations for each continuously-varying parameter. This means:
 - One continuous parameter would require 5 to 10 simulations.
 - Two continuous parameters, 25 to 100 simulations.
 - Three continuous parameters would require 125 to 1000 simulations.
 - Four continuous parameters would require 625 to 10,000 simulations.
- Use constant values and discrete lists. Often, the line width of the metallization is limited and varies in a discrete way based on a given technology or application. If such cases, use discrete parameter lists that correspond to real world limitations.
- Replace a single model definition (with widely varying ranges) by multiple models of the same layout component, each valid in a subset of the original parameter space.
- Perform a worst case analysis of the library specifications (check the combination of max frequency, max width, and substrate) to make sure that no higher order modes occur. Momentum can be used to see if the simulation results (S-data) are accurate.

A typical workaround is to use discrete lists and limit ranges. With wider ranges and more continuously varying parameters, longer modeling times and more Momentum simulations will be required. (Carefully consider the parameter ranges and ensure that all values are physically possible.)

It should be noted that the AMC is designed for the purpose of component modeling, (i.e., components that are small compared to the wavelength). When modeling structures with resonances, the simulation time required to generate the model can increase significantly due to the fact that more Momentum simulations must be performed to get a converged model and that each of the Momentum simulations will require more frequency points.

Note

When the mesh density is set too low, mesh noise in the simulated models can influence the model generation process in such a way that the convergence of model generation is not reached within a reasonable number of samples. In this case, increase the mesh density.

It is recommended to simulate the component in the extremes of the parameter ranges prior to starting the tool; this will ensure that the Momentum model generation process for the selected simulation control options completes successfully.

AMC Model Generation Technique

AMC provides a method to build multidimensional parametrized analytical models for passive planar components. This produces highly accurate analytical models that can be used by all ADS circuit simulators. Model generation is based on EM simulation techniques, providing EM accuracy and generality at traditional circuit simulation speed.

The model generation technique, referred to as *Multidimensional Adaptive Parameter Sampling* (MAPS), selects a minimum number of EM simulations and builds a global analytical fitting model for the scattering parameters of general planar structures as a function of the geometrical parameters and of the frequency, with a predefined accuracy. Data points are selected efficiently and model complexity is automatically adapted. The algorithm consists of an adaptive modeling loop and an adaptive sample selection loop described in the following sections.

Adaptive Model Building Algorithm

The scattering parameters S are represented by a weighted sum of multidimensional orthonormal polynomials (multinomials) P_m . The multinomials only depend on the multidimensional coordinates \bar{x} in the parameter space R , while the weights C_m only depend on the frequency f :

(1-1)

$$S(f, \bar{x}) \approx M(f, \bar{x}) = \sum_{m=1}^M C_m(f) P_m(\bar{x})$$

The weights C_m are calculated by fitting the above equation on a set of D data points

$$\{x_d, S(f, x_d)\}$$

(with $d=1, \dots, D$). The number of multinomials in the sum is adaptively increased until the error function:

(1-2)

$$E(f, x) = |M(f, x) - S(f, x)|$$

is lower than a given threshold (which is function of the desired accuracy of the model) in all the data points. For numerical stability and efficiency reasons orthonormal multinomials are used, that is, the multinomials $P_m(x)$ meet the condition:

(1-3)

$$\sum_{d=1}^D P_k(\bar{x}_d) P_l(\bar{x}_d) = \begin{cases} 1 & \text{for } (k = l) \\ 0 & \text{for } (k \neq l) \end{cases}$$

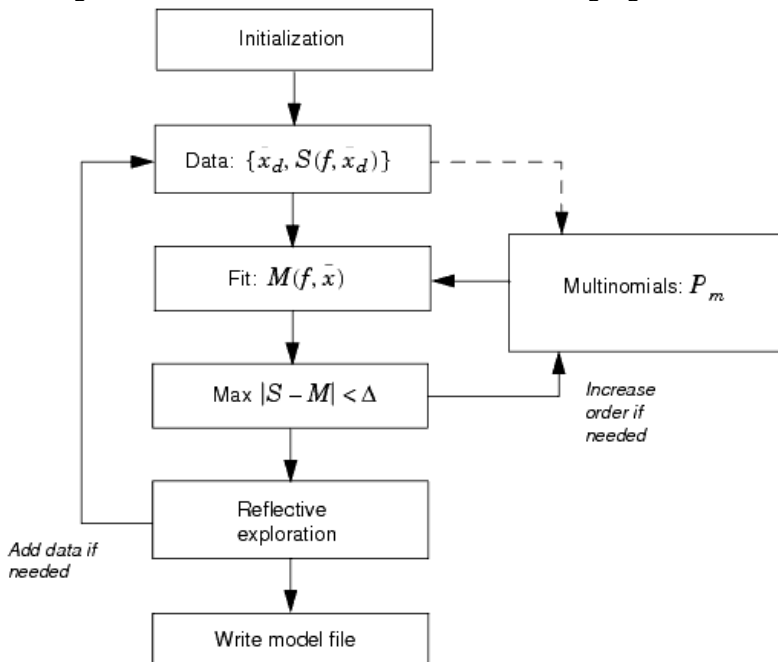
Adaptive Data Selecting Algorithm

The modeling process starts with an initial set of data points. New data points are selected adaptively in such a way that a predefined accuracy Δ for the models is guaranteed. The process of selecting data points and building models in an adaptive way is often called *reflective exploration*. Reflective exploration is useful when the process that provides the data is very costly, which is the case for full-wave electromagnetic (EM) simulators. Reflective exploration requires *reflective functions* that are used to select a new data point. The reflective function used in the MAPS algorithm is the difference between two different models (different order M in equation (1-1)). A new data point is selected near the maximum of the reflective function. When the magnitude of the reflective function becomes smaller than Δ over the whole parameter space, no new data point is selected.

If one of the scattering parameters has a local minimum or maximum in the parameter space of interest, it is important to have at least one data point in the close vicinity of this extremum in order to get an accurate approximation. Therefore, if there is no data point close to a local maximum or minimum of $M(f, \bar{x})$, the local extremum is selected as a new data point. For resonant structures, the power loss has local maxima at the resonance frequencies. Again, to get an accurate approximation, a good knowledge of these local maxima is very important.

The scattering parameters of a linear, time-invariant, passive circuit satisfy certain physical conditions. If the model fails these physical conditions, it cannot accurately model the scattering parameters. The physical conditions act as additional reflective functions: if they are not satisfied, a new data point is chosen where the criteria are violated the most.

The algorithm flowchart is shown in the following figure:



Using the Job Manager

You can manage non-interactive processing jobs by using the Job Manager window. A new managed job is automatically created for various time-consuming operations. These operations currently include physical EM simulations and 3D model generations. The Job Manager provides a user interface to show the job status and messages, and allows you to cancel running jobs. However, its scope is currently limited to physical EM simulation related jobs.

The Job Manager consists of an internal queuing concept. According to this concept, jobs submitted with the same queue name run sequentially. Queues can be suspended and resumed. Note that this internal queuing system is complimentary to the queuing system of distributed job management systems such as Platform LSF and Sun Grid Engine. For circuit simulation job management, refer *Circuit Remote Simulation* (cktsim).

Besides running locally, physical EM simulation setups can be configured to run on a remote host or a submission host for a distributed cluster. The Job Manager maintains a [PVM](#) cluster of hosts ready to accept jobs, and dynamically adds hosts to the cluster as needed. Complete cluster host configuration and management capabilities are provided inside the Job Manager user interface.

Starting the Job Manager

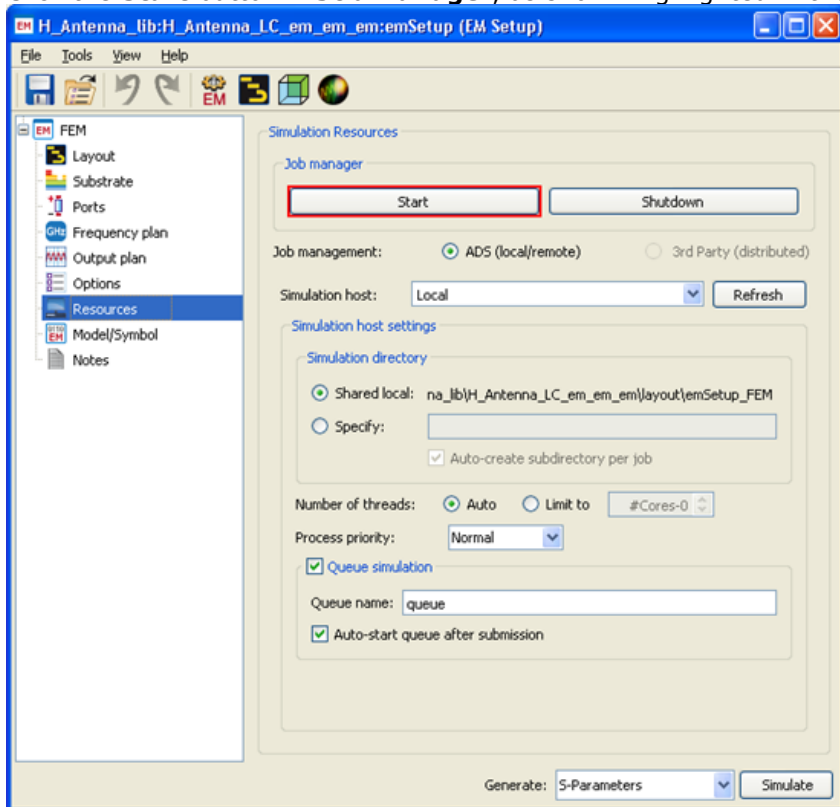
A Job Manager window is automatically displayed in the background when an EM Setup is opened or a physical EM simulation is started. It can be brought to the front by clicking



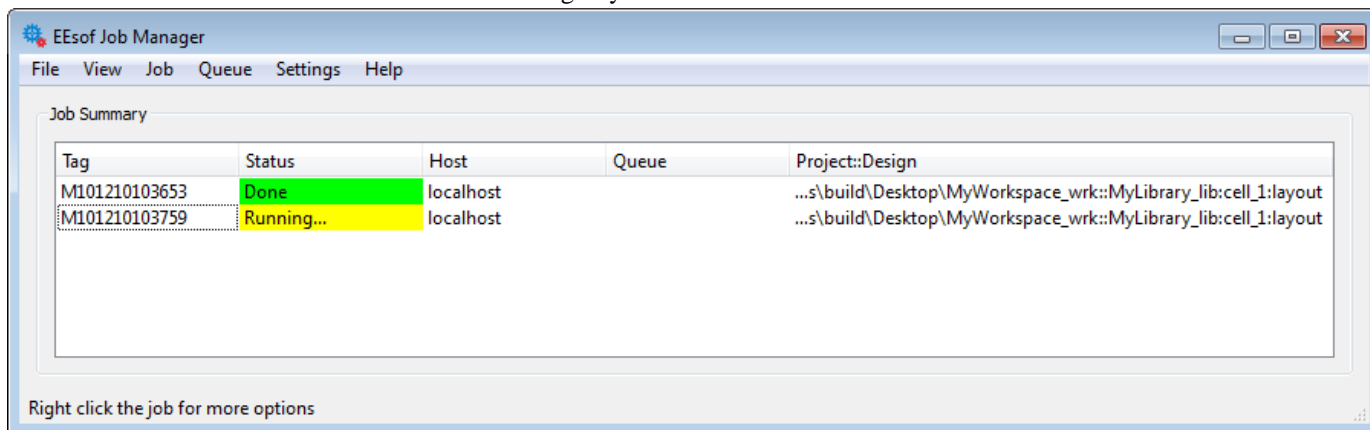
icon on the task bar.

Alternatively, you can display the Job Manager window by using the EM Setup window. Perform the following steps:

1. Select **EM > Simulation Setup** to open the **EM Setup** window.
2. Select **Resources** in the left pane of the EM Setup window.
3. Click the **Start** button in **Job Manager**, as shown highlighted in the following figure:



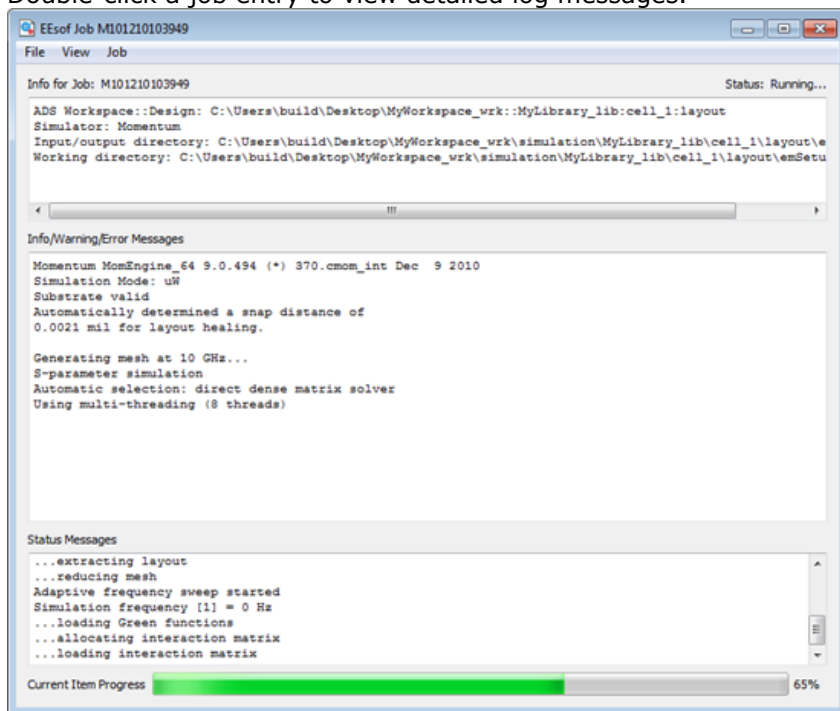
4. The Job Manager main window is displayed, which lists a summary of all jobs created since the Job Manager was launched:



Warning

The Job Manager window closes automatically if the main window is closed. If active jobs are still pending, a message box is displayed asking for confirmation. After confirmation, these jobs will be canceled without cleaning up their working directories. All status and logging information is discarded at exit. If you just want to hide the main window, minimize it instead of closing it.

5. Double-click a job entry to view detailed log messages.



Note

This log window is immediately displayed when newly created jobs do not need to wait in the internal queuing system. Select **View > Show Main Window** to open the main window from here. Newly created jobs that have to wait in a Job Manager queue will open the main window and highlight their summary line.

Job Progression

All jobs progress through the following states:

1. **Creating a new job:** A new job is created and the following job configuration information is displayed in the topmost part of the detailed log window.
 - **ADS Workspace::Design:** Specifies the workspace and design being simulated.
 - **Simulator:** Specifies the simulator **Momentum** or **FEM** used for simulation.
 - **Input/output directory:** Specifies the directory used for containing input files and output files.
 - **Working directory:** Specifies the host directory where the job will perform the main task. This can be the same directory as the input/output directory.

2. **Queuing... or Queuing (queue suspended):** After a job is created, it needs to wait until earlier jobs in the same Job Manager queue are complete.
3. **Connecting with the remote host:** The remote host is not yet part of the PVM cluster and is currently being dynamically added to the cluster.
4. **Preparing Files:** This preparation step consists of the following stages:
 - The input/output directory and working directory are locked to prevent double usage. The lockfile is an empty directory named *.simlock*.
 - Local preparation work is performed in the input/output directory.
 - The contents of the input/output directory are copied to the working directory.
5. **Performing the main task:** The job is inside its main task. In the **Local/remote** mode, the status displays **Running**. For the **Distributed** mode, the following third party cluster submission and status tracking is performed:
 - **Submitting to cluster**
 - **Cluster response: submitted.**
In response to choosing **Job > Query Status:**
 - **Requesting status update**
 - **Cluster response: created/submitted/pending/running/suspended/ended or Cluster did not answer**
6. **Waiting:** This step allows dying processes and network disk operations to settle.
7. **Retrieving results:** The contents of the working directory are copied back to the input/output directory.
8. **Processing results:** The final datasets are installed in this step.
9. **Cleaning up:** The working directory is cleaned up (only if this is not also the input/output directory). The *.simlock* lockfile is removed from the input/output directory.
10. **Done**

Managing Jobs

You can control the following operations using the **Job** menu, which is available from the Main window and log window:

- **Stopping or Canceling a Job:** You can immediately cancel a job and perform the necessary cleanup operations by selecting **Job > Stop/Cancel**.
- **Querying Status:** You can query a status update for jobs submitted to a third party distributed cluster system by selecting **Job > Query Status**. The updated status will be shown in the status line. If the status indicates that the simulation is done, all results will be automatically fetched, available datasets are installed, and the remote working directory is cleaned up.
- **Retrieving Intermediate Results:** You can synchronize the remote working directory to the input/output directory and install currently available datasets by selecting **Job > Fetch Intermediate Results**. This option is applicable only to jobs submitted to a third party distributed cluster system.



Note

It is not necessary to use **Job > Fetch Intermediate Results** at the end of a distributed simulation. Use **Job > Query Status** instead, which additionally cleans up the remote working directory and notifies other tools of the completion of the simulation.

- Deleting logs: You can completely remove a job from the Job Manager by selecting **Job > Delete Log** in the main window.



Warning

When using **Job > Delete Log** on a running job, a message box is displayed asking for confirmation. After confirmation, this job will be canceled without cleaning up its working directory. To avoid this, use **Job > Stop/Cancel Job** and delete the job only after the status has changed to **Done**.

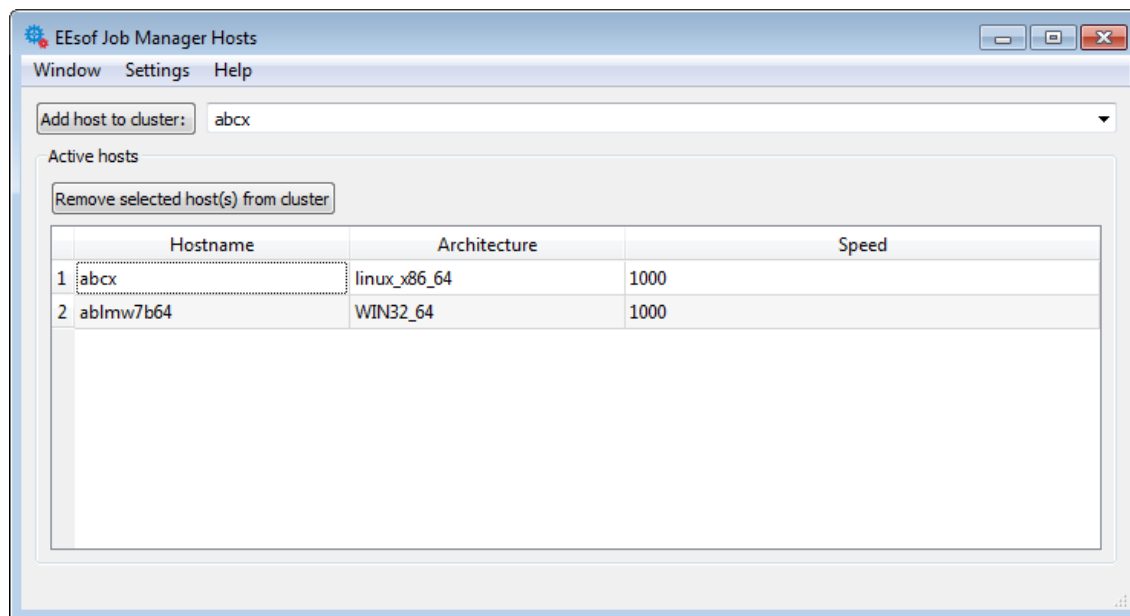
The Job Manager provides a simple queuing system where jobs submitted with the same queue name run sequentially. The queue name is *specified in the EM Setup (em)*. For complex queue management, use a dedicated job management tool such as Platform LSF and Sun Grid Engine.

Job manager queues can be suspended and resumed by selecting a queued job in the main menu and choosing **Queue > Suspend Queue** or **Queue > Resume Queue**.

Note
 It is not possible to suspend a queued job once the job has started. Still, **Queue > Suspend Queue** can be used to suspend execution of subsequent queued jobs.
 After a queue completes, it will automatically enter a suspended state. Use **Resources > Simulation Resources > Auto-start queue after submission** in the EM Setup if you want to automatically (re)start the queue when submitting a new simulation.

Managing Hosts

Select **View > Hosts** from the main Job Manager window to display the status of all hosts in the PVM cluster. This window will also automatically pop up whenever hosts are dynamically being added to the cluster.



One line is displayed for each running host in the cluster. The architecture and speed of each host are shown as well. The speed can be configured using the custom `sp=value` `eesofpvm` option and is currently used for display purposes only. Refer to *Setting up remote hosts* for host configuration information.

Select or type a host name and click **Add host to cluster** to manually add a new host to the cluster. This will bring up a password prompt if authentication is required. If the operation fails, information about the failure is shown in the status bar. To remove hosts from the cluster, select one or more entries in the **Active hosts** table and click **Remove selected host(s) from cluster**. Attempts to remove the local host from the cluster are ignored.

Setting Preferences

The Job Manager is an improved version of the former EEsosPVM Simulation Console. Complete cluster host configuration and management capabilities are now provided inside the Job Manager user interface. The Legacy setups using a cluster configuration file and environment variables are still supported but deprecated. This support may be removed in a future release.

- The Job Manager now automatically uses secure shell (SSH) to connect to other hosts. If it still needs to run some other third party remote shell tool for a legacy setup, the environment variable `EESOFPVM_RSH_NOAUTO=<any nonempty string>` is now additionally required besides the existing `EESOFPVM_RSH` variable.
- When migrating to the new GUI based configuration, it is advised to remove all old

eesofpvm configuration settings:

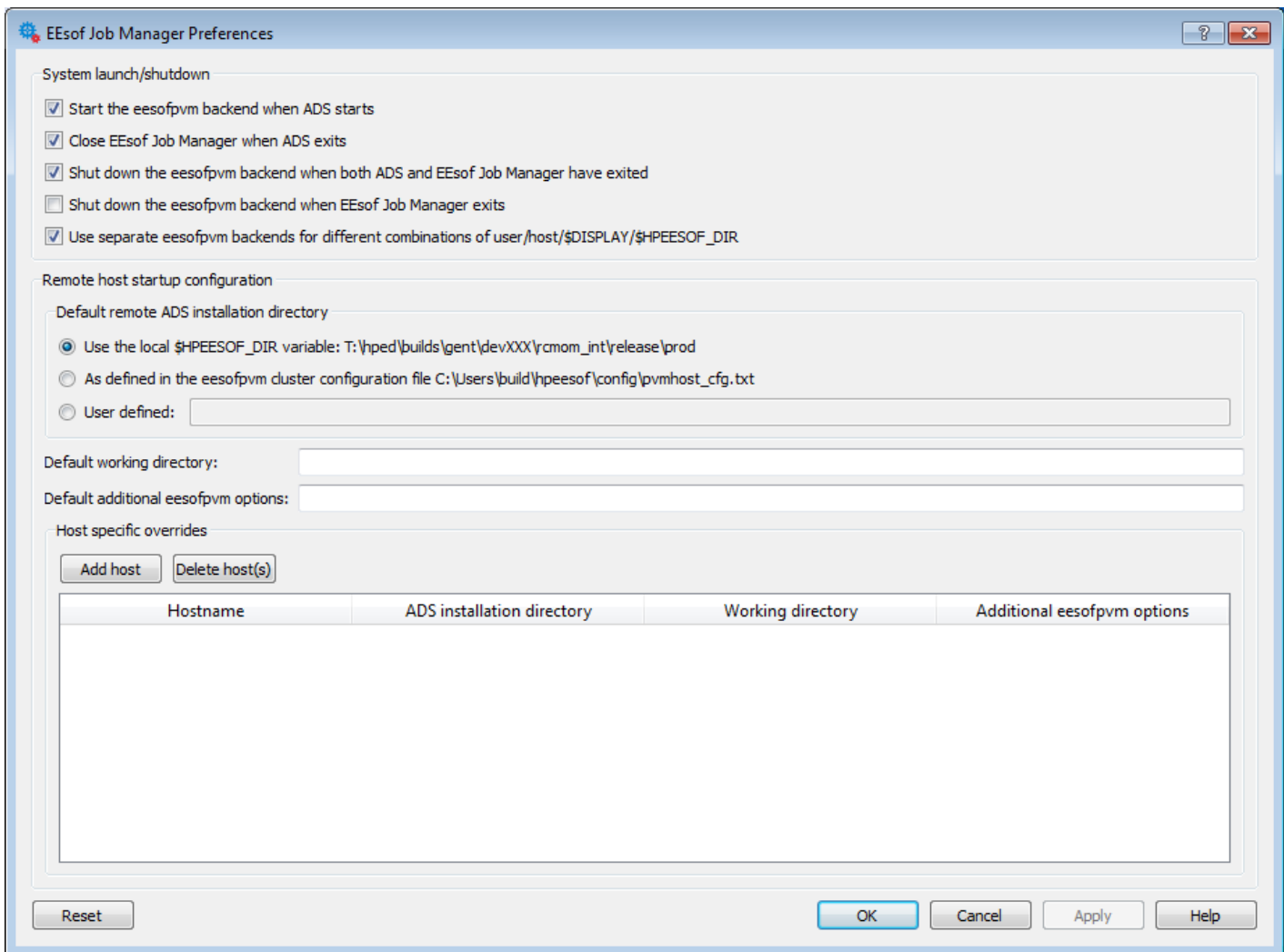
- If coexistence with ADS 2009 (Update 1) is not necessary, remove the cluster configuration file at *<home directory>/hpeesof/config/pvmhost_cfg.txt* and/or the path specified by the *<home directory>/hpeesof/config/hpeesof.cfg* variable `EESOFPVHOSTFILE` (and remove this variable).

However, if ADS 2009 (Update 1) coexistence is required, the legacy eesofpvm host file can still be obscured to the Job Manager by overriding the `hpeesof.cfg` variable `EESOFPVHOSTFILE` with an additional variable

`EESOFPVHOSTFILE:370=/nonexistent`. The number 370 is the internal Job Manager version number as shown when choosing **Help > About EEs of Job Manager** from the main window.

- If coexistence with ADS 2009 (Update 1) is not necessary, eliminate all environment variables containing `EESOFPVM` in the name from the Windows environment or Unix init scripts (e.g. `~/bashrc` or `~/kshrc`) and the `~/eesofpvmprofile` file. The only environment variable that does not lose its relevance is `EESOFPVM_TMP`, the directory where the PVM daemon displays its configuration and writes logging information.

Choose **Settings > Preferences** from the Main window or the hosts window to open the Preferences dialog box.



Specifying Startup/Shutdown Configuration

The Job Manager runs as a task of the PVM cluster it controls. The PVM infrastructure needs to start before the Job Manager starts, and can be allowed to continue running after the Job Manager has exited. To accommodate different requirements, several

startup/shutdown options are available. Refer to the following recommended settings:

- **Start the eesofpvm backend when ADS starts:** When selected, the PVM infrastructure will silently start when ADS is started. This reduces the response time to start the first simulation (at much lower cost). It is selected by default.
- **Close EEs of Job Manager when ADS exits:** When selected, the Job Manager will exit when ADS exits. A confirmation message box will appear when there are unfinished jobs. It is selected by default.
- **Shut down the eesofpvm backend when both ADS and EEs of Job Manager have exited:** When selected, the PVM infrastructure will be stopped when both ADS and EEs of Job Manager have exited. It is selected by default.
- **Shut down the eesofpvm backend when EEs of Job Manager exits:** When selected, the PVM infrastructure will be stopped when the EEs of Job Manager exits. This check box can be partially selected, which displays a message box asking for your confirmation. It is not selected by default.
- **Use separate eesofpvm backends for different combination of user/host/\$DISPLAY/\$HPEESOF_DIR:** When selected, all internal files that are shared between different executables of the same PVM session will get a suffix to separate different PVM sessions from one another. This prohibits conflicts between different PVM sessions running on the same system. The suffix consists of eight lowercase letters and/or digits. It is selected by default.

If the presence of a legacy PVM cluster configuration is detected (if `EESOF_PVM_HOSTFILE` is defined or `$HOME/hpeesof/config/pvmhost_cfg.txt` exists), the default settings will be different in order to mimic the original behavior:

Preference name	Normal default (recommended)	Legacy default
Start the eesofpvm backend when ADS starts	Selected	Not selected
Close EEs of Job Manager when ADS exits	Selected	Not selected
Shut down the eesofpvm backend when both ADS and EEs of Job Manager have exited	Selected	Not selected
Shut down the eesofpvm backend when EEs of Job Manager exits	Not selected	partially selected = prompt
Use separate eesofpvm backends for different combinations of user/host/\$DISPLAY/\$HPEESOF_DIR	Selected	Not selected
Default remote ADS installation directory	Use the local \$HPEESOF_DIR variable	As defined in the eesofpvm cluster configuration file

Setting up Remote Hosts

The *EM Setup* (em) window allows you to configure jobs to run on a remote host or a distributed cluster.

Note
 The configuration and operation of a third party distributed cluster happens completely outside the Job Manager. The Job Manager only communicates with a *submit host* having access to the distributed cluster. The requirements and configuration of a submit host are identical to those of any other remote host as far as the Job Manager is concerned.

Requirements for Running Jobs

In order to run jobs on a remote host you need the following:

- A compatible ADS installation on the remote host. Compatibility is only guaranteed if the versions of the local and remote ADS installations are identical.
- An account on the remote host accessible over secure shell (SSH). A SSH service is available on all contemporary Unix-like systems. Setting up SSH on Windows is described [below](#). *Passwordless SSH operation* is supported but not required.
- A read/write file system location on the remote host that can temporarily support the working directory of a job.

Configuring a Remote Host

To configure a remote host, add a configuration line to the *Host specific overrides*:

Host specific overrides				
<input type="button" value="Add host"/> <input type="button" value="Delete host(s)"/>				
	Hostname	ADS installation directory	Working directory	Additional eesofpvm options
1	linuxbox.company.com	/opt/ADS2011_01	/var/tmp/\$USER/jobs/\${SESSION:local}/\${TAG:local}:force	lo=unixuser
2	windowsbox.company.com	C:\ADS2011_01	%USERPROFILE%\Jobs\%SESSION:local%\%TAG:local%:force	lo=winuser

Specify the following fields:

- **Hostname:** This is the DNS name of the remote host or its dotted IP address. If you want to limit the configuration to a specific ADS version, add `:version` and optionally additionally `.revision` after the host name. The current values for `version` and `revision` are shown on the first line of the dialog shown when choosing **Help > About EEs of Job Manager** from the main window: `EEsof Job Manager (version.revision)`.
- **ADS installation directory:** This is the path to the main ADS installation directory.
- **Working directory:** The remote working directory is generally *specified in the EM Setup (em)*. The working directory entered in the Job Manager preferences is used as base directory to complete relative path specifications. However, if you append `:force` to the end of the path, the directory specified in the EM Setup is always ignored and the path preference is used exactly as specified. Environment variables specified as `$VAR`, `${VAR}` or `%VAR%` are expanded on the remote host. Use `${VAR:local}` or `%VAR:local%` to expand environment variables on the local host. Escape a literal `$` as `$$` and a literal `%` as `%%`. The following local variables have a special meaning:
 - `${HOSTNAME:local}`: the local hostname
 - `${PWD:local}`: the input/output directory
 - `${SESSION:local}`: an identifier unique to each PVM cluster instance (a string of eight lowercase letters and digits if *Use separate eesofpvm backends for different combinations of user/host/\$DISPLAY/\$HPEESOF_DIR* is selected, otherwise default)
 - `${TAG:local}`: a `M012345678901` identifier for the job that is unique within the Job Manager session
 In case you want to avoid all directory conflicts, a good choice would be
 - `%USERPROFILE%\Jobs\%SESSION:local%\%TAG:local%:force` for remote Windows hosts and


```

          /var/tmp/$USER/jobs/${SESSION:local}/${TAG:local}:force
          
```

 for remote Linux hosts.
- **Additional eesofpvm options:** Enter a list of space separated `name=value` options for PVM.
 - `lo=username` specifies an alternate login name to use.
 - `dx=<path to eesofpvm>` specifies an alternate path to the eesofpvm executable or script. For further options, consult the [PVM documentation](#).

If the entries for *ADS installation directory* or *Working directory* are empty, the specified defaults are used. These defaults can be used to avoid redundant configuration repetitions. Each *name* specified in *Default additional eesofpvm options* is only overridden

if the same *name* is specified in the *Host specific overrides*.

Mode of Operation

The Job Manager window dynamically adds remote hosts to the PVM cluster as required or requested. A host is added as follows:

1. A secure shell (SSH) connection is initiated to the remote host.
2. The local host prompts for passwords.
3. The SSH connection is now established.
4. (Only on Unix remote hosts) The Korn shell script file `~/.eesofpvmprofile` is sourced if it exists. This file can be used to set additional environment variables. A typical use case is extending the `PATH` and `LD_LIBRARY_PATH` environment variables to ensure accessibility of the tools required to control a third party job management framework.
5. The PVM daemon is started on the remote host. The choice between 32-bit and 64-bit executables is automatic (64-bit executables are used whenever possible). The command used to start the PVM daemon is `<ADS installation directory>/pvm3/bin/auto_eesofpvmd.exe`.
6. The remote PVM daemon contacts the local PVM daemon over a dedicated TCP/IP connection. The remote host is now part of the PVM cluster and remains in the cluster until this connection is closed.
7. The SSH connection is closed. The remote PVM daemon keeps running.

Note
In case of remote simulation issues, check your firewall configuration at both ends (try disabling it temporarily to check if there is an issue). Several executables in the ADS installation directory open outbound and/or inbound TCP/IP connections. The ports are always in the operating system's dynamic TCP/IP port range, except for SSH which uses port 22. On Windows versions prior to Windows Vista, the dynamic port range is 1025-65535. Windows Vista and later versions use the range 49152-65535. On Linux the dynamic port range is 32768-61000. Adjust the firewall rules to allow connections from/to these ports.

Setting up a SSH server on Windows

The free software package [Copssh](#) can be used to deploy a SSH server on a Windows host. A version of this package is provided in the `<ADS installation directory>\pvm3\tools\Copssh` directory. Install the package using the suggested settings (administrator privileges are required). Do not attempt to change the autogenerated password for the `SvcCOPSSH` user (remembering this password is also not necessary).

Follow these steps for each user:

1. Log on as a user having administrative privileges (possibly but not necessarily the user to be activated).
2. Launch **Start > Copssh > 01. Activate a user.**
 - Enter the name of an existing user on the host. The short `%USERNAME%` login name is required, not a first and last name. This name will be the login name for SSH. If the user name exists inside a Windows domain, prefix the user name with the domain name and a backslash (these items are not part of the login name for SSH).
 - Use `/bin/bash` as command shell
 - Select *Remove copssh home directory if it exists.*
 - Do not select *Create keys for public key authentication.*
 - Select *Create link to user's real home directory.*
3. Click **Next**. An acknowledgment message is displayed.
4. Click **OK**.
5. Browse to the required directory `C:\Program Files\ICW\home/username` (`C:\Program Files (x86)\ICW\home/username` on a 64-bit system).
6. Open the file `.bashrc` with a text editor. It is important to use an editor that upholds end-of-line conventions (do not use Notepad). A suitable editor is [Notepad++](#), which is provided as `<ADS installation directory>\pvm3\tools\Notepad++\notepad++.exe`.
7. At the end of the `.bashrc` file, add an empty line followed by the following lines:

```
export EESOFPVM_TMP="$USERPROFILE\\PVM"
export HOME="$USERPROFILE"
```

cd

Note
Make sure that the file ends again with an empty line.

8. Save the `.bashrc` file and exit the editor.
9. Log on as the user to be activated and open a command prompt (**Start > Run > cmd.exe**)
10. Execute `mkdir "%USERPROFILE%\PVM"` in the command prompt.
11. Close the command prompt.
12. The user account is now accessible through ssh.

You can verify correct operation by using the shell login command `ssh <user>@<hostname>` from a Unix terminal or from a Windows command prompt (choose **Start > Run > cmd.exe**, then first type `cd <ADS installation directory>\tools\bin <return>`). Log on over SSH and run `<ADS installation directory>/pvm3/bin/auto_eesofpvm.exe` (replace all backslashes by forward slashes). If the output ends with a short string resembling either `7f000001:d789` or `/tmp/eesofpvmtmp025108.0`, configuration was successful (the exact (hexa)decimal digits can be different). Otherwise an error message indicating the problem is shown (messages about an unreadable file `pvmhost_cfg.txt` are normal and can be safely ignored).

Note
When accessing a Windows host remotely, network shares are generally not available as drive letters. When referring to remote network drives, use the UNC name (`<double backslash> <hostname> <backslash> <sharename>`) instead of the drive letter.

The Copssh installer assigns some special rights to the administrative `SvcCOPSSH` user in order to support passwordless operation. However, when the host is part of a Windows domain, the domain's security policy is often configured to periodically reject these rights to all users, rendering Copssh unusable. Typically Copssh appears to be working after installation, but all SSH logins fail the next day (as a result of the domain's security policy being applied overnight).

To permanently fix this issue:

1. Make sure you are logged in as a user in the Administrators group.
2. Move the file `C:\Program Files\ICW\etc\sshd_config` (`C:\Program Files (x86)\ICW\etc\sshd_config` on a 64-bit system) to a writeable location, such as the Desktop.
3. Open the moved file with a text editor. It is important to use an editor that upholds end-of-line conventions (do not use Notepad). A suitable editor is [Notepad++](#), which is provided as `<ADS installation directory>\pvm3\tools\Notepad++\notepad++.exe`.
4. Find the following line of text:

```
#PubkeyAuthentication yes
```

5. Change this line into (be sure to remove the pound sign):

```
PubkeyAuthentication no
```

6. Save `sshd_config` and exit the text editor.
7. Move `sshd_config` back to its original position.
8. Open the Services control panel (As privileged user, choose **Start > Run > services.msc**).
9. Double-click the entry for **Openssh SSHD**.
10. Stop the service.
11. In the *Log On* tab, change *Log on as:* to use *Local System Account* instead of the `SvcCOPSSH` user.
12. Click **Apply**.
13. Go back to the *General* tab and start the service.

After this fix, you cannot connect to the remote machine without needing to specify the password, yet normal password based operation will work again. If you plan to use password-based operation, this fix can be applied up-front (without running into the issue

first).

Passwordless SSH Operation

SSH can be configured to use public key authentication without the need to enter a password. In this mode of authentication, the local host presents a private key that must match a public key known to the remote host. Perform the following steps:

1. Generate the public/private key pair on the local host (the one that runs the Job Manager)
 1. On Windows, open a command prompt (**Start > Run > cmd.exe**) and type `cd <ADS installation directory>\tools\bin <return>`.
On Unix, open a terminal.
 2. Type `ssh-keygen -t dsa <return>`. Just hit the return key when prompted for the file location and the passphrase.
 3. Remember the location of the public key which is displayed.
2. Copy the public key `id_dsa.pub` from the local host to the remote host.
3. On the remote host, check for the existence of the `.ssh` directory and create it if missing. This directory must have read/write rights for the targeted user, and no rights assigned to anyone else.
On Windows, the `.ssh` directory is `C:\Program Files\ICW\home\username\.ssh` (`C:\Program Files (x86)\ICW\home\username\.ssh` on a 64-bit system).
On Unix, the `.ssh` directory is `~/.ssh`.
4. On the remote host, append the contents of `id_dsa.pub` to `<.ssh directory>/authorized_keys`. This file must not be writeable by other users than the targeted user.
If the `authorized_keys` file did not yet exist, just rename `id_dsa.pub` to `authorized_keys` instead of creating an empty file and appending.

The configuration is now complete. If passwordless operation does not appear to work in the Job Manager, verify operation using the shell login command `ssh <user>@<hostname>` from a Unix terminal or from a Windows command prompt (choose **Start > Run > cmd.exe**, then first type `cd <ADS installation directory>\tools\bin <return>`).

Warning

On Windows remote machines, network shares on the remote machine may refuse operation in passwordless mode because the user is not authenticated to the Windows domain.

Specifying Configuration Files and Environment Variables

Specifying Job Manager Preferences

The Job Manager preferences are stored as `<name>=<value>` lines in the file `<home directory>/hpeesof/config/simsys.cfg`. These settings override those in the file `<ADS installation directory>/custom/config/simsys.cfg`, which in its turn overrides settings in the file `<ADS installation directory>/config/simsys.cfg`. The file `<ADS installation directory>/custom/config/simsys.cfg` can be used for site-wide deployment of Job Manager preferences. Note however that the Job Manager GUI will always incrementally save preference changes to the file `<home directory>/hpeesof/config/simsys.cfg`.

The preferences are mapped to the `simsys.cfg` file as follows:

Advanced Design System 2011.01 - EM Simulation

Preference name	Variable name in <code>simsys.cfg</code>	Acceptable values
Start the eesofpvm backend when ADS starts	ADS_STARTS_EESOFPVM	TRUE or FALSE
Close EEs of Job Manager when ADS exits	ADS_STOPS_JOBMANAGER	TRUE or FALSE
Shut down the eesofpvm backend when both ADS and EEs of Job Manager have exited	STOP_ORPHAN_EESOFPVM	TRUE or FALSE
Shut down the eesofpvm backend when EEs of Job Manager exits	JOBMANAGER_STOPS_EESOFPVM	TRUE, FALSE or ASK
Use separate eesofpvm backends for different combinations of user/host/\$DISPLAY/\$HPEESOF_DIR	SEGREGATE_EESOFPVM_SESSIONS	TRUE or FALSE
Use the local \$HPEESOF_DIR variable	DEFAULT_HPEESOF_DIR	<local> (include the angular brackets)
As defined in the eesofpvm cluster configuration file	DEFAULT_HPEESOF_DIR	<i>empty</i>
User defined:	DEFAULT_HPEESOF_DIR	<i>string</i>
Default working directory:	DEFAULT_WORKING_DIRECTORY	<i>string</i>
Default additional eesofpvm options:	DEFAULT_EESOFPVM_OPTIONS	<i>string</i>
Host specific overrides (one entry per override)	HOST_CONFIG_<unimportant unique suffix>	<Hostname> <ADS installation directory> <Working directory> <Additional eesofpvm options>

Running PVM Boot Script

When a Unix host is added to the PVM cluster, the optional Korn shell script file `~/ .eesofpvmprofile` is sourced before the PVM daemon is started on the host. This file can be used to set additional environment variables. A typical use case is extending the `PATH` and `LD_LIBRARY_PATH` environment variables to ensure accessibility of the tools required to control a third party job management framework.

Specifying Environment Variables

The optional environment variable `EESOFPVM_TMP` indicates the directory in which the PVM daemon advertises its configuration and writes logging information. You can specify the following settings:

- Windows: the environment variable `TEMP`, or if this is unset, `C:\TEMP`
- Unix: `/tmp`

Specifying Legacy Settings

The following settings are deprecated but supported for legacy setups:

- The PVM cluster configuration file: this is by default `<home directory>/hpeesof/config/pvmhost_cfg.txt`. Its path can be overridden by the `<home directory>/hpeesof/config/hpeesof.cfg` variable `EESOFPVMHOSTFILE`.
- The environment variable `EESOFPVM_RSH` pointing to a remote shell executable. This variable now additionally requires `EESOFPVM_RSH_NOAUTO=<any nonempty string>` to be set in order to be recognized.

Using EMPro 3D Components in ADS

You can import EMPro default 3D library components and parametrized 3D Components in ADS. This section provides information about how to use the EMPro 3D components in ADS 2011.01.

Using EMPro Default 3D Components in ADS

The default 3D library components are provided as part of the EMPro installation. The following types of default library components are provided by EMPro:

- Basic Shapes: Block, SolderBall, Cylinder
- Span: Basic shapes that span a substrate layer.
- Arrays: NxM arrays of basic shapes
- Span Arrays: NxM arrays of spanning shapes
- Array Outlines: $2(N-1)+2(M-1)$ basic shapes that form the outline of the array.
- Span Array Outline: $2(N-1)+2(M-1)$ spanning shapes that form the outline of the array.

Note
For more information about the default 3D library components provided by EMPro, refer [EMPro Default 3D Library Components](#)

Installing EMPro Design Kit in ADS

Before installing the EMPro design kit containing 3D library components in ADS 2011.01:

1. Get the latest version of the design kit.
2. The ADS license **emds_3d_param** is required for 3D preview and EM simulation of the EMPro default 3D Components. This license is free; but without it the 3D components are silently ignored. If you do not have this license, you can install the design kit, insert the 3D default components into designs, and perform 3D Preview and EM operations. However, 3D components will be excluded from these operations without any warning or indication of any kind.
3. 3D components include a 2D footprint that is visible in the ADS layout window, and, additionally, a full 3D model that is used in 3D operations. These components do not include a Schematic symbol or representation, and are not directly supported by ADS circuit simulation.

Importing EMPro Default 3D Library Components in ADS 2011.01

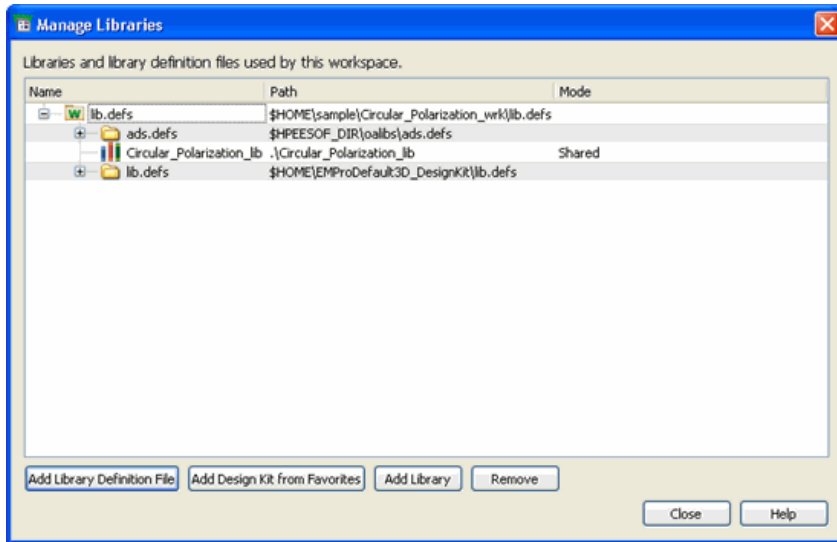
To install EMPro default 3D library components in ADS:

1. Select **DesignKits > Unzip DesignKit** from the ADS Main window.
2. Browse to the location of the default component design kit, which is the EMPro 2011 installation directory under **data\ADS_integration_\EMProDefault3D_DesignKit_ADS2011.zip**.
3. Select a directory to unzip the design kit.

For more information on installing design kits, refer to *Design Kit Installation and Setup* in [ADS documentation](#) .

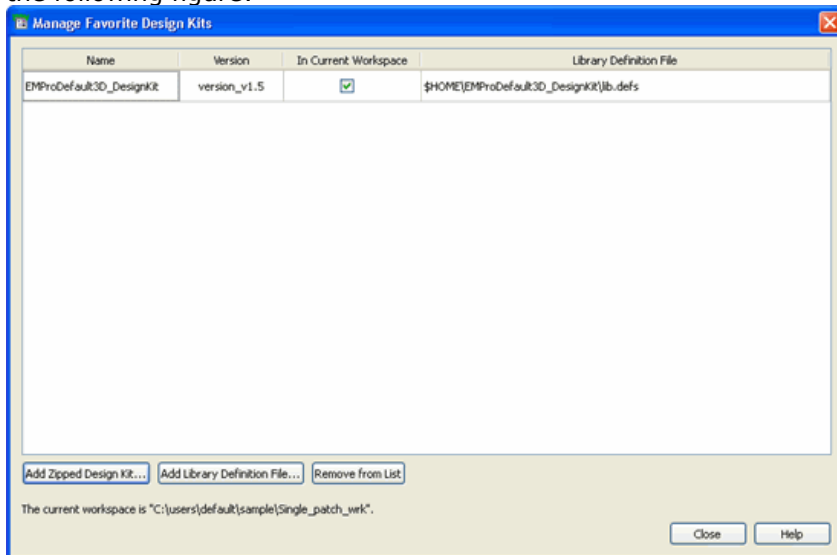
Adding the EMPro Default 3D Library to an Existing Workspace in ADS 2011.01

1. Select **DesignKits > Manage Libraries** from the ADS Main window.
2. Click **Add Library Definition File** to select the lib.defs file of the library you want to add in the workspace. A message window appears to confirm if you want to add this library to the current workspace.
3. Click **Yes**. The library is added to your current workspace.

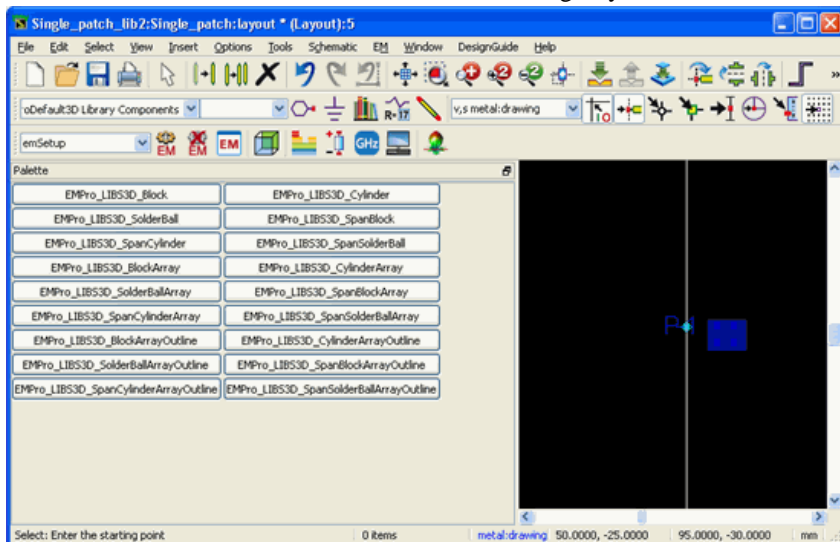


To use EMPro default 3D library components:

1. Ensure that you have a license for the Agilent FEM Simulator.
2. Install the design kit: *EMProDefault3D_DesignKit_ADS2011*.
3. Select **DesignKits** > **Manage Favorite Design Kits** from the ADS Main window. The design kit is displayed in the Manage Favorite Design Kits window, as shown in the following figure:



4. Create a new design and open the Layout window.
5. From the **Library** drop-down list, select the library containing the 3D components. The palette will change showing a single component. Click the component and drag it on the Layout window. The parameter window of the component is displayed, as shown below:



6. Customize the layout layers (optional).
7. In a Layout window, insert the 2D shapes, ADS layout components, and any other required objects.
8. Specify the substrate (such as, layer properties, thickness, and expansion model).

Note
Only substrate layers that are mapped to layout layers are *seen* by EM.

9. Insert one or more 3D components, picking them from the palette or the component library. Assign parameter values. The layout is now ready for 3D EM Preview.
10. Select **EM > 3D EM Preview** to see a preview of the layout.

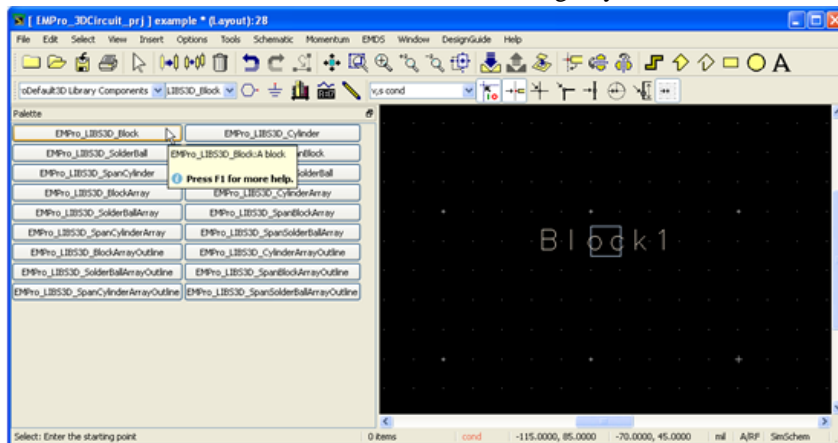
Select the required values of the parameters and apply the settings. Single-clicking a parameter will give a description of the parameters use. For reference, a description of the parameters used in this example are repeated below:

- **Layer:** Specifies the physically mapped layer that will be used to define the final z-position of the 3D component.
- **PlaceCustomComponentOnTop:** Set this parameter to **Yes** to place the 3D Component on **top** of the physically mapped layer selected by the **Layer** parameter.
- **Layer2:** Specifies the layer that is used to display the first cross-section. This layer may or may not be physically mapped. The main intent is to provide guidance about the final positioning of the component with respect to other elements present in ADS Layout.
- **Layer3:** Specifies the layer that is used to display the second cross-section. This layer may or may not be physically mapped. The main intent is to provide guidance about the final positioning of the component with respect to other elements present in ADS Layout.
- **package_height:** Specifies the EMPro generated parameter. Changing this parameter will be reflected in the final 3D geometry that can be verified in the 3D Viewer.

Inserting Design Kit Components into ADS Layouts

After installing the design kit, insert 3D components in ADS layouts by performing the following steps:

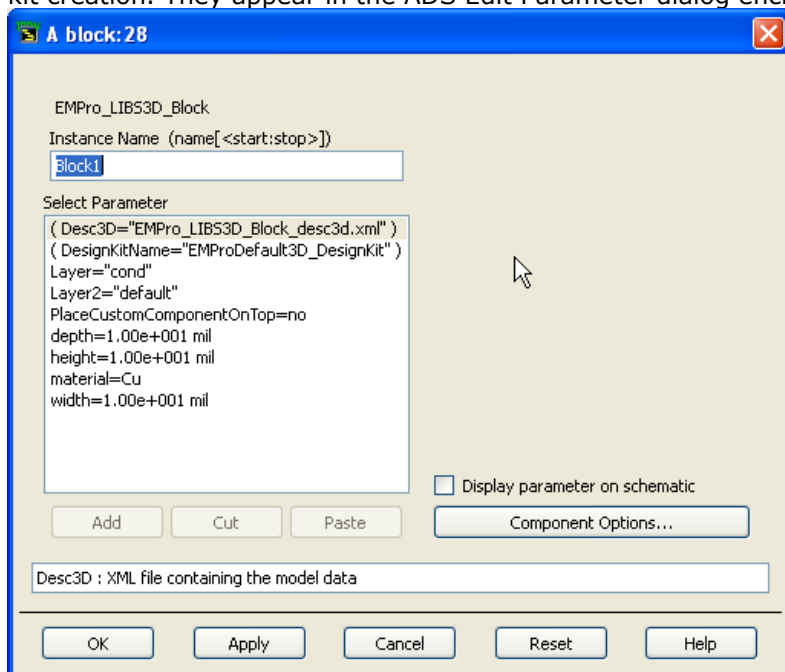
1. Select *EMProDefault3D Library Components* in the Layout window to install the 3D Component palette, as shown in the following figure:



- Select a 3D Component from the palette to insert into the Layout. Note that all 3D Component names begin with *EMPro_LIBS3D*.

Setting Parameter Values for the 3D Components: Example EMPro_LIBS3D_Block

Parameter values for the 3D Components are set in the same way as other ADS Layout components. The 3D Components, however, have some parameter values that cannot be modified: **Desc3D** and **DesignKitName**. These values are fixed at the time of the design kit creation. They appear in the ADS Edit Parameter dialog enclosed in parenthesis.



All 3D Components have two layer parameters: *Layer* and *Layer2*. *Layer* specifies the layout layer on which the component either:

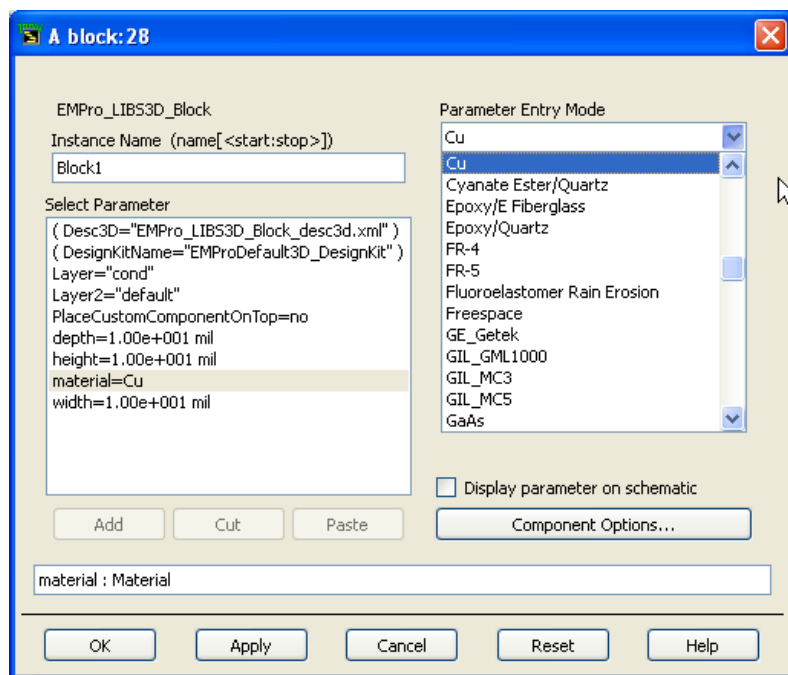
- spans, if the component is a spanning component (with 'Span' in its name).
- is placed on the top of, if the value of the parameter **PlaceCustomComponentOnTop** is set to yes.
- is placed on the bottom of, if the value of the parameter **PlaceCustomComponentOnTop** is set to no.

Note
The value of the parameter **PlaceCustomComponentOnTop** is relevant only if the associated layout layer is specified for an EM simulation as having a *Thick* model, with up or down expansion.

In the above example, the parameter *Layer* specifies the layout layer ("cond") that the block is placed on the bottom of, because it is not a spanning component ('Span' is not in its name), the layout layer "cond" is defined with a Thick, expansion up model, and the value of its parameter **PlaceCustomComponentOnTop** is set to **no**.

The parameter **Layer2** is used to specify the layout layer to be used to draw the 2D *footprint* or *mark-up* of the component. By default, it is set to the layout layer named "default" which, by convention, is not mapped to a substrate layer. This means that the 2D markup will appear in the ADS layout window (if visibility is turned on for the layer "default") and therefore will provide a visual indication of where the 3D component will be situated relative to the other layout elements, but this 2D shape will not contribute in any way to the analysis of the 2D or 3D component. The value of *Layer2* can be set to a layout layer that is mapped to a substrate layer (and there are reasons why it is sometimes a good idea to do this that will be explained in the next section of this document), but in these cases it will be included in 2D and 3D analysis.

All 3D Components also have the parameter material. A material is a property defined in EMPro, which is used in EM calculations, and is stored with the 3D component model. The design kit configures a long list of available materials that can be selected for this parameter value.



In addition to these common parameters, each component has a unique set of parameters that are used to specify its shape. For example, the *EMPro_LIBS3D_Block* component has parameters depth, height, and width for specifying its dimensions.

Establishing Compatibility: Substrate Definition and Layer Mapping

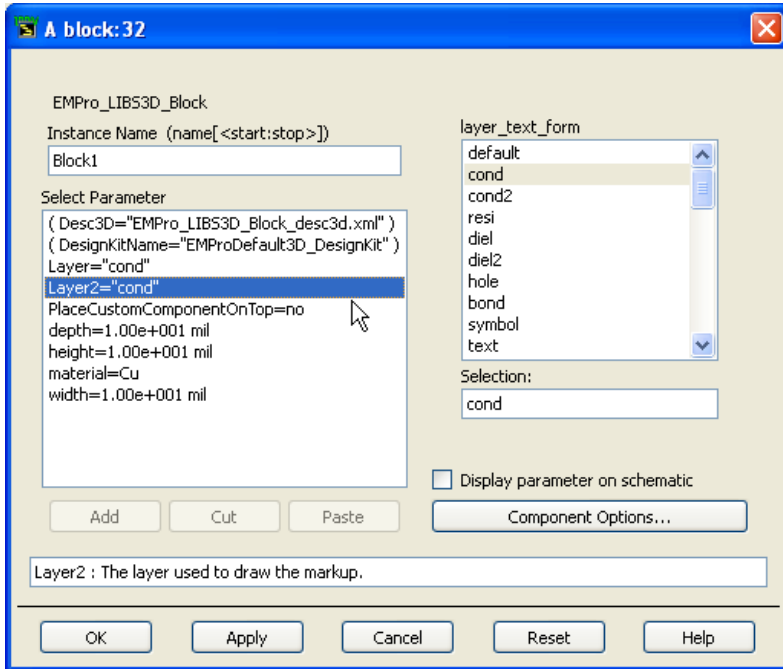
Before performing EM simulations or 3D preview of layouts that include 3D components, the substrate has to be defined and its layers mapped to layout layers in a way that is consistent with the 3D component parameter values. Open the window to specify the substrate definition and layer mapping by selecting the layout menu: **EM > Substrate**.

All layout layers that are used as values for a 3D Component Layer parameter must be mapped to a substrate layer. As shown below, the layout layers "cond", "pcvia1" and "cond2" are all mapped to substrate layers, and can be used as values for the 3D components' Layer parameter. Refer to the ADS on-line documentation if further help is needed to understand EM requirements with respect to substrate definition and layer mapping.

Special Considerations for 3D Operations

First, there must be at least one 2D object on each layer that a 3D Component either spans or has been placed on. If not, the ADS 3D Preview and EM layout processor will not be able to access the 3D Component model. You should ensure that there is some 2D shape on the 3D Component Layer. One easy solution is to put the markup on the same

layer as the 3D object; this is done by setting the value of the *Layer2* parameter to the value of the *Layer* parameter.

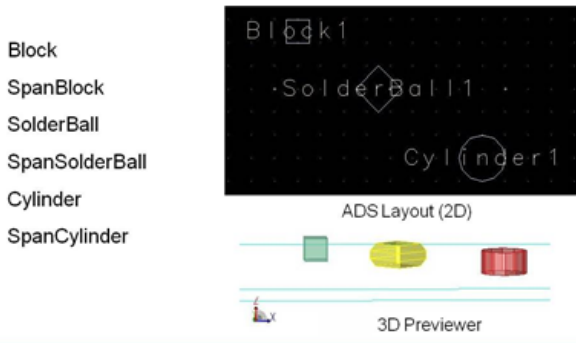


Troubleshooting Hints

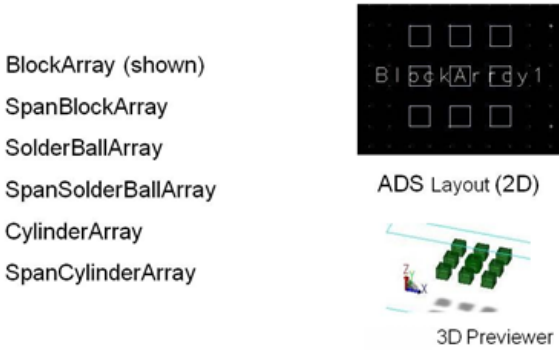
- If you have inserted one or more 3D component into your layout, but do not see any 3D components in the 3D Preview, then most likely you do not have the necessary codeword (emds_3d_param).
- If you have inserted a 3D component and changed some parameters that affect the dimensions of the object, but see in the Preview window the object with the default dimensions, then there is a problem with the regeneration of the 3D object. (If any type of a problem is encountered when the component is regenerated, the default object will appear in the viewer.) This is caused by an invalid configuration, a corrupt database, or some other problem of that nature. About the only thing to try is to re-start ADS and try using the 3D components in a new design, and/or re-installing the design kit, and/or re-installing ADS.
- If you insert one or more 3D component into your layout, an error might be displayed after selecting a 3D Preview. It is likely that you have placed a 3D component on a layer that does not have any 2D shapes on it. The solution to this problem is to either specify the *Layer2* parameter to be the same as its *Layer* value (effectively inserting the shape of the 2D markup on the layer that the 3D model was placed on), or insert some 2D shape on any layer that has only 3D Components on it.
- If you see 3D components extending outside the bounding box in a 3D Preview window, then you will need to auto-extend the boundary so that the 3D components are fully contained in the bounding box.

EMPro Default 3D Library Components Summary

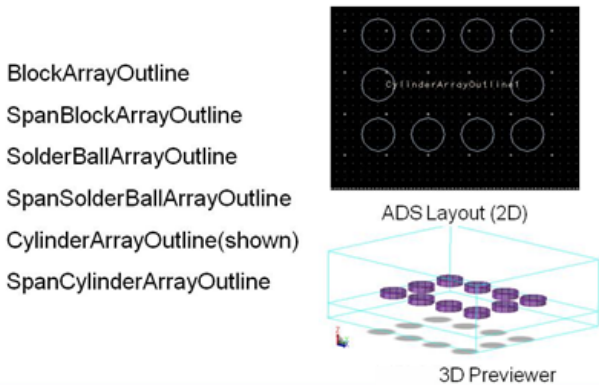
Basic and Span



Arrays and Span Arrays

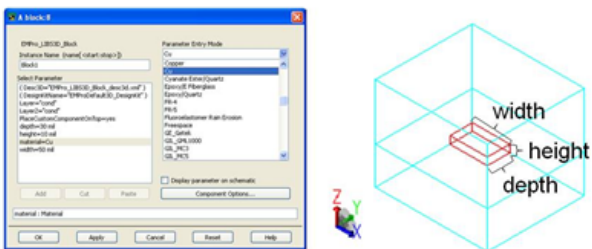


Array and Span Array outlines

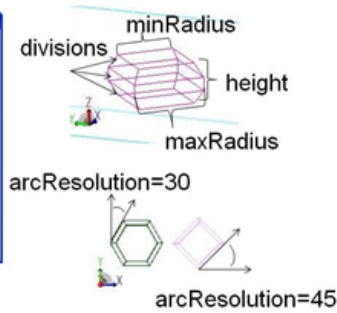
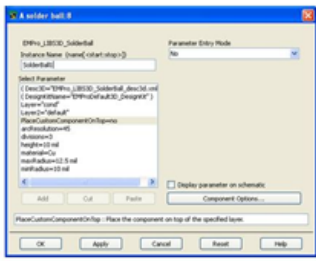


Parameterization of Components

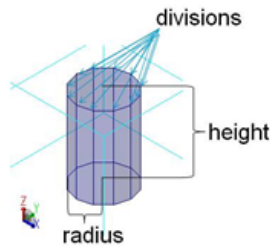
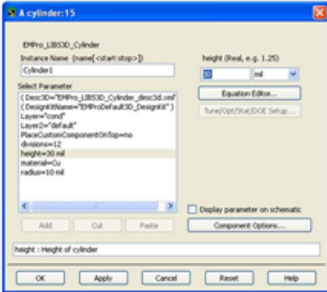
Basic Shape Drawing - Block



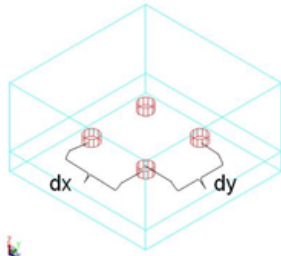
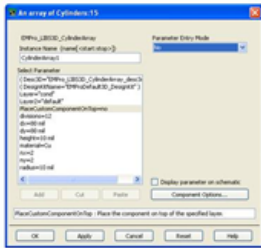
Basic Shape Drawing - SolderBall



Basic Shape Drawing - Cylinder



Array and Array Outline Drawing



nx = number of components in X direction
ny = number of components in Y direction

Using Parametrized EMPro 3D Components in ADS

Before using the parametrized EMPro 3D components in ADS 2011.01, you need to convert the design kits generated with EMPro by using the `de_convert_3D_component()` tool available in ADS 2011. To use parametrized EMPro 3D components in ADS 2011.01:

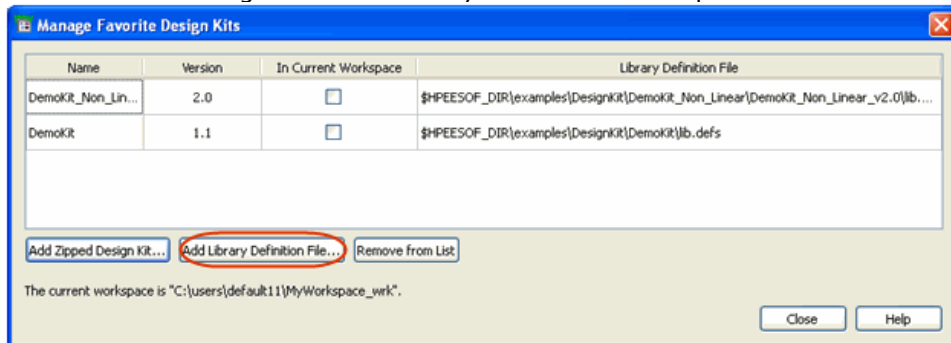
1. Close all open workspaces in ADS 2011.01.
2. If a 3D component is contained in a zip file, extract the file by using any regular unzip program.

Note
Do not select **DesignKits > Unzip DesignKit** from the ADS Main window.

3. From the ADS Main window select **Tools > Command Line**.
4. Enter the command `de_convert_3D_component()`.
5. Click **Apply**.
6. An information message is displayed asking you to select a directory of the unzipped 3D components. Click **OK**.
7. Select a directory in the **Convert EMPro 3D Component** dialog box.
8. Click **Choose**. Once a design kit is converted, you can use it both in ADS2011 and older versions of ADS. However, you should not add components to a kit that already has been converted.

To use EMPro 3D library components in ADS 2011.01:

1. Ensure that you have a license for the Agilent FEM Simulator.
2. Click **DesignKits > Manage Favorite Design Kits** from the ADS Main window.
3. Click **Add Library Definition File** and select the **lib.defs** file. You will receive a prompt if you want to add it to the current workspace.
4. Click **Yes**. The design kit is added to your current workspace.



5. Create a new design and open the Layout window.
6. From the **Library** drop-down list, select the library containing the 3D components. The palette will change showing a single component. Click the component and drag it on the Layout window. The parameter window of the component is displayed, as shown below:
7. Select **EM > 3D EM Preview** to see a preview of the layout.

Select the required values of the parameters and apply the settings. Single-clicking a parameter will give a description of the parameters use. For reference, a description of the parameters used in this example are repeated below:

- **Layer**: Specifies the physically mapped layer that will be used to define the final z-position of the 3D Component
- **PlaceCustomComponentOnTop**: Set this parameter to **Yes** to place the 3D Component on **top** of the physically mapped layer selected by the **Layer** parameter.
- **Layer2**: Specifies the layer that is used to display the first cross-section. This layer may or may not be physically mapped. The main intent is to provide guidance about the final positioning of the component with respect to other elements present in ADS Layout.
- **Layer3**: Specifies the layer that is used to display the second cross-section. This layer may or may not be physically mapped. The main intent is to provide guidance about the final positioning of the component with respect to other elements present in ADS Layout.
- **package_height**: Specifies the EMPro generated parameter. Changing this parameter will be reflected in the final 3D geometry that can be verified in the 3D Viewer.

Performing SI/PI Analysis

Overview

The SI/PI Analyzer feature assists in setting up SI or PI EM and circuit simulations. This feature consists of two parts:

- Setup wizard: The setup wizard guides you through the process of creating a new cell that contains only selected nets (physical interconnects) to be analyzed. The output of the setup wizard is a cell that is ready for EM analysis.
- Analysis guide: The analysis guide assists in setting up typical SI or PI circuit simulation testbench. In this release, you can create Power Distribution Network (PDN) analysis. The output of the analysis guide is a circuit schematic, ready to be simulated, a data display template that captures relevant output figures. In addition, the analysis guide supports Momentum current visualization based on the results.

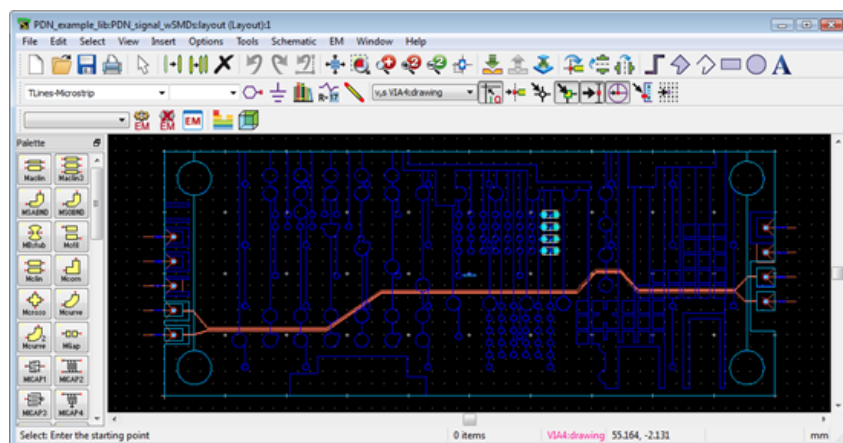
You can access SI/PI Analyzer by selecting **Tools > SI/PI Analyzer** in a Layout window.

To set up an SI/PI analyzer, refer to the following sections:

- *Setting up SI and PI Analyzer (em)*
- *Using SI and PI Analyzer (em)*

Setting up SI/PI Analyzer

The SI/PI Analyzer Setup wizard guides you through the process of creating a new hierarchical cell that contains only selected nets (physical interconnects) and components to be analyzed. All the objects that were not selected are removed from the output cell created by the wizard. You can use the hierarchical cell for circuit and EM simulations. The structures to be modeled by the Momentum EM simulator are grouped in a sub cell. Any components mounted on the interconnects are in the top level cell. The following figure displays the design of a hierarchical cell:



This cell will be used to illustrate how the power distribution network can be setup for analysis.

Using the SI/PI Analyzer Setup

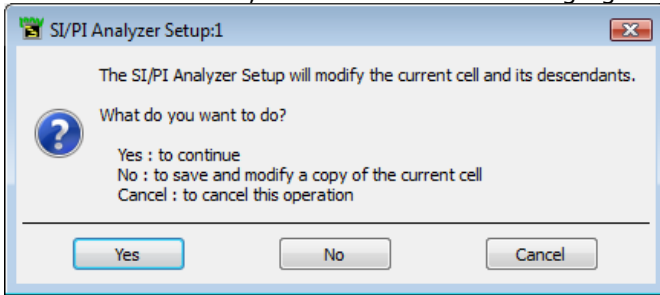
1. Open the layout view of the cell you want to analyze.

Note

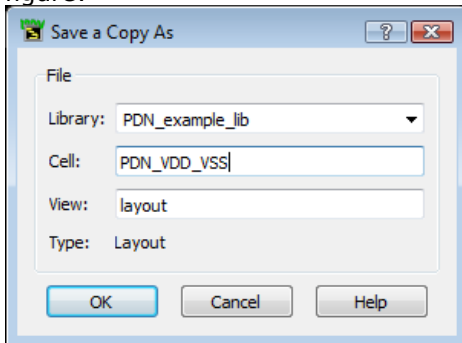
The example workspace used in this section is present at the following location:
 %EESOFHOME%\examples\Momentum\HighSpeedDigital\PDN_example_wrk.7zap.

2. Enable **Physical Connectivity Engine (PCE)**.
3. Select **Tools > SI/PI Analyzer > Setup** to open the Setup wizard.

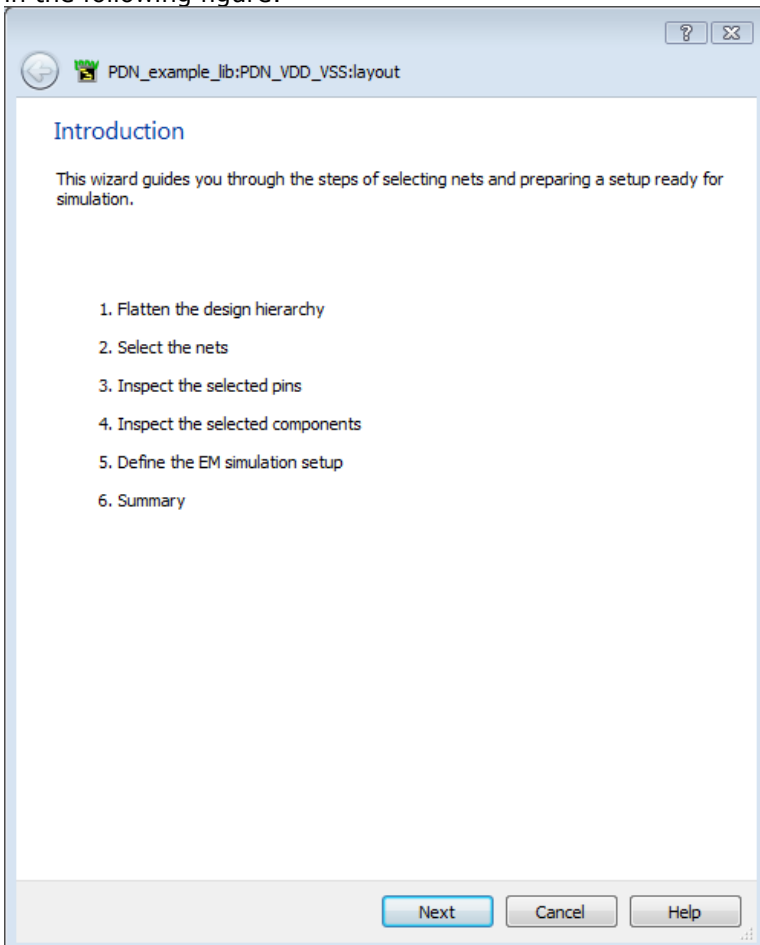
4. A message box appears stating that SI/PI Analyzer Setup will modify the current cell and its descendants, as shown in the following figure:



- Click **Yes** to work in the existing cell. Note that the cell will be modified, for example unselected objects are removed from the cell.
- Click **No** to create a new cell. A copy of the current cell is saved under a new name and values in the current cell are not changed, as shown in the following figure:



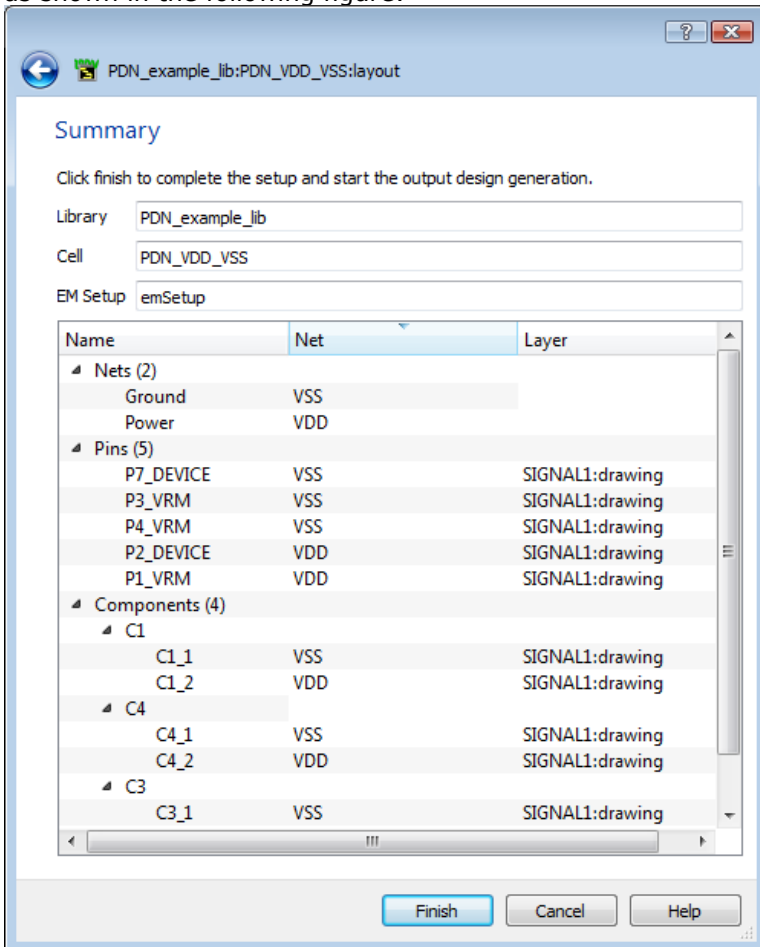
5. The **Introduction** screen is displayed that outlines the steps in the wizard, as shown in the following figure:



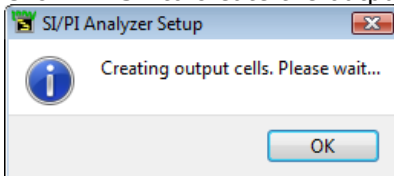
6. Click **Next** to display the **Flatten the Design Hierarchy** screen.
 7. Specify the options for flattening the design hierarchy (applies to cells with

components only). For more information, refer to [Flattening the Design Hierarchy](#).

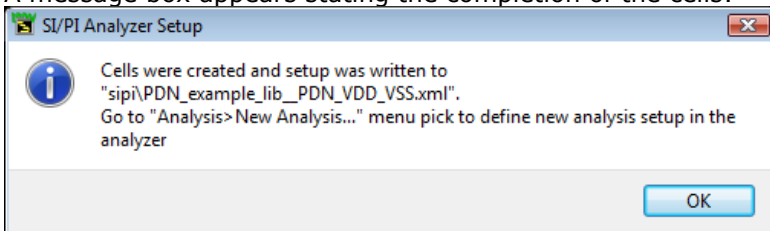
8. Click **Next** to display the **Nets** screen.
9. Select the nets to be modeled by EM. For more information, refer to [Selecting Nets](#).
10. Click **Next** to display the **Pins** screen.
11. Inspect the pins connected to the selected nets. For more information, refer to [Analyzing Pins](#).
12. Click **Next** to display the **Components** screen.
13. Inspect the components connecting the selected nets (applies to cells with components only). For more information, refer to [Inspecting Components](#).
14. Click **Next** to display the **EM Simulation Setup** screen.
15. Specify the EM simulation setup. For more information, refer to [Specifying EM Simulation Setup](#).
16. Click **Next** to display the **Summary** screen.
17. The Summary screen provides information about the library, cell, EM setup, and pins, as shown in the following figure:



18. Click **Finish** to create the output cells.

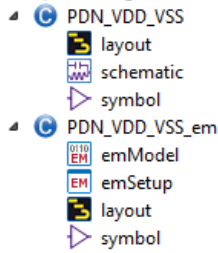


19. A message box appears stating the completion of the cells:

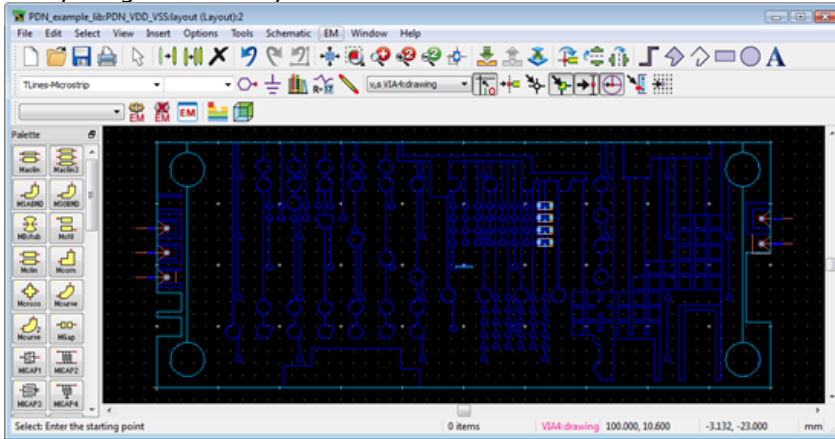


20. Click **OK** to open the SI/PI Analyzer analysis guide.

21. This creates two new cells as shown below:



22. The top level PDN_VDD_VSS cell has a schematic, layout, and symbol view. This cell contains all component instances and an instance of the sub cell that bundles everything modeled by EM:



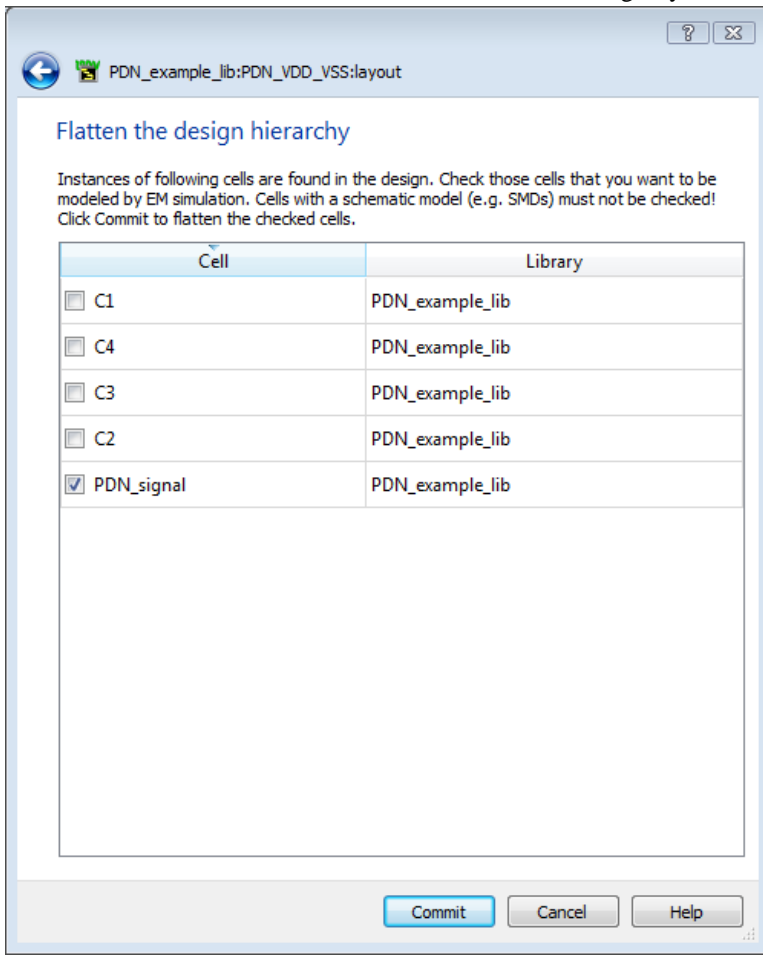
The sub level cell PDN_VDD_VSS_em cell has a layout, symbol, emSetup, and emModel view. Everything is properly connected, the schematics used named connections, and are ready to be used in circuit and EM simulations.

Flattening the Design Hierarchy

Using the Setup wizard, the **Flatten the Design Hierarchy** screen is displayed only when the instances of other cells are found in the current cell. You need to select all the instances that must be modeled by the EM simulation. Do not select cells with other components, such as schematic model cells SMDs. You must provide models for these cells in a schematic view of the component cells.

Warning
 SI/PI setup wizard will add a top level pin per inst pins of the components which you don't select here at the same location/layer of the inst pins so that other models such as schematic model can be connected later. Make sure that the layer of such component's pins are correctly mapped in the substrate. Otherwise having such top level pins are causing EM simulation error later as these pins are not mapped in the substrate layers.

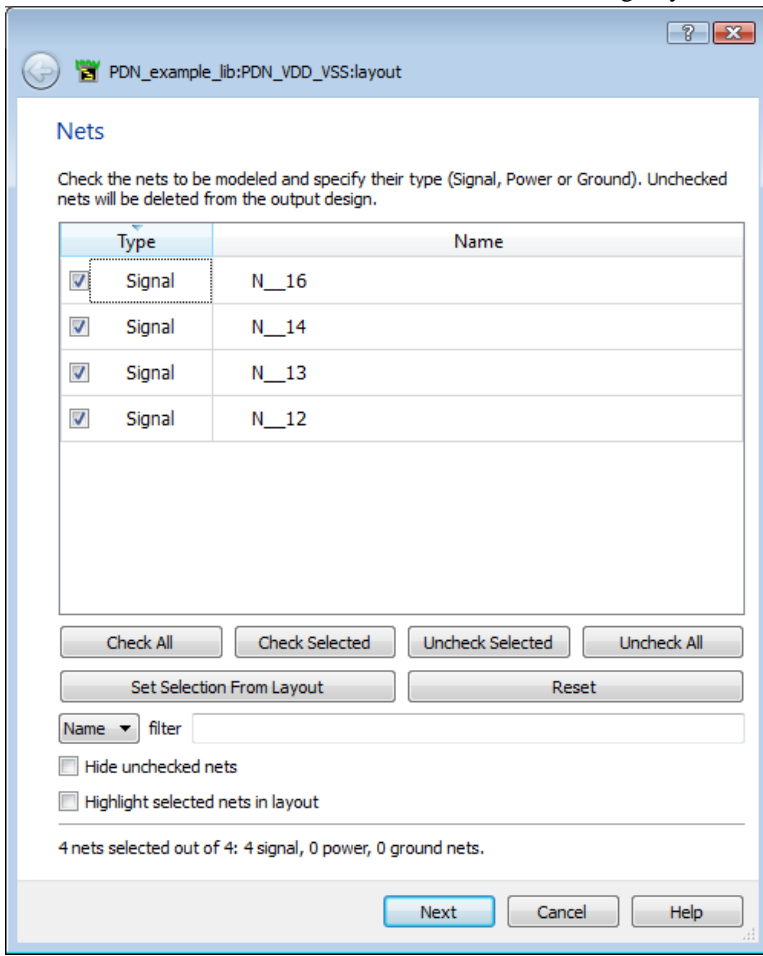
In the example shown below, the PDN_signal cell contains a power net, a ground net and two signal nets from a differential trace. This cell will be modeled by EM. The other cells are instances of surface mounted device (SMD) components.



Click **Commit** to flatten the selected cells. This step is irreversible and modifies the current cell for the first time.

Selecting Nets

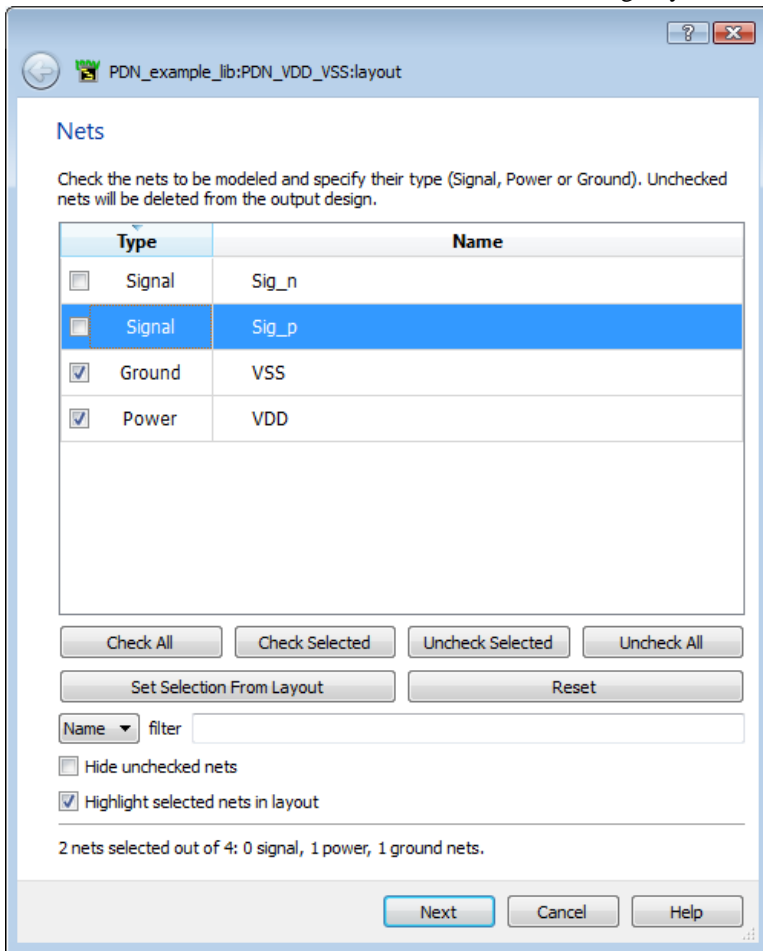
In the **Nets** screen, you need to specify the nets you want to analyze. The nets, extracted by the Physical Connectivity Engine, are shown in a table. Each net has a type and a name. The default net type is **Signal**. Other choices are **Power** or **Ground**. The net type facilitates sorting or filtering the nets. The default PCE net names start with **N__**. Double-click the net name to specify the required name. The new name is stored in the cell and automatically updated in the table. Multiple PCE nets given the same name will be considered as one net in the SI/PI analyzer. The following figure displays the Nets screen in the SI/PI Analyzer Setup wizard.



All selected nets are saved in the output cell. However, the nets that are not selected are excluded from the analysis. In the Nets screen, you can perform the following tasks:

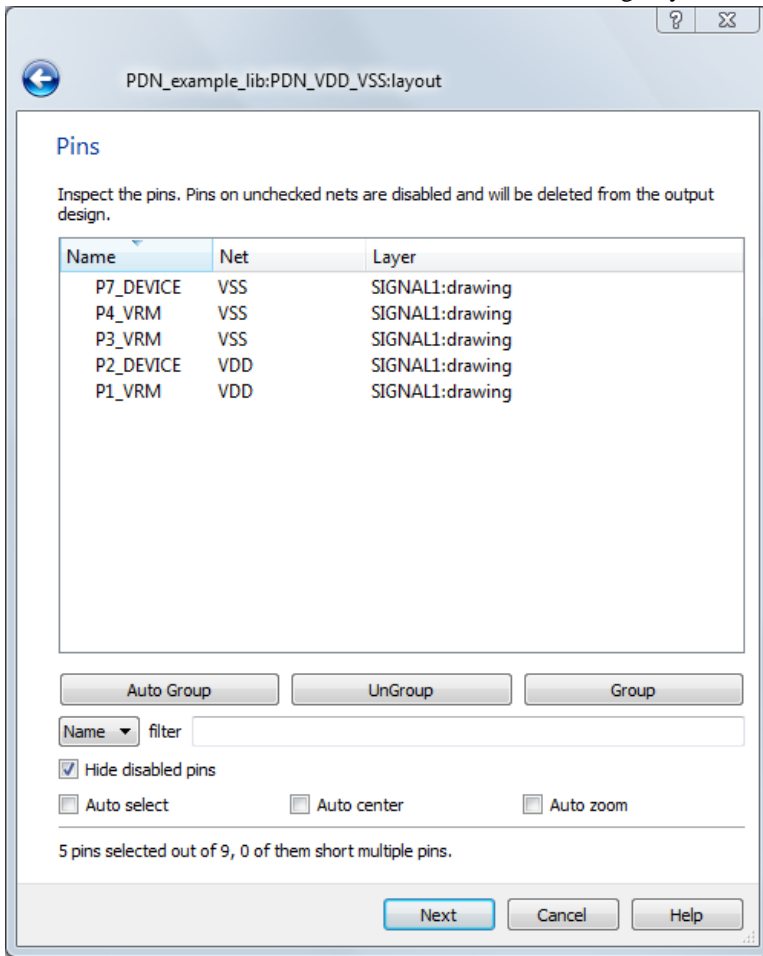
- **Setting the Selection From Layout:** You can select all nets in the table based on the selection made in the layout by clicking **Set the Selection From Layout**. The layout is scanned for all selected shapes and pins. The corresponding net is selected in the table.
- **Filtering:** You can display the required nets in table based on type or name from the **Filter** list. Regular expressions are also supported while filtering the nets.
- **Selecting Nets:** You can hide the nets that are not selected in the Nets screen by selecting the **Hide unchecked Nets** option. You can also highlight selected nets in the layout window by selecting the **Highlight selected nets in layout** option.

Renaming the nets, setting the appropriate net type and removing selection from the signal nets generates the following screen.



Analyzing Pins

The Pins screen enables you to analyze the pins that are connected to the selected nets, as shown in the following figure:

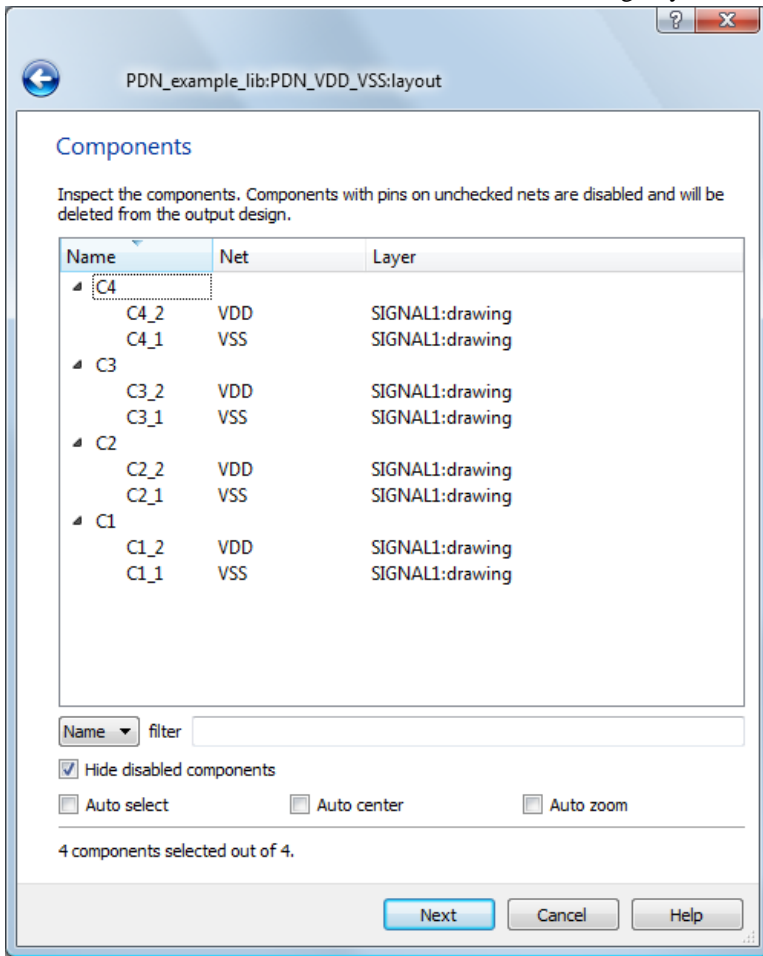


In the Pins screen, you can perform the following tasks:

- **Grouping pins automatically:** You can select pins in the table and create groups automatically for selected pins by selecting **Auto Group**. Pins on a same net and a same layer will become a group.
- **Removing pins from a group:** You can remove grouped pins by selecting the required group and click **UnGroup**.
- **Grouping Pins:** You can select the required pins and create a group by selecting **Group**. Grouping is possible for pins on a same net only.
- **Filtering:** You can display the required nets in table based on type or name from the **Filter** list. Regular expressions are also supported while filtering the nets.
- **Selecting Nets:** You can hide the nets that are not selected in the Nets screen by selecting the **Hide unchecked Nets** option. You can also highlight selected nets in the layout window by selecting the **Highlight selected nets in layout** option.

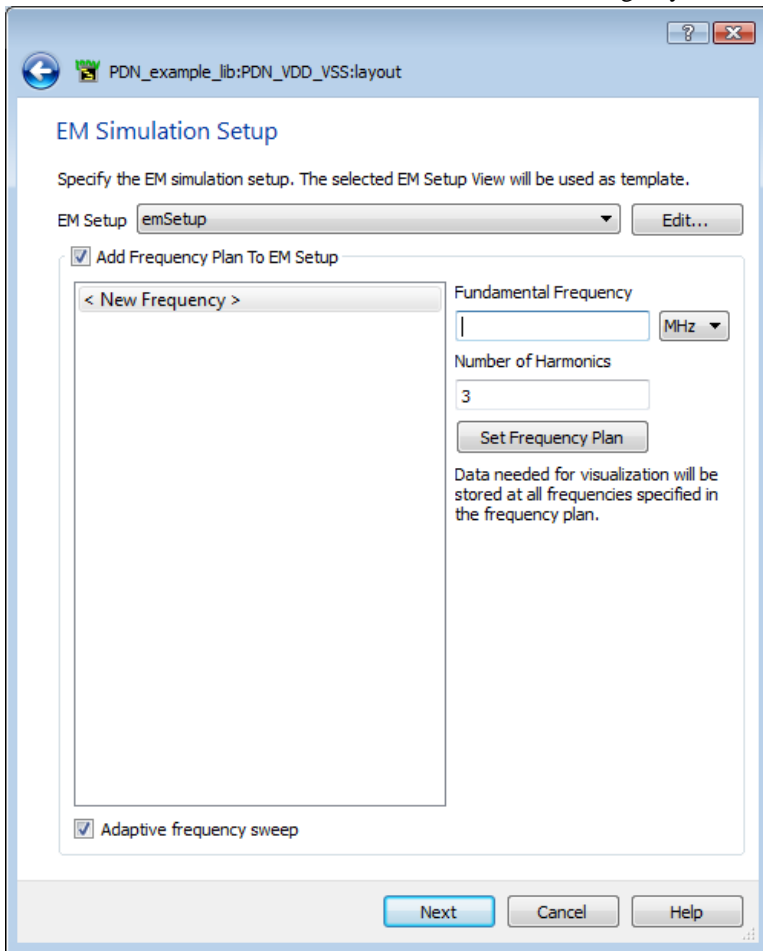
Inspecting Components

All components connecting the checked nets are shown on this page. Components with pins on unchecked nets will not be included in the output cells.

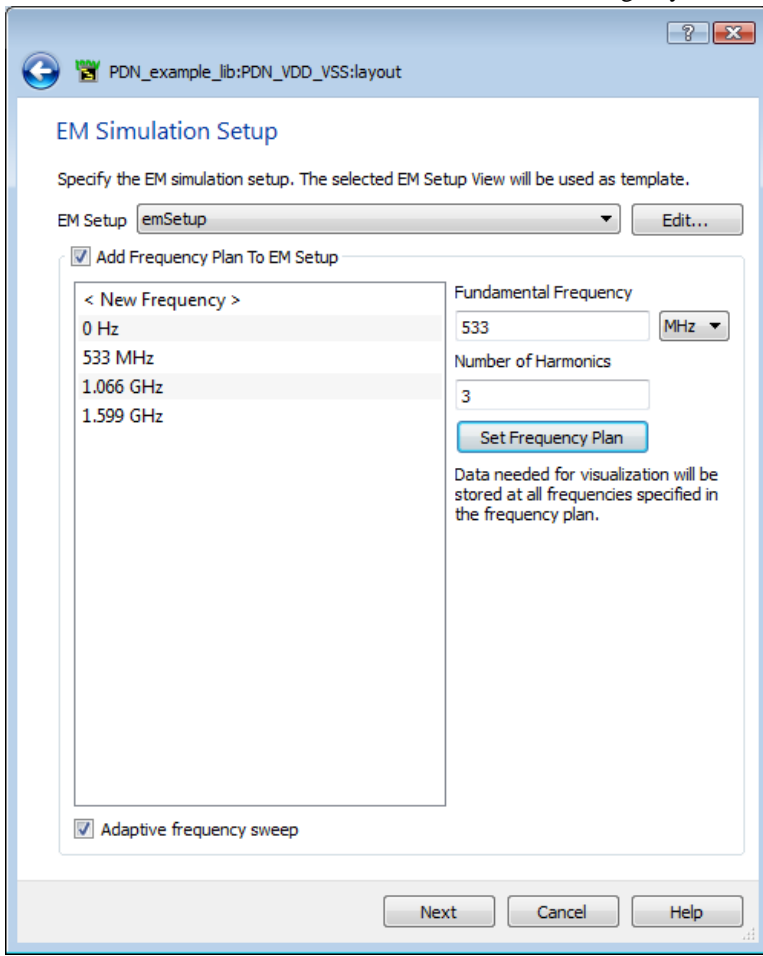


Specifying EM Simulation Setup

The EM Simulation Setup screen allows you to specify the EM Setup will be used for the EM Model generation. You can select the required view from the **EM Setup** list box. In the absence of existing views, a new EM Setup view named emSetup will be created. Click **Edit** to open the EM Setup window to define the required EM simulation settings.



In the EM Simulation Setup screen, you can define or extend the frequency plan of the EM Setup. Specify a fundamental frequency and a number of harmonics and click on **Set Frequency Plan**. You can specify additional frequencies by double-clicking **<New Frequency >**.



Using SI and PI Analyzer

Using the SI/PI analyzer, you can perform the following types of analysis:

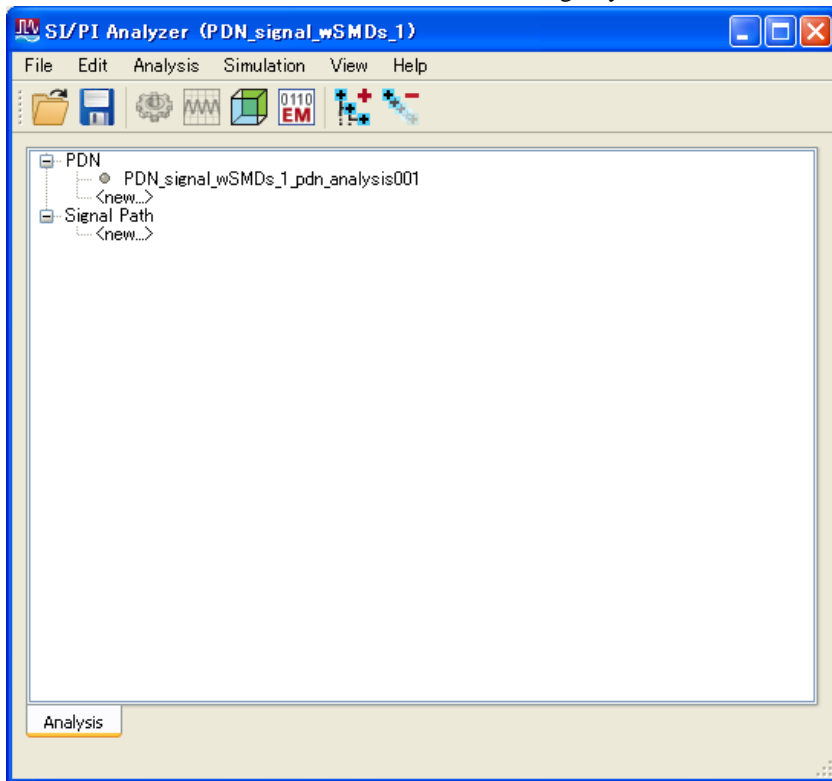
- **Power Distribution Network(PDN) Analysis:** The PDN analysis enables you to:
 - Extract PDN impedance
 - Visualize current distribution with SMD components (if any) in 3D Visualizer.
- **Signal Path Analysis:** The signal path analysis enables you to extract signal path characteristics such as R, L, G, and C characteristic impedance.

Before performing the analysis, you need to set up SI/PI Analyzer. For more information, refer *Setting up SI and PI Analyzer (em)*.

Opening SI/PI Analyzer

To open the SI/PI Analyzer window:

1. Open the layout view of the cell you want to analyze.
2. Enable **Physical Connectivity Engine (PCE)**.
3. Select **Tools > SI/PI Analyzer > Analysis**. The SI/PI Analyzer window is displayed, as shown in the following figure:

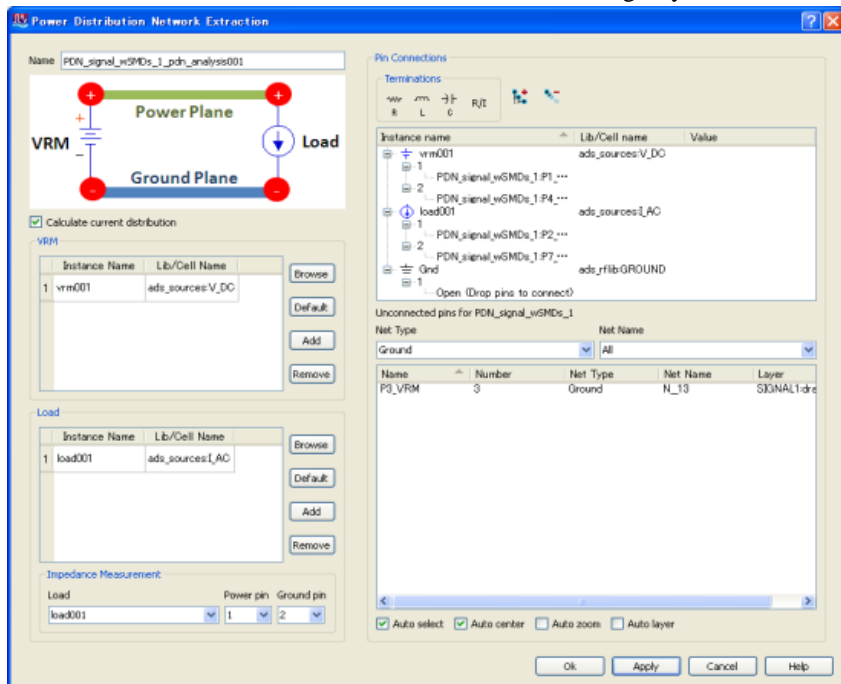


The SI/PI Analyzer window displays a list of analysis that you have created. In this window:

- A grey (●) icon indicates that the analysis is not performed yet or finished with an error.
- A green (●) icon indicates that the analysis is finished successfully.

Defining Power Distribution Network Analysis

To define a PDN analysis, select **Analysis > New analysis > PDN impedance** in the SI/PI Analyzer window. You can also double-click the item displayed as <new...> to create an analysis for PDN. The **Power Distribution Network Extraction** window is displayed, as shown in the following figure:



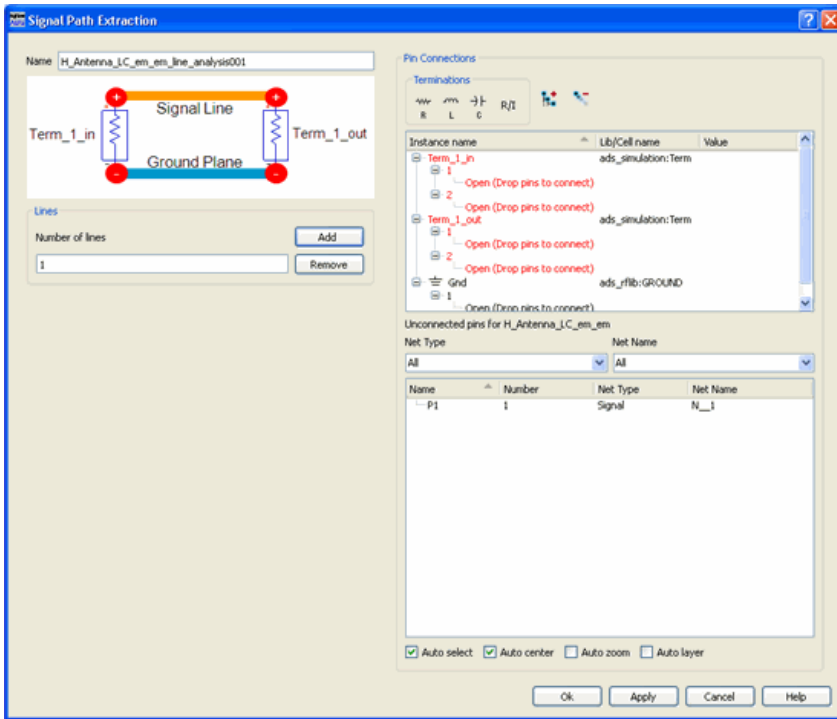
In the Power Distribution Network Extraction window, you can specify:

- On or off capability of calculating currents flowing in the PDN including SMDs (if any).
- Models for Voltage Regulator Module (VRM) and Load.
- Connections between VRM/Load models and DUT.

Editing the PDN Analysis Setup

Using the Power Distribution Network Extraction window, you can perform the following tasks:

- **Calculating current distribution:** You can calculate current distribution by selecting the **Calculate current distribution** check box.
- **Selecting a VRM and Load:** In the VRM or Load frame, you can:
 - Select a source or load model: Click **Browse** to select a source or load model to use. Note that only frequency domain model can be used.
 - Restore models: Click **Default** to restore a default model.
 - Add a new source or load: Click **Add** to include a new source or load into the analysis.
 - Delete a source or load: Click **Remove** to delete a source or load from the analysis.
- **Measuring Impedance:** To calculate the impedance of a power distribution network, you need to monitor the current that flows a load and voltage between power pin and ground pin. You need to provide the following values in **Impedance Measurement**:
 - Load value: Specify a load model which you want to make measurement of your impedance in the **Load** field.
 - Power pin value: Specify a pin number which should connect to the power nets in the **Power pin** field.
 - Ground pin value: Specify a pin number which should connect to the ground nets in the **Ground pin** field.
- **Specifying Pin connections:** You can specify the following connections between your model that is being analyzed and a source or load in **Pin Connections**:
 - You can terminate unconnected pins in a model by using the icons in the Termination section. You can also drag and drop unconnected pins to end a connection.
 - You can view instances of source/load and termination are shown with these



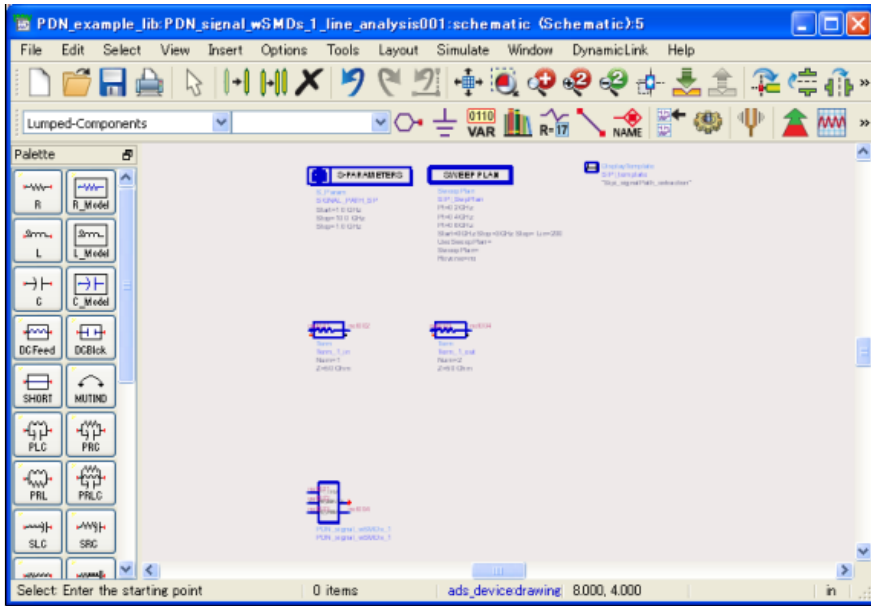
The Signal Path Extraction window allows you to specify the:

- Number of paths you want to analyze.
- Connections between S-parameter terms and your DUT.

Editing the Signal Path Analysis

You can perform the following tasks for the Signal Path Extraction window:

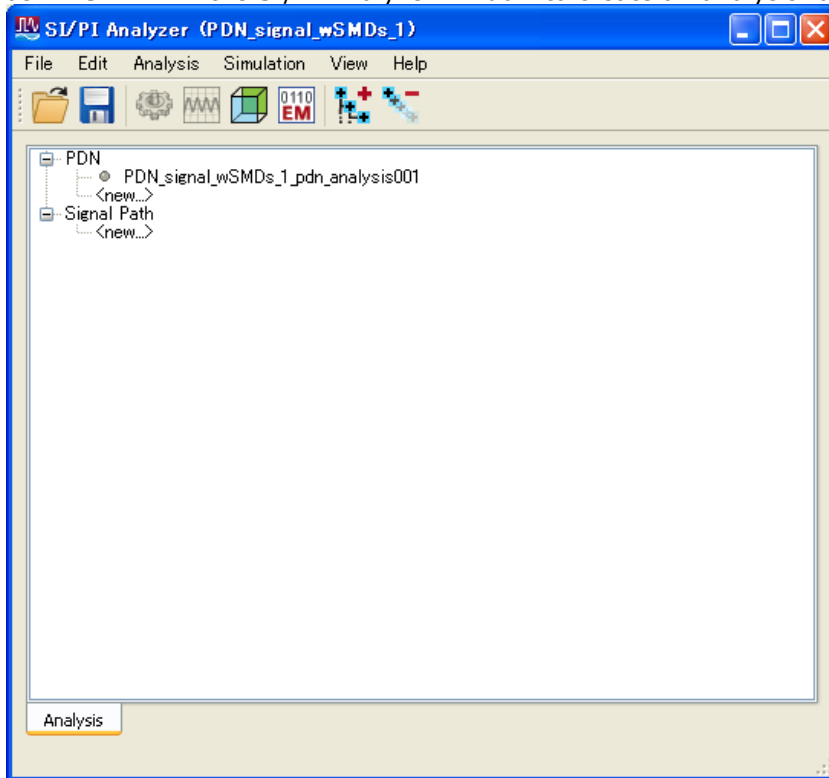
- **Specifying Lines:** You can specify number of lines (paths) to analyze. Click **Add** or **Remove** to increase or decrease the number of lines, respectively.
- **Specifying pin connections:** You can specify the following connections between your model that is being analyzed and S-parameter terms in **Pin Connections**:
 - Terminate unconnected pins in a model by using the icons in the Termination section. You can also drag and drop unconnected pins to end a connection.
 - View instances of S-parameter terms and termination are shown with these pins in the top pane.
 - View unconnected pins of your model that are analyzed in the bottom pane.
 - Generate a connection between pins of your model and S-parameter terms by using the drag and drop operation.



Creating a Signal Path Analysis


To analyze a signal path:


1. Select **Analysis > New analysis > Signal Path** or double-click the item displayed as **<new...>** in the SI/PI Analyzer window to create an analysis for signal path:



2. Type the required number of lines or click **Add** in the **Lines** frame.
3. In **Pin Connections**, generate a connection between pins of your model and source/loads by dragging pins from the unconnected pins section and drop on the required pin in the upper pane.
4. Click **OK** to create test bench schematic for the PDN analysis.

Performing Analysis

After creating an analysis, you can view it in the Analysis tree. By default, an analysis appears with a gray icon  next to the analysis name, which means that the analysis has

not been performed yet. If the analysis is successful, this icon is green  in color.


To perform an analysis:

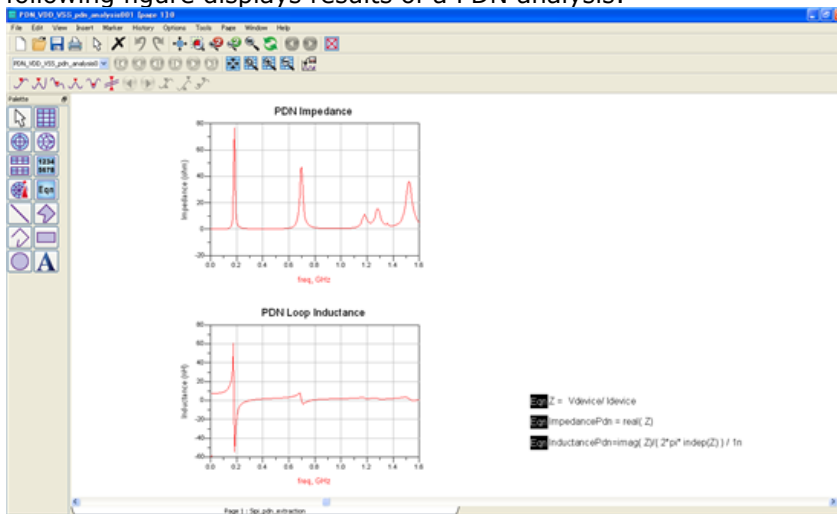
1. Select the required analysis for performing simulations.
2. Choose **Simulation > Run** or click the **Analysis** button to initiate simulations. Circuit simulations including EM co-simulation are performed sequentially.

Viewing results

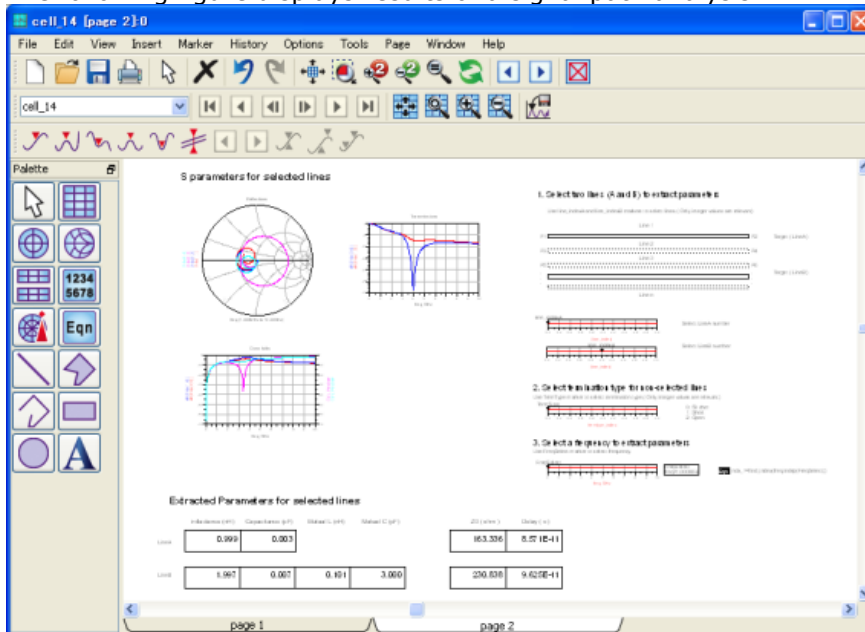
After successfully completing your analysis, you can view the analysis result in the Data Display window. You can also view the current distribution in the 3D Visualizer, if current distribution calculation is specified. However, this feature is applicable for a PDN analysis only.

To view analysis results in the Data Display window:

1. Select an analysis with green icon in the SI/PI Analyzer window.
2. Click **Data Display** () to display results in the **Data Display** window. The following figure displays results of a PDN analysis:



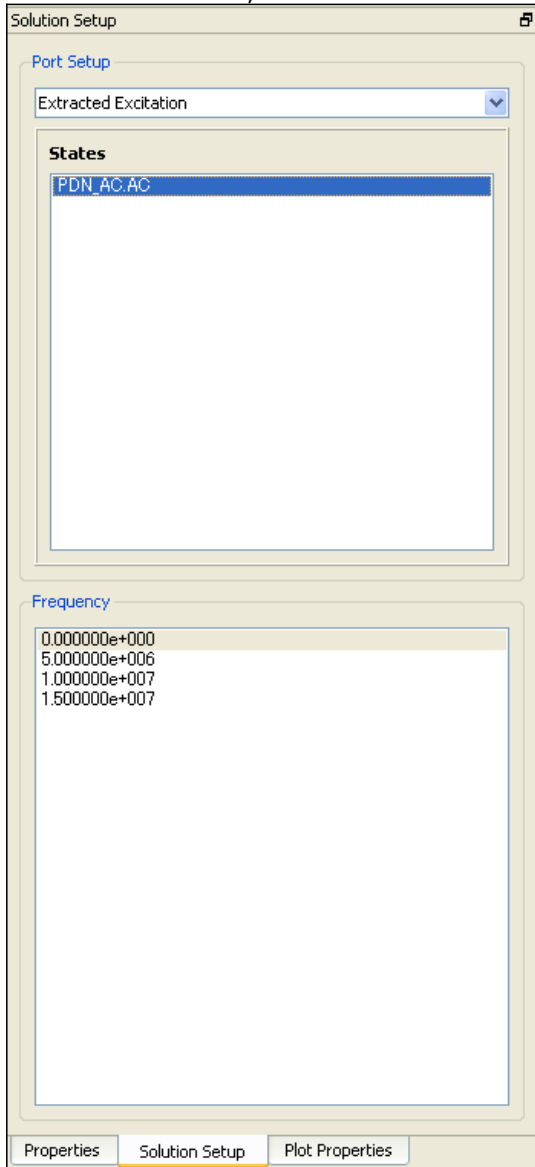
The following figure displays results of a signal path analysis:



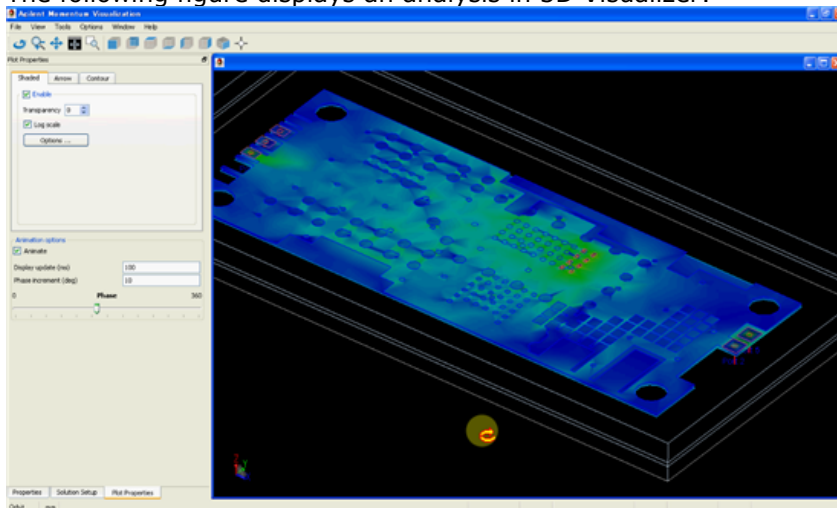
To view the current distribution in the 3D Visualizer (PDN analysis only):

1. Select a PDN analysis with green icon.

2. Select **Show Current Distribution** menu in the context menu.
3. In the 3D visualizer, select the **Solution Setup** tab.



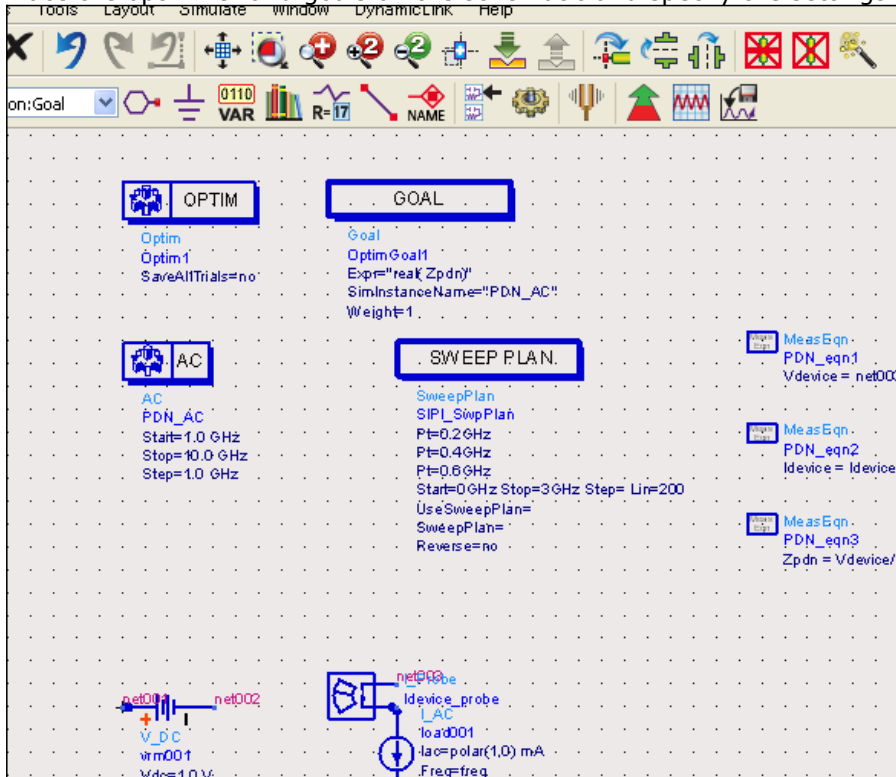
4. Select **Extracted Excitation** in the top drop-down combo box.
5. Select a state in the **States** list.
6. Select a frequency that you want to display.
7. The following figure displays an analysis in 3D Visualizer:



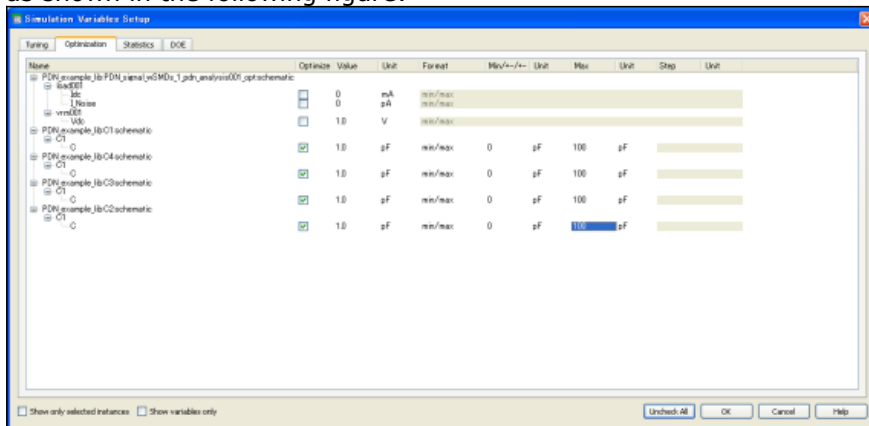
Example

The following example describes how to optimize decoupling capacitor values. It also describes how to view the current distribution after the optimization.

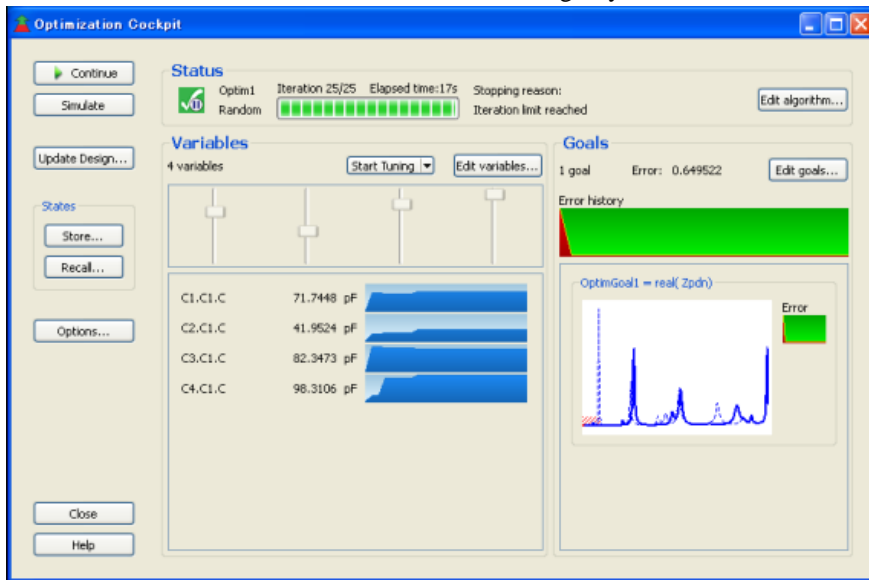
1. Create a PDN analysis and setup it correctly.
2. Generate a test bench schematic from the above analysis.
3. Save the test bench schematic with a different name by selecting **File > Save as** in the schematic window.
4. Place the optimizer and goals on the schematic and specify the settings accordingly.



5. Setup parameters of decoupling capacitor for optimization. Select **Simulate > Simulation Variable Setup** to open the **Simulation Variables Setup** dialog box, as shown in the following figure:



6. Click the **Optimize** icon to optimize the decoupling capacitors and update optimized values to the schematic when optimization is finished.



7. Open the SI/PI analyzer and copy the original PDN analysis with a different name and edit it to make sure **Calculate current distribution** is selected.
8. Perform a simulation for the new pdn analysis.(This pdn analysis will be done with the optimized decoupling capacitor values)
9. Select **Show Current Distribution** menu in the context menu to show current distribution with optimized decoupling capacitors.

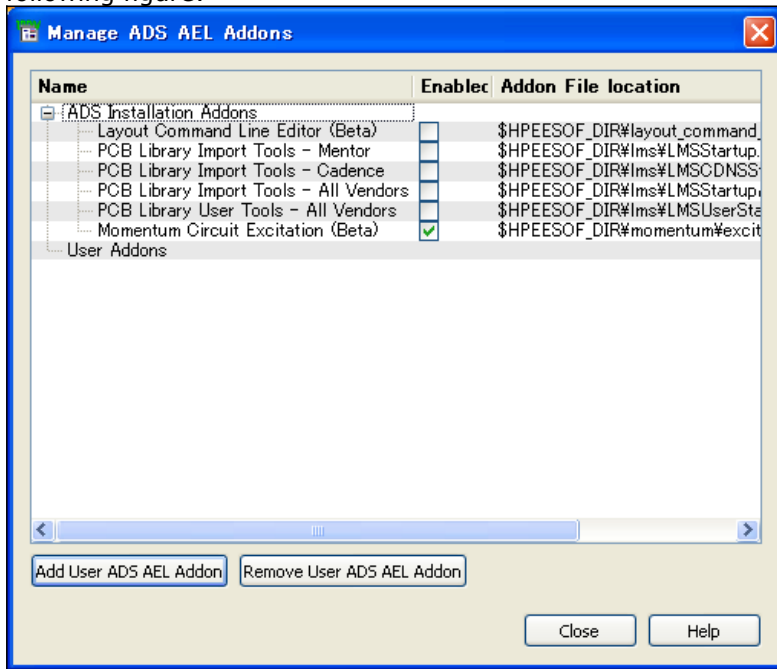
Performing Momentum Circuit Excitation

You can use the Momentum Circuit Excitation feature in ADS 2011.01 to view surface currents and far fields of your EM Model with the SMDs and/or any other circuit elements of your circuit schematic. This feature is available for the AC or Harmonic Balance circuit simulations.

Enabling Momentum Circuit Excitation AEL

To enable the Momentum Circuit Excitation AEL add-on:

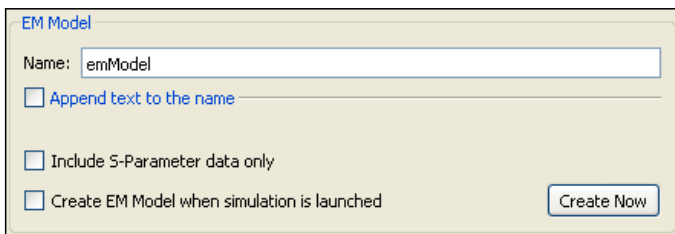
1. Select **Tools > Manage ADS AEL Addons** in the ADS Main window.
2. Select the **Enabled** check box for **Momentum Circuit Excitation**, as shown in the following figure:



3. Click **Close**.

Creating EM Models

You need to create an EM Model for each EM setup. While creating the EM model, do not select the **Include S-Parameter data only** check box. This allows the EM Model to include not only the S-Parameter data, but also the required data for an EM visualization. If you select this check box, instances with such setup will not be picked up by subsequent processes.



Note
Momentum Circuit Excitation currently supports only **Momentum RF** and **Momentum Microwave** as simulators for EM Models.

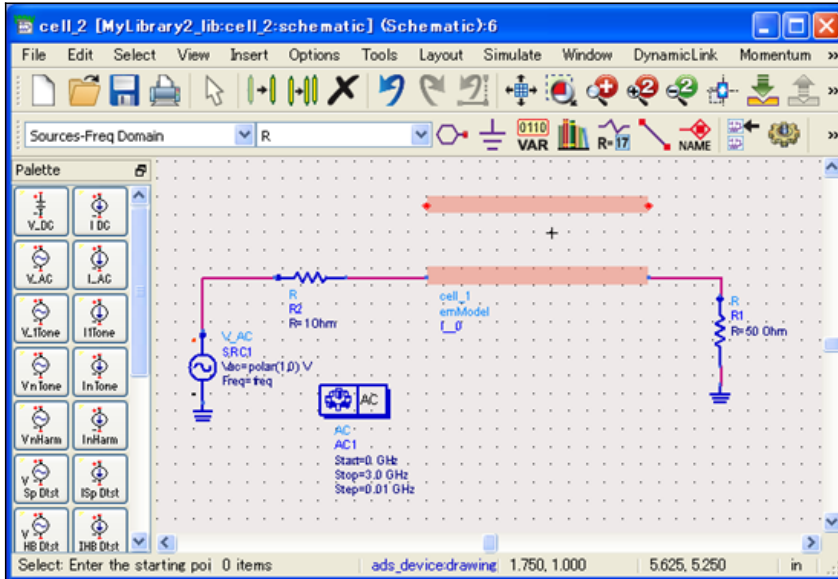
For more information, refer *Generating EM Model and Symbol (em)*.

Note

Frequencies calculated by Momentum can only be shown in the EM visualization. If you want to see surface currents and/or far field at explicit frequencies, add such frequencies in your EM Setup.

Creating Circuits for AC/HB

Momentum Circuit Excitation works for circuits with single or multiple EM Models and a AC/Harmonic Balance simulation controller. Parameter sweep is also available. The following figure displays an example of a circuit with AC simulation controller:



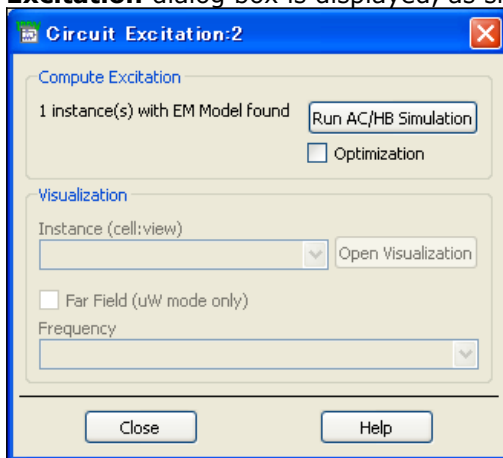
For creating circuits for AC/HB, choose an EM Model view explicitly for the required instances. Only such instances will be available for EM Visualization. The excitation voltages for each instances for EM Visualization will be interpolated or extrapolated based on available frequencies of the AC/Harmonic Balance simulation.

Note

Currently AC and Harmonic balance simulation are supported.

Running AC/HB Simulation

1. Select **Momentum > Circuit Excitation** in the schematic window. The **Circuit Excitation** dialog box is displayed, as shown in the following figure:



2. Click **Run AC/HB Simulation** to start circuit simulation and post processing.

**Note**

If you have included an optimization controller in your circuit and want to run optimization, select the **Optimization** check box. However, if you have selected **Enable Optimization Cockpit** in the optimization controller, optimization flow may not work properly from this tool. To avoid this issue, it is recommended to disable this option. In the schematic, double-click Optimization controller, select the **Parameters** tab, and remove selection from **Enable Optimization Cockpit**.

3. A message box stating that **Compute excitation is all finished** is displayed.

Viewing Results

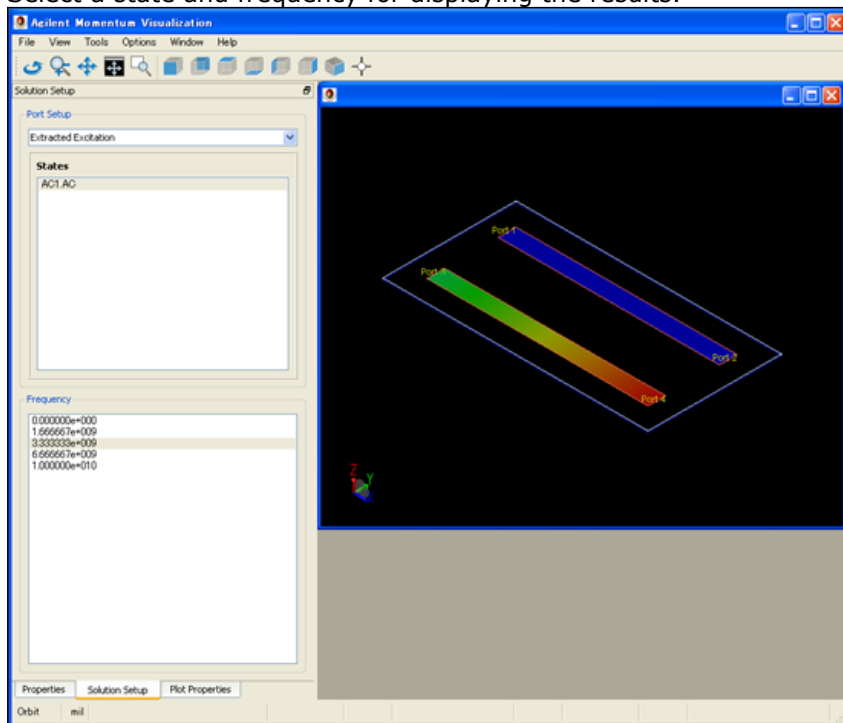
To view surface currents:

1. Select an instance from the **Instance** drop-down list.
2. Click **Open Visualization**.
3. In the EM Visualization window, select the **Solution Setup** tab.
4. Select **Extracted Excitation**.

**Note**

It is recommended to reset the maximum amplitude for plots after selecting **Extracted Excitation** because it has largely different maximum amplitude than **Single Mode Excitation**. To reset maximum amplitude for each plots type, click **Options > Restore System Max/Min**.

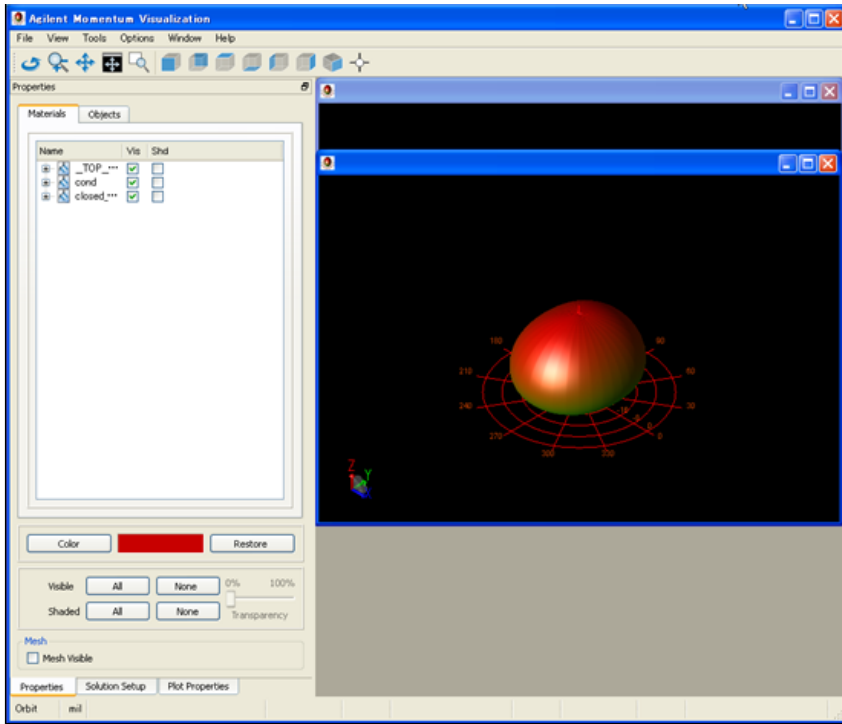
5. Select a state and frequency for displaying the results.

**Note**

If you swept parameters by parameter sweep, each of these simulation becomes a **State**.

To view far fields:

1. Select an instance in the **Instance** drop-down combo box.
2. Select the **Far Field (uW mode only)** check box and select a frequency.
3. Click **Open Visualization** to view results, as shown in the following figure:



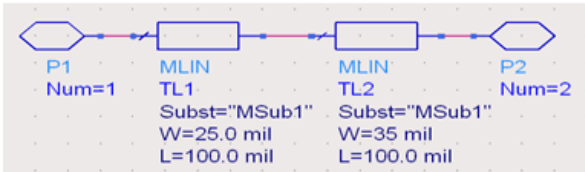
Designing a Microstrip Line

In this section, you will be introduced to the concept of setting up EM simulation for a Microstrip Line design. You will use default settings and a simple circuit to create a Microstrip Line design with step in width.

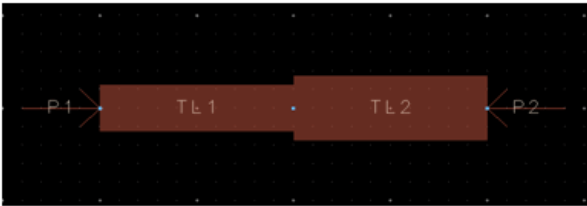
For designing a Microstrip Line component, you will perform the following tasks:

- Draw a simple Microstrip Line design with step in width as a schematic.
- Generate a corresponding layout.
- Create a simple substrate.
- Define a mesh.
- Perform an EM simulation using both Momentum and FEM.
- Analyze the results.

The schematic representation of the Microstrip Line design is displayed below:



The layout representation of the Microstrip Line design is displayed below:



Drawing the Circuit

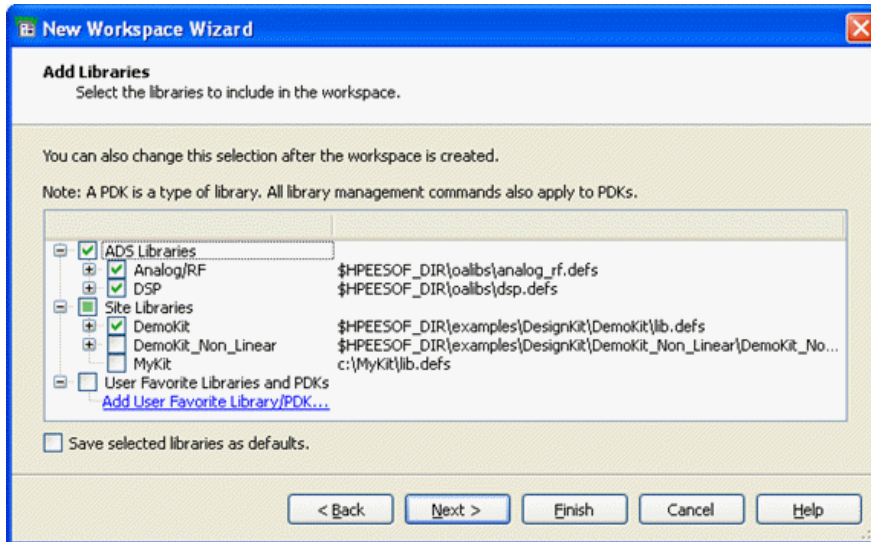
To start the process of designing a Microstrip Line component, you need to draw the circuit by:

- Creating a new workspace.
- Adding Microstrip components to the schematic.
- Converting the schematic to a layout.

Creating a New Workspace

To create a new workspace:

1. From the ADS Main window, choose **File > New > Workspace** to display **New Workspace Wizard**.
2. Click **Next**.
3. In the **Workspace Name** field, type **step1**.
4. Enter a path in **Create In**. Click **Browse** to select the location.
5. Click **Next**.
6. Under **Add Libraries**, select the libraries to be included in the Workspace, as shown in the following figure:



7. Click **Next**.
8. Under **Library Name**, enter a unique library name.
9. Click **Next**.
10. Under **Technology**, choose **ADS Standard: length unit - mil**.
11. Click **Finish**.

For more information about a workspace, refer *Using Workspace* (adstour).

Adding Microstrip Line components to the Schematic

To create a new schematic:

1. Choose **File > New > Schematic**.
2. Create a new schematic step_1 in library step_1_lib.

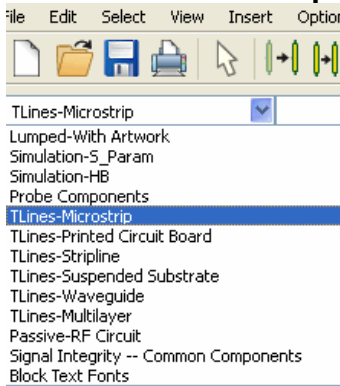
For more information about a schematic, refer to *Creating a New Schematic* (adstour).

After creating a schematic, select a component and place it in the Schematic window.

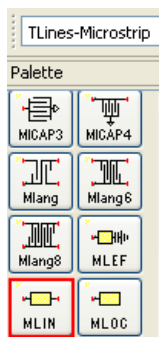
Placing Components in a Schematic

To select and place components in a Schematic window:

1. Click the **Palette** drop-down list.
2. Select **TLines-Microstrip** from the list.




3. Click **MLin** in the Microstrip Transmission Lines palette.



4. 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.
5. Move the cursor to place the crosshairs directly on the right pin of the first component, and then click once to place the second component.

Canceling Commands

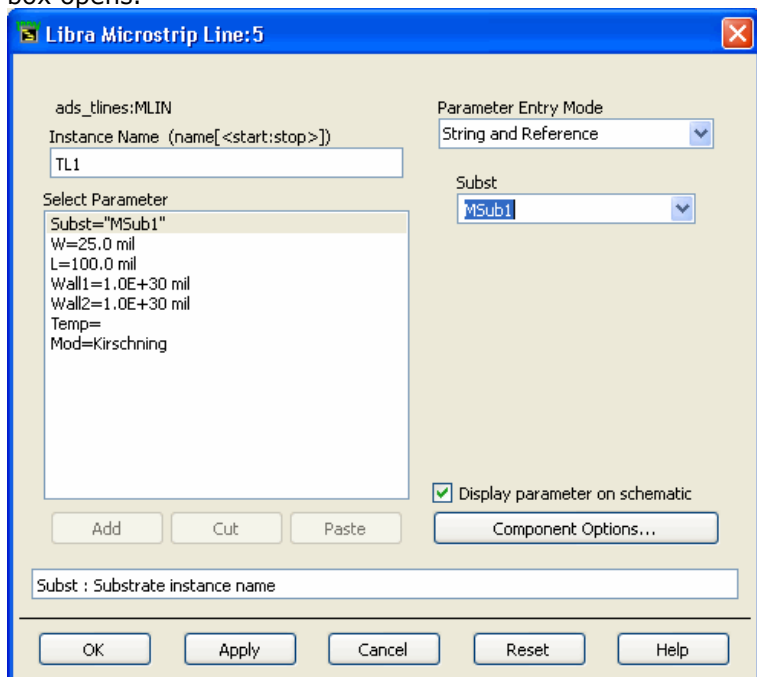
If you continue to click without ending the current command, you will add another component with each click. To cancel a command, click the arrow button (). You can also end a command by pressing the **Escape** key.

Editing Component Parameters

This section describes how to change the width of one of the strips to create a Microstrip Line step in width transmission line.

To edit components:

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** to specify the width. When the field to the right shows the value of the width, change the value in this field to 35 mil.
3. Click **Apply**.
4. Click **OK**.



Note



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

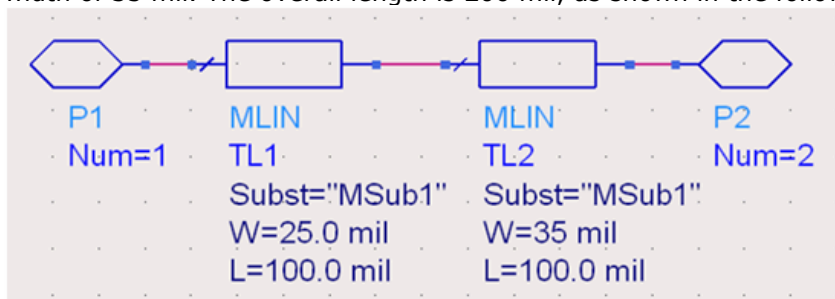
5. Verify that the width of component on the left is set to 25 mil.

Adding Pins to the Circuit

To complete the circuit, you must add pins, one at the beginning of the Microstrip Line component step in width and one at the end. This section describes how to add pins to the circuit.

To add pins:

1. Click **Pin** () in the menu bar.
2. Move the cursor over the Schematic window and note the orientation of the ghost icon of the pin.
3. Click **Rotate** to rotate the port to the necessary orientation.
4. Move the cursor over the open pin on the left side of the left component, then click once.
5. 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 once.
6. Click  to cancel the current command. Your schematic displays 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, as shown in the following figure:



Note
Diamond-shaped pins indicate that pins are not connected. You need to select and move components to form complete connections.

Saving a Design

To save your design, choose **File > Save**.

Generating a Layout

Since EM simulations require a circuit in the layout format, it gives you the option of drawing your circuits either as schematics or as layouts.

Note
If you 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.

To convert the Microstrip Line design with step in width schematic:

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**. The Layout Generation message appears indicating that the conversion is complete.
3. Click **OK**.
4. A Layout window appears, which displays 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.
5. From the Layout window, choose **File > Save**. You now have a layout and a schematic as part of your library step_1.

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 connecting traces on different layers
- Air surrounding the circuit board

In ADS 2011.01, substrates are part of the Technology dialog box used for a library. For the Microstrip Line design having step in width example, a substrate having the following layers is used:

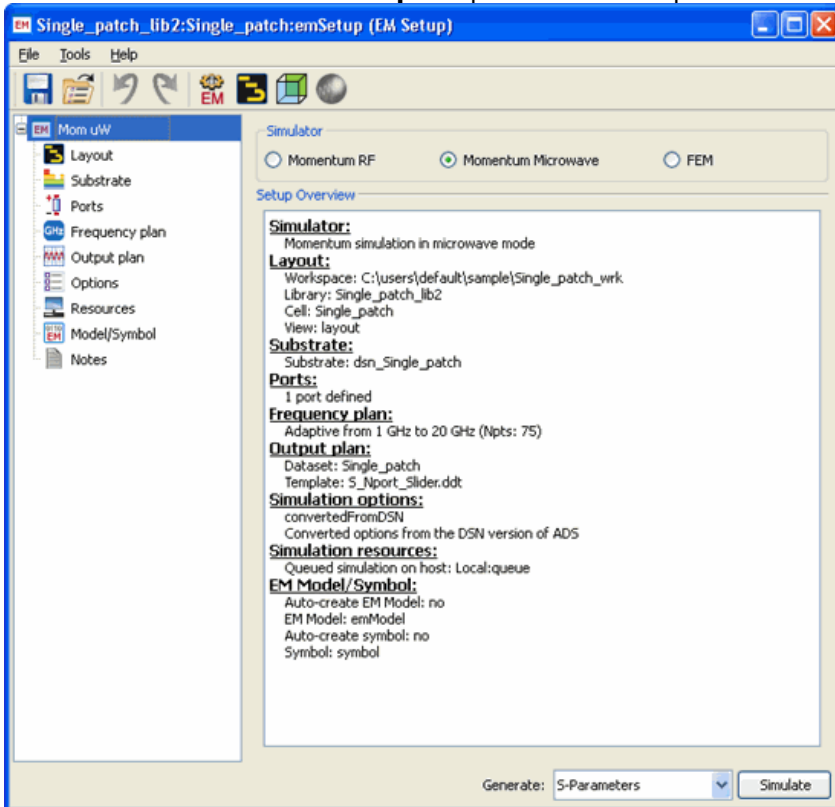
- Ground plane
- Dielectric(insulation), such as Alumina
- Metal layer for the Microstrip Line component
- Air layer above the Microstrip component

Defining the EM Setup

In the EM Setup window, you can specify settings for running a successful EM simulation. If you do not provide correct information, an invalid symbol () appears against the options in the left pane of the EM Setup window.

To specify the EM simulation settings:

1. Choose **EM > Simulation Setup** to open the EM Setup window.



2. Select **Substrate** in the left pane of the EM Setup window.
3. Select **Frequency Plan** and add the frequency range for simulation. By default, an adaptive frequency plan from 0 GHz to 10 GHz is added. The EM setup becomes valid on adding the frequency plan. For more information, refer to *Defining the Frequency and Output Plan (em)*.

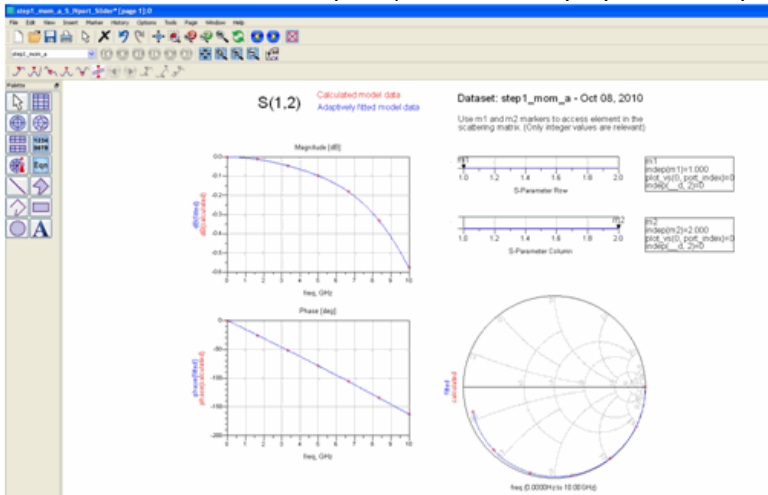
Note

At this stage, if you want to start the simulation, click the **Simulate** button at the bottom of EM Setup window.

4. Select **Layout** to gather information about the Layout for which this EM setup is

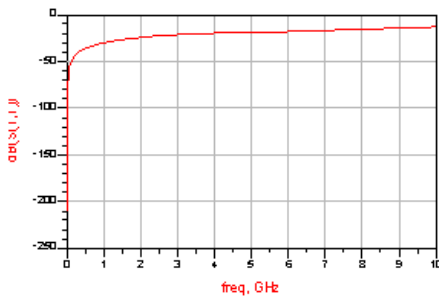
prepared. For more information, refer to *Viewing Layout Information* (em).

5. Select **Substrate** to display the substrate used for the simulation. For more information, refer to *Assigning Substrate* (em).
6. Select **Ports** to gather information about the ports being generated from the pins defined in the layout. EM simulations need ports to inject energy into system. For more information, refer to *Generating Ports* (em).
7. Select **Output plan** to specify a name for the EM dataset.
8. Select **Options** to specify mesh, preprocessor, and solver options. For more information, refer to *Defining Simulation Options* (em).
9. Select **Resources** to specify computer resources options. For more information, refer to *Specifying Simulation Resources* (em).
10. Select **Model/Symbol** to generate an EM model and symbol. For more information, refer to *Generating EM Model and Symbol* (em).
11. Select **Note** to add any specific note about the simulation.
12. Save the EM settings by selecting **File > Save**.
13. Click **Simulate** in the EM Setup window to start the EEsof Job manager and EEsof Job log window for the currently running momentum simulation.
14. After the simulation is complete, the Data Display window opens automatically:



Viewing Results

Load S11 plot on the Data Display window, as shown in the following figure:

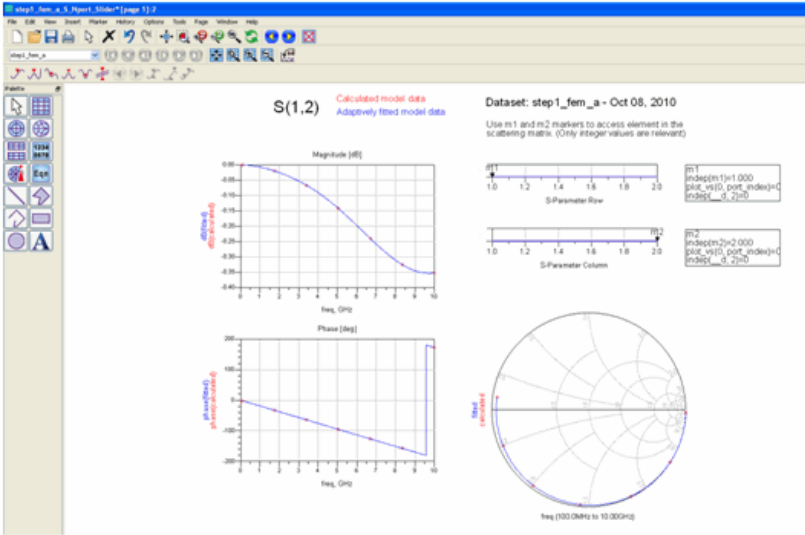


Switching Simulation Engines

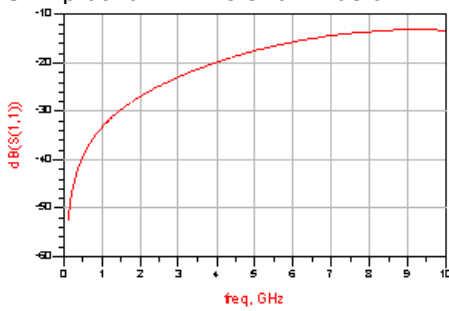
For changing the mode of simulation to FEM, open the **EM Setup** window and select **FEM**. Once the mode is changed to FEM, the EM setup remains valid for the FEM simulator.

Note
Selecting the FEM simulator changes the dataset name in Output plan to **step1_fem**.

Clicking **Simulate** in the EM Setup window launches the EEsof Log window for the FEM simulation. After completion of the FEM simulation, a data display window is displayed:



S11 plot for FEM is shown below:



Comparing Momentum and FEM Results

Load the plots from both Momentum and FEM and compare the results. Comparison plots are shown below:

