

Advanced Design System 2017

Electromagnetic



Notices

© Keysight Technologies Incorporated, 2002-2017

1400 Fountaingrove Pkwy., Santa Rosa, CA 95403-1738, United States

All rights reserved.

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 Keysight Technologies, Inc. as governed by United States and international copyright laws.

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause.

Use, duplication or disclosure of Software is subject to Keysight Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Portions of this software are licensed by third parties including open source terms and conditions. For detail information on third party licenses, see Notice.

Contents

EM Simulation Overview	10
EM Simulation Overview	10
Getting Started with an EM Simulation	10
Process for Simulating a Design	11
Momentum	13
Momentum	13
Contents	13
Momentum Overview	13
Momentum Overview	13
Theory of Operation for Momentum	16
Theory of Operation for Momentum	16
Conductor Loss Models in Momentum	35
Setting up Momentum Simulations	44
Setting up Momentum Simulations	44
Examples- Setting up Momentum Simulations	49
Examples- Setting up Momentum Simulations	49
Defining Momentum Settings for LPF	50
Designing a Coplanar Waveguide Bend	61
HSD Applications: Guidelines for Specifying EM Settings	70
HSD Applications: Guidelines for Specifying EM Settings	70
FEM	77
FEM	77
Contents	77
FEM Overview	//
	//
Theory of Operation for FEM	/9
Theory of Operation for FEM	/9
Setting up FEM Simulations	100
Setting up FEM Simulations	100
Example- Designing a Microstrip Filter	103 100
Example- Designing a Microstrip Filter	IU3 110
Guidelines For Optimal Performance of FEM Simulation	IIU 110
Derellel And Distributed Simulations	IIU 111
Memory usage	110.
Cotting Started with a Layout for EM Simulations	∠
Gotting Started with a Layout for EM Simulations	נוו . 112
Creating a Layout for EM Simulations	ווט 112
Creating a Layout for EM Simulations	נוו . 110
Using the Cookie Cutter	110 101
Using the Cookie Cutter	101 .
Defining Component Parameters	יבי . 100
	120

Defining Component Parameters	128
Converting Layers between Strip and Slot Representation	134
Converting Layers between Strip and Slot Representation	134
Example- Designing a Microstrip Line	136
Example- Designing a Microstrip Line	136
Substrates	146
Substrates	146
Defining Substrates	146
Defining Substrates	146
Sweeping Substrate Name and Mesh Density	169
Sweeping Substrate Name and Mesh Density	169
Adding Boxes, Waveguides, and Symmetry Planes	171
Adding Boxes, Waveguides, and Symmetry Planes	171
Modeling Through-Silicon Vias	178
Modeling Through-Silicon Vias (TSV)	178
Variables in Substrates and Materials	183
Variables in Substrates and Materials	183
Overview of Using Variables in Substrates	183
Defining Variables in a Defaults Design	184
Using Variables in Substrates and Materials	188
Simulation Flows	194
Update EM Model Database when Parameter Set is Modified	203
Dielectric Substrate Modeling	208
Dielectric Substrate Modeling	208
Generating Thermal Technology Files Using Substrate Editor	211
Generating Thermal Technology Files Using Substrate Editor	211
Ports	214
Ports	214
Ports Overview	214
Ports Overview	214
Defining Ports	217
Defining Ports	217
Ports for Momentum	233
Ports for Momentum	233
Ports for Momentum Overview	233
Applying TML Ports Calibration in Momentum	237
Applying SMD Port Calibration in Momentum	246
Applying Delta Gap Port Calibration in Momentum	250
	252
	252
Setting up EM Simulations	250
Setting up ENI Simulations	200
EIVI Setup Window Overview	200 257
Viewing Layout Information	70/ 70/
Viewing Layout Information	203
viewing Layout information	203

Automatic EM Circuit Part	itioning	. 264
Automatic EM Circu	it Partitioning	. 265
Viewing the Substrate	- 	. 272
Viewing the Substra	ite	. 272
Viewing Ports		. 272
Viewing Ports		. 273
Defining a Frequency Plar	۱	. 276
Defining a Frequence	cy Plan	. 276
Defining Simulation Optio	ns	. 278
Defining Simulation	Options	. 278
Specifying Physical	Model Settings for Momentum	. 280
	Lumped and wire via model options	. 283
Specifying Physical	Model Settings for FEM	. 287
	Substrate LATERAL Extension	. 287
	Substrate VERTICAL Extension	. 288
	Substrate Wall Boundary	. 288
	Merge Adjacent Layers with same Material Propertie	S
	Option	. 288
Defining Preprocess	or Settings	. 293
	Local Via Array	. 299
	Stacked Conductors	. 300
	Global	. 300
	Layer Specific	. 301
	Global	. 303
	Layer Specific	. 304
Defining Mesh Setti	ngs for Momentum	305
Defining FEM Mesh	Settings	319
	Conductor Mesh	. 324
	Number of Bondwire Segments	. 325
	Global Vertex, Edge, and Surface Mesh	. 325
	Vertex Mesh	. 326
	Edge Mesh	. 326
	Surface Mesh	. 327
Guidelines for Speci	ifying FEM Mesh Settings	329
Performing FEM Bro	badband Refinement	. 331
Defining Solver Sett	ings	335
Defining Expert Opt	ions for Momentum	. 340
Defining an Output Plan		. 341
Defining an Output	Plan	. 342
Specifying Simulation Res	sources	. 345
Specifying Simulation	on Resources	. 346
Generating an EM Model		. 355
Generating an EM N	10del	355
Using EM Setup Window	100ls	358
Using EM Setup Wil	naow Iools	358
Creating an SnP Scl	nematic	362

Using an EM Setup Template	363
Managing EM Setup Templates	363
Finding EM Setup Templates	366
Deprecated	369
Managing EM Simulations	371
Managing EM Simulations	371
Running EM Simulations	371
Running EM Simulations	371
Using the Job Manager	374
Using the Job Manager	374
Since Windows 10	386
Before Windows 10	386
Viewing Simulation Summary	391
Viewing Simulation Summary	391
Running a Momentum simulation from Command Line	397
Running a Momentum simulation from Command Line	397
Viewing Simulation Results in 3D	401
Viewing Simulations Results in 3D	401
Starting the 3D Visualization	401
Visualizing 3D View before EM Simulations	402
Visualizing 3D View before EM Simulation	402
Visualizing Momentum Simulations	410
Visualizing Momentum Simulations	410
Visualizing FEM Simulations	415
Visualizing FEM Simulations	415
Computing Radiation Patterns	431
Computing Radiation Patterns	431
EM Circuit Cosimulation	447
EM Circuit Cosimulation	447
Contents	447
EM Circuit Cosimulation Overview	447
EM Circuit Cosimulation Overview	447
Using the EM Cosimulation View	447
Using the EM Cosimulation View	448
EM Cosimulation View	448
Parameterized Layout View	452
Circuit Simulated Component	453
Design	453
Resulting EM Simulated Partition	454
Performing an EM Circuit Cosimulation with an EM Cosimulation View	
458	
EM Cosimulation Controller Component	466
Using the EM Model View	468
Using an EM Model View	468
Creating and Using an EM Model View	468
Performing an EM Circuit Cosimulation with an EM Model View	471

Show me How to Perform EM Circuit Cosimulation	. 476
The EM Model View	. 476
Using the EM Circuit Excitation AEL Addon	. 482
Using the EM Circuit Excitation AEL Addon	. 482
Manual versus Automatic EM/Circuit Partitioning	. 489
The manual approach in ADS 2013.06 or before	. 490
The automatic approach in ADS 2014.01	. 491
EMPro and ADS Integration	. 497
EMPro and ADS Integration	. 497
Contents	. 497
EMPro and ADS Integration Process	. 497
EMPro and ADS Integration Process	. 497
Saving EMPro Designs in a Library	. 500
Saving EMPro Designs in a Library	. 500
Creating Designs in EMPro	. 503
Adding an EMPro Library in ADS	. 506
Adding an EMPro Library in ADS	. 507
Adding EMPro Components in ADS Layout	. 511
Adding EMPro Components in ADS Layout	. 511
Using EMPro Components in ADS Schematic	. 516
Using EMPro Components in ADS Schematic	. 516
Adding EMPro Components to Schematic	. 516
Exporting EMPro Simulation Results as emModel	. 519
Exporting ADS Layouts to EMPro	. 524
Exporting ADS Layouts to EMPro	. 524
EMPro ADS Integration FAQs	. 525
EMPro ADS Integration FAQs	. 525
Example- Using the EMPro Connector Design in ADS	. 533
Example- Using the EMPro Connector Design in ADS	. 533
Using the EMPro FEM Engine in ADS	. 538
Using the EMPro FEM Engine in ADS	. 539
Package Model Extraction	. 540
Package Model Extraction	. 540
Enabling the Package Model Extraction Add-on	. 540
Using the Package Model Extraction Tool	. 540
Example	. 545
CollSys	. 548
CollSys	. 548
Supported Platforms	. 548
	. 548
	. 548
Collonunt Parameter	. 556
Uolizhad Parameter	. 55/
Using Package in Virtuoso Flow	. 228
Draview Levente	. 559
Fleview Layouts	. 004

Preview Layouts 5	564
Displaying the 3D Viewer Window	564
Viewing the Components of a Layout Design	565
Customizing the View 5	566
Layout Display Settings 5	567
Customizing the Layout View 5	567
Connectivity Tool 5	569

EM Simulation Overview

EM Simulation Overview

ADS provides EM simulation tools for designing and evaluating modern communications systems products. It provides a unified interface for Momentum and FEM simulators. These are electromagnetic simulators that compute Sparameters, surface currents, and fields for general planar circuits, including microstrip, stripline, coplanar waveguide, and other topologies.

The EM simulation use model 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.
 - Ensures easier, faster, and accessible EM simulations.

Getting Started with an EM Simulation

The 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 to define frequency and output plan, configure engine options, generate ports and EM model. For more information, see Setting up EM Simulations.
- 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.
- Stopping a simulation: To end a simulation before it is finished, choose EM> Stop and Release Simulator. You can use this option to 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, farfields, antenna parameters, and transmission line data by choosing EM> Post-Processing> Visualization. For more information, refer to Visualizing 3D View before EM Simulations.

NOTE The SIO file format (replacement of CITI file format) is a compact internal representation of the S-parameters as generated by the EM simulators (Momentum/FEM).

- 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.
- Defining radiation patterns: You can compute the electromagnetic fields by choosing EM> Post-Processing> Far Field. For more information, refer to Computing Radiation Patterns.
- 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

Related Videos

You might find the following online video useful in understanding the topics covered on this page:

ADS Momentum and FEM Interface

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. ADS can import files in various types of formats.
 For information on converting schematics or drawing in Layout, see Schematic Capture and Layout and Creating a Layout for EM Simulations. For information on importing designs, see Importing and Exporting Designs.
- 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, see Selecting an EM Simulator.
- 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, see Defining Substrates.
- Assigning port properties: Ports enable you to inject energy into a circuit 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 Ports.
- 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 a Frequency Plan.
- Setting Simulation Options: You set up a simulation by specifying the parameters of preprocessor, substrates, and mesh. You can specify various mesh parameters to customize the mesh to your design. For more information, refer to Defining Simulation Options.
- 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 an EM setup file from one library to another library, all the information is not copied. For example, you must define a substrate in the new library. Similarly, if a layout does not have pins referenced in the original layout, these pins are not visible and all the ports are invalid.

Momentum

Momentum

Momentum is a complete 3D planar EM tool set to predict the performance of high-frequency circuit boards, antennas, ICs, and modules. It identifies parasitic coupling between components and goes beyond simple analysis and verification to design automation of passive distributed circuits. It also visualizes current flow and 3D display of far-field radiation.

Momentum Simulator comes with the following list of features:

- 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
- Installs with ADS and runs locally on the desktop, on a remote server, or on a distributed compute cluster
- State-of-the-art meshing and solving technologies provide breakthrough capacity, speed, and accuracy
- Seamless EM/circuit co-simulation and co-optimization
- Complete feature list for the Momentum G2 Element

Contents

- Momentum Overview
- Theory of Operation for Momentum
- Setting up Momentum Simulations
- Examples- Setting up Momentum Simulations
 - Designing a Coplanar Waveguide Bend
 - Defining Momentum Settings for LPF

Momentum Overview

Momentum Overview

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 that enable you to simulate multilayer RF/microwave printed circuit boards, hybrids, multichip modules, and integrated circuits. Momentum provides a complete tool set to predict the performance of high-frequency circuit boards, antennas, and ICs.

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

Momentum enables you to:

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

Key features of Momentum include:

- An electromagnetic simulator based on the Method of Moments.
- Adaptive frequency sampling for fast, accurate, simulation results.
- Optimization tools that alter geometric dimensions of a design to achieve performance specifications.
- Comprehensive data display tools for viewing results.
- Equation and expression capability for performing calculations on simulated data.
- Full integration in the ADS circuit simulation environment allowing EM /Circuit co-simulation and co-optimization.

Related Videos

You might find the following online video useful in understanding Momentum:

Momentum Overview

Momentum Simulation Modes

You can use Momentum RF and Momentum Microwave simulation modes. 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. Typical RF applications include RF components and circuits on chips, modules, and boards, as well as digital and analog RF interconnects and packages.

When compared to the Momentum Microwave mode, the Momentum RF mode uses new technologies enabling it to simulate physical designs at RF frequencies with several useful benefits. The RF mode is based on quasi-static electromagnetic functions enabling faster simulation of designs. Momentum RF has the same usemodel as Momentum in ADS, and works with Momentum Visualization and Optimization. Choose the mode that matches the application

Momentum RF is usually the more efficient mode when a circuit

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

For descriptions about electrically small and geometrically complex circuits, see Matching the Simulation Mode to Circuit Characteristics.

NOTE For infinite ground planes with a loss conductivity specification, the MW mode of Momentum incorporates the HF losses in ground planes, however, the RF mode of Momentum will make an abstraction of these HF losses.

Locating Momentum Examples

You can refer the examples of designs that are simulated using Momentum RF and Momentum Microwave in the Examples directory.

To open a Momentum design example:

- 1. Choose File > Open > Example from the ADS Main window. The *Select an Archived File* dialog box is displayed.
- 2. Select Momentum. A list of examples is displayed, as shown in the following figure:

😋 🗢 🚽 « exam	ples 🕨	Momentum 🕨	- 4→ Se	earch Momentum	٩
Organize 🔻 New f	older			!≡ ▼	
 Downloads Dropbox Recent Places Libraries Documents Music My Documents Pictures Videos 		Antenna Antenna emcktcosim HighSpeedDigital Microwave ModelComposer Optimization RF		Date modified 8/21/2012 12:49 PM 8/21/2012 12:49 PM 8/21/2012 12:49 PM 8/21/2012 12:49 PM 8/21/2012 12:49 PM 8/21/2012 12:49 PM 8/21/2012 12:49 PM	Select a file to preview.
Fi	▼ •		▼ Zip	▶ ped Archive Files (*.72 Open ►	ads *.7 ▼ ancel

- **3.** Select the required example directory. For example, you can select Antenna, ModelComposer, or Microwave.
- 4. Click Open. A list of all examples present in the selected example category is displayed.
- 5. Select the required example.

6. Click Open.

Theory of Operation for Momentum

Theory of Operation for Momentum

Momentum is based on a numerical discretization technique called the *method of moments*. This technique is used to solve Maxwell's electromagnetic equations for planar structures embedded in a multilayered dielectric substrate. The simulation modes available in Momentum (microwave and RF) are both based on this technique, but use different variations of the same technology to achieve their results.

Momentum has two modes of operation, Microwave or full-wave mode and the RF or quasi-static mode. The main difference between these two modes lies within the Green functions formulations that are used. The full-wave mode uses full-wave Green functions, these are general frequency dependent Green functions that fully characterize the substrate without making any simplification to the Maxwell equations. This results in L and C elements that are complex and frequency dependent. The quasi-static mode uses frequency independent Green functions resulting in L and C elements that are real and frequency independent. Because of the approximation made in the guasi-static mode, the RF simulations run a lot faster since the matrix L and C elements only have to be calculated for the first frequency simulation point. The approximation also implies that the quasi-static mode typically should be used for structures that are smaller than half the wavelength. Both engine modes are using the so-called star-loop basis function, ensuring a stable solution at all frequencies. Both engines also make use of a mesh reduction algorithm which reduces the number of unknowns in the simulation by generating a polygonal mesh. This mesh reduction algorithm can be turned on or off, with a toggle switch.

The sources applied at the ports of the circuit yield the excitations in the equivalent network model. The currents in the equivalent network are the unknown amplitudes of the rooftop expansion functions. Solving the equivalent network for a number of independent excitation states yields the unknown current amplitudes. A port calibration process is used to calculate the S-parameter data of the circuit from the current solution, when calibration is requested.

The following sections contain more information about:

- The method of moments technology
- The Momentum solution process
- Special simulation topics
- Considerations and software limitations

The Method of Moments Technology

The method of moments (MoM) technique is based upon the work of R.F. Harrington, an electrical engineer who worked extensively on the method and applied it to electromagnetic field problems, in the beginning of the 1960's. It is based on older theory which uses weighted residuals and variational calculus. More detailed information on the method of moments and Green's theorem refer to, *Field Computation by Moment Methods* Reference 1).

In the method of moments, prior to the discretization, Maxwell's electromagnetic equations are transformed into integral equations. These follow from the definition of suitable electric and magnetic Green's functions in the multilayered substrate.

In Momentum, a mixed potential integral equation (MPIE) formulation is used. This formulation expresses the electric and magnetic field as a combination of a vector and a scalar potential. The unknowns are the electric and magnetic surface currents flowing in the planar circuit.

Using notations from linear algebra, we can write the mixed potential integral equation in very general form as a linear integral operator equation:

$$\iint dS \overline{\overline{G}}(r,r') \cdot J(r) = E(r) \tag{1}$$

Here, J(r) represents the unknown surface currents and E(r) the known excitation of the problem. The Green's dyadic of the layered medium acts as the integral kernel. The unknown surface currents are discretized by meshing the planar metalization patterns and applying an expansion in a finite number of sub-sectional basis functions B1(r), ..., BN(r):

$$J(\mathbf{r}) \approx \sum_{j=1}^{N} I_j B_j(\mathbf{r})$$
(2)

The standard basis functions used in planar EM simulators are the subsectional rooftop functions defined over the rectangular, triangular, and polygonal cells in the mesh. Each rooftop is associated with one edge of the mesh and represents a current with constant density flowing through that edge as shown in the following illustration. The unknown amplitudes Ij, j=1,...,N of the basis function expansion determine the currents flowing through all edges of the mesh.



Discretization of the surface currents using rooftop basis functions.

The integral equation (1) is discretized by inserting the rooftop expansion (2) of the currents. By applying the Galerkin testing procedure, that is, by testing the integral equation using test functions identical to the basis functions, the continuous integral equation (1) is transformed into a discrete matrix equation:

for i=1,...,N

$$\sum_{i=1}^{N} Z_{i,j} I_j = V_i \quad \text{or} \quad [Z] \cdot [I] = [V] \quad (3)$$
with

$$Z_{i,j} = \iint_{S} dS \boldsymbol{B}_i(\boldsymbol{r}) \cdot \iint_{S'} dS' \overline{\boldsymbol{G}}(\boldsymbol{r}, \boldsymbol{r}') \cdot \boldsymbol{B}_j(\boldsymbol{r}) \quad (4)$$

$$V_i = \iint_{S} dS \boldsymbol{B}_i(\boldsymbol{r}) \cdot \boldsymbol{E}(\boldsymbol{r}) \quad (5)$$

The left hand side matrix [Z] is called the interaction matrix, as each element in this matrix describes the electromagnetic interaction between two rooftop basis functions. The dimension N of [Z] is equal to the number of basis functions. The right-hand side vector [V] represents the discretized contribution of the excitations applied at the ports of the circuit.

The surface currents contribute to the electromagnetic field in the circuit by means of the Green's dyadic of the layer stack. In the MPIE formulation, this Green's dyadic is decomposed into a contribution from the vector potential A(r) and a contribution from the scalar potential V(r):

$$\overline{\overline{G}}(\boldsymbol{r},\boldsymbol{r}') = j\omega G^{A}(\boldsymbol{r},\boldsymbol{r}')\overline{\overline{I}} - \frac{1}{j\omega}\nabla[G^{V}(\boldsymbol{r},\boldsymbol{r}')\nabla']$$
⁽⁶⁾

The scalar potential originates from the dynamic surface charge distribution derived from the surface currents and is related to the vector potential through the Lorentz gauge.

By substituting the expression (6) for the Green's dyadic in the expression (4) for the interaction matrix elements, yields the following form:

$$Z_{i,j} = j\omega L_{i,j} + \frac{1}{j\omega C_{i,j}}$$
(7)

with

$$\begin{split} L_{i,j} &= \iint_{S} \mathrm{d} S \boldsymbol{B}_{i}(\boldsymbol{r}) \cdot \iint_{S'} \mathrm{d} S' \boldsymbol{G}^{A}(\boldsymbol{r},\boldsymbol{r}') \boldsymbol{B}_{j}(\boldsymbol{r}') \\ & \frac{1}{C_{i,j}} = \iint_{S} \mathrm{d} S \nabla \cdot \boldsymbol{B}_{i}(\boldsymbol{r}) \iint_{S'} \mathrm{d} S' \boldsymbol{G}^{V}(\boldsymbol{r},\boldsymbol{r}') \nabla \cdot \boldsymbol{B}_{j}(\boldsymbol{r}') \end{split}$$
(9)

This allows the interaction matrix equation to be given a physical interpretation by constructing an equivalent network model. In this network, the nodes correspond to the cells in the mesh and hold the cell charges. Each cell corresponds to a capacitor to the ground. All nodes are connected with branches which carry the current flowing through the edges of the cells. Each branch has in inductor representing the magnetic self coupling of the associated current basis function. The equivalent circuit is built by replacing each cell in the mesh with a capacitor to the ground reference and inductors to the neighboring cells, as shown in the following figure:



All capacitors and inductors in the network are complex, frequency dependent and mutually coupled, as all basis functions interact electrically and magnetically. The ground in this equivalent network corresponds with the potential at the infinite metallization layers taken up in the layer stack. In the absence of infinite metallization layers, the ground corresponds with the sphere at infinity. The method of moments interaction matrix equation follows from applying the Kirchoff voltage laws in the equivalent network. The currents in the network follow from the solution of the matrix equation and represent the amplitudes of the basis functions. Equivalent network representation of the discretized MoM problem are displayed in the following figure:



The Momentum Solution Process

Different steps and technologies enable the Momentum solution process:

- Calculation of the substrate Green's functions
- Meshing of the planar signal layer patterns
- Loading and solving of the MoM interaction matrix equation
- Calibration and de-embedding of the S-parameters
- Reduced Order Modeling by Adaptive Frequency Sampling

Calculation of the Substrate Green's Functions

The substrate Green's functions are the spatial impulse responses of the substrate to Dirac type excitations. They are calculated for each pair of signal (strip, slot and /or via) layers mapped to a substrate level. Although it is necessary to know which signal layers are mapped to a substrate level, since only impulse responses are being calculated, it is not necessary to know the patterns on these signal layers. This implies that the Green's functions can be pre-calculated and stored in a substrate database. This allows the substrate Green's functions to be reused for other circuits defined on the same substrate.

The high frequency electromagnetic Green's functions depend upon the radial distance and the frequency. The computations are performed up to very large radial distances over the entire frequency band specified by the user. The frequency points are selected adaptively to ensure an accurate interpolation with respect to frequency. Computations performed over very wide frequency ranges can consume more CPU time and disk space to store the results. To increase speed, the RF mode uses quasi-static electromagnetic Green's functions based on low-frequency approximation and scales the quasi-static Green functions at higher frequencies.

Meshing of the Planar Signal Layer Patterns

The planar metallization (strip, via) and aperture (slot) patterns defined on the signal layers are meshed with rectangular and triangular cells in the microwave simulation mode. As translational invariance can be used to speed up the interaction matrix load process, the meshing algorithm will maximize the number of uniform rectangular cells created in the mesh. The meshing algorithm is very flexible as different parameters can be set by the user (number of cells/wavelength, number of cells/width, edge meshing, and mesh seeding), resulting in a mesh with different density. It is clear that the mesh density has a high impact on both the efficiency and accuracy of the simulation results. Default mesh parameters are provided which give the best accuracy/efficiency trade-off. Both the RF and microwave modes use mesh reduction technology to combine rectangular and triangular cells to produce a mesh of polygonal cells, thus reducing demand for computer resources. Mesh reduction eliminates small rectangles and triangles which from an electrical modeling point of view only complicate the simulation process without adding accuracy. As mentioned previously, this feature may be turned on or off with a toggle switch.

Loading and Solving of the MoM Interaction Matrix Equation

The loading step of the solution process consists of the computation of all the electromagnetic interactions between the basis functions and the filling of the interaction matrix and the excitation vector. The interaction matrix as defined in the rooftop basis is a dense matrix, that is, each rooftop function interacts with every other rooftop function. This electromagnetic interaction between two basis functions can either be strong or weak, depending on their relative position and their length scale. The matrix filling process is essentially a process of order (N²), (i. e. the computation time goes up with the square of the number of unknowns).

In the solving step, the interaction matrix equation is solved for the unknown current expansion coefficients. The solution yields the amplitudes of the rooftop basis functions which span the surface current in the planar circuit. Once the currents are known, the field problem is solved because all physical quantities can be expressed in terms of the currents. With the release 2005A, an iterative matrix solve scheme was introduced in Momentum. For large problem sizes, the iterative matrix solve performs as an order (N2) process. Momentum still uses a direct matrix solve process if the structure is small or when convergence problems are detected in the iterative matrix solver process.

Calibration and De-embedding of the S-parameters

Momentum performs a calibration process on the single type port, the same as any accurate measurement system, to eliminate the effect of the sources connected to the transmission line ports in the S-parameter results. Feedlines of finite length (typically half a wavelength at high frequencies, short lines are used at low frequencies) are added to the transmission line ports of the circuit. Lumped sources are connected to the far end of the feedlines. These sources excite the eigenmodi of the transmission lines without interfering with the circuit. The effect of the feedlines is computed by the simulation of a calibration standard and subsequently removed from the S-parameter data. A built-in cross section solver calculates the characteristic impedance and propagation constant of the transmission lines. This allows to shift the phase reference planes of the S-parameters, a process called deembedding. Results of the calibration process includes the elimination of low-order mode mismatches at the port boundary, elimination of high-order modes, and removal of all port excitation parasitics.

Besides transmission line ports, Momentum offers the user the ability to define direct excitation or internal ports. These ports can be specified at any location on the planar metallization patterns as either a point or a line feed. They allow to connect both passive and active lumped components to the distributed model of the planar circuits. The S-parameters associated with these ports are calculated from the excitation consisting of a lumped source connected to the equivalent network model at the locations of the internal ports. The S-parameters results.

Reduced Order Modeling by Adaptive Frequency Sampling

A key element to providing fast, highly accurate solutions using a minimum of computer resources is the Adaptive Frequency Sampling (AFS) technology. When simulating over a large frequency range, oversampling and straight line interpolation can be used to obtain smooth curves for the S-parameters. Oversampling however implies a huge amount of wasted resources. Momentum allows the user to benefit from a smart interpolation scheme based on reduced order modeling techniques to generate a rational pole/zero model for the Sparameter data. The Adaptive Frequency Sampling algorithm selects the frequency samples automatically and interpolates the data using the adaptively constructed rational pole/zero model. This feature allows important details to be modeled by sampling the response of the structure more densely where the S-parameters are changing significantly. It minimizes the total number of samples needed and maximizes the information provided by each new sample. In fact, all kinds of structure can take advantage of the AFS module. The Adaptive Frequency Sampling technology reduces the computation time needed for simulating large frequency ranges with Momentum significantly.

Special Simulation Topics

Some special simulation topics are discussed in this section:

- Simulating slots in ground planes

- Simulating metallization loss
- Simulating internal ports and ground references

Simulating Slots in Ground Planes

Slots in ground planes are treated in a special manner by Momentum. An electromagnetic theorem called the equivalence principle is applied. Instead of attempting to simulate the flow of electric current in the wide extent of the ground plane, only the electric field in the slot is considered. This electric field is modeled as an equivalent magnetic current that flows in the slot.

Momentum does not model finite ground plane metallization thickness. Ground planes and their losses are part of the substrate definition.

By using slot metallization definitions, entire structures, such as slot lines and coplanar waveguide circuits, can be built. Slots in ground planes can also be used to simulate aperture coupling through ground planes for multi-level circuits. Structure components on opposite sides of a ground plane are isolated from each other, except for intermediate coupling that occurs through the slots. This treatment of slots allows Momentum to simulate slot-based circuits and aperture coupling very efficiently.

Simulating Metallization Loss

When using Momentum, losses in the metallization patterns can be included in the simulation. Momentum can either treat the conductors as having zero thickness or include the effects of finite thickness in the simulation. In the substrate definition, the expansion of conductors to a finite thickness can be turned on/off for every layer. Conductors with arbitrary height/width ratio can be accurately modeled with Momentum.

Momentum uses a complex surface impedance model for all metals that is a function of conductor thickness, conductivity, and frequency. At low frequencies, current flow will be approximately uniformly distributed across the thickness of the metal. At high frequencies, the current flow flows dominantly on the outside of the conductor and Momentum uses a complex surface impedance that closely models this skin effect.

For more information on the surface impedance model can be found here.

The mesh density can affect the simulated behavior of a structure. A denser mesh allows a better finer distribution of the current flow what can slightly increase the loss. This is because a more uniform distribution of current at a low density mesh corresponds to a lower resistance.

Losses can be defined for ground planes defined in the substrate definition. This uses the same formulation as for loss in microstrips (i.e., through a surface impedance approximation). It should be noted however that since the ground planes in the substrate description are defined as infinite in size, only HF losses are incorporated effectively. DC losses are zero by definition in any infinite ground plane. DC metallization losses in ground planes can only be taken into account by simulating a finite size ground plane as a strip metallization level.

For infinite ground planes with a loss conductivity specification, the MW mode of Momentum incorporates the HF losses in ground planes, however, the RF mode of Momentum will make an abstraction of these HF losses.

Simulating with Internal Ports and Ground References

NOTE

Momentum offers the ability to use internal ports within a structure. Internal ports can be specified at any location on the planar metallization patterns, and they make possible a connection for both passive and active lumped components to the distributed model of the planar circuits. The following figure displays an internal port and equivalent network model.



The S-parameters associated with these ports are calculated from the excitation consisting of a lumped voltage source connected to the equivalent network model. The ground reference for these ports in the resulting S-parameter model is the ground of the equivalent network, and this ground corresponds *physically* to the infinite metallization layers taken up in the layer stack. In the absence of infinite metallization layers, the ground no longer has a physical meaning and corresponds *mathematically* with the sphere at infinity.

It is important to mention that in this case, the associated S-parameters also lose their physical meaning, as the applied voltage source is assumed to be lumped, that is electrically small, since it sustains a current flow from the ground to the circuit without phase delay. To overcome this problem, a ground reference must be specified at a distant electrically small from the internal port. Failure to do so may yield erroneous simulation results.

The following sections illustrate the use of internal ports with ground planes and with ground references, and the results.

Internal Ports and Ground Planes in a PCB Structure

The following figure shows the layout for a PCB island structure with two internal ports.



The infinite ground plane is taken up in the substrate layer stack and provides the ground reference for the internal ports. The magnitude and phase of the S21-parameter calculated with Momentum. The simulation results are validated by comparing them with the measured data for the magnitude and the phase of S21. The following figure displays the Magnitude and phase of S21:



The thicker line is Momentum results, the thinner line is the measurement.

Finite Ground Plane, no Ground Ports

The same structure was resimulated with a finite groundplane. The substrate layer stack contains no infinite metallization layers. No ground reference was specified for the ports, so the sphere at infinity acts as the ground reference for the internal ports. The S-parameters obtained from a simulation for these ports no longer have a physical meaning. The following figure displays PCB substrate layer stack and metallization layers:



Using such ports in the simulation yields incorrect simulation results as shown in the following figure:



The thicker line is Momentum results, the thinner line is the measurement.

Finite Ground Plane, Internal Ports in the Ground Plane

One way to define the proper grounding for the internal ports is to add two extra internal ports in the ground plane, as shown in the following figure:



four internal ports

The resulting four-port structure is simulated with Momentum. Note that because no ground reference is specified for the internal ports, the resulting S-parameters for the four port structure have no physical meaning.

However, by properly recombining the four ports, a two-port structure is obtained with the correct ground references for each of the ports. Care should be taken with this recombination, as the ground reference for the four individual ports does not act as the ground reference for the two recombined ports. Therefore, the top recombination scheme is incorrect. The correct recombination scheme is the bottom one shown in the following figure:



Finite Ground Plane, Ground References

The process of adding two extra internal ports in the ground plane and recombining the ports in the correct way can be automated in Momentum by defining two ground reference ports, as shown in the following figure:



The S-parameters obtained with Momentum are identical to the S-parameters of the two-port after the correct recombination in the section above, also shown in the following figure:



The thicker line is Momentum results, the thinner line is the measurement.

Internal Ports with a CPW Structure

The following figures show the layout for a CPW step in width structure with an integrated sheet resistor (45 μ m x 50 μ m, 50 $^{\Omega}$).



The following figure display the CPW Step-in-width structure with integrated sheet resistor, port configurations and results:



Six internal ports are added to the metallization layers. The six-port S-parameter model is recombined into a two-port model to describe the CPW-mode S-parameters. Different recombination schemes are possible, yielding different simulation results. Scheme (c) is the correct one, as the ground reference for the internal ports is the sphere at infinity and this cannot act as the ground reference for the CPW ports.

The process of adding extra internal ports to the ground metallization patterns and recombining the ports in the correct way can be automated in Momentum by defining ground reference ports, as shown in the following figure:



Limitations and Considerations

This section describes some software limitations and physical considerations which need to be taken into account when using Momentum:

- Comparing the microwave and RF simulation modes
- Matching the simulation mode to circuit characteristics
- Higher-order modes and high frequency limitation
- Substrate waves and substrate thickness limitation
- Parallel-plate modes and high frequency limitation
- Slotline structures and high frequency limitation
- Via structures and metallization thickness limitation
- Via structures and substrate thickness limitation
- CPU time and memory requirements

Comparing the Microwave and RF Simulation Modes

Momentum has two simulation modes: the Microwave and RF mode. The Microwave mode uses full-wave formulation, the RF mode uses a quasi-static formulation. In the quasi-static formulation, the Green functions are low-frequency approximations to the full-wave and more general Green functions. Because of the approximations made in the RF mode, the simulations are more efficient. The approximation is valid for structures that are small compared to the wavelength (size of the circuit smaller than half a wavelength).

NOTE For infinite ground planes with a loss conductivity specification, the MW mode of Momentum incorporates the HF losses in ground planes, however, the RF mode of Momentum will make an abstraction of these HF losses.

Matching the Simulation Mode to Circuit Characteristics

The Momentum RF mode can be used to simulate RF and microwave circuits, depending on your requirements. However, Momentum RF is usually the more efficient mode when a circuit is electrically small, geometrically complex, and does not radiate. This section describes these characteristics.

Radiation

Momentum RF provides accurate electromagnetic simulation performance at RF frequencies. However, this upper limit depends on the size of your physical design. At higher frequencies, as radiation effects increase, the accuracy of the Momentum RF models declines smoothly with increased frequency.

Similarly, if the substrate allows the propagation of surface waves (these are guided waves that propagate in the substrate layers) the accuracy of the RF mode will gradually decline because surface waves are not included in the RF calculation.

Electrically Small Circuits

Momentum RF works best for electrically small circuits as its accuracy smoothly decreases with increasing electrical size relative to a given frequency. A circuit is considered electrically small relative to a given frequency if its physical dimension is smaller than half the wavelength of the frequency. Depending on which value you know, maximum circuit dimension or maximum simulation frequency, you can determine a qualitative approximation of the circuit's electrical size.

For space wave radiation, you can use one of the following two expressions strictly as a guideline to have an awareness about the circuit's electrical size relative to the maximum frequency you plan to run the simulation. When you know the value of D, use the first expression to approximate the maximum frequency up to which the circuit is electrically small. When you know the maximum simulation frequency, use the second expression to approximate the maximum allowable dimension:

$$F < \frac{150}{D}$$

or

$$D < \frac{150}{F}$$

where:

D = the maximum length in mm diagonally across the circuit.

F = the maximum frequency in GHz.

The following expression provides a guideline up to which frequency the substrate is electrically small for surface wave radiation.



During a simulation, Momentum RF calculates the maximum frequency up to which the circuit is considered electrically small, and displays that value in the status display. This is similar to using the expressions above since the dimension and thickness of a layout is typically fixed, and it is the simulation frequency that is swept.

Geometrically Complex Circuits

The mesh generated for a simulation establishes a geometric complexity for a circuit. A circuit is considered geometrically complex if its shape does not fit into a uniform, rectangular mesh, and the mesh generation produces a lot of triangles. Layouts containing shapes such as circles, arcs, and non-rectangular polygons usually result in meshes with many triangles. A measure of increasing geometric complexity is when the ratio of triangular cells to rectangular cells grows larger:

Complexity = Triangles Rectangles

The mesh-reduction technology offered by Momentum RF and Microwave eliminates electromagnetically redundant rectangular and triangular cells. This reduces the time required to complete the simulation thus increasing efficiency. This mesh reduction algorithm can be turned on or off, using a toggle switch.

Higher-order Modes and High Frequency Limitation

Since Momentum does not account for higher-order modes in the calibration and de-embedding process, the highest frequency for which the calibrated and deembedded S-parameters are valid is determined by the cutoff frequency of the port transmission line higher-order modes. As a rule of thumb for microstrip transmission lines, the cutoff frequency (in GHz) for the first higher-order mode is approximately calculated by: Cutoff frequency fc = 0.4 ZO / height where ZO is the characteristic impedance of the transmission line. For a 10 mil alumina substrate with 50 ohm microstrip transmission line, we obtain a high frequency limit of approximately fc = 80 GHz.

Parallel Plate Modes

In the region between two infinite parallel plates or ground planes, parallel plate modes exist. Any current flowing in the circuit will excite all of these modes. How strong a mode will be excited depends on how well the field generated by the current source matches with the field distribution of the parallel plate mode.

A distinction can be made between the fundamental mode and the higher order modes. The fundamental mode, which has no cut-off frequency, propagates at any frequency. The higher order modes do have a cut-off frequency. Below this frequency, they decay exponentially and only influence the local (reactive) field around the source. Above this frequency, they propagate as well and may take real energy away from the source.

Momentum takes the effect of the fundamental mode into account. The higher order modes are taken into account as long as they are well below cut-off. If this is not the case, a warning is issued saying that higher-order parallel plate modes were detected close to their cut-off frequency. Simulation results will start to degrade from then on.

The Effect of Parallel Plate Modes

The fundamental mode has its electric field predominantly aligned along the Z-axis from the top plate to the bottom plate. This means that this mode creates a difference in the potential between both plates. This mode can be excited by any current in your circuit: electric currents on a strip or via, magnetic currents on a slot. It behaves as a cylindrical wave that propagates to infinity where it feels a short circuit (since both plates are connected there). However, since infinity is very far away, no reflected wave ever comes back.

In a symmetric strip line structure, the current on the strip won't excite the fundamental mode due to this symmetry. However, the presence of a feature such as a slot in one of the plates creates an asymmetry. The slot will excite the fundamental mode!

The accuracy of the Momentum results for calibrated ports degrades when the parallel plate modes become important. For calibrated ports is assumed that they excite the circuit via the fundamental transmission line mode. However, this excitation is not pure. The source will excite the circuit via the parallel plate modes, too. This effect cannot be calibrated out, since those contributions are not orthogonal. The consequence is that the excitation doesn't correspond with a pure fundamental transmission line mode excitation.

Avoiding Parallel Plate Modes

You can short circuit the parallel plate modes by adding vias to your structure at those places where parallel plate modes will be excited. Similar problems will exist with the measurements. Both plates must be connected explicitly with each other! This won't take place at infinity but somewhere at the side of the board. If no vias

are used in the real structure, the fundamental mode may be excited. This mode will propagate to the borders and reflect. Thus, the real structure differs in that from the simulations where no reflection will be seen since a reflection only happens at infinity.

One way to counter the difference between measurements in reality and simulation is to take into account the package (box) around the structure. Another possibility is adding vias to your structure, both in real life and in the simulations. Try to make the physical representation of your circuit as close as possible to reality. However, the actual shape of the via should not be that important. Small sheets can be used to represent the small circular vias that are realized in the real structure.

Surface Wave Modes

If the substrate is not closed (open or half open), and not homogeneous, surface wave modes can exist. The parallel plate mode can be seen as a special case of a surface wave mode. Their behavior is identical. Both are cylindrical waves that propagate radially away from the source. They are *guided* by the substrate. Both fundamental and higher order surface wave modes exist. Similar conclusions can be drawn with respect to limitations such as the effects of the modes.

Slotline Structures and High Frequency Limitation

The surface wave accuracy deterioration of the calibration process is somewhat more prominent for slotline transmission lines than for microstrip transmission lines. Therefore, slotline structures simulated with Momentum will exhibit a somewhat higher noise floor (10-20 dB higher) than microstrip structures. For best results, the highest simulation frequency for slotline structures should satisfy:

Substrate thickness < 0.15 * effective wavelength

Simulation of narrow slot lines are accurate at frequencies somewhat higher than the high frequency limit determined by this inequality, but wide slotlines will show deteriorated accuracy at this frequency limit.

Via Structures and Substrate Thickness Limitation

The vertical electrical currents on via structures are modeled with rooftop basis functions. In this modeling, the vertical via structure is treated as one cell. This places an upper limit to the substrate layer thickness as the cell dimensions should not exceed 1/20 of a wavelength for accurate simulation results. Momentum simulations with via structures passing through electrically thick substrate layers will become less accurate at higher frequencies. By splitting the thick substrate layer into more than one layer, more via-cells are created and a more accurate solution is obtained.

CPU Time and Memory Requirements

Both CPU time and memory needed for a Momentum simulation increase with the complexity of the circuit and the density of the mesh. The size N of the interaction matrix equation is equal to the number of edges in the mesh. For calibrated ports, the number of unknowns is increased with the edges in the feedlines added to the transmission line ports.

CPU Time

The CPU time requirements for a Momentum simulation can be expressed as:

CPU time =
$$A + B N + C N^2 + D N^3$$

where:

N = number of unknowns

A, B, C, D = constants independent of N

The constant term A accounts for the simulation set up time. The meshing of the structure is responsible for the linear term, BN. The loading of the interaction matrix is responsible for the quadratic term and the solving of the matrix equation accounts for:

- part of the quadratic term (when using the iterative solver)
- the cubic term (when using the direct solver)

It is difficult to predict the value of the constants A, B, C and D, because they depend on the problem at hand.

Memory Usage

The memory requirement for a Momentum simulation can be expressed as:

 $Memory = X + Y N + Z N^2$

where

N = number of unknowns

X, Y, Z constants independent of N

Like with the CPU time expression, the constants X, Y and Z are difficult to predict for any given structure.

For medium to large size problems, the quadratic term, which accounts for storing of the interaction matrix, always dominates the overall memory requirement. For small structures, memory usage can also be dependent upon the substrate. The substrate database must be read and interpolated, which requires a certain amount of memory. Algorithms to make trade-off between time and memory resources are implemented in the simulator. These algorithms result in additional usage above that required to solve the matrix. Total memory consumption is typically less than 1.5 times what is required to store the matrix, for large matrices.

References

1. R. F. Harrington, *Field Computation by Moment Methods* . Maxillan, New York (1968).

Conductor Loss Models in Momentum

Conductor Loss Models in Momentum

Surface Impedance Models

Sheet Conductor



- model 0: DC resistance

$$Z_{s}(t, \sigma, \omega) = \frac{1}{\sigma t}$$

- model 1: single sided skin effect

$$Z_{S}(t, \sigma, \omega) = Z_{C} \coth(jk_{C}t)$$

- model 2 (default): double sided skin effect

$$Z_{s}(t, \sigma, \omega) = \frac{1}{2} Z_{c} \operatorname{coth}(jk_{c} \frac{t}{2})$$

With

$$Z_{c} = \sqrt{\frac{j\omega\mu}{\sigma + j\omega\varepsilon}}$$

$$jk_c = \sqrt{j\omega\mu(\sigma+j\omega\varepsilon)}$$

In momentum.cfg file: MOM3D_USE_SHEETLOSSMODEL=0,1,2

Thick Conductor (2D Distributed Model)

No horizontal currents on the conductor sides are allowed. Deprecated setting.

- model 0: 2-layer sheet conductor
 - each layer has half the conductor thickness
 - each layer is modeled with DC resistance sheet model


- model 1: 2-layer sheet conductor Note: this is default in ADS 2003C
 - each layer has half the conductor thickness
 - each layer is modeled with single-sided skin effect sheet model
- model 2 (default): 2-layer sheet conductor
 - 2 surface current layers at top and bottom
 - both modeled with coupled skin effect



$$Z_{s,s}(t, \sigma, \omega) = \frac{1}{2}(Z_1 + Z_2)$$
$$Z_{s,m}(t, \sigma, \omega) = \frac{1}{2}(Z_1 - Z_2)$$

$$\begin{aligned} Z_1 &= \frac{Z_{c,1} Z_{c,2}}{Z_{c,1} \tanh(jk_{c,2} \frac{t}{2}) - Z_{c,2} \tanh(jk_{c,1} \frac{t}{2})} \\ Z_2 &= \frac{Z_{c,1} Z_{c,2}}{Z_{c,1} \coth(jk_{c,2} \frac{t}{2}) - Z_{c,2} \coth(jk_{c,1} \frac{t}{2})} \end{aligned}$$

medium 1 = background layer, medium 2 = conductor In *momentum.cfg* file: MOM3D_USE_THICKLOSSMODEL=0,1,2

Thick Conductor (3D Distributed Model)

Horizontal currents on the conductors sides are included.

The same models as for thick conductors without horizontal side currents apply, however, the thickness is replaced by the effective thickness of the solid object represented by the thick conductor.



Example: Single Trace in Free Space





Convergence Using N-layer Sheet Conductor Model



Results Using Sheet Conductor Models



Results Using Thick Conductor Models

Example: Folded Trace in Free Space





Results Using Thick Conductor Models

Using Infinite and Finite Ground Planes

Any closed boundary condition or slot conductor layer in ADS is considered to be an infinite Ground Plane. It is important to understand what is modeled for the various definitions for these infinite planes. Since return currents are modeled in the infinite ground plane for all three models (perfect conductor, impedance, conductivity), the inductance of the Ground Plane is included in the simulation results for any case with an infinite Ground Plane.

When using the impedance and conductivity models for infinite Ground Planes, Momentum and FEM models and captures an equivalent surface impedance. The surface impedance is described as follows:

- For both the impedance and conductivity models, the High Frequency resistive losses are included but not the DC resistive losses.
- For the impedance model, the surface impedance is constant versus frequency.
- For the conductivity model, the surface impedance is frequency dependent. (Note: for infinite ground planes with a loss conductivity specification, the MW mode of Momentum incorporates the High Frequency (HF) losses in ground planes; however, the RF mode of Momentum will make an abstraction of these HF losses.)

If DC resistive losses need to be modeled in a Ground Plane, then a finite Ground Plane must be used. The DC resistance between two points on an infinite ground plane is always zero (independent of the impedance value). This is a theoretical limit that does not exist in real life (ground planes are always finite in size). Hence, Momentum will always compute a zero DC resistance for an infinite Ground Plane.

Specifying a finite conductivity in the Ground Plane provides an additional way to model a loss mechanism that happens in a real world situation. In this case, it is the resistive losses of the 'return current' that flows in the Ground Plane. In most practical cases the Ground Plane losses are much smaller than the losses in the signal lines because the Ground Plane metallization is both thicker and wider than the signal lines (thus the return current is distributed on more metal which reduces the losses). As such, modeling the Ground Plane as a "Perfect Conductor" is less of an approximation than modeling the signal lines as "Perfect Conductors".

Another way to think about this would be to consider the difference between modeling a big chunk of metal (an 'area fill' as a 'finite Ground Plane') as either a "Perfect Conductor" or with a finite conductivity. The effect will typically be a slight increase in the simulated losses, but usually the difference will be pretty small.

Defining Infinite Ground Planes in ADS

This section demonstrates how to define infinite Ground Planes within ADS. When an infinite Ground Plane is required, the 'Closed' boundary condition is used. This can be selected at the top and/or bottom of the substrate stack-up. Note that intermediate layers in the substrate stack-up can also be used as infinite Ground Planes, but are defined by using the 'Layout Layers' tab of the substrate editor to define 'Slot' planes at the desired locations.

Access the substrate editor, to define the 'Closed' boundary condition (or 'Slot' layers) that is used for an infinite Ground Plane. The Slot layers may also be used as infinite Ground Planes. This can be used in many different applications. Consider the case of a simple transmission line over an infinite Ground Plane that has a hole in it. If you want to create this structure, perform the following steps in the subsequent Figures.

ADS Momentum Layout for a very simple case of a transmission line over an infinite Ground Plane that has a hole in it. (The signal trace is on the "cond' layer and the hole for the Ground Plane is drawn on the "resi" layer, which is displayed in green and in outline mode. There is no significance to which layer upon which this is drawn, we just need to make note of it for the next steps.)



After the layout is created, the next step is to properly define the substrate stackup. This is shown in the next two Figures. These illustrations will conclude the main concepts that you need to know to setup infinite Ground Planes using 'Slot' layers.

Defining Finite Ground Planes in ADS Momentum This section will pictorially demonstrate how to define finite Ground Planes within ADS Momentum.

When a finite Ground Plane is desired, 'Strip' conductor layers are used with "Ground Reference" ports attached to them such that an explicit ground definition is created. In the case of a simple microstrip structure, the bottom 'Closed' boundary is replaced with an 'Open' boundary definition and then the Layout Layer with the object representing the finite Ground Plane is mapped as a 'Strip' layer into the appropriate substrate stack-up location. In the Figures on the following pages, the geometry (a simple rectangle in this case, but it could be any polygon) to be used as the finite Ground Plane was drawn on the "cond2" layout layer.

Note that the layout layer with the finite Ground Plane object ("cond2" in this case) is mapped as a 'Strip' layer at the bottom conductor location when using the open boundary at the very bottom of the substrate stack-up. In this case, we will consider a scenario with edge ports placed on the signal and finite Ground Plane objects.

For the case above, we observed some nuances to consider when using "Single Mode" ports with the finite Ground Plane configuration. Now let's look at a similar case that uses "Internal" ports instead. Once again, note that the layout layer with the finite Ground Plane object ("cond2" in this case) is mapped as a 'Strip' layer at the bottom conductor location when using the open boundary at the very bottom of the substrate stack-up. In this next case, we will consider a scenario with "Internal" point ports placed inside the signal and finite Ground Plane objects.

Setting up Momentum Simulations

Setting up Momentum Simulations

This section provides information about how to create and simulate a design with Momentum.

Creating a Physical Design

You start with the physical dimensions of a planar design, such as a patch antenna or the traces on a multilayer printed circuit board. There are three ways to create a design in ADS:

- 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, see Schematic Capture and Layout and Creating a Layout for EM Simulations. For information on importing designs, see Importing and Exporting Designs.

Applying and Drawing Vias in Layout

Momentum creates a via by extruding the object that is mapped as a via through the substrate layer it is applied to. Vias can be drawn as polylines (open or closed) or polygons. A simple line segment via is the simplest and most practical way of drawing a via. Vias drawn as open polylines are often called sheet vias because, when the line is extruded through the substrate, it is treated as a vertical metal sheet. Vias drawn as closed polylines or polygons will be treated as if they are filled. For example, for a cylinder via you draw a circle. When the shape is mapped to a via metallization layer in Momentum, the vertical dimension is added to the object in order for the shape to cut through the substrate. Thus a line becomes a sheet, a circle becomes a cylinder. To model a hollow via, use boolean operations to subtract the inner cylinder from the outer one keeping the resulting shape. During simulations, it will be assumed that the inner cylinder is filled with the material of the substrate layer it is applied to. A via cannot be coincident with another via if they cut through the same substrate layer and they are on different layout layers. But two vias can be coincident on the same layout layer.

Selecting the Momentum Mode

After creating a physical design in the layout window, you need to specify simulation setup options by using the EM Setup window. Select EM > Simulation Setup in the Layout window. Most Momentum simulation setups are specified in the EM Setup window.

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

From the EM Setup window, 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.

Momentum RF	Momentum Microwave
High-Speed Digital (SI, BGA) RF Board (FR4, Duroid) RF Padkage (Plastic)	
	Planar Antennas
RFIC (Silicon)	
RF Module (MCM, LT	rcc)
Micro	wave - hybrid (Alumina)
	Microwave - IC (GaAs)
Initial design &	Final design & ► optimization

Deciding which mode to use depends on your application. Each mode has its advantages. In addition to specifically RF applications, Momentum RF can simulate microwave circuits. 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. Later, you can switch to Momentum to include the 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.

To select a Momentum mode:

- 1. Choose EM > Simulation Setup in the Layout window. The EM Setup window is displayed.
- 2. Choose either Momentum RF or Momentum Microwave in the Simulator panel, as shown in the following figure:

EM Simulator		
Momentum RF	Momentum Microwave	◎ FEM

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, as shown in the following figure:



For more information about the EM Setup window, see EM Setup Window Overview and Setting up EM Simulations.

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.
- Partitioning: The Automatic EM/Circuit Partitioning feature facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator. For more information, see Automatic EM Circuit Partitioning.
- Substrate: You can open a predefined substrate file from ADS by selecting Substrate. For more information, refer Defining Substrates.
- 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, see Ports, Defining Ports, and Ports for Momentum Overview.
- Frequency Plan: You can add or remove frequency plans for your EM simulations by selecting Frequency Plan. For more information, see Defining a Frequency Plan.
- Output Plan: You can specify the data display settings for your EM simulations by selecting Output Plan. For more information, see Defining a Frequency Plan.
- Options: You can define the preprocessor, mesh, simulation, and expert settings by selecting Options. For more information, see Defining Simulation Options.

WARNING

You can control the number of threads used by Momentum by setting OMP_NUM_THREADS environment variable. This variable takes precedence over what you set in the EM Setup window. If you limit the number for cores via OMP_NUM_THREADS, and try to specify more cores using the EM Setup window, you will not get extra cores.

- Resources: You can specify the local, remote, and third party settings by selecting Resources. For more information, see Specifying Simulation Resources
- Model: You can generate an EM model by selecting Model. For more information, see Generating an EM Model.
- Notes: You can add comments to your EM Setup window by selecting Notes.

Defining Substrates

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

For specifying a substrate for your Momentum simulation, see Defining Substrates in an EM Simulation.

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 in Momentum. There are several different types of ports that you can use in your circuit, depending on your application.

For assigning ports in a Momentum simulation, see Ports.

Defining the Frequency and Output Plan

You can set up multiple frequency plans for a Momentum simulation. You can also specify the data display settings for your EM simulations by using the EM Setup Window.

For specifying a frequency and output plan, see Defining a Frequency Plan and Defining an Output Plan.

Defining Simulation Options

You can specify global options, such as physical model, preprocessor, and mesh for Momentum simulations. You can either use a predefined set of simulation options or or create a new set in Simulation Options of the EM Setup window. You can specify the following simulation options for a Momentum simulation:

- Defining Preprocessor Settings
- Specifying Physical Model Settings for Momentum
- Defining Solver Settings
- Defining Mesh Settings for Momentum

Setting up Local, Remote, or Third Party Simulation

You can run a Momentum simulation 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.

For more information, see Specifying Simulation Resources.

Adding a Box or Waveguide

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

Running the Momentum Simulation

You set up a simulation by specifying the parameters of a frequency plan, such as the frequency range of the simulation and the sweep type. When the setup is complete, you run the simulation. The simulation process uses the mesh pattern, and the electric fields in the design are calculated. S-parameters are then computed based on the electric fields. If the Adaptive Frequency Sample sweep type is chosen, a fast, accurate simulation is generated, based on a rational fit model.

For more information, see Running Momentum Simulations.

Viewing Momentum Simulation Results

You can display the results of a Momentum simulation by using the Data Display and Visualization window. You can display the various type of results from a Momentum simulation such as S-parameters, radiation patterns (far-field plots) and derived antenna parameters.

For more information, see Visualizing 3D View before EM Simulation and Visualizing Momentum Simulations.

Computing Radiation patterns

Once the electric fields on the circuit are known, you can compute electromagnetic fields. The electromagnetic fields can be expressed in the spherical coordinate system attached to your circuit. For more information on radiation patterns, see Computing Radiation Patterns.

Examples- Setting up Momentum Simulations

Examples- Setting up Momentum Simulations

- Designing a Coplanar Waveguide Bend
- Defining Momentum Settings for LPF

Defining Momentum Settings for LPF

Defining Momentum Settings for LPF

This example describes the recommended settings for Momentum by designing a MMIC Low Pass Filter (LPF).

Application Workspace

This application is available in the following location: \$HPEESOF_DIR/examples/Tutorial/LPF_Design_Demo_wrk.7zap

Defining a Substrate

To open a predefined substrate:

- 1. Select EM > Simulation Setup to open the EM Setup window.
- 2. Select Substrate.
- 3. Select the LPF_Design_Demo_lib substrate file from the drop-down list.
- 4. To open the Substrate window, click Open. The following figure displays the LPF_Design_Demo_lib substrate window.



Defining Materials

- 1. Select a substrate layer in the Substrate window. The substrate layer properties are displayed.
- 2. Select Demo_Nit21 from the Material drop-down list.
- 3. You can also select a material by clicking Edit Materials (.). The Materials Definitions window is displayed.

	Material		Permittivity (E	7)	Perme	abiity (MUr)		
Material Name "	Lbrary	Real	Imaginary	TanD	Real	Imaginary	Type	^
Demo_GaAs	DemoKit_Non_Linear_tech	12.9		0.0036	1	0	Prequency Independent	
Demo_GaAs1	LPF_Design_Demo_lib	12.9		0.0036	1	0	Frequency Independent	
Demo_Nit1	DemoKit_Non_Linear_tech	7.5		0.0005	1	0	Frequency Independent	
Demo_Nit11	LPF_Design_Demo_lib	7.5		0.0005	1	0	Frequency Independent	~
<							·	>

- 4. Choose the required material and click OK.
- 5. Type 0.12 in the Thickness field and select micron.
- 6. Select <inherit from substrate> from the Bounding area layer list.

Substrate L	ayer	
Material	Demo_Nit21	v
Thickness	0.12	micron 💌
Bounding a	rea layer: <pre> <inherit from="" substrate=""></inherit></pre>	···

Defining a Conductor Layer

- 1. Select a conductor layer in the Substrate window.
- 2. Specify the settings displayed in the following figure:

Conductor La	er
Layer	11 (6) 💌 📖
	Only pins and pin shapes from layer
Material	u 🔽 🛄
Operation	 Sheet Intrude into substrate Expand the substrate
Position	 Above interface Below interface
Thickness	micron 💌
Surface roughness model	Top <none> Bottom <none></none></none>
Precedence	2n 1 [⊥]
To move the down.	nductor up or down on the substrate, just drag it up or

Defining Ports

Defining TML Ports

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select Ports in the left pane of the EM Setup window.
- 3. Select TML from the drop-list available in the Calibration column.
- 4. Click once in the Ref Impedance [Ohm] column of a port and specify the required value.
- 5. Click once in the Ref Offset [um] column of a port and specify the required value.

Assigning Pins to an S-parameter Port

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.
- **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 Delete Ports and Continue in the message box.

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

	× 121 11	s T		
-parameter Port	S A Lu			
Number	- Name	Ref Impedance [Ohm]	Calibration	
□ <u>1</u> 1 □ 0 P1 □ Gnd	P1	50 + 0i	TML	
☐ 2 ☐ 0 P2 ☐ 0 Gnd	P2	50 + 0i	TML	
<				
Inconnected Lav	out Pins			
Name 🔶 Co	nnected to	Layer Num Layer	Purpose Num	

Specify 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 left pane of the EM Setup window.
- 3. To add a new frequency plan, click Add.
- 4. Select Adaptive sweep from the Type column.
- 5. Specify 0 GHz in the Fstart column.
- 6. Specify 10 GHz in the Fstop column.

- 7. Specify 50(max) of N points in the Npts column.
- 8. Do not specify any value in the Step column.
- **9.** Select the check box in the Enabled column to enable the frequency plan. The following figure displays the recommended frequency settings:

	Туре 🔍	Fstart	Fstop	Npts	Step	Enabled
1	Adaptive	0 GHz	10 GHz	50 (max)	-	

See frequency plan for more information.

Defining 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 left pane of the EM Setup window.
- **3.** The default dataset name is displayed. To specify a new dataset, click Edit. For more information, see Defining an Output Plan#Selecting an Output Dataset.
- 4. Select Simulation Type as an appending text to the dataset name.
- 5. Select Open data display when simulation completes.
- 6. Select Auto-select based on number of Ports as a template to be used for displaying the data in Template.
- 7. Select Reuse files from previous simulation.
- 8. Select All generated frequencies.

Output Plan	
Dataset	
Start of the name: LPF_Finite_Thickness	Edit
Append text to the name	
From the simulation setup view:	
View name: emSetup	
Simulation type: MomRF	
Simulation options: Default	
Finally:	
This text: mySuffix	
Name: LPF_Finite_Thickness_MomRF	
⊂Data Display	
Open data display when simulation completes	
Auto-select based on number of norts	
O User speatied file [S_Nport_P.ddt	Browse
Reuse files from the previous simulation	
Save currents for:	
O two requencies	
User specified frequencies	

NOTE See Define an Output Plan for more information.

Specifying Physical Model Settings

Defining Engine-specific Modeling Options

To specify physical model definition globally:

- 1. Choose EM> Simulation Setup to open the EM Setup window.
- 2. Select Options in the left pane of the EM Setup window.
- 3. Click the Physical Model tab.
- **4.** Ensure that the Global tab is displayed.
- 5. Select 3D-distributed from the Thick Conductor drop-down list.

- 6. Select 2D-distributed from the Via drop-down list.
- **7.** Select the required lumped and wire via model options, as shown in the following figure:

Simulation Optic	ons		(
Default		Y [Co	py as	Rename	Remove
Description	Physical Mode	el Preprocess	or Mesh	Simulation	Expert
Global	Layer Specific				
Model typ	e for currents —				
Thick co	onductor	3D-distributed			*
Via		2D-distributed			*
Model	type info				
	umped: Self R, I	(no mutual coup	ling)		
	Wire: R, L (mu	utually coupled)			
2D Dist	ributed: Vertical	currents only			
3D Dist	ributed: Vertical	and horizontal cu	rrents		
Lumped a	nd wire via mode move the via out nove pads not co ep pads at the o nal pad radius:	l options ine innected to a trac uter ends of the 3 via r	e pad stack adii 🔽		
🗹 Rer	nove antipads wi	thin radius	5	5 via radii 🔽	
Rer	nove thermal reli	efs within radius	5	i via radii 🗸	

Specifying Layer-specific Modeling Options

To specify physical model 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. Click the Physical Model tab.
- 4. Click the Layer Specific tab.
- 5. Select a value from the Model Type for Via drop-down list in the required layer, as shown in the following figure:

Description Phy	vsical Model Preprocessor	Mesh Simulation Expert
Global Layer	Specific	
	Model Type for Currents Thick Conductor	Model Type for Currents Via
<global></global>	3D-distributed	2D-distributed
BVia	-	2D-distributed
MO	-	-
M1	-	-
M2	-	-
Via0_1	-	2D-distributed
Via1_2	-	2D-distributed
Via_Nit1	-	2D-distributed
Via_Pass	-	2D-distributed
demo_dummy	-	-
mesa	-	-
nicr	-	-

Defining 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. Click the Preprocessor tab.
- 4. Select Heal the layout.
- 5. Select Auto-determine a safe snap distance (conservative).
- 6. Select Merge shapes touching each other where possible to preserve the edges shared between adjacent shapes.
- **7.** Select Simplify the layout to generate a conformal mesh containing minimum number of edges.
- 8. Type 1 in the Displacement field.
- **9.** Select Generate and replace shapes on derived layers to enable the processing of derived layers.

- **10.** Select Save Preprocessor Messages as a DRC Result to display the locationbound information and warning messages.
- 11. Type EmPpmsgs in the Job Name field.
- **12.** Type 255 in the DRC Layer field. The following figure displays the recommended preprocessor settings for Momentum.

Simulation Options				
Default Copy as Rename Remove				
Description Physical Model Preprocessor Mesh Simulation Expert				
Show Visual Aid				
Heal the layout				
nearby edges and vertices will be snapped together				
 Auto-determine a safe snap distance (conservative) 				
O User specified snap distance: 0 um 🖂				
Merge shapes touching each other where possible				
shared edges in the mesh will not be preserved				
Simplify the layout				
vertex count will be reduced, without changing the topology				
Constraints (upper limits)				
Displacement 1 % of the wavelength 💌				
Shrinkage/growth 7.6 % or arc resolution 45 degrees				
Use different constraints for vias				
Displacement 1 % of the wavelength 🖂				
Shrinkage/growth 7.6 % or arc resolution 45 degrees				
Never change lines whose length exceeds 0 um				
Generate and replace shapes on derived layers				
Save preprocessor messages as a DRC result				
Job name EmPpMsgs				
DRC layer 255				

Defining Mesh Settings

Define global and layer-specific mesh settings for Momentum.

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. Select Highest Simulation Frequency.
- 6. Select Cell/Wavelength and type 20.
- 7. Enable Edge Mesh
- 8. Select Auto-determine Edge Width.
- **9.** Enable Mesh reduction to obtain an optimal mesh with fewer small cells and an improved memory usage and simulation time.
- **10.** Enable Thin layer overlap extraction o 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. The Normal setting is recommended for most layouts. The Aggressive setting is only recommended when tiny overlap differences are expected to lead to significant result variations as this option will considerably increase the problem size. For more information, refer to Defining Mesh Settings for Momentum#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 Specificsubtab 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 Densitysection. 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 Defining Mesh Settings for Momentum#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

Creating an EM Model and Symbol

To create an EM model and symbol:

- 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.** Accept the default *emModel* name.
- 4. Select Create EM Model when simulation is launched. If the Create Now button is labeled Update Now, then the EM Model already exists for this cell.
- 5. Type Symbol_look_alike in the View Name field.
- 6. Select Layout Lookalike.
- 7. Select min pin-pin distance from the Size drop-down list.

8. Select Create Symbol when simulation is launched to create a symbol after the simulation process.

Designing a Coplanar Waveguide Bend

Designing a Coplanar Waveguide Bend

There are two methods for constructing a coplanar waveguide in layout:

 The objects that you draw are the slots on a groundplane. In the substrate definition, the layout layer that the slots are drawn on is mapped to a slot metallization layer. This method gives you infinite grounds on the coplanar plane. An example is the coplanar waveguide bend, shown below.



 The objects that you draw represent metal. For example, you can draw strips and ground metal with gaps in between. This layer is mapped to a metallization layer defined as a strip. This method gives you finite grounds on the coplanar plane. Typically, this method is used when coupled microstrip lines are created initially as schematic elements.



Drawing slots on a ground plane is the preferred method because a mesh with fewer edges or unknowns will result, compared to meshing the strips. If a coplanar waveguide design is drawn with strips, it can be converted to a slot pattern with a boolean operation if a bounding box is provided.

Designing a Coplanar Waveguide Bend

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

- Creating a layout that is drawn on more than one layer
- Using vias as connections between layers

- Specifying new port types
- Working with new mesh parameters

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



Creating the Layout

In this example, the circuit is created entirely within the Layout window, and a schematic representation is not used. Be sure to save your work periodically. The layout consists of the following components:

- Coplanar waveguide slots
- Bridges that span the two slots
- Vias that vertically connect the bridges to the slots
 The slots will be drawn on one layer, the bridges will be drawn on a second layer, and the vias on a third layer. This will be important for when these layers are mapped to the substrate. An illustration of the circuit is shown on the left, the layout is on the right.



- Set your preferences so that a Layout window will open instead of a Schematic window. From the Main window, choose Tools > Preferences and enable Create Initial Layout Window. Disable Create Initial Schematic Window.
- 2. Create a new workspace named cpw_bend.
- **3.** From the Layout window, choose Options > Preferences. Scroll to the Layout Units tab. Set the layout units to um and the resolution to 0.001. Click OK.
- 4. There is a difference between the length units that you chose when you created the new project, and the units that you just set. The units chosen when you created the project set the units of the layout window in which you are working. The units chosen through *Options > Preferences* sets the units for objects that are drawn in the Layout window.
- 5. You will rename the layout layers that are used for drawing the slots, bridges, and vias of the coplanar waveguide bend. Using more descriptive names can help identify which layer that each part of the circuit is drawn on. Choose Options > Layers. Select the Advanced tab. From the Layers list, select cond. In both the Name and Layer Binding fields, enter slot. Click Apply (located just below the Layers list). Select cond2. In both the Name and Layer Binding fields, enter bridge. Click Apply. Select hole. In the Name field only, enter via and click Apply. Click OK.

Drawing the Waveguide Slots

The section describes how to draw the coplanar waveguide slots.

- 1. Verify that *slot* is the current layer. The slots will be drawn on this layer. The name of the current layer is displayed in the tool bar and at the top of the Layout window. If slot is not displayed, select it from the list.
- 2. Choose Insert > Rectangle , then choose Insert > Coordinate Entry. The toolbar arrow is activated and the *Coordinate Entry* dialog box opens.
- **3.** You will define four rectangles to make two waveguide bends. Establish the following values in the X,Y coordinate window, and click Apply after each entry:

X, Y coordinates	Define
0, 0	First corner of first rectangle
+45, +290	Second corner of first rectangle
0, +335	First corner of second rectangle
+335, +290	Second corner of second rectangle
+110, 0	First corner of third rectangle

X, Y coordinates	Define
+155, +180	Second corner of third rectangle
+110, +180	First corner of fourth rectangle
+335, +225	Second corner of fourth rectangle

- 4. Click OK when finished, to dismiss the dialog box.
- 5. Select View > View All to see your drawing.
- 6. End the command. The drawing in the layout window should resemble the figure here.



Drawing the Bridges

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

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

NOTE

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

2. Choose Insert > Rectangle , then choose Insert > Coordinate Entry.

3. Using the *Coordinate Entry* dialog box you will define two rectangles that make the bridges. Establish the following values in the X,Y coordinate window, and click Apply after each entry:

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

- 4. Click OK when finished, to dismiss the dialog box.
- 5. End the current command. The drawing in the Layout window should now look like the following figure.



Drawing a Via

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

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

- 1. From the layout layer list on the toolbar, select via.
- Choose Options > Preferences. Select the Grid/Snap tab. Under Active Snap Modes, enable Vertex and deselect Grid. Click OK. By using the vertex snap mode, the vias that you draw will snap to the vertex of a nearby object (a bridge) and not to a grid point. This ensures that the vias will be positioned precisely on the edges of the rectangles.

- 3. Choose Insert > Polyline. Referring to the points identified on the drawing below, create the line segments shown by performing these steps:
 - Click once on A1 and click twice on A2.
 - Click once on B1 and click twice on B2.
 - Click once on C1, click once on C2, and click twice on C3. Then, click the toolbar arrow. End the command. The drawing in the Layout window should now appear as shown below.



Opening a Substrate

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

- 1. From the Layout window, choose EM > Simulation Setup.
- 2. Select Substrate.
- 3. Select the substrate file *cpw_bend_example*.
- 4. Click Open.

Map the conductor layers

- 1. Right-click a layer and select Map Conductor Layer. You can see how the layers are mapped between the Free_Space, air, and GaAs layers:
 - There is a thin layer of air between the bridges and the slots.
 - The vias are mapped directly to the air substrate layer. The vias, although drawn as lines, are extruded by Momentum and effectively cut vertically through this layer of air, connecting the edges of the slots and the bridges.
 - A strip defines the layer such that the bridges are metal and what surrounds the microstrips on that layer is air. A slot is the opposite of a strip: the layout layer would be treated as a ground plane, and the

bends that you have drawn are treated as open areas in the ground plane. *Vias* represent vertical sections that cut through other layers vertically.

Adding Ports

This section describes how to add ports to the layout and how to apply the coplanar port type to them. Coplanar is one of the several port types that are available in Momentum. It is designed specifically for coplanar waveguide circuits, where an electric field is likely to build up between two ports. Each of the two ports is excited with the same absolute electrical field, but with opposite polarity. The currents are equal but in opposite in direction. For more information, see Ports.

- 1. The ports must be added to the waveguide slots and so they must be drawn on the same layer as the slots. From the Layout Layers list box, select slot.
- 2. The ports must also be positioned in the center of each slot edge. To help position the ports, set the following snap modes:
 - Enable Options > Midpoint Snap
 - Enable Options > Edge/Centerline Snap
 - Disable *all* other snap modes except Snap Enabled.
- 3. Click the Pin icon. The Properties dialog box appears.
- 4. Verify that Num=1. If it is not, select Num and set it to 1. Click OK.
- 5. Note the ghost image of the port. If it appears very large, or if it cannot be seen, cancel the command. Depending on your system defaults, you may need to change the value of the Port/Ground Size (Options > Preferences > Placement) and the value of the Font Definition Size (Options > Preferences > Component Text) before your ports and labels will be an appropriate size for this layout.
- 6. When you are ready to add ports, apply them in the order and direction as shown here, then end the port command.



- **7.** From the Layout window, chose EM > Simulation Setup. Select Ports to display port properties.
- 8. In the Layout window, click P1. In the EM Setup window, verify that the calibration is set to TML.
- 9. Click P2. Verify that the calibration is set to TML.
- 10. Select the 🗢 terminal of P2 and drop it on the 😳 terminal of P1.
- 11. Click P3. Verify that the calibration is set to TML.
- 12. Click P4. Verify that the calibration is set to TML.
- 13. Select the 🗢 terminal of P3 and drop it on the 😳 terminal of P3.
- 14. Save your changes.

Generating the Mesh

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

- 1. From the Layout window, choose EM > Simulation Setup.
- 2. Select Options.
- 3. Click the Mesh tab.
- 4. Set the following Global parameters:
 - Mesh Frequency = 40 GHz
 - Number of Cells per Wavelength = 30
 - Enable Edge Mesh
 - Global parameters will be applied to all layers, unless other values are specified from the Layer tab.
- 5. Click the Layer tab.
- 6. In the Layout Layers list, select bridge. Set the mesh density to 30 and select *No Edge Mesh.* These mesh parameters will be applied to the objects drawn on the bridge layer only.
- 7. Select Preprocessed geometry from the Generate list
- 8. Click Simulate to start the computations. The mesh will be computed and displayed on the layout, as shown here. Zoom in for a closer look at the mesh.



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

Performing a Simulation

- 1. From the Layout window, choose EM > Simulation Setup.
- 2. Select Frequency Plan in the EM Setup window.
- 3. From the Type list, select Adaptive and set these parameters:
 - Fstart = 1 GHz
 - Fstop = 40 GHz
 - Npts = 50
- 4. Click Add.
- 5. Click Simulate.

Viewing Simulation Results

The figure shows some of the automatically plotted simulation results.



HSD Applications: Guidelines for Specifying EM Settings

HSD Applications: Guidelines for Specifying EM Settings

This section provides guidelines for setting up an EM simulation with Momentum for HSD users.

Select the Momentum RF Mode

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

Momentum RF	Momentum Microwave
High-Speed Digital (SI, BGA) RF Board (FR4, Duroid) RF Package (Plastic)	
	Planar Antennas
RFIC (Silicon)	,
RF Module (MCM, LTC	C)
Microw	ave - hybrid (Alumina)
	Microwave - IC (GaAs)
Initial design &	Final design & ptimization

To select a Momentum mode:

- 1. Choose EM > Simulation Setup in the Layout window. The EM Setup window is displayed.
- 2. Select Momentum RF, as shown in the following figure:

Simulator	
- China Con	
O Momentum RF	Momentum Microwave

Selecting a Set up Type

You can configure a circuit simulation to use an EM-based model for a cell instance by selecting EM Cosimulation. For more information, see EM Circuit Cosimulation.

Specifying Momentum Simulation Settings

To set up a Momentum simulation:

- Select the Momentum RF Simulator.
- Define a substrate.
- Specify a Frequency Plan.
- Specify Physical Model Settings.
- Specify Preprocessor Settings.
- Create Ports.
- Create an EM Cosimulation View.



For more information about the EM Setup window, see EM Setup Window Overview and Setting up EM Simulations.

Partitioning

Using partitioning, you can combine the EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator or for which a model already exists. For more information, see Automatic EM Circuit Partitioning.

Define a Substrate

For HSD applications, it is recommended to specify the sheet option for conductor layers.

To define a substrate:

- 1. Open the Substrate Editor by double-clicking the imported substrate.
- 2. Select the conductor layer.
- **3.** Select the Sheet option for the conductor layers highlighted in the following figure:
| 🗋 📁 🔚 🕂 🎺 🤕 | B | | | | |
|-------------|--|--|--|--|---|
| 4 | DR1L_1_2 DR1L_2_3 DR1L_3_4 DR1L_4_5 DR1L_5_6 | AR
PCV1MDE
0.017 millinear
P, 4
0.13 millinear
P, 4
0.13 millinear
P, 4
0.13 millinear
P, 4
0.13 millinear
P, 4
0.12 millinear
P, 4
P, 4 | Use right mouse
Select items on
Conductor L
Layer
Material
Operation
Thickness
Surface
roughness
Precedence
To move the
down. | context menus to add or delete substrate it
the substrate and view their properties below
ayer
ETCH_IN2_L4 (1004)
Only pins and pin shapes from layer
COPPER
Sheet
Diffunde into substrate
Expand the substrate
Below interface
Below interface
Below interface
Below interface
1 C | ems,
v,
v are
milimeter v
v are
v are
t drag it up or |

Specify a Frequency Plan

For HSD applications, it is recommended to select the Adaptive type frequency. To create a frequency plan:

- 1. Select Frequency Plan in the left pane of the EM Setup window.
- 2. To add a new frequency plan, click Add.
- 3. Specify the Type, Fstart, Fstop, Npts values, as shown in the following figure:

Fre	equency Plan					
	Add	Remove				
	Туре	Fstart	Fstop	Npts	Step	Enabled

For more information about how to specify a frequency and output plan, see Defining a Frequency Plan and Defining an Output Plan.

Specify Physical Model Settings

For faster EM simulation results, it is recommended to select the Wire option for Via globally. You can also specify the layer-specific and net-specific model types. For more information, see Specifying Physical Model Settings for Momentum.

Defining Net Specific Physical Model Options

You can specify the physical model definition for a specific net by using the Net Specific tab in Physical Model options. To define a model type for specific nets in your design:

- 1. Select EM> Simulation Setup to open EM Setup window.
- 2. Select Options in the left pane of the EM Setup window.
- 3. Click the Physical Model tab.
- 4. Click the Net Specific tab, as shown in the following figure:

S	imu	ula	tion Options						
	De	fa	ult						•
	C)es	scription	Physical Model	Preproces	sor	Mesh	Solver	Expert
	ļ	(Global La	ayer Specific	Net Specific				
				Model Type for	r Currents	Mov	del Type f	or Currents	
				Model Type for Currents Model Type for Thick Conductor Via		i currents			
			<global></global>	3D-distributed	3D-d				
	- 1								

- 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.

Specify Preprocessor Settings

It is recommended that the user specified snap distance is set at 20 % of max trace Width (Max trace width:10 mil). For more information, see Defining Preprocessor Settings.

To specify Preprocessor settings:

- 1. Click the Preprocessor tab in the EM Setup window.
- 2. Specify the User-defined snap distance as 20 % of the maximum trace Width.
- 3. Specify the arc resolution value, such as 60 degrees.

Create Ports

To assign pins:

1. Open the Port Editor window from a layout by clicking the Port Editor icon () present on the EM toolbar.

2. Move all GND pins and drop them on the respective VDD part of the component, which is the negative terminal, as shown below:

Number	<u>^</u>		Name	Ref Imped	ance [Ohm]	Calibrat	ion	Ref Offset [mm]	Term Type	
4 <u>10</u> 1 0	VDD-C1_1		VDD-C1_1	50 + 0i		None		N/A	inputOutput	-[
4 <u>10</u> 2	VDD-C3_1		VDD-C3_1	50 + 0i		None		N/A	inputOutput	
4 ¹ 1 3	0140-05_2		VDD-C23 1	50 + 0i		None		N/A	inputOutput	
wout Dine										,
Jame	Laver	Net	Connect	ed to	Purpose	X [mm]	V[mm]	Number	Laver Num	D
	ETCU TOD	CNID	1()	euto	Junio	21,261	26 505	1	1001	-
		GND	2()		drawing	21,501	20.363	2	1001	-1
De GND	ETCH TOP	GND	A(-)		drawing	8 661	26 585	4	1001	1
De GND-	FTCH TOP	GND	F(-)		drawing	8 661	16 249	5	1001	4
De GND-	FTCH TOP	GND	2(-)		drawing	21 361	14 254	2	1001	-1
De GND-	ETCH TOP	GND	5(-)		drawing	100.061	26 585	6	1001	-1
7. GND	FTCH TOP	GND	7(-)		drawing	112.761	26,585	7	1001	-1
	ETCH TOP	GND	8(-)		drawing	112.761	14.254	8	1001	-1
	ETCH TOP	GND	9(-)		drawing	18.211	13.75	9	1001	-1
										•

- **3.** Ensure that the port calibration is set to None for all ports in the Calibration drop-down list.
- 4. Click Refresh (🔽).
- 5. Open the EM Setup window.
- 6. Select Ports in the left pane of the EM Setup window.
- 7. Click Refresh () to update ports definition in the EM Setup window.

For more information about assigning ports, see Ports.

Create an EM Cosimulation View

After finalizing the EM setup, generate an EM cosimulation view:

- 1. Select EM Cosimulation in the Setup Type panel.
- 2. Ensure that Views is selected in the Generate drop-down list.
- **3.** Click Go. An EM Cosimulation view is created, as shown in the following figure:



Running the Momentum Simulation

You set up a simulation by specifying the parameters of a frequency plan, such as the frequency range of the simulation and the sweep type. When the setup is complete, you run the simulation. The simulation process uses the mesh pattern, and the electric fields in the design are calculated. S-parameters are then computed based on the electric fields. If the Adaptive Frequency Sample sweep type is chosen, a fast, accurate simulation is generated, based on a rational fit model.

For more information, see Running Momentum Simulations.

Viewing Momentum Simulation Results

You can display the results of a Momentum simulation by using the Data Display and Visualization window. You can display the various type of results from a Momentum simulation such as S-parameters, radiation patterns (far-field plots) and derived antenna parameters.

For more information, see Visualizing 3D View before EM Simulation and Visualizing Momentum Simulations.

FEM

FEM is a powerful finite-element EM simulator that solves a wide array of applications. It enables you to model arbitrary 3D shapes such as bond wires and finite dielectric substrates. It is a 3D EM simulator, with a full 3D electromagnetic field solver, and fully automated meshing and convergence capabilities.

FEM Simulator comes with the following list of impressive features:

- Conductors, resistors, isotropic dielectrics, isotropic linear magnetic material modeling that allows 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.
- Installs with ADS and runs locally

Contents

- FEM Overview
- Theory of Operation for FEM
- Setting up FEM Simulations
- Example- Designing a Microstrip Filter
- Guidelines For Optimal Performance of FEM Simulation

FEM Overview

FEM Overview

An FEM simulator provides a complete solution for electromagnetic simulation of arbitrarily-shaped and passive three-dimensional structures. FEM simulators create 3D EM simulation for designers working with RF circuits, MMICs, PC boards, modules, and Signal Integrity applications. It provides a 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 simulators provide RF and microwave engineers access to some of the most comprehensive EM simulation tools in the industry.

Developed with the designer of high-frequency/high-speed circuits in mind, FEM Simulator offers a powerful finite-element EM simulator that solves a wide array of applications with impressive accuracy and speed.



By combining fast solution times, efficient memory usage, and powerful display capabilities, FEM Simulator for ADS provides performance solution to complex highfrequency 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

FEM Simulator comes with a list of impressive features. These key technological enablers demonstrate the advantages of full 3D EM design and verification which include:

- 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 provides an integrated approach to EM /Circuit design for both co-simulation and co-optimization.

Application Areas

EM modeling tools are known for their great accuracy. FEM Simulator redefines this term with broad application coverage, including the following:

- Microstrip, stripline, CPW elements (filters, couplers, spiral inductors, via holes, air bridges, meander lines...)
- Multilayer structures

- Ceramic filters
- Adapters/transitions
- Antennas
- Couplers
- Power splitters/combiners

Visualization and Display of Results

The visualization and animation capabilities in FEM Simulator enable you to evaluate simulation results thoroughly.

To aid in analyzing your designs, EM field animation and dynamic rotation of structures can be performed simultaneously. Choose from shaded plots, contour lines, or vectors. 3D far-field plots illustrate beam shapes in both azimuth and elevation on a single plot. To aid in analyzing your designs, EM field animation and dynamic rotation of structures can be performed simultaneously. Choose from shaded plots, contour lines, or vectors. 3D far-field plots illustrate beam shapes in both azimuth and elevation on a single plot.

Theory of Operation for FEM

Theory of Operation for FEM

The simulation technique used to calculate the full three-dimensional electromagnetic field inside a structure is based on the finite element method. Although its implementation is largely transparent, a general understanding of the method is useful in making the most effective use of FEM Simulator.

This section provides an overview of the finite element method, its implementation in FEM Simulator, and a description of how S-parameters are computed from the simulated electric and magnetic fields.

The Finite Element Method

To generate an electromagnetic field solution from which S-parameters can be computed, FEM Simulator employs the finite element method. In general, the finite element method divides the full problem space into thousands of smaller regions and represents the field in each sub-region (element) with a local function.

In FEM Simulator, the geometric model is automatically divided into a large number of tetrahedra, where a single tetrahedron is formed by four equilateral triangles.

Representation of a Field Quantity

The value of a vector field quantity (such as the H-field or the E-field) at points inside each tetrahedron is interpolated from the vertices of the tetrahedron. At each vertex, FEM Simulator stores the components of the field that are tangential

to the three edges of the tetrahedron. In addition, the component of the vector field at the midpoint of selected edges that is tangential to a face and normal to the edge can also be stored. The field inside each tetrahedron is interpolated from these nodal values.

In the following figure, the field quantities are interpolated from nodal values:



The components of a field that are tangential to the edges of an element are explicitly stored at the vertices.

The component of a field that is tangential to the face of an element and normal to an edge is explicitly stored at the midpoint of the selected edges.

The value of a vector field at an interior point is interpolated from the nodal values. By representing field quantities in this way, Maxwell's equations can be transformed into matrix equations that are solved using traditional numerical methods.

Basis Functions

A first-order tangential element basis function interpolates field values from both nodal values at vertices and on edges. First-order tangential elements have 20 unknowns per tetrahedra.

Size of Mesh Versus Accuracy

There is a trade-off between the size of the mesh, the desired level of accuracy, and the amount of available computing resources.

On one hand, the accuracy of the solution depends on how small each of the individual elements (tetrahedra) are. Solutions based on meshes that use a large number of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each tetrahedron must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

On the other hand, generating a field solution for meshes with a large number of elements requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh that is fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power. To produce the optimal mesh, FEM Simulator uses an iterative process in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh based on suitable error criteria and generates a new solution. When selected S-parameters converge to within a desired limit, the iteration process ends.

Field Solutions

During the iterative solution process, the S-parameters typically stabilize before the full field solution. Therefore, when you are interested in analyzing the field solution associated with a structure, it may be desirable to use convergence criteria that is tighter than usual.

In addition, for any given number of adaptive iterations, the magnetic field (H-field) is less accurate than the solution for the electric field (E-field) because the H-field is computed from the E-field using the following relationship:

$$\boldsymbol{H} = \frac{\nabla \times \boldsymbol{E}}{-j \, \boldsymbol{\omega} \boldsymbol{\mu}}$$

thus making the polynomial interpolation function an order lower than those used for the electric field.

Implementation Overview

To calculate the S-matrix associated with a structure, the following steps are performed:

- 1. The structure is divided into a finite element mesh.
- 2. The waves on each port of the structure that are supported by a transmission line having the same cross section as the port are computed.
- **3.** The full electromagnetic field pattern inside the structure is computed, assuming that each of the ports is excited by one of the waves.
- 4. The generalized S-matrix is computed from the amount of reflection and transmission that occurs.

The final result is an S-matrix that allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals, reducing the full three-dimensional electromagnetic behavior of a structure to a set of high frequency circuit values.

The Solution Process

There are three variations to the solution process:

- Adaptive solution
- Non-adaptive discrete frequency sweep
- Non-adaptive fast frequency sweep

Adaptive Solution

An adaptive solution is one in which a finite element mesh is created and automatically refined to increase the accuracy of succeeding adaptive solutions. The adaptive solution is performed at a single frequency. (Often, this is the first step in generating a non-adaptive frequency sweep or a fast frequency sweep.).

Non-adaptive Discrete Frequency Sweep

To perform this type of solution, an existing mesh is used to generate a solution over a range of frequencies. You specify the starting and ending frequency, and the interval at which new solutions are generated. The same mesh is used for each solution, regardless of the frequency.

Non-adaptive Fast Frequency Sweep

This type of solution is similar to a discrete frequency sweep, except that a single field solution is performed at a specified center frequency. From this initial solution, the system employes asymptotic waveform evaluation (AWE) to extrapolate an entire bandwidth of solution information. While solutions can be computed and viewed at any frequency, the solution at the center frequency is the most accurate.

The Mesher

A mesh is the basis from which a simulation begins. Initially, the structure's geometry is divided into a number of relatively coarse tetrahedra, with each tetrahedron having four triangular faces. The mesher uses the vertices of objects as the initial set of tetrahedra vertices. Other points are added to serve as the vertices of tetrahedra only as needed to create a robust mesh.

After the initial field solution has been created, if adaptive refinement is enabled, the mesh is refined further.

2D Mesh Refinement

For 2D objects or ports, the mesher treats its computation of the excitation field pattern as a two-dimensional finite element problem. The mesh associated with each port is simply the 2D mesh of triangles corresponding to the face of tetrahedra that lie on the port surface.

The mesher performs an iterative refinement of this 2D mesh as follows:

- 1. Using the triangular mesh formed by the tetrahedra faces of the initial mesh, solutions for the electric field, E, are calculated.
- 2. The 2D solution is verified for accuracy.
- **3.** If the computed error falls within a pre-specified tolerance, the solution is accepted. Otherwise, the 2D mesh on the port face is refined and another iteration is performed.

Any mesh points that have been added to the face of a port are incorporated into the full 3D mesh.

The 2D Solver

Before the full three-dimensional electromagnetic field inside a structure can be calculated, it is necessary to determine the excitation field pattern at each port. The 2D solver calculates the natural field patterns (or modes) that can exist inside a transmission structure with the same cross section as the port. The resulting 2D field patterns serve as boundary conditions for the full three-dimensional problem.

Excitation Fields

The assumption is that each port is connected to a uniform waveguide that has the same cross section as the port. The port interface is assumed to lie on the z=0 plane. Therefore, the excitation field is the field associated with traveling waves propagating along the waveguide to which the port is connected:

$$\boldsymbol{E}(x,y,z,t) = Re[\boldsymbol{E}(x,y)e^{j\omega t - yz}]$$

where:

Re is the real part of a complex number or function. E(x,y) is a phasor field quantity.

j is the imaginary unit, $\sqrt{-1}$

 $\boldsymbol{\omega}$ is angular frequency, $2\pi f$.

 $\gamma = \alpha + j \beta$ is the complex propagation constant, α is the attenuation constant of the wave.

 β is the propagation constant associated with the wave that determines, at a given time *t*, how the phase angle varies with *z*.

In this context, the x and y axes are assumed to lie in the cross section of the port; the z axis lies along the direction of propagation.

Wave Equation

The field pattern of a traveling wave inside a waveguide can be determined by solving Maxwell's equations. The following equation that is solved by the 2D solver is derived directly from Maxwell's equation.

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}(x,y)\right) - k_0^2 e_r \boldsymbol{E}(x,y) = 0$$

where:

E(x,y) is a phasor representing an oscillating electric field.

$$k_{0}$$
 is the free space wave number, $\sqrt[\omega]{\mu_{0}\epsilon_{0}}$,

\mathbf{w} is the angular frequency, $2\pi f$.

 $\mu_r(x, y)$ is the complex relative permeability.

 $\boldsymbol{\varepsilon}_{r}(x, y)$ is the complex relative permittivity.

To solve this equation, the 2D solver obtains an excitation field pattern in the form of a phasor solution, E(x,y). These phasor solutions are independent of z and t; only after being multiplied by $e-\gamma^z$ do they become traveling waves.

Also note that the excitation field pattern computed is valid only at a single frequency. A different excitation field pattern is computed for each frequency point of interest.

Modes

For a waveguide or transmission line with a given cross section, there is a series of basic field patterns (modes) that satisfy Maxwell's equations at a specific frequency. Any linear combination of these modes can exist in the waveguide.

Modes, Reflections, and Propagation

It is also possible for a 3D field solution generated by an excitation signal of one specific mode to contain reflections of higher-order modes which arise due to discontinuities in a high frequency structure. If these higher-order modes are reflected back to the excitation port or transmitted onto another port, the S-parameters associated with these modes should be calculated.

If the higher-order mode decays before reaching any port-either because of attenuation due to losses or because it is a non-propagating evanescent modethere is no need to obtain the S-parameters for that mode. Therefore, one way to avoid the need for computing the S-parameters for a higher-order mode is to include a length of waveguide in the geometric model that is long enough for the higher-order mode to decay.

For example, if the mode 2 wave associated with a certain port decays to near zero in 0.5 mm, then the "constant cross section" portion of the geometric model leading up to the port should be at least 0.5 mm long. Otherwise, for accurate S-parameters, the mode 2 S-parameters must be included in the S-matrix.

The length of the constant cross section segment to be included in the model depends on the value of the mode's attenuation constant, α .

Modes and Frequency

The field patterns associated with each mode generally vary with frequency. However, the propagation constants and impedances always vary with frequency. Therefore, when a frequency sweep has been requested, a solution is calculated for each frequency point of interest.

When performing frequency sweeps, be aware that as the frequency increases, the likelihood of higher-order modes propagating also increases.

Modes and Multiple Ports on a Face

Visualize a port face on a microstrip that contains two conducting strips side by side as two separate ports. If the two ports are defined as being separate, they are treated as two ports are connected to uncoupled transmission structures. It is as if a conductive wall separates the excitation waves.

However, in actuality, there will be electromagnetic coupling between the two strips. The accurate way to model this coupling is to analyze the two ports as a single port with multiple modes.

The 3D Solver

To calculate the full 3D field solution, the following wave equation is solved:

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}(x,y,z)\right) - k_0^2 e_r \boldsymbol{E}(x,y,z) = 0$$

where:

E(x,y,z) is a complex vector representing an oscillating electric field. mr(x, y) is the complex relative permeability.

k $_0$ is the free space phase constant, $\sqrt[\omega]{\mu_0 \epsilon_0}$.

ω is the angular frequency, 2πf.

 $\boldsymbol{\varepsilon}_{r}(x, y)$ is the complex relative permittivity.

This is the same equation that the 2D solver solves for in calculating the 2D field pattern at each port. The difference is that the 3D solver does not assume that the electric field is a traveling wave propagating in a single direction. It assumes that the vector E is a function of x, y, and z. The physical electric field, E(x,y,z,t), is the real part of the product of the phasor, E(x,y,z), and $\mathbf{e}^{j} \mathbf{w} t$:

$$\boldsymbol{E}(x,y,z,t) = Re\left[\boldsymbol{E}(x,y,z)e^{j\omega t}\right]$$

Boundary Conditions

FEM Simulator imposes boundary conditions at all surfaces exposed to the edge of the meshed problem region. This includes all outer surfaces and all surfaces exposed to voids and surface discontinuities within the structure. The following types of boundary conditions are recognized by the 3D solver:

- Port
- Perfect H
- Symmetric H
- Perfect E
- Symmetric E
- Ground plane
- Conductor
- Resistor
- Radiation
- Restore

Port Boundaries

The 2D field solutions generated by the 2D solver for each port serve as boundary conditions at those ports. The final field solution that is computed for the structure must match the 2D field pattern at each port.

FEM Simulator solves several problems in parallel. Consider the case of analyzing modes 1 and 2 in a two-port device. To compute how much of a mode 1 excitation at port 1 is transmitted as a mode 2 wave at port 2, the 3D mesher uses the following as boundary conditions:

- A "mode 1" field pattern at port 1.
- A "mode 2" field pattern at port 2.

To compute the full set of S-parameters, solutions involving other boundary conditions must also be solved. Because the S-matrix is symmetric for reciprocal structures (that is, S $_{12}$ is the same as S $_{21}$), only half of the S-parameters need to be explicitly computed.

Perfect H Boundaries

A Perfect H boundary forces the magnetic field (H-field) to have a normal component only. A symmetric H boundary can be used to model a plane of symmetry for a mode in which the H-field is normal to the symmetry plane.

Perfect E Boundaries

By default, the electric field is assumed to be normal to all surfaces exposed to the background, representing the case in which the entire structure is surrounded by perfectly conducting walls. This is referred to as a Perfect E boundary. The final field solution must match the case in which the tangential component of the electric field goes to zero at Perfect E boundaries.

It is also possible to assign Perfect E boundaries to surfaces within a structure. Using Perfect E boundaries in this way enables users to model perfectly conducting surfaces. The surfaces of all objects that have been defined to be perfectly conducting materials are automatically assigned to be Perfect E boundaries.

Conductor Boundaries

Conductor boundaries can be assigned to surfaces of imperfect conductors or resistive loads such as thick film resistors. At such boundaries, the following condition holds:

$$E_{tan} = Z_s(\hat{n} \times H)$$

where:

ñ

is the is the unit vector that is normal to the surface. E tan is the component of the E-field that is tangential to the surface. H is the H-field.

Z s is the surface impedance of the boundary. $(1+j)/(\delta\sigma)$.

δ is the skin depth, $\sqrt{2/(ωσμ)}$, of the conductor being modeled. ω is the frequency of the excitation wave. σ is the conductivity of the conductor.

The fact that the E-field has a tangential component at the surface of imperfect conductors simulates the case in which the surface is lossy. The amount of loss will be proportional to the component of ExH that flows into the surface.

The field inside these objects is not computed; the conductor boundary approximates the behavior of the field at the surfaces of the objects.

Resistor Boundaries

Resistor boundaries model surfaces that represent resistive loads such as thin films on conductors. The following condition holds at resistor boundaries:

$$E_{\text{tan}} = R(\hat{n} \times H)$$

where:

 ${\it E}$ tan is the component of the E-field that is tangential to the surface. ${\it H}$ is the H-field.

R is the resistance at the boundary in ohms per square meter.

Radiation Boundaries

Radiation boundaries model surfaces that represent open space. Energy is allowed to radiate from these boundaries instead of being contained within them. At these surfaces, the second order radiation boundary condition is employed:

$$(\nabla \times \boldsymbol{E})_{tan} = jk_0 \boldsymbol{E}_{tan} - \frac{j}{k_0} \nabla \times \hat{\boldsymbol{n}} (\nabla \times \boldsymbol{E})_n + \frac{j}{k_0} \nabla_{xan} (\nabla_{tan} \bullet \boldsymbol{E}_{xan})$$

where:

E tan is the component of the E-field that is tangential to the surface.

 \hat{n}

is the unit vector normal to the radiation surface.

k $_0$ is the free space phase constant, $\sqrt[\omega]{\mu_0 \epsilon_0}$

j is equal to $\sqrt{-1}$

To ensure accurate results, radiation boundaries should be applied at least one quarter of a wavelength away from the source of the signal. However, they do not have to be spherical. The only restriction regarding their shape is that they be convex with regard to the radiation source.

Computing Radiated Fields

Electromagnetic Design System maps the E-field computed by the 3D solver on the radiation surfaces to plane registers and then calculates the radiated E-field using the following equation:

$$\boldsymbol{E}(x,y,z) = \int_{\sigma} ((j \,\omega \mu_0 \boldsymbol{H}_{sin}) G + (\boldsymbol{E}_{tan} \times \nabla G) + (\boldsymbol{E}_{normal} \nabla G)) ds$$

where:

s represents the radiation surfaces.

j is the imaginary unit, $\sqrt{-1}$.

 \mathbf{w} is the angular frequency, $2\pi f$.

 μ_0 is the relative permeability of the free space.

H tan is the component of the magnetic field that is tangential to the surface. H normal is the component of the magnetic field that is normal to the surface. E tan is the component of the electric field that is tangential to the surface. G is the free space Green's Function, given by:

$$G = \frac{e^{-jk_0|\bar{r}-\bar{r}'|}}{|\bar{r}-\bar{r}'|}$$

where:

k $_0$ is the free space wave number, $\sqrt[\omega]{\mu_0 \epsilon_0}$.

r and \vec{r} represent, respectively, points on the radiation surface and points beyond the surface as shown in Implementing Green's Function When Computing Radiated Fields.



Implementing Green's Function When Computing Radiated Fields

Displaying Field Solutions

The 3D solver is also used to manipulate field quantities for display. The system enables you to display or manipulate the field associated with any excitation wave at any port-for example, the field inside the structure due to a discrete mode 2 excitation wave at port 3. Waves excited on different modes can also be superimposed, even if they have different magnitudes and phases-for example, the waves excited on mode 1 at port 1 and mode 2 at port 2. In addition, far-field radiation in structures with radiation boundaries can be displayed.

The available fields depend on the type of solution that was performed:

- For adaptive solutions, the fields associated with the solution frequency are available.
- For frequency sweeps, the fields at each solved frequency point are available.
- For fast frequency sweeps, the fields associated with the center frequency point are initially available.

Ports-only Solutions and Impedance Computations

This section addresses how impedances are computed for multi-conductor transmission line ports. Some examples of such structures are:

- Two coupled microstrip lines
- Coplanar waveguide modeled with 3 separate strips
- Shielded twin-wire leads

For structures with one or two conductors, you will need to define a single line segment, called an "impedance line", for each mode. Some examples of such structures include:

- microstrip transmission line (two-conductor structure)
- grounded CPW (two-conductor structure) where the CPW ground fins are also attached to the *OUTER* ground

For these structures, the port solver will compute the voltage V along the impedance line which is used to calculate Z $_{pv}$ = V 2 /(2*Power). The power is always normalized to 1 Watt.

If one models N-conductor structures where N>2, then FEM Simulator uses a different algorithm for computing Z $_{\rm pv}$ and Z $_{\rm pi}$. The user must define an impedance line for each interior conductor. The impedance lines should go from the center of each interior conductor to the outer conductor.

The port solver computes a voltage along the first N line segments for each of the first N modes when the port solver detects that there are N+1 conductors. This

becomes a "voltage vector" V

(of length N) for each of the N quasi-TEM modes. Then, when computing Z $_{\rm pv}$, the square of the scalar voltage is now replaced by the dot product of the voltage vectors, for example:

$$Z_{pv} = \bar{V} \bullet \bar{V} / (2 \cdot Power)$$

For a more detailed explanation as to why this is done, refer to reference [1] at the end of this section.

For Z $_{pi}$, the current is generally computed by adding the currents flowing into and out of the port and taking the average of the two. (If the simulator computed currents to perfect accuracy, the inward and outward currents would be identical.) For all mode numbers >= N, where N = number of conductors, the currents are calculated in this way.

For the first (N-1) quasi-TEM modes, the currents are computed on the N-1 interior conductors producing a current eigenvector. Then the impedance:

$$Z_{pi} = (2 \cdot Power) / (I \bullet I)$$

The result is that the FEM Simulator impedance computations for such structures as coupled microstrip lines match the published equations for even- and odd-mode impedances.

As an example, take a CPW modeled as three interior strips surrounded by an enclosure. The ground strips do not touch the enclosure. Such a model is in the examples directory of FEM Simulator and is called *cpwtaper*. The port solver shows us that the desired CPW mode is not the dominant mode, but is actually mode 3. To identify the modes, one can use the arrow plots in the Port Calibration menu or the Arrow display of the E-field in the post processor.

Each port consists of a 4 conductor system, the outer (ground) conductor, the inner strip, and the two "ground" strips. This results in 3 quasi-TEM modes. Mode 1 has E-field lines predominately in the substrate, all pointing in the same direction. This is the common mode (+V, +V, +V). Mode 2 also has E-field lines predominately in the substrate, but in opposite directions under the two "ground" strips. This is the slot mode (-V, 0, +V). Mode 3 has nearly zero E-fields everywhere because the fields are predominately between the inner strip and the "ground" strips. This is the CPW mode (0, +V, 0).

For further help in identifying modes in such a structure, one can look at the distributions with the "full" scale. One will notice that modes 1 and 3 obviously have the same "even" symmetry in the E fields, while mode 2 has an odd symmetry. The CPW mode has an "even" symmetry, so it has to be mode 1 or 3.

Modes 1 and 2 have significant E-field strengths in the substrate, especially under the "ground" strips. So, there is a potential difference between the "ground" strips and the outer ground for these two modes. However, for mode 3, the "ground" strips are at the same potential as the outer ground, which is consistent with the CPW mode. Thus one can identify the modes. For mode 3, the "ground" strips are at 0 volts with respect to the outer ground, and the signal line has +V. The Z _{pv} impedance computed for this mode using a dot product of the voltage vector for mode 3 gives the same Z _{pv} as by computing the simple voltage between center strip and either "ground" strip. That is because the voltage vector for mode 3 along the three

impedance lines is V = [0, V, 0] and $\overline{V} \cdot \overline{V} = [0, V, 0] \cdot [0, V, 0] = 0 \cdot 0 + V^2 + 0 \cdot 0 = V^2$

However, the impedances for the other modes now match accepted impedance definitions found in the literature for multi-conductor transmission lines.

1. G.G. Gentili and M. Salazar-Palma, "The definition and computation of modal characteristic impedance in quasi-TEM coupled transmission lines," IEEE Trans. Microwave Theory Tech., Feb. 1995, pp. 338-343.

Calculating S-Parameters

A generalized S-matrix describes what fraction of power associated with a given field excitation is transmitted or reflected at each port.

The S-matrix for a three port structure is shown below:

$$\begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

where:

All quantities are complex numbers.

The magnitudes of a and b are normalized to a field carrying one watt of power.

 $|a_i|^2$ represents the excitation power at port i.

 $|b_i|^2$ represents the power of the transmitted or reflected field at port i.

The full field pattern at a port is the sum of the port's excitation field and all reflected/transmitted fields.

The phase of a _i and b _i represent the phase of the incident and reflected

/transmitted field at t=0, $\angle a_i$

represents the phase angle of the excitation field on port i at t=0. (By default, it is zero.)

 $\angle b_i$

represents the phase angle of the reflected or transmitted field with respect to the excitation field.

S $_{ij}$ is the S-parameter describing how much of the excitation field at port j is reflected back or transmitted to port i.

For example, S $_{31}$ is used to compute the amount of power from the port 1 excitation field that is transmitted to port 3. The phase of S $_{31}$ specifies the phase shift that occurs as the field travels from port 1 to port 3.

NOTE When the 2D solver computes the excitation field for a given port, it has no information indicating which way is "up" or "down". Therefore, if ports have not been calibrated, it is possible to obtain solutions in which the S-parameters are out of phase with the expected solution.

Frequency Points

The S-parameters associated with a structure are a function of frequency. Therefore, separate field solutions and S-matrices are generated for each frequency point of interest. FEM Simulator supports two types of frequency sweeps:

- Discrete frequency sweeps, in which a solution is generated for the structure at each frequency point you specify.
- Fast frequency sweeps, in which asymptotic waveform evaluation is used to extrapolate solutions for a range of frequencies from a single solution at a center frequency.

Fast frequency sweeps are useful for analyzing the behavior of high Q structures. For wide bands of information, they are much faster than solving the problem at individual frequencies.

NOTE Within a fast frequency solution, there is a bandwidth where the solution results are most accurate. This range is indicated by an error criterion using a matrix residue that measures the accuracy of the solution. For complex frequency spectra that have many peaks and valleys, a fast sweep may not be able to accurately model the entire frequency range. In this case, additional fast sweeps with different expansion frequencies will automatically be computed and combined into a single frequency response.

Renormalized S-Matrices

Before a structure's generalized S-matrix can be used in a high frequency circuit simulator to compute the reflection and transmission of signals, the generalized S-matrix must be normalized to the appropriate impedance. For example, if a generalized S-matrix has been normalized to 50 ohms, it can be used to compute reflection and transmission directly from signals that are normalized to 50 ohms, as in:

$$\begin{bmatrix} Vo_1 \\ Vo_2 \\ Vo_3 \end{bmatrix} = \begin{bmatrix} S_{50} \end{bmatrix} a \begin{bmatrix} Vi_1 \\ Vi_2 \\ Vi_3 \end{bmatrix}$$

where the input signals, Vi $_{\rm i}$, and output signals, Vo $_{\rm i}$, are both normalized to 50 ohms.

To renormalize a generalized S-matrix to a specific impedance, the system first calculates a unique impedance matrix associated with the structure. This unique impedance matrix, Z, is defined as follows:

$$Z = \sqrt{Z_0}(I-S)^{-1}(I+S)\sqrt{Z_0}$$

where:

S is the $n \ge n$ generalized S-matrix.

I is an $n \ge n$ identity matrix.

Z $_{\rm 0}$ is a diagonal matrix having the characteristic impedance (Z $_{\rm 0}$) of each port as a diagonal value.

The renormalized S-matrix is then calculated from the unique impedance matrix using this relationship:

$$S_{\Omega} = \sqrt{Y_{\Omega}} (Z - Z_{\Omega}) (Z + Z_{\Omega})^{-1} \sqrt{Y_{\Omega}}$$

where:

Z is the structure's unique impedance matrix.

 Z_{Ω} and Y_{Ω} are diagonal matrices with the desired impedance and admittance as diagonal values. For example, if the matrix is being renormalized to 50 ohms, then

 Z_{Ω} would have diagonal values of 50.

Visualize the generalized S-matrix as an S-matrix that has been renormalized to the characteristic impedances of the structure. Therefore, if a diagonal matrix

containing the characteristic impedances of the structure is used as Z_{Ω} in the above equation, the result would be the generalized S-matrix again.

Z- and Y-Matrices

Calculating and displaying the unique impedance and admittance matrices (Z and Y) associated with a structure is performed in the post processor.

Characteristic Impedances

FEM Simulator calculates the characteristic impedance of each port in order to compute a renormalized S-matrix, Z-matrix, or Y-matrix. The system computes the characteristic impedance of each port in three ways-as Z $_{\rm pi}$, Z $_{\rm pv}$, and Z $_{\rm vi}$ impedances.

You have the option of specifying which impedance is to be used in the renormalization calculations.

PI Impedance

The Z $_{\rm pi}$ impedance is the impedance calculated from values of power (P) and current (I):

$$Z_{pi} = \frac{2P}{I \bullet I}$$

The power and current are computed directly from the simulated fields. The power passing through a port is equal to the following:

$$P = \frac{1}{2} \oint_{s} E \times H ds$$

where the surface integral is over the surface of the port.

The current is computed by applying Ampere's Law to a path around the port:

$$I = \oint_{l} H \bullet dl$$

While the net current computed in this way will be near zero, the current of interest is that flowing into the structure, I^- or that flowing out of the structure, I^+ . In integrating around the port, the system keeps a running total of the contributions to each and uses the average of the two in the computation of impedances.

PV Impedance

The Z $_{\rm pv}$ impedance is the impedance calculated from values of power (P) and voltage (V):

$$Z_{pv} = \frac{V \bullet V}{2P}$$

where the power and voltage are computed directly from the simulated fields. The power is computed in the same way as for the Z $_{\rm pi}$ impedance. The voltage is computed as follows:

$$V = \oint_{l} E \bullet dl$$

The path over which the system integrates is referred to as the impedance line, which is defined when setting up the ports. To define the impedance line for a port, select the two points across which the maximum voltage difference occurs. FEM Simulator cannot determine where the maximum voltage difference will be unless you define an impedance line.

VI Impedance

The Z $_{vi}$ impedance is given by:

$$Z_{vi} = \sqrt{Z_{pi}Z_{pv}}$$

For TEM waves, the Z $_{pi}$ and Z $_{pv}$ impedances form upper and lower boundaries to a port's actual characteristic impedance. Therefore, the value of Z $_{vi}$ approaches a port's actual impedance for TEM waves.

Choice of Impedance

When the system is instructed to renormalize the generalized S-matrix or compute a Y- or Z-matrix, you must specify which value to use in the computations, Z $_{\rm pi}$, Z $_{\rm pv}$, or Z $_{\rm vi}$.

For TEM waves, the Z $_{\rm vi}$ impedance converges on the port's actual impedance and should be used.

When modeling microstrips, it is sometimes more appropriate to use the Z $_{\rm pi}$ impedance.

For slot-type structures (such as finline or coplanar waveguides), Z $_{\rm pv}$ impedance is the most appropriate.

De-embedding

If a uniform length of transmission line is added to (or removed from) a port, the Smatrix of the modified structure can be calculated using the following relationship:

$$\begin{bmatrix} S' \end{bmatrix} = \begin{bmatrix} e^{\gamma l} \end{bmatrix} \begin{bmatrix} S \end{bmatrix} \begin{bmatrix} e^{\gamma l} \end{bmatrix}$$

where:

εγ

is a diagonal matrix with the following entries:

$$\begin{bmatrix} e^{\gamma_1 l_1} & 0 & 0 \\ 0 & e^{\gamma_2 l_2} & 0 \\ 0 & 0 & e^{\gamma_3 l_3} \end{bmatrix}$$

 γ = α + j β is the complex propagation constant, where:

 α i is the attenuation constant of the wave of port *i*. β i is the propagation constant associated with the uniform transmission line at port *i* *l* is the length of the uniform transmission line that has been added to or removed from the structure at port *i*. A positive value indicates that a length of transmission line has been removed from the structure.

The value of $\boldsymbol{\gamma}$ for the dominant mode of each port is automatically calculated by the 2D solver.

Equations

The sections below describe some of the equations that are solved in a simulation or used to define elements of a structure.

Derivation of Wave Equation

The solution to the following wave equation is found during a simulation:

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}\right) - k_0^2 \boldsymbol{\varepsilon}_r \boldsymbol{E} = 0$$

where:

E(x,y,z) is a phasor representing an oscillating electric field

k $_0$ is the free space wave number, $\sqrt[\omega]{\mu_0 \epsilon_0}$.

 $\boldsymbol{\omega}$ is the angular frequency, $2\pi f$.

 $\mu_r(x,y,z)$ is the complex relative permeability.

 $\boldsymbol{\varepsilon}_{r}(x,y,z)$ is the complex relative permittivity.

The difference between the 2D and 3D solvers is that the 2D solver assumes that the electric field is a traveling wave with this form:

$$\boldsymbol{E}(x,y,z,t) = Re[\boldsymbol{E}(x,y)e^{(j\omega t - \gamma z)}]$$

while the 3D solver assumes that the phasor E is a function of x, y, and z:

$$\boldsymbol{E}(x,y,z,t) = \Re[\boldsymbol{E}(x,y,z)e^{j\omega t}]$$

Maxwell's Equations

The field equation solved during a simulation is derived from Maxwell's Equations, which in their time-domain form are:

$$\nabla \times \boldsymbol{H}(t) = \boldsymbol{J}(t) + \frac{\partial}{\partial t} \boldsymbol{D}(t)$$

$$\nabla \times \boldsymbol{E}(t) = \frac{\partial}{\partial t} \boldsymbol{B}(t)$$

$$\nabla \bullet \boldsymbol{D}(t) = \rho$$

$$\nabla \bullet \boldsymbol{B}(t) = 0$$

where:

E (t) is the electric field intensity.

D (t) is the electric flux density, ϵ E (t), and ϵ is the complex permittivity.

H (t) is the magnetic field intensity.

B (t) is the magnetic flux density, μ H (t), and μ is the complex permeability.

J (t) is the current density, σ E (t).

 ρ is the charge density.

Phasor Notation

Because all time-varying electromagnetic quantities are oscillating at the same frequency, they can be treated as phasors multiplied by $e^{j} \mathbf{w} t$ (in the 3D solver) or by $e^{j} \mathbf{w} t - \mathbf{\gamma} z$ (in the 2D solver).

In the general case with the 3D solver, the equations become:

$$\nabla \times \boldsymbol{H} e^{j \, \omega t} = \boldsymbol{J} e^{j \, \omega t} + \frac{\partial}{\partial t} \boldsymbol{D} e^{j \, \omega t}$$

$$\nabla \times \boldsymbol{E} e^{j\,\omega t} = \frac{\partial}{\partial t} \boldsymbol{B} e^{j\,\omega t}$$

$$\nabla \bullet \boldsymbol{D} e^{j \, \omega t} = \rho e^{j \, \omega t}$$

$$\nabla \bullet \boldsymbol{B} e^{j\omega t} = 0$$

By factoring out the quantity $e^{j} \mathbf{w} t$ and using the following relationships:

$$\frac{\partial}{\partial t} \boldsymbol{E} e^{j\omega t} = j \boldsymbol{\omega} \boldsymbol{E} e^{j\omega t}$$

$$\frac{\partial \boldsymbol{H}}{\partial t} = j \boldsymbol{\omega} \boldsymbol{H}$$

Maxwell's Equations in phasor form reduce to:

$$\nabla \times \boldsymbol{H} = \boldsymbol{J} + j \boldsymbol{\omega} \boldsymbol{E}$$

 $\nabla \times \boldsymbol{E} = -j \boldsymbol{\omega} \boldsymbol{B}$

 $\nabla \bullet D = \rho$

 $\nabla \bullet \boldsymbol{B} = 0$

where B , H , E , and D are phasors in the frequency domain. Now, using the relationships B = μ H , D = ϵ E , and J = α E , Maxwell's Equations in phasor form become:

 $\nabla \times \boldsymbol{H} = (j\omega\varepsilon + \sigma)\boldsymbol{E} = j(\omega\varepsilon)\boldsymbol{E}$

for = 0 $\nabla \times \boldsymbol{H} = \boldsymbol{j}(\boldsymbol{\omega}\boldsymbol{\varepsilon})\boldsymbol{E}$

$$\nabla \bullet \varepsilon \boldsymbol{E} = \rho$$

$\nabla \bullet \mu \boldsymbol{H} = 0$

where H and E are phasors in the frequency domain, μ is the complex permeability, and ϵ is the complex permittivity.

H and E are stored as phasors, which can be visualized as a magnitude and phase or as a complex quantity.



Electric and Magnetic Fields Can Be Represented as Phasors

Assumptions

To generate the final field equation, place H in the equation in terms of E to obtain:

$$\boldsymbol{H} = \frac{1}{j\boldsymbol{\omega}\boldsymbol{\mu}} \nabla \times \boldsymbol{E}$$

Then, substitute this expression for H in the $\nabla \times H$ equation to produce:

$$\nabla \times \left(-\frac{1}{j \omega \mu} \nabla \times \boldsymbol{E} \right) = j \omega \epsilon \boldsymbol{E}$$

Conductivity

Although good conductors can be included in a model, the system does not solve for any fields inside these materials. Because fields penetrate lossy conductors only to one skin depth (which is a very small distance in good conductors), the behavior of a field can be represented with an equivalent impedance boundary.

For perfect conductors, the skin depth is zero and no fields exist inside the conductor. Perfect conductors are assumed to be surrounded with Perfect E boundaries.

Dielectric Loss Tangent

Dielectric losses can be modeled by assuming that the relative permittivity, ${}^{m{\epsilon_r}}$, is complex:

$$\hat{\varepsilon} = \varepsilon' - \varepsilon''$$

Expressed in terms of the dielectric (electric) loss tangent, $\tan \delta_{\epsilon} = \epsilon''_r / (\epsilon'_r)$

the complex relative permittivity, ${}^{m{\epsilon_r}}$ becomes:

$$\hat{\varepsilon}_r = \varepsilon_r' - j\varepsilon_r' \tan \delta_e$$

Magnetic Loss Tangent

Losses in magnetic materials can be modeled by assuming that μ_r is complex.

$$\hat{\mu}_r = \mu_r' - j\mu_r''$$

Expressed in terms of the magnetic loss tangent, $(\mu''_r)/(\mu'_r)$, the complex relative permeability becomes:

$$\hat{\mu}_r = \mu_r' \left(1 - \frac{j\mu_r''}{\mu_r'} \right) = \mu_r' (1 - j\tan \delta_m)$$

Definition of Freespace Phase Constant

Using the relationships $\epsilon = \epsilon_0 \epsilon_r$ and $\mu = \mu_0 \mu_r$, the wave equation being solved can be placed in this form:

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}\right) - \boldsymbol{\omega}^2 \mu_0 \boldsymbol{\varepsilon}_0 \boldsymbol{\varepsilon}_r \boldsymbol{E} = 0$$

Now, if the free space phase constant (or wave number) is defined as,

 $k_0^2 = \omega^2 \mu_0 \epsilon_0$, the above reduces to the following equation that the 2D and 3D engines solve.:

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \boldsymbol{E}\right) - k_0^2 \boldsymbol{\varepsilon}_r \boldsymbol{E} = 0$$

Setting up FEM Simulations

Setting up FEM Simulations

This section describes the process of creating and simulating a design with FEM.

Creating a Physical Design

You start with the physical dimensions of a planar design, such as a patch antenna or the traces on a multilayer printed circuit board. You can create a design in the following ways in ADS:

- 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 about converting schematics or drawing in Layout, see Schematic Capture and Layout documentation. For information on importing designs, see Importing and Exporting Designs.

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, as shown in the following figure:



For more information about the EM Setup window, see EM Setup Window Overview and Setting up EM Simulations.

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.
- Partitioning: You can configure a circuit simulation to use an EM-based model for a cell instance or parts by using the EM Cosimulation view and EM Model view. For more information, see EM Circuit Cosimulation.
- Substrate: You can open a predefined substrate file from ADS by selecting Substrate. For more information, refer Defining Substrates.
- 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, see Ports, Defining Ports, and Ports for FEM.
- Frequency Plan: You can add or remove frequency plans for your EM simulations by selecting Frequency Plan. For more information, see Defining a Frequency Plan.
- Output Plan: You can specify the data display settings for your EM simulations by selecting Output Plan. For more information, see Defining a Frequency Plan.
- Options: You can define the preprocessor, mesh, simulation, and expert settings by selecting Options. For more information, see Defining Simulation Options.
- Resources: You can specify the local, remote, and third party settings by selecting Resources. For more information, see Specifying Simulation Resources
- Model: You can generate an EM model by selecting Model. For more information, see Generating an EM Model.
- Notes: You can add comments to your EM Setup window by selecting Notes.

Defining Substrates

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

For specifying a substrate for your FEM simulation, see Defining Substrates in an EM Simulation.

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 in FEM Simulator. There are several different types of ports that you can use in your circuit, depending on your application.

For assigning ports in an FEM simulation, see Ports.

Defining the Frequency and Output Plan

You can set up multiple frequency plans for an FEM simulation. You can also specify the data display settings for your EM simulations by using the EM Setup Window.

For specifying a frequency and output plan, see Defining the Frequency and Output Plan.

Defining Simulation Options

You can specify global options, such as physical model, preprocessor, and mesh for FEM simulations. You can either use a predefined set of simulation options or or create a new set in Simulation Options of the EM Setup window. You can specify the following simulation options for an FEM simulation:

- Defining Preprocessor Settings
- Specifying Physical Model Settings for FEM
- Defining Solver Settings for FEM
- Defining FEM Mesh Settings

Setting up Local, Remote, or Third Party Simulation

You can run FEM 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.

For more information, see Specifying Simulation Resources.

Adding a Box or Waveguide

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

For more information, see Adding Boxes, Waveguides, and Symmetry Planes.

Running FEM Simulations

You set up a simulation by specifying the parameters of a frequency plan, such as the frequency range of the simulation and the sweep type. When the setup is complete, you run the simulation. The simulation process uses the mesh pattern, and the electric fields in the design are calculated. S-parameters are then computed based on the electric fields. If the Adaptive Frequency Sample sweep type is chosen, a fast, accurate simulation is generated, based on a rational fit model.

For more information, see Running FEM Simulations.

Viewing FEM Simulation Results

You can display the results of an FEM simulation by using the Data Display and Visualization window. You can display the various type of results from an FEM simulation such as S-parameters, radiation patterns (far-field plots) and derived antenna parameters.

For more information, see Visualizing 3D View before EM Simulation and Visualizing FEM Simulations.

Computing Radiation Patterns

Once the electric fields on the circuit are known, you can compute electromagnetic fields. The electric fields can be expressed in the spherical coordinate system attached to your circuit. For more information on radiation patterns, see Computing Radiation Patterns.

Example- Designing a Microstrip Filter

Example- Designing a Microstrip Filter

This section describes the process of designing a microstrip coupled-line filter.

Drawing the Circuit

The circuit is drawn as a schematic, then converted to a layout. To start the process of designing a Microstrip filter, you need to draw the circuit by:

1. Creating a new workspace.

- 2. Adding Microstrip components to the schematic.
- 3. 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 Workspace.

Adding Components to the Schematic

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

- 1. In the Schematic window, click the Palette List arrow. The Palette List drops down.
- 2. Scroll and select T-lines Microstrip __ from the Palette List.
- **3.** In the component palette, locate and click Mcfil. This selects the Libra Microstrip Coupled-Line Filter Section component.
- 4. Crosshairs and a ghost icon of the component appear as you move the cursor over the Schematic window. Position the cursor and click.
- 5. End the command either by clicking the Cancel Command button or by choosing *Insert* or the *arrow* button.

Editing the Component

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

- 1. To edit the component parameters, double click the component.
- 2. Select a parameter and set the values for these parameters:

- W = line width = 0.25 mm
- S = spacing between lines = 0.044 mm
- L = line length = 1.8 mm
- W1 = width of the line that connects to pin 1 = 0.25 mm
- W2 = width of the line that connects to pin 2 = 0.25 mm
- 3. Click OK to accept the edits and dismiss the dialog box.

Copying and Placing another Component

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

- 1. Click the filter component on the schematic to select it. A black outline appears around it.
- 2. From the Schematic window, choose Edit > Copy.
- 3. Choose Edit > Paste. ** Move the crosshairs so that they are directly over the left connector of the first component and click.
- 4. End the current command.

Adding Ports

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

With FEM, you can define additional characteristics to a port.

Generating the Layout

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

- 1. From the Schematic window, choose Layout > Generate/Update Layout. The Generate/Update Layout dialog box and a Layout window appear.
- 2. From the *Layout* window, choose Options > Preferences. Click the Layout Units _____ tab and set the Resolution to 0.001 mm. Click OK.
- **3.** In the Generate/Update Layout dialog box, enter -1.8 in the X field and 0.294 in the Y field. This step identifies the placement of P1 on the layout. This step is usually optional, but specifying the location will be helpful later when the box is added to enclose the filter. Click OK.
- **4.** A window displaying the results of the conversion is displayed. Click OK. A layout representation of the schematic appears in the Layout window. It should resemble the figure here.



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

Defining a Substrate

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

- A ground plane representing the top of the box
- A layer of air
- The microstrip filter traces
- A layer of Alumina
- A ground plane representing the bottom of the box.
- 1. 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 create a substrate.
- 4. Specify the library and view name and select a template.
- 5. Click OK.
- 6. Select the top layer in the Substrate Layers field. Specify the bounding area layer. The top ground plane is added to the substrate.
- **7.** In the Substrate Layers list, select *Free_Space*. Change the Thickness to 2846 um. This sets the layer of air to an appropriate thickness.
- 8. In the Substrate Layers list, select *Alumina*. Change the Thickness to 254 um.
- **9.** Keep the bottom ground plane, but select and Cut any other layers that may appear in the substrate.

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

You currently see four of the five substrate layers that you need. In this instance, the metal layer is automatically positioned between the Alumina and air layers.

Note the following points about this layer:

- Metal layers are identified by a dashed line.
- The word *cond* identifies the layout layer that is mapped to this position. Refer to the Layout window, and you will see that the microstrip circuit was automatically applied to the layout layer named *cond* during the translation process.
- The word *strip* defines the layer such that the microstrips are metal and what surrounds the microstrips on that layer is air or dielectric. Other choices are slot and via. These are described in the exercise in the next section.
- 11. Choose File > Save As. Specify the file name as filter and click Save. The substrate file *filter.slm* is saved as part of the project.

Adding Ports to the Layout

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

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

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

- 1. From the Layout window, click the pin icon. If necessary, set layer = cond. The ports must be on the same layout layer as the microstrip filter. Click OK.
- 2. Move the cursor over the open port on the left side of the layout and click. Move the cursor over the open port on the right side of the layout and click. End the command. The layout should resemble the figure here.



- 3. From the Layout window, select EM > Simulation Setup to open the EM Setup window.
- 4. Select Ports in the left pane of the EM Setup window.
- 5. If necessary, drag the window away from the Layout window so that both ports are visible. Click the connector P1.
- 6. Select TML.
- 7. Click the connector P2. Select TML.

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

Finishing the Box around the Filter

Boxes and waveguide enclosures are useful because they introduce vertical boundaries to the design. Adding a box enables you to look at box resonance, which can have a significant effect on S-parameters in a small band centered around the box resonance frequency.

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

- 1. From the Layout window, choose EM > Box & Waveguide > Add Box.
- 2. Type x, y coordinates for the left and right sides of the box. Choose Insert > Coordinate Entry. Type -4.5, -4.5 in the X,Y fields of the Coordinate Entry dialog box and click Apply.
- **3.** Type +4.5 , +4.5 in the X,Y fields of the Coordinate Entry dialog box and click Apply. The box appears as shown below (you may need to click View All to see the box). Click Cancel to close the dialog box.


Performing the Simulation

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

- 1. From the Layout window, choose EM > Simulation Setup.
- 2. Select FEM.
- 3. Select Frequency Plan in the left pane of the EM Setup window.
- 4. Select the Adaptive Sweep Type and set these parameters to the following values:
 - Fstart = 1 GHz
 - Fstop = 40 GHz
 - Npts = 50
- 5. Click Add.
- 6. Make sure that *Open data display when simulation completes* is enabled The S-parameters will automatically be plotted and displayed at the end of the simulation.

- 7. Click Simulate. The simulation will be performed and its progress and completion will be indicated in the status window.
- 8. Select EM > Show Most Recent > Mesh Generation Summary to view simulation details. You can also print the report.

Viewing Simulation Results

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





Guidelines For Optimal Performance of FEM Simulation

The default settings are a good starting point to get an optimal performance of the FEM simulation. The following guidelines can be used to further improve simulation performance. The usage of these settings is application specific.

Mesh and Solver Settings

Target Mesh Size

The default Target Mesh size in the Initial Mesh tab is by default set to Automatic. For the complex designs where the small detail of the design is not expected to have any meaningful impact on the electrical behavior, you can use the Minimal memory option to reduce the size of the original mesh. Going one level deeper, you can specify the value of the initial mesh size using the Use initial minimal mesh size setting under the Advanced tab.

In the following situations, the Use initial minimal mesh size setting can be useful:

- In case there are curved surfaces where the meshing process potentially over-meshes the structure. Here, use a value that is small enough to describe the curved shape. For example, choose a distance value that is equivalent to 15-45 degree resolution of an arc or a curved surface.
- For stacked conductors in an IC technology where the stacked metal layers are not strictly aligned. Here, set the Minimal Mesh size to a number that is smaller than the typical line width but larger than the misalignment distance.

Conductor And Edge Mesh

By default, no conductor edge mesh is applied, which means that refinement of the meshes near the edges of conductors is a result of the automated mesh refinement process. In some applications, for example, planar filter or planar patch antennas, seeding the initial mesh using an edge mesh can speed up the mesh convergence process:

- If the edge mesh is expected to be required on specific conductor objects in the geometry, you can use the object specific mesh setting.
- For designs where the width of the ports is representative for all the conductor widths in the design, the Automatic Conductor Mesh option can be used

For more details on how to use these settings, see Defining Mesh Settings for FEM.

Solver Settings

The Discretization Order in the Solver tab is by default set to 2nd order. For designs where there are large regions with low field values and small field variation, you can use the Mixed order setting.

Parallel And Distributed Simulations

The calculations of the frequencies in the frequency sweep can be parallelized . For more details, see Specifying Simulation Resources. Performance improvements to expect depend on the specific design. The designs requiring more frequency points typically see a bigger performance improvement when doing parallel frequency point calculations.

NOTE Parallel and Distributed simulation requires one or more Distributed Computing 8-pack licenses. One Distributed Computing 8-pack license allows to do 8 processes in parallel. Specifying a number **N** that is bigger than 8 requires multiple Distributed Computing 8-pack licenses.

Memory usage

The log files of a FEM simulation report the amount of physical memory that is used by the FEM simulation process. This is a change in EMPro 2017 versus earlier releases where virtual memory was reported.

If the memory required for a simulation job exceeds the amount of physical memory, the reported memory will be maximized by the amount of physical memory.

NOTE Simulation logs where during the mesh refinement steps, the reported memory no longer increases monotonically, is a sign that the simulation is exceeding the computer resources and that the simulation process is going into memory swap, leading to slower performance.

Getting Started with a Layout for EM Simulations

Getting Started with a Layout for EM Simulations

- Creating a Layout for EM Simulations
- Using the Cookie Cutter
- Defining Component Parameters
- Converting Layers between Strip and Slot Representation
- Example- Designing a Microstrip Line

Creating a Layout for EM Simulations

Creating a Layout for EM Simulations

You can create a layout in the following ways:

- Directly in a Layout window.
- From the Schematic window.
- Import a layout from another simulator or design system.

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

Creating a Layout from a Complete Schematic

To create a layout from a schematic:

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



By default, the layout will begin with P1, at 0,0, with an angle of 0 degrees. There is no existing layout, so the Equivalence (the layout component that corresponds to the starting component in the schematic) is shown as not created.

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

2. Click OK.

The Status of Layout Generation dialog box appears. It displays the number of designs processed, the number of items regenerated (created) in the layout, the number of items that are oriented differently in the layout than in the schematic, and the number of schematic components that were not placed in the layout.

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



Creating a Layout Directly

Launch Advanced Design System and create a workspace. To display a Layout window, choose Window > New Layout from the ADS main window, or choose Window > Layout from an open Schematic window.

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

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

Connecting Components with a Trace

As in a Schematic window, you can connect components without having them actually touch. In a Layout window, this is done by placing a trace between the component (in the same way a wire is used in the Schematic window). Either choose Component > Trace, or click the Trace button in the toolbar.

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

Shapes

In Layout you can insert shapes, using either Insert commands or toolbar icons: With each shape, you can either click and drag to place it, or define points by coordinate entry (choose the shape, then choose Insert > Coordinate Entry).

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

NOTE The final segment of polygons and polylines can be entered by pressing the space-bar rather than double-clicking the mouse.

Layers

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

Changing the Insertion Layer

There are many ways to change the insertion layer:

- In the toolbar, retype the name of the layer and press Enter.
- In the toolbar, click the arrow next to the layer name. Choose a name from the list of currently defined layers.
- Choose the command Insert > Entry Layer and select a layer from the list.
- Choose the command Options > Layers and select a layer from the list of defined layers in the Layer Editor dialog box.
- Choose the command Insert >

Change Entry Layer To , and click an object whose layer you wish to make the current insertion layer.

 Use the Layers window, that is opened when a Layout window is opened and select a name in the list of currently defined layers.
 Experiment with placing shapes on different layers. Remember to click OK to accept a change in a dialog box.

Copy to a Different Layer

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

Default Layer Settings

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

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

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

Adjust Grid Visibility and Color

- 1. In the Display area, choose Major, Minor, or both.
- 2. Choose the Type of display (Dots or Lines). You may have to zoom in to see the grid display.
- **3.** Click the colored rectangle next to the word Color, and choose the desired color for the grid. Click OK to dismiss the color palette.
- 4. Click Apply. Experiment with different settings

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

Adjust Snap and Grid Spacing

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

- 1. In the Spacing area, change the Minor Grid display factors for both X and Y. The larger the number, the wider the grid spacing.
- 2. Click Apply. Experiment with different settings. If a display factor makes the grid too dense to display, it is invisible unless you zoom in.
- 3. Now experiment with the Major Grid.

Adjust Pin/Vertex Snap Distance

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

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

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

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

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

Experiment with Snap Modes

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

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

	Snap Mode	Priority
You can restrict or enhance the manner in which	Pin	1
snap modes. This table lists the snap modes, and their priorities.	Vertex	2
	_ Midpoint	_
	Intersect	
	Arc/Circle Center	
	Edge	3
	Grid	4

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



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

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

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

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

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

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

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

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

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

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

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

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

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

All other snap modes have priority over grid snap mode.

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

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

Drawing Tips

This section provides suggestions and examples when drawing layouts to be simulated using Momentum or FEM. These examples take into consideration things that are necessary during the drawing phase to ensure that requirements for an EM simulation are met.

Using Grid Snap Modes

Working in Layout enables you to control all geometry precisely. The variety of available snap modes can help you draw and position shapes. As an example, if you are connecting shapes, you should have the vertex snapping mode turned on. You will also want to review snap mode settings prior to adding ports to a layout.

Snap grid spacing also helps control the positioning shapes. In general, snap grid spacing should be about 1/2 of the minor grid point value. For example, if the minor grid points are 1.0, then the snap grid spacing should be 0.5. This makes it easy to know how your geometry will snap into place.

For more information on snap modes and snap grid spacing, refer to Layout.

Choosing Layout Layers

The standard set of layout layers begins with the layer named *default*. Do not use this layer for drawing your circuit. The first valid layer for an EM simulation is *cond*. For more information on layout layers, refer to Schematic Capture and Layout.

Keeping Shapes Simple

In general, when drawing shapes, you should use a minimal number of vertices per shape, since this will make the mesh easier to compute. When drawing curved objects, consider using a relatively large value for Arc/Circle Radius (under *Options* > *Preferences* > *Entry/Edit*). This will minimize the number of vertices, and facets used to represent the shape.

Merging Shapes

Merging shapes is often useful for eliminating small geometry overlaps and can also, in some cases, result in simpler mesh patterns. For example, if you draw a layout using multiple polygons and you suspect there is some overlap, merging the polygons will prevent the system from returning an overlap error. To merge the shapes, select each shape that you want to merge, then choose *Edit > Merge*.

Viewing Port and Object Properties

Any item selected in a layout, including ports, shapes, or other components, has associated properties. One way to view the properties is to select the object of interest, then choose *Edit > Properties*. This method also enables you to change object properties. If you want to change properties that have been attributed to an object for an EM simulation, you should change these properties only through the EM Menu selections.

To view the properties of the entire layout, choose *Options > Info.*

Adding a Port to a Circuit

You can add a port to a circuit either from the Schematic window or a Layout window. The procedures below include considerations for adding ports to a circuit that will be simulated using an EM simulation. For more information, see Ports.

Considerations

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

- The components or shapes that ports are connected to must be on layout layers that are mapped to metallization layers that are defined as strips or slots. Ports cannot be directly connected to vias.
- Make sure that ports on edges are positioned so that the arrow is outside of the object, pointing inwards, and at a straight angle.
- Make sure that the port and the object you are connecting it to are on the same layout layer. For convenience, you can set the entry layer to this layer; the Entry Layer listbox is on the Layout tool bar.
- A port must be applied to an object. If a port is applied in open space so that is not connected to an object, Momentum will automatically snap the port to the edge of the closest object. This will not be apparent from the layout, however, because the position of the port will not change.
- If the Layout resolution is changed *after* adding ports that are snapped to edges, you must delete the ports and add them again. The resolution change makes it unclear to which edges the ports are snapped, causing errors in mesh calculations.

Using the Cookie Cutter

Using the Cookie Cutter

The Cookie Cutter feature enables you to crop only the required area from an original design. During cookie cut, a new cell view is created, which contains the cookie cut area around the selected objects. This feature is useful when you want to analyze the behavior of a specific area or nets in your design.

You can select nets/objects by using the Navigator tab or manually create a shape as a boundary box. Navigator is preferred for selecting the desired nets in HSD designs. This section provides information about how to create a desired portion of your board/design using cookie cutter.

Selecting Nets Using Navigator for Cookie Cutter

The Navigator tool allows you to view and work with all the objects such as traces, paths, rectangles, polygons, circles, pins, components, vias, or cell instances in a design (layout or schematic).

To generate a cookie cutter boundary based on the selected nets in a layout:

- 1. Display the Navigator tab. For more information about how to access the navigator, see Using the Navigator.
- 2. Select the required nets in the Navigator tab, as shown highlighted in the following figure. You can use the search feature in the Navigator to search for nets.





- 3. Select EM > Tools > Cookie Cutter. The Cookie Cutter dialog box is displayed.
- 4. Select a boundary type. For more information, see Specifying a Boundary
- 5. Specify the Cut proximity nets within distance to expand the cookie cutter boundary and include proximity nets. Setting the distance to zero will create a tight boundary around selected objects.

Cookie Cutter
Boundary
Туре
Convex Hull O Bounding Box
Cut proximity nets within
0.5 mm 💌
Temporary Drawing Layer
Boundary shape that is used for the layout processing will be temporarily drawn on this layer. Specify a layer which is not used in your design.
EEM_CUTTER:drawing
PC3-ODIMIM_01_07_IID:cell_1:layout
- Option
☑ Include instance bounding box for boundary creation
Cut instances if it's on the boundary
Highlight boundary for Preview
Keep boundary after cut
Preview Cut Cancel

- 6. Accept the Temporary Drawing layer, which is a boundary shape drawn temporarily on the selected layer.
- 7. Accept the default output layout. To create a new output design layout, Click
- 8. Expand the Option button and select the required boundary option.
- 9. Click Preview to verify your boundary shape.
- **10.** Click Cut. The cookie cut layout is displayed in a different cell called cell_1, which includes all the nets, component footprints, and component pins inside the cookie cutter.



Selecting Objects Manually for Cookie Cutter

You can manually create an arbitrary shape polygon around the original design for creating a new design using the Cookie Cutter feature.

NOTE To use a cookie cutter, you need to define a boundary layer, which is drawn for previewing and cutting operation. Note that cookie cutter is not applied to the shapes on a boundary layer. Therefore, you need to select a layer that does not include real patterns where you want to use this feature.

To create a cookie cut cell view:

- 1. Select EM > Tools > Cookie Cutter.
- 2. Create an arbitrary polygon shape on any unused layer and select it in the layout window.



- 3. Select the boundary type.
- 4. Specify the Cut proximity nets within distance.
- 5. Accept the Temporary Drawing layer, which is a boundary shape that will be drawn temporarily on the selected layer.
- 6. Accept the default output layout. To create a new output design layout, Click
- 7. Expand the Option button and select the required boundary option.
- 8. Click Preview to check boundary shape.
- 9. Click OK to create a cut boundary. Another layout window opens, which contains the area that was cut by the boundary.



Copying the Selected Design to a New Cell View To copy selected objects:

1. Select EM > Tools > Copy To Design.

🔁 Copy Selected To Design	×
Target Design PDN_example_lib:cell_3:layout	—
Copy Cano	el

- 2. Select the required objects in the layout window by manual selection, using the Navigator, or any other method
- 3. Click with to select a target layout.
- 4. Click Copy to copy selected objects to the layout.

Using the *Copy to design* feature, you can copy only those shapes and components that you have selected in the layout design. An error message is displayed, if you do not select any shape.

Example

In the following example, it is illustrated how to copy selected nets using the *Copy* to design feature.

- 1. Display the Navigator tab. For more information about how to access the navigator, see Using the Navigator.
- 2. Select the required nets in the Navigator tab, as shown highlighted in the following figure. You can use the search feature in the Navigator to search for nets.

- **3.** Right-click on the selected nets and choose Select shapes and component on Net.
- 4. Select EM > Tools > Copy To Design.
- 5. Click Copy to copy selected objects to the layout. The selected nets are copied to cell_2.

Cookie Cutter Options

This section describes the various options provided in the Cookie Cutter dialog box.

Specifying a Boundary

Cookie cutter is operated on a boundary that can be specified by selecting nets or selecting shapes. You can choose the following boundary types:

- Bounding Box: Represents the smallest rectangle that contains the selected nets or shapes.
- Convex Hull: Represents the smallest convex set that contains the selected nets or shapes.

You can select nets or shapes either manually or using the Navigator feature. This section provides information about how to create the cookie cutter design by selecting objects manually and using the navigator.

Specifying the Cut Proximity Nets within Option

The Cut Proximity nets within option sets the boundary away from the nets by a specified distance.

Temporary Drawing Layer

Temporary Drawing layer, which is a boundary shape that will be drawn temporarily on the selected layer. To use a cookie cutter, you need to define a boundary layer, which is drawn for previewing and cutting operation. Note that cookie cutter is not applied to the shapes on a boundary layer. Therefore, you need to select a layer that does not include real patterns where you want to use this feature.

Output

You can create a new layout for displaying the design that you are copying using Cookie Cutter. The New Layout window is displayed where you can specify the required library, cell, and view. The shapes and nets of the objects inside the cookie cut boundary is preserved. Objects on the boundary are converted to polygons during cookie cut operation. If the original design has instances, then the instances on or inside the boudary is preserved by default.

Options

The following options are provided by Cookie Cutter for specifying a boundary layer:

- Include Instance bounding box for boundary creation

- Cut instances if it is on the boundary: If you select this option, the instances with its box intersects with the boundary will be cut and the instance itself will be deleted. Otherwise, such instances are not displayed in the output design.
- Highlight boundary for preview
- Keep boundary after cut
- NOTE Cookie cutter works with objects that are selected by any type of selection mechanism provided by ADS, such as manual selection and Physical Connectivity Engine. Therefore, you can use the Cookie cutter, even if the Physical Connectivity Engine is switched off.

Edge and Area Pin Manipulation Utilities

The EM Tools menu also provides utilities for quickly creating and manipulating edge and and area pins. The following utilities are available in the EM > Tools menu:

- Create One Pin From Selection: Creates a single pin containing all selected shapes that are not yet part of a pin.
- Create Multiple Pins From Selection: Creates a different pin for each selected shape that is not yet part of a pin.
- Add Selection To Touching Pins: Adds each selected shape to a touching pin if such a pin exists and is unique.
- Add Selection To One Selected Pin: Adds each selected shape that is not yet part of a pin to the unique selected pin.
- Dissociate Selection From Pins: Dissociates each selected shape from its associated pin.
- Force Center As Snap Point For Selected Pins: Removes the dot shape from each selected pin that consists of a unique dot shape and at least one other figure.
- Force Explicit Snap Point For Selected Pins: Adds a dot shape at the snap point of pins without any dot shape.

Defining Component Parameters

Defining Component Parameters

You can create user-defined components from a layout window in ADS. You can insert these components 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 lookalike symbol. You can also create an EM Model to model a component by using the S-parameters generated by an EM simulation.

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

- You can directly include the component in a schematic. During circuit simulation from the schematic environment, a solver is called automatically to generate a model. The user-defined layout parameters and the relevant simulation parameters (e.g., model type, mesh density) can be set from the schematic page. The EM model consists of a built-in database mechanism that stores previously calculated Momentum simulation results. Therefore, after Momentum simulation is done for a certain layout component, no new EM simulations are required, unless there is a change in the parameters for this component.
- You can use the components as the starting point of Advanced Model Composer (AMC), for more details see Using Advanced Model Composer. AMC is a generalization of Model Composer, for details see Model Composer , enabling you to generate a parameterized electrical model for a Component from within the layout window. After defining and creating the component, 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 Circuit cosimulation feature, AMC is a 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 simulations that run in the background. No EM simulations need to be performed when the component is used in schematic. Using an AMC component in schematic is as fast as using any other ADS built-in component. The AMC components can be added to a standard ADS design kit for easy sharing between workspaces and users.

Components and EM Circuit Cosimulation

The Electromagnetic-Circuit cosimulation 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) component in a schematic. The symbol representing this component in a schematic can either be a black box or a layout lookalike symbol.

For more information, see Performing an EM Circuit Cosimulation with an EM Model View.

Editing 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, and circles). 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.

Tesign Parameters: 10	
Design Name Single_patch_lib2:cell_2_design1:layout	
Select Parameter	Create/Edit Parameter
	Name kk. Type Nominal/Perturbed 💌
	Nominal Value 9 mm Perturbed Value 8 mm
	Edit/View Perturbation
Add Cut Paste Update	
ОК Арріу	Cancel Help

- 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.

NOTE The values that are specified as Nominal and Perturbed value correspond to the parameter values later, when an EM component is created. The Nominal value entered in this dialog will be the default value for this parameter in the EM component.

The following section describes the Linear Stretch, Radial Stretch, and Rotation 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.

The value that you enter for deltaX or deltaY is not necessarily the same as (Perturbed Value – Nominal Value) in the Design Parameters dialog box. You can use the same values, but it is not compulsory. The deltaX and deltaY values are defined only internally, to move certain vertices of the layout. If deltaX or deltaY do not correspond with (Perturbed Value – Nominal Value) in the Design Parameters dialog box, a mapping will take place.

🔋 Edit/View Perturbation: 2 🛛 💽 🔀	
Parameter L1	
Select an object or port in the layout to apply the perturbation.	LinearStretch
Perturbation	perturbation
Type LinearStretch 🔽	
deltaX mil	
deltaY mil	(deltaX, deltaY)
Apply Clear All	
OK Cancel Help	

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.

The value that you enter for 'factor' is not necessarily the same as (Perturbed Value /Nominal Value) in the Design Parameters dialog box. It is probably the most practical to use the same values, so they correspond, but it is not compulsory. The 'factor' value is defined only internally, to stretch certain parts of the layout. If 'factor' does not correspond with (Perturbed Value/Nominal Value) in the Design Parameters dialog box, a mapping will take place.

📓 Edit/Vi	ew Perturbation: 1			
Parameter	r len			
Select an (to apply th	Select an object or port in the layout to apply the perturbation.			
Perturbat	ion			
Type Ra	adialStretch 🛛 💌			
centerX	0	mm		
centerY	0	mm		
factor	4			
Apply Clear All				
ОК	Cancel	Help		



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 and centerY.

The value that you enter for 'angle' is not necessarily the same as (Perturbed Value – Nominal Value) in the Design Parameters dialog box. It is probably the most practical to use the same values, so they correspond, but it is not compulsory. The 'angle' value is defined only internally, to rotate certain parts of the layout. If 'angle' does not correspond with (Perturbed Value – Nominal Value) in the Design Parameters dialog box, a mapping will take place.

📓 Edit/View F	Perturbation: 2	? 🛛	
Parameter L1			
Select an objec to apply the pe	t or port in the layou! rturbation.	:	
Perturbation -			
Type Rotatio	n 💙		
centerX		mil	1
centerY		mil	
angle		deg	(
Apply	Clear	41	
ОК	Cancel	Help	



Using Existing Components

You can set up a parameterized layout by using a combination of several built-in components. Use the Design Parameters dialog box (EM > Component > Parameters) to specify subnetwork parameters for the layout component. 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. A parameterized layout can be set up using a combination of several built-in components. You can define a subnetwork for hierarchical components only.

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

To specify subnetwork parameters for the layout component:

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

📓 Design Par	rameters:4							
Design Name	Single_patch_lib2	:Single_patch	:layout					
Select Param	eter			Create/	Edit Parar	meter		
em1				Name	em1			
				Туре	Subnetw	ork	*	
				Defau	ult Value	1	None	~
				in the	layout to	set the parar	t a compon meter value	,
Add	Cut	Paste	Update]				
ОК	:	Apply		Cance			Help]

- 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. For example, in the following figure, subnetwork parameter L1 is used to set parameter value L of the MLIN instance.

Coupled_line_filter [Coupled_line_filter_lib:Coupled_li	ine_filter:layout] * (Layout):1	. 🗆 🛛
	Libra 3-Conductor Microstrip Asymmet	tric Coupled Lines:1 🛛 🗙
TLines-Microstrip 🗹 🗹 🕂 🛄		>
🧳 😫 🔤 😴	ads_tines:MACLIN3 Instance Name (name[<start:stop>])</start:stop>	Parameter Entry Mode Standard 💌 🕂 »
Palette 6	CLin2	
	Select Parameter Subst="MSub1" W1=73 um W3=73 um S1=15 um S2=15 um L=L1 Temp=L1 WA=10.0 um WC=10.0 um WD=10.0 um	Equation Editor Tune/Opt/Stat/DOE Setup
Horoso Hourre Hourre Hourre Hourre	Add Cut Paste	Display parameter on schematic Component Options
	L : Conductor length	
	OK Apply Cancel	Reset Help
Edit Param: Enter component location	MACLIN3 CLin2 cond:drawing 40	5.000, 1000.000 260.000, 25.000 um

Converting Layers between Strip and Slot Representation

Converting Layers between Strip and Slot Representation

You can convert between Strip and Slot Layer representations by using the EM >Tools > Convert Strip <-> Slot utility. This opens the following dialog box that enables you to convert strip and slot layer representation:

Layout Substrates		
Source E_plane_lib:dsn_E_plane		
Destination E_plane_lib:dsn_E_plane	· •	
Layer Defining Conversion Area(s)		
Cutter Layer default:drawing		
Cutter oversize distance	0.0	
Layers To Process		
Source Layer	Destination Layer	
RF:drawing	no change	
RF:drawing	Update Destination Layer in Table	

The Convert Strip to Slot dialog box processes a complete layout in one go for all the layers. It consists of the following options:

- Select the source and destination substrate. You can define a new one by typing the name in the destination field or use an existing one by selecting from the drop-down list.

- Cutter oversize Distance: The cutter/conversion area for the layout is defined by all polygon shapes on the layer selected under Cutter Layer. You can use an oversize setting to avoid slivers.
- Select the layers to process in the conversion. In the Source Layer list, all the layers from the source substrate are displayed. In the Destination list, all the destination layers are displayed. If you select --no change -- in the Destination Layer, the source layer is retained in the destination.

You can select a source layer in the table and at the bottom fill in or select a destination layer. If not specified the layer purpose will be "drawing".

Example- Designing a Microstrip Line

Example- 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:

You can a	lso change this selection after th	e workspace is created.
Note: A P	DK is a type of library. All library	management commands also apply to PDKs.
	ADS Libraries Analog/RF DSP Site Libraries DemoKit DemoKit_Non_Linear MyKit Jser Favorite Libraries and PDKs	\$HPEESOF_DIR\oalibs\analog_rf.defs \$HPEESOF_DIR\oalibs\dsp.defs \$HPEESOF_DIR\examples\DesignKit\DemoKit\lib.defs \$HPEESOF_DIR\examples\DesignKit\DemoKit_Non_Linear\DemoKit_No c:\MyKit\lib.defs

- 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 Workspace.

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.

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 (¹). 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.

🖥 Libra Microstrip Line:5	
ads_tlines:MLIN Instance Name (name[<start:stop>]) TL1 Select Parameter Subst="MSub1" W=25.0 mil L=100.0 mil Wall1=1.0E+30 mil Wall2=1.0E+30 mil Temp= Mod=Kirschning</start:stop>	Parameter Entry Mode String and Reference
Add Cut Paste	Display parameter on schematic Component Options
Subst : Substrate instance name	
OK Apply Cance	I Reset Help

- 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.



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 covert the Microstrip Line design with step in width schematic:

- 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, 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 a Frequency Plan.



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.
- **5.** Select Substrate to display the substrate used for the simulation. For more information, refer to Defining Substrates.
- 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 Ports.
- 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.
- **9.** Select Resources to specify computer resources options. For more information, refer to Specifying Simulation Resources.
- **10.** Select Model/Symbol to generate an EM model and symbol. For more information, refer to Generating an EM Model.
- **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:



Substrates

Substrates

- Defining Substrates
- Sweeping Substrate Name and Mesh Density
- Adding Boxes, Waveguides, and Symmetry Planes
- Modeling Through-Silicon Vias
- Variables in Substrates and Materials
- Dielectric Substrate Modeling

Defining Substrates

Defining Substrates

A substrate defines the cross-section of a physical design. It is a pre-requisite for 3D viewing and/or EM simulations. 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. The substrate editor enables you to specify properties such as the number of layers in the substrate, select the materials, specify the height of each layer, etc. You can save the substrate definitions and use with other circuits.

A substrate consists of the following types of alternating items:

- Substrate Layer: This layer defines the dielectric media, ground planes, covers, air, or other layered material.
- Interface Layer: This is the conductive layer in between the substrate layers. Typically conductive layers are the geometries that are drawn on the layout layers in layout window. By mapping layout layers to interface layers, you can position the layout layers that your circuit is drawn on within the substrate.

Another way to look at the substrate is: Substrate provides a side view of your design (not to scale, for ease of illustration) while the geometries drawn on layout layers in layout window provides a top view of your design. They together give a 3D picture of your design. The top and bottom of the substrate either end with a Cover (Interface) or an infinitely thick Substrate Layer, as shown in the following figure:

		~	top
101.34		/	Demo_Nit3 0.12 micron
101.22	Via_Pass M2	/	Demo_Poly 1 micron
100.22	Via1_2 M1	/	Demo_Nit2 0.1 micron
100.12			Demo_Nit1 0.12 micron
100	Via_Niti mesa nigr demo_dun	my	Demo_GaAs 100 micron
0 micron	Evia		

To see if a substrate definition/feature is supported by an EM simulator, click here.

Opening the Substrate Editor

You can open the Substrate Editor window in the following ways:

- From the ADS Main window, choose File > New > Substrate and click OK.
- From the ADS Main window, select Library View tab. Right-click any library or cell and choose New Substrate.
- From the Layout window, choose EM > Substrate.
- From the EM Setup window, select Substrate in the left pane. Then, click New.

The Substrate Editor window is displayed, as shown in the following figure:

<u>File</u> <u>T</u> echnology	<u>E</u> dit <u>V</u> iew	Options Tools	<u>W</u> indow <u>H</u> elp							
D 📁 🗖	96.	il- 🗘 🧔 🗄								
Substrate Name:	dsn_LTCC_balur	n_fem			**	Alt Contract Stations Alt Stations Alt Alt Antenne Alt Antenne Alt Antenne Alt Antenne Alt Antenne Alt Antenne Alt Alt Alt Alt Alt Alt Alt Alt Alt Alt			•	Use right mouse context merus to add or delete substrate items. Select items on the substrate and view their properties below. Shortcuts in the Edit menu can be used to quiddy edit the next substrate item. Entre Substrate
			Image: second			Aliment Borten Borten Marten M				Bounding area layer: www.sea.org
Substrate Layer St	ackup		ø	×s	ubstrate Vias				0 ×	3
Туре	Name	Material	Thickness	Â.	Туре	Name	Тор	Bottom		▲
Cover		PERFECT_CON	0 um		Conductor Via	cond_2 (2)	Interface 14	Interface 15	cc	
Dielectric		AIR	140 um		Conductor Via	cond_3 (3)	Interface 12	Interface 13	cc	
Dielectric		AIR	10 um		Conductor Via	cond_4 (4)	Interface 10	Interface 11	cc	Select a substrate item to see
Dielectric		AIR	140 um	- 1	Conductor Via	cond_6 (5)	Interface 6	Interface 7	cc	more information about that item.
Dielectric		AIR	10 um		Conductor Via	cond_7 (6)	Interface 4	Interface 5	cc	
		AIP	140 um		Conductor Via	cond_8 (7)	Interface 2	Interface 3	cc	
Dielectric		MIN	140 4111		1					
Dielectric		AIR	10 um		Conductor Via	pcvia1 (24)	cond_1 (1)	Bottom Cover	pc	
Dielectric Dielectric Dielectric		AIR	10 um 140 um		Conductor Via Conductor Via	pcvia1 (24) pcvia2 (25)	cond_1 (1) Interface 15	Bottom Cover cond_1 (1)	pc pc	-

The key components of substrate editor are listed below:

- 1. Main Menu bar: Contains menu options to edit or create a new substrate.
- 2. Toolbar: Contains the most commonly used buttons.

- **3.** Substrate view: Displays 3D cross-section view of substrate stack with mask mappings, it has basic operations to edit the substrate definition.
- **4.** Layer table: Lists the layers included in the substrate to perform quick checks and edits.
- 5. Via table: Lists the Vias included in the substrate to perform quick checks and edits.
- 6. Status bar: Notifies about warnings or errors for the substrate.
- **7.** Properties panel: This panel, on the right, allows editing the properties of the currently selected item of 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.
- 3. Select an existing ADS substrate file from the drop-down list.
- 4. To open the Substrate window, click Open.

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:

New Substrate							
New Substrate Create a new substrate file in the specified library.							
Library: LTCC BALUN lib							
File name: substrate1							
Template: 25milAlumina 💌							
Create Substrate Cancel Help							

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

Saving a Substrate

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

NOTE A substrate definition file is saved with file extension *.subst* by default.

Substrate Editor provides three options to save:

- 1. Save: This option saves the changes in the current substrate.
- 2. Save As: The Save As command allows you to save the current substrate with a new name. Select the library from the *Library* drop-down list and type the File Name of the substrate. The specified substrate is created in the selected library and displayed in the Substrate Editor.

Save :	Substrate As	x
Library: File Name:	LTCC_BALUN_lib Newsubstrate1	•
	OK Cancel Hel	p

3. Save a Copy As: The Save a Copy As command allows you to save a copy of the current substrate. Select the library from the *Library* drop-down list and type the File Name of the substrate. A copy of the current substrate is created in the specified library.

Deleting a Substrate

This command removes the selected substrate file (.subst). If the substrate has been precomputed, the calculations remain in the database.

To delete a substrate:

- 1. Right-click the required substrate in the ADS Main window
- 2. Select the Delete option, as shown in the following figure:

	cell_1_em.dds						
🗄 🕒 cell_2							
substra	ata 1						
	Open	Г					
	Copy	Ctrl+C					
	Copy File						
	Rename						
	Delete						
	Filter View Show Library	Name					

Adding, Moving, and Deleting Layers

The Substrate View enables you to visualize the substrate stack and do basic editing. To add or delete an item in the substrate, right-click in the substrate view and select from the list of option displayed in the pop-up menu. After selecting the desired action, the properties associated with it are displayed in the right panel of the Substrate Editor.



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

To add a substrate layer:

- 1. Right-click a substrate layer in the *Substrate Name* list that has the same basic property (open boundary, closed boundary, or finite thickness) as the layer you are adding.
- 2. Click Insert Substrate Layer Above or Insert Substrate Layer Below to add a new layer.

Context menus

Right-click on a Substrate Layer and you may see some of the following menus depending on the interface position and properties:

- *Insert Substrate Layer Above* - Inserts a new substrate layer with an interface layer above the selected layer.

- *Insert Substrate Layer Below* inserts a new substrate layer with an interface layer below the selected layer.
- *Delete with Upper Layer* Deletes the substrate layer above the selected layer.
- *Delete with Lower Layer* Deletes the substrate layer below the selected layer.
- Map Conductor Via Inserts a new conductor via in the selected substrate.
- *Map Semiconductor Via* Inserts a new semiconductor via in the selected substrate.
- Map Dielectric Via Inserts a new dielectric via in the selected layer.
- *Move Up With Upper Interface* Moves the Substrate Layer and the Interface above it up, along with items on that interface.
- *Move Up With Lower Interface* Moves the Substrate Layer and the Interface below it up, along with items on that interface.
- *Move Down With Upper Interface* Moves the Substrate Layer and the Interface above it down, along with items on that interface.
- *Move Down With Lower Interface* Moves the Substrate Layer and the Interface below it down, along with items on that interface.

Depending on the position of the substrate layer you can move the layer up or down the stack. If the layer is either at the top or bottom, you can add cover above or below the substrate layer, as applicable.

Right-click on a Interface Layer and you may see some of the following menus depending on the interface position and properties:

- *Map Conductor Layer* Inserts a new conductor layer on the selected interface.
- *Map Semiconductor Layer* Inserts a new semiconductor layer on the selected interface.
- *Map Dielectric Layer* Inserts a new dielectric layer on the selected interface.
- *Insert Nested Substrate* Inserts a new Nested Substrate on the selected interface.
- *Delete Cover* Deletes the Cover leaving the adjacent Substrate Layer as an infinite thickness layer.

To unmap the already mapped item, right-click on a *Conductor Layer* or *Via* and select Unmap option from the pop-up menu.



Moving Conductor Layers

Use the left mouse button to move a Conductor Layer up or down to a different interface or to drag the item to the new location. This method also works for Semiconductor Layers, Dielectric Layers, and Nested Substrates.

Moving Vias

Using the left mouse button, drag a via on the upper or lower 1/3 of its body and you will be able to stretch the via so that it goes through more or less Substrate Layers. Dragging it from the middle of its body allows you to move the via up or down without stretching it.

To move a substrate layer:

- 1. Right-click a substrate layer that you want to move in the *Substrate Name* list.
- 2. Select one of the following options:
 - Move Up With Upper Interface
 - Move Up With Lower Interface
 - Move Down With Upper Interface
 - Move Down With Lower Interface

To delete a substrate layer:

- 1. Right-click a substrate layer.
- 2. Select Delete with Upper Layer or Delete with Lower Layer to delete a layer definition.

Using Layer and Via Table Views

Starting ADS 2017, the layer and via table views are introduced in the substrate editor. The two table views provides the following adequate functionalities:

- Enables you to cross-check and edit the essential data such as material and thickness.
- Selecting a via or layer on substrate automatically highlights the respective row in the table.

- Double-click the Material or Thickness to edit a particular layer or view from the table.
- You can change the table position from View > Change Table Position.
- You can view or hide the views from View > Layer Table or View > Via Table.

ostrate Layer Stackup			0	×s	Subs	trate Vias				0	×
Туре	Name	Material	Thickness	^		Туре	Name	Тор	Bottom	Material	-
Cover		PERFECT_CON -	0 um			Conductor Via	cond_2 (2)	Interface 14	Interface 15	cond_2	
Dielectric		PERFCTOR	140 um			Conductor Via	cond_3 (3)	Interface 12	Interface 13	cond_3	
Dielectric		cond_3	10 um			Conductor Via	cond_4 (4)	Interface 10	Interface 11	cond_4	
Dielectric		cond_4 cond_5	140 um		\mathbb{Z}	Conductor Via	cond_6 (5)	Interface 6	Interface 7	cond_6	
Dielectric		cond_6	10 um			Conductor Via	cond_7 (6)	Interface 4	Interface 5	cond_7	
Dielectric		cond_8	140 um			Conductor Via	cond_8 (7)	Interface 2	Interface 3	cond_8	
Dielectric		pcvia1 pcvia2	10 um		1	Conductor Via	pcvia1 (24)	cond_1 (1)	Bottom Cover	pcvia1	
Dielectric		AIR	140 um		1	Conductor Via	pcvia2 (25)	Interface 15	cond_1 (1)	pcvia2	
				T	11						

- You can sort the table using the gear icon (

Editing Substrate Properties

After adding a substrate layer, you can set the properties in the right panel of the Substrate Editor window.

Editing Properties for the Entire Substrate

Click in the background of the substrate to deselect any specific item. This allows you to edit the properties of the entire substrate on the right panel of the window.

Entire Substrate	
Bounding area layer:	<none></none>

You can define a bounding area layer, which is a layout layer, to specify an area delimiting the design. It specifies the extent of substrate layers, slot layers and covers for simulators operating in a finite simulation domain. This includes the finite element simulator and exporting to EMPro. It excludes Momentum, which continues extending these layers to infinity. All layout outside the bounding area layer is discarded.

All layout outside the bounding area layer will be discarded.

The bounding area layer can be selectively overridden on substrate and interface layers in their respective property sheet. Note that individual vias can only be discarded as a whole; having them simultaneously inside the bounding area layer of one substrate layer and outside the bounding area layer of another substrate layer in unsupported and leads to undefined behavior.

The top level bounding area layer definition does not descend into nested substrates.

However, the shared interface layer between a substrate and a nested substrate uses the Boolean OR of all defined bounding areas. In this respect, the absence of a bounding area layer definition in one of the substrates is being treated as if the entire infinite plane is to be used as bounding area. For the entire substrate, you may select one bounding layer that applies to the entire substrate, unless it is overridden on a specific interface or substrate layer.

- 1. Open the Substrate Editor window.
- 2. Select in the open area that is not part of the substrate. This will show properties of the "Entire Substrate" on the right.
- 3. Select a layer from the Bounding area layer drop-down list.You can also click Edit Layer 🚾 to open the Layer Definitions window if you need to add an additional layer.
- 4. Click Save.

After adding a bounding layers the substrate will appear to be cut on the right edge and will display the name of the layer that bounds each interface and substrate layer.



Note that individual vias can only be discarded as a whole; having them simultaneously inside the bounding area layer of one substrate layer and outside the bounding area layer of another substrate layer is unsupported and leads to undefined behavior.

The top level bounding area layer definition does not descend into nested substrates. However, the shared interface layer between a substrate and a nested substrate uses the Boolean OR of all defined bounding areas. In this respect, the absence of a bounding area layer definition in one of the substrates is being treated as if the entire infinite plane is used as a bounding area.

Editing Substrate layer Properties

The bounding area layer can be selectively overridden on substrate and interface layers in their respective property sheet. You can choose to have no bounding layer or inherit the bounding layer defined for the entire substrate or choose a different layer to bound this interface or substrate layer. Following are the properties that can be edited:

 Material - This property allows you to select the layer material from the Material drop-down list. Materials are defined in the Materials Definition dialog box. Click the ... button to open the Material Definition dialog box where you can define a new material. The defined material is added automatically in the Materialdrop-down list.



- Thickness - This property allows you to define the thickness of the layer. The units can be selected from the *Thickness* drop-down list.

Editing Interface Layer Properties

Select an Interface layer to define the interface layer as one of the following:

- Strip: A *Strip* allows the mapping of Conductor, Semiconductor, and Dielectric Layers, and the insertion of Nested Substrates.defines the objects on the layout layer as conductive, the area surrounding the objects is not, as shown in the following figure:



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

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





- Cover: A cover is available only for the top or bottom Interface. If it is enabled, you can also set the following properties:
 - 377 Ohm Termination Select this option to enable this termination. If this option is selected, you cannot specify Material and Thickness.
 - Material This property allows you to select the layer material from the *Material* drop-down list. Materials are defined in the Materials Definition dialog box. Click the ... button to open the Material Definition dialog box where you can define a new material. The defined material is added automatically in the *Material* drop-down list.
 - Thickness This property allows you to define the thickness of the layer. You can also select the units for thickness.

Some examples of using strips and slots include:

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

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

Editing Via Properties

Select a Conductor Via layer to edit any of the following properties:

Conductor Via					
Layer	pcvia3 (26) 🗸 🗸 🗸				
Process Role	Conductor Via 🗸				
Material	pcvia3 🔹				
Side surface roughness model	<none> •</none>				
Precedence					
Plated					
Thickness	micron 🔻				
Dielectric Material	▼				
Drag the top or bottom of a via to stretch it.					
Drag the middle of a via to move it up or down.					

- Layer Allows you to map the mask layer with layout layer from the *Layer* drop-down list. To add new layout layer click the ... button (next to Layer drop-down list).
- Process Role Allows to select a process role to s pecify the role that the layer represents in a design.
- Material Defines material property for the mask layer from the *Material* drop-down list. To add new material layer click the button (next to Material drop-down list).
- Surface roughness model Allows you to select Surface roughness model for the sides of the via.
- Precedence Precedence specifies which layout layer has precedence over another if two or more layout layers are assigned.
- Plated Allows you to specify a plated via. Select the *Plated* option and then specify the *Thickness* and *Dielectric Material*. A non-plated via is a cylinder filled with the specified *Material*. A plated via consists of a cylinder wall with a thickness and the center of the via has a dielectric, as illustrated in the following figure:



To see if a substrate definition/feature is supported by an EM simulator, click here.

Mapping a Layout Layer

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

To map a layout layer:

- 1. Open the Substrate Editor window.
- 2. Select an interface layer.
- 3. Select the Strip Plane or Slot Plane option.
- **4.** To map the layer as a via, right-click the substrate layer that you want the via to cut through.

5. Select either Map Conductor Via, Map Semiconductor Via, or Map Dielectric Via.

NOTE After mapping vias, it is recommended to convert the vias to rectangles or square vias in the layout design to reduce memory consumption.

- 6. To define the conductivity characteristics of this layer, see Editing Conductor Layer Properties.
- 7. If you have overlapping layers, you need to set overlap precedence. For more information, see Setting Overlap Precedence.
- 8. Click Save.

Unmapping a Layer

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

To unmap a layer:

- 1. Open the Substrate Editor window.
- 2. Select a mapped layout layer from the Layers list.
- 3. Right-click a substrate layer or the interface where the layout layer is mapped.
- 4. Click Unmap. A message box appears.
- 5. Click OK. This removes all layout layers assigned to this position.
- 6. To remap a layer, refer to the steps in Mapping a Layout Layer.

Editing Conductor Layer Properties

You can define a layout layer as a conductor or a semiconductor. If the layout layer is defined as a strip or via, you define the conductivity of the strip or via. However, if the layout layer is defined as a slot, it is not possible to define the conductivity of the metal around the slot. This is because a slot is assumed to be surrounded by a perfect conductor.

Select a Conductor layer to define any of the following properties:

- Layer Allows you to map the mask layer with layout layer from the *Layer* drop-down list. To add new layout layer click the ... button (next to *Layer* drop-down list).
- Only pins and pin shapes from layer Allows you to only map the pins and pin shapes into the substrate, but not the geometry.
- Material Defines material property for the mask layer from the *Material* drop-down list. To add new material layer click the ... button (next to *Material* drop-down list).
- Operation The operation transforms 2D shapes drawn on a mask into 3D objects. For example, select the proper expand operation to define the thickness of a conductor mask.

- Position Defines the position of the layer.
- Thickness Defines the thickness of the layer.
- Surface roughness model Allows you to select Surface roughness model at Top and Bottom.
- Precedence Precedence specifies the precedence of a layout layer over another layer, if two or more layout layers are assigned to the same interface or substrate layer and objects overlap. Precedence is used by the mesh maker so that objects on the layer with the greatest precedence number are meshed and any overlap with objects on layers with lesser numbers are logically subtracted from the circuit. If you do not set the precedence, and there are overlapping objects, a mesh will automatically and arbitrarily be created, with no errors reported.

To define conductor properties:

- 1. Open the Substrate Editor window.
- 2. Right-click an interface layer and select Map Conductor Layer or Map Semiconductor Layer. A conductor layer is added, as shown in the following figure:



3. You can set properties in the conductor layer panel, as shown in the following figure:

Conductor Layer						
Layer	cond_1 (1)					
Process Role	Conductor	•				
	Only pins and pin shapes from layer					
Material	PERFECT_CONDUCTOR					
Operation	 Sheet Intrude into substrate Expand the substrate 					
Position	 Above interface Below interface 					
Thickness	15 micron	•				
Angle	90 degrees					
Surface roughness model	Top <none></none>					
Precedence						
To move the layer up or down on the substrate, just drag it up or down.						

- 4. Select the required layer from the Layer drop-down list.
- 5. Select the Process Role to that s pecifies the role that the layer represents in a design.
- 6. Select a material from the Material drop-down list.
- 7. Select an option from the Operation panel. For more information, see Automatic 3-D Expansion for Thick Conductors.
- 8. Type the conductor thickness in the Thickness text box and select a unit.
- 9. Specify the surface roughness model.
- **10.** Set overlap precedence. For more information, see Setting Overlap Precedence.
- **11.** Repeat these steps for the remaining mapped layout layers.
- 12. Click Save.

Automatic 3-D Expansion for Thick Conductors

Conductors with finite thickness can be modeled using the 3D metal expansion feature. This feature automatically expands the mask of a conductor with finite thickness in the direction orthogonal to the layered medium, using the specified thickness of this conductor.

3D expansion is activated by selecting either the *Intrude into Substrate* or *Expand the Substrate* option. In both cases, an extra dielectric layer is included in the internal substrate model. This is done for each metal layer that is expanded. The thickness values of the dielectric layers in between the metal layers are not changed, which preserves the capacitance value between two conductors lying on top of each other in the substrates.

To select the automated 3-D Metal Expansion Substrate items:

- 1. Open the Substrate Editor window.
- 2. Select the cond layer.
- **3.** In the Conductor Layer panel, select the substrate layer or the interface where the layout layer is mapped.
- 4. Choose either Intrude into Substrate or Expand the Substrate.
- 5. Select the required position: Above Interface or Below Interface.

Operation	 Sheet Intrude into substrate Expand the substrate
Position	 Above interface Below interface

Simulations with automatic expansion require more simulation time and memory, but result in more accurate simulation results. Typically, when the width/thickness aspect ratio is smaller than a factor of 5, the effect of accounting for the finite thickness of the conductors need to be allowed in Momentum simulations.

Consider an example, where an extra dielectric layer is inserted (indicated with [new] in the figure), which in the case of an above interface expansion has the dielectric properties of the layer above the metal layer. In the case of a below interface expansion, the new layer has the material properties of the layer below the metal layer. The following figure illustrates the internal substrate model when using the Expand the Substrate option and selecting the Above Interface expansion for a conductor.



The following figure illustrates the internal substrate model when using the Expand the Substrate option and selecting the Below Interface expansion for a conductor.



If you select Intrude into Substrate, the above expansion is illustrated in the following figure:





Trapezoidal Cross-section Angle Parameter

If you activate 3D expansion by selecting either the *Intrude into Substrate* or *Expand the Substrate* option, you can also specify an angle for the conductor layer. For example, the following figure displays a conductor layer at 135 degrees:



Currently, only the Controlled Impedance Line Designer and components on the TLines-LineType palette support trapezoidal cross sections. The trapezoidal conductor shape is not yet supported by an EM simulator.

Angle Parameter Conventions

Angle Parameters

- W: Width of widest edge (≈ width of etch resist). The widest edge can be either the top edge or the bottom edge, either the edge at the interface or not.
- Lateral Etch = Δ W / 2, where Δ W = difference in width between bottom and top edge = Wbottom Wtop
- 1/Etch Factor = Lateral Etch / Thickness = $(\Delta W / 2) / T = \cot(\alpha)$

You can enter the angle (α) in Substrate Editor. The top and bottom width are derived from it.

In the Substrate Editor, only absolute values are shown and radio buttons replace the signs.



Substrate editor		Technology and Netlist				
Position		Thickness	1/Etch Factor	ΔW	α	
Above interface	Widest edge at interface	> 0	> 0	> 0	0-90 °	
	Rectangular		= 0	= 0	90°	
	Narrowest edge at interface		< 0	< 0	90-180°	
Below interface	Widest edge at interface	< 0	> 0	< 0	0-90 °	
	Rectangular		= 0	= 0	90 °	
	Narrowest edge at interface		< 0	> 0	90-180°	

Setting Overlap Precedence

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



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

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

To specify overlap precedence:

- 1. Open the Substrate Editor window.
- 2. Select one of the overlapping layout layers.
- **3.** Assign its order of precedence in the Precedence field. Either type a value directly into the field or use the arrow keys to select a value.

		 <u> </u>
Precedence	0	\$ 1-

- 4. Click Save.
- 5. Repeat these steps for the remaining overlapping layers.

Updating the Layer Bindings from a Substrate

The ADS Physical Connectivity Engine enables you to establish electrical interconnects based on polygon shaped layout artwork and performs interconnect information extraction. The Physical Connectivity Engine requires a proper definition of the shape and pin layer binding. The layer binding setup can be derived automatically from a valid Momentum or FEM substrate setup. For more information on the Physical Connectivity Engine, refer to Physical Connectivity Engine

- 1. Click Edit Layer (🔤) to open the Layer Definition window.
- 2. Click Update Layer Bindings to synchronize the layer bindings from the substrate setup. The Update Layer Binding dialog box is displayed.
- 3. Select the required substrate.

4. Click OK. An information dialog box is displayed to confirm that the layer bindings are updated.

The layer binding rules applied are:

- 1. All layout layers mapped at a specific dielectric interface (as STRIP or SLOT) or through a specific dielectric layer (as VIA) will bind.
- 2. Every via layer additionally binds to all strip layers just above, just below, or somewhere along the via.
- 3. The layer binding of layout layers that are not mapped will be cleared.

CAUTION SLOT layers require special attention! With this command, SLOT layers only bind to themselves or to other SLOT layers mapped at the same interface. The interconnect highlighting will not go through a SLOT layer. If you want a different behavior, you have to update the layer binding manually.

Editing Nested Substrate Properties

Select a Nested Substrate to edit any of the following properties:

Nested Substrate				
A nested substrate is another substrate that is placed within this substrate.				
Use a nested technology. This allows access to a library with a different technology.				
-Nested technology				
Nested technology	top0 🔹			
Nested Library	DemoKit_Non_Linear_tech			
Flipped	No			
Nested substrate	DemoKit_Non_Linear_tech:demo 🔹 🛄			
Alignment within nested substrate				
Bottom interface				
Top interface				
Bottom ▼ of strip, slot or via layer				
Offset 0	micron 👻			
Precedence increase 1				

 Use a nested technology - Check this if your nested substrate represents a substrate from a different technology. With this un-checked you may nest two substrates from the current technology.

- Nested Technology Choose a Nested Technology from the Nested Technology drop-down list. The chosen Nested Technology determines if the substrate appears flipped or not. Click the button to create or edit Nested Technologies.
- Nested Library This is the name of the library specified in the chosen Nested Technology. This is not editable.
- Flipped The shows if the nested technology is flipped. This is not editable.
- Nested Substrate Choose a Substrate from the Nested Library to specify the EM properties of the layouts that will be placed on layouts using this substrate.
- Alignment within nested substrate This chooses a location within the nested substrate that will be aligned with the interface of the current substrate. Choosing the *Bottom interface* of the nested substrate places the nested substrate on top of the current substrate interface. Choosing the *Top interface* of the nested substrate places the nested substrate on bottom of the current substrate interface. You may also choose a strip, slot, or via layer of the nested substrate that will be aligned with the current substrate interface. The *Bottom* or *Top* choice option is used to choose the bottom or top of the strip or via layer that was selected.
- Offset This allows you to move your Nested Substrate up or down relative to the interface it is on.
- Precedence increase Precedence specifies which substrate has precedence over another if two or more substrates are at the same location. The substrate with the higher precedence occupies the overlapped area.

See Nested Technology and Setting up Multi-Technology Designs for more information about using Nested Technologies and Nested Substrates.

Verifying Substrate Definition

After creating the substrate definition, choose File > Check (from the Substrate Editor window) to verify the created substrate definition. The verification result or errors (if any) are displayed in the Check Substrate message window.

Setting a Default Substrate

A default substrate can be specified for each library. This substrate can be any substrate in the library or from any library referenced by the technology of the library. A default substrate will be the initial value of the substrate name when any new emSetup views are created. A design kit can have a default substrate and any other libraries that uses the design kit technology will inherit the default substrate from the design kit.

The Set Default Substrate dialog can be opened from the Options > Set Default Substrate menu on the Substrate Window. It can also be open from the Set Default Substrate context menu on substrates in the ADS Main window.

Choose the default substrate for a specified library.				
Library				
MyLibrary1_lib				
Default Substrate				
Inherit from referenced libraries: <no default=""></no>				
OK Cancel Help				

The Library combobox specifies which library the default substrate is for. The Default Substrate combobox specifies the default substrate. The substrate choices include those in the specified library and any libraries referenced by the technology of the specified library. It also includes the choice to Inherit from referenced libraries. If this is chosen, the referenced libraries are searched for a default substrate specification and the first found will become the default substrate.

Importing a Substrate

You can import an SLM substrate file, substrate from database, substrate from Schematic, and an ltd substrate file. To import a substrate choose File > Import from the Substrate editor window or the ADS Main window. The following import options are available:

- SLM Substrate File: Select File > Import >SLM Substrate file.
- Itd Substrate File: Select File > Import > Itd Substrate file.
- Substrate From Database: Select File > Import >Substrate From Database.
- Substrate From Schematic: Select File > Import > Substrate From Schematic

	Momentum	FEM	PIPro	SIPro
Slot layer	Supported	Supported	NA	NA
Svensson/Djordjevic dielectric model	Supported	Supported	Supported	Supported
Plated via	NA	NA	Supported for PI-DC	NA
Surface impedance conductor model incl. skin effect (sheet/thick)	Supported	Supported	Supported	Supported

	Momentum	FEM	PIPro	SIPro
Field modeling of conductor (Meshed interior)	NA	Supported	NA	NA
Conductor roughness model (surface top/bottom)	Supported	NA	NA	Partially supported (traces only
Conductor roughness model (via sides)	Supported	NA	NA	NA

Sweeping Substrate Name and Mesh Density

Sweeping Substrate Name and Mesh Density

To support sweeps over a substrate name, you can add a cell parameter, *Substrate*, of the string type. Use the Substrate Editor to define and save different substrate setups to sweep through. Add a custom parameter to the design called Substrate by performing the following steps:

- 1. In the layout window, select File > Design Parameters.
- 2. Click the Cell Parameters tab.
- **3.** Add a parameter named Substrate, of type String, with a Default Value set to an existing substrate, using the "libraryName>:<substrateName>" format. Double quotes are also included.
- 4. Click Add.

🔁 Design Parameters:22					
Library: Sweep_Substr_lib Cell: xline1					
General Cell Definition Cell Parameters					
Select Parameter Style Temp Trise Tnom TC1 TC2 wBV InitCond Model Width Length _M Add Cut Paste	Edit Parameter Parameter Name Style Value Type String Default Value Narrow Optional Parameter Type/Unit String Parameter Description Capacitance Display parameter on schematic Optimizable Allow statistical distribution Not edited Not edited				
Copy Parameters From Note: Adding parameters does not automatically make the layout artwork parameterized. To make instances of a specific layout change based on parameters, please set the Artwork Type to 'Parameterized' on the layout's Customize Pcell dialog box.					
OK Save AEL file	Cancel Help				

5. Click OK.

For more information about how to define a parameter, see Defining Parameters. Similarly, for Momentum only, you can define MeshDensity as a cell parameter of the real type to support sweeps over the mesh density.

Setting up Substrate Sweep

You can instantiate the *xline1* cell in a testbench schematic and setup a sweep over the substrate name. By using the *StringList* component, which can be accessed from the *Simulation-Batch* palette, you can sweep over multiple substrates in an EM/Circuit co-simulation.

- 1. A StringList component is used to sweep through a list of substrate file names. This component consists of an index parameter that is used to return the filename with the associated index value.
- 2. Define variables for substrate name and index to sweep. Note the syntax for defining the substrate variable. The *Substrate* parameter expects a string value of the "<libraryName>:<substrateName>" format. The order in which these components are placed/defined in the schematic does not matter.

- 3. Set the component's Substrate parameter to the substrate variable.
- 4. Add a Simulation controller and a Parameter Sweep controller to sweep the index variable.



You can now view results in the data display window:



Substrate Thickness Sweep = 2, 12, 22 mil

You can also use an example for illustrating the sweeping substrate name by performing the following steps:

- 1. Select File > Open > Examples. The *Choose an Archived Example to open* dialog box is displayed.
- 2. Double-click Momentum and then open emcktcosim.
- 3. Select Sweep_Substrate_Parameters_wrk.7zap.
- 4. Click Open.

Adding Boxes, Waveguides, and Symmetry Planes

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 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.



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 losse 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.

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:

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



- 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.
- 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.

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 is you suspect that the presence of these sidewalls will have an immediate effect on the behavior of the circuit. For example, "broad-coupled filters" are placed in metal enclosures (a box) and the sidewalls can have an significant influence on the filter characteristics.
- 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 $\boldsymbol{\epsilon}$ and μ (make sure that $\boldsymbol{\epsilon}$ 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.

NOTE A symmetry plane is not supported by Momentum simulations.

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.

Modeling Through-Silicon Vias

Modeling Through-Silicon Vias (TSV)

Through silicon via (TSV) technology is a key enabling technology for 3D integration of System on Package (SoP). Modeling of the TSV interconnect array will be important for high speed or high frequency applications.



The TSV structure provides a conductive path through the semiconductor die between a bond pad on the front side and a land on the rear side. The conductive path is isolated from the semiconductor by a linear dielectric.



Modeling Theory

Important effects to model are resistance, self and mutual inductance and capacitance of the TSVs.



There are two effects that influence the capacitive behavior of the TSV.



The Liner Oxide Capacitance

The TSV oxide layer introduces an excess oxide capacitance that blocks the TSV metal - silicon bulk DC contact.

The MOS Depletion Capacitance

The TSV Metal - Oxide - Semiconductor (silicon) structure introduces an effect of accumulation, depletion or inversion of the major carriers in the silicon that depends upon the intrinsic semi-conductor properties, the substrate doping and

the TSV metal – silicion bulk bias voltage. The voltage dependent depletion layer in the silicon bulk changes the TSV oxide capacitance. The MOS structure yields a non-linear voltage dependent capacitive effect as shown below.



For a p-type substrate, when applying a negative bias voltage on the TSV, we get an accumulation of majority carriers at the TSV surface and therefore a high capacitance to the semiconductor bulk. As a result the total TSV capacitance is close to the oxide capacitance. When the negative voltage is reduced sufficiently, a depletion layer is formed which acts as a dielectric in series with the oxide and the total capacitance decreases. When further increasing the voltage, the capacitance goes through a minimum and then increases again as an inversion layer of minority carriers is formed at the surface. The increase of the capacitance depends on the ability of the minority carriers to follow the applied signal. This only happens at low frequencies where the recombination-generation rates of the minority carriers can keep up with the signal variations. Experimentally, it is found that for a metal-oxidesilicon system this happens only below 100 Hz. As a consequence, C-V curves measured at higher frequencies do not show the increase of capacitance. We obtain the C-V curves under depletion, where the capacitance flattens out as the voltage increases more.

Tayloring the fixed oxide charge density (Qf) during the TSV oxide fabrication process gives extra shift in the C-V curve. This enables to obtain minimum depletion capacitance in the desired operation voltage region (0-Vdd). Typical values for Qf depend on the TSV oxide deposition process:

- Wet oxidation: Qf = 5.1e10 cm-2
- SACVD (SubAtmospheric Chemical Vapor Deposition): Qf = 7.8e10 cm-2


References

- G. Katti, M. Stucchi, J. Van Olmen, K. De Meyer, and W. Dehaene, "Through-Silicon-Via Capacitance Reduction Technique to Benefit 3-D IC Performance", IEEE Electron Device Letters, Vol. 31, No. 6, June 2010.
- L. Zhang, H. Y. Li, S. Gao, and C. S. Tan, "Achieving Stable Through-Silicon Via (TSV) Capacitance with Oxide Fixed Charge", IEEE Electron Device Letters, Vol. 32, No. 5, May 2011.
- **3.** J. Sercu and T. Schwartzmann, "Modeling of Through-Silicon Via's (TSV) with a 3D Planar Integral Equation Solver", NEMO-IEEE 2014 Conference, Pavia, Italy, May 14-16, 2014.

Example workspace

\$HPEESOF_DIR/examples/EM/RF_Microwave/tsv_example_wrk

Modeling TSVs with Momentum

Momentum includes the effect of both the liner oxide and MOS depletion capacitances in the EM model without the introduction of extra unknowns in the matrix. The model does not require extra computational resources. There is no performance and/or capacity penalty and can be applied to cover large arrays of TSV's.

For the MOS depletion capacitance model, four depletion modes can be selected:

- No depletion (accumulation)
- HF depletion (default, high frequency behavior)
- LF depletion (inversion, low frequency behavior, below 100 Hz)
- Deep depletion

NOTE

The TSV liner oxide dielectric is not physically created for a Momentum simulation. Consequently, the 3D viewer does not show the silicon oxide coating on the TSV.

The TSV conductor radius is the radius of the through hole (the circle drawn on the TSV layer) minus the liner thickness! The Momentum mesh is located at the conductor surface.

Modeling TSVs with FEM

FEM includes the effect of the liner oxide capacitances only. The fixed oxide charge density and MOS depletion effects are ignored. Changing the corresponding parameters will have no effect on the FEM model.

NOTE The TSV liner oxide dielectric is created for a FEM simulation. Consequently, the 3D viewer shows the silicon oxide coating on the TSV.

Defining a TSV in the Substrate Editor

You can define a Through Silicon Via in the Substrate Editor similar to a Conductor or Dielectric Via.

To define a TSV:

- 1. Open the Substrate Editor.
- 2. Right-click on a substrate layer and select Map Through Silicon Via. A TSV is added.



- 3. Specify the Through Silicon Via attributes on the right side.
 - Shapes on the selected Through Silicon Via Layer designate where a hole through the silicon will be etched.
 - The Liner Thickness and Liner Material specify the oxide coating that is deposited on the side walls of the through holes
 - The remainder of the hole is filled with the conductive Through Silicon Via Material. The TSV conductor radius is the radius of the through hole (the circle drawn on the TSV layer) minus the liner thickness!
 - The depletion capacitance is influenced by the Depletion mode, the Fixed charge and Bias voltageparameters.

Besides the bias voltage, the doping type of the semiconductor material where the TSV passes through plays a role in the calculation of the depletion capacitance. Semiconductor materials have a doping property, p-type or n-type. The default is p-type.

Variables in Substrates and Materials

Variables in Substrates and Materials

Contents

- Overview of Using Variables in Substrates
- Defining Variables in a Defaults Design
- Using Variables in Substrates and Materials
- Simulation Flows
- Update EM Model Database when Parameter Set is Modified

Overview of Using Variables in Substrates

Overview of Using Variables in Substrates

You can use variables in a technology substrate after defining the variables in a defaults design. All variables defined in the defaults designs of a library and the defaults designs of referenced libraries are visible in the entire library, and can be used in the substrate and materials of that library.

You can replace real numbers in the substrate and material definitions with expressions that use the variables, such as layer thickness and angle, conductivity, permittivity, roughness, and TSV bias voltage.

The expressions will be evaluated when running a simulation. If the variables are swept, optimized, tuned or used in a statistical simulation, the substrate is evaluated for each iteration.

The different components and simulation flows that use technology substrates are:

- EM simulation from layout
- EM simulation triggered by EM Model view or EM Cosimulation view.
- Line Type components
- Controlled Impedance Line Designer

NOTE The variables defined in the defaults design are not only accessible in the substrate definitions, but also accessible and controllable from any design.

The following flowchart depicts the process of using variables in substrates and materials:



The following sections provide more information about how to use variables in substrates and materials:

- Defining Variables in a Defaults Design
- Using Variables in Substrates and Materials
- Simulation Flows
- Update EM Model Database when Parameter Set is Modified

Defining Variables in a Defaults Design

Defining Variables in a Defaults Design

Technology substrates are defined at the library level. Therefore, to use variables in a technology substrate, these variables must be defined at the library level. This is done by defining the variables in a defaults design. All variables defined in the defaults designs of a library and the defaults designs of referenced libraries are visible in the entire library, and can be used in the substrate and materials of that library.

WARNING To avoid unexpected results, use different names for variables in defaults designs, variables on normal schematics and cell parameters.

There are two methods to define variables in a defaults design:

- 1. Using the Library Variables Dialog
- 2. Manually creating a defaults design and placing variables on it like on any other schematic

Library Variables Dialog

The library variables dialog can be opened from and design window or the ADS main window from the menu Options > Technology > Variable Definitions.

In the substrate editor it can be opened by clicking on the VAR icon: VAR .

ڬ 🕢 Library Variat	les [Sweep_Subst	r_lib]											\odot	×
Show variables for libr	ary:					Sweep_S	ubstr_lib							•
Tuning Optimizati	on Statistics D	OE												
Name		Tune	Value	Unit	Format	Min/+-/+-%	Unit	Max	Unit	Step	Unit			
B-Sweep_Substr_lit VAR1 Fr Fr H Cond300 Cond300 Alpha T	o:defaults:schematic		3 25 4.1e7 0.00393 300		linear linear linear linear linear									
<u> </u>				-							<u> </u>			<u> </u>
			Add V	ariable De	elete Variable					Uncheck	C All OF	Cancel	Help	

To add a variable

- 1. Select the library.
- 2. Click on Add Variable.
- 3. Enter a name and value in the dialog that pops up.
- 4. If multiple defaults designs and/or multiple variable component instances exist, select one in the drop down list.

If no defaults design exists in the library, a new design will be created.

늘 💽 Add Vari	able 💿 🛇 🔿	\otimes
Name	н	
Value	25	
Design	<new 'sweep_substr_lib:defaults:schematic'="" defaults="" design=""></new>	•
Var Component	<new component="" var=""></new>	•
	OK Can	cel

Additional variables are added to the existing design and variables components.

0110

늘 💽 Add Vari	Add Variable (Er 3.3 gn Sweep_Substr_lib:defaults:sch Component VAR1		R
Name	Er		-
Value	3.3		
Design	Sweep_Substr_lib:defaults:sch	iematic 💌]
Var Component	VAR1	-]
	ок	Cancel	

To delete a variable:

- 1. Select the variable.
- 2. Click on Delete Variable.

Manually Creating a Defaults Design and Placing Variable Components

Creating variables at the library level without the Library Variable Dialog requires three steps:

- 1. Create a schematic.
- 2. Add the schematic to the defaults designs.
- 3. Place a variable on the schematic.

Creating a Defaults Design

To create a new schematic with a defaults design:

- 1. Click New Schematic Window in the Main window.
- 2. Select a library from the Library drop-down list.
- 3. Type a name in the Cell field, e.g., "defaults".
- 4. Accept the default view.
- 5. Remove selection from the Use Schematic Wizard option.
- 6. Click OK. A schematic is created in the cell.



Adding the Schematic to the Defaults Designs

After creating a schematic, add it to default designs:

- 1. Select Options > Technology > Defaults Designs. The Defaults Designs window is displayed.
- 2. Click Add Defaults Design. The Open Cell View window is displayed.
- 3. Select the new cell, which was created in the previous step.

W Open Cell View		×
Type: Schematic	Read-only	Show ADS libraries
Library:	Cell:	View:
LineDesigner_lib	defaults	schematic
LineDesigner_lib	Channel_Simulation ReadMe X_Talk_Simulation breakout_region cell_1 defaults	schematic
	ОК	Cancel Help

4. Click OK. The selected schematic is added to the Defaults Designs list, as shown in the following figure:

M Defaults Designs	? —————————————————————————————————————
View Technology for this Library: LineDesigner_lib	•
	Defaults Designs
	LineDesigner_lib:defaults:schematic
Components from these designs will be netlisted into the top design of any circuit simulation that uses a component from this library. Component instance names will be prefixed with " <libname>", where <libname> is the name of the library the component comes from.</libname></libname>	
Components in defaults designs should be pinless components, such as circuit substrate, model, and VAR components.	
	Add Defaults Design Remove Defaults Design OK Cancel

5. Click OK.

Placing a Variable on the Defaults Design

To place a variable on the defaults design:

- 1. Open the schematic in the defaults cell.
- 2. Click the Insert Var: Variable Equation button on the toolbar.
- 3. Place a variable on the schematic.
- 4. Rename the variable.
- 5. Specify a new value, as shown in following figure:



Using Variables in Substrates and Materials

Using Variables in Substrates and Materials

After placing a variable in the defaults design, you can use the variable in substrates and materials. All real numbers in the substrate and material definitions can be replaced with expressions that use the variables. For example, layer thickness and angle, conductivity, permittivity, roughness, and TSV bias voltage. The expressions will be evaluated when running a simulation. If the variables are swept, optimized, tuned or used in a statistical simulation, the substrate is evaluated for each iteration.

To see which variables are used in a substrate or its materials, click in the white area that surrounds the substrate. Double clicking on a variable in the table will open the Library Variables Dialog. Right clicking will open a context menu that leads you to the locations where the variable is used.



Using a Variable in a Substrate

To use a variable in a substrate:

- 1. Place a variable in a defaults design. For more information, see Defining Variables in a Defaults Design.
- 2. Open a substrate.
- 3. Select the required layer or via.
- 4. Change the required field with an expression using the variable. For example, in the following substrate, the thickness of the dielectric between ETCH_TOP and SLOT_L2_GND is changed to H mil. Note that the value of the variable H, itself, should not contain a unit. For example, H=5 will result in a thickness of 5 mil.



It is possible to fill in more complex expressions. Below is a list of valid example expressions (assuming all variables are defined)

- 2*H
- H + T

- ln(T)
- c0*f

In the last example c0 is a built-in constant (speed-of-light), not a variable.

Using a Variable in Materials

To use variables in a material:

- 1. Place a variable in a defaults design. For more information, see Defining Variables in a Defaults Design.
- 2. Open the Material Definitions window. This can be done in two ways:
 - a. Options >Technology > Material Definitions.
 - b. Select a layer in the substrate editor and click the Edit materials button (....) next to the Material drop-down list.
- **3.** Replace the value in the required field with an expression that uses the variable. For example, in the following figure the Permittivity field is replaced with the variable Er:

Μ	laterial Defini	tions						
iew	/ Technology fo	r this Library:	SubstVarDemo_lib		•			
F	Conductors	Dielectrics	Semiconductors	Surface Roughnes	SS			
			Material		Permittivity (Er)			
	Material Name		Libr	ary	Real	Imaginary	TanD	
Alumina		SubstVarDemo_lib		Er				
	M	Material Definit iew Technology fo Conductors Materia Alumina	Material Definitions iew Technology for this Library: Conductors Dielectrics Material Name Alumina	Material Definitions iew Technology for this Library: SubstVarDemo_lib Conductors Dielectrics Semiconductors Material Material Material Libr Alumina SubstVarDemo_lib	Material Definitions iew Technology for this Library: SubstVarDemo_lib Conductors Dielectrics Semiconductors Surface Roughnes Material Material Material Material Alumina SubstVarDemo_lib	Material Definitions iew Technology for this Library: SubstVarDemo_lib Conductors Dielectrics Semiconductors Surface Roughness Material Material Material Alumina SubstVarDemo_lib Er	Material Definitions iew Technology for this Library: SubstVarDemo_lib Conductors Dielectrics Semiconductors Surface Roughness Material Permittivity (Er) Material Name Library Real Imaginary Alumina SubstVarDemo_lib Er	

4. Click OK.

Using Variables before they are Defined

Instead of first creating a variable and then using it, it is also possible to first use a variable and then define it.

In the example below "H_al" is used in the thickness expression of the dielectric layer. Though, it is not defined yet as can be seen in the variable table.

Let Sub [Sweep_Substr_lib] * (Substrate):7 File Technology View Options Tools Window Help	\odot \odot
j 🗋 🚰 🔚 🕂 🖓 🥥 🖾 🚟 👯	
Substrate Name: sub	Use right mouse context menus to add or delete substrate terms. Select items on the substrate and view their properties below. Entire Substrate Bounding area layer: <pre> oversite view view view view view view view vie</pre>

Double clicking on it will directly open the Add Variable dialog of the library variables dialog, with the name of the variable filled in.

ڬ 🖸 🗠 Add Va	ariable \bigcirc \bigcirc \bigotimes
Name	H_al
Value	
Design	Sweep_Substr_lib:default
Var Componer	nt VAR1
	OK Cancel

Example: Using a Variable in a Material Parameter

Open the example workspace Sweep_Substrate_with_Variables_wrk from the following location:

\$HPEESOF_DIR/examples/EM/Cosim/Sweep_Substrate_with_Variables_wrk

We will create a new variable for the conductivity of the layer cond.

- 1. Open the substrate sub.subst.
- 2. Click on the layer mapped on the layer cond.
- 3. Click on the Edit materials button (....) next to the Material drop-down list.

늘 💿 sub [Sweep_Substr_lib] (Substrate):13		\odot \otimes \otimes
<u>File</u> <u>Technology</u> <u>View</u> <u>Options</u> <u>Tools</u> <u>Window</u> <u>H</u> elp		
] 🗋 📁 🔚 📫 🧔 🧔 🖾 🚟 👯		
Substrate Name: sub	Use right m items.	ouse context menus to add or delete substrate
	Select item	s on the substrate and view their properties below.
	Conduct	or Laver
	Laver	[cond (1)
	,	C Only pins and pin shapes from layer
	Material	cond
	Oneratio	Sheet C Intrude into substrate
	operade	C Expand the substrate
Emstaura		
riesjace	Position	Above interface Relow interface
H Alumina H mil		
0 mil	Thicknes	ss 0 micron I
	Angle	90 degrees
	Surface	Top <none></none>
	roughne model	Bottom <none></none>
		Assume metal fill is present
	Metal fill	Area fill fraction
		Area Intraction
	Precede	
	To move drag it u	the conductor up or down on the substrate, just p or down.
		10

4. Select the real part of the conductivity and copy the value to the clipboard using Ctrl-C

a	onig our	. 0									
10	Material Definition	s									8
Vie	ew Technology for this Libr	rary: Swee	p_Substr_lib]							
	Conductors Dielectric	s Semic	onductors Surface Roug	nness							
		Mater	ial		Los	ss Parameters			Permeability (MUr)		
	Material Name	1	Library	Parameter Type	Real		Imaginary	Real	Imaginary		-
	cond		Sweep_Substr_lib	ConductNity	4.1e+07	Siemens/m 🗵	0 Siemens/m	1			
									Add Conductor Add From Database Remo	we Conductor	d
E	Help								ОК Арр	dy Canc	el

5. Type "Cond" as value in the conductivity field.

Material Definitions View Technology for this Libra Conductors Dielectrics	ny: Sweep_Substr_lib Semiconductors Surface Ro	vughness					۵ ک	8
	Material		L	oss Parameters		Permeability (MUr)		11
Material Name	Library	Parameter Type	Real	Imaginary	Real	Imaginary		-
cond	Sweep_Substr_lib	Conductivity	Cond	Siemens/m 💌 0 Siemens/m	1			41
						Add Conductor Add From Database Remo	ve Conducto	2
Help						ОК Арр	ly Canc	el

- 6. Click OK.
- 7. Click in the white area that surrounds the picture of the substrate stack-up. We can see that "Cond" is not defined yet.

: ○ sub [Sweep_Subst_lib] (Substrate):7 e_Technology View Options Tools Window Help)					0 0
bistrate Name: sub	FreeSpace Aumina H mil	Use r items Selec Va	ight mouse cor tit items on the itire Substrate- unding area la iriables used in Name ond r	ntext menus to add c substrate and view t ver: <none> substrate: Default Value 25 Not defined 3</none>	er delete substrate heir properties belo Unit(s) mil Siemens/m
	×		Sel more	ect a substrate item information about th	to see nat item.

- 8. Double click on "Cond" or on "Not defined". The Add Variable dialog will open.
- 9. Select the Value field and paste the value using Ctrl-V.

ڬ 🖸 🗠 Add Vari	able 🔤 📀 🔗 🛞
Name	Cond
Value	4.1e+07
Design	Sweep_Substr_lib:default
Var Component	VAR1
	OK Cancel

- 10. Click OK.
- **11.** We see that the variable has been added in the Library Variables dialog.

v variables for library:					Sweep_Su	bstr_lib				
ning Optimization Statis	tics DOE				,					
ame	Tune	Value	Unit	Format	Min/+ -/+ - %	Jnit Max	Unit	Step	Unit	
Sweep_Substr_lib:defaults:so	hematic:									
-H Er Cond		25 3 4.1e+07		linear linear linear						
	_									

12. Click OK to save the changes.

늘 💮 sub [Sweep_Substr_lib] (Substrate):7			\odot \otimes \otimes
<u>File Technology View Options Tools Window Help</u>			
] 🗋 🚰 🔚 🕂 🤣 🥏 🖾 🚟 🎆			
Substrate Name: sub	FreeSpace Atumina H mil	Use right mouse context menus to items. Select items on the substrate and w Entire Substrate Bounding area layer: <none> Variables used in substrate: Name Default Va H 25 Cond 4.1e+07 Er 3</none>	add or delete substrate view their properties below.
0 mil	2	Select a substrate more information ab	item to see out that item.

Simulation Flows

Simulation Flows

The different components and simulation flows that use technology substrates are

- EM simulation from layout
- EM simulation triggered by EM Model view or EM Cosimulation view
- Line Type Components
- Controlled Impedance Line Designer

EM Simulation from Layout

At the start of an EM simulation, the substrate will be evaluated using the variable values assigned in the defaults designs.

WARNING It is not allowed to use a cell parameter in a substrate or material. If a cell parameter has the same name as a variable used in the substrate or material there will be an error message. Either the cell parameter or the variable will have to be renamed.

EM Simulation triggered by an EM Model View or EM Cosimulation View

Variables defined in a defaults design can be used in expressions in substrate and material parameters. By sweeping these variables, you can sweep over substrate parameter values in an EM/Circuit co-simulation. Also optimizations and statistical simulations are possible.

Creating and Using an EM Model View is done in the same way as for substrates that do not use variables.

The parameters of the EM Model database is the union of the cell parameters and the variables used in the substrate and materials. When placing an instance of a cell, it is required to only assign values to the cell parameters. The variables used in the substrate and materials are passed automatically to the EM Model. The Database and Interpolation tab of the EM Model will show which parameters are cell parameters and which are variables used in the substrate or materials.

EM C) cell_1 [e	emModelView_test	[lib:cell_1:emModel] (EM	Model) 📃 📀 🔗	\otimes
<u>F</u> ile	<u>E</u> dit <u>T</u> ools	<u>H</u> elp			
	89	୯ 🖪 🗊 🌲			
Sir	nulation S	etup Database	Interpolation Options]	-1
A	vailable da	ta			
		Cell Parameters	Variables in Subst/Mat		
	Name	L	Dl		
	data.000	125	1		
	data.001	125	2		
	Delete	Delete All			



It is not allowed to have a cell parameter with the same name as a variable on a defaults design.

Example

The example workspace examples/Momentum/emcktcosim /Sweep_Substrate_with_Variables_wrk.7zads illustrates how to sweep the thickness or the permittivity of a dielectric in an EM/Circuit co-simulation.

To open this workspace, select File > Open > Example and select Momentum /emcktcosim/Sweep_Substrate_with_Variables_wrk.7zads.

The substrate "sub" uses the variables "H" and "Er" as can be seen after opening the substrate editor. The dielectric thickness is "H mil" and the permittivity of the material Alumina is "Er".



"H" and "Er" are defined in the defaults design "defaults":

1	🛓 🕢 Library Variables [Sweep_Substr_	lib]										\odot	×
9	Show variables for library:					Sweep_Substr_lib							
	Tuning Optimization Statistics DOB	=]											
	Name	une 🕅	Value L	Jnit	Format	Min/+-/+-% Unit	Мах	Unit	Step	Unit			
	- H	3	25		linear								
		_ ·	5		intear								
	×												
-			Add Var	iable De	lete Variable				Unched	K All OK	Cancel	Help	

NOTE For more information about how to place variables, see Defining Variables in a Defaults Design.

The schematic "Sweep_Substrate_Thickness" has a parameter sweep component that sweeps "H" from 2 to 22 with a step of 10. The variable "Er" is not swept and thus keeps its value from the defaults design, i.e. 3.

			· ·		· ·	1.1	
Term Term1 Num=1 Z=50 Ohm	xline1 emModel xline1_1	· · ·	· ·	· · ·	 		erm erm2 lum=2 /=50 Ohm
₹	· · · · ·	· · · ·	· ·	· · · ·		1 	
		S-PAR	AMETE	RS.			
ParamSweep Sweep1	S_Pai SP1	ram					
SweepVar="H" SimInstanceName[1]="SP1"	Start= Stop=	1.0 GHz 2.0 GHz	· ·	 	· ·	• •	
SimInstanceName[2]= SimInstanceName[3]=	Step=	0.1 GHz					
SimInstanceName[4]= SimInstanceName[5]=				· · ·			
SimInstanceName[6]=						• •	
Stop=22 Step=10							
						· ·	

The cell "xline1" does not have cell parameters. It uses the substrate "sub". As a result, "H" and "Er" will become the parameters of the EM Model database.

The schematic simulation will trigger an EM simulation for each value of "H". The results are stored in the database of the EM Model and picked up by the circuit simulator.

EM C) xline1 [Sweep_Subs	tr_lib:xline1:	emModel 🖂 🔗	\otimes
<u>F</u> ile	<u>E</u> dit <u>T</u> ools	<u>H</u> elp			
	89	۴ 🖪 🕯] 🌲		
Sir	nulation S	etup Datak	oase Interp	olation Options	
A	vailable da	ita			
		Variables ir	n Subst/Mat		
	Name	Н	Er		:
	data.000	2	3		
	data.001	12	3		
	data.002	22	3		
	Delete	Delete All			

Substrate Thickness Sweep = 2, 12. 22 mil



The schematic "Sweep_Substrate_Dk" is very similar. Here, "Er" is swept from 2 to 6 with a step of 2 and "H" is kept constant at 25.

Line Type Components

With respect to variables in substrates, Line Type components are very similar to components with an EM Model. The variables used in the substrate are passed automatically to the simulation model. The Line Type component parameters are the same as for a fixed substrate.

Example

The example workspace examples/HSD/ControlledImpedanceLineDesigner_wrk. 7zads illustrates how a channel simulation with Line Type components can be performed with varying substrate and material parameters. To open this workspace go to the ADS Main Window and choose File > Open > Example and select HSD/ControlledImpedanceLineDesigner_wrk.7zads.

The folder "03_Pre_Layout_Substrate_Variables Channel Design" contains two example schematics.

- 1. 'Channel_Simulation_Substrate_Variables' analyzes a single-ended serial channel. The thickness of the microstrip's dielectric is varied to analyze the impact on the eye opening at the pre and post equalization stages.
- 2. 'X-Talk_Simulation_Substrate_Variables' analyzes the same channel with 2 aggressor microstrip lines running in parallel. Tuning of the line spacing, dielectric thickness, Er and tanD allows to quickly see the impact on the eye opening.

The substrate "6_Layer_Substrate_With_Variables" uses the variables "H", "Er" and "tanD" as can be seen after opening the substrate editor. The dielectric thickness between ETCH_TOP and SLOT_L2_GND is "H mil" and the permittivity and loss tangent of the material FR_4_Var are "Er" and "tanD", respectively.



The schematic "Channel_Simulation_Substrate_Variables" has a Batch Simulation component that varies "H" from 2.8 to 4.8 with a step of 0.5. The variables "Er" and "tanD" are not swept and thus keep their value from the variable definition, i.e. 4.3 and 0.03 respectively.



The variables used in the substrate and materials are not visible on the schematic in contrast with the line_width variable. To shortest way to access them is by opening the Simulation Variables Setup: Simulate > Simulation Variables Setup....In this dialog the value and tuning, optimization, ... settings of all variables and component parameters can be set.

ne	Tune	Value	Unit	Format	Min/+-/+-% Unit	Мах	Unit	Step l	Jnit
ineDesigner lib:Channel Simulation Substrate Variables:scl	nematic								
⇒ TL2	_								
Length		2	in	linear					
the SingleEnded1									
H- RX_SingleEnded1									
- VARL		E mil		lineer					
		5 1111		IIIIeai					
length		2	in	linear					
the Channel Sim1		-		micai					
ineDesigner lib:breakout region:schematic									
TLD1									
ineDesigner lib:defaults:schematic									
÷-VAR1									
н	v	3.8		linear	2	5		0.2	
Er		4.3		linear	4	5		0.1	
1		0.03		linear	0.02	0.05		0.01	

The two LTLINE1 components model a microstrip with a signal line on ETCH_TOP and SLOT_L2_GND as ground plane. This is shown in the component parameter dialog and in the Line Type edit dialog. We can observer that Thickness_2 is "H mil"

🔛 💮 Line Type 1	ransmission Lin	e:2			\odot \odot \otimes
ads_tlines:LTLINE1	Instance Name				
TL1					
Setup Display					
Line Type 🛛 🔜 L	ineDesigner_lib:T0	P_MS_Substrat	e_∨ariables		Edit New
Length 2					iņ
Substrate Line	Designer_lib:6_La	yer_Substrate_V	Vith_Variables		
+ .					<u> </u>
	AIR				
3.8+(H)	CH .				
	514			Thickness_1	2 + 1.8 mil
н		1			ETCH_TOP
	FR_4_Var			Thickness 2	Hmil
				_	
0 mil	ER / Var				SLOT_LZ_GND
Nicke of the second of	in in motor and a stimu			Thickness 3	5 mil
failed. Variables	defined outside t	actual dimensio ne dialog are not	ns because the t evaluated.	evaluation for so	me of the parameters
Use values	from Line Type				
Width	Spacing	Spacing Type	Clearance		
line_width					
+1 Advanced (Options				
ОК		Apply	C	ancel	Help
	-				

The schematic simulation will trigger a channel simulation for each value of "H". The results are displayed in the datadisplay as below. We see how the dielectric thickness modifies the eye size.



The second example "X_Talk_Simulation_Substrate_Variables" is very similar. Here, the substrate and material variables are tuned in addition to the spacing between the central signal line and the two aggressors.

													•																																																1								
													•																																												*	~	~	~	╢	l.							
												XtI	k2	Si	ģ	εĒ	nď	be													bn	eak	ou	U P	eg	jor																						Теп	m_	Si	ngle	eEr	Ide	t i					
												Xtl	k2_	Si	ngl	эEr	nde	d1														Г	00	ĥ																								Теп	m	Si	ngle	eEr	Ide	11					
												Ph	ase	To	Τx	100	;e=	Ra	ndo	m.							1		_		-	P1		P3	-	-	٦																					ĻOa	ia=	= QU	ų Ol	лщ							
													<	÷	÷	1					2						•				1	5	-	-			÷			÷			-																		1			_		1	÷		
		1										X	tik	≻	-	-	-	_		-	Ł	1		⊢	-		_				bn	eak aak	ou	닕	eg	jor	1		-	Ŀ	1		÷	-													2			4	_	3	۶Ŀ		Eye	- 2	rob	е.	
-		Г	~	1											•	1					Г	_		1.1						-	-	Г		h			-			Г			٦	÷	1												Γ	~	÷.		6			-		-	4		
•		1	Tx	>	-											-	-	_		-	÷	2		F	-				-		-	P1		P2	_	-	-		-	÷	2		÷	-												-	R	×	≻		1								
•		L	/		÷	•							-	2	1	1		•	1		늘									-		5	-	1			•			늘			4	•	1										•		L	/	1	•	1								
-			Tx	_Sir	ngle	Eng	bet					X	lik	≥	-	-	-	_		_	Ŀ	3		⊢	_	_		11			bn	eak aak	ou ou	님	eg	301	11			Ŀ	3	1	÷	-	_	_	_										Rx	Sir	ngl	eE	nde	ed .							
-			- Ex	_Sir	Igle	Ene	led'					/		1	1	1	1		1		E	TLI	NE	3			1					ĩ		ň			1			Ľ	FLI	NE	3	÷	1	1	1										Rx	Sir	ngl	eE	nde	ed 1							
-			VE	indh:	e=1 =1 (0 V	pps					Xtl	k2_	Si	١ġ١	эÉ	nde	be			T	L3-						_			-	Ľ	÷.	P3			_		Ľ	, FI	L4			1													EN En	able	IC.	oa n.t	a=r E=n	10. M							
-			vi	ow=	0.0	v						Xtl	k2_	Si	ŋgl	εEr	nde	d2	1		- î	ub; enc	st=. sth=	12 i	eL n	les	gr	ier	-"	0:0	-L	aya	-		DSI	ra	e_	vyr	n_	va L		ste ath	SLI =2	in	De	sig	ne.	-	0:0	La	yer	-54	DSI	rate	9_V	yiu	En	able	FF	Ê.	no	~ ·							
-			Ri	séFa	allT	'nе	≐15	pse	ec (Ph	ase	10	DX	100	je=	Ra	ndo	m	ŭ	ine	Tvp	e='	Lir	ne[) es	sia	ne	r li	b:1	Ó	'n	лś	3	Su	bs	rat	e'،	Vali	iat	Ter	pe:	÷	ine	De	sic	ine.	r lil	b:T	OP	MS	33	Sut	bstr	ate	E٥	ahla	R	FF	=yie	s i							
-			M	ode:	=Ma	xim	al L	eng	th I	FS	ŝR		1		1	1	1				S	[1]	=lin	e_ś	spa	icir	ŋġ	Ť		Ξ.			T							S	[1]	=lir	ie_	Śр	ac	inģ	1										Τ.												
					de	200	= no	EI		ne					1	1					s	[2]	=lin	e_s	spa	icir	ŋg													S	[2]	=lir	ne_	sp	ac	ing												÷.,	<u>.</u>		÷.	. 1							
1			Ļ	2 ivio	ųo.	φp	șen;		1 10	pa			•		1																																				_	_	_	_	_	_	*	~~	~	~		U.							
																																																										Ten	m_	Si	ngle	eEt	Ide	1.1					
																																																										Torr						12.					
													•				-		-	- 1	- 1			_	ς.						4.		Ċ,																									Lon	m_ 	50			ICIE						
																			ŀ	Ch	anir	nel	Sin	n '	ì				I	Var Togn	j	/AF	2.																									Loa	m_ ad=	50	QI	hm							
	-												•				6		ľ	Ch	anir	nel	Siń	n]				(Var Togn	: קיני ייני	/AF /AF	۲. ۲. ۲.		in	g=		mil	{t}																			Loa	m_ ad=	50	O	nm							
	-												•					Chu		Ch	anir	neľ	Siń	n .)				(Var Togn	ן ני	/AF /AF ne	₹. 21 _st	bac	in	g=	10	mil	(†)																			Loa	m_ ad=	50	QI	שם חות							
	-												•					Cha Cha Cha	ann ann de=	Ch elS elS Bit	ani im im	neľ	Sin	n .]				(Var Togn	ן ני	/AF /AF	2. 21 _st	bac	in	g=	0	mil	(†)																			Loa	m_ ad=	50	QI	שם חודר יייייייייייייייייייייייייייייייייייי							





Controlled Impedance Line Designer

The nominal value of variables in the Controlled Impedance Line Designer is initialized by evaluating the expressions in the substrate and materials.

NOTE The **Update Substrate/Material Parameters** option will only fill in constant numbers in the substrate and materials. It will not modify the value of variables in defaults designs.

Update EM Model Database when Parameter Set is Modified

Update EM Model Database when Parameter Set is Modified

The EM Model database items are indexed with the values of the cell parameters and the variables used in the substrate. When adding a variable to the substrate, or removing one, the database of the EM Model needs to be updated to correspond with its new parameter set. This is possible without loosing all the simulation results stored in the database. This will be illustrated with a simple example.

Open the following example:

Examples/Momentum/emcktcosim/Sweep_Substrate_with_Variables_wrk.7zads

This example contains a substrate that depends on the variables H and Er. After running the simulation in the schematics Sweep_Substrate_Dk and Sweep_Substrate_Thickness, the EM model is populated as shown in the following figure:

EM (🖸 xline1	[Sweep_Sub	str_lib:xline	1: 💿 🔿 🛛 🛞
<u>F</u> ile	<u>E</u> dit <u>T</u> oo	ls <u>H</u> elp		
	89	9 🖪 🎍	= 📶 🌲	
Sin	nulation Se	etup Datak	oase Interp	olation (
A	vailable da	ita		
		Variables ir	n Subst/Mat	
	Name	Н	Er	
	data.000	25	2	
	data.001	25	4	
	data.002	25	6	
	data.003	2	3	
	data.004	12	3	
	data.005	22	3	
	Delete	Delete All		

By sweeping Er, you can analyze the impact of variations in the permittivity of the dielectric material.

ЛЛ	Material Definiti	ons									\odot	\otimes
Vie	ew Technology for this I	Library: Sweep Substr lib										
	Conductors Dielect	trics Semiconductors Surf	ace Roughr	ess								
		Material		Permittivity (B	Er)	Permeab	ility (MUr)		Djordjevic]
	Material Name 🛆	Library	Real	Imaginary	TanD	Real	Imaginary	Туре	TanD Freq	Low Freq	High Freq	
	Alumina	Sweep_Substr_lib	Er		0	1	0	Frequency Independent				
	FreeSpace	Sweep_Substr_lib	1		0	1	0	Frequency Independent				
	1											-
								Add	Dielectric Add Fi	rom Database F	lemove Dielectric	
	Help									ОК	Apply Cance	el

Removing a Variable

If you do not want to analyze the impact due to variations in the material parameter Er, replace the value of the permittivity of the material with its fixed value.

Л٨	Material Definiti	ions									\odot	×
Vie	ew Technology for this	Library: Sweep Substr lib	×									
	Conductors Dielectrics Semiconductors Surface Roughness											
		Material		Permittivity (B	ir)	Permeat	oility (MUr)		Djordjevic			
	Material Name 🗡	Library	Real	Imaginary	TanD	Real	Imaginary	Туре	TanD Freq	Low Freq	High Freq	
	Alumina	Sweep_Substr_lib	3		0	1	0	Frequency Independent				
	FreeSpace	Sweep_Substr_lib	1		0	1	0	Frequency Independent				
	Help OK Apply Cancel											

Now, if you open the EM model, or start a simulation that uses it, the following message box is displayed:

EM (EM Model Needs to be Updated	\odot	\otimes
0	The parameter list of Sweep_Substr_lib:xline1:emModel has b modified. Deleted parameters: Er	een	
	Do you want to update the EM Model's database to the new parameter set? Choose "No" to delete the database content.		
	Yes <u>N</u> o Cancel		

If you click No, all simulation data in the EM model database will be lost.

If you click Yes, the following dialog box enables you to select the data you want to keep. In this example, the value of permittivity is modified from Er to 3. Therefore, you need to keep the simulation results that were done for Er=3.

🏗 😳 Transfer Data to New Parameter Set [Sweep_Substr_lib:xline1:emModel] ? 📀 🔗 🛛 🛞						
The parameter set of the emModel has _ Deleted Parameters	The parameter set of the emModel has been modified. — Deleted Parameters ————————————————————————————————————					
Select the values of the deleted parar results as the design in which the par If you are unsure delete all items.	Select the values of the deleted parameter(s) that gave the same simulation results as the design in which the parameters have been removed. If you are unsure delete all items.					
Er 3 💌						
Database with old parameters	Da	atabase wit	h ne	w parameters		
Name H Er		Name	Н			
1 data.000 25 2	1	data.003	2			
2 data.001 25 4	2	data.004	12			
3 data.002 25 6	з	data.005	22			
4 data.003 2 3	Г					
5 data.004 12 3						
6 data.005 22 3						
<u>U</u> pdate Database		tems			<u>H</u> elp	

After clicking Update Database, the EM model database is updated, as shown in the following figure:

🔣 🖸 xline1 [Sweep_Substr_lib:xline1:emModel 😔 🔗 🛛 🛞						
<u>F</u> ile	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>H</u> elp					
	📁 🗔 🤌 🔁 🐂 🎒 🌲					
Sin	nulation Se	etup Database Inter	oolation Options			
Available data						
		Variables in Subst/Mat				
	Name	Н		:		
	data.003	2				
	data.004	12				
	data.005	22				
Delete All						

Adding a Variable

If you want to analyze the impact of variations in the loss tangent of our dielectric, replace the 0 loss tangent column of our dielectric material with tanD.

Л	💿 Material Definiti	ons									\odot	\otimes
Vi	/ew Technology for this Library: Sweap Substr IIb											
	Conductors Dielectrics Semiconductors Surface Roughness								_			
		Material		Permittivity (B	ir)	Permeab	oility (MUr)		Djordjevic			1
	Material Name 🛆	Library	Real	Imaginary	TanD	Real	Imaginary	Туре	TanD Freq	Low Freq	High Freq	
	Alumina	Sweep_Substr_lib	3		tanD	1	0	Frequency Independent				
	FreeSpace	Sweep_Substr_lib	1		0	1	0	Frequency Independent				
	Add Dielectric Add From Database Remove Dielectric											
	Help OK Apply Cancel						91 					

Since there is no variable with the name tanD, the following message is displayed after closing the material editor.

늘 💿 Undefined variable	\odot	×
The variable tanD is not defined yet. Do you want to d Yes No	efine it r	iow?

Click Yes and assign value 0 to the variable, which corresponds with the value the loss tangent had in all our simulations run until now.

\blacksquare \bigcirc Add Variable \bigcirc \bigcirc \bigotimes					
Name	tanD				
Value	0				
Design	Sweep_Substr_lib:default				
Var Component	VAR1				
	OK Cancel				

Click OK to create the variable and save the variables and the substrate.

Now, if you open the EM model, or start a simulation that uses it, the following dialog box is displayed:

🖭 💿 EM Model Needs to be Updated 🔤	\odot	\otimes
The parameter list of Sweep_Substr_lib:xline1:emModel has modified. Added parameters: tanD	been	
Do you want to update the EM Model's database to the new parameter set? Choose "No" to delete the database content.		
Yes <u>N</u> o Cancel		

If you click No, all simulation data in the EM model database will be lost.

Clicking Yes the following dialog box enables you to enter which value of the new EM Model parameter, tanD, corresponds with all the simulation results saved in the database. Enter 0, since you replaced the fixed value 0 with the variable tanD.

🌇 🕢 Transfer Data to New Parameter Set [Sweep_Substr_lib:xline1:emModel] 🚃 🧿 📀 🔗 🛛 🛞						
The parameter set of the emModel has been modified. Added Parameters Enter the value(s) of the new parameter(s) that would result in the same simulation results as those that are currently in the model's database. If you are unsure delete all items. tanD 0						
Database with old parameters	Database with new parameters					
Name H	Name H tanD					
1 data.003 2	1 data.003 2 0					
2 data.004 12	2 data.004 12 0					
3 data.005 22	3 data.005 22 0					
Update Database Delete All Items Help						

After clicking Update Database, the EM model database is updated as follows:

🔛 💽 xline1 [Sweep_Substr_lib:xline1:err 📀 🔗 🛛 🛞							
<u>F</u> ile	<u>File Edit T</u> ools <u>H</u> elp						
	🖆 🔚 🤌 🥙 🖪 🏪 🎁 🌲						
Sin	nulation Se	etup Data	abase Inter	oolation Opt			
A	Available data						
		Variables	in Subst/Mat				
	Name	Н	tanD				
	data.003	2	0				
	data.004	12	0				
	data.005	22	0				
	Delete All						

Dielectric Substrate Modeling

Dielectric Substrate Modeling

This section provides information about the following dielectric parameters:

- Dielectric Permittivity
- Dielectric Loss Tangent
- Dielectric Conductivity
- Dielectric Relative Permeability
- Dielectric Magnetic Loss Tangent

Dielectric Permittivity

The relative permittivity of all dielectrics is as follows:

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

which can also be expressed as:

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

where ϵ ' is the real portion of ϵ and ϵ''/ϵ' is the dielectric loss tangent.

Examples of materials and their typical permittivity are shown in the following table:

Material Relative Permittivity

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

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

Dielectric Loss Tangent

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

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

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

Dielectric Conductivity

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

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

where:

 $\boldsymbol{\omega}$ = 2 π frequency

 ϵ 0 = 8.85e-12 F/m (absolute dielectric constant free space)

Often, resistivity (ρ) is specified instead of conductivity. The relationship between resistivity and conductivity is as follows:

$\rho = 1/\sigma$

Resistivety is usually specified in Ω cm, conductivity is specified in S/m.

Example:

A typical value for resistivity of a Silicon material is as follows:

$$\rho$$
 = 10 Ω cm

which corresponds to a conductivity value of the following:

 σ = 10 S/m.

Dielectric Relative Permeability

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

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

The relative permeability can also be expressed as:

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

where μ ' is the real portion of μ and μ ''/ μ ' is the magnetic loss tangent. Examples of relative permeabilities are listed in the following table:

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

NOTE

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

Dielectric Magnetic Loss Tangent

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

Generating Thermal Technology Files Using Substrate Editor

Generating Thermal Technology Files Using Substrate Editor

To create thermal technology files, you can consider the following options:

- 1. Define the thermal technology in text editor. This makes the thermal technology files generation process less intuitive, time-saving, and less error-prone.
- 2. Define the thermal technology in UI. This is the recommended option.

This section describes the option 2. Let us look at an example. The thermal technology file consists of the following files:

- 1. streamLayerTable.txt
- 2. heatLayers.cfg
- 3. tech.tcl
- 4. user_tech.tcl (optional)

Here is a sample tech.tcl file:

```
#{name thickness(m) bkgnd-material {layer1 material1}...
{lyrN matN}}
    set layerList {
        { substrate 100.000e-6 GaAs
{Layer10 Au} }
        { mesa 0.978e-6 Si3N4 {Layer01 GaAs}
{Layer02 NiCr} }
        { nicr 0.022e-6 Si3N4 {Layer01 GaAs}
{Layer02 NiCr} } }
# {name volumetric-heat-capacity in J/(K*m*3) = c*d }
        set materialCvList {
            { GaAs 1.73e6 }
            { Si3N4 2.75e6 } }
```

You do multiple calculations to slice layers when multiple processes are laterally juxtaposed, as in a multi-technology or nested-technology setup.

In contrast to the option 1, the option 2 offers the following advantages:

1. The Substrate Editor generates the thermal technology files (all three files) directly using a thermal substrate.

- 2. You can define the substrate that represents the thermal cross-section, the thermal properties in the material database, and the heat sources.
- **3.** You can export the thermal technology files from a substrate after ensuring that all the layers with relevant thermal information are defined in the substrate. The technology files are manually editable if required for customizations.
- 4. You can map the layers that are important for thermal simulation in the top technology for multi-technology setup.

To avail these advantages, you must meet the following prerequisites.

Prerequisites

To use this option efficiently, you must be familiar with:

- ADS Substrate Editor Terminology
- Thermal Technology

Generating Thermal Technology Files

The process of generating the thermal technology files involves the following steps:

- 1. Define Substrate. See Defining Substrates.
- 2. Specify the following thermal properties in Material Definition Dialog:
 - Conductivity (W/mK)

Measure of a material's ability to conduct or transfer heat. It varies with temperature.

- Heat Capacity (J/m^3*K)

Ability of a given volume of a substance to store internal energy while undergoing a given temperature change, but without undergoing a phase transition.

- e- Mean Free Path (m)

Average distance traveled by a moving electron between successive impacts which modify its direction or energy or other properties.

- 3. Identify the heat source layer in the layout. An additional set of layers is added to the layout for the heat source. You must have a layerItem in the substrate whose process role is heat source. In this case, MyHEAT is the "outname" that is used for the third (and fourth) column of heatLayers. cfg file.
- 4. Define temperature dependent property material database. Use the user_tech.tcl file that allows you to add custom tcl variables for material properties and use these variables later in the Substrate Editor. The file must be saved in the THERMAL_DIR directory.
- Specify the layer mask precedence.
 Precedence for layer in the substrate reflects in thermal technology as well.
 Higher is the number, higher is the precedence. If there are two shapes at the same location (say, M0anM1), the thermal properties of the layer with

higher precedence are used. The other layer is ignored (at that particular location). You can control this behavior by setting the precedence of the layer and/or via objects in the substrate. For example, if a layer has M0 and M1 and user needs M1 to take precedence, then M1 precedence number must be greater than M0 precedence number (say M0 take 0 as default and M1 is set to 1).

M1 [1] > M0 [0] Thermal tech output -> sub M1 metal1 {M0 Metal0} M0 [1] > M1 [0] Thermal tech output -> sub M0 metal0 {M1 Metal1}

6. Select File > Export > Thermal Tech file from the substrate window. Currently, there is no UI, so the three files (tech.tcl, streamLayerTable.txt and heatLayers.cfg) are created in the workspace directory.

Now, you can choose the exported tcl file for ElectroThermal Simulation.

Ports

Ports

- Ports Overview
- Viewing Ports
- Defining Ports
- Ports for Momentum
 - Ports for Momentum Overview
 - Applying TML Ports Calibration in Momentum
 - Applying SMD Port Calibration in Momentum
 - Applying Delta Gap Port Calibration in Momentum
- Ports for FEM

Ports Overview

Ports Overview

Before an electromagnetic simulation can be performed on a design, you need to define points of excitation/measurements on your design. These points are identified in ADS using ports. A port is a set of two terminals on the layout at which ADS computes the circuit impedance, etc. Momentum and FEM generate an N-port S-parameter model for your circuit. The S-parameter ports define where energy can enter or leave the structure. They are also required to connect the physical structure with other electrical circuit components in a circuit/EM co-simulation.



A port (1) always consists of two terminals, a • terminal (signal) and • terminal (reference). The port terminals make a connection with the layout through one or more layout pins (). Current will enter or leave the structure through the layout pins. The port voltage will be measured between the two terminals of the port.



Using the Port Editor window, you can edit, view, and set up ports and layout pins in an EM simulation. This window consists of two panels S-parameter Ports and Layout Pins:

- Layout Pin: Specifies a location in layout, either point, edge or area.
- S-parameter Port: Specifies a circuit port connecting two Layout Pins

Layout Pins

The physical size of a layout pin depends on the location of the pin symbol (the top of the arrow) in the layout. A layout pin can be a point, an edge or an area pin.

- Point pin: the pin symbol is inside a metal shape. Current can enter the structure through this point or a small area around it. The EM simulator will determine that.



- Edge pin: the pin symbol is at an edge segment of a metal shape. By default, the size of the edge pin will be all collinear segments. Current can enter the structure through the edge segments. The entire edge is forced at the same electric scalar potential.



The size of an edge pin can be limited by explicitly associating a polyline with the pin. See Using Edge and Area Pins to learn how to create such pins.



An edge pin can be defined internal to a metal shape.



NOTE FEM does support edge pins when the edge is a line segment. Arcs and polylines with two or more segments are not supported. A point pin, at the location of the pin symbol, will be used for the unsupported edge pins.

 Area pin (Momentum only): the pin symbol is internal to a metal shape and a (polygon) shape has been associated with the pin. See Using Edge and Area Pins to learn how to create such pins. Current can enter the structure through the area. The entire area is forced at the same electric scalar potential.



NOTE

ADS FEM does not support area pins. The cylinder ports are created automatically and does not depend whether an area pin is defined or not, i.e., these ports are created even if the pin is not an area pin. The generation of cylindrical ports is automated, as it has better behavior related to the parasitic port effects. For any port (pin) that is in the middle of some metal, cylindrical port are created and its size is determined automatically. It can be referred as an 'automated area pin'. Therefore, "wire" ports no longer exist in FEM. You cannot control the size and it does not depend on the area size defined by area pin
S-parameter Ports

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:

Vp = V(r+) - V(r-)

The scalar potential V(r) at infinity is zero. S-parameters model the difference in voltage between the + and - terminal only. One never knows the absolute voltage of the reference terminal(s). On a schematic, an N-port S-parameter model can have either N+1 or 2N terminals. The two are equivalent because you don't know the absolute voltage at any of the N reference terminals. It doesn't matter if you short them all together or not. In case of a symbol with 2N terminals, only the voltage difference will be applied to the S-parameter model.

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 explicit terminal defined and no infinite ground plane, the port voltage is defined as the value of the electric scalar potential at the + terminal: Vp = V(r+). You can rewrite this as follows:

Vp = V(r+) – V (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.
- CAUTION Caution must be taken when making a connection with a port that has an explicit reference terminal. Incoming current through the plus terminals must be the same as the current leaving through the minus terminals. Only pure differential components can be connected between the + and - terminals. There should be no shunt path that allows current to leak to ground or other components. If this requirement is violated, S-parameter data will be useless.

Defining Ports

Defining Ports

Ports are defined in a two-step process:

1. Add layout pins in the circuit layout. These pins define the places where current can enter or leave the circuit. You can also import pins from other designs.

2. Specify how the layout pins are used to define S-parameter ports in an EM simulation by using the Port Editor window.

Adding Pins to a Layout

A pin can be applied at the midpoint of the edges of a component or object (such as a rectangle or polygon), usually at the midpoint or at the surface of an object. To add a pin:

- 1. Select Insert > Pin or click the Pin icon from the toolbar.
- 2. Position the mouse where you want the port and click. Verify that the connection has been made.

For more information about adding pins, see Using Edge and Area Pins, Edge and Area Pins, and Designating Edge and Area Pins.

Use Layout Pins to Define S-parameter Ports

To define S-parameter ports:

- Open the Port Editor window.
- Create connections between the terminals of a port.
- Specify Feed Type.
- Specify Reference Impedance.

Open the Port Editor

To define ports, open the Port Editor by using one of the following ways:

- From a Layout window:
 - Select EM > Port Editor or
 - Click the Port Editor icon (🛄) on the EM Toolbar.
- From an EM Setup window:
 - Select Ports in the left pane of the EM Setup window.
 - Click Edit to open the Port Editor.

Port definitions are stored with the layout. The Layout window will open along with the Port Editor dialog.

Port Editor	S Port Editor					8		
S-parameter Ports	View Edit Tools Help							
Number	Gnd Layer	Name	Feed Type	Ref Impedance	[Ohm]	Ref Offs	et [mil]	
 ▲ 10 ● P1 ● Gnd 	ground_bottom	P1	Direct	50 + 0i		N/A		
 ▲ 10 / 2 ④ P2 ● Gnd 	ground_bottom	P2	Direct	50 + 0i		N/A		
▲ 10 3 ● P3 ● Gnd	ground	P3	Direct	50 + 0i		N/A		
Layout Pins	Laver	Net		Connected to	Purpe		X [mil]	
	cond	PEin		1(+)	drawi	30	0	_
O+ P2	cond	RFou	ıt	2(+)	drawi	na	760	
O• P3	cond_bottom	VDD		3(+)	drawi	ng	500	
•								Þ
Hide connected layout pins								
V Auto-select	Auto-center	Auto-zo	oom					

NOTE

The port definitions are stored in a layout.

- If any layout change occurred since the dialog was opened the refresh button will be orange Pressing it will refresh the port definition display but only modify it if the layout change affected the ports
- Multiple EM Setup views for a given Layout view will share the same port definition.
- It is recommended to enable the Save all designs when simulation starts under Design Management in the Main Preference dialog box that can be opened from the ADS Main window, **Options** > **Preferences**.

Show me How Do I Define Ports



Port Editor window Overview

S-Parameter Ports

The S-Parameter Ports section displays the plus and minus terminals of a port. The name besides the plus and minus pins specifies the pin name that is connecting with these terminals. The following table describes the S-parameter Ports fields:

Field	Description
Number	Specifies the S-parameter port number.
Gnd Layer	Only applicable for ports with a ' Gnd' minus pin! Specifies the name of the layer where a reference pin will be inserted automatically during the EM simulation flow.
Name	Specifies the name of the first positive pin added. You cannot edit a port name.
Feed Type	Allows you to select a feed type: TML, TML (zero length), SMD, DeltaGap, or None. For more information, see Specify Feed Type.
Ref Impedance [Ohm]	Specifies the reference impedance value in ohms.
Ref Offset [mil]	Specifies the Ref Offset value (shown in layout units).
Term Type	No longer in use.

Layout Pins

The Layout Pins section displays the list of unconnected pins. You make connections between the positive and negative terminals of the port and layout pins by dragging/dropping layout pins For more information, see Create Connections. The following table describes the Layout Pins fields:

Field	Description
Name	Specifies the layout pin name.
Layer	Specifies the layer name.
Net	Specifies the net name.
Connected To	Specifies the connecting terminal name.
Purpose	Specifies the purpose.

Field	Description
X [mm]	Specifies the X axis value (shown in layout units).
Y [mm]	Specifies the Y axis value (shown in layout units).
Number	Specifies the number.
Layer Num	Specifies the layer number.
Purpose Num	Specifies the purpose number.
Pin Type	Specifies the type of pin used in the design.

Refreshing layout pins and S-parameter ports information

Click Refresh pins and ports (<u>)</u>) 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 (IPP) 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.

Setting the Minus Pin

Specifying a minus pin appropriately is critical for EM simulation: it determines the return path of the current.

You can either set it manually (see Create connections between the terminals of a port.) or automatically.

There are two ways to set the minus pin automatically:

By Layer

If no explicit layout pin is assigned to the minus pin of the port, Gnd will be displayed. In this case you can dedicate a layer where the EM engine will autocreate pin(s) on that layer by copying the properties (except the layer) of the pin(s) assigned to the + terminal of the port. If you have not dedicated a specific layer <Implicit> will be shown as the Gnd layer. For Momentum, not using <Implicit> means that the substrate will define the return path of the current. This is normally OK if the substrate itself has a cover or contains a slot plane. If this is not the case Momentum will warn you that its results may not be accurate due to numerical problems and advise you to set the minus pin.

By Pin

You can search a pin on the following parameters:

This search is done based on the preferences that you can specify in the dialog. You can choose search type from "Net", "Layer:Purpose" and "Within". (Combination of them is allowed. In case these are used with combination, it means "AND".)

- Net: Search will be limited to pins only on specified net.
- Layer:Purpose: Search will be limited to pins on specified Layer:Purpose.
- Within: Search will be limited to pins within a certain distance from plus pin (s). (In case you have multiple plus pins, center of gravity of plus pin coordinates is used.)

To assign a minus pin:

- 1. Click the Set Nearest Minus Pin 🌻 .
- 2. Select a Minus Pin Candidate option: All non-used Pins or All non-used Pins with Restriction.

🔁 Set Nearest Minus Pin	? 🗙
Minus Pin Candidates	
All Non-Used Pins	All Non-Used Pins with Restriction
Restrict Search To	
Net 📃	N_30
Layer:Purpose	cond:drawing 🔹
Within	mm
Apply To:	All Ports Selected Only
	OK Cancel

- **3.** Search appropriate pins for the minus terminal of ports that do not have an explicit minus pin.
- 4. Assign the pin to all ports or selected ports.
- 5. Click OK.

Deleting ports

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

Resequencing ports

Click Resequence Ports (128) 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 (). Type the required string in the Search text box. In the Layout Pins panel, you can also right-click any required field and select Set texts to filter. This option picks up the text of current item in Layout Pins and sets it in the filter, as shown in the following figure:

Layout Pins					
Search	(+)P1	cond2	drawing	Search	Sea
Name	Connected to	Layer	Purpose	X [mm]	Y [mn
O+ P1	(+)P1	cond2	drawing	-11	4

Viewing Controls

You can specify how pins are selected in a layout by selecting 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).

Create Connections between the Layout Pins and Terminals of a Port

You can create a connection between the positive (•) and negative (•) terminal of the port and layout pins by dragging/dropping layout pins. To create a port:

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

S-parameter	Ports				
Number	🔶 Name	Ref Impedanc	e [Ohm]	Calibratio	on
	Port1 P1 Gnd	50 + Oi		TML	
i <u>1</u> 2	Port2 P2 Gnd	50 + Oi		TML	
<]		>
Layout Pins					
Name 🔶	Connected to	Layer Pt	urpose	X [mil]	Y [mil]
O• P1	(+)Port1	cond dr	awing	-170	20
()• P2	(+)Port2	cond dr	awing	130	20
<					>

2. 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:

S-parameter Ports	5				
Number	👚 Name	Ref Imp	edance [Ohm]	Calibr	ation
	Port1	50 + Oi		TML	
<					>
Layout Pins					
Name 👘 Cor	nnected to	Layer	Purpose	X [mil]	Y [mil]
O• P1 (+)I O• P2 (−)I	Port1 Port1	cond cond	drawing drawing	-170 130	20 20
<		11			>

NOTE

To specify a minus pin automatically, use the *Set Nearest Minus Pin* option. For more information, see <u>Set Nearest Minus Pin Automatically</u>.

3. A message box is displayed if your target pin is already used by other ports.

- 4. Click Delete Ports and Continue in the message box.
 - NOTE You can drag and drop multiple layout pins and connect with a Sparameter port.
- CAUTION Caution must be taken when making a connection with a port that has an explicit reference terminal! Incoming current through the plus terminals must be the same as the current leaving through the minus terminals. Only pure differential components can be connected between the + and - terminals. There should be no shunt path that allows current to leak to ground or other components. If this requirement is violated, S-parameter data will be useless.

Specify the Feed Type

The default Feed Type is Auto. The EM simulator will automatically determine the most appropriate port feed mechanism. The Auto feed type will be treated in the EM simulators (Momentum and FEM) as Direct feed type, except for Momentum ports on slot layers which will be treated as TML feed type ports.

The Direct feed type connects the source directly at the S-parameter port terminal pins. It is the simplest way to feed a structure. By selecting another feed type, you can add or remove a particular effect of the feed mechanism. This is known as 'calibration'. Calibration in simulation is distinct from what you do in measurements. The source in a measurement setup is in the measurement instrument and, by calibrating, you want to remove the effect of everything in between that source and the DUT. In simulation, you can position the source directly at the right location. By calibrating, you can add or remove an effect of the feed.



The following types of calibration are available:

The calibration is only applied for ports that connect to pins that are on the edges of a conducting shape. In case the pin is internal to a conducting shape, no calibration process will be performed by the EM simulator. The following table describes the feed types used in Momentum and FEM. For more details, refer to respectively Ports for Momentum Overview and Ports for FEM.

Feed Type	Momentum	FEM
Direct	If none of the available feed types is valid for the connecting component, specify Direct. Depending on placement, this will excite either an edge or a point and will include self- induced parasitics.	If none of the available feed types is valid for the connecting component, specify Direct. Depending on placement, this will excite either an edge or a point and will include self-induced parasitics. When placed on an edge, a sheet port is created that inherits the edge widths at the + and – pins.
TML	This feed type assumes the structure is fed through a transmission line. The open end effect is removed. Mutual coupling between parallel feed lines is removed, but mutual coupling induced by the feeding transmission line to the rest of the structure is added.	This feed type assumes that the structure is fed through a waveguide and enforces the waveguide mode field distribution at the port. The feedline is fully calibrated.
TML (zero length)	This feed type removes the open end effect through an RLC lumped-element calibration.	The TML (zero length) option is automatically converted to TML calibration for FEM. No unique calibration technique for TML (zero length) exists for FEM.
SMD	This feed type is defined across a physical gap in the layout artwork. It removes the open-end effect at the port and adds the mutual coupling between the current flowing from the + to – pins and the rest of the circuit, assuming the area between both pins would be filled with metal.	While the reference offset specification is ignored for this option in FEM, a sheet port is created with calibration that removes the self-inductance effect.
Delta Gap	This feed type is defined across a physical gap in the layout artwork. It adds the effect of metal filled in the area between the + and – pins of the port. A gap in the middle remains during the calibration process for the source connection.	The Delta Gap option is automatically converted to SMD calibration for FEM. No unique calibration technique for Delta Gap exists for FEM.
NO	TF The global scope calibration settin	g available in 2016.01 is deprecated in this

The global scope calibration setting available in 2016.01 is deprecated in this version. If a design with a global scope is found its value will be forwarded as the Feed Type for each port.

To assign a feed type:

- 1. Open the Port Editor window.
- 2. Click 🏴 . The Specify the Port Feed Type dialog box is shown:

Specify the Port Feed Type
Feed Type: Direct 🔹
Apply To: O All Ports O Selected Only
OK Cancel

- 3. Select a Feed Type from the drop-down list.
- 4. Select an option for applying the setting to All Ports or Selected Only.
- 5. Click OK.

Reference Offset

The calibrated ports enable you to shift the location of the reference lines used for the S-parameters. In TML, TML (zero length) or SMD calibration, you can additionally shift the reference plane over a certain distance. The transmission line (Momentum) or waveguide (FEM) effect will be removed (offset > 0) or added (offset < 0).

- 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 to 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 Sparameter 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.

Specify Reference Impedance

A reference impedance is needed for each port to compute the S-parameter model. By default, the reference impedance is 50 Ohm. To modify the reference impedance: 1. Click once in the Ref Impedance (Ohm) column in the S-parameter Ports panel, as shown in the following figure:

S-parameter Ports						
Number	Name	Ref Impedance [Ohm]	Calibr			
▲ 1 ● P1	P1	40 + 3	TML (
▲ 10 Gnd ▲ 10 P2	P2	40 + 3i	TML (
Gnd Gnd	83	10 01				

2. Specify the required values in the Ref Impedance (Ohm) field.

Customizing Ports Options

Copy/Paste Port Properties

You can also copy the value of an S-parameter Ports field and paste it on multiple fields. This enables you to update multiple S-parameter Ports fields. This applies for reference impedance, calibration type and reference offset. To copy/paste values:

- 1. Edit the required S-parameter Ports field.
- 2. Right-click and copy the value that you have edited in the field.

Ref Imped	n] C	alib	
40 + 3i		IT	ML
	Сору	Ctrl+C	
10 + 3	Paste	Ctrl+V	L
_			_
30 + 3i		т	ML
50 + 3i		TI	ML

- 3. Select the fields for updating the copied value.
- **4.** Right-click and paste the value on the selected fields, as shown in the following figure:

Ref Impeda	ance [Ohm]	Calibr
40 + 3i		TML (;
10 + 3i		TML
	Сору	Ctrl+C
20 . 2:	Paste	Ctrl+V
50 + 3i		TML (;

Selecting 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 View Option. This opens the View Option dialog box, as shown in the following figure:

🎮 View Option 🛛 🛛 💽 🔀	
Item Item Item Item Index Item Index Item Item Item Item Item Item Item Item	
OK Cancel	

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

Using Layout Pins

Creating Ports with Selected Layout Pin

In the Layout pins panel, you can create port in the following ways:

- Creating a port with selected pins: Right-click the required layout pin and select Create Ports. It creates a port with the selected layout pin.
- Creating a port with clustering pins: Right-click the required layout pin and select Create a Port with clustering pins. It creates a port with all selected layout pins for the plus pin of a S-parameter port.

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 select View Option. This opens the View Option dialog box, as shown in the following figure:

Ite	m
~	Name
~	Connected to
	Layer Num
~	Layer
	Purpose Num
~	Purpose
~	X [mil]
~	Y [mil]
~	Net Name
	Number
	From Tool
	Name (From Tool)
	Net (From Tool)
	Design (From Tool)
	Number (From Tool)
	Port Name (From Tool)
	Pin Type

- **NOTE** The Tool-related parameters such as Name (from Tool) are available only when the pins are imported through Allegro ADFI. Otherwise, these parameters are shown as **N/A**, if you try to select in the View Option dialog box.
- 2. Select the required fields.
- 3. Click OK.

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

Hiding Layout Pins

You can hide the layout pins that are connected to S-parameter Ports by selecting the Hide connected layout pins check box. The layout pins that are connected with S-parameter Ports are hidden in the Layout Pins panel. For example, if all layout pins are connected to S-parameter ports, no layout pins are displayed in the layout pins panel.

The following figure highlights the Hide connected layout pins check box:

Layout Pins					
Name	Connected to	Layer	Purpose	X [mm]	Y [mn
O• P1	(+)P1	cond2	drawing	-11	4
O• P2	(+)P2	cond2	drawing	4	-11
•		111			•
Hide conr	nected layout pins				

Displaying Hidden Layout Pins

You can display the layout pins that are connected to S-parameter Ports by deselecting the Hide connected layout pins check box. The layout pins that are connected with S-parameter Ports are hidden in the Layout Pins panel. For example, if all layout pins are connected to S-parameter ports, no layout pins are displayed in the layout pins panel.

See Also Viewing Ports Ports for Momentum Ports for FEM

Ports for Momentum

Ports for Momentum

- Ports for Momentum Overview
- Applying TML Ports Calibration in Momentum
- Applying SMD Port Calibration in Momentum
- Applying Delta Gap Port Calibration in Momentum

Ports for Momentum Overview

Ports for Momentum Overview

The port setup determines how the structure is excited. The source excitation mechanism is specific to the EM simulator that is used. In this chapter, details about the Momentum source excitation mechanism are provided.

Momentum connects a voltage source with series source impedance between the + and - terminal of each port. Then, it will solve the structure for as many independent excitation states as there are ports (N). For each excitation state, the source amplitude at a particular port is set to 1 Volt and all other ports are terminated by their source impedance. The S-parameters can be computed from these N solutions.

Port Size Restriction

Due to the lumped character of the source used by Momentum, there are two restrictions with regard to the port size. A first has to do with the distance between the + and – terminal. The other has to do with the physical size of the pins. Failing to comply with these restrictions may lead to unphysical S-parameters.

In the source, electric current flows from the – to the + terminal without delay. This is a valid assumption only when the distance between the + and – pins (or infinite ground plane) is electrically small. Rule of thumb is that the physical distance must be smaller than 1/10 of the wavelength at the highest frequency.



Momentum enforces a constant potential over the entire edge or area pin. This assumption is only valid when the length of the edge pin or the diameter of the area pin is electrically small. Rule of thumb is that the size is smaller than 1/10 of the wavelength at the highest frequency.



NOTE To comply with these restrictions in case of a circuit without infinite ground plane in the substrate:

- S-parameter ports should always be defined between two explicit layout pins, or
- The S-parameter model (which can be unphysical because the reference of the ports is at infinity) should only be used in a circuit schematic with sources and loads connecting pins that are electrically close.

What's the Port Feed Type

In a Momentum simulation, by specifying the port feed type you want to add or remove a specific effect of the feed.

In case you choose 'Auto' as the feed type, Momentum will auto-determine one of the feed types described below from the layout of the port terminal pins.

In case you choose 'Direct' as the feed type, the source will be directly connected at the S-parameter port terminal pins. By choosing another feed type, you can add or remove a particular effect of the feed mechanism. The following feed types apply only for ports that connect to pins that are on the edges of a conducting shape. In case at least one pin of a port is internal to a conducting shape, no feed effects can be added/removed.

- TML: Assumes the structure will be fed through a transmission line. This calibration type removes the open end effect and adds the mutual coupling induced by the current flowing on the calibration feed line.
- TML (zero length): Removes the open end effect. Nothing gets added.

- SMD: Removes the open end effect and adds the mutual coupling between the current flowing from the + to – pins and the rest of the circuit assuming this current flows through extended feed lines from + and – pin that are joined in the middle.
- Delta gap: Extends the + and pin with feed lines that are joined in the middle of the gap with the source. This adds the effect of metal filled in the area between the + and - pins.
- Direct: The source will be directly connected at the S-parameter port terminal pins.

Direct Feed Ports

The simplest setup is to not add/remove any feed effects. Therefore, set the Feed Type to Direct in the Port Editor. The following figure shows the source excitation used for direct feed ports, which applies to point (shown on right), edge (shown on left) and area pins (not shown here).



Connection to an edge pin

An illustration of how the source is connected for an edge pin is shown below.



The source is connected to all of the edges of the mesh cells generated for that edge pin. In the example shown above, there are 3 mesh edges affected by the edge pin. The sum of the currents through the pin edges make up the source current. All 3 pin edges are forced at the same potential, they are short-circuited.

When an edge pin becomes electrically wide and multiple mesh cell edges make up the pin, spurious current loops can be induced as shown below that do not flow through the source. When this loop current becomes significant, the simulation result with the port left open will be different from the simulation result with the port removed.



Limit the size of the edge pin if you want to avoid such spurious current loops.

Connection to a point or area pin

An illustration of how the source is connected for a point pin is shown below.



The lumped source is directly connected to the node in the equivalent LC network corresponding with the cell in the mesh around the point pin. In case of an area pin, one or more mesh cells will make up the pin area. The source is connected to all of these mesh cells in parallel. The sum of the currents through the cell nodes make up the source current. All pin mesh cells are forced at the same potential, they are short-circuited.

In case multiple mesh cells make up the pin, spurious current loops can be induced, similar to what was described for edge pins.

Connection to a reference pin

In ADS, any negative – pin can serve as a reference pin, whether it is a local potential/voltage reference or a global ground. A direct port connected between a + pin and a - pin is considered to excite the structure in a differential way. The – pin is just a reference point for the excitation port. The S-parameters associated with direct ports are calculated by exciting the circuit with a lumped voltage source connected to the branches in the equivalent LC network corresponding with the port edges in the mesh.

Note for direct ports associated with reference pins: ADS uses a lumped voltage source to excite the circuit at the direct port + pin and reference - pin combinations. Lumped implies that the distance between + pin and - pin is electrically small (< λ /10) at all frequencies. You can define direct ports associated with a reference pin that do not follow this rule. However, you might get unphysical simulation data as a result.

For reference pins associated with direct ports placed either on edges of structures or within structures, a lumped source excitation is applied and no feed type technique is used. Also, local ground recombination and grouping is applied to the associated ports.

Assumptions for reference pins associated with direct ports are:

- Same as for direct ports.
- The port voltage is referenced to a local ground. The reference points for the excitation ports are explicit definitions of the negative terminal of the associated port. This explicitly defines a ground return path for the uncalibrated port.
- A local open end effect is included in the DUT model. This is a result of the direct feed being applied to the edge port.

An illustration of the excitation used for structures with this port type is shown in the following figure:



In this figure, the direct edge port scenario is displayed on the left side, while on the right side is the direct point port scenario. From this connection approach, the recombined/grouped S-parameters can be directly calculated. This applies to both edge ports (shown on left) and to point ports (shown on right). The direct port is the + pin while the reference point is the - pin of the recombined port. The recombined S-parameters are derived from this.

Show me How Do I Define Ports



See Also Defining Ports

Viewing Ports

Applying TML Ports Calibration in Momentum

Applying TML Ports Calibration in Momentum

If you select the TML calibration, Momentum automatically adds a transmission line feed line to the circuit. A source is added at the end of the feed line to excite the structure.



In case the feed line overlaps with another object in the layout, the port calibration type will be changed to 'TML (Zero Length)'. A warning message will be issued indicating the change.

A model for the feed structure is extracted from simulating a calibration standard. That model allows to deembed the feed structure but leaves the mutual coupling effect with the circuit in the S-parameter model.

Applying TML Calibration

The Transmission Line Calibration (TML) technique is used in Momentum for TML port positive pin, common, and differential ports.

- This calibration technique removes the "open end" discontinuity effects of TML mode ports.
- This calibration technique is based on extending the ports with finite feed lines (uniform transmission lines) of length λ /2 at each analysis frequency. The characteristic impedance and propagation constant is calculated for this line at each frequency.
- The extended feed lines are modeled through the simulation of a calibration standard.
- The effect of the feed lines is removed from the extended circuit simulation results.
- Extended feed lines are filtering out the (non-propagating) higher-order modes excited by the source.
- The port model after calibration is valid for a single propagating fundamental transmission line mode.

Port with single positive pin

The following figure illustrates how the DUT gets extended with calibration feed lines at the input and output pins.



A calibration standard is simulated behind the scenes and the resulting model is used to deembed the calibration feed lines.



Ports sharing the same reference plane

Parallel feed lines of ports that share the same reference plane will be calibrated as a group. This is necessary to remove the mutual coupling effect between the feed lines.



In case the parallel feed lines do not share the same reference plane, they will be calibrated separately. The mutual coupling between the feed lines will not be taken into account (is not added to the coupling matrix) when simulating the structure to improve the accuracy of the calibration process.

	S-parameter Ports					
	Number	* Name	Ref Impedance [Ohm]	Calibration	Ref Offset	
₽.⊅	G • 1 1	P1	50 + Di	TML	D	R 2
	☐ 1 2 ○ P2 ○ Gnd	P2	50 + Di	TML	0.	
p ج	☐ 10 3 ☐ 0 P3 ☐ 0 P3 ☐ 0 Gnd	P3	50 + Di	TML	0	P.4
		P4	50 + 0	TML	٥	

There may be significant coupling between the parallel calibration feed lines that are added to the circuit.



A single calibration standard modeling these coupled feed lines is used to deembed them.



Parallel ports not sharing the same reference plane will calibrated individually. The mutual coupling between the calibration feed lines will be neglected in the computation process .

Common Mode Port

For two or more ports having common polarity, a coupled transmission line (TML) excitation is applied and a coupled extended calibration line technique is used. For example, two ports having + terminals can be assigned as common ports, as shown in the following figure:

Ports Ports are stored w S-parameter Ports	with the layou	ut. To undo port edits: use	undo from the la	yout window then refresh.
Number	Name	Ref Impedance [Ohm]	Calibration 🔺	Ref Offset [mil]
	P2	50 + 0i	None	N/A
 1 P1 P3 Gnd 	Ρ1	50 + 0i	TML	0

TML excitation setup for a DUT with associated common polarity ports attached at the input and output



Assumptions for common polarity ports are:

- Same as for TML Port positive pin.

- The ports are aligned in the same reference plane and are desired to be considered as same-polarity (common-mode) combined ports.

The following figure displays the calibration standard used for structures with common ports in the following figure:



From this, the calibrated and recombined S-parameters can be calculated.

Differential, Coplanar or GSG Port

These ports do have one or more explicit reference pins.

For a differential mode port, a coupled transmission line (TML) excitation is applied and a coupled extended calibration line technique is used.

S-parameter Por	us				
Number	Name	Ref Impedan	ce [Ohm]	Calibration	Ref Offset
1 1 0 1 0 1 0 1 1 0 1	P1	50 + Oi		TML	0
 1 2 0 P2 0 P4 	P2	50 + Oi		TML	0
<					>
Layout Pins					
Name 🔶 C	onnected to	Layer Num	Layer	Purpose Num	Purpose
O• P1 (+	+)P1	1	cond	-1	drawing
O• P2 (+	+)P2	1	cond	-1	drawing
O D3 ()P1	1	cond	-1	drawing
0.12	, <u>-</u>				

TML excitation setup for a DUT with associated differential ports attached at the input and output



Assumptions for differential ports are:

- Same as for TML Port positive pin.
- The ports are aligned in the same reference plane and are desired to be considered as opposite-polarity (differential-mode) combined ports.

Coplanar ports are used specifically for coplanar waveguide (CPW) circuits. It is similar to differential ports, but coplanar ports are applied to objects on slot layers.

In case of multiple ground reference pins, a local ground recombination and grouping is applied to the associated ports.

Ports are s	tored w	vith the l	ayout. To un	do port e	edits: use	undo f	rom the la	yout windo	w then refr	resh.
S-paramete Number	er Ports	Name	RefIm	edance	[Ohm]	Calibr	ation 🔺	Ref Offse	t [mil]	
	P1 P3 P5	P1	50 + 0i	counce	[orm]	TML		0	. []	
⊡ <u>1</u> 2	P2 P4 P6	P2	50 + 0i			TML		0		
Layout Pins	s			1.	-		-			
	Conne	cted to	Layer Num	Layer	Purpose	e Num	Purpose	X [mil]	Y [mil]	Net Na
O P1	(+)P1 (+)P2		1	cond	-1		drawing	-50	15	N_3
O• P3	(-)P1		1	cond	-1		drawing	-50	65	N 5
O• P4	(-)P2		1	cond	-1		drawing	150	65	N_0
O• P5	(-)P1		1	cond	-1		drawing	-50	-35	N_6
O• P6	(-)P2		1	cond	-1		drawing	150	-35	N_7

The TML excitation setup for a DUT with reference pins associated with a TML port positive pin attached at the input and output



Assumptions:

- Same as for TML Port positive pin.
- The ports are aligned in the same reference plane and are desired to be considered as combined ports where the reference pins are explicit definitions of the negative terminal of the associated port.

The TML calibration standard used for a DUT with reference pins associated with TML port positive pins attached at the input and output is displayed in the following figure:



From this, the calibrated and recombined S-parameters can be calculated.

Source Type Switch for TML Calibrated Ports in Microwave Mode Simulations

The Momentum TML calibration procedure becomes less accurate when radiation effects become important. The feed lines added in the Momentum simulation process will radiate as well. The radiation by the feed line will be different when in combination with circuit versus when in combination with a mirrored equivalent of itself as part of the calibration standard. Consequently, the radiation effect of a calibration line cannot be perfectly removed.

To lower the radiation of the feed line, above a certain frequency, a floating source is used instead of the default grounded source. The displacement current through the substrate will ensure a closed loop with the ground. This source type only works above a certain frequency because the capacitive internal impedance of the source through the substrate blocks the current flow at low frequencies.

For Microwave-mode simulations, the simulator automatically uses a grounded source below the quasi-static frequency and a floating source above the quasi-static frequency (fqs). The quasi-static frequency is chosen automatically by Momentum. The simulator switches between source types at the quasi-static frequencies to avoid these issues.



Grounded source excitation



For the grounded source, consider the following factors:

- This source is used in Momentum RF for all frequencies.
- This source is used in Momentum MW at low frequencies (below ~10 MHz, the exact transition frequency is substrate dependent).
- This source is a grounded source (connected between the port and the ground). The + terminal is connected to port and the - terminal connected to ground below port.
- This is a "lumped" source. (Based on the assumption that the distance between port and ground is electrically small (< \u03c6 /10), hence yields good results at low frequencies down to DC and yields less accurate results at higher frequencies where the "electrically small" distance assumption is violated. In the RF mode, this assumption should always be met, since it is based on a quasi-static approach.)
- At some high frequency, when distance between two terminals becomes electrically large (substrate thickness > λ /10), the grounded source can yield unphysical results.
- The grounded source can be used from DC to fmax.



For the floating source, consider the following:

- This source is used for TML calibrated ports in Momentum MW at higher frequencies (above around 10 MHz, the exact transition frequency is substrate dependent).
- This source is a "floating" source (not physically connected to the ground, it is capacitively connected to ground). The + terminal is connected to a port and the - terminal connected to the floating local ground.
- The distance between two terminals is "lumped" at all frequencies. Hence, this source is electrically small (< λ /10) at all frequencies by construction; however, due to the capacitive connection to ground, it blocks the DC current and yields incorrect results at low frequencies (due to breakdown of the solution).
- Towards lower frequencies, the (capacitive) source impedance to ground becomes so large that it is no longer a suitable source to excite the structure.
- The floating source can be used from fmin to fmax.

The transition between the grounded and floating source in Momentum MW is set at a frequency that is substrate dependent, typically around 10 MHz, where both are valid sources, hence, the S-parameters should be identical and the transition can be made safely. In some cases, the transition frequency is too low, causing a discontinuity in the S-parameters. In case of an adaptive frequency sweep, this discontinuity may lead to non-convergence. In such case, consider to use the TML zero length calibration instead or force to use the grounded source at all frequencies. You can do this by adding following configuration variable in a momentum.cfg file.

MOM3D_USE_SOURCETYPE=0

A value of 1 forces the floating source at all frequencies, but note that this is not a suitable source at low frequencies.

This variable can be set in either one of following locations:

- \$HPEESOF_DIR/Momentum/<version>/custom/config/momentum.cfg
- \$HOME/hpeesof/config/momentum.cfg

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.



See Also Defining Ports Viewing Ports

Applying SMD Port Calibration in Momentum

Applying SMD Port Calibration in Momentum

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.



Defining SMD Ports

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.
- **3.** Select SMD from the drop-list available in the Calibration column, as shown in the following figure:

Ports			
Ports are stored w	ith the layout. To	undo port edits: use undo f	rom the layout window then refresh.
😒 🏚 🖞 🖊	< 🛃 👫 🔨	T	
S-parameter Ports			
Number	Name	Ref Impedance [Ohm]	Calibration Rei 📤
🖻 🛄 1	P1	50 + 0i	None 🗸 🗸
• • • P1			None
	P2	50 ± 0i	TML (zero length)
- 0 P2	12	30 1 0	SMD
Cod			Delta gap 🗸 🗸
<			>

- 4. Click once in the Ref Impedance [Ohm] column of a port and specify the required value.
- 5. Click once in the Ref Offset [um] column of a port and specify the required value.
- 6. Select a pin row in Layout Pins.
- 7. Drag the pin from Layout Pins and drop it on the 😏 or 🗢 terminal of the required port in S-parameter Ports.
- 8. A message box is displayed if your target pin is already used by other ports.
- 9. Click Delete Ports and Continuein the message box.

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

Show me How Do I Define SMD Ports

Define SMD Ports

Example SMD Port Calibration

A circuit model for the SMD component is displayed in the following figure:



This model:

- Includes parasitic self impedance (between the two port pins).
- Does not include mutual impedance (coupling) with the interconnection traces.

For creating this circuit model in the layout:

- Define two edge pins (P1 and P2).
- Define an S-parameter port connected to pins P1 and P2.
- Set feed type to SMD.
- Specify the reference offset (OPTIONAL).

For creating this circuit model in the schematic:

- The (layout lookalike) symbol view includes 2 pins for the SMD port.
- The SMD component is connected between these two pins.



SMD calibration in the Momentum engine:

- Feed lines are automatically added to create a delta-gap.
- Delta-gap lumped source used to excite the port.
- Calibration standard used to de-embed the feed lines. Note that the mutual coupling effect of the feed lines with the rest of the circuit remains in the S-parameter model.



Assessment of calibration accuracy

Simulation with SMD replaced by matching line section should give SAME result as simulation of straight line without SMD ports



Perfectly calibrated means these two simulations give identical results

SMD circuit component parasitic self impedance included mutual impedance NOT included



SMD port calibration:

- Feed lines with delta-gap source
- Includes self and mutual impedance
- SMD calibration removes the parasitic self impedance (NOT the mutual impedance)
- Reference shift can be used to control amount of self impedance removed



	s	-par	am	eter	Por	ts
--	---	------	----	------	-----	----

ľ	o por onne		0.00				
	Number		Name	Ref Impedance [Ohm]	Calibration	Ref Offset [mil]	
	🕀 🛄	1	P1	50 + 0i	TML	0	
	÷.	2	P2	50 + 0i	TML	0	
	1	3	P3	50 + 0i	TML	0	
	1	4	P4	50 + 0i	TML	0	
	<u>أ</u> - ط	5	P5	50 + 0i	SMD	0	

Assessment of calibration accuracy

Simulation with very small (delta) gap in layout with NO calibration gives SAME result as simulation of physical SMD gap with DeltaGap calibration.

Show me How Do I Define Ports

How to Define Ports

See Also Defining Ports

Viewing Ports

Applying Delta Gap Port Calibration in Momentum

Applying Delta Gap Port Calibration in Momentum

If you select the Delta gap calibration, Momentum automatically adds two feed lines to the two edge pins of the port. The lines will fill the gap and a (differential) 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

- when connecting with a 2-pins SMD component whose model does not yet includes the parasitics associated with 'bridging the gap'.
- when you want to create a gap on a transmission line to, for example, measure the current passing through in a circuit simulation.



Defining Delta Gap Calibrated Ports

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select Ports in the left pane of the EM Setup window.
- 3. Select Delta Gap from the drop-list available in the Calibration column.

Example Delta Gap port Calibration

In this section, a physical structure is connected with a Delta Gap component. Assumptions for the circuit model for the Delta Gap component:

- Does NOT include parasitic self impedance (between the two port pins).
- Does NOT include mutual impedance (coupling) with the interconnection traces taken into account through DeltaGap port calibration.
- User defines two edge pins (P1 and P2).
- User defines an S-parameter port connected to pins P1 and P2.
- User sets feed type to DeltaGap.



Number	Name	Ref Impedance [Ohm]	Calibration	Ref Offset [mil]
🖻 🚺 🚺	P1	50 + 0i	TML	0
🕀 🏥 2	P2	50 + 0i	TML	0
🕀 🚺 3	P3	50 + 0i	TML	0
🕀 🚺 4	P4	50 + 0i	TML	0
🗄 – 🛄 5	P5	50 + 0i	Delta gap	N/A

The following Use Model is used in schematic:

- The (layout lookalike) symbol view includes 2 pins for the DeltaGap port.
- The SMD component is connected between these two pins.

	ł	Term Term1 Num=1 Z=50 Ohm		-	-	•	•	•	•	-	•	•	•	•	-	•	•	 	-	-	•	-	•	•	•	-	-	-		ſ	-			Ten Ten Nur Z=6	n n2 ⁻ n=2 i0 Oh	- - -
1	1				÷	í.	÷	i.	÷	í.		÷		÷	i.	i.	i.			í.	÷	í.		í.	i.	i.	i.	i.	í.		Ì	1	1	F		
	1				÷	÷	÷	÷		ċ	÷	÷	÷	÷	1	:	1			÷		÷		÷	ċ	ċ	÷	ċ	÷	ŀ	-	•	•	 	n .	1
	Ş	Term Term3 Num=3	1			oup	ledl	Line	s I	NoN	1° 0	Delt	aĞı	ap	I											•		÷		ļ				Terr Nur Z=8	n4 n=4 0 Oh	m
l	ŧ	.Z=50 Ohm	· ·		e	mM _11	ode	10M	16M	_D	elta	Ga	p'	1	L						•	:		•		-	•	ł			Ì			■		•

DeltaGap calibration in the Momentum Engine:

- Feed lines are automatically added to create a delta-gap
- Delta-gap lumped source used to excite the port

Show me How Do I Define Ports



See Also Defining Ports

Viewing Ports

Ports for FEM

Ports for FEM

Ports enable energy to flow into and out of a circuit. Energy is applied to a circuit as part of the simulation process. A circuit solved using FEM Simulator must have, at the minimum, one port. Ports are defined in a two-step process. First, layout pins are added to a circuit when the circuit is drawn. Then, you specify the type of port in order to tailor the port to your circuit. This facilitates the simulation process.

Considerations

While adding pins to circuits to be simulated using FEM:

- The components or shapes that pins are connected to must be on layout layers that are mapped to metallization layers that are defined as strips or slots. Pins cannot be directly connected to vias.
- Ensure that pins on edges are positioned so that the arrow is outside of the object, pointing inwards, and at a straight angle.
- Ensure that the pin and the object you are connecting it to are on the same layout layer. For convenience, you can set the entry layer to this layer; the *Entry Layer* listbox is on the Layout tool bar.
- A pin must be applied to an object. If a pin is applied in open space so that it is not connected to an object, Momentum will automatically snap the pin to the edge of the closest object. However, FEM leaves the pin unattached to the structure. This difference is not apparent from the layout, however, because the position of the pin will not change.
- ADS FEM does not support area pins. The cylinder ports are created automatically and does not depend whether an area pin is defined or not, i.e., these ports are created even if pins are not defined as area pins. The generation of cylindrical ports is automated, as it has better behavior related to the parasitic port effects. For any port (pin) that is in the middle of some metal, cylindrical port are created and its size is determined automatically. It can be referred as an *automated area pin*. Therefore, *wire* ports no longer exist in FEM. You cannot control the size and it does not depend on the area size defined by area pin.

NOTE

When impedance lines are displayed in the 3D EM preview mode, the endpoints for the lines will be placed in the middle of thick conductors. During the solution process, the lines are processed, moving their endpoints to the correct surface, based on the solution setup.
Selecting a Feed Type

Ultimately, FEM supports three levels of calibration:

- Fully calibrated ports
 - Ports that exist on the virtual 'bounding-box' that strictly surrounds all polygons on layers mapped in the substrate AND have the feed type TML are fully calibrated.
- Sheet port calibration
 - Ports that are attached to at least one edge are realized as sheet ports. The self-inductance of an equivalent sheet of current is calibrated for ALL feed types (unless the port meets the fully calibrated criteria above).
 - This calibration does not remove port parasitic effects due to port capacitance, current fan-in/fan-out, or mutual inductance between sheet port currents.
- Uncalibrated ports
 - Ports that are not attached to at least one edge (i.e. they only attach to a points within polygons) are not calibrated
 - These ports include parasitic effects due to port current selfinductance as well as port capacitance, current fan-in/fan-out, and mutual inductance between port currents.
- **NOTE** In ADS 2014.01, an automatic calibration routine is added for FEM sheet sources to remove the self-inductance of the sheet of current that flows through the source. This calibration is always enabled for ports attached to at least one edge (geometry edge or edge port) this calibration takes place for for **ALL** feed types, including *None*. Ports that do not connect to any edges are not yet calibrated (and thus include excess inductance).

TML Calibration

TML is the default calibration in FEM ports. It has the following properties:

- The port is connected to an object that is on either a strip or slot metallization layer.
- The port is external and calibrated. The port is excited using a calibration process that removes any undesired reactive effects of the port excitations (mode mismatch) at the port boundary.
- The port boundary can be moved into or away from the geometry by specifying a reference offset. S-parameters will be calculated as if the phase reference were at this position. For more information, refer to Applying Reference Offsets.

In FEM, TML ports have the following restrictions:

 The positive pin of the port must be attached to edge of an polygon, or be an edge pin. - The pins that define the port must be on the strict *bounding box* that surrounds all polygons on layers mapped in the substrate definition.

NOTE TML is the default feed type, but it will revert to None if the port does not satisfy the restrictions above.

To define a TML port calibration:

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select FEM.
- 3. Select Ports in the left pane of the EM Setup window.
- 4. Select TML in the Calibration column.

FEM uses the Waveguide ports or TML calibration to shift the location of the reference lines used for the S-parameters. It consists of the following features:

- Calibrated 2D-plane source
- User defines a 2D plane for the port in the simulation domain with a source impedance

You can select the following types of reference offsets:

- Specifying a positive reference offset: Shifts the reference planes in the direction towards the circuit.
- Specifying a negative reference offset: Shifts the reference planes in the direction away from the circuit.

TML (zero length) Calibration

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

SMD Calibration

While performing an FEM simulation, the SMD ports are treated as sheet ports and the reference offset specification is ignored (i.e. the reference offset is always zero). The self-inductance of an equivalent sheet of current is calibrated out of the results.

Delta Gap Calibration

While performing an FEM simulation, the Delta Gap calibration is equivalent to an SMD calibration.

None Calibration

The None port calibration enables you to apply a port to the surface of an object in your design. By using None calibration, all the physical connections in a circuit can be represented, so your simulation can take into account all of the EM coupling effects that will occur among ports in the circuit. These coupling effects caused by parasitics are included in your simulation results because ports are not calibrated. You should avoid geometries that allow coupling between TML and None calibrated ports to prevent incorrect S-parameters.

An example of where an internal None port is necessary is a circuit that consists of transmission lines that connect to a device, such as a transistor or a chip capacitor, but this device is not part of the circuit that you are simulating. An internal None port can be placed at the connection point, so even though the device is not part of the circuit you are simulating, the coupling effects that occur among the ports and around the device will be included in your simulation.

A None calibrated port has the following properties:

 None ports that are connected to either a geometry edge or are connected to an internal edge created with a polyline to create an edge pin will be implemented as sheet ports.

NOTE The self-inductance is calibrated out for **ALL** sheet ports, even if the feed type is *None*. If the following configuration variable does not exist in the fem.cfg file, by default 0.5 of the port inductance is subtracted:

FEM_PORT_CALIBRATION_FACTOR = 1

- It can be applied to the interior of a circuit by applying it to the surface of an object.
- It can be applied to the edge of an object.
- It can be applied to objects that are on strip layers only.
- The orientation of the port is not considered if it is on the surface of an object.

FEM uses None Calibration Ports to inject energy into a structure as an alternative to ports. Sources are useful for modeling internal ports within a structure. Some examples are:

 Your structure consists of transmission lines that connect to a device, such as a transistor, but this device is not part of the structure that you are simulating. A source can be connected at that point, so even though the device is not part of the circuit you are simulating, the coupling effects that occur among the connections around the device will be included in your simulation.

To define a None calibration:

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select FEM.
- 3. Select Ports in the left pane of the EM Setup window.
- 4. Select None in the Calibration column.

See Also Defining Ports

Setting up EM Simulations

Setting up EM Simulations

The EM Setup view and its associated editor centralize all settings for your EM simulation (Momentum or FEM) from a single interface. You can select a substrate, inspect port definitions, define a frequency sweep, specify EM simulator options, specify simulation resources, and required output data.

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

- EM Setup Window Overview
- Viewing Layout Information
- Automatic EM Circuit Partitioning
- Viewing the Substrate
- Viewing Ports
- Defining a Frequency Plan
- Defining Simulation Options
 - Specifying Physical Model Settings for Momentum
 - Specifying Physical Model Settings for FEM
 - Defining Preprocessor Settings
 - Defining Mesh Settings for Momentum
 - Defining FEM Mesh Settings
 - Defining Solver Settings
 - Defining Expert Options for Momentum
- Defining an Output Plan
- Specifying Simulation Resources
- Generating an EM Model
- EM Setup Tools
 - Open and Create a Symbol View
 - Creating an SnP Schematic
- Customization
 - Using an EM Setup Template

EM Setup Window Overview

EM Setup Window Overview

ADS 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 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.

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:

Mom RF Layout Partitioning Substrate Ports Frequency plan Output plan Options Resources Model Notes	Setup Type EM Simulation/Model EM Cosimulation EM Simulator Momentum RF Momentum Microwave FEM Setup Overview EM Simulator: Momentum simulation in RF mode EM Simulator: Morkspace: C:\Users\psinghal\RF_Board_NewFlo_wrk Library: RF_Board_Flow_lib Cell: new1_design View: Layout Partitioning between EM and circuit: EM simulation/model of all items Substrate: FR4_DoubleSided (defined in library: RF_Board_Flow_lib) Ports: Substrate: FR4_DoubleSided (defined in library: RF_Board_Flow_lib) Ports: Substrate: FR4_DoubleSided (defined in library: RF_Board_Flow_lib) Ports: Substrate: FR4_DoubleSided (defined in library: RF_Board_Flow_lib) Ports: Dataset: new1_design_MomRF Template: Auto select Mom Simulation options are initialized and ready for simulation. EM simulation options are initialized and ready for simulation. EM simulation on bost:Local Simulation on host:Local	
	EM Model/Symbol:	
	Generate: S-Parameters	Simulate

The following sections describe the EM Setup window:

- Setup Type
- EM Setup Options
- EM Simulators
- EM Setup Summary
- EM Setup Window Menu Options
- EM Toolbar Options

Setup Type

You can select the following types of setup for performing a simulation:

- EM Simulation/Model: This option generates the S-parameter model extracted by an EM simulator for the entire (flattened) layout.
- EM Cosimulation: This option facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator (e.g. a nonlinear device) or for which a model already exists (e.g. a built-in primitive, a schematic, or an EM Model).

EM Setup Options

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, as shown in the following figure:



- Layout: You can view information about the workspace, library, cell, and view by selecting Layout. For more information, refer Viewing Layout Information.
- Substrate: You can open a predefined substrate file from ADS by selecting Substrate. For more information, refer Defining Substrates.
- 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 Ports.
- Frequency Plan: You can add or remove frequency plans for your EM simulations by selecting Frequency Plan. For more information, see Defining a Frequency Plan.
- Output Plan: You can specify the data display settings for your EM simulations by selecting Output Plan. For more information, see Defining a Frequency Plan.
- Options: You can define the preprocessor, mesh, simulation, and expert settings by selecting Options. For more information, see Defining Simulation Options.
- Resources: You can specify the local, remote, and third party settings by selecting Resources. For more information, see Specifying Simulation Resources

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

EM Simulators

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

- Momentum Microwave
- Momentum RF
- FEM

For more information, see FEM and Momentum.

EM Setup Summary

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

EM Setup Window Menu Options

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

Saving 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.

File	
Library:	QFN_Designer_lib
Cell:	example_board_3x3_QFN_chip_Bondw
View:	emSetup
File path:	example_board_3x3_%Q%F%N_chip_%Bondw\em%Setup
Type:	EM Setup View

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.

File	
Library:	QFN_Designer_lib
Cell:	example_board_3x3_QFN_chip_Bondw
View:	emSetup
File path:	example_board_3x3_%Q%F%N_chip_%Bondw\em%Setup
Type:	EM Setup 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.

Import from another EM Setup

To import the contents of another EM Setup in the EM Setup window:

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

🐻 QFN_Designer_lib:exa	mple_board_3x3_QF	N_open_chip 💌
Import from		
Cellview		Browse
Filter		
 Sections to include 		
Setup	Mom options	Fem options
Substrate	Frequency plan	Output plan
Cosimulation	Model	Notes
Partitioning	Compute resource	es
Notes		
I	import Cance	Help

- 2. Click Browse to open the Specify EM Cell View dialog box. For more information, see Opening an EM Setup View.
- 3. Select the required sections from the Sections to include list. Clicking on the Sections to include Checkbox will toggle between selecting or deselecting all the sections. A summary of the content can be seen by hovering over the section label or by pressing the toggle near the Sections to include Checkbox. To view the full content in a read-only EM Setup dialog, press the Inspect icon (
- 4. Click Import.

Opening an EM Setup View

To open an EM setup view, perform the following the steps:

1. Start ADS and open an existing workspace.

- 2. From the ADS Main window, choose File > Open > EM Setup View to open the Specify EM Cell View dialog box.
- 3. Select EM Setup View from the Type drop-down list.

EM Specify EM CellView		×
Type: EM Setup View	Read-only	Show built-in libraries
Library:	Cell:	View:
QFN_Designer_lib	Chip_Thru_Line	emSetup_Mom
QFN_Designer QFN_Designer_lib QFN_Designer_tech	Chip_Thru_Line example_board_3x3_QFN_ example_board_3x3_QFN_	emSetup_Mom
	ОК	Cancel Help

- 4. If you want to open a built-in ADS design (read-only), check the Show ADS Libraries check box to display the list of all libraries under Library.
- 5. Under Library, select the Library name where the view exists.
- 6. Under Cell, select the cell name.
- 7. Under View, select the required EM setup view name.
- 8. Click OK to open the selected EM setup view.

Opening the 3D Preview window

Choose Tools > 3D EM Preview or click the 3D EM Preview icon (

Opening the Layout window

Choose Tools > Open Layout Window or click the Open Layout Window icon (

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.

EM Toolbar Options

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

- 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 (^{EM}) 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 (1) 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

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 more information about a workspace, library, cell, and view, see New Terminologies.

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.

⊿ 🛃 Mom uW	Input Layout	
🔁 Layout	Workspace: C:\Users\osingbal\EllipticEilter wrk	
🛞 Partitioning	Workspecch C. (Beralpangharpinpuchical_wrk	
🛀 Substrate	Library: EllipticFilter_lib	
10 Ports	Cell: momentum	
🔤 Frequency plan	View: lavout	
🚧 Output plan	New, byour	
Options	Show in Layout Window	Show 3D View
E Resources	Show in Layout window	SHOW SO VIEW
EM Model		
Notes		
	Generate: S-Parameters	 Simulate

- 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.

Automatic EM Circuit Partitioning

Automatic EM Circuit Partitioning

The Automatic EM/Circuit Partitioning feature facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator (e.g. a nonlinear device) or for which a model already exists (e.g. a builtin primitive, a schematic, or an EM Model). Automatic EM/circuit partitioning reduces cosimulation setup time.

The partitioning policy defined for an EM Setup and the partitioning rules defined for a cell allow ADS to automatically determine which instances will be EM simulated and which will be circuit simulated. Traditional EM-only simulation of a layout is prohibited if it uses instances not supporting EM simulation. Instead, an EM Cosimulation View can be generated from the EM Setup window.

A circuit simulation is required to use an EM Cosimulation view. It is a netlistable view that integrates an EM model for the EM simulated partition with circuit models for the circuit simulated instances. You can either directly use an EM Cosimulation View as 'View for Simulation', or alternatively, integrate it into a synchronized schematic using an EM Cosimulation Controller.

Partitioning Rules

A partitioning rule determines how a cell instance can be processed, the Action, during the generation of the EM Cosimulation view. There can be multiple rules defined for a cell.

By default, the following rules are assigned to all components:

- A rule "em" to use EM simulation.
- A rule "circuit" to use circuit simulation.

The following table provides a description of these default rules:

Rule Name	Action	Simulators
circuit	Do not descend into the instance. Add cell ports to the EM Partition. The circuit simulation hierarchy policy is used to resolve the model that is used for simulation. This rule requires that the component can be simulated using circuit simulation.	All
em	Descend into the instance. Model all shapes by EM and re-examine the cell instances	All

For cells that haven't been specifically configured, both default rules are assumed to be valid choices. However, user-defined rules can be used to define a special behavior for a cell, see **#Creating User-defined Partitioning Rules for a Cell**.

How Does ADS Select Partitioning Rules?

ADS applies partitioning rules based using the following scheme:

- If you select a rule in the EM Setup window, the specified rule is applied. For more information, see From the EM Setup window.



- Otherwise, ADS looks for a rule specified on the instance. A particular partitioning rule can be enforced for a particular instance of a cell, see #Enforcing Partitioning Rules for an Instance.

 · 🖞 ·						Info					
SANTA		· · · · S	<u>A Ń T A</u>			(\$) 🚖 (\$	0				
ro	End Command		Esc		TL3	Instance: rosa		. ni			
_	Edit In Place		•			Subview for Placed at: (a	simul	atio			
	C <u>o</u> mponent		•	R-17 Ec	dit <u>C</u> omponen	t Parameters					
	Push Into Hierar	chy		E	dit Componen	t Art <u>w</u> ork	â	" Choo	se View for S	imulation	
 	Open Instance V	ïew						Circuit	Simulation	EM/Circuit Partitioning	
 	Move Compone	int Text	F5		eactivate/Activ	/ate		0 1e	t the EM/Circi	it partitioning policy deter	mine which rule to use
 · ·	0.4		Chilly V		eactivate and s	Short/Activate	-	@	e die Eniferie	are partitudorning policy detern	Three which have to use
 0	Cu <u>r</u>		Ctri+X	C N	hoose View fo	r Simulation	≯	O Us	e this rule		
 - 12P	<u>C</u> opy		Ctrl+C	45					circuit		
						And a second			em		
clara		lib3_are	ea_pins	santa	a <u>circuit</u>	EM C					
rosa		lib3_are	ea_pins	santa	a <u>circuit</u>	EM C					
			(di	spla	y: <u>unde</u>	rline)				ОК Са	ncel Help

- If there is no override in the EM setup or on the instance, ADS uses this partitioning policy:

preferred em circuit

The names are tested in order against the available rule names for the cell. The first matching rule will be used. You can override the policy at the top of the partitioning. For more information, see Partitioning Policy.

Creating User-defined Partitioning Rules for a Cell

The cell specializations can be configured from the EM/Circuit Partitioning Rules Of Cell dialog box. While creating a new rule, you need to specify the action and EM simulator.

You can set up your own rules and actions in the partitioning rule editor. Use the *EM Simulator* column to define simulator specific rules. You can also switch to a specific view of the same cell:

- To select a particular port setup as circuit simulated instance.
- To select a simplified layout or empro view for EM simulation.

Rules for view switching:

- Ordering and geometry of the pins must be identical to the original layout.
- Port setup, instances and shapes may be different.

Name	Action	EM Simulator		
circuit_diff	💹 layout_diff All			
circuit	🔛 circuit using ports from placed layout 🔻			
	💭 circuit using ports from placed la			
	💭 circuit using ports from other view			
	🥶 descend in	to placed layout		
	🗺 descend into other view			
	🔀 deactivate view			
	隊 ignore rule	for EM simulator		

The following table describes the various types of actions that can be assigned to a rule:

Action	Description
circuit using ports from placed layout	Simulate with the circuit simulator using the pins and ports defined in the placed layout to connect to the EM simulation.
circuit using ports from other views	Simulate with the circuit simulator using the pins and ports defined in the named layout to connect to the EM simulation.
descend into placed layout	Include shapes and vias inside the placed layout in the EM simulation. The partitioning analysis is repeated for all instances in this layout.
descend into other view	Include shapes and vias inside the named layout or EMPro view in the EM simulation. In the case of a normal layout, the partitioning analysis is repeated for all instances in this layout.
deactivate view	Act as if this cell was not instantiated.
Ignore rule for EM simulator	The partitioning policy will not select this rule, even if it contains its name.

To create a new partitioning rule:

- 1. Open a Layout view of a cell and select File > EM/Circuit Partitioning.... You can also right-click a cell and select File > EM/Circuit Partitioning.... The EM /Circuit Partitioning Rules Of Cell dialog box is displayed.
- 2. Select Specify custom rules.

an addrining reace		
partitioning rule speci	fies a supported modeling	action for instances of this cell in a
M simulated layout		
O Use factory def	aults 💿 Specify	custom rules
Add partitioning rule	Remove partitioning rule	Reset to factory defaults
Name	Action	EM Simulator
em	<pre>>>></pre>	All
circuit	💭 <circuit></circuit>	All
Name and a later		

- 3. Click Add partitioning rule. A new row of rule is created.
- 4. Click once in the Name column and specify a name for the new rule.
- 5. Select a action from the drop-down list in the Action column.
- 6. Select an EM simulator from the drop-down list.
- 7. Click OK. A new partitioning rule is created.

NOTE To remove a rule, select the rule and click **Remove partitioning rule**. To assign default rule, click **Reset factory defaults**.

Enforcing Partitioning Rules for an Instance

A specific rule can be enforced for a specific instance in the following two ways:

- From the Layout window
- From the EM Setup window

From the Layout window

To apply a partitioning rule for a selected instance in the layout window:

1. Select an instance in the layout design.

- 2. Click the Choose View For Simulation button () on the EM Toolbar. The Choose View for Simulation window is displayed.
- 3. Select the EM/Circuit Partitioning tab
- 4. Override the policy selection by selecting a specific rule and click OK.

🚰 Choose View for S	mulation	—
Circuit Simulation	EM/Circuit Partitioning	
Let the EM/Circu	it partitioning policy determine which rule to us	e l
O Use this rule		
circuit		
		,
		OK Cancel Help

From the EM Setup window

To apply a partitioning rule for a selected instance in the EM Setup window:

- 1. Select an instance in the layout design.
- 2. Select EM > Simulation Setup to open the EM Setup window.
- 3. Select Partitioning in the left pane of the EM Setup window.
- 4. Select the row of the instance for which you want to specify the partitioning rule.
- 5. Select the required rule from the drop-down list in the Partition Rule column:

Ell RF_Board_Flow_lib:new1_design:emSetup (EM Setup for cosimulation)									
R 9 8 B	\triangleright								
Mom RF Cosim Layout Partitioning Substrate	Partitioning Policy: Enter space s	eparated list to o	override standard	l policy: preferred er	n dirault				
Ports Frequency plan	Parent Library	Parent Cell	Parent View	Instance Name	Instance Lib:Cell	Partitioning Rule	Modelling ^		
Cutput plan	RF_Board_Flow_lib RF_Board_Flow_lib	new1_design new1_design	layout layout	C1 C2	Board_Design_Kit:C Board_Design_Kit:C	circuit circuit	- Use layout setting -		
Cosimulation	RF_Board_Flow_lib RF_Board_Flow_lib	new1_design	layout	C3	Board_Design_Kit:C Board_Design_Kit:C	<u>circuit</u>	circuit em		

6. You can select the Auto-select, Auto center, or Auto Zoom check box. If you select these options and open the layout window of the parent cell by clicking on the instance name (e.g. I_0), it will perform the appropriate action in the layout window. To open the layout window a right-click menu has been added to the Instance-name cell : Open Parent Layout Window.



It is recommended to uncheck the Auto select, Auto center, and Auto Zoom options, if you do not want to open the layout window, as it may slow down the operation process.

Partitioning Policy

The partitioning policy is an ordered list of rule names. When processing a cell instance, the cell rule that appears earliest in the list will be applied. The default partitioning policy is as follows:

- 1. preferred
- **2**. em
- 3. circuit

NOTE If no matching rule can be found for the cell, an error message will be issued.

The following figure highlights the partitioning policy in the EM Setup window:

Mom RF Cosim	Partitioning				
Layout	Policy: Enter space	separated list to r	werride standard	policy: preferred er	n circuit
Substrate			veniae standard	policy, preferred er	
GHz Frequency plan	Parent Library	Parent Cell	Parent View	Instance Name	Ins ^ Lil
Output plan	RF_Board_Flow_lib	new1_design	layout	C1	Board_Design
Resources	RF_Board_Flow_lib	new1_design	layout	C2	Board_Design
Cosimulation	RF_Board_Flow_lib	new1_design	layout	C3	Board_Design
Notes	RF_Board_Flow_lib	new1_design	layout	C4	Board_Desigr [≡]
	RF_Board_Flow_lib	new1_design	layout	C5	Board_Design
	RF_Board_Flow_lib	new1_design	layout	L1	Board_Design
	RF_Board_Flow_lib	new1_design	layout	L2	Board_Design
	RF_Board_Flow_lib	new1_design	layout	L3	Board_Design
	RF_Board_Flow_lib	new1_design	layout	Q1	Board_Desigr
	RF_Board_Flow_lib	new1_design	layout	R1	Board_Desigr
	RF_Board_Flow_lib	new1_design	layout	R2	Board_Desigr 👻
	Rule text styl	iii le: Defined by pol ses an EM Cosimu	licy - <u>Instance ov</u> lation	erride - Defined he	re

Removing a Partitioning Rule

You can remove a partitioning rule to prohibit EM or circuit simulation. For example, if you remove the *em* rule, ADS will not perform any EM simulation for this cell. ADS will automatically choose circuit simulation.

To remove a rule:

- 1. Open a Layout view of a cell and select File > EM/Circuit Partitioning.... You can also right-click a cell and select File > EM/Circuit Partitioning.... The EM /Circuit Partitioning Rules Of Cell dialog box is displayed.
- 2. Select a rule.
- 3. Click Remove partitioning rule.

A partitioning rule spi EM cimulated layout	ecifies a supported mod	deling action for instances of this cel	l in a
 Use factory of 	defaults 💿 Sp	ecify custom rules	
Add partitioning rule	Remove partitioning	Reset to factory defaults	
Name	Action	EM Simulator	
em	Interplaced layers	out> All	
circuit	<pre>circuit></pre>	All	

4. Click OK.

See Also EM Circuit Cosimulation Manual Versus Automatic EM Circuit Partitioning

Viewing the Substrate

Viewing the Substrate

A drop-down list shows the available substrate definitions. Select the appropriate substrate from the list. The graphical area underneath provides a view on the cross-section.

You cannot edit the substrate from the EM Setup window. Instead, click Open to open the Substrate Editor. In that window, editing of the substrate is possible.

In case there is no substrate defined yet, click New, which will allow you to define a substrate.

Viewing Ports

Viewing Ports

You can view ports in the EM Setup window. However, you cannot define or edit ports from the EM Setup window.

To view ports:

- 1. Select EM > Simulation Setup to open the EM Setup window.
- 2. Select Ports in the left pane of the EM Setup window. The ports are displayed, as shown in the following figure:

	X 🖪 🏜 🗁	🗇 📶 🐁			
FEM Layout Partitioning Substrate	Ports Defined by: O T	'his EM setup 💿 In , riew only)	nput layout Op	en Layout Port Edito	r
🕮 Frequency plan 🐺 Output plan	Number	Name (Calibration	Ref Impedance [C Ref Offset
 Options Resources Model Notes 	□ □ □ □ P1 □ □ P1 □ □ Gnd □ □ □ 1 □ □ □	P1 1	EMI	50 + 0i	• • •
	Layout Pins (view on	ly)	Dumana V I	1 M. Franz I. M. Harris	han I area Nirra
	Q ⁴ P1 cond 1 N	115 1(+)	drawing -44	mj r (umj Num 0 -1513 1	1
	O P2 cond_1 N	0 2(+)	drawing -616	i.306 475 2	1
	O• P3 cond_1 N	_116 3(+)	drawing -127	3.69 -4655 3	1
	•				•
	Hide connected	layout pins Auto-center 🔲 Au	to-zoom		

To edit ports, open the Port Editor window from a layout or from the EM Setup window.

- To open the Port Editor window from a layout, click the Port Editor icon (
) present on the EM toolbar.
- To open the Port Editor window from the EM Setup window:
 - Select EM > Simulation Setup to open the EM Setup window.
 - Select Ports in the left pane of the EM Setup window.
 - Click Edit to open the Port Editor dialog in the Layout window.
- NOTE See
- See Defining Ports for more information on how to define ports.

Port Overview in the EM Setup Window

This section provides information about the port options available in the EM Setup window.

S-Parameter Ports

The S-Parameter Ports section displays the plus and minus terminals of a port. The name besides the plus and minus pins specifies the pin name that is connecting with these terminals. The following table describes the S-parameter Ports fields:

Field	Description
Number	Specifies the S-parameter port number.
Name	Specifies the name of the first positive pin added. You cannot edit a port name.
Ref Impedance [Ohm]	Specifies the reference impedance value in ohms.
Calibration	Allows you to select a feed type: TML, TML (zero length), SMD, DeltaGap, or None. For more information, see Defining Ports.
Ref Offset [mil]	Specifies the Ref Offset value (shown in layout units).

Layout Pins

The Layout Pins section displays the list of unconnected pins. You make connections between the positive and negative terminals of the port and layout pins by dragging/dropping layout pins For more information, see Create Connections. The following table describes the Layout Pins fields:

Field	Description
Name	Specifies the layout pin name.
Layer	Specifies the layer name.
Connected To	Specifies the connecting terminal name.
Net	Specifies the Net 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).

Field	Description
Number	Specifies the number.
Layer Num	Specifies the layer number.
Purpose Num	Specifies the purpose number.
Pin Type	Specifies the type of pin used in the design.

Refreshing Layout Pins and 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.

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 Ports

For easy filtering and sorting commands, click Toggle Search Filter On/Off (). Type the required string in the Search text box. In the Layout Pins panel, you can also right-click any required field and select Set texts to filter. This option picks up the text of current item in Layout Pins and sets it in the filter, as shown in the following figure:

Layout Pins

Search	(+)P1	cond2	drawing	Search	Sea
Name	Connected to	Layer	Purpose	X [mm]	Y [mn
O• P1	(+)P1	cond2	drawing	-11	4

Viewing Controls

You can specify how pins are selected in a layout by selecting 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).

See Also Defining Ports Ports for Momentum Ports for FEM

Defining a Frequency Plan

Defining a Frequency Plan

You can set up multiple frequency plans for a simulation. For each plan, you can specify that a solution be found for a single frequency point or over a frequency range. You can also select a sweep type for your frequency plan. However, the collection of frequency plans 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
Туре	Specifies the type of frequency:
	 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. Two datasets will be the written, one with the actually simulated samples (<name>. ds) and another one (<name>_a.ds) which is a densly sampled dataset based on the rational fitting model.</name></name>
	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 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 stop frequencies in the Start and Stop fields, and select frequency units for each. Enter the number of frequency points to be simulated in the Points/Decade field, and select units.
	 Single: It simulates at a single frequency point. Type a value in the Frequency field and select units.
	Dec: This option creates a frequency plan with logarithmically distributed frequencies with the number of points specified per decade. It is similar to the frequency plan created by using the log option. For example, Logs from 1 to 100 with 21 points, is same as dec from 1 to 100 with 10 points/decade.

Column	Description
Fstart	Specifies the frequency start value. Set a unit, such as Hz, KHz, MHz, GHz, or THz, for the selected value.
Fstop	Specifies the frequency stop value. Set a unit, such as Hz, KHz, MHz, GHz, or THz, for the selected value.
Npts	Specifies the number of frequencies to simulate. For Adaptive, this is the maximum number of samples to use.
Step	Specifies the step value. Set a unit, such as Hz, KHz, MHz, GHz, or THz, for the selected value.
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 left pane of the EM Setup window.

✓ EM Mom uW Layout ③ Partitioning	Fre	equency Plan	Remove]			
Substrate		Туре	Fstart	Fstop	Npts	Step or pts/dec	Enabled
GHB Frequency plan	1	Adaptive	0 GHz	15 GHz	25	-	
Output plan							
Resources							
Model							

- **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.

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:

NOTE

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

Defining Simulation Options

Defining Simulation Options

You can specify engine-specific options, such as physical model, preprocessor, and mesh for EM simulations. You can either use a predefined set of simulation options (e.g. from an EM Template) or your own customized options.

To view the 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:

Mom uW	Simulation Options									
Partitioning	Preset:	MMIC		• R	▼ Rename		/e			
Substrate Substrate Ports Frequency plan Output plan Options Resources Model Notes	All si	iption	Physical Model	Preprocessor	Mesh IMIC simu	Solver	Expert			

To modify the simulation options, either select another preset from the drop down box -if multiple presets are defined- or modify a setting in the dialog. In the latter case the preset <none> will be created with the contents of the selected preset and the modification just made. The selected preset will change to <none>. The content of the previously selected preset is not altered.

Mom uW	Simulation Options					
 Eayout Partitioning 	Preset: <none></none>	Save As				
Substrate	Descr <none></none>	Expert				
Frequency plan Output plan	Global Layer Specific	Net Specific				
Options	Simulation temperature 30 °C 🔹					
Resources	Model type for currents					
Notes	Thick conductor 3D-distri					
	Via 3D-distri	ibuted 🔻				

All other modifications to fields in the options dialog will be stored in <none>. If you are pleased with the modified settings you can save them as your own preset using Save As... and provide a meaningful name.

The names of presets imported from EM Templates or other EM Setups are reserved: they cannot be used as the name for a Save As... . They can be removed or renamed. The presets created with Save As... can be overwritten, removed or renamed. When you expand the preset drop down box, presets created with Save As will be shown in bold, imported presets in plain text as shown below.

🔁 Layout		
Partitioning	Preset: <none></none>	Save As
Substrate	MMIC	
1 Ports	Descr Preset_MMIC	Expert
	<none></none>	
Frequency plan	Global Layer Specific Net Specific	
M Output plan		
E Options	Simulation temperature 35 °C •	
Resources	Model type for currents	
Model	Annual and a second sec	
Notes	Thick conductor 3D-distributed -	
in Notes		
	Via 3D-distributed •	

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:

- Specifying Physical Model Settings for Momentum
- Specifying Physical Model Settings for FEM
- Defining Preprocessor Settings
- Defining Mesh Settings for Momentum
- Defining FEM Mesh Settings
- Guidelines for Specifying FEM Mesh Settings
- Performing FEM Broadband Refinement
- Defining Solver Settings
- Defining Expert Options for Momentum

Specifying Physical Model Settings for Momentum

Specifying Physical Model Settings for Momentum

The physical model options provide control over how the surface current flow on thick conductors and vias will be modeled. You can specify these options either globally, for selected layers or for selected nets.

Thick Strip Conductor Models

3D Distributed

This option provides the most accurate modeling approach.



- Top, bottom and side-wall currents seperately modeled
- Any surface current flow

2D Distributed

This option is preserved for compatibility with older versions. It may also serve to more efficiently model a stacked conductor configuration where a via array interconnects top and bottom conductor.



- Top and bottom currents seperately modeled
- Vertical side-wall currents only

Sheet

This option provides an approximate but faster modeling approach that can be useful to speed up the simulation of thick planes.



- Top and bottom currents collapsed
- No side-wall currents
- Via Conductor Models

3D Distributed

This option provides the most accurate modeling approach.



- DC resistance and skin effect

- All self and mutual inductance and capacitance
- Any surface current flow

2D Distributed

This option is useful when the via object is the result of a via array simplification. In such via array, no horizontal current can flow and this model will exclude such current.



- DC resistance and skin effect
- All self and mutual inductance and capacitance
- Vertical side-wall currents only

Wire

This option removes the via from the structure before meshing and adds a circuit element network connection between mesh cells at the top and bottom of the via.



- DC resistance and skin effect
- Wire self inductance
- Mutual wire-wire and wire-infinite ground inductance
- Mesh-less equivalent circuit model

Lumped

This option removes the via from the structure before meshing and adds a circuit element network connection between mesh cells at the top and bottom of the via.



- DC resistance and skin effect
- Wire self inductance
- Mesh-less equivalent circuit model

Lumped and wire via model options

Additional layout simplification options are available when the Lumped or Wire via model is selected. They can be chosen for faster simulation of layouts with a large number of vias. The following figure shows the lumped and wire via options:

The following table describes the new options:

Option	Description
Remove the via outline	Disabling the Remove the via outline option includes via outline in the mesh, which generates an accurate modeling of the via current flow. If you enable this option, the via outline is ignored during meshing, which results in a mesh with fewer cells around the via connections.

Option	Description
Remove pads not connected to a trace	Disabling the Remove pads not connected to a trace option includes the via pads in the mesh, which generates an accurate modeling of the via current flow. If you enable Remove pads not connected to a trace , the via pads (not connected to a trace) are ignored during meshing, which results in a mesh with fewer cells.
Remove antipads within radius	Disabling the Remove antipads within radius option includes the via antipads (clearances in the ground plane) in the mesh, which generates accurate modeling of the ground plane current flow. If you enable Remove antipads within radius , the via antipads within a radius smaller than the specified number of via radii is ignored during meshing, which results in a mesh with fewer cells.
Remove thermal reliefs within radius	Disabling the Remove thermal reliefs within radius option includes the via thermal reliefs in the mesh, which generates accurate modeling of the via current flow. If you enable Remove thermal reliefs within radius , the via thermal reliefs within a radius smaller than the specified number of via radii is ignored during meshing, which results in a mesh with fewer cells.

NOTE If you convert a project from an older ADS version, the wire vias options are disabled, by default. Therefore, results are same in the original project and the converted workspace.

The lumped and wire via models are subject to the following restrictions:

- Vias that are strictly partially connecting to a pad, trace, or plane cannot be simulated using a lumped or wire model. The 3D Distributed model will be used instead.
- Via shapes whose center is outside the shape uses the 3D Distributed model for simulation.
- Via outlines can only be removed from the mesh if they stay clear of other outlines.
- No current can flow in a lumped or wire via stub having an unconnected end. For example, if an outer pad is removed, current cannot flow in a lumped or wire via. Such vias are automatically discarded.

Defining Physical Model Options

You can specify the physical model options either global (lowest precedence), for selected layers or for selected nets (highest precedence).

Defining Global Physical Model Options

To specify physical model definition globally:

- 1. Choose EM> Simulation Setup to open the EM Setup window.
- 2. Select Options in the left pane of the EM Setup window.
- 3. Click the Physical Model tab.

4. Ensure that the Global tab is displayed, as shown in the following figure:

fault	Copy as Rename Remov
escription Physical N	1odel Preprocessor Mesh Solver Expert
Global Layer Specif	fic Net Specific
Thick conductor	3D-distributed
Via	3D-distributed
Wire: R, L 2D Distributed: Verti 3D Distributed: Verti	(mutually coupled) ical currents only ical and horizontal currents
Lumped and wire via m	odel options outline
Remove pads no	t connected to a trace ne outer ends of the pad stack
Maximal pad radius	: 3 via radii 💌
Remove antipad	s within radius 5 Via radii 💌
Demove thermal	reliefs within radius

- 5. Select a value from the Thick Conductor drop-down list.
- 6. Select a value from the Via drop-down list.
- 7. Select the required lumped and wire via model options.

Defining Layer Specific Physical Model Options

To specify physical model 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. Click the Physical Model tab.
- 4. Click the Layer Specific tab, as shown in the following figure.

efault					Copy a	as	Rename		Remove
De	scription	Physical I	Model	Preprocess	or M	1esh	Solver	Expert	
-(Global	Layer Speci	fic	Net Specific					
			Mog	del Type for Cu Thick Conduc	urrents tor	Mod	lel Type fo Via	r Current	s
	<global> DRILL_THROUGH ETCH_BOTTOM</global>		3D-distributed - 3D-distributed		3D-distributed 3D-distributed -				
	ETCH_L3		3D-distributed		-				
	ETCH_L4		3D-distributed		-				
	ETCH_TOP		3D-distributed		-				
	SLOT_L2_	GND	-			-			
	SLOT_L5_	VDD	-			-			

- 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 Net Specific Physical Model Options

To specify physical model definition for a specific net:

- 1. Select EM> Simulation Setup to open EM Setup window.
- 2. Select Options in the left pane of the EM Setup window.
- 3. Click the Physical Model tab.
- 4. Click the Net Specific tab, as shown in the following figure:

Simulation Op	tions						
Default							•
Description	1	Physical Model	Preproces	sor	Mesh	Solver	Expert
Global	La	yer Specific	let Specific				
	Model Type for Currents Thick Conductor			Mod			
<glob< th=""><th colspan="2"><global> 3D-distri</global></th><th colspan="2">uted</th><th colspan="3">3D-distributed</th></glob<>	<global> 3D-distri</global>		uted		3D-distributed		

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.

Specifying Physical Model Settings for FEM

Specifying Physical Model Settings for FEM

You can specify the physical model parameters such as model type, substrate layout and outline, and merge adjacent layers with same material properties in an FEM simulation. You can specify these parameters at the global or layer level and set advanced physical model options for FEM simulations.

Specifying Global Options

Model type for metals

The fields inside a good conductor decay exponentially with the decay rate given by the skin depth. To get accurate results for the fields inside a good conductor, it is necessary to generate a mesh that has a cell size that is smaller than the skin depth. At high frequencies, this would require very small cells and is thus computationally expensive. The FEM simulator offers a choice to approximate the fields inside good conductors to reduce this expense. You can specify these choices by specifying the "Model type for metals" options: Surface Impedance and Mesh Interior.

In FEM simulations, the default behavior is to not model the fields inside conductors. Instead, an approximate Surface impedance model is used on the surface of conductors. This gives the correct loss modeling at high frequencies. It also gives the correct DC losses for conductors with a uniform cross section. Slightly more accurate DC and low frequency losses for structures of varying cross section can be obtained by selecting the Meshed interior option. However, this can significantly degrade the accuracy of the high frequency losses from FEM simulations.

Substrate and Layout Outline

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

Additionally, since the FEM Simulator uses a Finite Element simulation technique, the EM problem domain needs to have a finite extent. Even if your actual substrate is much larger than the circuit you want to simulate, the problem domain will be automatically truncated by the FEM Simulator. The following options enable you to control how this truncation is computed:

Substrate LATERAL Extension

Prior to simulation, the FEM Simulator will create a bounding box that surrounds the layout. All the four edges of this bounding box are extended in the X- and Y- directions by the distance specified, unless:

- A Box is defined (the Box specification is used)
- A Waveguide line is parallel to the bonding box edge (the Waveguide specification is used)
- A TML calibrated port lies on the bounding box edge (the domain is truncated at the port)

The FEM Simulator problem domain is truncated by this modified bounding box. You can view the effect of this truncation by selecting FEM > 3D EM Preview.

Substrate VERTICAL Extension

If the uppermost or lowermost substrate layer is specified as an Open Boundary, then the uppermost/lowermost substrate will be constructed as a layer with the given Vertical Extension thickness and an Open (absorbing) boundary condition to truncate the problem domain.

The FEM Simulator problem domain is truncated by this modified bounding box. You can view the effect of this truncation by selecting FEM > 3D EM Preview.

Substrate Wall Boundary

The selected boundary condition is applied to all the side walls of the problem domain. The boundaries that can be selected are:

- Open: an absorbing boundary is applied to approximate an infinite extension.
- Perfect Conductor: the side walls are perfect (lossless) conductors.
- Perfect MagWall: the side walls are perfect (lossless) magnetic walls.

Merge Adjacent Layers with same Material Properties Option

using the Merge Adjacent Layers with same Material Properties option, you can merge adjacent layers that share the same material properties.

In an FEM simulation setup, you can specify three types of merge options: Merge Adjacent Layers with same Material Properties option in the Physical Model tab, Merge shapes touching each other where possible in the Preprocessor tab, and Merge Objects with Same Material in the Advanced subtab of the Mesh tab. The Preprocessor merge is done first. Layout objects, drawn on the same layer, touching each other are merged. Shared edges are not preserved during mesh generation. The merge option in the Physical Model merges adjacent layers that share the same material properties. Finally, during FEM mesh generation, and after the 3D objects are added if any, another merge can happen, based on material properties.

To specify the physical model settings 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 Physical Model tab.
- 5. Ensure that the Global subtab is displayed, as shown in the following figure:

Global	Layer Specific	Advanced					
Model ty	pe for metals		Surface impedance	•			
Substra	te and layout outli	ne					
O	ustom						
	Substrate LATE	RAL extension	3.125	mm 🔻			
	Substrate VER	TICAL extension	5	mm 🔻			
Substrate wall boundary Open							
Merge adjacent layers with same material properties							

- 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 Options

In the Layer Specific tab, you can set the model type for metals value for each layer independently. There are following three options:

- <Global>: take the model type for metals defined in the Global tab
- Surface impedance: See #Model type for metals.
- Meshed interior: See #Model type for metals.

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 Physical Model 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.

Simulation Options	S	
Default	Copy as Rename	Remove
Description	Physical Model Preprocessor Mesh Solver E	Expert
Global La	ayer Specific Advanced	
	Model Type for Metals (only if material = resistivity or conductivity)	
<global></global>	Surface impedance	
diel	Surface impedance	
pc1	Surface impedance	
pc2	Surface impedance	
pcvia1	Surface impedance	

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

Specifying Advanced Options

The Advanced options are used to determine the port surface for calibrated (TML) ports that are on the bounding box of the geometry. You can set the Port Surface Scaling and Port Surface Boundary options.

Port Surface Scaling

The port surface scaling options control how large the port surfaces are that the FEM simulator will generate for calibrated (TML) ports on the bounding box of the layout. You set the following options:

- Port LATERAL Scale: The port surface will be extended laterally at both sides *Port LATERAL scale* times the width of the conductor connected to the positive pin of the port in case of an infinite ground plane. In case of a finite ground plane the port surface will be extended *Port LATERAL scale* times the widht of the smallest bounding box containing both strips.
- Port VERTICAL Scale: The port surface will be extended vertically *Port VERTICAL scale* times the vertical distance between the positive and negative pin of the calibrated port. In case of a finite ground plane the port surface is extended both above and below the ground plane. In case of an infinite ground plane only at one side, the side where the positive pin of the port is located.

By setting these options to a very large value (say 1.0e6 or larger) you can ensure that the port surface covers the entire boundary surface. This will give the most consistent value for port de-embedding and is recommended when the Reference Offset of the port is non-zero. In the following example, a port has a positive pin on a strip and a negative pin on a strip or plane that is wider and is a vertical distance H below the positive pin. The smallest bounding box containing the two strips has width W:



The port surface consists of a vertical extension from a distance 'Port VERTICAL scale x H' above the positive pin and the same distance below the negative pin. The horizontal extension has a distance span of 'Port LATERAL scale x W' at both sides of the bounding box containing the two strips.

The scaling factors are 0 are larger, which means that the minimal port surface is the box with dimension W ${\rm x}$ H.

If the scaling factors are large, the port surface can be cropped by the bounding box of the geometry.

Treat finite strip (ground plane) as infinite plane

In cases where one of the pins, typically the negative or ground pin, is on a strip or plane that is much wider than the strip on which the other pin, typically the positive pin, lies, it is normally desired to model this strip as an infinite plane and set it as a border of the port surface. For this, you can use the parameter Finite (ground) plane detection factor, below abbreviated to 'f'. If the smallest strip has a width W and the widest strip spans a distance of at least f x W at both sides of the smallest strip, the widest strip/plane is treated as an infinite plane. Note that this does not only influence the vertical extension, but also the horizontal extension as the horizontal extension is now a factor of the smallest strip instead of the bounding box containing both strips.



The *Finite (ground) plane detection factor* does not take care of the polarity of the pins. If the positive pin is on the widest strip, then this strip is modelled as an infinite plane and becomes the border of the port surface. Hence, the brackets around 'ground' in the name of the parameter.

Setting Advanced 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 Physical Model tab.
- **5.** Select the Advanced subtab to specify advance options for your FEM simulation.

efault	 Copy as. 	Re	name	Remove
Description Physical Model	Preprocessor	Mesh	Solver	Expert
Global Layer Specific	Advanced			
Port Surface Scaling				
Port LATERAL scale	100			
Port VERTICAL scale	100			
Port Surface Boundary				
Finite (ground) plane	detection factor	4		

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

8. Type a value in the Finite (ground) plane detection factor text box.

Defining Preprocessor Settings

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.

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. The global preprocessor options applied to all shapes alter the way the EM meshers interpret the input layout geometries to improve the quality and complexity reduction of a generated mesh.

In addition, preprocessor options are provided that simplify specific patterns in the layout: metal fill and via arrays. Metal fill

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. Click the Preprocessor tab.

escription	Physical M	odel Pre	processor	Mesh	Solver	Expert	
Global: All Sha	apes G	lobal: Patter	ns Laye	Specific			
Show Visual A	Aid						
Heal the la	avout						
nearby edge	es and vert	tices will be s	napped toge	ther			
Auto-de	termine a s	safe snap dis	tance (cons	ervative)			
O User spe	ecified snap	o distance:	0		um	-	
V Merge sha	pes touchi	ng each othe	er where pos	sible ——			
shared edg	es in the m	esh will not b	preserved	I			
✓ Simplify th	e layout —						
vertex coun	t will be re	duced, witho	ut changing	the topolo	gy		
Constraints	(upper limi	ts)					
Displace	ment 1		% of the v	vavelengtl	h 🔻		
Shrinkag	e/growth	7.6	% or arc re	solution	45 0	degrees	
📃 Use diffe	erent const	traints for via	as				
Displace	ment 1		% of the v	vavelengt	h 💌		
Shrinkag	e/growth	7.6	% or arc re	solution [45 0	degrees	
Never ch	nange lines	whose lengt	th exceeds	0		um	-
Ignore sha	apes on the	e following pu	urposes:				
dummy dun	nmy2						
Concrate :	and replace	a change on	derived lave				
		e shapes on		5			
Scale the layout after preprocessing							
Scale Tactor	1						
Save prep	rocessor m	essages as a	a DRC result				_
Job name	EmPpMsgs						
DRC layer	255						

The **Preprocessor** options are organized in 3 tabs.

Preprocessor Options Applied to All Shapes

The preprocessor settings alter the way the EM meshers interpret the input layout geometries to improve the quality and complexity reduction of a generated mesh. You can write the preprocessed version of the layout to a new cell view.

NOT

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

You can specify the following global preprocessor options that will be applied to all shapes:

- Layout healing: Layout healing merges vertices and edges in close proximity. It corrects numerical issues in the layout that obstruct the creation of a highquality 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 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/√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:



- In an FEM simulation setup, you can specify three types of merge options: Merge Adjacent Layers with same Material Properties option in the Physical Model tab, Merge shapes touching each other where possible in the Preprocessor tab, and Merge Objects with Same Material in the Advanced subtab of the Mesh tab. The Preprocessor merge is done first. Layout objects, drawn on the same layer, touching each other are merged. Shared edges are not preserved during mesh generation. The merge option in the Physical Model merges adjacent layers that share the same material properties. Finally, during FEM mesh generation, and after the 3D objects are added if any, another merge can happen, based on material properties.
- 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.





Layout Scaling after Preprocessing: The layout scaling feature scales an entire design in the X and Y direction, but not in the Z direction. Pins are also taken into account. The scale factor value is forwarded to the simulation engine that multiplies all X and Y coordinates with the given scale factor. For example, a scale factor of 0.9 will reduce the layout by 10%. A scaled version of the layout is simulated. Any 3D visualization and simulation is done including the XY-scaling. When the mesh is written, the scaling is reversed to have the mesh match the original layout. The 3D viewer will also show the non-scaled layout. The currents visualization will show the dimensions of the simulated structure, therefore, it is recommended to turn off the visibility of the 3D objects, as they are not scaled. To set the scale factor, select the Scale the layout after preprocessing option. Type a value in the Scale factor field. The default scale factor is 1.0 (no scaling).

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.
	This option controls whether edges shared between adjacent shapes need to be preserved in the result.

Option	Description
Merge shapes touching each other where possible	
Simplify the Layout	This option toggles the usage of the layout simplification feature. In this section, you can specify the following parameters:
	 Constraints (upper limits):
	 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: You may need to simplify the vias more than the rest of the layout. You can specify a different set of constraints for vias: Displacement and Shrinkage/growth limit.
Ignore shapes on the following purposes	Shapes on the indicated purposes are ignored. An application of this feature is to ignore dummy metal fill, although more advanced options are now available, see Preprocessor Options to Model Unrelated Metal Fill. Use spaces to separate the purposes.
Generate and replace shapes on derived layers	When enabled, existing shapes on derived layers are ignored and the layers are regenerated. When disabled, only the existing shapes are being used.
Scale the layout after preprocessing	When enabled, scales an entire design in the X and Y direction, but not in the Z direction
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). Refer the DRC result or the layout processing report to verify these changes.
- Do not use layout healing as a polygon simplification tool. It is recommended to avoid choosing a large snap distance. For this purpose, the layout simplification feature provides a better and safe alternative.

Preprocessor Options to Simplify Via Arrays

Simulating via arrays in detail requires significant simulation resources. For typical RFIC designs, accurate results can obtained much faster by merging them upfront. The underlying assumption is that the impedance presented by the via array is low and has little effect on the device performance.

Merging Standard Via Arrays

When this option is enabled, standard (i.e. OA) via arrays are merged before being passed to the EM simulation engine. The merge replaces the individual cuts of standard via arrays by their hull. Non-standard (non-OA) via array cannot be merged by this option. Consider to use the alternative method to merge rectangular via shapes during the layout preprocessing step of an EM simulation.

Merging Rectangular Via Shapes

The merge of rectangular via shapes occurs during the layout preprocessing step of an EM simulation. Two distinct types of via clustering are provided: Local Via Array (recommended) and Stacked Conductor.

Local Via Array

Only axis aligned or 45 degree turned rectangles are being considered. The other shapes are left unchanged. First the shapes are grouped per contiguous area where the top and bottom via cover presence are the same (present or absent). Shapes having their center exactly on a boundary between areas are left unchanged. In each such group, the algorithm looks for arrays of exactly identically sized rectangles. The arrays can be axis aligned or 45 degree turned. The spacing between the cuts across the array axes does not need to be uniform. The combination of axis aligned rectangles and 45 degree arrays (or 45 degree rectangles are square.

You can optionally limit the distance between cuts where arrays are detected by setting Max Space, either in absolute terms (e.g. 1 um) or as a multiple of the cut size (e.g 3 via sides) along the direction of the array (i.e. for 45 degree array of axis aligned squares, the diagonal is taken into account). Specify a strictly positive value to turn this on. 0 will turn it off.

The shapes of arrays with only one cut are left unchanged. Otherwise, the algorithm looks at the array as a bitmap of cuts. For each bitmap, a hull around the cuts is constructed and limited to fit inside the contiguous top/bottom area. The resulting shapes become "cluster shapes". The bitmap can be modified before cluster shape construction occurs. You can optionally divide an array into smaller arrays not larger than not larger than a specific dimension by specifying a strictly positive Max Array Dim. Arrays will not be larger than this amount of pixels in either dimension. This allows to find a middle ground between all original cuts and merging everything.

In the second stage, final shapes are generated from cluster shapes according to the selected clustering Method:

- keep boundary: returns the cluster shapes as-is.
- keep area: shrinks the cluster shapes until they have the same area as the original via cuts they are replacing
- keep boundary and area: constructs donut shapes with the same outer boundary as keep boundary and the same area as keep area.
- increment and clip to lid boundary: extends the cluster shape bitmap horizontally, vertically and diagonally by 1 pixel. The cut distance applied to the extension is the average cut distance in that dimension (or in the other dimension if the array is 1D). This allows you to make arrays that almost fill the top/bottom conductor they are in to fill the conductor surface up to the boundary.

Stacked Conductors

In the first stage, each contiguous intersection area of interface layer masks present at both the top and bottom via interfaces becomes a cluster shape, provided there is at least one via shape present in the intersection. Sections of via shapes where there are no shapes on both top and bottom interface masks are discarded. In the second stage, final shapes are generated from cluster shapes according to the selected clustering Method:

- <upper> AND <lower>: returns the cluster shapes as-is.
- shrink <upper> AND <lower>: shrinks the cluster shapes with the distance amount specified by Shrink Size.

Specifying the Preprocessor Settings

The via simplification options are specified globally, i.e. applied to patterns on all via layers, but can be overriden for a specific layer. **Global**

Description	Physica	Model	Preproce	essor	Mesh	Solver	Expert	
Global: All S	hapes	Global: I	Patterns	Layer	Specific			
Unrelatad m	netal fill —							
Presence : Not used								
Model :	not ap	plicable						-
Via simplifica	ation —							
Merge	standard	oa via ar	rays					
🔽 Merge	rectangu	lar via sha	apes —					
Type:	۲	Local Via	Array 🤇) Stack	ed Conduc	tor		
Method	:	keep bo	undary				•	
Max Sp	ace:	3				v	ia sides 🔻	
Max Arr	ray Dim:	0						

Layer Specific

escription	Physical Model Preprocessor Mes	sh Solver Expert
Global: All Sh	apes Global: Patterns Layer Speci	ific
Strip Via		
	Destruction (Constitution	r
(Clabal)	Rectangular via Simpli	- Dire
< Global>	Local Array, keep bound, space = 3 via	aDim
KV	Local Array, keep bound, space = 3 via	aDim
VIAI	Local Array, keep bound, space = 3 via	aDim
VIA2	Local Array, keep bound, space = 3 via	aDim
VIA3	Local Array, keep bound, space = 3 via	aDim
VIA4	Local Array, keep bound, space = 3 via	aDim
VIA5	Local Array, keep bound, space = 3 via	aDim
VIA6	Local Array, keep bound, space = 3 via	aDim
у0	Local Array, keep bound, space = 3 via	aDim
y1	Local Array, keep bound, space = 3 via	aDim

Preprocessor Options to Model Unrelated Metal Fill

RFIC designs often contain unrelated metal fill (aka dummy fill or tiling) on selected strip layers. Simulating those in detail requires significant simulation resources. Often, when keep-out regions are respected, the metal fill will have limited impact and can be ignored during the simulation. In case of a uniform metal fill distribution underneath the RF device, one can obtain fast simulation results by replacing the metal fill with an equivalent dielectric layer. The preprocessor tries to identify the metal fill and takes action based on the selected modeling approach.

Presence

Presence option specifies how unrelated metal fill is distinguished from functional metal.

- Not used: there will be no identification of unrelated metal fill.
- Used but not present in layout: the metal fill is not drawn in the layout but a certain percentage of the area might be uniformly filled on selected layers before tape out. This selection is specified per strip layer and goes with the equivalent dielectric layer modeling approach.
- Drawn in layout on dedicated purposes: all shapes drawn in the specified Dedicated purposes will be treated as unrelated metal fill. Use spaces to separate the purposes.
- Drawn in layout as individual shapes: all shapes that fit within the Enclosing bounding box size will be treated as unrelated metal fill.

Modeling Approach

The Model option specifies the modeling approach for the unrelated metal fill.

- Remove unrelated metal: the unrelated metal fill will be ignored in the simulation.
- Keep unrelated metal: the unrelated metal fill will be simulated as-is.
- Replace by equivalent layer: the unrelated metal fill is removed from the layout passed to the simulator and the substrate stack is adjusted with an equivalent layer. This approach assumes a uniform distribution of the unrelated metal fill, no other usage of that layer and requires an area fill fraction which is the fraction of the area that gets filled. This is a value between 0 and 1.

Specifying the Preprocessor Settings

The unrelated metal fill options are specified globally (i.e. applied to patterns on all strip layers) but can be overriden for a specific per layer.

Global

Description	Physica	l Model	Preproce	essor	Mesh	Solver	Expert		
Global: All S	hapes	Global: I	Patterns	Layer	Specific				
Unrelatad m	netal fill —								
Presence	Presence : Not used								
Model ·	not an	plicable						_	
Model .	[IIUC ap	plicable							
Via simplifica	ation ——								
Merge	standard	oa via ar	rays						
🔽 Merge	rectangul	lar via sha	apes —						
Type:	۲	Local Via	Array 🤇) Stack	ed Conduc	tor:		_	
Method	:	keep bo	undary				•		
Max Spa	ace:	3					via sides 🔻		
Max Arr	ray Dim:	0							

Layer Specific

Unrelated Metal Fill Not used
Not used
Not used
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Replace shapes in bbox < 5 um by using 0.15 as fill factor for e
Not used
Not used
Not used

Generating Preprocessed Geometry

Before simulating your design, you can preprocess your design to automatically perform layout healing, merge shapes, and simplify the layout. To generate a preprocessed geometry:

- 1. Select Preprocessed geometry from the Generate drop-down list in the EM Setup window.
- 2. Click Go.

Defining Mesh Settings for Momentum

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 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.

Defining a Mesh

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



Creating a mesh consists of two parts:

- Defining mesh parameters
- Previewing the mesh

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

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.

Adjusting The 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, you specify 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.

About the Edge Mesh

An edge mesh is important when conductors are modelled as a sheet. 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 conductors, 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 sheet 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 use of an edge mesh is typically not required when the conductors are defined as thick conductors with a 3D-Distributed model. In the latter case, the current crowding at the edges is already accounted for by the horizontal current flowing on the side walls of the thick conductor. Enabling the edge mesh for thick conductors will additionally allow to capture the current crowding effect in corners. However, that is secondary effect that in the vast majority of cases is not required for a sufficient accuracy.

NOTE

- The edge mesh option is available at all levels of meshing: global, per layer, per net or per object meshing.
- When enabled for thick conductors, an edge mesh is created at all 4 sides of the conductor: top, bottom and side walls.

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 transmission line simulations 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.

NOTE

- When enabled for thick conductors, a transmission line mesh is created at all 4 sides of the conductor: top, bottom and side walls.
- Specifying a transmission line mesh will significantly increase the number of unknowns and consequently increase the simulation time and memory requirements.

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 and edge mesh. Although the simulation that includes the edge mesh takes more time, the results are closer to the actual measured results of the filter.



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.



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.

All possible overlaps are being imprinted when using the Aggressive setting. This can lead to a very dense mesh. Therefore, when using the Normal option, 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. Edges of relevant capacitive overlap regions that don't pass both of the following two limit checks are imprinted:
 - The parameter oe_capacitor_accuracy sets an upper limit for the ratio between the effective maximal cell size of the victim and the effective overlap width.
 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.
 - The product of the parameters oe_capacitor_accuracy and

oe_capacitor_inv_rel_sens_to_delta_w_div_eff_h sets an upper limit for the ratio between the maximal cell size and the effective z-plane distance.

- 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. The Aggressive overlap extraction setting has been implemented in practice by forcing oe_pedantic to ON.

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)	momentum.cfg variable name
oe_capacitor_width_threshold	MOM_MESHER_OVERLAP_EXTH
oe_capacitor_accuracy	MOM_MESHER_OVERLAP_EXTH
oe_inductor_length_threshold	MOM_MESHER_OVERLAP_EXTH
oe_inductor_width_threshold	MOM_MESHER_OVERLAP_EXTH
oe_inductor_accuracy	MOM_MESHER_OVERLAP_EXTH
oe_capacitor_inv_rel_sens_to_delta_w_div_eff_h	MOM_MESHER_OVERLAP_EXTH
oe_maximal_cell_size_height_shrink_factor	MOM_MESHER_OVERLAP_EXTH
oe_overlap_width_height_grow_factor	MOM_MESHER_OVERLAP_EXTH
oe_maximal_overlap_width_change	MOM_MESHER_OVERLAP_EXTH
oe_pedantic	EMPREP_PEDANTIC_OVERLAI

To specify expert options, see Defining Expert Options for Momentum.

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 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.

⊿ Mom uW	Simulation Options
🔁 Layout	
Partitioning	Copy as Rename Remove
Substrate	Description Physical Model Preprocessor Mesh Solver Expert
10 Ports	
🔤 Frequency plan	Giobal Layer Specific Net Specific Shape Specific
WW Output plan	Mesh frequency
Options	Highest simulation frequency
Resources	Mesh frequency 4 GHz
Model	
Notes	Mesh density
	Maximum cellsize 0 um 💌
	Cells/Wavelength 60
	✓ Edge mesh
	Auto-determine edge width
	O Use edge width 0 mm 🔻
	Transmission line mesh
	Number of cells in width 0
	Mesh reduction
	Thin layer overlap extraction

- 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 Meshto 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 Autodetermine 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.
- **11.** Enable Thin layer overlap extraction o 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. The Normal setting is recommended for most layouts. The Aggressive setting is only recommended when tiny overlap differences are expected to lead to significant result variations as this option will considerably increase the problem size. For more information, refer to #Processing Object Overlap.



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:

▲ EM Mom uW	Simulation Options					
Layout	convertedFromD	SN	•	Copy as Rename Remove		
💮 Partitioning			(
🛀 Substrate	Description	Physical Model	Preprocess	or Mesh Solver Expert		
Dorts						
GH2 Frequency plan	Global	ayer Specific	Net Specific	Shape Specific		
MM Output plan		Mesh Density	Edge Mesh	Transmission Line Mesh		
Options						
Resources	<global></global>	60 CpW	Auto-det	No Transmission Line		
Model	cond	60 CpW	Auto-det	No Transmission Line		
Notes	cond2	60 CpW	Auto-det	No Transmission Line		

- 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.

Defining Net-specific Mesh Options

To specify mesh definition for a specific net:

- 1. Select EM> Simulation Setup to open EM Setup window.
- 2. Select Options in the left pane of the EM Setup window.
- 3. Click the Mesh tab.
- 4. Click the Net Specific tab, as shown in the following figure

▲ Mom uW	nulation Options
Layout	onvertedFromDSN Copy as Rename Remove
Substrate	Description Physical Model Preprocessor Mesh Solver Expert
10 Ports	
GHz Frequency plan	Global Layer Specific Net Specific Shape Specific
Output plan	Mesh Density Edge Mesh Transmission Line Mesh
Options	<global> 60 CpW Auto-det No Transmission Line</global>
Resources	
Model	N_2 60 CpW Auto-det No Transmission Line
Notes	N_3 60 CpW Auto-det No Transmission Line

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.

▲ EM Mom uW	Simulation Options
🔁 Layout	
Partitioning	Converted follows
Substrate	Description Physical Model Preprocessor Mesh Solver Expert
±₫ Ports	Children Lawren Caracife - Mark Caracife - Change Caracife
🖽 Frequency plan	Global Layer Specific Net Specific Shape Specific
MM Output plan	Attention
E Options	Shape specific options are applied to the layout as properties of the shape. Save the layout after Clear/Apply to make this data persistent
Resources	If override is checked the entered value will be used otherwise either the pet specific layer
Model	specific or global setting will be used.
Notes	
	The layout for shape selection is not available Retry
	Override mesh density
	🖉 Maximum cellsize 🛛 🗤 🔻
	Cells/Wavelength
	Override edge mesh
	🖉 No edge mesh
	Auto-determine edge width
	🔘 Use edge width 🛛 🗤 🔻
	Override transmission line mesh
	No transmisson line mesh
	Number of cells in width
	Clear Apply

WARNING

To enable shape-specific options, you need to select a shape in the layout before specifying the options. As soon as you click the **Shape Specific** tab from another tab, immediately move your mouse over to the layout window. Use the mouse to select a shape in layout and then go back to Shape Specific tab to select the required options. You must have the white cross hair at the tip of the mouse before selecting a shape. Otherwise, options will not be enabled. You can always go to another tab and then come back to this tab to invoke the white cross hair at the tip of mouse when you move your mouse to layout window.

- 6. Enable the Override Layer Specific Mesh Densitysection. 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 Meshsection. You can select one of the following options:

- No Transmission Line Mesh
- Cells in Width

Defining FEM Mesh Settings

Defining FEM Mesh Settings

To perform a FEM simulation, the entire 3D problem domain is divided into a set of tetrahedra (or cells). The pattern of cells, called the mesh, is based on the geometry of a layout and optionally, user-defined parameters, so each layout has a unique mesh calculated for it. The mesh is then applied to the geometry to compute the electric fields within each cell. It also helps to identify any coupling effects in the layout during simulation. From these calculations, S-parameters are then calculated for the layout.

The FEM simulator implements an adaptive meshing algorithm, where an initial mesh is generated and the electric fields (and S-parameters) are computed on that initial mesh for a single frequency or multiple frequencies if Performing FEM Broadband Refinement is used. An error estimate is generated for each tetrahedron. The tetrahedrons with the largest estimated errors are refined to create a new mesh where the electric fields (and S-parameters) are computed again. The S-parameters from consecutive meshes are compared. If the S-parameters do not change significantly, then electric fields (and 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.

Specifying Mesh Settings for FEM

To define FEM mesh settings using the EM Setup window:

- 1. Select EM > Simulation Setup to open the EM Setup window.
- 2. Select the FEM simulator.
- 3. Select Options in the left pane of the EM Setup window.
- 4. Click the Mesh tab, as shown in the following figure:

Layout	convertedFromDSN Copy as Rename Remov	ve
Substrate	Description Physical Model Preprocessor Mesh Solver Expert	
_ Ports GHz Frequency plan	Stop criterium Refinement Initial Mesh Advanced	
M Output plan	Delta error: 0.015	
Doptions	Consecutive passes of delta error required:	
Resources	Minimum number of adaptive passes: 1	
Notes	Maximum number of adaptive passes: 5	
	Use memory limit: 0 B	

In the Mesh tab, you can specify the following settings:

- Specifying Stop Criterium Settings
- Specifying Refinement Settings
- Specifying Initial Mesh Settings
- Specifying Advanced Settings

Specifying Stop Criterium Settings

Using the Stop criterium tab, you can specify delta error and adaptive phase values. The following figure displays the stop criterium options:

Description	Physical Model	Preprocessor	Mesh	Solver	Expert	
Stop criteri	ium Refinement	Initial Mesh	Advanc	ed		Â
		Delta error:	0.01			
Consecutiv	ve passes of delta e	rror required:	1		▲ ▼	
Mini	mum number of ada	ptive passes:	5		<u>*</u>	
Maxi	mum number of ada	ptive passes:	15		<u>.</u>	
🔽 Use me	emory limit: 0			В	•	
Automa	atically open mesh c	onvergence plo	t			Ξ

The following table describes the stop criterium options:

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. It is the allowable change in the magnitude of the vector difference for all S-parameters for at least the number of consecutive refinement passes specified in the field "Consecutive passes of delta error required".
Consecutive passes of delta error required	Specify the required number of consecutive passes with a delta error smaller than or equal to the value specified in the field "Delta Error".
Minimum number of adaptive passes	Enter the minimum number of refinement passes. The number of refinement passes is at least this number even if the convergence criterium based on the delta error is met with less refinement passes. If you have concerns about convergence, you can increase this value, otherwise, use the default value of 2.
Maximum number of adaptive passes	Enter the maximum number of passes to be attempted. If the number of refinement passes entered is reached before the delta error criterium is met, the refinement process ends, based upon this limit. Typically, a value between 10 and 20 is recommended.
Use Memory Limit	If the memory used by the FEM solver exceeds the specified memory limit, the mesh refinement process will end and the simulation will continue with the solve phase, even if the delta error does not satisfy the convergence criterium.

Specifying Refinement Settings

Using the Refinement tab, you can control the frequencies at which the mesh will be adaptively refined. The following figure displays the Refinement tab options:

escription	Phy	vsical Model	Preprocessor	Mesh	Simulation	Expert
Stop criteri	ium	Refinement	Initial mesh	Advanc	ed	
Refineme	ent fre	equency				
O Cho	sen au	utomatically afte	er each pass			
O Cho	sen au	utomatically afte	er initial pass			
🔿 Man	ual se	lection				
					CH ₇	
					GHZ	
				Add	De	lete

The following table describes the Refinement tab options:

Option	Description
Maximum frequency	The refinement frequency will be the highest frequency found in the frequency plan.
Chosen automatically after initial pass	Refinement frequencies are chosen by the solver. This strategy is recommended when you are not aware about the important characteristics that can occur in your frequency, but you know the frequency transitions are not extremely sharp (i.e. are not high Q resonances).
Chosen automatically after each	Refinement frequencies are not fixed, but can vary from level to level. This is an automated option and tends to generate accurate results, but it also tends to result in the longest simulation times. This strategy is recommended:
pass	 For structures with high Q resonances.
	 During final analysis of a structure where accuracy is preferred over fast simulations.
Manual Selection	Refinement frequencies are specified manually.

NOTE Performing FEM Broadband Refinement

Specifying Initial Mesh Settings

Using the Initial mesh tab settings, you can control how an initial mesh is generated. If the initial mesh is made finer, the adaptive mesh refinement may converge faster. However, a finer mesh results in higher computation times. Therefore, to avoid overmeshing, the initial mesh size should not be too small.

For some resonating structures, such as patch antennas, the adaptive refinement may fail to converge, if the initial mesh is not fine enough. This is because the resonance frequency depends heavily on the mesh and a small difference in this frequency causes a large delta S. In these cases, the initial mesh settings are required to get the adaptive refinement converging.

Global Initial Mesh Settings

Using the Global tab, you can set initial target size, automatic, and edge mesh length. The following figure displays the initial mesh options:

Stop criterium Refineme	nt Initial Mesh Advanced
Global Layer Specific	
Target mesh size	
Automatic (recomm	iended)
Minimal memory	
Custom 2.22045	e-16
Conductor Mesh	
Automatic	
Fixed for all conduct	ctors
Conductor edge mesh l	length: 0
Conductor vertex mesh	n length: 0
Number of bondwire Segn	nents: 4
Global Vertex, Edge and	I Surface Mesh
Vertex Mesh Length:	0
Edge Mesh Length:	0
Surface Mesh Length:	0

You can specify Target Mesh Size and Conductor Mesh options for FEM mesh.

Target Mesh Size

You can specify the following target mesh size options:

- Automatic (recommended): The initial target mesh size is determined automatically. The length is computed based on the frequency plan available when the request is made.
- Minimal memory: Minimum memory is used for calculating the mesh.
- Custom: Provides an initial denser mesh. If used, no edge in the initial mesh will be longer than the given length.

Automatic is the recommended setting. The target mesh length will be one third of the free space wavelength at the maximum frequency. It is a compromise between accuracy and simulation performance.

The target mesh length is used for the full simulation domain. No tetrahedron in the simulation domain will contain an edge which is longer than the target mesh length. If there is a large area of the simulation domain that does not contain any fields, then using a global target mesh length can generate too many tetrahedral and it is advised to revert to the "custom target mesh size" setting or use the "minimal memory".

When Minimal Memory is chosen the initial mesh will generate a mesh with the least amount of tetrahedra as possible. A typical mesh is then determined by the geometric detail of the design, the wavelength does not play any role at all. The advantage of using this setting is that no part of the design will contain more tetrahedral than necessary. However, there might be insufficient tetrahedra to have an accurate simulation result. When using the "Minimal Memory" it is advised to either:

- have a sufficient number of mesh refinement steps;
- or to further customize the mesh with additional dedicated settings.

The custom target mesh works similar to the automatic target mesh except that as a user you have the liberty to specify the mesh size. This choice can be used when the automatic derived mesh length is not sufficient. For example: not the maximum frequency but another frequency has to be used, or the design contains little to no free space and a different wavelength is important.

Conductor Mesh

You can specify the following conductor mesh options:

- Automatic: When automatic conductor meshing is selected, the mesher estimates where mesh lengths need to be reduced to improve the accuracy and convergence speed of the simulation. The algorithm to produce this mesh is based on the DC connectivity of geometry with ports. Only parts electrically connected to a port are affected. For those parts the edge and vertex mesh is set. The length of the edge and vertex mesh is based on the estimated width assuming the structure being excited starts with a strip-like construction. The mesh process will report the estimated width in the simulation output window.
- Fixed for all conductors: The following options are used for specifying the mesh for all conductors:
 - Conductor edge mesh length: Specifies a target mesh size of all edges that belong to the geometry of a conductor. Edges do not necessarily have to be straight lines. A value of 0 means that no initial conductor edge mesh length is applied.
 - Conductor vertex mesh length: Specifies a target mesh size for all vertices that belong to the geometry of a conductor. A value of 0 means that no initial conductor vertex mesh length is applied.
For all lengths a parameter or formula is allowed. The formula is always evaluated at the time the simulation is being created.

"Conductor Mesh" specifies the mesh lengths on conductive objects in the mesh. To obtain an accurate simulation result, typically the mesh on conductive objects is finer than on its surrounding elements.

There are two ways to specify the mesh lengths to be applied to conductors:

- Automatic conductor mesh: during the simulation process will extract the DC connectivity of the design and consider all conductive objects connected to a port. At each of the pins of the port the actual width of the conductive object is determined. The estimated width is reported at the start of the mesh generation process. That width is used to set the edge mesh length on the connected pieces of conductive objects. The length set for the edges is by default 1/5 of the width. This factor can be tweaked on the "Advanced" tab of the "Mesh/Refinement Properties".

Automatic conductor mesh performs a best-effort to determine the widths at all conductors and will report when there was a failure. The effective port /pin location contributes to the success of the estimation. Pins located at the edges of a conductor help disambiguating the width. The automatic conductor mesh will typically back out when a pin is connected to a curved surface or curved edge.

Fixed for all conductors: all conductors receive the specified mesh length. In contrast with the automatic conductors is there no difference between a conductor attached through a conductive path to a pin and a so called floating conductor.

Number of Bondwire Segments

This option enables you to specify the number of bondwire segments.

Global Vertex, Edge, and Surface Mesh

You can specify the following options:

- Global Vertex Mesh Length: Defines the value of the Global field used in Layer Specific Initial mesh tab.
- Global Edge Mesh Length: Defines the value of the Global field used in Layer Specific Initial mesh tab.
- Global Surface Mesh Length: Defines the value of the Global field used in Layer Specific Initial mesh tab.

NOTE To avoid difficult convergence during adaptive refinement and large number of unknowns, use the initial mesh options, Initial target mesh size, and edge mesh size.

Layer-specific Initial Mesh Settings

Using the Layer Specific tab, you can set the vertex, edge, and surface mesh options for each layer. The following figure displays the Layer Specific tab options:

Stop criterium	Refinement	Initial Mes	h Advanced
Global La	ayer Specific		
	Vertex Mesh	Edge Mesh	Surface Mesh
<global></global>	0 mm	0 mm	0 mm
diel	0 mm	0 mm	0 mm
pc1	0 mm	0 mm	0 mm
pc2	0 mm	0 mm	0 mm
pcvia1	0 mm	0 mm	0 mm

If a mesh setting of a layer is set to <Global>, it will inherit its value from the corresponding global mesh setting on the Global Initial Mesh tab, which is also shown on the first row in the Layer Specific tab.

It is also possible to enter a maximum cell size that differs from the global value. The value is then displayed in bold. A value of 0 always means that no initial mesh size is applied.

Vertex Mesh

Vertex meshing reduces the mesh length around vertices. When combined with edge meshing the shortest length of both is chosen in any given region. The region where the value is applied by the vertex mesh length setting is roughly twice its length in each direction: the actual region is still being optimized. The region affected by it depends on local geometry.

The following figure displays the result of a vertex mesh setting:



Edge Mesh

Edge meshing reduces the mesh length around the edges. A fan-out effect is created to maintain the tetrahedron quality. The edges do not need to be straight lines, any shape is allowed.

The following figure displays the result of an edge mesh setting.



Surface Mesh

Surface meshing reduces the mesh length on surfaces of objects. The following figure displays the result of a surface mesh setting.



Specifying Advanced Settings

Using the Advanced tab, you can specify the target mesh growth, initial mesh size, and automatic conductor mesh settings. The following figure displays the advanced options:

C	escription	Physical N	1odel	Prepro	cessor	Mesh	Solver	Expert
	Stop criteri	um Refi	nement	Initia	al Mesh	Advanc	ed	
		Targe	t mesh gr	rowth:	25			
	📃 Use init	tial minimal n	nesh size:		2.2204	5e-16		mil 💌
	Merge	objects with	same ma	aterial				
	Automati	ic conductor	mesh set	ttings: -				
	🔽 Edge	e meshing:	0.2			x estimate	d conducto	or width
	Verte	ex meshing:	0.06			x estimate	d conducto	or width

The following table describes advanced options:

Option	Description
Target Mesh Growth	Specifies the percentage by which each adaptive mesh grows relative to the previous mesh.
Use Initial Mesh Size	Enables you to set a minimum length of edges in the initial mesh. During mesh refinement, the individual length of edges in the mesh can be shortened where deemed necessary by the adaptive refinement process. Using this option, you can restrict the meshing process to a more limited set of possible solutions. It is advised to only use this option when the initial mesh is unreasonably dense in some areas due to some combination of geometric features triggering that behavior. This situation can arise close to curved surfaces or arcs or discretized curved surfaces or arcs with small angle results where the meshing process potentially over-meshes the structure. When this happens, you should use a value that is small enough to describe the curved shape. For example, choose a distance value that is equivalent to 15-45 degree resolution of a curved surface or arc.
Merge Objects with Same Material	This option controls if two objects that have the same material properties may be merged prior to meshing. This option can be used to reduce the number of tetrahedrons generated. Object merging is applied only when the objects also have the same mesh priority. In an FEM simulation setup, you can specify three types of merge options: Merge Adjacent Layers with same Material Properties option in the Physical Model tab, Merge shapes touching each other where possible in the Preprocessor tab, and Merge Objects with Same Material in the Advanced subtab of the Mesh tab. The Preprocessor merge is done first. Layout objects, drawn on the same layer, touching each other are merged. Shared edges are not preserved during mesh generation. The merge option in the Physical Model merges adjacent layers that share the same material properties. Finally, during FEM mesh generation, and after the 3D objects are added if any, another merge can happen, based on material properties.
	This option is used when automatic meshing is applied. It determines the length of the edge mesh in relation to the estimated width of the conductor.

Option	Description
Automatic conductor mesh setting Edge meshing	
Automatic conductor mesh setting: Vertex meshing	This option is used when automatic meshing is applied. It determines the length of the vertex mesh in relation to the estimated width of the conductor.

NOTE

Advanced meshing controls should only be changed in case of meshing problems.

Guidelines for Specifying FEM Mesh Settings

Guidelines for Specifying FEM Mesh Settings

Target Mesh Size

The Automatic Target Mesh size is a good choice for most applications. For strong radiating designs, it can be recommended to set a smaller Custom Target Mesh Size, for example, one third of the wavelength at which the radiation occurs. For these radiating designs a second order discretization is recommended. If first order discretization is used further reducing the custom target mesh size to one fifth of the wavelength of radiation produces more accurate results.

Use the Minimal Memory setting for non-radiating designs. The initial mesh is primarily determined by the shapes of the design itself instead of a wavelength. This is useful for designs where the complexity of the shapes of the designs is ensuring dense enough starting meshes in the regions where the electromagnetic fields are concentrated. A set of thick conductors running through the design where the thickness is smaller than the width of the conductors is a typical example.

Common designs sharing these traits are package and module designs.

Minimal Mesh Size

The Minimal Mesh Size Setting located on the advanced properties are used to prevent the meshing process from generating too small tetrahedra. This situation can arise close to curved surfaces where the meshing process potentially overmeshes the structure. When this happens you should use a value that is small enough to describe the curved shape: e.g. choose a distance value that is equivalent to 15-45 degree resolution of an arc or curved surface.

Another situation where the minimal mesh size can be used is when you like to continue using the automatic conductor mesh setting but on some locations the estimated width leads to an unnecessary dense mesh.

Merge Objects with Same Material

Merging objects with the same material takes all objects that are assigned exactly the same material and share the same mesh priority and lumps them together in one object. Any seam lines that would arise from making the union of these objects are removed. The figure below shows the effect of merging of objects:



By default objects merged with same material is switched on. Using this option will typically give a smaller mesh with similar EM performance. Only in cases where it is deemed mandatory to keep seam lines between objects in the final mesh the option should be turned off. A possible use for this is to force objects lines to be in the mesh for seeding purposes.

Using Conductor Meshing

Using the Conductor Meshing you can control the density of the initial meshes close to conductors, e.g. to create a denser Edge Mesh type mesh there where you can expect a dense mesh is needed to get an accurate simulation. Using Conductor meshing to seed the initial mesh is useful to speed up the adaptive mesh refinement process.

You can use the Automatic conductor setting when the location of the pins of the port allow to get a good estimate on the edge and vertex values that would be required to obtain an accurate simulation. The estimated widths are reported by the simulator in the log window. Note that this setting will apply the edge to all conductors that are electrically connected at DC to the port (net). Any metal not connected to the port will not get this mesh seeding.

For the designs where metal not electrically connected to the ports is important for the electrical behavior, like a coupled line filter, the Automatic setting is not advised. In case multiple ports are connected to the same net the smallest estimate will be chosen. The Fixed for all conductors setting applies the specified Edge or Vertex setting to all the metal shapes in the designs which can be overkill in more complex designs. Applying the setting to specific models can be used to be more selective in applying the conductor mesh setting.

Choosing the Edge and Vertex Mesh Values

For microwave applications, like a coupled line filter, you can set the edge mesh value to 1/5th of the width of the strip.

For RF-type of applications, such as RF connectors, modules and packages, you can set the edge mesh value to 1/3th of the width of the strip.

Use vertex meshing to generate a locally denser mesh around vertices. The region in which the mesh is denser is limited and the impact on the total number of tetrahedral is often quite minimal. The results are that the accuracy of the inductance computation along the conductive paths will typically be more accurate.

If it is deemed overkill to apply the edge meshing on all conductors, apply the edge meshing at the minimum to the objects closest to a port.

Performing FEM Broadband Refinement

Performing FEM Broadband Refinement

You can automate the FEM broadband refinement process. You can select one of the following refinement strategies depending on your requirements:

- Multiple fixed refinement frequencies
- Refinement frequencies chosen automatically after the initial pass
- Refinement frequencies chosen automatically after each pass
- Manual refinement frequency

This section describes the refinement frequencies and provides guidelines for choosing a strategy.

Importance of Refinement Frequencies

Mesh refinement procedures in FEM typically consist of the following steps:

- 1. Simulate at the refinement frequencies.
- **2.** If the calculated S-parameters did not reach convergence, perform the following steps:
 - Estimate the discretization errors of the field solution.
 - Refine the mesh at locations with high values for the error.
 - Repeat step 1.

In this procedure, the refinement frequencies play a key role:

- Convergence monitoring: the convergence of the refinement process is monitored at the refinement frequencies (step 2).

- Field error estimation: the field error estimates, on which mesh refinement is based, are calculated at the refinement frequencies (step 2a).

Consequently, the choice of the refinement frequencies can have an important impact on the accuracy of the simulation results:

- The mesh convergence is monitored at frequencies where the S-parameters change most during mesh refinement. If this is not the case, the refinement process might stop too early, i.e., *before* the mesh refinement has reached convergence over the entire frequency range of interest.
- All important electromagnetic phenomena should occur at one of the refinement frequencies. If this is not the case, this phenomena cannot be taken into account during mesh refinement. As a consequence, it is possible that the final mesh is not suited to represent this phenomena accurately.

Choosing a Refinement Strategy in FEM

You can select one of the following refinement strategies depending on your requirements:

- Select the maximum simulation frequency for the adaptive refinement
- Refinement frequencies chosen automatically after the initial pass of the adaptive refinement
- Refinement frequencies chosen automatically after each pass of the adaptive refinement
- Manual specification of one of more refinement frequencies

This section describes these refinement frequencies and provides guidelines for choosing a strategy.

Performing Automatic Broadband Refinement

Multiple Fixed Refinement Frequencies

You can specify a set of refinement frequencies by selecting the button 'Manual selection', and adding frequencies to the frequency table. This refinement strategy behaves as follows:

- All refinement frequencies are simulated on each mesh level.
- Convergence is reached if the convergence criterium is satisfied for *all* refinement frequencies. In other words, convergence depends on the *worst* case refinement frequency on each level.
- The mesh refinement depends on the error estimates of *all* refinement frequencies.

The corresponding logfile shows:

- Delta(S) for each simulated frequency
- Worst case Delta(S) for each mesh level (last line)

REFININ	•										88
level	fraq(GHz)	ł	nb/Tetr	HESHING Elapsed time	CPU tine	nbUnknowns	men(GB)	SOLVING Elapsed time	CPU time	DeltaS	m
L	2.000 6.000 8.000		272	00:00:02.2	00:00:01.1	1848	0.300 0.300 0.300	00:00:00.8 00:00:00.1 00:00:00.0	00:00:00.9 00:00:00.2 00:00:00.1	/ /	
3	2.009 6.009 8.009 (8.009)		513	00:00:00.4	00:00:3	3478	0.308 0.308 0.308	00:00:01.1 00:00:00.1 00:00:00.1	00:00:01.4 00:00:00.2 00:00:00.2	0.007[->0.010] 0.020[->0.010] 0.026[->0.010] 0.026[->0.010]	
3	2.000 6.000 8.000 (8.000)		1219	00:00:00.8	00:00:00.7	8135	0.337 0.337 0.337	00:00:00.7 00:00:00.2 00:00:00.2	00:00:01.2 00:00:00.0 00:00:00.6	0.007[->0.010] 0.020[->0.010] 0.025[->0.010] 0.025[->0.010]	
•	2.009 6.009 8.009 (8.009)		1969	00:00:01.3	00:00:01.2	1 3048	0.364 0.364 0.364	00:00:00.9 00:00:00.3 00:00:00.3	00106/02 0	0.003[->0.010] 0.009[->0.010] 0.012[->0.010] 0.012[->0.010]	

Refinement Frequencies chosen Automatically After Initial Pass

In this refinement strategy, the refinement frequencies are not user-defined, but chosen by the solver. For this purpose, the entire target frequency plan is simulated on the initial mesh. Based on an analysis of the resulting S-parameters, a set of refinement frequencies is chosen. Typically, these frequencies are the frequencies for which the diagonal elements of the S-parameter matrix reach a minimum. The selected refinement frequencies are used in all subsequent refinement passes.



Except for the AFS sweep on the initial mesh level, this refinement strategy is identical to the strategy based on user-defined refinement frequencies.

	1	HESTRING			\$01//TING		
evel	freq(GHz)	nbTetr Elapsed time CPU time	abUnitnovna	nem(GB)	Elapsed time	CPU time	Delta(S)
1	0.200	8501 00:00:09.4 00:00:08.4	54582	0.596	00:00:03.4	00:00:07.3	1
	20.000			0.596	00:00:01.5	00:00:04.9	1
	11.400			0.596	00:00:01.5	60:00:04.9	1
	6.800		12 12 12 12 12	0.596	00:00:01.5	00:00:04.9	1.1.1
	3.500	AFS sweep on initial level,		0.596	00:00:01.5	00:00:04.9	1
	10.100	> used to determine the		0.596	00:00:01.5	0.00:05.00	1
	8.450		19 10 19 19	0.596	00:00:01.6	0.00:05.0	1
	11.750	refinement frequencies		0.596	00:00:01.5	00:00:04.8	1
	17.800			0.596	00:00:01.5	00:00:05.0	1
	16.333			0.596	00:00:01.5	00:00:05.4	1
	18.500			0.596	00:00:01.5	00:00:05.0	1
	10.350	V !		0.596	00:00:01.5	00:00:05.3	1
2	8.800	10524 00:00:04.7 00:00:04.5	67892	0.686	00:00:03.8	0.00:00:09	0.020[->0.010]
	18.300			0.586	00:00:02.1	00:00:00	0.026[->0.010]
	(18.300)	Fixed refinement freque	ncies				0.026[->0.010]
3	8.800	12516 00:00:05.9 00:00:05.7	80896	0.772	00:00:07.7	00:00:11.9	0.009[+>0.010]
	18.300			0.772	8.20:00:00	0.00:00:6	0.005[->0.010]
	(8,800)						0.009[->0.010]

Refinement Frequencies Chosen Automatically After Each Pass

In this refinement strategy, the refinement frequencies are not fixed, but can vary from level to level. This refinement strategy behaves as follows:

- The entire target frequency plan is simulated *on each mesh level using an adaptive frequency sweep*.
- The difference of the S-parameters of the *rational models is monitored*, instead of the difference between the S-parameters of the simulated frequencies only. In other words, convergence depends on the worst case frequency in the entire frequency range of interest. This is the most detailed convergence monitoring possible.
- Mesh refinement depends on the error estimates of *all simulated frequencies*

REFINING	1								
		1	MESHING		1		SOL//ING		
level	freq(GHz)	nbTetr	Elapsed time	CPU time	nbUnitnowns	nen(GB)	Elapsed time	CPU time	Delta(S)
1	0.200	8501	00:00:09.2	00:00:08.2	54582	0.658	00:00:02.9	00:00:06.6	/
	20.000	1			1	0.658	00:00:01.5	00:00:04.8	/
	13.400	1			1	0.658	00:00:01.5	00:00:04.7	/
	6.800	1			1	0.658	00:00:01.5	00:00:04.8	/
	3.500	1			1	0.658	00:00:01.5	00:00:05.0	/
	10.100	1			1	0.658	00:00:01.5	00:00:04.7	/
	8.450	1			1	0.658	00:00:01.5	00:00:04.9	/
	11.750	1			1	0.658	00:00:01.5	00:00:05.0	/
	17.800	1			1	0.658	00:00:01.5	00:00:04.9	/
	16.333	1			1	0.658	00:00:01.5	00:00:05.2	/
	18,900	1			1	0.658	00:00:01.5	00:00:05.4	/
	18.350	1			1	0.658	00:00:01.5	00:00:05.2	/
2	0.200	10524	00:00:04.6	00:00:04.5	67892	0.627	00:00:09.3	00:00:09.4	1.5e-04[->0.010]
	20.000	1			1	0.627	00:00:02.0	8.80:00:00	0.015[->0.010]
	13.400	1			1	0.627	00:00:02.0	00:00:07.1	0.019[->0.010]
	6.800	1			1	0.627	00:00:02.0	00:00:07.0	0.005[->0.010]
	3.500	1			1	0.627	00:00:02.0	00:00:06.8	0.002[->0.010]
	10.100	1			1	0.627	00:00:02.0	00:00:06.8	0.034[->0.010]
	8.450	1			1	0.627	00:00:02.0	00:00:06.9	0.015[->0.010]
	11.750	1			1	0.627	00:00:02.1	00:00:06.8	0.021[->0.010]
	17.800	1			1	0.627	00:00:02.1	00:00:07.0	0.037[->0.010]
	16.333	1			1	0.629	00:00:02.0	00:00:06.9	0.057[->0.010]
	18.900	1			1	0.629	00:00:02.1	00:00:06.9	0.020[->0.010]
	18.350	1			1	0.629	00:00:02.1	00:00:07.2	0.025[->0.010]
	(16.670)	1			1				0.060[->0.010]
	~							1 /	
		N Wor	rst case fre	equency a	nd corresp	onding	Delta(S)		
		1				5.151.16	2 2110(0)		

Selecting a Refinement Strategy

You can refer the following guidelines while selecting a refinement strategy:

- Multiple fixed refinement frequencies: This strategy computes the fewest frequency points during the adaptive refinement stage and therefore usually lead to the fastest simulations. This strategy is recommended for:
 - Electrically short transition structures. These can include matching networks, packages, and impedance transformations. If the transmission parameters accumulate less than 180 degrees of phase, it is sufficient to refine only at the highest frequency.
 - Designs where you know the frequencies of the critical features such as dual (or multi) band antennas and band pass filters. 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). It is advisable to 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 where the important characteristics occur in your frequency. However, you might 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 in the following situations:
 - Structures with high Q resonances.
 - During the final analysis of a structure where accuracy is preferred over fast simulations.

Defining Solver Settings

Defining Solver Settings

You can define the solver 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. The solver settings are EM simulator specific.

Defining the Solver Settings for Momentum

Specifying the Matrix Solver Settings

You can select the following types of matrix solvers used by Momentum during a simulation:

- Auto-select: If you select the Auto-select solver, Momentum automatically selects the most appropriate matrix solver. It is the default matrix solver selection option.
- Direct Dense: If you select Direct dense, Momentum selects the direct dense matrix solver for a simulation. This implies that the matrix is stored in a dense matrix format (requiring order N² memory) and solved using a direct matrix factorization technique (requiring order N³ computer time). The direct dense matrix solver guarantees to provide a solution using a predetermined number of operations. The main disadvantage is that the computer time it requires scales qubic with the matrix size N, yielding larger simulation times for large problem sizes.
- Iterative Dense: If you select Iterative dense, Momentum selects the iterative dense matrix solver for a simulation. This implies that the matrix is stored in a dense matrix format (requiring order N² memory) and solved using an iterative matrix solve technology (requiring order N² 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 monitors its convergence rate and automatically switches 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 NlogN memory) and solved using a direct compressed matrix factorization technique (requiring order (NlogN)^{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 it the preferred matrix solver for large problem sizes.

Defining the Compression Level

The optimal compression level is not automatically determined, but can be usercontrolled. You can select one of the following options:

- Normal: This selection compresses the matrix to a level that preserves the accuracy of the solution for the large majority of cases.
- Reduced: This selection lowers the amount of matrix compression. It can be used in cases where the normal compression level fails to provide an accurate solution. It may provide an accurate enough solution faster than the direct dense solver.
- Aggressive: This selection increases the amount of matrix compression. It can be used to explore speeding up a simulation.

Enabling the Port Solver

The Port Solver is a 2D EM cross-section solver that computes the characteristics of the transmission lines attached to TML-calibrated ports. Its simulation results are propagation constant and characteristic impedance. The propagation constant is written to dataset as 'GAMMA' and the characteristic impedance is written to dataset as 'Z0'.

You can either enable or disable the port solver. Enabling the port solver also produces an additional set of S-parameters in the dataset which are renormalized to this characteristic impedance. The port solver works only on conductors defined as infinitely thin sheet conductor in the Substrate Editor, and it is automatically disabled when ports are connected to thick conductors. When available, the propagation constants are used in the estimation of the wavelength for the mesh cell size.

To specify the simulation solver options:

- 1. Select EM > Simulation Setup to open the EM Setup window.
- 2. Choose either Momentum RF or Momentum Microwave in the Simulator panel.
- 3. Choose Options in the left pane of the EM Setup window.
- 4. Click the Solver tab.

Mom uW	Simulation Options	
🔁 Layout	convertedFromDSN	
🛞 Partitioning	converteurrombalv	Copy as Rename Remove
🋀 Substrate	Description Physica	al Model Preprocessor Mesh Solver Expert
🖞 Ports	Market and the	
🚥 Frequency plan	Matrix solve method	Auto-select
🚧 Output plan	Compression level	🔘 Reduced 💿 Normal 🔘 Aggressive
E Options	Port solver	Enabled Disabled
Resources	Davias files from th	a ana ing a kata
Model	Reuse files from th	e previous simuladon
Notes		

5. Specify the Momentum solver settings.

Defining the Solver Settings for FEM

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

- Auto-select: On selecting Auto-select, the FEM Simulator automatically determines to choose the Direct or Iterative solver based on a heuristic rule related to number of ports in the setup. When the design has less than 15 ports, the iterative solver is used. In other cases, the direct solver is chosen. For most designs, this leads to optimal performance of the simulation.
- Direct: On selecting Direct, the FEM Simulator uses a multi-threaded sparse direct solver. The memory requirements and computing time of this solver typically scale as N^(3/2) to N^2 respectively, where N is the matrix size. This solver does not suffer from potential convergence problems and is guaranteed to yield a solution if sufficient memory is available.
- Iterative: On selecting Iterative, FEM Simulator uses the iterative matrix solver. The memory requirements and computing time of this solver scale as N to N^(3/2) respectively where N is the the matrix size. In other words, the computing resources of the iterative solver are typically one order of magnitude lower then the computing resources of the direct solver. Especially, the iterative solver requires significantly less memory then the direct solver. However, contrary to the direct solver, the iterative solver is not guaranteed to converge.

You can also specify the Order of basis functions used to approximate the electric fields in all tetrahedra. You can set the following values:

- 2: Electric fields will be approximated with a quadratic variation in each tetrahedron, and the magnetic fields will be approximated with a linear variation in each tetrahedron. This gives the best accuracy.
- 1: Electric fields will be approximated with a linear variation in each tetrahedron, and the magnetic fields will be approximated as being constant in each tetrahedron. This requires less memory than the Order 2 basis functions given the same mesh, but typically requires a larger mesh to get the same level off accuracy. This setting is useful for electrically small structures.
- Mixed order: The mixed order scheme uses a combination of first and second order bases functions depending on the location in the mesh. This is an automated scheme that uses first order basis functions in regions with lower field variation. It leads to faster overall simulations for designs where there are big enough regions with low field values and/or small field variation.

Using Modal TML Port Solver

If you enable the Use modal TML port solver option, all the waveguide ports are solved using the legacy port solver algorithm. This option provides the following advantages:

- Numerically stable to a much lower frequency.

- Correct results for waveguides with independent conductors that are not associated with a port pin. Therefore, you need to setup only one port for a pair of coupled microstrip lines that terminate in the same plane.

However, it consists of the following disadvantages relative to the legacy port solver:

- Does not support de-embedding.
- Less accurate for dispersive waveguides.

```
NOTE Starting with ADS 2017 to generate Gamma, Zpi, Zpv, and Zvi outputs, you must enable this option.
```

Reusing Simulation Data

To reduce the time required to complete the current simulation, you can use sampled data from the previous simulation. Enabling the Reuse files from the previous simulation option reuses previously calculated S-50 parameters. This function is useful, if you extend or shift the frequency plan and re-simulate, or if you want more samples near a resonance frequency. If you change the circuit geometry, substrate, ports, or mesh parameters, the simulation data cannot be reused.

NOTE ADS does not verify that the design has not changed. Make sure that only the frequency plan has changed before attempting to reuse data.

To reuse data from the last simulation:

- 1. Select EM > Simulation Setup to display the EM Setup window.
- 2. Choose FEM in the Simulator panel.
- 3. Select Options in the left pane of the EM Setup window.
- 4. Click the Solver tab.
- 5. Enable Reuse files from the previous simulation. Sampled data from the previous simulation will be used in the current simulation, which can decrease the simulation time.

a Em FEM	Simulation Options
Layout Layout Partitioning Layout Substrate Dorts Frequency plan Output plan Output plan Options Resources W Model	Default Copy as Rename Remove Description Physical Model Preprocessor Mesh Solver Expert Matrix solve method Iterative Order of basis functions 2 Use modal TML port solver Ø Reuse files from the previous simulation

Using the Advanced Options

The following simulation settings are available by clicking the Advanced button:

- Maximal Number of Iterations: Specifies the maximum number of iterations allowed during the iterative solve process. This value is increased to achieve a lower absolute tolerance. Typically, a value 500 is sufficient.
- Tolerance iterative solver: Specifies the absolute error tolerance that the solver process must reach to achieve convergence. A typical value would be 1.0e-5. You may need to increase the Maximum Number of Iterations, if you significantly decrease the tolerance.

Defining Expert Options for Momentum

Defining Expert Options for Momentum

You can define configuration variables for a Momentum simulation in the EM Setup window. Use the *Expert* tab, which is present in *Options* of the EM Setup window.

WARNING These variables can alter the default behavior of the simulator. They should only be used to workaround a specific issue.

Defining Configuration Variables

- 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 Expert tab.

Simulation Option	าร		▼ Cop	oy as	Rena	ame	Remove
Description	Physica	al Model	Preproc	essor	Mesh	Solver	Expert
Ke	у	Va	lue				

- 5. Click Add to insert a new row.
- 6. Specify the name of the configuration variable in the Key field. Momentum Configuration Variables provides links to the appropriate sections where configuration variables are described.
- 7. Specify a value in the Value field.

NOTE

Variables specified on the Expert tab apply for this specific simulation only. All key-value pairs are written in the momentum.cfg file in the simulation directory (some variables are written in the proj.opt). This file has the highest priority when searching for configuration variables.

Momentum Configuration File

By default, the simulator searches for variables in momentum.cfg files in the following order, and uses the first one found:

- The simulation directory, <workspaceDirectory>/simulation
/<libName>/<cellName>/<layoutName>/<emSetupName>_MoM
/momentum.cfg

Variables defined on the Expert tab apply to that specific simulation.

- Your personal directory, \$HOME/hpeesof/config/momentum.cfg Define configuration variables here that apply to all your simulations.

NOTE On the PC, %HOME% represents the path you specified as the *Home Folder* during installation (e.g. C:\users\default)

- The custom configuration directory under the ADS Momentum installation, \$HPEESOF_DIR/Momentum/<version>/custom/config/momentum. cfg

Define configuration variables here that apply to all users. Configuration variables defined here will not be overwritten by installation of subsequent program patches or updates.

The configuration directory under the ADS Momentum installation directory
 \$HPEESOF_DIR/Momentum/<version>/config/momentum.cfg
 Define the default installed configuration variables here that apply to all users. Configuration variables should not be customized here.

Momentum Configuration Variables

The links below lead to specific locations in the documentation describing configuration variables.

- Mesh, processing object overlap
- Source type
- Conductor Loss Model
- Passivity Check

Specify (MOM3D_WRITE_PASSIVITY,TRUE) as Key-Value pair to verify the passivity of the S-parameters.

At each simulated frequency, the maximum eigenvalue of conj([S]^T)*[S] is computed. The square root of this maximum eigenvalue is available as PASSIVITY in the resulting dataset. Passivity requires that this value is smaller or equal than 1.

- Phase center for Far Fields

Defining an Output Plan

Defining an 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 run as a single simulation. You can also specify the data display settings for your EM simulations by using the EM Setup Window.

You can specify a dataset for storing results by using the EM Setup window. You can perform the tasks described in this section.

Selecting an Output Dataset

When you run an EM simulation, the results are stored in a dataset. This dataset is then used by 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 where you want simulation data to be saved:

- 1. Select EM > Simulation Setup to display the EM Setup window.
- 2. Select Output Plan in the left pane of the EM Setup window.
- 3. Click Edit. The Choose Output Names dialog box is displayed.

NOTE You can also select **EM** > **Choose Output Names** to open the Choose Output Names dialog box.

- **4.** By default, the Use cell name option is selected. To type a name for the dataset, remove selection from this option.
- 5. You can also click Browse to view the existing dataset names in the current workspace, as shown in the following figure:

Choose Output Names:3
Dataset ✓ Use cell name
cell_3 Browse
Append text specified in simulation setup
Data Display Image: Object of the second sec
cell_3 Browse
OK Cancel Help

- 6. Select the required dataset and click OK.
- **7.** The Append text specified in simulation setup check box is selected by default.
- 8. Click OK.

Customizing the Dataset Naming Convention

You can customize the naming convention of the dataset generated during a simulation by adding a prefix or suffix to the data name. To generate the dataset with the required naming convention, you can select Appending text to the dataset name. You can select one of the following options for customizing the data name in Append text to the name:

- View name: Select the View name option in the From the layout view section to add text from the layout view.
- Adding simulation type: Select Simulation type include the type of simulator in the dataset name by selecting Simulation Type.
- Adding simulation options: Select Simulation options in the From the simulation setup view section to add text from the simulation setup view.
- Adding user-defined text: You can add a suffix to the dataset name in the This text field.
- In case of an adaptive frequency sweep, two datasets will be the generated: one with the actually simulated samples (<name>.ds) and another one (<name>_a.ds) which is a densly sampled dataset based on the adaptively fitted rational model.

Specifying Data Display Settings

You can select options to open data display after completing the simulation process.

Open data display when simulation completes: Automatically opens the data display window on the completion of a simulation.

You can also insert a template in a new or empty data display by using one of the following options:

- Auto-select based on number of ports: Selects a template automatically based on the number of ports.
- User-defined file: Enables you to specify a template. You can also perform the following steps to select the required template:
- 1. Click Browse to open the Data Display Insert Template window, as shown in the following figure:

= DOS Template Libraries	Search		Search	See
Customized	Name	-	Description	1
Momentum Product	EM_2D_Polar			
Momentum Customized	EM_FarFieldCut			
Momentum_User	FEM_convergence		FEM Convergence Plots	Aglen
	FF_2D_Polar		2D Far Field Cut	Aglen
	S_1port_P		S-Parameters for 1-Port, Multi-Page	Agilen
	S_2port		S-Parameters (Smith and Linear) for 2-Port	Aglen
	S_2port_P		S-Parameters for 2-Port, Multi-Page	Aglen
	S_2port_Spiral		2-port Spiral Characteristics	Aglen
	S_3port_P		S-Parameters for 3-Port, Multi-Page	Aglen
	S_3port_Spiral		3-port Spiral Characteristics	Aglen
	S_4port_P		S-Parameters for 4-Port, Multi-Page	Aglen
	S_Nport		S-Parameters for N-Port	Aglen
	S_Nport_B8SMG		Broad Band Spice Model Generator: compare input and fitt	Aglen
	S_Nport_P		S-Parameters for N-Port, Multi-Page with slider bar	Aglen
	<			>

- 2. By default, the Momentum Product option is selected in the left pane of the Data Display Insert Template window.
- 3. Select the required template.
- 4. Click OK.

Saving Currents and Fields

You can select the following options from the Save fields for list for saving the current simulation settings:

- All generated frequencies: This option enables you to save the field values for all the frequencies that are solved, including all the frequencies adaptively determined in any Adaptive sweep.
- No frequencies: This option does not save the field value for any frequency.
- User specified frequencies: This option enables you to describe all the frequencies explicitly and define a priority in the Frequency Plan. These include all instances of:
 - Fstart and Fstop in 'enabled' Adaptive plans (but none of the adaptively determined frequencies in between)
 - Fstart, Fstop, and all the intermediate frequencies in 'enabled' Linear plans
 - Fstart, Fstop, and all intermediate frequencies in 'enabled' Log plans

- Fstart, Fstop, and all intermediate frequencies in 'enabled' Dec plans
- Fstart from 'enabled' Single plans

NOTE For more information about a frequency plan, see Defining a Frequency Plan.

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 left pane of the EM Setup window.

🔁 Layout	Dataset	
 Partitioning Substrate Ports Frequency plan Output plan Options Resources Model Notes 	Start of the name: amplifier_die_passives Edit Append text to the name From the simulation setup view: View name: femSetup View name: femSetup Simulation type: FEM Simulation options: Default Finally: This text: mySuffix	=
	Name: amplifier_die_passives_FEM Data Display ✓ ✓ Open data display when simulation completes Template (only inserted in a new or empty data display) Auto-select based on number of ports ✓ User specified file S_Nport_P.ddt Save fields for: ✓ All generated frequencies ✓ No frequencies	
	Generate: S-Parameters Vision	late

- **3.** The default dataset name is displayed. To specify a new dataset, click Edit. For more information, see Selecting an Output Dataset.
- 4. Select an option for appending text to the dataset name.
- **5.** Select Open data display when simulation completes, if you want to open the data display window automatically.
- 6. Specify the template to be used for displaying the data in Template. You can also click Browse to select the required template.
- 7. Select the required option in the Save fields for section.

Specifying Simulation Resources

Specifying Simulation Resources

You can run EM simulations on a local or remote host 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:

- Local and Remote Simulation
- Third Party (Distributed) Simulation

Momentum simulations launched on a remote host or load balancing system have the license auto-retry enabled. For more information, see License Retry. The license auto-retry feature is disabled for simulations running on the Local host. If no license is available, a simulation will fail immediately.

Setting up Local and Remote Simulation

From the EM Setup window, you can 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.

You can specify the setup options described in the following sections for local or remote simulations:

Selecting 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 Autocreate subdirectory per job, sub-directories are created automatically.

Setting the Number of Threads Option

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.

The multi-thread calculations of a simulation are time consuming. The workload is then distributed over a number of threads of execution. The hardware of presentday computers can run several threads in parallel, thereby reducing the simulation time. Using more threads, allows more parallelization. However, if you use this concept above a certain amount, it might not be beneficial due to the following reasons:

- The hardware can only run as many threads in parallel as it has logical CPU cores. The number of physical CPU cores times the number of hyper-threads per core. You cannot use more threads than this limit.
- Several computing resources are shared, such as non-cached memory access hardware or the execution units on a hyper-threaded CPU core. Once a shared resource is saturated, adding more threads will not offer further speedups.

The number of threads can be specified in the simulation resources pane. The Auto setting will use as many threads as is deemed appropriate for an otherwise unoccupied machine. The Limit to setting allows to explicitly specify either the maximal number of logical CPU cores to use, or the minimal number of logical CPU cores to remain unused (to reserve them for other software than this simulation).

WARNING You can control the number of threads used by Momentum by setting OMP_NUM_THREADS environment variable. This variable takes precedence over what you set in the EM Setup window. If you limit the number for cores via OMP_NUM_THREADS, and try to specify more cores using the EM Setup window, you will not get extra cores.

Selecting a 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. You can set the following options:

- Queue name: Specify a queue name.
- Auto-start queue after submission: Selecting this option starts the simulation automatically after you have submitted the changes.



Specify a **Simulation host** other than 'Local' if you have multiple queues and you want the license auto-retry enabled.

Setting 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).

Mom RF Layout Substrate Substrate Output plan Output plan Output plan Resources Model/Symbol Notes Notes	Simulation Resources Job manager Start Shutdown Job management: ADS (local/remote) 3rd Party (distributed) Simulation host: Local Refresh Simulation host settings Simulation directory Shared local: k\simulation\EM_Cosimulation_lib\cell_1\ayout\emSetup_MoM Specify: Auto-create subdirectory per job
	Queue simulation Queue name: queue
	Generate: S-Parameters Simulate

- 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 deamon on the submit host. On a Linux submit host, add the necessary environment variables, or source the required configuration script for your LSF/SGE/PBS system into the PVM boot script ~/.eesofpvmprofile. For instance, on LSF systems most configurations require to source the local profile.lsf script for the LSF cluster here. I.e. you need to place a line similar to . _path_to_local_LSF_configuration_dir_/profile. lsf in ~/.eesofpvmprofile. Remember to use a dot (.) to source a ksh(sh) configuration script as .eesofpvmprofile requires Korn Shell syntax to work correctly. For SGE and PBS environments a similar configuration is also required. Contact the local LSF/SGE/PBS administrator for the appropriate configuration can differ depending on the installation.

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.

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 deamon 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: https://www.ibm.com/support/knowledgecenter/en/SSETD4_9.1.2 /lsf_foundations/lsf_introduction_to.html
- PBS Professional: http://www.pbsgridworks.com/Default.aspx
- Sun Gridengine: http://www.oracle.com/technetwork/oem/grid-engine-166852.html

Make sure that the execution environment has the essential variables such as *HPEESOF_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 machines, you can add a ksh script .adsrc with the appropriate ADS environment settings inside your home directory on the cluster machines. The \sim /. *adsrc* file is picked up by the simulation scripts just before a simulation starts. The .

adsrc script is run with two arguments the name of the simulator component (Momentum or fem) and the expected version of that simulator component. This allows the user to select and configure for the appropriate ADS installation on his system.

Third Party (Distributed) Setup Options

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.
- 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 (Momentum Only): 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. See #Using Momentum Turbo.
- 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.
- 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 for the job. Due to name limitations on various management systems the name can only contain upper and lower case a to z, 0-9 and an underscore and may not start with a 0-9.
- Shell command at finish: Allows you to use shell commands.
- Resource String: Allows you to specify a resource and option string that is passed to the submit command without modifications.
- 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.

Hom RF	Simulation Resources
Substrate	Start Shutdown
Gutput plan Output plan Options Resources Model/Symbol Notes	Job management: ADS (local/remote) 3rd Party (distributed) Submit host: Local Refresh Submit queue: Refresh Parallel jobs for frequency sweep: 1 Simulation host settings Simulation directory
	Shared local: imulation\EM_Cosimulation_lib\cell_1\ayout\emSetup_FEM Specify: Auto-create subdirectory per job
	Number of threads: Auto Limit to #Cores-0 Process priority: Normal Job control settings Submit user: Email notification: Tota areas Auto
	Generate: S-Parameters 🗸 Simulate

- 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 Momentum Turbo

Momentum Turbo accelerates a Momentum simulation by solving frequencies in parallel on a compute cluster. This is especially useful for large simulations requiring many frequency samples.

A dedicated Momentum Turbo Element license (W2343) is required in addition to the standard Momentum Element license (W2341). One Momentum Turbo Element license allows that the frequency sweep runs on up to 8 nodes in parallel. Multiple Turbo Elements can be stacked to go above 8.

NOTE The compute nodes in the cluster must share the same OS and have similar performance for proper operation of Momentum Turbo.

- The slowest machine will determine the actual speedup that can be obtained. The master process that distributes frequencies to the child processes waits until they all return before distributing a next batch of frequencies.
- A mix of Linux and Windows in the cluster may introduce numerical noise in the solution (e.g. due to a slightly different mesh) that makes it hard to impossible for the adaptive frequency sampling to converge.

Amdahl's Law

The multi-processing performance can be measured using Amdahl's law

$$S(N) = \frac{1}{(1-p) + \frac{p}{N}}$$

Whereby S is the maximum speedup, p is proportion of the program that is parallelized, and N is the number of processors allocated.

The following graph shows the speedup versus parallelism for respectively 4, 8 and 16 nodes.



The law illustrates that, in order to obtain a decent speedup, the parallelism needs to be high enough.

For a Momentum simulation, this implies that:

- A sufficient number of frequency samples is requested or required. The higher the number, the higher the parallelism and consequently the speedup.
- The speedup of a microwave mode simulation will be higher than an RF mode simulation on the same design. An RF mode simulation has a higher fixed cost. Every node needs to load the quasi-static matrix, independent to the number of frequencies points that it will actually solve.
- An adaptive frequency sweep will most likely require more frequency samples to converge. The 'next' samples are chosen in bulk, one for each parallel process, which doesn't guarantee they are chosen at the optimal location compared to a sequential simulation.

Example

An example (see Examples > Momentum > RF > PCBlines) was used to characterize the speedup on a Sun Grid Engine cluster that consists of 1 submit host and 8 compute nodes.

The layout of 'PCBlines16', shown below, was simulated by Momentum.

Following simulation settings were changed so that a large enough simulation time is needed per frequency point.

- Mesh frequency = 4 GHz
- Cells per wavelength = 50
- Edge mesh = on
- Mesh reduction = off

8 different simulations are compared:

- microwave and RF mode simulation,
- using an adaptive (Fstart = 0 GHz, Fstop = 4 GHz, Npts = 200 (max)) and a linear frequency sweep (Fstart = 0 GHz, Fstop = 4 GHz, Npts = 112),
- performing the frequency sweep on 1 node (= standard Momentum simulation) and on 8 nodes in parallel (= Momentum Turbo simulation).

Simulator	Sweep Type	# Nodes	# Freqs	Speedup
Momentum UW	Adaptive	1	108	
Momentum UW	Adaptive	8	128	4.8
Momentum UW	Linear	1	112	
Momentum UW	Linear	8	112	6.7
Momentum RF	Adaptive	1	101	
Momentum RF	Adaptive	8	128	3.6
Momentum RF	Linear	1	112	
Momentum RF	Linear	8	112	4.6

This example illustrates that:

- The speedup is larger in microwave mode versus RF mode
- The speedup of a linear (or log) frequency sweep is higher than an adaptive sweep.
- That an adaptive sweep typically requires a couple of extra passes to converge. For example, a standard adaptive sweep in microwave mode requires 108 samples to converge. Ideally, this would require each parallel

node to compute 108/8=int(13.5)=14 frequencies. In the actual parallel adaptive sweep simulation, 16 passes or 16*8=128 samples were required to reach convergence. This is related to the fact that a bulk of 'next' samples are chosen at once in a parallel simulation.

Using the Job Manager

The EM Setup window provides the capability to start and stop the Job Manager. For for more information, refer Using the Job Manager. The Job Manager is automatically started in mimimized 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:

	Subar	Hat	Questi	Project Durige
4001210109008	Rest:	Incalized		. As Mitchieg/MW/department/Mitchieg/Read/Likese
ALC: NO	In one part	lantheri		 Album (Dentity) MyWashapare saladilyi darey biaseli Talapa

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. 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 an EM Model

Generating an EM Model

An EM Model is used by a circuit simulator to model a component by using the Sparameters 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 circuit simulation performance. For more information about how to use an EM model, see Creating and Using an EM Model View.

You can generate an EM model from the EM Setup window.

How Do I Generate an EM Model

To create an EM model:

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select Model in the left pane of the EM Setup window. The EM Model options are displayed:



3. Accept the default *emModel*name or type a new 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 Reduce to decrease size, 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 Ensure the EM Model exists when the simulation is launched to create an EM model at the start of the simulation process if it does not exist yet.
- 6. Select Update the existing EM Model when the simulation finishes to autoupdate the existing EM Model to use the results of this simulation. If there is no EM Model it will not be created.
- 7. To generate an EM model immediately, click Auto-create Now.

NOTE If the **Auto-create Now** button is labeled **Update Now**, an EM Model already exists for this cell.

The following figure displays an EM model:

🔣 Coupled_Stubs_tune_lib:CoupledStubs:emModel (EM M 🔳 🗖 🔀
File Edit Tools Help
1 🚰 🔚 🍠 🔍 🖪 🗇 🌲
Simulation Setup Database Interpolation Options
Summary
Simulator! Momentum simulation in microwave mode Layout: Workspace: C: Users\default\sample\multi_band_wrk\model\Coupled_Stubs_tune_wrk Library: Coupled_Stubs_une_lib Cell: CoupledStubs View: layout Substrate: Substrate: Substrate: CoupledStubs Ports: 2 ports defined Prequency plan: Adaptive from 0 GHz to 30 GHz (max:200) Dutput plan: Dataset: CoupledStubs Template: S_Nport_Slider.ddt Simulation options: ConvertedFromDSN Converted options from the DSN version of ADS Simulation resources: Queued simulation on host: Local:queue Pt Model/Symbol: Auto-create EM Model: no EM Model: emModel Auto-create symbol: no No symbol name specified
Edit

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 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:

EM Model
Start of the view name: emModel
Append text to the name
From the layout view:
From the simulation setup view:
Simulation type: MomRF
Finally: This text:

- 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.

Using EM Setup Window Tools

Using EM Setup Window Tools

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

Saving 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.

™ Save D	esign As 🛛 🔀		
File			
Library:	H_Antenna_lib		
Cell:	H_Antenna_LC_em_em_em_em		
View:	emSetup		
Type: EM Setup View			
ОК	Cancel Help		

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.

🍽 Save a Copy As 🛛 🔀
File
Library: H_Antenna_lib
Cell: H_Antenna_LC_em_em_em
View: emSetup
Type: EM Setup View
OK Cancel Help

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 from an EM Setup

To import from another EM Setup in the current EM Setup window:

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

Import from		
Cellview		Browse
Filter		
Sections to include	n de la compañía de l	(R
V Setup	Mom options	Fem options
Substrate	Frequency plan	Output plan
Cosimulation	Model	Votes
Partitioning	Compute resource	ces
The imported setup data w	ill overwrite the existing	g setup data.
	Import Cance	Help

- 2. Click Browse to select the cellview to import from. For more information, see #Opening an EM Setup View.
- **3.** Select the required sections in the Filter. You can also select/deselect all the sections by clicking the checkbox Sections to include.
- 4. Click Import.

Opening an EM Setup View

To open an EM setup view, perform the following the steps:

- 1. Start ADS and open an existing workspace.
- 2. From the ADS Main window, choose File > Open > EM Setup View to open the Specify EM Cell View dialog box.
- 3. Select EM Setup View from the Type drop-down list.
| EM Specify EM CellView | | |
|---|--------------|--------------------|
| Type: EM Setup View 💌 | | Show ADS libraries |
| Library: | Cell: | View: |
| Double_Patch_lib | Double_Patch | emSetup |
| Double_Patch_lib
EMProDefault3D_DesignKit
IMPORT_ADFI | Double_Patch | emSetup |
| | ОК | Cancel Help |

- 4. If you want to open a built-in ADS design (read-only), check the Show ADS Libraries check box to display the list of all libraries under Library.
- 5. Under Library, select the Library name where the view exists.
- 6. Under Cell, select the cell name.
- 7. Under View, select the required EM setup view name.
- 8. Click OK to open the selected EM setup view.

Opening the 3D Preview window

Choose Tools > 3D EM Preview or click the 3D EM Preview icon (

Opening the Layout window

Choose Tools > Open Layout Window or click the Open Layout Window icon (**1**) 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 (EM) 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.

Creating an SnP Schematic

You can create a new SnP schematic view by using the EM Setup window. This schematic can be used as a subcircuit for further circuit simulations.

To create an SnP schematic from the EM Setup window:

- 1. Open the EM Setup window.
- 2. Select Tools > Create SnP Schematic. A new schematic view is created.
- **3.** A message box is displayed that asks if you want FileName and FileType to be cell parameters.
 - If you select FileName and FileType as cell parameters, the SnP data item will contain FileName and FileType as file parameters. The default value for cell parameters will be Dataset for FileType and the dataset name in emSetup for FileName.
 - If you do not select FileName and FileType as cell parameters, the file parameters for SnP will be dataset for FileType and the dataset name in emSetup for FileName.
- 4. A new SnP schematic view is created, as shown in the following figure:



Creating an SnP Schematic

Creating an SnP Schematic

You can create a new SnP schematic view by using the EM Setup window. This schematic can be used as a subcircuit for further circuit simulations.

To create an SnP schematic from the EM Setup window:

- 1. Open the EM Setup window.
- 2. Select Tools > Create SnP Schematic. A new schematic view is created.
- **3.** A message box is displayed that asks if you want FileName and FileType to be cell parameters.
 - If you select FileName and FileType as cell parameters, the SnP data item will contain FileName and FileType as file parameters. The default value for cell parameters will be Dataset for FileType and the dataset name in emSetup for FileName.

- If you do not select FileName and FileType as cell parameters, the file parameters for SnP will be dataset for FileType and the dataset name in emSetup for FileName.
- 4. A new SnP schematic view is created, as shown in the following figure:



Using an EM Setup Template

While creating an EM Setup view, the data fields of the view are initialized with hard-coded factory default settings. To override these settings, an EM setup template with user-defined default settings can be used. You can use the settings for Simulator, Substrate, Frequency Plan, Output Plan, Momentum Options, FEM Options, Compute Resources, and EM Model found in the template to initialize a new EM Setup view.

Managing EM Setup Templates

Create EM Setup Template

Before creating an EM Setup Template prepare an EM Setup to contain all the values that have to be stored in the template.

Once the values have been set, from the menu bar select File > Create Template... . The below dialog will open.

MyLibrary1_lib:cell_1:emSetup (Create EM Template)
Location
Template name: TestTemplate .emSetupT
O Library MyLibrary1_lib ▼
ADS configuration directory Iocal user site
Creating: MyLibrary1_lib:TestTemplate.emSetupT
Filter
 Sections to include
Setup I Mom options Fem options
✓ Substrate ✓ Frequency plan ✓ Output plan
Cosimulation 📝 Model
Partitioning Compute resources
Notes
This document contains no notes
Create Cancel Help

Enter a meaningful name for the template, as this name will be presented when creating a new EM Setup.

Choose the directory where the template will be stored. It can either be stored with a library or in an ADS configuration directory.

The ADS configuration directory mapping is:

- local \rightarrow current working directory
- user → \$HOME/hpeesof/em/setupTemplates (@EMSETUPT_USER)
- site → \$ADS_CUSTOM_DIR/em/setupTemplates (@EMSETUPT_SITE)

The templates' absolute filepath will be displayed in a pop-up box when hovering over the location display after the label Creating:

You can filter out the section(s) you do not want to be part of the template, and you can add/modify some notes to ease template management (reload/delete)

Once the data is entered, Click the Create button.

NOTE The template can only be used when the storage directory is part of the EM Setup Template search path defined in emSetupTemplate.loc

Load EM Setup Template

To reset your current EM Setup to use the values in an EM Setup Template, from the File Menu in the EM Setup Dialog choose Load Template... . The following dialog will open:

MyLibrary1_lib:cell_1:emSetup (Load EM Template)
Available templates
MyLibrary1_lib:TestTemplate
Notes
This document contains no notes
Load Cancel Help

Select the desired entry, optionally inspect the notes, and press Load.

NOTE The values in the current EM Setup for which no values are defined in the EM Setup Template will keep their value.

Delete EM Setup Template

To delete an EM Setup Template, from the File Menu in the EM Setup Dialog, choose Delete Template... . The following dialog will open:

X MyLibrary1_lib:cell_1:emSetup (Delete EM Template)
Existing templates
MyLibrary1_lib:TestTemplate
Notes
This document contains no notes
Delete Cancel Help

Select the entry to be deleted and click Delete.

Templates can only be deleted one after the other.

Finding EM Setup Templates

The search for EM Setup Templates is a 2-step process:

1. Finding emSetupTemplate.loc, the file that defines the EM Setup Template search path.

2. Finding the available EM Setup Templates

Finding emSetupTemplate.loc

The first emSetupTemplate.loc file found browsing the following, ordered directory locations will be used and stop the search:

1		The current working directory
2	@ADS_WORKSPACE	The current workspace
3	\$ADS_EMTEMPLATELOC_FILEPATH	Environment variable with the full PATH to the file
4	\$HOME/hpeesof	The hpeesof directory in your HOME directory
5	\$ADS_CUSTOM_DIR	If \$ADS_CUSTOM_DIR not set use \$HPEESOF_DIR/custom instead
6	\$HPEESOF_DIR/config	The config directory shipping with ADS

The config directory shipped with ADS contains a predefined emSetupTemplate.loc file, so an emSetupTemplate.loc file will always be found.

The search will always look for an emSetupTemplate.loc file in the designated directory, except for \$ADS_EMTEMPLATELOC_FILEPATH that defines the full filePath, i.e. the directory and the file name.

NOTE To find out which emSetupTemplate.loc is used, open the Delete EM Template dialog (see below) and hover with your mouse just below the Existing templates text, an info box will pop-up with the full path to the emSetupTemplate.loc file.

Finding the available EM Setup Templates

The available EM Setup Templates is the list of ALL EM Setup Templates found in the search locations defined in the emSetupTemplate.loc file. I.e. unlike for the search for the emSetupTemplate.loc file the search is not stopped after finding the first template.

EM Setup Templates are files with a .emSetupT suffix (in case of legacy libraries . emsetup is also accepted). They can be created using the methods described below or for the more expert users using an xml editor.

The available search locations that can be used in the emSetupTemplate.loc file are

	search the current working
directory for templates	
@LIBRARY	search all the design
libraries for templates	
<pre>@LIBRARY_myLibrary</pre>	search the myLibrary design
library for templates	

```
@ADS_WORKSPACE search current user workspace
for templates (in most cases the current working
directory)
$ADS_EMSETUPT_SEARCHDIR search the directory defined
by the environment variable that may be set during
startup for templates
@EMSETUPT USER
                       search for user templates
stored in $HOME/hpeesof/em/setupTemplates
@EMSETUPT_SITE
                       search for site templates
stored in $ADS_CUSTOM_DIR/em/setupTemplates
                           if ADS_CUSTOM_DIR is not
;
set search in $HPEESOF_DIR/custom/em/setupTemplates
@EMSETUPT_SUPPLIED search for shipped templates
stored in $HPEESOF_DIR/em/setupTemplates
$ANY_ENV_VARIABLE search the directory defined
by the environment variable that may be set during
startup for templates
"/a/fixed/path/to/a/directory"
                                       search
directory for templates
"/a/fixed/path/to/a/template.emsetupT" add this
template to the list if found.
```

The format of the file is

- one entry per line
- everything after blank or tab is comment (quote entries if containing blanks)
- ; or '#' or '//' at the start of a line is a comment.
- \$envvar, \${envvar} are expanded as needed. If an environment variable is not defined, then that entry is ignored.

No errors are generated during the parsing of the file.

A customizable example can be found in \$HPEESOF_DIR/config/emSetupTemplate. loc

Using the EM Setup Template

A list of available EM setup Templates is constructed using the algorithm described above.

- If the list is empty, the factory default settings are used
- If the list holds one entry, the settings from that template will be used to initialize the new EM Setup view.
- If the list holds more entries, you can select from the list the one previously used will be preselected -

<mark>𝗚</mark> New	EM Setup View
EM	New EM Setup View Create a new EM setup view to contain your EM simulation settings.
Library:	portRefactoring_lib
Cell:	cell_5
View:	emSetup
Use tem ./altern ./altern	plate ateSubstrate ateSubstrate
portRef	actoring_lib:defaults
Related What	I topics at is a cell?

NOTE Give your EM Templates meaningful names, it will be easier to determine which one to choose.

Deprecated

Installing an EM Setup Template in a Library using scripting (for Experts)

To add an EM Setup template:

- 1. Create a (dummy) EM Setup cellview in the library, e.g. libname>: <cellName>:<emSetupViewName>, and initialize its data.
- 2. Optionally using an xml editor, edit the file emStateFile.xml in the emSetup cellview directory to remove the entries where you want to always use the factory defaults.
- **3.** From the ADS Main window, select Tools > Command Line to open the ADS Command Line Editor.
- 4. Type the following AEL command:

```
deemif_installSetupViewAsTemplateInLibrary("<libName
>", "<cellName>", "<emSetupViewName>", "<emSetupTemp
lateName>")
```

The template gets installed in the library and a file with the name <emSetupTemplateName>.emSetupT is saved in the library. After installation of the template, you can delete the EM Setup cellview.

Removing an EM Setup Template from a Library

To remove an EM Setup template from a library:

1. From the ADS Main window, select Tools > Command Line to open the ADS Command Line Editor.

2. Type the following AEL command:

```
dex_em_removeEmSetupTemplateFromLibrary("<libName>",
"<emSetupTemplateName>")
```

Managing EM Simulations

Managing EM Simulations

- Running EM Simulations
- Using the Job Manager
- Viewing Simulation Summary
- Running a Momentum simulation from Command Line

Running EM Simulations

Running EM Simulations

Before running a simulation, the following criteria must be met:

- A substrate definition must be specified for the circuit.
- The layout must contain at least one port.
- A simulation frequency plan must be specified.

To run a simulation:

- 1. Create an EM setup.
- 2. Select an option from the Generate drop-down list in the EM Setup window.

Generate:	Views 🗸	Go
	Views	1
	EM Simulation input files	
_	Preprocessed geometry	_
	Substrate	
	Mesh	

3. Click the Go button. Alternatively, press F7, which will start the action specified in the EM Setup currently associated with the layout.

When you start a simulation, a new job is created in the Job manager. The Job Manager window provides a user interface to show the job status and messages, and allows you to cancel running jobs. For non-queued simulations, by default, a window with job status and messages will be opened. In the case of a queued simulation, double-click the job to open the detailed log window. 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 stop an EM simulation, choose EM > Stop and Release Simulator. Alternatively, the simulation can be stopped using Job > Stop/Cancel Job in the Job Manager.

As a simulation progresses or after the simulation finished, 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. See Viewing Simulation Summary

Generating EM Cosimulation Views

To generate EM Cosimulation Views:

- 1. Select Views from the Generate drop-down list in the EM Setup window.
- 2. Click Go.

Generating EM Simulation Input Files

You can use the simulation input file for running command line simulations. To generate the input files:

- 1. Select EM Simulation Input Files from the Generate drop-down list in the EM Setup window.
- 2. Click Go.

Generating Substrate (Momentum only)

The substrate output files contain the Green functions used by Momentum. These are precomputed and stored in a substrate database so that they can be reused for a subsequent simulation.

To generate substrate output files:

- 1. Select Substrate from the Generate drop-down list in the EM Setup window.
- 2. Click Go.

NOTE This mode always recalculates the substrate's Green functions. Existing local and/or remote substrates are removed from the database prior to the generation.

Viewing Substrate Generation Status

After the substrate calculations are started, any messages regarding the calculations will appear in the Job Manager status window. Messages usually refer to any errors found, and indicate when the calculations are complete. You can open the Job Manager window from the Layout menu bar choose Window > Simulation Status.

Viewing the Substrate Generation Summary

If the substrate generation is successful, you can view the job status summary which informs about the time to compute and the resources required. To view the summary, choose EM > Show Most Recent > Substrate Generation Summary. This displays the Substrate Summary window.

Substrate Database Locations

By default, Momentum searches for substrate output files containing the Green functions in 3 locations. The locations are defined by following configuration variables in a momentum_ads.cfg configuration file.

Variable Name	Default Value	File Location
SUPL_GF_DATABASE	^/	\$HPEESOF_DIR/Momentum/ <version>/substrates</version>
SITE_GF_DATABASE	~/	\$HOME/substrates ()
LOCL_GF_DATABASE	./	<workspacedirectory>/substrates</workspacedirectory>

If no output files are found that can be reused, new substrate output files will be computed and stored in the SITE_GF_DATABASE (when writable) or LOCL_GF_DATABASE location. By default, substrate files will be written to \$HOME /substrates.

You can change the default search location by adding a configuration variable into either \$HPEESOF_DIR/custom/config/momentum_ads.cfg, \$HOME/hpeesof/config /momentum_ads.cfg, or <workspaceDirectory>/momentum_ads.cfg (create the directory path and configuration file if it does not yet exist).

Some examples are provided here. A 'substrates' subdirectory will be created automatically at the specified location! Environment variables can be used as well (Note: the unix notation with '\$' and '/' are used).

To store new substrate output files

1. Under <workspaceDirectory>/substrates, set the SITE_GF_DATABASE variable to a non-existing directory

SITE_GF_DATABASE=/non_existing_dir

2. Under the C:/tmp/substratesdirectory", set

SITE_GF_DATABASE=C:/tmp

3. Under a design kit, assuming that an environment variable \$my_designkit_directorypoints to the location of a design kit, set

SITE_GF_DATABASE = \$my_designkit_directory/circuit

will store substrate output files under \$my_designkit_directory /circuit/substrates.

Embedding the substrate output files in the workspace

Computing the substrate output files can be time consuming. To avoid recomputing them when a workspace is shared with another user, you can move the substrate output files into the substrates directory of a workspace.

To identify which substrate output files to move, browse to the <workspaceDirectory>/simulation/<libName>/<cellName>/<layoutName> /<emSetupName>_MoM directory and open the proj.qry file. The line at the bottom will be like

```
Corresponding potentialname : C:\Users\default\\substrat
es\0214.sdb
```

Browse to this substrates directory and move the corresponding .sdb and .sci file to <workspaceDirectory>/substrates. No need to update the .ndx file there, this will be done automatically. Your next simulation will reuse the substrate output files from the workspace.

Generating a Mesh

To generate mesh at a specified frequency:

- 1. Select Mesh from the Generate drop-down list in the EM Setup window.
- 2. Click Go.

```
Status Messages

...extracting layout

...reducing mesh

Mesh processing finished

Meshing finished on: Mon Mar 05 14:42:38 2012

Job finished on: Mon Mar 5 14:42:38 2012

Exit status: Done
```

Using the Job Manager

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 window 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.

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 in the following figure:

🖉 🖬 Mom RF 🛛 🚺	Simulation Resources			
🔁 Layout	Job manager			
ڬ Substrate 🛛 🕄				
Dorts	Start Shutdown			
Frequency plan Output plan Options Resources Model/Symbol Notes	Job management: ADS (local/remote) Tremote Simulation host: Local Refresh Simulation host settings Simulation directory Shared local: on\EMPro_ADS_handson_lib\board\Jayout\emSetup_MoM Specify: Auto-create subdirectory per job			
	Process priority: Normal Queue simulation Queue name: queue Auto-start queue after submission			
	Generate: S-Parameters Simulate			

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

🍓 EEsof Job Manager				
File View Job Q	ueue Settings Help)		
Job Summary				
Tag	Status	Host	Queue	Project::Design
M101210103653	Done	localhost		s\build\Desktop\MyWorkspace_wrk::MyLibrary_lib:cell_1:layout
M101210103759	Running	localhost		s\build\Desktop\MyWorkspace_wrk::MyLibrary_lib:cell_1:layout
Right click the job for I	more options			h.

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.

One exception: jobs that are actively running on a 3rd party cluster survive closing and reopening the Job Manager. To access these jobs, reopen the Job Manager from an EM Setup and the jobs will reappear in the main Job Manager window.

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 the 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 Distributedmode, 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 Job 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.
- Skipping simulation steps: Some iterative simulation steps can be interrupted, forcing the simulation to proceed with the next step. The generic name of this menu item is Job> Interrupt simulation. When enabled, this changes to a more specific name indicating precisely what will be interrupted. You can interrupt the adaptive mesh refinement or solve phase (frequency sweep) of an FEM simulation process before convergence. The simulation continues with the current mesh or the already calculated frequency points. Intermediate data of interrupted simulations can be reused in consequent simulations. Select Job> Skip mesh refinement during the mesh refinement step of an FEM simulation. Select Job> Skip frequency sweep to stop frequency sweep, as shown in the following figure:



- Querying Status: You can query a status update for jobs submitted to a thirdparty 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. You can configure an interval for automatic status polling in the general Job Manager preferences (on by default using 60 second intervals). Automatic polling is suspended if the submit host is dropped from the PVM cluster. A manual status query triggers reconnecting to the submit host, after which automatic polling will resume.
- 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. For more information, see Specifying Simulation Resources. 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.

indow Settings Help			
Active hosts Remove selected host(s) from cl	uster		
Hostname	Architecture	Speed	
1 abcx	linux_x86_64	1000	
2 ablmw7b64	WIN32_64	1000	

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 $s_P = value$ eesofpvm option and is currently used for display purposes only. See #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 EEsofPVM 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
 EESOFPVMHOSTFILE (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 EESOFPVMHOSTFILE with an additional variable EESOFPVMHOSTFILE:390=/nonexistent. The number 390 is the internal Job Manager version number as shown when choosing Help > About EEsof 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.

These preferences are saved in <*home directory*>/hpeesof/config/simsys. cfg, which can be distributed towards other users if they all use the same remote simulation hosts.

See	2 ×
General Host configuration	
Allow jobs to pop up their log	
Automatically query 3rd party cluster status every 60 seconds	
System launch/shutdown	
Start the eesofpvm backend when ADS starts	
Close EEsof Job Manager when ADS exits	
Shut down the eesofpvm backend when both ADS and EEsof Job Manager have exited	
Shut down the eesofpvm backend when EEsof Job Manager exits	
Use separate eesofpvm backends for different combinations of user/host/\$DISPLAY/\$HPEESOF_DIR	
Reset	OK Cancel Apply Help

EEsof Job Manager Preferences				? ×
General Host configuration				
Default remote ADS installation dire	ctory			
Use the local \$HPEESOF_DIR va	riable: C:\Keysight\ADS 2014_0	1		
As defined in the eesofpvm clus	ter configuration file C:\Users\mo	dewilde\hpeesof\config\pvmhost_cfg.bd		
O User defined:				
Default working directory:				
Default additional eesofrym ontions:				
Default memory limits	unlimited			
Hort cracific quarridas	uniimiteu 💌			
Host specific overfices				
Add host Delete host(s)				
Hostname	ADS installation directory	Working directory	Additional eesofpvm options	Memory limit
1 linuxbox.company.com	/opt/ADS 2014_01	/var/tmp/\$USER/jobs/\${SESSION:local}/\${TAG:local}:force	lo=unixuser	۵
2 windowsbox.company.com	C:\Keysight\ADS 2014_01	%USERPROFILE%\Jobs\%SESSION:local%\%TAG:local%:force	lu=winuser	8192MB
Reset		01	Cancel Apply	Help

General Options

- Allow jobs to pop up their log: When selected, a new window pops up for each new job. Otherwise, new jobs will only be highlighted in the Job Summary list. It is selected by default.
- Automatically query 3rd party cluster status every xxx seconds: When selected, the status of jobs running on a 3rd party cluster will be automatically polled using fixed time intervals. Automatic polling is suspended if the submit host is dropped from the PVM cluster. A manual status query triggers reconnecting to the submit host, after which automatic polling will resume. This option is selected by default using an interval of 60 seconds.

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 EEsof 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 EEsof Job Manager have exited: When selected, the PVM infrastructure will be stopped when both ADS and EEsof Job Manager have exited. It is selected by default.
- Shut down the eesofpvm backend when EEsof Job Manager exits: When selected, the PVM infrastructure will be stopped when the EEsof 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 EESOFPVMHOSTFILE 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 EEsof Job Manager when ADS exits	Selected	Not selected
Shut down the eesofpvm backend when both ADS and EEsof Job Manager have exited	Selected	Not selected
Shut down the eesofpvm backend when EEsof Job Manager exits	Not selected	partially selected = prompt

Preference name	Normal default (recommended)	Legacy default
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 window allows you to configure jobs to run on a remote host or a distributed cluster.

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). An SSH service is available on all contemporary Unix-like systems. Setting up SSH on Windows is described. 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*:

He	st	specific overrides				
		Add host Delete host(s)				
		Hostname	ADS installation directory	Working directory	Additional eesofpvm options	Memory limit
	1	linuxbox.company.com	/opt/ADS2012_08	/var/tmp/\$USER/jobs/\${SESSION:local}/\${TAG:local}:force	lo=unixuser	-
	2	windowsbox.company.com	C:\Keysight\ADS2012_08	%USERPROFILE%\Jobs\%SESSION:local%\%TAG:local%:force	lo=winuser	8192MB

Specify the following fields:

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.

- 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 EEsof 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 specified in the EM Setup window. 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, AR 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 eesofpvmd> specifies an alternate path to the eesofpvmd executable or script.
 For further options, consult the PVM documentation.
- Memory limit: This field specifies a hard limit for the memory available to individual job processes. It can be used to automatically stop jobs with an out of memory error before they use so much memory that the computer enters an unresponsive trashing state from which it is hard to recover. It does not allow you to run a simulation with less memory. This configuration option is applicable to local and remote processes, but not to processes running on 3rd party clusters. You can either specify unlimited, or an absolute amount of memory (*xxx*MB), or a negative offset from the total amount of RAM (RAM-*xxx*MB).

If the entries for *ADS installation directory*, *Working directory* or *Memory limit* 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 Bourne 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 communicate over dynamically allocated TCP and UDP ports. On each host involved you can limit the port range for inbound connections through the environment variable

EESOFPVM_PORTRANGE=<*from>-*<*to>* (e.g. using an export directive in the ~/.eesofpvmprofile file). Adjust the firewall rules to allow connections from/to these ports.

Setting up an SSH server on Windows

Since Windows 10

A SSH server is automatically deployed when the machine is configured into developer mode.

Use Start > Settings > Update & security > For developers and click the Developer mode radio button.

Before Windows 10

The software package Copssh can be used to deploy a SSH server on a Windows host. A free 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).

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++.

6. At the end of the .bashrcfile, 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.

- 7. Save the .bashrc file and exit the editor.
- 8. Log on as the user to be activated and open a command prompt (Start > Run > cmd.exe)
- 9. Execute mkdir "%USERPROFILE%\PVM" in the command prompt.
- 10. Close the command prompt.
- 11. 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_eesofpvmd.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 specifying the password, yet normal password-based operation 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)
 - a. On Windows, open a command prompt inside ADS (Tools > Command Line... > invoke_process("cmd",list(),"",FALSE, TRUE,"");) and type cd <ADS installation directory>\tools\bin <return>. On Unix, open a terminal.
 - **b.** Type ssh-keygen -t dsa *<return>*. Just hit the return key when prompted for the file location and the passphrase.

- c. 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 Tools > Command Line... and run invoke_process("cmd", list(), "", FALSE, TRUE, "");, 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:

Preference name	Variable name in simsys.cfg	Acceptable values
Allow jobs to pop up their log	JOB_LOG_POPUP	TRUE OF FALSE
Automatically query 3rd party cluster status	AUTOQUERY_CLUSTER	TRUE OF FALSE

Preference name	Variable name in simsys.cfg	Acceptable values
Interval between queries in seconds	AUTOQUERY_CLUSTER_INTERVAL	integer
Start the eesofpvm backend when ADS starts	ADS_STARTS_EESOFPVM	TRUE OF FALSE
Close EEsof Job Manager when ADS exits	ADS_STOPS_JOBMANAGER	TRUE OF FALSE
Shut down the eesofpvm backend when both ADS and EEsof Job Manager have exited	STOP_ORPHAN_EESOFPVM	TRUE OF FALSE
Shut down the eesofpvm backend when EEsof 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 OF FALSE
Use the local \$HPEESOF_DIR variable	DEFAULT_HPEESOF_DIR	<local> (include the angular brackets)</local>
As defined in the eesofpvm cluster configuration file	DEFAULT_HPEESOF_DIR	empty
User defined:	DEFAULT_HPEESOF_DIR	string
Default working directory:	DEFAULT_WORKING_DIRECTORY	string
Default additional eesofpvm options:	DEFAULT_EESOFPVM_OPTIONS	string
Default memory limit:	DEFAULT_MEMORY_LIMIT	unlimited, XXXMB Or RAM-XXXMB
Host specific overrides (one entry per override)	HOST_CONFIG_ <unimportant unique<br="">suffix></unimportant>	<hostname> <ads installation directory> <working directory=""> <additional eesofpvm options> <memory limit=""></memory></additional </working></ads </hostname>

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. For instance in LSF environments a user very often needs to source the local profile.lsf script here to ensure that the PVM environment can find the LSF commands.

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

The optional environment variable **EESOFPVMNETSOCKPORT** indicates the socket port number which the PVM daemon uses to connect to other cluster clients. If it set PVM will use the indicated port number to set up the cluster. If it is not set it will search for an available valid port number.

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.

Viewing Simulation Summary

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. The mesh view is selected in the circuit, as shown in the following figure:



NOTE You cannot view FEM mesh.

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 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 have to compute the mesh again.

NOTE You cannot delete the mesh view of an FEM simulator.

Viewing S-param Simulation Summary

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

ы	EM Simulation Summary:1		
			^
		MOMENTUM STATISTICS (NW mode)	
	Moment	um MomEngine 9.0.006 (*) 370.502 Jan 1 2011	
		Wed Jan 05 09:32:36 2011	
	MESH		
	Frequency	: 20 GHz	
	Pectangular cells	. 500	
	Quadrangular cells	: 16	
	Triangular cells	: 2	
	Via cells	: 0	
	Edge currents	. 989	
	Eage Carrenos	. 505	
	SOLVER	USER TIME ELAPSED TIME MEMORY	
		(n:m:s) (n:m:s) (nb)	
	SIMULATION SET-UP		
	Initialization		~
	<u>N</u>		
_			
	ОК	Refresh Print	

In the EM Simulation Summary window, you can view information about the matrix size after mesh reduction and the statistics for quasi-static calculations. The user time and elapsed time values are listed as separate entries in the Summary window. The elapsed time represents the overall time taken for completing the simulation, while user time only presents the time taken by the CPU to complete the calculations. Therefore, if the CPU exclusively works only on the simulation, the user time and elapsed time values are 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

For an FEM simulation, you can view information about the frequency, meshing time, and simulation time. The following figure displays the S-param summary for an FEM simulation:

в	EM Simulat	ion Summary:3						
								^
	Layout s	simplification	n finished o	. Tue Oet 26	14.41.99 201	^		
	Starting	the FFM simi	lation		11.11.00 201	·		
	FEM engi	ne	: 23	0.100 2010-10-	21			
	Machine		: SG	H002S6LN				
	Date		: Tu	e Oct 26 14:41	:46 2010			
	Design		: C:	\users\default	ADS2010\FEM	exam	ple\on26thOc	t\LOWPA
	INITIAL	MESH						
	nbPoints	: 94						
	nbTetrah	iedra : 287						
	REFINING	3						-
			l i i i i i i i i i i i i i i i i i i i	MESHING		1		
	level	freq(GHz)	nbTetr	Elapsed time	CPU time		nbUnknowns	mem (GB
		20.000			E	!	1014	0.00
	2	20.000	∠0/ £10	00:00:00.9	00:00:00.5	1.1	4104	0.02
	3	20.000	1033	00:00:00.7	00:00:00.2	1.1	7040	0.02
	4	20.000	1376	00:00:00.8	00:00:00.2	1.1	9292	0.02
	5	20.000	1785	00:00:00.5	00:00:00.3	1	11984	0.02
	6	20.000	2232	00:00:00.8	00:00:00.4	- i -	14830	0.03
	7	20.000	3259	00:00:01.1	00:00:00.6	- i -	21606	0.03
	8	20.000	3944	00:00:01.1	00:00:00.7	i.	26048	0.04
	9	20.000	4912	00:00:01.3	00:00:00.8	1	32288	0.04
	10	20.000	6338	00:00:01.6	00:00:01.0	1	41562	0.05
	11	20.000	8857	00:00:02.4	00:00:01.5	_	57942	0.06
		19						
—								
	l	OK		Refresh			Print	

Viewing Mesh Generation Summary

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

Mesh Summary:1	
-	MOMENTUH STATISTICS (MW mode)
Nomen	um MomEngine 9.0.006 (*) 370.502 Jan 1 2011 Wed Jan 05 09:32:11 2011
MESH	
STATISTICS	
Quadrilateral cell	3 : 516
Triangular cells	: 2
Via cells	: 0
Edge currents	: 989
RESOURCES	
Process size :	3.55 MB
<	>
ОК	Refresh

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. The Layout Processing Report window is displayed. 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:
Substrate Summary:1
······
MOMENTUH STATISTICS (MW mode)
Nomentum MomEngine 9.0.006 (*) 370.502 Jan 1 2011 Wed Jan 05 09:32:09 2011
SUBSTRATE GENERATION
Process size : 0.21 MB
User Time : Oh Om Os
Elapsed Time : Oh Om 2s
OK Refresh Print

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

NOTE The FEM simulator does not provide any substrate summary information.

Running a Momentum simulation from Command Line

Running a Momentum simulation from Command Line

You can run the Momentum simulator from Command Line providing the necessary input files.

CAUTION The files, file formats and process described below can change from release to release.

- 1. Create the layout and define the associated EM simulation setup in an ADS workspace.
- From the EM Setup window, select Generate: Simulation input files and hit the Go button to write the required input files. Following files will be written to the <workspace>/simulation/<library>/<cell> /<layout>/<emSetup>_<simulator> directory:

Input File	Content
proj.prt and proj. pin	S-Parameter port and associated pin information
proj_a	original geometries before layout pre-processing
proj.ltd	substrate information
proj.opt	simulation options (e.g. layout pre-processing, mesh and simulation related settings)
proj.sti	frequency stimulus plan
proj.cfg	configuration file (e.g. paths to user/site/supplied substrate database locations)
proj.vpl	far field view plan, optional (for far field computation only)

Alternatively, use one of the AEL functions below.

```
dex_em_writeSimulationFiles(layoutContext,
simulationDirectory)
dex_em_writeSimulationFiles(libName, cellName,
emSetupViewName, simulationDirectory)
```

- **3.** This step is platform dependent and will set all necessary environment variables, e.g. HPEESOF_DIR, ADS_LICENSE_FILE, PATH and LD_LIBRARY_PATH (for linux).
 - On Linux- Open a command line window. For example, when using a ksh or bash:

```
export
HPEESOF_DIR=<path_to_ADS_installation_directory
> # for example: /hfs/dl/eesof/ADS2014_11
export
ADS_LICENSE_FILE=<path_to_ADS_license_file_or_s
erver> # for example: /hfs/dl/eesof/licenses
/license.lic
export PATH=$HPEESOF_DIR/bin:$PATH
```

- On Windows- Open a Command Prompt window.

NOTE: do NOT use double quotes.

```
set
HPEESOF_DIR=<path_to_ADS_installation_directory
> # for example: set HPEESOF_DIR=C:\Progam
Files\Keysight\ADS2014_11
set
ADS_LICENSE_FILE=<path_to_ADS_license_file_or_s
erver> # for example: set ADS_LICENSE_FILE=C:
\Program
Files\Keysight\EEsof_License_Tools\licenses\lic
ense.lic
set PATH=%HPEESOF_DIR%\bin;%PATH%
```

You can verify whether the setup is correct by executing following command: adsMomWrapper --version. This should return the Momentum engine version. If not, verify the environment variable settings.

4. Change directory (cd) to the simulation directory mentioned above. A complete Momentum simulation consists of different steps.

Simulation Step	Engine Option	Output Files
Substrate file pre-processing (e.g. expanding substrate for thick conductors)	-T	proj.sub including expanded layers for thick conductors
Substrate database generation (Green function calculation)	-DB	proj.qry that contains information on where the Green function database file is located
Layout pre-processing and mesh generation	-M	proj.mmd, proj.mrp, proj.stm, and others
S-parameter model generation	-3D	proj.cti, proj.afs, proj.msf, proj.sta, and others
Far Field calculation	-FF	proj.ant, proj.fff, proj.FFe

You can either invoke each step individually or when specifying the -O (capital letter O, not the number zero). option, the engine will run all steps from the beginning up to the one specified. For example:

- To run the simulation flow up to and including mesh generation, execute:

```
adsMomWrapper -O -M proj proj
```

- To run the simulation flow up to and including the S-parameter generation, execute:

adsMomWrapper -O -3D proj proj

The S-parameters are written to proj.cti (simulated points), proj.afs (interpolated curves) - both are in the citifile format - and statistics to proj.sta.

- To run the far field generation step only (assumes that the flow up to S-parameter model generation has been completed before), execute:

```
adsMomWrapper -FF proj proj
```

Extra command line options:

 You can override the simulation mode (RF = quasi-static or MW = full wave) by providing an extra command line option:

```
adsMomWrapper -O -3D --objMode=RF proj proj
```

Set to objMode to RF or MW to force respectively a quasi-static or fullwave simulation.

- By default, the S-parameter model generation writes results in the CITI file format. Add following options to additionally generate the dataset.

adsMomWrapper -0 -3D --dsName=<datasetName>
--dsDir=<datasetDirectory> proj proj

Omit the '.ds' extension in the dataset name. If you don't specify the dsDir, the dataset file(s) will be written in the current working directory.

- Generating the dataset(s) after a simulation with existing results can also be done using the -CD option

adsMomWrapper -CD proj proj

Viewing Simulations Results in 3D

In the 3D Viewer, you can view and analyze the following types of simulation data:

- Surface currents
- Fields (FEM only)
- Far fields and antenna parameters

Starting the 3D Visualization

Before using the visualization feature, you should 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.

This section provides information about the following topics:

- Visualizing 3D View before EM Simulations

- Visualizing Momentum Simulations
- Visualizing FEM Simulations
- Computing Radiation Patterns

Visualizing 3D View before EM Simulations

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 viewer consists of the same definition that will be used in the EM simulator. If a view is incorrect in the 3D viewer, 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:

- 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.
- **NOTE** If pins are not properly aligned on the bounding box of a design, it causes an invalid simulation set up. The 3D View from the Momentum perspective does not cause failure, since it does not support Waveguide Ports. On the FEM side, the waveguide ports are validated and if they fail validation, no design is exported for preview. To avoid this problem, move the pins in your ADS design so that they are properly aligned on the bounding box.

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.

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:

Name Vis Shd	Name Image: Name Image: Description Image: Des	Vis Shd
Image: Second system Image: Second system	 TOP top_met SubstrateLayer2 	
	 SubstrateLayer1 Closed_bottom 	

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:

	Vis	Shd
PORT_B		
FreeSpace	×	
_TOP	×	
cond	×	×
Subst	×	×
FreeMetal		
closed_b		
	PORT_B FreeSpace TOP cond Subst FreeMetal closed_b	Vis PORT_B FreeSpace X TOP X cond X Subst X FreeMetal closed_b

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:

isible	All	Conductors	None
haded	All	Conductors	None

In the Visible area:

- Click All to apply visibility to all the objects and materials.
- Click Conductors to apply visibility to all the conductors.
- 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 Conductors to shade all the conductors and remove shading from all the non conductors.

- 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.

erbes	8	
taterials Objects		And and a second s
Name Vis Shd > M OPRT_BOUNDARIES 2 > M OTHER BOUNDARIES 2 > M FreeSpace_box 2 > M FreeSpace_box 2 > M StrateSpace_box 2		
Color 0%	100% amparency	

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

Select Color	X
Basic colors	
Custom colors	Hue: 0 - Red: 200 - Sat: 255 - Green: 0 - Val: 200 - Blue: 0 -
	OK Cancel

- 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:

Measure 🛛 🔀
Reference Point
Object
Location
X 0.000 Y 0.000 Z 0.000
Current Point
Object
Location
X 0.000 Y 0.000 Z 0.000
Move Current Point to Reference
X 0.000 Y 0.000 Z 0.000
Distance 0.000mil

 Z-Scaling: You can change the geometry dimension of a model in the zdirection 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.

z	Scale		×
		Scal	e
	- [] –	10
	-	-	
	-	-	
	-	-	
	-	-	
	-	-	1
	-	-	
	-	-	
	-	-	
	- 6	5-	1

 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:

Cut Plane		×
-YZ Plane	-XZ Plane	XY Plane
323	140.5 	221.9
323 	 140.5	221.9
123	140.5	0
Value 323 Enable Cut Plane Flip Cut Show Plane	Value -140.5 Enable Cut Plane Flip Cut Show Plane	Value 221.9 Enable Cut Plane Flip Cut Show Plane

- Viewing 3D Connectivity: 3D Connectivity can be viewed by using the button on the tool bar. When this button is pressed, the viewer does an initial processing of the project connectivity. A progress bar is displayed

during this process. It is only done once per invocation of the viewer. After the initial processing, you can view the connectivity by clicking on the object. The selected object is highlighted. If the object is a conductor, it is shaded and all the connected conductors are shaded as well.

 Viewing Object Statistics: You can view detailed information about the 2D and 3D object parameters such as substrate layer, mask, and top statistics.

Nome	Validity	Points	Faces	Minimal Length	First Point	Second Point
BOTTOM_2	Valid	8	6	1.6000000000000000000e+001	(3.500000000e+002, 1.000000000e+002, 0.000000000	e+000) (3.500000000e+002
_TOP_2	Valid	8	6	1.600000000000000e+001	(3.500000000e+002,1.000000000e+002,0.000000000	e+000) (3.500000000e+002
_SubstrateLaye	Valid	8	6	2.0000000000000000e+001	(3.500000000e+002,1.00000000e+002,0.000000000	e+000) (3.500000000e+002
N85K_1_2	100	•	1	4.97777777777777756+001	(-2.000000000+002, 5.000000000+001, 0.00000000	00+000)(-2.000000000000000000000000000000000000
•						
	-	-	a la	and the second		
Woland 4	Object 2	Canada	Minn E	wet Daint I Canonal Daint		
Object 1	Object 2	Separa	stion F.	irst Point Second Point		
Object 1	Object 2	Separa	ation Fi	irst Point Second Point		
Object 1	Object 2	Separa	ation Fi	irst Point Second Point		
Object 1	Object 2	Separa	ation Fi	irst Point Second Point		
Object 1	Object 2	Separa	ation Fi	Inst Point Second Point		
Object 1	Object 2	Separa	ation Fi	Inst Point Second Point		
Object 1	Object 2	Separa	ation Fi	Inst Point Second Point		
Object 1	Object 2	Separa	ation F	Inst Point Second Point		
Object 1	Object 2	Separa	ation [Fi	Inst Point Second Point		
Object 1	Object 2	Separa	ation Fi	Inst Point Second Point	Inter Object Inspection	
istance Resoluti	Object 2	Separa	ation Fi	Inst Point Second Point	Inter Object Inspection	6.20648

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

Visualizing Momentum Simulations

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

- Surface Currents
- Far fields and antenna parameters

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 Keysight 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 Keysight Momentum Visualization window, as shown in the following figure:



The Keysight 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 Keysight Momentum Visualization window includes the following modifications:

- Object shading is removed.
- A shaded surface current plot is displayed.
- In addition to the Properties tab, Solution Setup and Plot Properties tabs are added for controlling the view.

In the Keysight Momentum Visualization window, the Properties tab provides the same functionality as the EM Previewer properties tab. However, in the Keysight 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:

-Mesh		
Mesh Visible		

NOTE For more information about the Properties tab, refer to Visualizing 3D View before EM Simulations.

You can plot results 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.

Select the Solution Setup tab present at the bottom of the window. In this tab:

 Define the excitation (source voltage and series impedance) to be applied at each port.

Excitation Type	Details
Single Port Excitation	1 Volt source and 50 Ohm series impedance at the selected port, 50 Ohm terminations at all other ports
User Defined Excitation	User-defined source voltage (amplitude in Volt and phase in deg) and series impedance (real and imaginary part in Ohm) for each port
Extracted Excitation	Frequency dependent excitation computed by circuit simulator during EM /Circuit co-simulation

- Select the frequency for which you want to see results.

After the excitation and frequency selection is done, the plots are automatically updated for the new configuration.

Plotting Properties

The Plot Properties tab enables you to control 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. Volume current is in A/m^2 , surface current (which is what Momentum computes) is in A/m. The following figure displays the plot properties tab:



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.

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

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 Keysight 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 Keysight 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 Keysight 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 Keysight FEM Visualization window, you can specify the type of mesh: Surface or Volume Mesh, as shown in the following figure:

Mesh	
Surface	Volume Mesh

NOTE For more information about the Properties tab, refer Visualizing 3D View before EM Simulations.

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 Keysight 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 Port for ports to be excited individually or User Defined 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

In the Visualization window, click the Plot Properties tab to select fields, field sensors, basic plots, and animation settings. The following figure displays the Plot Properties tab:

Plot Properties					
Field E Plot Magnitude Display Maximum Field Locations					
Field Sensors					
Name Show Enable					
Add Edit					
Shaded Arrow Contour Volume Bound					
Animation options					
Display update (ms) 100 Phase increment (deg) 10					
0 Phase 360					

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.



Displaying Maximum Field Locations

You can view maximum field locations by clicking the Displaying Maximum Field Locations button in Plot Properties. This opens the Maximum E Field Locations window. This window provides information about the E field locations and their values. If you select a particular frequency, it is highlighted in the Advanced Visualization window. The following figure displays the Maximum Field Locations window:

M	aximum E Field Loo	ations				×
						1
	#	Value	Х	Y	Z	
	1	1285.24	6.5749	-1	2	1
	2	958.518	5.87503	1	2	
	3	958.156	-4.25694	1	2	
	4	956.359	5.64981	-1	2	
	5	907.583	-4.83499	1	2	
	6	848.295	7.5	-1	2	
	7	800.487	5.06254	1	2	
	8	798.747	0.471001	1	2	
	9	773.537	-4.46585	-1	2	
	10	749.688	4.6563	1	2	
	11	729.967	1.28022	-1	2	
	12	709.128	-3.92807	-1	2	
	13	708.916	-5.00363	-1	2	
	14	699.842	7.5	1	2	
	15	696.64	1.10084	1	2	
	16	679.614	4.72471	-1	2	
	17	654.39	-0.158842	1	2	
	18	651.23	-3.39029	-1	2	
	19	632.623	-3.10086	1	2	
	20	626.142	1.73069	1	2	
						J

The following figure displays the highlighted frequency in the Advanced Visualization window:



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.

Field Sensors		
Name	Show	Enable
X:-3		
Z:0		
Z:0.25		
X:3		
Add		Edit

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:

Add Plane			Car.	×				
Select Obje	ct Plane	Select Three Points	Select Object					
Please shad	Please shade an object by clicking on an edge. ᠺ							
	Save							
Surface Name Surface_1								
		Cancel						

- 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.

Ac	dd Plane		and the second second	×
	Select Object Plane	Select Three Points	Select Object	
	Select a shaded object	. 💽		
		Save Selected Object	t	
		Save Shaded Objects	S	
	Surface Name Surface_	1		
		Cancel		

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.

Modify Surface			×
Surface Name Z:0	.25		Rename
		1	4.28
Press button to sel	lect point		\square
Normal Increment	0.0856		
Normal Value	0.25		
Reset	Delete		Done

Plotting Properties

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

- Shaded Plot
- Arrow Plot
- Contour Plot
- Volume 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.

Displaying a Volume Plot

The volume plot enables you to scale and specify arrow size, as shown in the following figure:



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.

Options
Max Color
Max (V/m) 1716.15
Min (V/m) 0
Min Color
System Values
System Max 2212.18
System Min 0
Restore System Max/Min
X arrow density 🧯 🖨 Y arrow density 6 🜲

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

Shaded	Arrow	Contour	Boundaries	
Boundari	es			
🔲 Bou	ndaries Visib	ole		
√ Ra	adiation	[√ Port	
V Pe	erfect E		✓ Symmetric	
V Ur	nassigned	[√ Other	

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:

Animation options					
Display update (ms)		100			
Phase increment (deg)		10			
0	Phase				360
Q <u> </u>	1 1	I.	I	I	1 1

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.

Exporting Field Data

You can export E or H fields data by using the Advanced Visualization window. In addition to exporting the field data per tetrahedra, you can export the data per uniform grid.

To export field data:

1. Select File > Export Field Data. The Export Field Data dialog box is displayed, as shown in the following window:

port	Field Data		
Data	a Sampling -		
۲	Tetrahedra	O Unifor	m 3D Grid
Divi	sions	20	
File	Format 🔿	xml 🖲 tx	t
File			Browse

- 2. Select one of the data sampling options Tetrahedra or Uniform 3D Grid. If you select Uniform 3D Grid, type the number of divisions required per grid.
- 3. Select the type of field you are exporting E or H.
- 4. Select a file format option xml or txt.
- 5. Type a file name in the File text box.
- 6. Click Browse to save the file at the required location.
- 7. Select the required location and click Save.
- 8. Click Export.

The export file contains a list of tetrahedra each with 4 point values and 10 field values, as shown in the following sample file:

```
т 1
P_1="4.167964e+000 -1.415037e+000 2.636480e+001"
P_2="6.683468e+000 3.139348e-001 2.959784e+001"
P 3="4.260608e+000 3.055021e-001 2.803953e+001"
P_4="6.421932e+000 2.846765e-001 2.614055e+001"
N_1="2.379960e+007 2.757609e+007 -7.738133e+006
-8.965870e+006 -4.548292e+005 -5.237584e+005"
N_2="2.070606e+007 2.399198e+007 -3.023256e+006
-3.502968e+006 -3.397782e+005 -3.917465e+005"
N_3="2.430381e+007 2.816046e+007 -3.119658e+006
-3.614634e+006 -2.683231e+005 -3.074952e+005"
N_4="1.989725e+007 2.305460e+007 -3.246496e+006
-3.761620e+006 -3.040257e+005 -3.501238e+005"
N 5="1.745019e+007 2.021953e+007 1.869907e+006 2.166790
e+006 -1.937719e+005 -2.238154e+005"
N_6="2.117090e+007 2.453070e+007 1.722775e+006 1.996282
e+006 -2.439926e+005 -2.807636e+005"
N 7="1.672967e+007 1.938458e+007 1.558138e+006 1.805409
e+006 -1.685716e+005 -1.944367e+005"
N_8="2.481198e+007 2.874945e+007 1.584858e+006 1.836456
e+006 -1.704304e+005 -1.940710e+005"
N_9="2.035784e+007 2.358837e+007 1.409211e+006 1.632806
e+006 -2.056144e+005 -2.360936e+005"
N_10="1.599067e+007 1.852821e+007 1.252236e+006 1.450838
e+006 -1.420236e+005 -1.634942e+005"
```

In the export file, the point values (P_1 to P_4) represents the X, Y, Z coordinates of the four vertices of each tetrahedron. The ten field values (N_1 to N_10) represents the field strengths at each of the four vertices and the midpoints of the six edges of each tetrahedron. Each value contains the real and imaginary parts of the x, y and z field components.

Computing Radiation Patterns

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. These terms are called far field components (E_{ff}, H_{ff}).

$$E(r \to \infty, \theta, \varphi) = E_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$
$$H(r \to \infty, \theta, \varphi) = H_{ff}(\theta, \varphi) \frac{e^{-jkr}}{r}$$
$$H = \hat{r} \times E/Z_{w}$$

The **H** vector relates to the **E** vector as shown above. Z_w is the free space wave impedance.

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. The origin or phase center of this XYZ coordinate system for the far field computation is at the center of the bounding box around the layout.

NOTE For Momentum,

- The far field is computed separately for the upper and lower open half space. The Z-coordinate of the phase center is the Zcoordinate where the upper or lower half space starts.
- There is a configuration variable, MOM3D_USE_GRAVITYCENTERTRANSFORMATIONFARFIELDS = FALSE, to keep the XY of the phase center at the origin of the layout (0,0). Defining Expert Options for Momentum provides information on how to specify this variable.


NOTE In the direction parallel to the substrate (theta = 90 degrees), parallel plate modes or surface wave modes, that vary as 1/sqrt(r), are also present. Although the modes dominate in this direction, they are not considered to be part of the far-fields.

The radiated power is a function of the angular position and the radial distance from the circuit. The variation of power density with angular position is determined by the type and design of the circuit. It can be graphically represented as a radiation pattern.

The far-fields can only be computed at those frequencies that were calculated during a simulation. The far-fields will be computed for a specific frequency and for a specific excitation state. They will be computed in all directions (theta, phi) in the open half space above and/or below the circuit. Besides the far-fields, derived radiation pattern quantities such as gain and axial ratio 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.

To get the E-field strength at a specific (x,y,z) location in the far field zone, do following steps:

- 1. Compute r, θ and ϕ from x, y, and z
- 2. Get the E_{ff} for the θ and ϕ
- **3.** $E(x,y,z) = E_{ff}(\theta, \phi) * e^{-j k r} / r$

About Antenna Characteristics

Based on the far fields, you can derive antenna characteristics such as gain, radiated power, polarization and other .

Electric Field and Radiation Intensity

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

$$U(\boldsymbol{\theta}, \boldsymbol{\varphi}) = \frac{1}{2} (\mathbb{E}_{f\!f}(\boldsymbol{\theta}, \boldsymbol{\varphi}) \times H_{f\!f}^*(\boldsymbol{\theta}, \boldsymbol{\varphi})) = \frac{1}{2Z_w} (|\boldsymbol{E}_{\boldsymbol{\theta}}(\boldsymbol{\theta}, \boldsymbol{\varphi})|^2 + |\boldsymbol{E}_{\boldsymbol{\varphi}}(\boldsymbol{\theta}, \boldsymbol{\varphi})|^2)$$

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

$$\begin{split} U_{max} &= max(U(\mathbf{0}, \mathbf{\phi})) \\ & \mathbf{\theta} \mathbf{\phi} \end{split}$$

The corresponding maximum electric field intensity is given by:

$$E_{\max} = \max_{\theta\varphi} \left(\operatorname{sqrt} \left(\left| \tilde{E}_{\theta} \right|^{2} + \left| \tilde{E}_{\varphi} \right|^{2} \right) \right)$$

Radiated and Input 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.

The input power, in Watts, is the net power (incident – reflected) delivered to the antenna.

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 calculates the radiation intensity in a certain direction versus the radiated power. Directivity is dimensionless and calculated by using the following equation:

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

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 is the ratio of the radiation intensity in a certain direction versus the intensity of a uniform radiation of the input power. The gain of the antenna is calculated as:

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

where P_{input} is the real power, in watts, delivered into the circuit. The 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:

$$P_{rad} = G$$

$$\eta = \frac{1}{P_{inj}} = D$$

Effective Area

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

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

Polarization

The polarization of the far field describes the orientation of its vectors (usually the electric field component) at a given point and how it varies with time. For antennas, we are concerned with the polarization of the field in the plane orthogonal to the direction of propagation. This is the plane defined by the vectors of the far field ($E_{\rm ff}$).

The polarization is the locus traced by the extremity of the time-varying field vector at a fixed observation point.

According to the shape of the trace, three types of polarization exist: linear, circular and elliptical. Any polarization can be represented by two orthogonal linear polarizations, (E_{θ}, E_{ω}) , whose fields are out of phase by an angle of δ_{I} .

- If $\mathbf{\delta}_1 = 0$ or $n\pi$, then linear polarization results.
- If $\mathbf{\delta}_{L} = \pi/2$ and $|\mathbf{E}_{\mathbf{\theta}}| = |\mathbf{E}_{\mathbf{\omega}}|$, then circular polarization results.
- In the most general case, elliptical polarization is defined.
 Mathematically, the linear and circular polarizations are special cases of the elliptical polarization. In practice, the term elliptical polarization is used to indicate polarizations other than linear or circular.

Polarization in terms of two orthogonal linearly polarized components

The far field phasor vector can be decomposed into two orthogonal components, $\rm E_{0}$ and $\rm E_{o}.$ Both are in general complex phasors.

$$\boldsymbol{E} = \tilde{E}_{\theta} \widehat{\boldsymbol{\theta}} + \tilde{E}_{\varphi} \widehat{\boldsymbol{\varphi}}$$

Those fields are out of phase by an angle of $\boldsymbol{\delta}_{|}$, defined as

$$\frac{\tilde{E}_{\varphi}}{E_{\varphi}} / \frac{\tilde{E}_{\theta}}{E_{\theta}} = e^{j \, \delta_L}$$

The field vector at a given point traces an ellipse as a function of time.



An alternative decomposition into two orthogonal components has the field phasor vectors aligned with the major and minor axis of the ellipse, E_{co} and E_{cross} . Those can be derived as follows

$$\begin{split} \tilde{E}_{co} &= \tilde{E}_{\theta} \cos \tau + \tilde{E}_{\varphi} \sin \tau \\ \tilde{E}_{cross} &= -\tilde{E}_{\theta} \sin \tau + \tilde{E}_{\varphi} \cos \tau \end{split}$$

Where is the tilt angle of the polarization ellipse.

Polarization in terms of two circularly polarized components

Similarly, the field can be decomposed in terms of two circularly polarized components, one right-handed and the other left-handed. Although this decomposition is less easy to perceive, it is very useful in the calculation of the polarization ellipse parameters. The total field phasor is represented by

$$\boldsymbol{E} = \tilde{E}_R \left(\frac{\widehat{\boldsymbol{\theta}} - j\widehat{\boldsymbol{\varphi}}}{\sqrt{2}} \right) + \tilde{E}_L \left(\frac{\widehat{\boldsymbol{\theta}} + j\widehat{\boldsymbol{\varphi}}}{\sqrt{2}} \right)$$

Again, these are in general complex phasors. The relative phase difference between the two, ${\bf \delta}_{\rm C}$, is defined as

$$\frac{\tilde{E}_L}{E_L} / \frac{\tilde{E}_R}{E_R} = e^{j \, \delta_C}$$

The relation between the linear-component and the circular-component representations are give by

$$\widetilde{E}_{R} = \frac{1}{\sqrt{2}} \left(\widetilde{E}_{\theta} + j \widetilde{E}_{\varphi} \right)$$
$$\widetilde{E}_{L} = \frac{1}{\sqrt{2}} \left(\widetilde{E}_{\theta} - j \widetilde{E}_{\varphi} \right)$$

Polarization vector, polarization ratio and axial ratio

The polarization vector is the normalized phasor of the electric field vector. It is a complex-valued vector of unit magnitude.

$$\widehat{p}_L = \frac{E}{E_m} = \frac{\widetilde{E}_{\theta}}{E_m} \widehat{\theta} + \frac{\widetilde{E}_{\varphi}}{E_m} \widehat{\varphi}$$
, with $E_m = \sqrt{E_{\theta}^2 + E_{\varphi}^2}$

The linear polarization ratio is the ratio of the phasors of the two orthogonal polarization components. In general, it is a complex number.

$$\check{r}_L = r_L e^{j\,\delta_L} = \frac{\tilde{E}_{\varphi}}{\tilde{E}_{\theta}} = \frac{E_{\varphi} e^{j\,\delta_L}}{E_{\theta}}$$

In the case of the circular-component representation, the polarization ratio is defined as:

$$\check{r}_{C} = r_{C} e^{j\delta_{C}} = \frac{E_{L}}{\tilde{E}_{R}} = \frac{E_{L}}{E_{R}} e^{j\delta_{C}}$$

The axial ratio of the polarization ellipse can be computed from the circular polarization ratio.

$$AR = \frac{r_{C} + 1}{r_{C} - 1} = \frac{E_{L} + E_{R}}{E_{L} - E_{R}}$$

The tilt angle of the polarization ellipse is simply

$$\tau = -\frac{\delta_C}{2}$$

Antenna polarization

The polarization of a transmitting antenna is the polarization of its radiated wave in the far field zone. The polarization of a receiving antenna is the polarization of a plane wave, incident from a given direction, which, for a given power flux density, results in the maximum available power at the antenna terminals. The antenna polarization is defined by the polarization vector of the radiated (transmitted) wave. The polarization vector of a wave in the coordinate system of the transmitting antenna is represented by its complex conjugate in the coordinate system of the receiving antenna:

$$\widehat{\boldsymbol{p}}_w^r = (\widehat{\boldsymbol{p}}_w^t)^*$$

The conjugation is relevant in case of circularly or elliptically polarized waves. The transmitting-mode polarization of an antenna is the conjugate of its receiving-mode polarization vector.

Polarization loss factor and polarization efficiency

In general, the polarization of the receiving antenna is not the same as the polarization of the incident wave. This is called polarization mismatch. The polarization loss factor, PLF, characterizes the loss of power because of polarization mismatch:

$$\mathrm{PLF} = |\widehat{\boldsymbol{p}}_i.\widehat{\boldsymbol{p}}_a|^2$$

In the above, the two polarization vectors are respectively for the incident field and the antenna.

The PLF, also called polarization efficiency, is a value between 0 and 1. If the antenna is polarization matched, then PLF=1, and there is no polarization power loss. If PLF=0, then the antenna is incapable of receiving the signal. The PLF for an antenna of RH circular polarization (in transmitting mode) that receives a RH circularly polarized incident wave is 1. Simularly, the PLF = 0 for receiving a LH circularly polarized incident wave.

Calculating Far Fields

To calculate far fields, choose EM > Post-Processing > Far Field.

In case of an FEM simulation, far fields will be computed before opening the 3D Viewer. In case of a Momentum simulation, far fields will be compute when required by the 3D Viewer.

Viewing Far Fields

Two windows are displayed, 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 the port excitation and frequency as specified on the Solution Setup tab.

Image: Solution Solution Solution	File View Tools Options Window Help			
PROFENSE Image: Maximum Red Locations Animation regions Image: Maximum Red Locations Animation regions Image: Maximum Red Locations Parts: Incomment (Strip) Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Parts: Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations Image: Maximum Red Locations </th <th>ی 💭 💭 🗗 🔍 🔄 🖉 🖉 🖉 🖉</th> <th>I 🕲 🕂</th> <th></th> <th></th>	ی 💭 💭 🗗 🔍 🔄 🖉 🖉 🖉 🖉	I 🕲 🕂		
Solid goode Biscle Complay Maximum Red Locations Annata Dapkay Maximum Red Locations Annata Dapkay John (m) Doublay John (m) Point in the intervent (ke) Point in the intervent (ke) Doublay Antenna Parameters in Data Display Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Solution Statis Properties Solution Statis Properties Solution Statis Properties Solution Statis Solution Statis	Plot Properties	² (11		
B Sake Diplay Mamum Field Locations Annabic Diplay Mamum Field Locations Pase Increment (deg) Diplay Mamum Field Locations Mamum dB -0 Tangpeency Anterna Parameters Diplay Anterna Param	Scaling options			
Display Maxmun Field Loadsons Armaton options Armaton options Display Maxmun Field Loadsons Display Maxmun Field Loadsons O Procerosment (dog) Procerosment (dog) <td>🔄 d8 Scale</td> <td></td> <td></td> <td></td>	🔄 d8 Scale			
Annabas Diplay update (m) Pase norement (deg) Pase norement (deg) Pase norement (deg) Pase norement (deg) Pase norement (deg) Requestory Reque	Display Maximum Field Locations			
Preder 0 <td>Animation options</td> <td></td> <td></td> <td></td>	Animation options			
Procenter Solution Setue Por Properties	Display update (ms) 100			
Phase 500 Shaded Arrow Contox For Field Cut Frequency 8.00000000+0000 For Normalize	Phase increment (deg) 10			
Subded arrow Contour For Fields The Field Cut. Frequency 8.00000000+008 Pot Image: Pot contour For Fields are field Cut. Pot Image: Pot contour For Fields are field Cut. Image: Pot contour For Fields are field Cut. Pot Image: Pot contour For Fields are field Cut. Image: Pot contour For Fields are field Cut. Pot Image: Pot contour For Fields are field Cut. Image: Pot contour For Fields are field Cut. Digitize Anterna Parameters in Data Digitize Image: Pot contour For Field Cut. Progenese Solution Seture Pot Properties Outcome contour For Field Cut. Progenese Solution Seture Pot Properties Pot Properties	0 Phase 360			
Procetters Solution Setur Por Progetters Other Port Progetters Solution Setur Port Progetters	Y			
Prequency 8.0000000 + 008 Pot Importante Normable Importante Properties Solution Setue Properties Solution Setue Properties Solution Setue Port Properties Solution Setue Properties Solution Setue Port memory Pot Properties	Shaded Arrow Contour Far Fields Far Field Cut			
Port Important Promatice Important Minimum dB 0 Transparency 0 Dipolary Anterna Parameters Dipolary Anterna Parameters Dipolary Anterna Parameters 0 Dipolary Anterna Parameters 0 0	Frequency 8.0000000e+008			
I hormalize I tog sole Morrun d8	Plot E 💌			
Winnundb Imageneters Binnundb Imageneters Digitey Anterna Parameters in Data Digitey Imageneters Digitey Anterna Parameters Imageneters Opporters Solution Setue Pot Properties Other mm	V Normalize			
Properties Solution Setup Pot Properties	Minimum dBi -40			
Antenna Parameters in Data Digday Digday Antenna Parameters in Data Digday	Transparency 0			
Diplay Antenna Parameters in Data Diplay Diplay Antenna Parameters in Data Diplay	Antenna Parameters			
Properties Solution Setup Pot Properties	Display Antenna Parameters in Data Display			
Properties Solution Setup Pot Properties				
Properties Solution Setup Pot Properties Min 0.000e+000 Alm Image: Calcor Key Hele Calcor Key Properties Solution Setup Pot Properties Max B.895e+000 Max Hele Calcor Key				
Occore+000 A/m Occore+000 A/m Occore+000 A/m Occore+000 A/m Occore+000 Max Hele Color Key Hele Color Key Occore+000 Max B.899e+000 Use Gobal Min,Max Occore+000 Max S.899e+000 Use Gobal Min,Max				
Properties Solution Setup Pot Properties		Z_Y		
Properties Solution Setup Pol Properties		N 1997	7	
Occore+000 Alm Alm Cooce+000 Alm Cooce+000 Max Min O.cooce+000 Max B.895e+000 Uae Global Min,Max Hide Color Key			Text	
Properties Solution Setup Plot Properties Orbit mm		0.000++000 0.10		
Properties Solution Setup Plot Properties Orbit mm		Threshold Global Min Max	Hide Color Key	
Properties Solution Setup Pot Properties		Min 0.000e+000 Max	x 8.889e+000 Use Global Min/Max	
Orbit mm	Properties Solution Setup Plot Properties			
	Orbit mm			

Select the Plot type:

- E = sqrt(mag(E Theta)2 + mag(E Phi)2)
- E Theta
- E Phi
- E Left
- E Right
- E Co
- E Cross
- Axial Ratio

If you want the data normalized to a value of one, enable Normalize. For Axial Ratio choices, set the Minimum dB. 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 input power, radiated power and maximum directivity, gain, radiation intensity, E-field, and angles in the direction of maximum radiation.

📶 Antenna Parameters		8 ×
Frequency (GHz)		7.65953
Input power (Watts)		0.00237776
Radiated power (Watts)		0.00222541
Directivity(dBi)		7.60076
Gain (dBi)		7.31316
Radiation efficiency (%)		93.5925
Maximum intensity (Watts/Steradian)		0.00101924
Effective angle (Steradians)		2.18341
Angle of U Max (theta, phi)	2	22
E(theta) max (mag,phase)	0.346601	159.074
E(phi) max (mag,phase)	0.804875	149.636
E(x) max (mag,phase)	0.054846	-136.571
E(y) max (mag,phase)	0.874529	151.03
E(z) max (mag,phase)	0.0120962	-20.9262
OK		

In the Visualization window, click the Far Field Cut tab to view the far field in a 2D data display. Select the Enable flag and select to make a Theta or Phi cut. For a Phi cut, the angle phi is kept constant. The angle theta, which is relative to the z-axis, is swept. This produces a vertical cut that is perpendicular to the XY layout plane.





To view the cut data in a Data Display window, click *Display Cut in Data Display.

Exporting Far Field Data

Dataset file with Antenna Characteristics

The Antenna Parameters data display file contains following variables versus frequency:

Variable	Description
freq	Frequency in Hz
Gain, Directivity	Gain and Directivity values in dBi
Efficiency	Antenna efficiency in percent (%)
RadiatedPower	Radiated power in Watts
MaximumIntensity	Maximum radiation intensity in Watss/steradian

Variable	Description
E_phi, E_theta, E_x, E_y, E_z	Electric far field components in Volts in the direction of maximum radiation intensity
EffectiveAngle	Effective radiation angle in steradian

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
Frequency, InputPower, RadiatedPower, Efficiency	Quanties at the given frequency
E_max, Theta_max, Phi_max, Gain_max, Directivity_max	Quantities in the direction of maximum radiation.
Theta or Phi, CutType, CutAngle	The swept parameter of a cut. When the CutType is "Phi", Theta will be available. When the CutType is "Theta", the Phi variable will exist. The CutAngle is the angle of the cut.
Gain, Directivity, RadiationIntensity, EffectiveArea	Gain (dBi), Directivity (dBi), Radiation intensity (in watts/steradian), Effective area (in m ²).
Etheta and Ephi	Absolute field strength (in volts) of the theta and phi electric far-field components.
TiltAngle	Tilt angle of the polarization ellips
Htheta and Hphi	Absolute field strength (in amperes) of the theta and phi magnetic far-field components.
Eco and Ecross	Absolute field strength of co and cross polarized electric far-field components.
Elhp and Erhp	Absolute field strength of respectively left-hand and right-hand circular polarized electric far-field component.
AxialRatio	Axial ratio, derived from left-hand and right-hand circular polarized far-field components.

Variables in the Standard and AFS Dataset

Variable	Description
freq	The independent variable frequency, which was specified during the simulation setup. The number of data points is based on the sweep type and frequency plans specified during simulation.
GAMMA n	The (modal) propagation constant of Port <i>n</i> . It is used in de-embedding, and is calculated for single, differential, and coplanar ports only.
PORTZ n	The impedance of Port n as defined in the Impedance fields of the Port Editor dialog box. If an impedance is not specified, 50 Ω is assumed.
S	S matrix, normalized to PORTZ <i>n</i> .
S(<i>i, j</i>)	S-parameters for each pairing of ports in the circuit, normalized to PORTZ <i>n</i> .
S_50	S-matrix, normalized to 50 ohms.
S_50(<i>i, j</i>)	S-parameters for each pairing of ports in the circuit, normalized to 50 ohms.
S_Z0	S matrix, normalized to ZO.
S_ZO(<i>i, j</i>)	S-parameters for each pairing of ports in the circuit, normalized to Z0 of each port.
Z0 n	Reports the characteristic impedance of the transmission line connected to Port <i>n</i> . This is not calculated for internal or coupled ports, in these cases 50 ohms is substituted.
	70 values are not computed for Single Ports, if more than 2 ports are used in

NOTE Z0 values are not computed for Single Ports, if more than 2 ports are used in the same reference plane. In this case, the Z0 values are set to 50 ohms. To obtain a scalar Z0 for each port, they must be put on different reference planes.

ASCII file with 3D Far Field Data

The 3D far field data can be written in an ASCII formatted file.

- In case of an FEM simulation, launch the far field computation from the EM Setup window (Tools > Far Field). Search for the design.fff file under <workspace>/<library>/<cell>/<layout>/<emSetup>_FEM/proj.ep/... /level<i>/...
- In case of a Momentum simulation, create a proj.vpl file in the <workspace> /<library>/<cell>/<layout>/<emSetup>_MoM directory and run MomEngine from command line (see here) to generate the proj.fff file. The format for the Momentum proj.vpl file is:

VISUALIZATIONTYPE 1; # 3D output PARAMETER FREQUENCY, UNITS GHz, PT <f1> [, PT <f2>]...; PARAMETER PHI, UNITS DEG, PT 0; # Needed, but not used for 3D output VAR THETA, UNITS DEG, PT 0; # Needed, but not used for 3D output PORT <portNr1>, UNITS VOLT, UNITS DEG, AMPLITUDE <V1_ampl> PHASE <V1_phase>, UNITS OHM, UNITS RAD, AMPLITUDE <Z1 ampl> PHASE <Z1_phase> [,PORT <portNr2>, UNITS VOLT, UNITS DEG, AMPLITUDE <V2_ampl> PHASE <V2_phase>, UNITS OHM, UNITS RAD, AMPLITUDE <Z2 ampl> PHASE <Z2_phase>]...;

The data is saved in the following format:

```
// loop over frequencies
#Frequency <f> GHz
#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</pre>
(E_phi)> <imag(E_phi)>
<theta_n> <phi_0> <real(E_theta)> <imag(E_theta)> <real</pre>
(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</pre>
(E_phi)> <imag(E_phi)>
<theta_n> <phi_1> <real(E_theta)> <imag(E_theta)> <real</pre>
(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</pre>
(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 normalized 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 is E_max

$$\sqrt{\langle E_{theta_max} \rangle^2 + \langle E_{phi_max} \rangle^2}$$

with E_theta_max and E_phi_max respectively the E_theta and E_phi in the direction of maximum radiation intensity.

See Also Example Printed Dipole Antenna

EM Circuit Cosimulation

EM Circuit Cosimulation

Contents

- EM Circuit Cosimulation Overview
- Using the EM Cosimulation View
 - EM Cosimulation View
 - EM Cosimulation Controller Component
 - Performing an EM Circuit Cosimulation with an EM Cosimulation View
 - Using the EM Cosimulation View Tutorial
- Using the EM Model View
 - Creating and Using an EM Model View
 - Performing an EM Circuit Cosimulation with an EM Model View
- Using the EM Circuit Excitation AEL Addon

Reference

- Automatic EM Circuit Partitioning
- The EM Model View
- Manual Versus Automatic EM Circuit Partitioning

EM Circuit Cosimulation Overview

EM Circuit Cosimulation Overview

You can configure a circuit simulation to use an EM-based model for a cell instance or parts thereof by using the following view types:

- EM Cosimulation view: This view facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator (e.g. a nonlinear device) or for which a model already exists (e.g. a built-in primitive, a schematic, or an EM Model). You can directly simulate circuit components inside an EM simulated layout. For more information, see Using the EM Cosimulation View.
- EM Model view: This view contains the S-parameter model extracted by an EM simulator for the entire (flattened) layout. For more information, see Using the EM Model View.

Using the EM Cosimulation View

Using the EM Cosimulation View

- EM Cosimulation View
- EM Cosimulation Controller Component
- Performing an EM Circuit Cosimulation with an EM Cosimulation View
- Using the EM Cosimulation View Tutorial

EM Cosimulation View

EM Cosimulation View

An EM Cosimulation view is used by the circuit simulator to model a layout that is partly circuit simulated and partly EM simulated (using an autogenerated traditional EM model). When netlisted, it integrates the EM model for the EM simulated partition with circuit models for the circuit simulated instances. The EM model can be precomputed or will be computed during circuit simulation. It caches the S-parameters to improve performance for repeated simulations.

EM Cosimulation views are compatible with Dynamic Model Selection, which enables you to switch between different models for the same cell. For example, a faster, all schematic-based representation in the schematic view, and an accurate mixed EM/Circuit-based representation in the emCosim view.

Creating an EM Cosimulation View

An EM Cosimulation View is created from an EM Setup by performing the following steps:

- 1. Open the EM Setup window.
- 2. Select EM Cosimulation in the Setup Type panel.
- 3. Ensure that Views is selected in the Generate drop-down list.
- 4. Click Go after finalizing the setup.



Specify Automatic EM/Circuit Partitioning

The Partitioning tab of an EM Setup controls whether individual instances are EM simulated or circuit simulated.

Mom RF Cosim	Partitioning						
Layout Partitioning	Policy: Enter sp	ace separated list	to override stand	ard policy: preferred	d em circuit		
Substrate							
10 Ports							
He Frequency plan	Parent Library	Parent Cell	Parent View	Instance Name	Instance Lib:Cell	Partitioning Rule	Modelling Action
E Options	RF_Board_lib	LNA_stage_1	layout	C1	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
Resources	RF_Board_lib	LNA_stage_1	layout	C2	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
Cosimulation	RF_Board_lib	LNA_stage_1	layout	C3	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
INOTES 1	RF_Board_lib	LNA_stage_1	layout	C4	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	C5	Board_Design_Kit:C	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	L1	Board_Design_Kit:L	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	L2	Board_Design_Kit:L	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	Q1	Board_Design_Kit:HBFP-0420	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R1	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R2	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R3	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
	Rule text	style: Defined by s an EM Cosimulat	policy - <u>Instance</u> ion	override - Defined	here		

The partitioning policy at the top of the tab and the partitioning rules defined for a cell allow ADS to automate this EM/circuit modeling choice, which can be overridden in the tab. For more information on automating the modeling choice, see Automatic EM Circuit Partitioning.

Besides the Partitioning setup, all other settings of the EM Setup are relevant as they will be used for simulating the EM partition. You need to specify the substrate, define ports, set up the simulation options such as mesh parameters, and specify a frequency plan for the EM model generation. For more information, see Setting up EM Simulations.

Generating the EM Cosimulation View

After finalizing the EM simulation setup, click Go. This does not immediately start an EM simulation. Instead it generates several cellviews. For the EM simulated partition, the EM Setup generates a layout and traditional EM model in a new cell named after the original cell with an '_emCosim' suffix. The EM Setup also generates a EM Cosimulation view emCosim in the original cell, which can be used as regular netlistable circuit model for that cell. The netlisted EM Cosimulation view integrates the circuit simulated instances with the EM model.

To see the names of generated views, click the Cosimulation option in the left pane of the EM Setup window, as shown in the following figure:



The emCosim view is available in the ADS Main window, as shown in the following figure:



Double-click the emCosim view in the ADS Main window to open the EM cosimulation window. Selecting the Necessary views option from the Generate drop-down list regenerates the emCosim view (itself) and the cell representing the EM simulated partition (which typically contains a layout and EM model).

File Tools Help
·
Setup
Summary
EM Simulator used with cosimulation:
Workspace: C:\Users\mdewilde\RE Board wrk
Library: RE Board lib
Cell: LNA stage 1
View: layout
Substrate:
Substrate: FR4_DoubleSided (defined in library: Board_Design_Kit_tech)
Ports:
3 ports defined
Frequency plan:
Adaptive from 0 GHz to 4 GHz (Npts: 50 (max))
Default
All simulation options are initialized and ready for simulation.
EM simulation resources:
Simulation on host:Local
Cosimulation:
EM Model representing the EM partition:
RF_Board_lib:LNA_stage_1_emCosim:emModel
Edit Generate Necessary views Go Go

In addition, a new cell with '_emCosim' suffix is created, which represents the EM partition. The cell representing the EM partition consists of a layout and a traditional EM model. The EM model stores the EM simulation data. You can edit the layout, but must not change the pin ordering.



NOTE If there are significant changes in the original layout, you must regenerate the EM cosimulation view, which will also regenerate the cell representing the EM partition.

Parameterized Layout View

The intermediate layout view is not created when the layout of the EM partition is parameterized (scalable). The EM simulation input files are generated on the fly from the partitioning information once the parameter values are known during the EM cosimulation.

You can control to which cell parameters the EM Model is sensitive. Select 'Include all parameters except these:' and leave the edit box blank to include all parameters. Each real-valued cell parameter will become an EM Model parameter. You can exclude one or more parameters by entering their name(s) as a space separated list in the edit box. The other selection 'Include only these parameters:' may be more convenient when only a few real-valued cell parameters change the layout.

How are pins and ports handled?

The layout for the EM simulated partition contains copies of the pins of the top level layout and the pins of circuit simulated instance layouts (rotated 180 degrees). Pins that are no longer connected to the EM simulated partition according to the Physical Connectivity Engine are removed from the generated layout. The connectivity of mutually touching removed pins will be guaranteed by the netlist of the cosimulation.

Area pin shapes are always copied to the EM simulated partition layout. The new shapes retain their area pin function, unless the old shape is carrying the custom property eeExcludeFromEmPin=1 (also editable in AEL using db_add_property()). Alternatively, the property eeShrinkAreaPinToContactRegion can be used to reduce the size of the EM simulated area pin to the contact region.

A polyline edge pin in a circuit simulated instance allows a flexible connection without copying filled shapes as in the area pins case.

Only the effectively connected section of such a pin is copied as edge pin to the EM simulated partition layout.

For all pins that have been copied to the generated layout, EM port definitions for these pins are copied from the original toplevel and circuit simulated instance layouts as-is.

The port definitions of circuit simulated instances can be edited on their layout using EM > Port Editor or, for artwork macros, using db_set_port_definitions().

You can refer the shipping example Pin_Handling_For_EmCosim_wrk.7zap, which illustrates a number of different possible pin setups inside a circuit simulated component. To open this example, select File > Open > Example > Momentum > emcktcosim. The README design of this example workspace provides a detailed explanation. The following images displays the pin setup of the circuit simulated component, a design using this component, and the resulting EM simulated partition.



Circuit Simulated Component

Design



Resulting EM Simulated Partition



Precomputing the EM Model

The EM model of the EM simulated partition caches the S-parameters to improve circuit simulation performance. When it is created, all the settings specified in the EM setup window are copied to the EM Model. Later, during 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. For more information, see The EM Model View.

You can precompute the EM model for default parameter values:

- 1. In the Main window, double-click the EM Model present in the cell representing the EM partition. The EM Model window is displayed.
- 2. Click the Database tab.
- **3.** Click Evaluate model for parameter default values. A message box is displayed.

🗱 LNA_stage_1_emCosim [RF_Board_lib:LNA_stage_1_emCosi 💼 📼 💌
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>H</u> elp
📁 🖬 🤌 🤁 🖪 🚺 🌲
Simulation Setup Database Interpolation Options
The database is empty.
Evaluate model for default parameter values

4. After the evaluation is complete, a new database entry has been created:

File Edit Iools Help	🔛 LNA_stage_1_emCosim [RF_Board_lib:LNA_stage_1_emCosi 👝 📼 💌
Simulation Setup Database Available data Name data.000 Delete All	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>H</u> elp
Simulation Setup Database Interpolation Options Available data Name data.000 data.000	📁 🗟 🤌 🔁 🚺 🌲
Available data	Simulation Setup Database Interpolation Options
Name data.000	Available data
data.000	Name
Delete All	data.000
Delete All	
	Delete All

Using an EM Cosimulation View for Circuit Simulation

After creating an EM Cosimulation view, you can perform the EM/circuit cosimulation. For more information, see Performing an EM Circuit Cosimulation with an EM Cosimulation View.

Example

Consider the cell MyResistor in following workspace:



In this workspace, the layout view consists of a 50 Ohm resistor component ads_rflib:R_Pad1 in the middle with two transmission line components ads_tlines:MLIN on either side. The associated schematic view contains the ads_tlines:MSUB component, which defines the substrate for the circuit.



In this design, the ads_tlines:MLIN instances can be simulated by both EM and circuit simulation. On the other hand, the ads_rflib:R_Pad1 resistor component is not EM simulatable. The actual layout for that component is unknown. Only its pads to make connection with the rest of the circuit are specified. An EM Model generated for the layout 'as-is' would represent an open circuit due to the gap between the resistor pads. Since the resistor is circuit simulatable, an EM/Circuit cosimulation can provide a model for the cell.

In the EM Setup, the partitioning can be edited to select the 'circuit' partitioning rule for the resistor. If the 'em' rule has been deleted for the resistor component, the 'circuit' selection is performed automatically.

After clicking Go, the emCosim view an a new cell with a layout and EM Model for the EM partition are generated:



The emCosim view is a circuit simulatable cell view supported by the Dynamic Model Selection in ADS. Selecting this view for simulation in the MyTestbench schematic will allow to compute S-parameters for the MyResistor cell whereby the resistor is circuit simulated and the transmission lines are EM simulated.



Performing an EM Circuit Cosimulation with an EM Cosimulation View

Performing an EM Circuit Cosimulation Using the EM Cosimulation View

The EM/Circuit Cosimulation feature facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator (e. g. a nonlinear device) or for which a model already exists (e.g. a built-in primitive, a schematic, or an EM Model).

A cell that has an EM Cosimulation View can be instantiated in a testbench schematic to run the cosimulation.

Perform the following steps:

- 1. Set up a Layout
- 2. Specify Automatic EM/Circuit Partitioning.
- 3. Generate an EM Cosimulation View.
- 4. Insert a symbol in the Schematic.
- 5. Run EM Circuit Cosimulation.

Set up a Layout

There are two basic ways to create a layout. Use the one that best complements your own design methodology and suits your design needs:

- Layout synchronized from a schematic
- Directly drawn or imported layout

Configure the EM Setup

After you have set up a layout, you need to create an EM Setup and configure the partitioning, i.e. whether instances will be circuit simulated or EM simulated:

- 1. Select EM > Simulation Setup to open the EM Setup window.
- 2. Select EM Cosimulation in Setup Type.

Mom RF Cosim	Setup Type	
Layout	EM Simulation/Model	EM Cosimulation

- 3. Select Partitioning in the left pane of the EM Setup window.
- 4. Select an instance for which you want to specify a partitioning rule. By default, the available rules are circuit and em.
- 5. Select the desired rule from the drop-down list in the Partition Rule column:

RF_Board_Flow_lib:new1_des	ign:emSetup (EM Setup for	cosimulation)					
ile <u>T</u> ools <u>V</u> iew <u>H</u> elp							
🖥 🦻 🥙 🖪 🎴	\triangleright						
4 🎮 Mom RF Cosim	Partitioning						
🔁 Layout							
🛞 Partitioning	Policy: Enter space	separated list to (override standard	l policy: preferred er	m circuit		
🛀 Substrate	> >						
10 Ports							
Frequency plan Output plan	Parent Library	Parent Cell	Parent View	Instance Name	Instance Lib:Cell	Partitioning Rule	Modelling ^ Action
E Options	RF_Board_Flow_lib	new1_design	layout	CI	Board_Design_Kit:C	circuit	🗾 👷 circui fror
Resources	RF_Board_Flow_lib	new1_design	layout	C2	Board_Design_Kit:C	<u>circuit</u>	- Use layout settin
🙀 Cosimulation	RF_Board_Flow_lib	new1_design	layout	C3	Board_Design_Kit:C	circuit	circuit
Notes	PE Peard Eleve lib	newl decign	lavout	01	Board Design KityC	circuit	em



For more information on configuring the partitioning, see Automatic EM Circuit Partitioning.

Also, define an EM Simulation Setup to:

- Specify the substrate.

- Define ports.
- Set up the simulation options, such as mesh parameters.
- Specify a frequency plan for the EM model generation.

Generate the EM Cosimulation View and EM Partition

After finalizing the EM simulation setup, click the Go button. This does not immediately start an EM simulation. Instead, it generates several cellviews. For the EM simulated partition, the EM Setup generates a layout and traditional EM model in a new cell named after the original cell with an '_emCosim' suffix. The EM Setup also generates a EM Cosimulation view emCosim in the original cell, which can be used as regular netlistable circuit model for that cell. The netlisted EM Cosimulation view integrates the circuit simulated instances with the EM model.

Mom RF Cosim	Partitioning						
🔁 Layout							
Partitioning	Policy: Enter sp	ace separated list	to override stand	lard policy: preferred	l em circuit		
* Ports	<u> </u>						
Frequency plan	Parent Library	Parent Cell	Parent View	Instance Name	Instance Lib:Cell	Partitioning Rule	Modelling Action
E Options	RF_Board_lib	LNA_stage_1	layout	C1	Board_Design_Kit:C	circuit	👷 circuit using ports from layout
Resources	RF_Board_lib	LNA_stage_1	layout	C2	Board_Design_Kit:C	circuit	👷 circuit using ports from layout
Corimulation Notes	RF_Board_lib	LNA_stage_1	layout	C3	Board_Design_Kit:C	circuit	👷 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	C4	Board_Design_Kit:C	circuit	👷 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	C5	Board_Design_Kit:C	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	11	Board_Design_Kit:L	circuit	👷 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	L2	Board_Design_Kit:L	circuit	👷 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	Q1	Board_Design_Kit:HBFP-0420	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R1	Board_Design_Kit:R	circuit	👷 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R2	Board_Design_Kit:R	circuit	💹 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R3	Board_Design_Kit:R	circuit	👷 circuit using ports from layout

Optionally Precompute the EM Model

The autogenerated EM Model caches the calculated S-parameters of the EM simulated partition. This calculation will be automatically triggered during the cosimulation if the data is not yet cached.

If the testbench schematic is not yet ready, you can already precompute the EM model ahead of time, so you wouldn't have to wait for this during the actual cosimulation.

You can precompute the EM model for default parameter values:

1. In the Main window, double-click the autogenerated EM Model in the EM Cosimulation view:

INA_stage_1
emCosim
EM emSetup
🔁 layout
🔛 schematic
> symbol
LNA_stage_1_emCosim
EM emModel
🔁 layout

- 2. Click the Database tab.
- 3. Click Evaluate model for parameter default values.

🔣 LNA_stage_1_emCosim [RF_Board_lib:LNA_stage_1_emCosi 👝 📼 💌
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>H</u> elp
🖆 🗟 🤊 🥙 🔁 🗊 🌲
Simulation Setup Database Interpolation Options
The database is empty.
Evaluate model for default parameter values

Create a symbol for your design

You need a symbol for your design in order to be able to instantiate it in a testbench schematic.

To create a symbol:

- 1. Right-click the cell representing your design.
- 2. Choose New Symbol.
- 3. Select the required options. For more information, see Generate a Symbol.
- 4. Click OK.



Instantiate your design into the testbench schematic

The symbol view of your design can be inserted into a testbench schematic by typing the component name, or by dragging the component from the ADS Main window into the testbench schematic.

SP_LNA_stage_1 [RF_Boar	rd_lib:SP_LNA_stage_1:schematic] * (Schematic):2	- • ×
<u>File Edit Select View</u>	Insert Options Tools Layout Simulate Window DynamicLink DesignGuide EM Help	
🗋 🗋 🚰 🔚 🚔 🕻	\; X 🤊 🤊 🗶 🏚 🍭 🗣 😓 🏦 🚘 🖨 🖡	💥 🐹 🗮
Lumped-Components	🔹 p:LNA_stage_1:symbol 👻 🖓 🕂 🥶 🚺 👬 💦 🖍 🖍 👘 🖤 🆃 🌵 🌋 🚟 🕼	
Palette 🗗 🗙		
	S-PARAMETERS OPTIONS Board Design Kit Netlist File Includ	t Jej
1977. 1 L L_Model · · ·	S_Param Options SP1 Options1 Include Start=100 MHz Temp=16.85 Include	
r→ ⊢ r→ ⊢ C C_Model	Stop=4 GHz	
DCFeed DCBlok		
-стр- РЕС РВС	V_DC +	
PRL PRLC		
slc src		· · · · · ·
SRL SRLC T		
RF_Board_lib:LNA_stage_1:sy	ymbol: Enter component location 0 items ads_device:drawing 1.250, 0.000 -0.500, 0	0.125 in 🦽

Configure the testbench schematic to use the Cosimulation View

There are two equivalent ways to use a Cosimulation View:

- Either the emCosim view of your design is used as View For Simulation for the instance in the testbench.
- Or an EM Cosimulation component is placed on the schematic view of your design to integrate the emCosim view with the schematic. This schematic view is used as View For Simulation for the instance in the testbench.

The simulation results are identical in both cases. The important distinction is the view used to netlist circuit simulated instances:

- When using the emCosim view as View for Simulation, the circuit simulated instances are netlisted from the partitioned layout view.
- When using an EM Cosimulation controller, the circuit simulated instances are netlisted from the schematic view.

This distinction is relevant for tuning and optimization, where the view that owns the circuit simulated instances is used for selecting and updating parameters.

Using the Cosimulation View as View For Simulation

This approach does not require the presence of a synchronized schematic. Even if one is present, it is not used by the simulation.

The circuit simulated instances are netlisted from the partitioned layout view, and can be tuned and optimized from there.

- 1. Select a symbol for your design instance in the testbench schematic.
- 2. Click the Choose View for Simulation button (📅).
- 3. Select the emCosim view in the Choose View for Simulation window.

Choose View for Simulation	×
🔿 Latatha Uisasachu Dalisu datasasina uduktu sisuu ta usa	
Cet the Hierarchy Policy determine which view to use	
Ose this view for simulation:	_
emCosim	
🔁 layout	
schematic	
	_
OK Cancel Help	

Alternatively, you can have the Hierarchy Policy select the correct simulation view for you. For more information, refer to Dynamic Model Selection.

NOTE The Standard Hierarchy Policy will always select the placed layout view of circuit simulated layout instances as View For Simulation. To avoid this, ADS comes with a Hierarchy Policy Standard_em amounting to emCosim emModel schematic. When using this Hierarchy Policy, ADS will never autoselect a layout view as View For Simulation for the circuit simulator.

Using an EM Cosimulation controller

This approach requires the presence of a synchronized schematic. The circuit simulated instances are netlisted from the schematic view of your design, and can be tuned and optimized from there.

- 1. Keep using schematic as a view for simulation of your design instance in the testbench schematic.
- 2. In the synchronous schematic of your design, select Simulate > Insert EM Cosimulation Controller to add an EM Cosimulation controller.



The EM Cosimulation controller integrates the EM partition of the emCosim view into the schematic. The netlisted schematic will instantiate the EM model of the EM partition to replace those original schematic instances and nets that are now EM simulated.

NOTE Nets in the schematic can have a spatial extent in the layout, where several EM model pins connect to a single original net. The netlister will split and rename such nets to multiple different nets to avoid shorting a pin of a circuit simulated instance to an unrelated distant EM model pin on the same original net.

Global nets, such as the analog ground, are an exception. They are never split or renamed, in order to guarantee global connectivity. EM model pins on the same global net will always be shorted.

Run the EM Circuit Cosimulation

You can now run simulation from the schematic testbench selecting Simulate from the Simulate menu.

The Data Display window appears, which shows the result of the EM cosimulation:



See Also EM Circuit Cosimulation Tutorial- Using the EM Cosimulation View Manual Versus Automatic EM Circuit Partitioning

EM Cosimulation Controller Component

EM Cosimulation Controller Component

The EM Cosimulation controller component integrates the EM simulated partition of an EM Cosimulation View into a schematic of the same cell. The netlisted schematic will instantiate the EM model of the EM partition to replace the original schematic instances and nets that are EM simulated. For more information, refer to Performing an EM Circuit Cosimulation with an EM Cosimulation View.

Inserting an EM Cosimulation Controller

To insert an EM Cosimulation controller:

- 1. Open a Schematic window.
- 2. Select Simulate > Insert EM Cosimulation Controller.
- 3. Place the EM Cosimulation component in the schematic.



Alternatively, you can access the EM Cosimulation component from the Components pallet:

- 1. Select Simulation S-Param from the components drop-down list in a Schematic.
- 2. Select the EM Cosimulation component from the components pallet and place it in the schematic.

Using the EM Cosimulation Controller Component

You can use the EM Cosimulation Controller component to switch between multiple EM Cosimulation Views. Double-click the instance to open the EM Cosimulation window. The EM Cosimulation controller window displays all the EM cosimulation views. You can select the required EM Cosimulation view and click Open View to integrate with the schematic:

EM Cosimulation:2
Available EM Cosimulation Views:
emCosim
Selected View:
emCosim
Open View
OK Apply Cancel Help

NOTE

Alternatively, you can click the **Push into Hierarchy** icon to open the EM Cosimulation View.

See Also

EM Circuit Cosimulation Tutorial- Using the EM Cosimulation View Manual Versus Automatic EM Circuit Partitioning

Using the EM Model View

Using an EM Model View

- The EM Model View
- Creating and Using an EM Model View
- Performing an EM Circuit Cosimulation with an EM Model View

Creating and Using an EM Model View

Creating and Using an EM Model View

An EM Model view 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.

Creating an EM Model

To create an EM model:

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select Model in the left pane of the EM Setup window.
- **3.** Select Auto-create EM model when simulation is launched to create an EM model after the simulation process.
- **4.** To generate an EM model immediately, click Auto-create Now. The following figure displays an EM model:
| 🜃 Coupled_Stubs_tune_lib:CoupledStubs:emModel (EM M 🔳 🗖 🔀 |
|--|
| File Edit Tools Help |
| 📁 🔚 🍠 🥙 🖪 🗇 🌲 |
| Simulation Setup Database Interpolation Options |
| Summary Simulator: Momentum simulation in microwave mode Layout: Workspace: C: Usersidefault/sample/multi_band_wrk/model/Coupled_Stubs_tune_wrk Library: Coupled_Stubs_tune_lib Cell: CoupledStubs View: layout Substrate: Substrate: CoupledStubs Ports: 2 ports defined Frequency plan: Adaptive from 0 GHz to 30 GHz (max:200) Output plan: Dataset: CoupledStubs Template: S_Nport_Slider.ddt Simulation options: Converted options from the DSN version of ADS Simulation resources: Queued simulation on host: Local:queue EM Model: emModel Auto-create symbol: no No symbol name specified |
| Edit |
| |

E If this button is labeled **Update Now**, an 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 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.

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 Nowin 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 does not exist).

Using an EM Model for circuit simulation.

Using an EM Model View for Circuit Simulation

Default View used by a Circuit Simulation

By default, the view named *schematic* 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.

To find the view that is used for an instance during circuit simulation, open the toplevel schematic and choose Simulate > Hierarchy Editor. In the Hierarchy Editor dialog box, find the instance name in the Instance column. Then search in the View for Simulation column to find the view that is 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 View 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 uses the view named *emModel* for this instance.

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 Editor (choose Simulate > Hierarchy Editor 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 Editor 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.

NOTE In ADS 2009 Update 1 and earlier, a Layout Component is used instead of an EM Model. For more information, refer Layout Components and EM Models.

Performing an EM Circuit Cosimulation with an EM Model View

Performing an EM Circuit Cosimulation with an EM Model View

The Electromagnetic-Circuit cosimulation feature enables you to run a circuit simulation with EM Model generation. Insert a cell that has an EM Model view in a testbench schematic. During the circuit simulation, the circuit simulator reuses S-parameter model data from the EM Model. In the absence of available data, the EM simulator is invoked to extend the EM Model with the missing data.

Setting up a Layout

There are two basic ways to create a layout. Use the one that best complements your own design methodology and suits your design needs:

- Layout synchronized from a schematic
- Directly drawn or imported layout

Defining the EM Simulation Setup

Once you have a layout, you need to create an EM Setup view that specifies the EM simulation setup:

- 1. Specify the substrate
- 2. Define the ports
- 3. Set up the simulation options, such as mesh parameters.
- 4. Specify a frequency plan for the EM model generation

Ports Width

Specifying correct port width is an important factor during EM cosimulation. For example, if you want to connect a strip line of 0.1 mm to a layout design of width 10 mm, you need to create a small polyline of width 0.1 mm.



In this example, you can create a polyline at points where you want to connect another strip line and ports.(Edge ports can also be used in these scenarios).



Defining Component Parameters

You can define component parameters that parameterize the layout. For more information, see Defining Component Parameters.

Creating an EM Model and Symbol

A symbol is used to represent the component in a schematic. Symbols are stored in Symbol views and must be created before they can be used. For more information on symbols, see Generate a Symbol.

An EM Model is used by the circuit simulator to model a component by using the Sparameters 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 circuit simulation performance. For more information about how to use an EM model, see Creating and Using an EM Model View.

You can generate an EM model and symbol by selecting EM > Component > Create EM Model and Symbol in the layout window. You can also use the EM Setup window to generate an EM model and symbol. The EM Setup window provides options to customize the model and symbol generation. For more information, see Generating an EM Model.

In ADS 2009 Update 1 and earlier, a layout component was modeled using a schematic, EM-based S-parameters, and File-based S-parameters. From ADS 2011 release onwards, an EM Model can be inserted into any schematic as a layout component. 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. From ADS 2011 onwards, you can access 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.

Using Components in a Schematic

The Symbol view of the cell can be inserted into a schematic by typing the name of the component in the component name entry field, or by dragging the component from the ADS Main window in the Schematic window.

Running an EM Circuit Cosimulation

According to the standard simulation hierachy policy, when a cell instance has an EM Model view named "emModel", the circuit view is used for simulation. You can override the view choosen for simulation by selecting the component symbol and click the Choose View for Simulation button. Select the view you want from the Choose View for Simulation window. For more information, see Creating and Using an EM Model View.

Example

This section provides an exercise illustrating how to set up, perform and view the results of EM Circuit Cosimulation.

Opening an Example Project

Start by opening an ADS Momentum example project into your local directory.

- 1. From the Main window, select File > Example > Momentum.
- 2. Select Microwave and select Coupled_line_filter_wrk and click Open.
- **3.** Specify the workspace name and location for creating the example workspace.
- 4. Click OK. This opens the example and saves it to the new location.

Editing the Component

To edit a component:

- 1. Close the README window.
- 2. Select the Coupled_line_filter layout window.
- 3. From the Layout window, select EM > Clear Momentum Mesh.
- 4. Select EM > Component > Parameters to open the Design Parameters dialog box.
- 5. In the Create/Edit Parameter section, type L1 as the parameter name.
- 6. Select Subnetwork as the parameter type.
- 7. Set the default value and unit to 300 um.
- 8. Click Add.

🔁 Design Pa	rameters:2	
Design Name	Coupled_line_filter_lib:Coupled_line_filter:layou	t
-Select Param	eter	Create/Edit Parameter
L1		Name L1
		Type Subnetwork
		Default Value 300 um V Add the parameter and select a component in the layout to set the parameter value.
Add	Cut Paste Update]
Ok	Apply	Cancel Help

- 9. Click OK.
- **10.** In the Layout window, double-click the component to select it.



- 11. Select Parameter L in the MLIN column and set Line Length L to L1 in Length
- 12. Click OK.

.

Creating an EM Model

To create an EM model:

- 1. Choose EM > Simulation Setup to open the EM Setup window.
- 2. Select Model in the left pane of the EM Setup window.
- 3. Click Create Now.

Adding and Using the Component in a Schematic

To add the parameterized component in schematic:

- 1. Click New Schematic Window from the ADS Main window to open a new Schematic window.
- 2. Specify the cell name.
- **3.** Select ads_templates_s_params from the Schematic Design Templates dropdown list.
- 4. Click OK.
- 5. Type the cell name and press Enter to place the EM model component.
- 6. Click End Command.
- 7. Click Insert Wire to connect the symbol with the terminals.
- 8. Click End Command.

Performing EM Circuit Cosimulation

To perform EM circuit cosimulation:

- 1. Select the EM Model.
- 2. Click Choose View for Simulation (🔛

🖁 Cho	oose View for Simulation	×
⊙ Le ○ Us	t the Hierarchy Policy determine which view to use this view for simulation:	
	emModel layout schematic	
	OK Cancel Help	

- 3. Choose EmModel view.
- 4. Click OK.
- 5. Click Simulate to perform circuit Cosimulation.

You can now view the results of EM circuit cosimulation in the Data Display window.

Show me How to Perform EM Circuit Cosimulation

Video: Perform EM Circuit Cosimulation

The EM Model View

The EM Model View

An EM Model view is used by the 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. The EM Model caches the S-parameters to improve the circuit simulation performance.

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

Simulation Setup

This tab provides a summary of the EM simulation setup: 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 S-parameter data cached in the EM Model will be deleted.

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.

DF_9	Stub_LC [[DoubleFold	led_Stub	p_Filt	er_lib:DF_S	t 🕑 🔿	\otimes			
<u>F</u> ile	<u>E</u> dit <u>T</u> ool	s <u>H</u> elp			_					
	H 9	61 2	<u>i</u>	2						
Sir	nulation S	etup Dat	abase	Inter	rpolation	Options				
A	vailable da	ata								
		(Cell Parar	meter	s					
	Name	Ll	L2	2	S					
	data.000	94.8524 m	il 89.933	9 mil	5.9091 mil					
	,									
_	Delete	Delete Al								

Cell Parameter Interpolation

The settings on the Interpolation tab are used by the circuit simulator to determine whether for a given set of cell parameter values, model data is available, can be constructed by interpolation, or if a new EM simulation is required. In the absence of a unit name, MKS units are assumed. The search for available model data takes into account the Resolution value for each cell parameter. When new data is required, the cell parameter values for the new model generation is rounded to the specified Resolution.

You can enable interpolation for cell parameters. This interpolation setting only applies to the cell parameters, not to the frequency! When enabled, a linear interpolation scheme is used. The circuit simulator invokes an EM simulation only if the requested model sample cannot be obtained by interpolation between neighbor model samples. Neighbor model samples must be within a normalized distance (L1 measure) smaller than one from the requested sample. The normalization of the distance is with respect to the Delta values specified for each cell parameter. Extrapolation is never performed in the layout parameter space. An EM simulation is performed to calculate a new sample in such case.

DF	Stub_LC [Doul	bleFolded_Stul	b_Filter_lib:DF_	st 🕑 🔿	\otimes
	<u>Eait 1</u> 00is <u>H</u>	eip 📰 🏪 🍘			
			-₩- 📃		
S	imulation Setup	Database	Interpolation	Options	
	🗸 Use interpola	ition			
	Cell Parameter	:erpolation Del	Resolution		
	Ll	2.95276 mil	0.0001 mil		
	L2	2.95276 mil	0.0001 mil		
	S	2.95276 mil	0.0001 mil		

Options

Frequency Domain Simulation Options

The EM Model does not use datasets. The EM Model reads the .cti file that contains the simulated frequency samples and (optionally) a .rat file that contains the coefficients of the rational polynomial fit in case an adaptive frequency sweep was used. When the circuit simulator needs S-parameters at a specific frequency, the . cti file is checked whether data is available for that frequency first. If no data is present and the requested sample is within the adaptive range(s), the rational polynomial fit is evaluated. If that is not the case, a linear interpolation is tried using the .cti file samples. If that is not possible, a linear extrapolation with passivity enforcement is done. In case of a non-adaptive sweep, the frequency domain interpolation is always linear using the samples that were simulated from the .cti file. Within an adaptive range, the rational polynomial fit is evaluated. There is no control over the interpolation.

You can specify the following frequency domain simulation options:

Frequency options	
Extrapolation mode:	Linear 🔹
Enforce passivity:	Only when extrapolating

Extrapolation mode: 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. Sparameters at kept constant during extrapolation towards low frequencies. Towards high frequencies, S-parameters are e-damped in magnitude and linearly extrapolated versus phase.

Enforce passivity: 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.

Netlist Options

Multi-pins port terminal netlist options	
O Use shorts in netlist	
Use transformers in netlist	

The Multi-pins port terminal setting is only relevant when All pins is selected and the S-parameter ports were defined having multiple layout pins connected to either the plus or minus terminal, or both. The All pins selection implies that the symbol consists of similar number of pins as present in the layout.

- Use shorts in netlist: convenience setting that shorts all pins connected to the S-parameter port terminal in the netlist fragment for the EM Model. It is sufficient to connect/wire to one symbol pin only.
- Use transformers in netlist: a transformer is used in the netlist fragment for the EM Model to extract the common mode voltage and current from the multiple pins. This option is required when the pins cannot be shorted automatically.

Tools Menu

The Tools menu in the EM Model provides following capabilities:

- Open Layout Window opens the corresponding layout cell view in a Layout window. For a cell with parameters, this is the 'nominal' layout view with default parameter values.
- Open Substrate Window opens the substrate used in the simulation in a Substrate Editor window.
- Create Dataset allows you to create a dataset for the selected data item in the EM Model database. The name of the dataset is "<cellName>_<emModelName>_<index>" where by <index> represents the

index of the selected data item and the file is written in the data directory.

- M Data Display opens a Data Display window after writing the dataset for the selected item in the EM Model database.
- Far Field allows you to compute the far field for the selected data item in the EM Model database.
- UV Visualization opens the 3D Visualization window for the selected data item in the EM Model database.
- 🗐 Simulation Log opens the simulation log of the selected database items.

Far Field and/or Visualization driven from an EM Model is not possible when the EM Model was generated with the 'Include S parameter data only' flag enabled in the corresponding EM Setup. The following warning window is displayed:



Disabling the flag ensures that all data required for post-processing operations are stored with the EM Model. Changing the flag in the EM Setup and updating the EM Model does not invalidate existing EM Model data. The flag will only take effect for newly generated data samples. If you want to do post-processing operations on existing EM Model data items, delete and regenerate them.

NOTE

Far-field computation and 3D Visualization driven from the EM Model database is only possible for fixed parameter value EM Model data items, not for the scalable model data items that the Advanced Model Composer generates. The scalable model data can be recognized by the LIST or RANGE parameter values. Data for post-processing cannot be stored in the EM Model database because such EM Model data item spans a (discrete or continuous) range of instances of a parametric layout.

Using the EM Circuit Excitation AEL Addon

Using the EM Circuit Excitation AEL Addon

EM Circuit Excitation allows you to visualize field data such as surface currents and far-field of your EM Model/EM Cosim with any circuit elements contribution in your schematic. Key features are:

- DC, AC, Harmonic Balance, Transient and Envelope simulation can be used. (Support of Envelope simulation is new since ADS2016.01 release)
- Both Momentum and FEM EM Model/EM Cosim are supported.
- EMPro Component is supported.
- Parameterized EM Models/EM Cosims are supported.

By running above supported circuit simulation for a circuit which includes components with EM Model/EM Cosim views, EM Circuit Excitation can get and store actual simulated voltages at all pins of those components. Those voltages will be used as excitation for EM visualization. (Note: For Transient simulation, time domain signals will be converted to frequency domain before used in EM visualization.)

Enabling the EM Circuit Excitation AEL Addon

To enable the EM Circuit Excitation AEL Addon:

1. Select Tools > App Manager in the ADS Main window.

2. Select the Enabled check box for EM Circuit Excitation, as shown in the following figure:

Name	Enabled	Addon File location
 ADS Installation Addons 		
Layout Command Line Editor		\$HPEESOF_DIR\layout_command_line_editor\ael\boot.atf
PCB Library Import Tools - Mentor		\$HPEESOF_DIR\Ims\LMSStartup.atf
PCB Library Import Tools - Cadence		\$HPEESOF_DIR\Ims\LMSCDNSStartup.atf
PCB Library Import Tools - All Vendors		\$HPEESOF_DIR\Ims\LMSStartupAll.atf
PCB Library User Tools - All Vendors		\$HPEESOF_DIR\Ims\LMSUserStartup.atf
EM Circuit Excitation	1	\$HPEESOF_DIR\ael_addons\em_excitation\ael\MomExc_boot.atf
Via Drawing Utility		<pre>\$HPEESOF_DIR\ael_addons\Via_Utility_Add_on\ael\boot.atf</pre>
User Addons		
Add User ADS AEL Addon Remove User ADS A	EL Addon	
		Close Help

- 3. Click Close.
- 4. Restart ADS.
- 5. Open a Schematic window and select EM > Circuit Excitation.

Circuit Excitation:1	x
Compute Excitation	
1 instance(s) with EM Model/EM Cosim found	Run DC/AC/HB/Tran/Envelope Simulation
	Optimization
Visualization	
Instance (cell:view)	
Analysis	
Baramatar Sat	· · · · · · · · · · · · · · · · · · ·
	· · · · · · · · · · · · · · · · · · ·
Far Field for MomMW or FEM (ignored for Mo	mRF)
Interpolate Node Voltages	
	Frequencies
	Start Visualization
	Clear Results]
Close	Help

Before using this addon, ensure that you have selected either All generated frequencies or User specified frequencies in Output Plan options:



EM Circuit Excitation Flow

To use the EM Circuit Excitation AEL Addon, you need to perform the following steps:

- 1. Create a EM Model or EM Cosimulation view.
- 2. Create a circuit with supported simulation controller.
- 3. Run the DC/AC/HB/Tran/Envelope simulation.
- 4. View surface currents.
- 5. View Far Field.

To use EM Circuit Excitation, perform the following steps:

- Select EM > Circuit Excitation in the Schematic window. The Circuit Excitation dialog box is displayed.
- 2. Click Run DC/AC/HB/Tran/Envelope Simulation to start circuit simulation and post processing.

NOTE The Open Visualization button is enabled .

- **3.** Select the Far Field for MomMW or FEM checkbox and select 2.4GH frequency from the Frequency drop-down menu.
- 4. Select the beam you want to see in visualization.
- 5. In Visualization view, set the phi and theta values and click Display Cut in Data Display.

The following sections describe these steps.

Creating an EM Model or EM Cosimulation view

You can create an EM Model or an EM Cosimulation view from an EM setup window. For more information, refer Generating an EM Model and EM Cosimulation View.

Before creating the EM model or EM cosimulation view, uncheck the Include S-Parameter data only check box. Unchecking this check box includes not only the S-Parameter data, but also the required data for EM visualization. If you check this check box, circuit excitation visualization will not be possible.

-EM Mod	el	
Name:	emModel	
	end text to the name	
📃 Incl	ude S-Parameter data only	
Crea	ate EM Model when simulation is launched	Create Now

NOTE Only frequencies calculated by the EM simulator can be shown in the EM visualization. If you want to see surface currents and far field at explicit frequencies, add such frequencies in your EM Setup frequency plan.

Creating Circuits for DC/AC/HB/Tran

EM Circuit Excitation works for circuits with single or multiple EM Models and single or multiple DC/AC/Harmonic Balance/Transient/Envelope simulation controller. Parameter sweep is also available. The following figure displays an example of a circuit with AC simulation controller:

🗟 cel	II_2 [M	y Lib	rary	/2_li	b:c	ell_2	sch	hen	nati	c]	(So	che	m	atio	:):6												-			×
File	Edit	Selec	t ۱	View	ŀ	nsert	Op	ptior	ns	To	ols	L	ayo	out	Si	mul	late	1	₩in	dow	(Dyn	ami	cLir	ж	Μ	ome	ntu	m	»
	6				5	+	ŀ	QQ.	X	1	9	(2	1	2	+	i -	e	D.	Ç	9	Q	¢	2	¢		Ş	1	Ì.	»
Source	es-Freq	Dom	ain		1	✓ R						~	0	>•	닅	-	0110 VAF		İ	 R-	17	\		(AMI		Ð	1	٢		»
Palette		8]: :			1.1			1.1		1	1		1			1	1	1		÷	1		1	1		1	1		^
		7^	: :			: :	: :	-			;	1						÷			÷	Þ					÷			
		ĭ								ł	ł							+	-		÷.		÷	÷			÷		· ·	
V flone		ו					R R2 R=	10h	 m		ł		•	eliji mMc_0	del		ł	ļ			ł			ļ	ş	R R1 R=5			• •	
		<u>j</u> _		ę	SR Vac	C1 = polar q= freq	(1.D)	v.			Ĵ										÷			÷	Ì					
Vellare		ี่ไ		Ŧ				Į	40 AC	AÇ	ļ			:			:		•		÷.		;	÷	Ē		;		• •	
		, ן						-	AC1 Stat Stop Step	t=0.0 =3.0 =0.0	GH2 0 GH	lz Hz										· ·		:			;		· ·	
			<			::					:						;												>	~
Select	Enter t	he st	artin	e poi	0 i	tems				a	ds_	devi	ice:	drav	vine	1	.750	. 1.	000			5.	625	. 5.2	250			in		

For creating circuits for DC/AC/HB/Tran/Envelope, make sure an EM Model or EM Cosim view is chosen for the required instances. Only such instances will be available for EM visualization. The excitation voltages for each instance for EM visualization is interpolated or extrapolated based on available frequencies of the AC/Harmonic Balance simulation.

NOTE Currently DC, AC, Harmonic balance, Transient and Envelope simulation are supported.

Running DC/AC/HB/Transient/Envelope Simulation

NOTE Note that EM Circuit Excitation can only visualize data at frequencies where EM simulator calculates its solutions. If frequencies that you want to visualize are not calculated by EM simulators yet, make sure to setup EM frequencies to cover those frequencies before using EM Circuit Excitation.

1. Select EM > Circuit Excitation in the schematic window. The Circuit Excitation dialog box is displayed, as shown in the following figure:

Circuit Excitation:1	X	
Compute Excitation		
1 instance(s) with EM Model/EM Cosim found	Run DC/AC/HB/Tran/Envelope Simulation	
Visualization		٦l
Instance (celliview)	*	
Analysis Act AC	•	
Parameter Set	•	
Far Field for MomMW or FEM (ignored for Mo Figure 1 and the second se	mRF)	
	Frequencies Start Visualization	
	Clear Results	
Close	Help	

- 2. Select Far Field for MomMW or FEM option if you want to see Far Field results.
- **3.** Select Interpolate Node Voltages if you want to interpolate circuit simulated voltages to get voltages at EM frequencies. (This option is only for frequency domain circuit simulations)
 - **NOTE** Interpolate Node Voltages option should be avoided to use unless you are really sure use of interpolated or extrapolated voltages as excitation is OK. Instead of using this option, consider to adjust both circuit simulation frequencies and EM simulation frequencies to match frequencies you're interested in.

4. Select Time domain to Frequency domain conversion method.(This option is only for time domain circuit simulations)

Time To Freq Conversion Settings	
O Use fs()	Use DFT at each EM freqs
Advanced Options	
Note: All text entries are optional. Defau	ult value will be used for blanks.
Start Time	Stop Time
psec 🔻	psec 🔻
Num of Freq Samples	
Transformation Method	
Chirp-z	
Interpolation Order	
1	▼
Window Type None -	Window Constant

- a. Select Use fs() if you want to use fs() functionality to convert to Frequency domain. This will allow you to specify very detailed options. See here for more fs function details. Note that depending on optional settings frequencies you get will be different. It's recommended to check output frequencies first by clicking Frequencies... button. Interpolation will be performed to get excitation voltage at EM frequencies.
- **b.** Select Use DFT at each EM freqs if you want to apply DFT algorithm at each of EM frequencies.(recommended)
- 5. Click Run DC/AC/HB/Tran/Envelope 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 might not work properly from this tool. To avoid this issue, you can disable this option. In the schematic, doubleclick Optimization controller, select the **Parameters** tab, and remove selection from **Enable Optimization Cockpit**.
- 6. 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 list.
- 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:



See Also Designing Smart Antenna

Manual versus Automatic EM/Circuit Partitioning

The Automatic EM/Circuit Partitioning feature facilitates combining EM simulation of a layout with circuit simulation of instances that cannot be simulated by the EM simulator (e.g. a nonlinear device) or for which a model already exists (e.g. a builtin primitive, a schematic, or an EM Model). The automatic partitioning obsoletes the manual process of stripping off these instances, extracting an EM Model and reintegrating it with the stripped instances in a schematic.

The automatic partitioning and how it compares versus the manual approach are described using an example. Open the following example:

- 1. From the ADS Main Window, select File > Open > Example.
- 2. Browse to the Momentum/emcktcosim directory and select the RF_Board_wrk.7zads archive file.
- 3. Click Open.
- 4. Open the LNA_stage_1:layout view. Make all layers visible.



The manual approach in ADS 2013.06 or before



Consider a layout having 11 surface mounted devices (SMDs). Central to the layout, there is the Board_Design_Kit:HBFP-0420 transistor component. The input and output matching and the power supply bias make use of various R, L and C surface mounted devices provided by the Board_Design_Kit library.

The SP_LNA_stage_1:schematic contains the circuit simulation setup for an S-Parameter simulation for this first stage. First select the schematic as view for simulation. This schematic view uses ideal wires to interconnect the SMD devices. Run the circuit simulation to see the amplifier gain and its input and output match in the SP_LNA_stage_1 data display.

The refined analysis of the amplifier design, using an EM-based accurate model for the passive interconnect, traditionally required a very significant amount of manual steps. This is because the layout contains instances that cannot be simulated by the EM simulator: the nonlinear transistor device and the Rs, Ls and Cs for which a model is provided in the Board_Design_Kit library. Previously, you had to manually separate the items that must be simulated by EM from the items that have a circuit simulation model already.

The traditional approach was as follows.

- 1. The LNA_stage_1 cell was copied to a new LNA_stage_1_em cell.
- 2. In the layout view of the new cell, pins were inserted at all SMD instance pin locations.
- 3. The SMD devices were deleted or flattened.
- 4. When applicable, edge and/or area pins had to be reconstructed. E.g. the pads of the transistor are defined as area pins.

- 5. In the EM Setup, the ports for the EM simulation had to be configured. E.g. the R, L and C devices have one differential port defined.
- 6. An EM Model and new (layout lookalike) Symbol view were created from the EM Setup.
- 7. The LNA_stage_1 cell was copied to a new LNA_stage_1_reconnection cell.
- 8. In the schematic view of the new cell, all wires are deleted. The (layout lookalike) symbol of the LNA_stage_1_em cell is inserted and all SMD devices are manually reconnected. Optionally, the layout view was synchronized with the schematic and a new layout lookalike symbol was created.
- 9. The SP_LNA_stage_1 testbench cell was copied to a new SP_LNA_stage_1_em cell.
- **10.** In the schematic view of the testbench, the LNA_stage_1 instance is deleted and replaced by an instance of LNA_stage_1_reconnection

Only after all these steps, you could run the refined analysis from the SP_LNA_stage_1_em:schematic to see the impact of the actual physical interconnect.

The automatic approach in ADS 2014.01

Starting point is the LNA_stage_1 cell. Open the LNA_stage_1:emSetup view.



At the top the setup type has been set to EM Cosimulation. When clicking Go this EM Setup does not immediately start an EM simulation. Instead it breaks up the design in an EM simulated partition with circuit simulated instances. For the EM simulated partition, the EM Setup generates a layout and traditional EM model in a new cell RF_Board_lib:LNA_stage_1_emCosim. The EM Setup also generates a EM Cosimulation view RF_Board_lib: LNA_stage_1:emCosim, which can be used as regular netlistable circuit model for RF_Board_lib:LNA_stage_1. The netlisted EM Cosimulation view integrates the circuit simulated instances with the EM model.

Select the Partitioning tab. You will see a table with the configuration for instances found in the layout hierarchy.

EN RF_Board_lib:LNA_stage 1:emSetup (EM Setup for cosimulation)							
Hie tools View Help							
日 フ て 日							
Image: A set of the set of t							
Layout							
B Partitioning	Policy: Enter spi	ace separated list	to override stand	ard policy: preferred	l em circuit		
* Ports	S 🕇						
Frequency plan Output plan	Parent Library	Parent Cell	Parent View	Instance Name	Instance Lib:Cell	Partitioning Rule	Modelling Action
E Options	RF_Board_lib	LNA_stage_1	layout	C1	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
Resources	RF_Board_lib	LNA_stage_1	layout	C2	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
Cosimulation	RF_Board_lib	LNA_stage_1	layout	C3	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
I NOLES	RF_Board_lib	LNA_stage_1	layout	C4	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	C5	Board_Design_Kit:C	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	L1	Board_Design_Kit:L	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	L2	Board_Design_Kit:L	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	Q1	Board_Design_Kit:HBFP-0420	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R1	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R2	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
	RF_Board_lib	LNA_stage_1	layout	R3	Board_Design_Kit:R	circuit	🔛 circuit using ports from layout
Rule text style: Defined by policy - <u>Instance override</u> - Defined here							
	ine setup impose	s an Em Cosimulat	ION				
						Generat	e: Views 🔹 Go

The Partitioning table shows which instances will be EM simulated and which will be circuit simulated. Each library cell component has its own set of Partitioning Rules, where each rule declares a supported way of simulating that component. Unless configured otherwise, each component automatically supports the following two rules:

Rule Name	Action
circuit	Do not descend into the instance. The circuit simulation hierarchy policy is used to resolve the model that is used for simulation. This rule requires that the component can be simulated using circuit simulation.
em	Descend into the instance. Model all primitive layout objects (traces, polygons, etc.) by EM and re- examine the cell instances.
n this	example, all Board_Design_Kit library components have been

In this example, all Board_Design_Kit library components have been preconfigured to enforce the circuit partitioning rule. The em rule has been deleted, hence ADS will refuse EM-only simulation of these components. If multiple partitioning rules exist for a component, a policy is used to autoselect a rule unless you specifically select a rule in the dialog to override the automatic choice. For more information on partitioning rules, refer to Automatic EM Circuit Partitioning.

In this example, the first row of the table can be interpreted as follows: when partitioning RF_Board_lib:LNA_stage_1:layout for EM Cosimulation, the circuit partitioning rule is being applied to the Board_Design_Kit:C instance C1. The layout of the instance will be replaced by EM simulated pins and ports copied from Board_Design_Kit:C:layout, connected to a circuit model for Board_Design_Kit:C. This model is selected using the regular circuit simulation rules: either configured as primitive (applicable here), or using the view for simulation, or using the circuit simulation hierarchy policy.

Besides the partitioning, the other tabs in the EM Setup window allow you to specify substrate, ports, frequency plan, simulation options and resources. This is not different from a normal 'EM only' simulation.

An EM Setup with a partitioning that contains circuit-simulated components will create a EM Cosimulation view as output. You can specify the view name on the Cosimulation tab. This view can be used as regular netlistable circuit model for RF_Board_lib:LNA_stage_1. When netlisted, it integrates the circuit simulated instances with the EM model for the EM partition.

EM RF_Board_lib:LNA_stage_1:emSetup (EM Setup for cosimulation)				
<u>File T</u> ools <u>V</u> iew <u>H</u> elp				
🔚 📂 🤊 🥙 🖪				
 Mom RF Cosim Layout Partitioning Substrate Ports Frequency plan Output plan Options Resources Cosimulation Notes 	EM Cosimulation View name: emCosim Cell representing the EM partition Library: Library: RF_Board_lib Cell: LNA_stage_1_emCosim EM Model: emModel Mode of operation Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Image: Use an intermediate layout view (not parameterized) Ima			
Generate: Views 💌 Go				

On that tab, you can also specify the mode of operation for the cell representing the EM partition. By default the EM partition will be represented by a dedicated layout cellview.

From the EM Setup Window, click Go to generate the EM Cosimulation view. Besides the view, the cell representing the EM partition, LNA_stage1_emCosim, will be created as well. The EM simulation itself will not be launched by this action. If you prefer to run the EM model extraction upfront, you can do so from the EM Model view of the LNA_stage1_emCosim cell.

An EM Cosimulation view with a proper partitioning is all that is required. You can run the refined analysis from the SP_LNA_stage_1:schematic by selecting the emCosim view as 'View for Simulation' for the LNA_Stage_1 instance. The EM Cosimulation view contains all information to netlist the circuit instances together with the EM Model view of the cell represting the EM partition.

SP_LNA_stage_1 [RF_Board_lib:SP_LNA_stage_1:schematic] (Schematic):4					
File Edit Select View Ins	ert Options Tools Layout Simulate Window DynamicLink DesignGuide Help				
🗋 🗋 🔚 🚔 🗞	⊡ []] 🗡 🔊 🥙 🖄 🖶 🖲 🥺 🖗 🧶 👘 📩 🚔 👘 😹	🗙 💐			
Lumped-Components 👻	- 🕞 🕂 🖓 🛄 rên 🔪 🗫 📳 🌰 🕂 🛣 🛵				
Palette 🗗 🗙 👘 👘					
	S-PARAMETERS. Board Design Kit Netjist File Include	· · · · ·			
۹۳۲۰ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲	S_Param Options Include Include Include				
	Stop=4 GHz				
DCFeed DCBlock	LNA_stage_1 emCosim	 			
		· · · · · ·			
-tjp- PLC PRC	SRC1 Term SRC1 C=100 pF Term1 Vdc=5 V-T SNum=1				
PRL PRLC	Image: second				
sLC SRC					
SRL SRL		+			
Select: Enter the starting point	0 items ads_device:drawing 0.500, -1.500	in _{at}			

Alternatively, you can integrate the EM Cosimulation with the existing LNA_stage_1:schematic schematic.

In this alternative approach

- Keep using schematic as view for simulation of the LNA_stage_1 instance in SP_LNA_stage_1:schematic.
- Use Simulate > Insert EM Cosimulation Controller... to add an EM Cosimulation controller to LNA_stage_1:schematic.

The EM Cosimulation controller integrates the EM partition of the emCosim view into the schematic. The netlisted schematic will instantiate the EM model of the EM partition to replace those original schematic instances and nets that are now EM simulated.



The simulation results are identical regardless whether you directly use LNA_stage_1:emCosim for simulation or, alternatively, continue to simulate LNA_stage_1:schematic with an EM Cosimulation controller.

The important distinction is the view used to netlist circuit simulated instances:

- When directly using LNA_stage_1:emCosim for simulation, the circuit simulated instances are netlisted from the partitioned layout LNA_stage_1:layout.
- When using LNA_stage_1:schematic with an EM Cosimulation controller, the circuit simulated instances are netlisted from the schematic LNA_stage_1:schematic.

This distinction is relevant for tuning and optimization, where the view that owns the circuit simulated instances is used for selecting and updating parameters.

See Also

Automatic EM Circuit Partitioning EM Circuit Cosimulation

EMPro and ADS Integration

EMPro and ADS Integration

Keysight EEsof provides a way to combine the capabilities of EMPro and ADS by using a common database. The following features are supported in EMPro:

- Import EMPro results and design data in ADS.
- Perform EM circuit co-simulation from the ADS Schematic window.
- Improve usability of the existing EMPro components by adding EM ports in ADS.

Contents

- EMPro and ADS Integration Process
- Saving EMPro Designs in a Library
- Adding an EMPro Library in ADS
- Adding EMPro Components in ADS Layout
- Using EMPro Components in ADS Schematic
 - Adding EMPro Components to Schematic
 - Exporting EMPro Simulation Results as emModel
- Exporting ADS Layouts to EMPro
- EMPro ADS Integration FAQs
- Application- SI Analysis of Interconnects using EMPro-ADS Interoperability
- Example- Using the EMPro Connector Design in ADS
- Using the EMPro FEM Engine in ADS

EMPro and ADS Integration Process

EMPro and ADS Integration Process

EMPro designs are stored in a cell, which contains multiple views. You can use these views in ADS layout and schematic.

EMPro and ADS Integration



EMPro and ADS Integration Process

The following flowchart depicts the ADS and EMPro integration process:





Creating Designs in EMPro

You can create a new design or modify an existing design in EMPro. For more details, see Creating Designs in EMPro.

Saving EMPro Designs in a Library

EMPro designs are saved as cells in a library. You can import this library in ADS. For more details, see Saving EMPro Designs in a Library.

Adding an EMPro Library in ADS

To use EMPro designs in ADS, add the EMPro design library in an ADS workspace. For more details, see Adding an EMPro Library in ADS.

Using EMPro Components in ADS Layout

From an EMPro library, you can use the footprint or layout view in the ADS layout. For more details, see

Adding EMPro Components in ADS Layout

Using EMPro Components in ADS Schematic

From an EMPro library, you can use the symbol or lookalike view in the ADS schematic. For more details, see Using EMPro Components in ADS Schematic.

Saving EMPro Designs in a Library

Saving EMPro Designs in a Library

From EMPro 2012 release onwards, your EMPro designs are saved in a library. After you have saved the design, a cell with multiple views is created and stored in a library. You can import this library in ADS.

NOTE Designs created with EMPro 2012 that are saved as cells and views in a library cannot be opened with older releases. However, from within EMPro 2012, you have the option to save the design in the legacy format (.ep). Designs created with previous releases can be opened in EMPro 2012 and can be saved in the new library cell/view format (recommended) or in the legacy format (.ep). When saved in the new library cell/view format, the legacy files on your system remain unchanged.

While saving an EMPro project, you can specify the following project types:

- EMPro OpenAccess projects: Enables you to save EMPro projects in a library, unless you explicitly specify a project name that ends with .ep. If the chosen destination is not within a library, a new library is created with the same name as the project.
- EMPro Legacy projects: Enables you to save EMPro projects in the legacy (. ep) format.

- All OpenAccess cells: Enables you to save projects in a library, unless you explicitly type a new project name that ends with .ep. If the chosen destination is not within a library, a new library is created with the same name as the project.
- All files: Displays all files. It enables you to save a project in a library, unless you explicitly type a new project name that ends with .ep. If the chosen destination is not a library, a new library will be created with the same name as the project.

Saving Designs in a Library

To save a design in a library:

- 1. Select File > Save Project As to open the Save Project As dialog box.
- 2. Choose the required location for saving your project.
- 3. Click Create New OpenAccess Library (🏬). A new library is created in the Save Project As dialog box.



4. Specify a name for your library, as shown in the following figure:



- 5. Double-click the new library.
- 6. Type a name for your project or accept the default name.
- 7. Accept the default project type, EMPro OpenAccess projects.



8. Click Save.

If new cells are added to the EMPro library, select View > Refresh (F5) to refresh the library view in the ADS Main window. If a particular cell is changed from within EMPro, you need to insert a new instance in the ADS design to capture the changes. For example, if the number of ports changed, you need to add the updated cell in ADS.

Show me How Do I Save EMPro Designs in a Library

youtu.be

Video: How Do I Save EMPro Projects as a Library

Saving Existing 3D Components in a Library

You can also save components that were created using older versions of EMPro in a library. These projects, which were created in the *.ep* format, can be easily saved in the new format. You need to open an existing project and save it as a library. While opening a project, you can select one of the following project types:

- All EMPro Projects (OA,.ep): Displays all EMPro projects that are created in the OpenAccess (OA) and .ep format.
- EMPro OpenAccess projects: Displays only OpenAccess cells that are EMPro projects.
- EMPro Legacy projects: Displays only EMPro projects that are created in the legacy format (.ep).
- All files: Displays all files.

To open an older version project:

1. Choose File > Open Existing Projects to display the Open Existing Project dialog box.

Look in:	C:\Users\jo\Documents	- 3 0 0 💼 🗉 🗉
My Comp	Bluetooth Exchange Folder Camtasia Studio Custom Production Presets 7.0 empro_comp_lib test14_final.ep	MyConnectFiles PGP Projects SMA_Johnson_PCB_Edge_Mount.ep
		••
Project name:		
Project type:	All EMPro projects (OA, *.ep)	•
No preview available		
		Open Cancel

- 2. Select the required project type.
- 3. Select the existing project you want to open.
- 4. Click Open.

After opening the project, you can save it in the new library format. Perform the following steps:

- 1. Open a legacy EMPro design.
- 2. Select File > Save Project As to open the Save Project As dialog box.
- 3. Choose the required location for saving your project.
- 4. Type a name for your project.
- 5. Select EMPro OpenAccess projects as the project type.
- 6. Click Save.

Creating Designs in EMPro

Creating Designs in EMPro

To create an EMPro design, perform the following tasks:

- 1. Create a Geometry.
- 2. Assign Materials to components.
- 3. Set Mesh Priority.
- 4. Define Ports.

5. Specify Simulation Setup.

Step 1: Create a Geometry

To create new objects or modify existing objects, use the Create Geometry toolbar.

Example: Creating a Parametrized Geometry

To create a parametrized geometry:

- 1. Click Extrude in the Geometry window.
- 2. Select the object you want to create.
- 3. Click the Extrude tab and enter the Extrude Distance.
- 4. Click Done. The geometry view displays the objects that you have created.
- 5. Click the Specify Orientation tab to specify the origin of the model.
- 6. Click and select the Origin tool.
- 7. Click the Edit Cross Section tab.
- 8. Click the Extrude tab.
- 9. Type package_length in the Extrude Distance text box. The edit box will turn red as soon as the focus is shifted away from the edit box or when you press Enter. This indicates that the value is interpreted as an expression containing an unknown parameter.
- **10.** Click Done. The geometry view will not show the top box as the modeling sequence contains the unknown parameter package_height.
- 11. Click Parameters (Parameters) to open the Parameters window.
- 12. Click 🛨 to add a new parameter.

f	×) Parameters – – ×				
	+ 1				
	Name 🗸	Formula	Value	Description	
	🌇 timestep	1.91791e-12	1.917908	Simulation timestep in seconds	
	Revert			Apply	
13. Add the parameter package_height and set Formula to 1 mm.

Show Me How to Create a Parameterized Geometry

Create Parametrized Geometry

Assign Materials to Components

After creating a parametrized component, assign materials to the component, as follows:

- 1. Click Parts and choose Create New > Assembly. Drag the two generated objects into the Assembly.
- 2. Choose Materials > Select from Default Material Library to open the Material Library window.
- 3. Double-click Copper.
- 4. Click Add.
- 5. Drag the material Copper and drop it on the geometry.

Set Mesh Priority

To set the mesh priority of a component:

- 1. Double-click a material in the project tree to view its default priority.
- 2. Right-click Parts and choose View Parts List (All Parts).
- Double-click the mesh priority column and type the required mesh priority. OR
- 4. Right-click an object in the Parts list and choose Gridding/Meshing, Gridding- Meshing\Meshing Order> Set Priority.
- 5. Type the mesh priority.
- 6. Click OK.

Define Ports

You can create two types of ports in EMPro internal and waveguide port.

Add an Internal Port

To create an internal port:

- 1. Right-click Circuit Components/Ports from the Parts list and then select New Circuit Component with > New Feed Definition. The New Circuit Component window is displayed.
- 2. Define Endpoint1 and Endpoint2.
- 3. Click Apply.

NOTE Few materials are not supported by 3D components due to limitations in the FEM simulator.

Add a Waveguide Port

1. Right-click Circuit Components/Ports > Waveguide Ports from the Parts list. The EMPro Waveguide Port Editor is displayed.



- R
- 2. In the Location tab, click
- to select an edge of the component.
- **3.** Click the Properties tab.
- 4. Specify the Voltage source.
- 5. Click the Impedance Line tab.
- 6. Define Endpoint1 and Endpoint2.

Location	EditCrossSectionPage	Properties	Impedance Lines	
Endpoint	1: Fixed Position	Endpoint 2:	Fixed Position Mode Number 1	-
X: -7.5	mm 🔊	X: -7.5 m	m 🛛	
Y: 0		Y: 0		
Z: 0		Z: 2 mm		

- 7. Click OK. The impedance line appear on the waveguide port. The Waveguide port might show an invalid symbol. As per the message, waveguide port should lie on the faces of the geometry. Change the padding in x direction. Both lower and upper padding in X direction should be 0mm as waveguide port lie on X plane.
- 8. Click Done. This makes the waveguide port valid. Similarly, define other waveguide ports.

Adding an EMPro Library in ADS

WARNING This page is automatically copied from the Single Source area. All edits must be done to the original page. Click on this link to edit the page: Adding an EMPro Library in ADS Adding an EMPro Library in ADS

When an EMPro design is saved in a library, the following types of views are created:

- emModel:Represents a custom view type supported by ADS that allows cosimulation with an EM simulator by caching simulation results.
- empro: Consists of a 3D design, which is managed by EMPro. In the ADS cell containing EMPro library, right-click empro and select Open. The EMPro View window is displayed. You can view the design in EMPro by clicking Open in EMPro.
- layout and footprint: Represent a layout view in ADS. The layout and footprint views can be used interchangeably. These views are of the type layout and represent a 3D design, but as a 2D outline.
- symbol: Represents a schematic view and is the schematic black box-like symbol.
- lookalike: Represents a schematic view and is similar to the symbol view, but shows the outline of a 3D design in a 2D outline for use on a schematic.



The following figure describes various components of a library:



Adding an EMPro library in ADS

In ADS, a library includes multiple cells each containing a view that is used for a particular tool operating on the design. For example, symbol views are used in ADS Schematic, while a layout view can be manipulated by ADS Layout. Together, these

views form a cell. EMPro, similarly, allows you to manage a design stored in the database. For example, consider an SMA connector that is created in EMPro. After adding this connector in ADS, the cell can include an emModel or layout view. You can insert the emModel view directly in an ADS schematic and use for circuit/EM cosimulation. Similarly, you can insert a layout view on an ADS layout design for performing 3D simulations in ADS.

To add an EMPro library in ADS:

- 1. Select File > Manage Libraries from the ADS Main window. The Manage Libraries window is displayed.
- 2. Click Add Library.

Л٧	Man	nage	Libraries			
U	Libraries and library definition files used by this workspace.					
1	Name	e		Path	Mode	
	4	w	lib.defs	\$HOME\MyWorkspace_wrk\lib.defs		
		\triangleright	🌡 analog_rf.de	fs \$HPEESOF_DIR\oalibs\analog_rf.defs		
		Þ	퉬 dsp.defs	\$HPEESOF_DIR\oalibs\dsp.defs		
			MyLibrary_I	b MyLibrary_lib	Shared	
6	Add Li	brary	y Definition File	Add Design Kit from Favorites Add Library	Configure Library Remove	

- 3. Click Yes to close the message box. The Add Library dialog box is displayed.
- **4.** Specify the path of your EMPro library. You can also click Browse to access the required library.
- 5. Select the required library mode from the Mode drop-down list.
- 6. Click OK. The EMPro design is imported as a cell with multiple views in the ADS Main window.

Show me How Do I Add an EMPro Library in ADS

Video: Add an EMPro Library in ADS

Using Libraries, Cells, and Views in ADS

After saving your EMPro designs in a library, you can import the library in an ADS workspace. A workspace is used to store and organize the design work in *Library: Cell:View* hierarchical architecture. EMPro components are added to a cell in the workspace. A cell contains various types of *views* that represent different aspects of the component, such as layout, schematic, or symbol.

Library:Cell:View hierarchy

my_workspace	Workspace
🗉 🏨 my_library	Library
B ⊂ my_cell	Cell
 emModel empro footprint layout lookalike symbol 	View

ADS Library

In ADS, a library is a subdirectory that includes cells. Libraries are used by specifying the subdirectory in a library definition file, such as lib.defs. This file defines name of the library and the mode of operation (Read Only, Non-Shared, or Shared). Files within the library itself define the technology (layers, resolution, and layout units) to be used by Views created in that library. All types of designs are contained in libraries.

The process information such as layer definitions, units, and substrates are defined in the technology of a library. You can create multiple libraries within a complete design hierarchy. A library resides physically in the workspace directory.

Key Features of an ADS Library

The following are the key features of a library in ADS:

- Each Library has a unique name and path, specified in a library definition file. A library itself does not specify its name. The name must be specified correctly in a lib.defs file.)

NOTE Two libraries with the same name cannot be open at the same time.

- Associated with a Technology that defines physical layers, and units. The technology can be a native one created for a library, or it can be referenced to another library.
- Contains zero (0) or more Cells.

- Loaded/unloaded independently or collectively loaded through a lib.defs file.
- Defines the technology (layers, resolution, layout units). All views in a library use the same technology.
- Can be opened in following three modes:
 - Read-only Mode: In this mode, you cannot modify or commit changes to the library cells and the cell views. You need to use the Only Save As option. will work.
 - Shared Mode: In this mode, you can jointly modify and/or commit changes to the library cells and cell views with others that might have access to the same ADS library location.
 - Non-Shared Mode: In this mode, only you can commit any changes to the library's cells and cell views when library is open.
- Can reside physically anywhere in the system, with the following conventions:
 - All ADS components are inside Read-only ADS libraries which are stored with the product.
 - Normal end-user libraries are usually stored in a workspace. You can add a library from another workspace into the current workspace.
 - NOTE In ADS 2011, Workspaces are not included inside another workspace. Instead, a lib.defs file from another workspace can be added to the current workspace using **DesignKits** > Manage Libraries... from the ADS Main Window.

Cell

A Cell is a container of Views. Each Cell:

- Must have a unique name in the library, although cells with the same name may exist in another library.
- Contains zero or more views.
- May have multiple views of the same type.
- May have a component definition, edited by choosing File > Design Parameters... while editing one of its views.

View

A View is a subdirectory in a cell that stores design information such as schematic, symbol, or layout. Views may also store an EM simulation setup or an EM Model. Each view is a container that stores a file or a database object. It is a specific representation of a cell. All views in a given library use the same technology.

Each view:

- is associated with a Cell.
- must have a unique name in the Cell.

NOTE Schematic, Layout, and Symbol Views should generally be named as *schematic*, *layout*, and *symbol* unless you are using polymorphism.

- Has a type (schematic, layout, symbol, EM Setup, EM Model).

Adding EMPro Components in ADS Layout

Adding EMPro Components in ADS Layout

An EMPro library consists of a footprint and layout view that can be placed on an ADS Layout. After adding EMPro components in a layout, you can customize parameters and define EM ports on these components in ADS.

Perform the following steps to use EMPro components in ADS:

- 1. Save your EMPro designs in a library. For more information, see Saving EMPro Designs in a Library.
- 2. Import the EMPro library in ADS. For more information, see Adding an EMPro Library in ADS.
- 3. Place EMPro components in the ADS Layout window.
- 4. Defining EM Ports on EMPro components.
- 5. Assigning Pins to EMPro Ports.
- 6. Perform simulation using EMPro components.

This section provides information about how to add EMPro components in ADS layout, assign pins, and perform co-simulation with an ADS board.

Inserting EMPro Components in ADS Layout

Before using EMPro components in ADS Layout, the technology database of *empro_standard_layers* library must be referenced by the workspace library. The artwork (footprint) of 3D components and their port-pins are drawn on the *empro_3dc_artwork* and the *empro_3dc_pins* layers, respectively. These layers are saved in the empro_standard_layers library. The layer on which a 3D component pin is drawn together with the net determines its logical connectivity.

NOTE This is needed only if you want to create ports on the EMPro components in layout.

Referencing the EMPro Standard Layer Library

To link the *empro_standard_layers* library to your design in ADS:

- 1. Select Options > Technology > Technology Setup.
- 2. Click the Referenced Technology tab.
- 3. Click Add Referenced Library.

4. Add empro_standard_layers, as shown in the following figure:

M Add Referenced Library		×
Select a library		
Library Name	Layout Unit	Layout Resol 🔶
ads_standard_layers		=
empro_standard_layers		_
1xEV		
3GPPFDD		
3GPPFDD_10_99		
Antennas_and_Propagation		
CDMA		
СММВ		
Circuit_Cosimulation		
Controllers		
DTMB		-
<		
() indicates that the item is inherited by r	eferencing another li	brary.
	ОК	Cancel

5. Click OK.

Adding EMPro Components from the ADS Main Window

To insert an EMPro component in ADS layout:

- 1. Open a Layout window in ADS.
- 2. From the ADS Main window, select the layout or footprint view in the cell imported from EMPro.

⊿	<pre>empro_comp_lib</pre>						
	⊿	SMA_Johnson_PCB_Edge_Mount					
	EM emModel						
	empro						
		🔁 footprint					
		🔁 layout					
		🕞 lookalike					
		🖒 symbol					

- **3.** Drag and drop the selected component in the ADS layout window. The Properties dialog box is displayed.
- 4. Modify the component properties as per your requirements in the Properties dialog box.

5. Click OK. An EMPro component is included in the ADS layout. You can now rotate and position the component in your layout design.

Adding EMPro Components using the Component Library Window

Alternatively, you can also insert a component by using the Component Library window. Perform the following steps:

- 1. Click III in the ADS Layout window. The Component Library window is displayed.
- 2. Access the library imported from EMPro in Workspace Libraries.
- **3.** Double-click the required component from the Component column. The Choose View window is displayed.
- 4. Select footprint or layout from the Component column.
- 5. Drag and drop the component on the Layout window.

Customizing Component Parameters in ADS

You can customize the following EMPro component parameters in ADS

Parameter	Description
CustomComponentLayer	Layer in the substrate corresponding to Z=0 in the 3D Component.
CustomComponentOnTop	Alignment of Z=0 in the 3D Component with top/bottom of the layer.
CustomComponentRotateX	Rotates the component around the X axis.
CustomComponentRotateY	Rotates the component around the Y axis.
CustomComponentMirrorZ	Mirrors the component about the XY plane, boolean that triggers the projection f (z)=-z.
CustomComponentOffsetZ	Adds an additional translation in Z direction to the component, z distance specified by user.
minFreq	Minimum frequency of interest for the project.
maxFreq	Maximum frequency of interest for the project.

Defining EM Ports on EMPro Components

If you have defined ports in the EMPro design, the instance pins are visible in layout (diamond shape) corresponding to the connection points of the ports defined in EMPro. You can also add pins in the top layout, connected to the instance pins.

NOTE When a pin is recognized as a 3D pin, it displays text 3D.

To add pins:

- 1. Select Insert > Pin in the ADS Layout window.
- 2. Place the pins at the required location of the ports imported from EMPro, as shown in the following figure:



NOTE If two 3D pins map on the same 2D coordinates on footprint, the 2D projection of one pin is displaced with respect to other to select each pin. However, the 3D locations are not changed.

Waveguide Port

A waveguide port definition on an EMPro component can be used to create either a calibrated port or non-calibrated port when using this component in ADS layout. By default, the waveguide port definition will lead to a calibrated port. To do this, reference the empro_standard_layers technology to your design (see Adding EMPro Components in ADS Layout), add pins to the + and – instance pins of the EMPro component, and associate both pins to the TML port in the EM Setup window.

The location of the + and – instance pins correspond to the location of the end points of the impedance lines of the waveguide port. The waveguide port setup parameters as defined in EMPro will be used as the port parameters in the simulation driven from layout.

In case the calibrated port is not be located on the boundary of the simulation domain (e.g. because of the presence of other metal shapes), it will be automatically translated into a circuit component port.

You can manually force the waveguide port definition to be translated to a noncalibrated port by marking the port to be of calibration type None in the EM Setup window.

Circuit Component Port

A circuit component port definition on an EMPro component can only be used to create a non-calibrated port definition in ADS layout. To do this,reference the empro_standard_layers technology to your design (see Adding EMPro Components in ADS Layout), add pins to the + and – instance pins of the EMPro component and associate both pins to a non-calibrated port in the EM Setup window. The location of the + and – instance pins of the end points of circuit component port.

ADS allows a geometrical connection between ADS and EMPro components. You need to assign EM pins to an EMPro component only when you need to measure power of the component.

Assigning Pins to EMPro Ports

To assign layout pins to EMPro ports:

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

S-naramete	r Ports								
Number	/	Mama		Defin	nnadanca [Ohm]		Calibrat	tion	Г
	7	Name	Sec. 1	Refin	npedance [Onm]		Calibrat	uon	R
		P1		50 +	01		TML		0
	-1								
the the second s	-2	D3		50 1	oi		тм		0
	23	r J		50 +	UI III		1 ML		U
	24								
	4								
•				*******			e erer ere		
ayout Pins									
Name 🛛	Connec	ted to	Layer	Num	Layer	Purpo	se Num	Purpose	
🕪 P1	(+)P1		-777		empro_3dc_pins	-6		annotation	
🕪 P2	(-)P1		-777		empro_3dc_pins	-6		annotation	
🕪 P3	(+)P3		-777		empro_3dc_pins	-6		annotation	
🕪 P4	(-)P3		-777		empro_3dc_pins	-6		annotation	
•									

- **3.** Select an EMPro pin in Layout Pins.
- 4. Drag the pin from Layout Pins and drop it on the 🗳 or 🗖 terminal of the required port in S-parameter Ports.
- 5. A message box is displayed if your target pin is already used by other ports.
- 6. Click Delete Ports and Continuein the message box.

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

After assigning layout pins, launch FEM from layout and simulate your design. The EM Setup window determines simulation parameters.

For more information, see Defining Ports.

Performing 3D Visualization

You must complete the simulation process to view data for your EMPro design imported in ADS. If you have already simulated a design, start the Visualization feature directly to view the existing data. In the ADS layout, select EM > Post-Processing > Visualization to open the Keysight 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 to preview your design.

For more information, see Viewing Simulation Results in 3D.

Show Me How Do I Use EMPro Components in the ADS Layout Video: Use EMPro Components in the ADS

Using EMPro Components in ADS Schematic

Using EMPro Components in ADS Schematic

- Adding EMPro Components to Schematic
- Exporting EMPro Simulation Results as emModel

Adding EMPro Components to Schematic

Adding EMPro Components to Schematic

Using EMPro designs, you can perform EM circuit cosimulation in the ADS Schematic window. An EMPro component is represented by a symbol or lookalike in ADS Schematic. You can also perform cosimulation by using the S-parameters generated by an EMPro simulation. For more information, see Exporting EMPro Simulation Results as emModel. Perform the following steps to use EMPro components in ADS Schematic:

- 1. Save your EMPro designs in a library. For more information, see Saving EMPro Designs in a Library.
- 2. Import the EMPro library in ADS. For more information, see Adding an EMPro Library in ADS.
- 3. Place EMPro components in the ADS Schematic window.
- 4. Perform EM circuit cosimulation using EMPro components.

This section provides information about how to add EMPro components in ADS Schematic and perform EM circuit cosimulation with an ADS board.

Inserting EMPro Components in Schematic

You can insert the symbol components imported from an EMPro library in ADS Schematic. To insert a component, type the name of the component in the component name field or drag the component from the ADS Main window. The EMPro ports become schematic pins that can be connected to other components.

Adding EMPro Components from the ADS Main Window

To add an EMPro component in the ADS Schematic window:

- 1. Open a Schematic window in ADS.
- 2. From the ADS Main window, select symbol or lookalike from the cell imported from EMPro.
- **3.** Drag and drop the selected component in the Schematic window, as shown in the following figure:



Adding EMPro Components using the Component Library Window

Alternatively, you can also insert a component by using the Component Library window. Perform the following steps:

- 1. Click III. The Component Library window is displayed.
- 2. Access the library imported from EMPro in Workspace Libraries.

 All Libraries 	Search	Search	
 Workspace Libraries comp 	Component		-
EMPro_ADS_handson_lib	board		
ADS Analog/RF Libraries	cell_1 cell_2		
Read-Only Libraries	test_layout		
	lest_schem		
			-

- **3.** Double-click the required component from the Component column. The Choose View window is displayed.
- 4. Select symbol or lookalike from the Component column.
- 5. Drag and drop the selected component on the Schematic window.

Edits made to symbol views for EMPro Component

Symbol views for EMPro component are automatically created upon saving an EMPro design as an OpenAccess library.

It is possible to make changes to the symbol views from within ADS by opening the Symbol view editor, like adding additional objects or text, scaling the symbol view, or moving the existing objects or pin locations. These edits will be retained after saving the cell from within EMPro (e.g. after making a change in the simulation setup).

Only in case there is a change in the port definition (e.g. a new port is added to the design in EMPro), previous edits in the pins locations made from within the Symbol View editor are omitted.

Performing EM Circuit Cosimulation

After inserting a symbol or lookalike component in the ADS Schematic window, you can perform EM circuit cosimulation. The EM circuit cosimulation feature enables you to combine EM and circuit simulations from the schematic.

To perform the EM circuit cosimulation using EMPro components in ADS Schematic:

- 1. Start ADS and open a workspace.
- 2. Place the symbol or lookalike component in the Schematic window.
- 3. Connect the imported symbol with other components.
- 4. Select the component and click Choose View for Simulation 🔛
- 5. Select the emModel view for simulation.

6. Click OK.

7. Click Simulate

Show me How Do I Add EMPro Components in Schematic



Video: Add EMPro Components to Schematic

Exporting EMPro Simulation Results as emModel

Exporting EMPro Simulation Results as emModel

You can export EMPro simulation results as an ADS-compatible cell. The export process creates a cell with two views, emModel and symbol. You can use these cells in the ADS Schematic window. In ADS, a circuit simulator uses the Sparameters results of an EMPro simulation. If you have performed simulations from within EMPro, the resulting S-parameter models are available as emModels for use in the circuit simulation flow. The emModel names refer to the simulation id, such as 000001 and 000002.

Perform the following steps to export EMPro simulation results:

- 1. Create an emModel and symbol in EMPro.
- 2. Import the EMPro cell in ADS.
- 3. Place the cell in the ADS Schematic window.
- 4. Perform simulation using the imported cell.

This section describes the process of creating an emmodel and symbol, importing EMPro cell, and performing simulation using S-parameters results.

Creating emModel in EMPro

To create an EM model component in EMPro:

1. Select File > Export > Simulation Results as emModel in EMPro. The Export Simulation Results as emModel dialog box is displayed, as shown in the following figure:

📅 Export Simulation Results as emModel 🛛 🔀					
Simulation Result:	(000002) FEM, From 0 GHz to 10 GHz (Adaptive), Iterative Solver, User Sper				
OpenAccess Library:	C:\Keysight\EMPro2014_11				
Cell Name:	Microstrip_50_Ohm				
	🔀 Open Libraries Window after creation of cell				
	Create Cell Cancel				

2. Select the required simulation result from the Simulation Result drop-down list.

3. Specify the path for saving the library file in OpenAccess Library. You can

also click to choose a location for saving the library file. Entering a new library name creates a library during the export.

- 4. Specify a cell name. By default, the project name is used as cell name.
- **5.** Ensure that the Open Libraries window after creation of cell option is selected to display the Libraries window automatically.
- 6. Click Create Cell. The Libraries window is displayed, which consists of an EM model and symbol that is generated as result of saving the EMPro simulation results. The following figure displays a Libraries window:

<i>4</i>	Libraries			- 0	×	
Library						
Libraries						
C:/Users/anp/Documents/t1						
	U	X		10	Î	
						//,

NOTE Alternatively, drag and drop a simulation from the Simulation Window onto an OpenAccess library to export the results from the simulation.

Using the Exported Cell in ADS

To add an EMPro library containing emModel and symbol in the ADS workspace:

- 1. Start ADS and open or create a workspace.
- 2. From the ADS Main window, choose File > Manage Libraries to open the Manage Libraries window.
- 3. Click Add Library to open the Add Library dialog box.

Add Library	×
Path	
C:\Users\anp\Documents\t1	Browse
Name	
(The name specified when the library was created)	
t1	
Mode	
Shared	•
OK Cancel	Help

- 4. Click Browse and select the library that you have generated in EMPro. A valid library name is displayed automatically in the Name text box.
- 5. Select the required mode from the Mode drop-down list.
- **6.** Click OK. The library containing the exported cell is added to the workspace, as shown in the following window:

emModel and Symbol cell

An emModel provides an S-parameter simulation model based on the EM simulation for use in a circuit simulator. The following figure displays the emModel window:



The following figure displays an example of a symbol imported from EMPro:



You can place the cell in a Schematic design in several ways, such as using the Place Component command or by typing the cell name in the inserted component history field. Alternatively, drag and drop the cell from the libraries view on the Schematic window.

Performing a Simulation in ADS Schematic with the EMPro Exported Cell

To perform the circuit simulation and using the EMPro simulation results:

- 1. Start ADS and open or create a workspace.
- 2. Place the emModel or symbol in the Schematic window.
- 3. Connect the imported symbol with the terminals of other components.
- 4. Select the symbol and click Choose View for Simulation 🔛
- 5. Choose the emModel view for simulation.

Cho	ose View for Simulation					
Let the Hierarchy Policy determine which view to use						
Output Use this view for simulation:						
	emModel					
	OK Cancel Help					

- 6. Click OK.
- 7. Click Simulate 🥮 .

Exporting ADS Layouts to EMPro

Exporting ADS Layouts to EMPro

You can transfer an ADS layout to a new EMPro project by using the following options:

- Using the Launch EMPro and import this design Option: You can use the Launch EMPro and import this design option in ADS 2011 to export designs from ADS and import the designs in EMPro.
- Using the Create a self-contained import script Option: You can use the Create a self-contained import script option in ADS 2011 to specify a location for saving a self-contained Python script. To complete the conversion, import and execute this script in an empty EMPro design. You can use this process when ADS and EMPro are not installed on the same machine.

To transfer ADS layouts:

- 1. Select EM > Simulation Setup to display the EM Setup window.
- 2. Select FEM in the EM Setup window, as shown in the following figure:



- **3.** Click Launch EMPro and import this design. You can also choose Tools > Launch EMPro and import this design.
- 4. Select the directory where you have installed EMPro.
- 5. Click Choose.

NOTE You need to select the EMPro installation location only once, ADS automatically launches EMPro if you have already specified the directory.

If you encounter a problem while exporting ADS layouts, such as despite entering a positive value for the thickness of a STRIP layer of the substrate definition, the layer appears as a sheet in EMPro. In such cases, to obtain thick metallization layers, edit the substrate definition in ADS and set the STRIP layer model to Thick (Expansion Up) or Thick (Expansion Down) instead of Sheet (No Expansion).

EMPro ADS Integration FAQs

EMPro ADS Integration FAQs

This section provides step-by-step solutions for using EMPro 3D designs in ADS.

How Do I Add an EMPro Library in ADS

To add an EMPro library:

1. Open a workspace in ADS.

- 2. From the ADS Main window, select Design Kits > Manage Libraries.
- **3.** Click Add Library. ADS displays a message about selecting a lib.defs file. For libraries that contain only 3D cells or libraries that do not have references cells in other libraries, there is no lib.defs required.
- 4. Click Yes. ADS automatically provides a name to the selected library. The default name is used when cells from the selected library are put on other cells and be used to match the library name back to the physical location of the library.

Manage Libraries	definition flag used by this up	
Name	Path	Mode
4 👿 lib.defs	\$HOME\MyWorksp	ace_wrk\lib.defs
Þ 퉬 ana	log_rf.defs \$HPEESOF_DIR\oali	bs\analog_rf.defs
b 🎍 dsp	.defs SHPEESOF_DIR\oali	bs\dsp.defs
My	Library_lib MyLibrary_lib	Shared
Þ 퉬 lib.e	defs \$HOME\import_ad	s_adfi_3.4_dk\lib.defs
Add Library Definition	File	orites) Add Library Configure Library) Remove
		Close Help

5. Click OK.

How Do I Reuse Results from an EMPro Simulation in ADS Schematic

If a simulation is successful in EMPro, an emModel is created automatically from the simulation results. After simulation, the cell managed by EMPro consists of an additional view. The first of these views is called 000001 and subsequent simulations have increasing numbers. These views are fixed simulation results and are directly used for cosimulation on ADS Schematic. The following figure displays a 3D cell containing a simulation result from EMPro:



To reuse Results from an EMPro Simulation in ADS Schematic:

- 1. Start ADS and open or create a workspace.
- 2. Place the emModel or symbol in the Schematic window.
- 3. Connect the imported symbol with other components.
- 4. Select the symbol and click Choose View for Simulation 🔛
- 5. Select 000001 for simulation.

	Use this view for simulation:
	12 000001
	😥 emModel
•	S footprint
+ + 1 2 	OK Cancel Help

- 6. Click OK.
- 7. Click Simulate 🍄



How Do I Sweep an EMPro 3D Design using FEM in ADS

A complete EMPro 3D design is encapsulated in a cell. Cosimulation is supported through the emModel and empro views.

To sweep an EMPro 3D Design using FEM in ADS:

- 1. Drag and drop the symbol or lookalike view on ADS Schematic. Optionally, verify if the emModel view is selected for simulation.
- 2. Select the symbol and click Choose View for Simulation 🔛



3. Select emModel. The default hierarchy policy is to use the emModel. The following figure displays parameter sweeping a 3D design using ADS Schematic:



How Do I Simulate an ADS Board with an EMPro 3D Design in ADS Schematic

You can simulate a board created in ADS together with a full 3D design created in EMPro by using ADS Schematic as driver. Perform the following steps:

- 1. Simulate the board using ADS Layout and create an emModel from the simulation.
- 2. Open an ADS Schematic window and place the generated emModel on the schematic.
- 3. Add the library with the 3D design to your ADS workspace.
- 4. Place the *emModel* components of the 3D design on the schematic and connect all the pins to create your required schematic.
- 5. Click Simulate . If the emModel databases are already populated with simulation results, the circuit simulator reuses the existing results. If the emModel database is empty, then proper cosimulation is performed.

```
NOTE Cosimulation with a 3D design from ADS Schematic is supported only if the 3D design is set up for simulation with FEM. If another simulator is desired, simulate from within EMPro and reuse the existing results.
```

How Do I Simulate an ADS Board with an EMPro 3D Design in ADS Layout

You can simulate a board created in ADS together with a full 3D design created in EMPro using ADS Layout as driver. Perform the following steps:

- 1. Open the board in ADS Layout.
- 2. Add the library containing the 3D design to your ADS workspace.
- **3.** From the library, drag and drop the footprint view of the cell containing the 3D design on to your ADS Layout window containing the board.
- 4. Set the simulation options by using the EM Setup window.
- 5. Select the FEM simulator to perform the simulation.
- 6. Click Simulate

NOTE

You can perform ADS layout simulation of EMPro 3D designs only by using the FEM simulator (the Momentum simulator does not support layout simulation).

How Do I Simulate a Board Created in ADS Layout together with a EMPro 3D Design using EMPro as Driver

To simulate an EMPro design in ADS layout:

- 1. Open the board in ADS layout.
- 2. Set up the board for EM simulation.

- 3. Use the EM Setup window to perform the necessary setup steps. You can modify all simulation setup options later in EMPro. Therefore, you can use either the ADS EM Setup window or the tools from within EMPro to setup ports, simulation boundaries, sweeps, and other parameters. In general, it is more efficient to perform port setup from within ADS.The layout is now ready to be exported to EMPro.
- 4. Click Launch EMPro and import this design. Alternatively, a stand-alone script can be generated that you can run from within EMPro to reconstruct the board, including any setup made, in EMPro.

How Do I Move the Height of the Instantiation of a 3D Component

The height where a 3D component is instantiated is controlled by two parameters: CustomComponentLayer and CustomComponentOnTop. The parameter CustomComponentLayer is a layer being used to determine the Z location of the 3D component. It implies the layer has to be mapped into the substrate. The second parameter CustomComponentOnTop allows for thick layers to select between the bottom or the top of the layer to be used for the effective Z location.



How Do I Set the Origin of a 3D Component

The effective location of a 3D design in an ADS workspace is also determined by the location of the original 3D design. The absolute origin (0,0,0) of the 3D designs corresponds to the anchor point used in simulation driven from ADS layout to determine the final position of the 3D design in a combined board and 3D connector analysis. EMPro renders the absolute origin when the bounding box visualization of geometry is turned on.

The following figure shows origin of a 3D cell displayed in EMPro:



How do I add a Port on a 3D Component

To add a port on a 3D component:

- 1. Ensure that the simulator is set to FEM in the EM Setup window.
- 2. Insert the footprint of the 3D design in the layout on ADS Layout window.
- **3.** Insert a pin in the top-level design in the same manner as a regular 2D pin is added.
- 4. Snap the pin location to the instance pin that is part of the 3D component.
- 5. When a pin is recognized as a 3D pin, it displays text 3D, as highlighted in the following figure:

Ports									
Ports are stored with the layout. To undo port edits: use undo from the layout window then refresh.									
😒 🕼 🗶 😥 🔛 📉 🝸									
S-parameter Ports									
Number		Name	- I	Ref Imp	edance [Ohm]	Calibrat	ion	Ref	Offset [r 🔺
4 <u>1</u> 1	P1 P3	P1	5	50 + Oi		TML		0	E
▲ <u>10</u> 2 •	P2	P2	5	50 + Oi		TML		0	-
<				I	11				•
Layout Pins									
Name	Connect	ed to	Layer	Num	Layer	Purpose Nui	Purpose		X [mm]
🕪 P1	(+)P1		231		ads_device	240	ads_anno	tate	-25.8355
Image: P2	(+)P2		231		ads_device	240	ads_anno	tate	7.7215
🐵• P3	(-)P1		231		ads_device	240	ads_anno	tate	-25.8355
Image: P4	(-)P2		231		ads_device	240	ads_anno	tate	7.7215
						1			
•			111						· · ·

How Do I Add Text to Look-alike or Footprint from a 3D Design

To add text to lookalike or footprint view in ADS:

- 1. Open a workspace in ADS.
- 2. Add the library where the cell containing the 3D design is contained.

- 3. Access the cell and open the lookalike view.
- 4. Open the ADS Symbol editor to add the desired text or figures, as shown in the following figure:



NOTE

EMPro does not modify any markup created by a user when the lookalike is updated within EMPro. Every shape generated by EMPro on the lookalike and footprint view can be updated in EMPro, but not in ADS.

How Do I Flip a 3D Component

There are presently two ways to flip a 3D component when inserted on a board. The first one is to open the original 3D design in EMPro and flip the component there and then save the 3D design again. The component is immediately displayed as flipped when a new 3D view is inserted in ADS.

This method modifies the original 3D design. If you do not want to modify the existing design, you can place the existing 3D component on a layer of a nested technology, which is placed in a flipped way. The following figure displays a flipped component:



How Do I Change the Layer of the Footprint View To change the layer of a footprint view in ADS:

- 1. Open the footprint view in ADS.
- 2. Click Select or Select All to select all the components present on the view.
- 3. Select Edit > Properties. The Properties dialog box is displayed.

Standard	Custom					
Property		Value	,			-
All Shap	es					ſ
Layer		laver	1:drav	vina		18
Text and	Text Dis	plays				
Text		(mult	iple va	lues)		۲
Font		(mult	iple va	alues)		
Size		1.333	3			
Location	X	-1.95	57			
Location	Y	0				
Horizont	tal justif	Left				÷
4				_	Þ.	
Edit layer						

4. Select the required layer.



If the technology of the library does not have any layers, the layer selection will be empty. In that case, first add a layer to the technology of the library.

After changing layers, if the design is updated and the footprint view needs to be updated, EMPro will not change the layer position, but update the footprint but not the layer. EMPro verifies whether all EMPro-generated layouts are moved consistently to another layer. If all layers are not moved, EMPro will move all layout to the original layer.

NOTE 3D cells can be saved in any library, so it is often beneficial to save them in a library that already contains the technology definition that can be used in ADS. When EMPro creates a library, the technology does not contain any layers by default.

How Do I Scale or Offset a Look-alike Symbol

By default, look-alike symbols are generated using a set of rather simple heuristics to determine the final size. To customize the scale value of a lookalike view:

- 1. Open the view in ADS layout.
- 2. Click Select or Select All to select all the components present on the view.
- 3. Select Edit > Scale/Oversize \ Scale.
- 4. Specify the required scale factor. After editing the scale factor, you still need to select a center of scaling.

File	Edit	Select View Option	s Insert	Tools Window Help
D	9	Undo	Ctr1+Z	▶ 21 + ● ◆ ◆ ◆ ◆ 幸 幸 命
Palet	6	Redo	Ctrl+Y	
	8	End Command	Esc	Additional info
-	X	Cut	Ctrl+X	Sale8
	s.	Сору	Ctrl+C	Enten 🐝
		Paste	Ctrl+V	ScaleX
-		Advanced Copy/Paste	•	Sceley
	×	Delete	Del	1.000
	×	Delete All		
		Move	,	OK Cancel Help
	7	Rotate	Ctrl+R	
	9	Mirror About X	Shift+X	
	Ð	Mirror About Y	Shift+Y	
		Advanced Rotate	,	
		Scale/Oversize		Scale.
		Vertex		Oversize.
		Align		
		Power Pin		
		Properties		
	-			
				-01-
Sele	ct En	ter the starting point		66 items -0.875, 1.000 -0.375, 0.625 in

Similarly, you can offset by selecting Edit > Move or Move Relative.

NOTE If a lookalike view is scaled or offset, EMPro does not change the scaling and offset. Therefore, if the original lookalike is 3 inches wide on ADS Schematic, any update of the lookalike from within EMPro keeps the outline of the design within the lookalike view within 3 inches. Similarly, if the lookalike view was shifted 2 inches up, any updates to the lookalike view is generated with the same offset value.

How Do I Use a 3D Component Created in an Earlier Version of EMPro

To use a 3D component created in previous versions of EMPro:

- 1. Open the original 3D design (typically stored in an .ep directory or zep file) in EMPro 2012.
- 2. Click Save As.
- **3.** Select the OpenAccess library where you want to store the 3D design. Now, you can use the 3D design as a 3D component.

Example- Using the EMPro Connector Design in ADS

Example- Using the EMPro Connector Design in ADS

This section describes how to use the EMPro connector design, SMA Johnson Edge Mount, in ADS. The Connector design used in this example consists of:

- Parameter definition for permittivity dielectric coax, eps_coax.
- Design has two component ports:
 - Port 1 on the side that connects to the ADS board.
 - Port 2 on the coax side, which is a component port.



Perform the following tasks:

- Open the SMA_Johnson_Edge_Mount Project and modify the design.
- Save the Connector Design in an EMPro Library.
- Import the EMPro library in ADS.
- Use EMPro Connector in Schematic.
- Use EMPro Connector in ADS Layout

Open the SMA Johnson Edge Mount Project in EMPro

To display SMA Johnson Edge Mount project in EMPro:

- 1. Open the Libraries window in EMPro 2012.09.
- 2. Select SMA Johnson Edge Mount in Basic Libraries.

4	Libraries	- - ×
<u>F</u> ile <u>E</u> dit		
Libraries Basic Lik Parte 1 1 1 1 1 1 1 1 1 1 1 1 1	brary s Indexsar Hand Model - CTIA Dat Indexsar Hand Model - CTIA Fol Indexsar Hand Model - CTIA Mo	ta Hand Grip d Hand Grip ono Hand Grip
	Indexsar Hand Model - CTIA PD/ SAM HEAD SMA Johnson Edge Mount SMA PCB Edge Mount	A Hand Grip
Exte	uit Components rnal Excitations	
⊕ 🗍 Defi	nitions	-
	U X	

3. Drag and drop SMA Johnson Edge Mount on Parts in the Project tree. The Material Assignments window is displayed.

4. Accept the default materials and click OK. The SMA Johnson Edge Mount project opens in EMPro.

Create Ports

You need to add two ports in the SMA Johnson Edge Mount design:

- Port 1 on the side that connects to the ADS board.
- Port 2 on the coax side, which is a component port.

Save the Connector Design in an EMPro Library

To save the EMPro connector design in a library:

- 1. Select File > Save Project As to open the Save Project As dialog box.
- 2. Choose the required location for saving your project.
- 3. Click Create New OpenAccess Library (順). A new library is created in the Save Project As dialog box.
- 4. Double-click the new library.
- 5. Specify empro_comp_lib as name for your library.
- 6. Accept the default name for your project, SMA_Johnson_PCB_Edge_Mount.
- 7. Accept the default project type, EMPro OpenAccess projects.
- 8. Click Save. This saves the EMPro design in the new format, ready for use in ADS.

Import the EMPro library in ADS

To open EMPro library containing the connector design in ADS:

- 1. Select File > Manage Libraries from the ADS Main window. The Manage Libraries window is displayed.
- 2. Click Add Library. A message box is displayed.
- 3. Click Yes to close the message box. The Add Library dialog box is displayed.
- 4. Click Browse to access the required design.
- 5. Select empro_comp_lib.
- **6.** Click OK. The EMPro design is imported as a cell with multiple views in the ADS Main window as shown in the following figure:
 - empro_comp_lib
 - ✓ SMA_Johnson_PCB_Edge_Mount
 () emModel
 () empro
 () layout
 () lookalike
 () symbol

Open Design in ADS

To integrate EMPro components with an ADS design, open the following workspace:

ADS Install \$HPEESOF_DIR/examples/EM/RF_Microwave /EMPro_ADS_handson_wrk

To open the EMPro_ADS_handson_wrk.7zads workspace in ADS:

- 1. Select File > Open > Example. The Select an Archive File dialog box is displayed.
- 2. Double-click the FEM folder.
- **3.** Select EMPro_ADS_handson_wrk.7zads.
- 4. Click Open. The Unarchive Wizard is displayed.
- 5. Provide a destination path for unarchiving your example workspace.
- 6. Click Next.
- 7. Select EMPro_ADS_handson_wrk.
- 8. Click Finish.

Use EMPro Connector in Schematic

To add an EMPro component in the ADS Schematic window:

- 1. Open the simulate_schematic window from the EMPro_ADS_handson_wrk. 7zads workspace.
- 2. From the ADS Main window, select symbol from the cell, SMA_Johnson_PCB_Edge_Mount, imported from EMPro.
- 3. Drag and drop the symbol component in the Schematic window.
- 4. Connect the imported symbol with other components by adding connectors, mind polarity and port numbering, as shown in the following figure:



- 5. Select the symbol and click Choose View for Simulation.
- 6. Select the emModel view for simulation.
- 7. Click OK.
- 8. Click Simulate. It triggers the following sequence of actions:
 - Retrieve existing EM model for the board.
 - FEM simulation for connector, eps is equal to 2.
 - FEM simulation for connector, eps is equal to 4.
 - Perform Circuit simulation.

Use EMPro Connector in ADS Layout

To use connectors in a layout:

- 1. Open the layout design from the EMPro_ADS_handson_wrk.7zads workspace:
 - C:\Users\psinl\EMPro_ADS_handson_wrk
 EMPro_ADS_handson_lib
 board
 simulate_layout
 emSetup
 layout
 simulate_schematic
- 2. From the ADS Main window, select footprint from the cell imported from EMPro.
- **3.** Drag and drop the footprint component in the ADS layout window. The Properties dialog box is displayed.
- 4. Insert the first connector, and select pc2 as the layer to match the z-location.
- 5. Accept the default values and click OK. The EMPro connector is added to the ADS layout.
- 6. Insert the second connector, rotate and position, and select pc2 as the layer to match the z-location.
- **7.** Add four pins on layer ads_device: drawing, connected to the instance pins of the connectors
- **8.** Place the pins at the required location of the ports imported from EMPro, as shown in the following figure:



- 9. Create two EM ports from the four pins using the EM Setup window.
- 10. Select EM > Simulation Setup to open the EM Setup window.
- 11. Select Ports in the left pane of the EM Setup window. The EMPro pins are displayed in Layout Pins.
- **12.** Select an EMPro pin row in Layout Pins.
- **13.** Drag the pin from Layout Pins and drop it on the + or terminal of the required port in S-parameter Ports.
- 14. A message box is displayed if your target pin is already used by other ports.
- **15.** Select FEM in the EM Setup window.
- **16.** Specify the required settings in the EM Setup window for a layout, substrate, port, frequency plan, output plan, options, resources and model or symbol.
- 17. Click Simulate in the EM Setup window to simulate your design.
- **18.** Select EM > Post-Processing > Visualization to view your design.



Using the EMPro FEM Engine in ADS

Using the EMPro FEM Engine in ADS

While running FEM in ADS, the internal FEM engine that comes with the ADS installation is used. However, you can configure ADS to use the EMPro FEM engine by using the *ComponentRefs.ini* file located in the main ADS installation directory.

To use EMPro FEM Engine in ADS:

- 1. Close all the open sessions of ADS.
- 2. Open the *ComponentRefs.ini*file present at the following location:

<ADS_HOME>/ComponentRefs.ini

WARNING The *ComponentRefs.ini* file is write protected. Therefore, you must use an editor in the administrator mode.

3. To point ADS to FEM bits in a different location, modify the file or include an additional line. If you make the new line start with a higher number, this will be the version that is used. Example:

```
[fem]
2012.09\Location=%HPEESOF_DIR%\\fem\\260
2013.09\Location=C:\\keysight\\Mosaic\\fem\\2012.09
```

4. Run ADS.

Package Model Extraction

The Package Model Extraction add-on allows you to create a low frequency RLCK model from S-parameter data representing the package interconnect. It models the structure as a series of RLk block followed by a GC block to ground. In between the RLk and GC block, the interconnect block W models the fan out from inputs to outputs. Mutual coupling between the different branches is considered. The equivalent circuit topology is illustrated below. The mutual R components are not shown.



Enabling the Package Model Extraction Add-on

To enable the Package Model Extraction add-on:

- 1. Select Tools > App Manager... from the ADS Main window. The Manage ADS Application Features and User Addons dialog is opened.
- 2. Select the *Enabled* check box for Package RLCk Model Extraction. A confirmation message window is displayed.
- 3. Click Yes to enable.
- **4.** Restart ADS. The Package Model Extraction functionality is added in the Add-Ons menu of the Layout and Schematic window.



Using the Package Model Extraction Tool

To extract a package model from an S-parameter file.
- 1. From a Schematic or Layout window, select Add-Ons > Package Model Extraction > Start...
- 2. Select File > Open menu to open a file browser. Browse to the S parameter file of your package in the dataset, touchstone, or citi file format. Click Open.
- **3.** Configure the Port Partitioning by selection the appropriate setting for each port: Input, Output of Terminated.
- 4. Select the appropriate output you want: Data Display or Sub Circuit
- 5. Click the Extract button on the toolbar.

The Package Model Extraction dialog consists of the following elements:

S-Parameter Summary

Displays a summary of the basic information about the selected s-parameter fle such as name and number of available frequency points.

Port Input/Output Partitioning

If port names are available, the names are picked from the S-parameter file. When no port names exist, P1 up to Pn is used with n the number of ports in the file.

The Port Partitioning allows you to specify whether an S-parameter port should be considered as an Input, Output or be Terminated. The die ports are usually specified as input, package pin ports are typically outputs.



The initial value will be Terminated unless the tool is launched from a layout view and a port is found in the layout with a matching name, then the term type of that port is used to initialize the value. In that case, the settings can be pushed to

A valid setup requires at least 1 Input and 1 Output port. Normally, all ports need to be correctly defined. This can be done by picking up the information from a layout

port configuration by clicking the left icon 📤 . The right icon 😂 allows you to update the layout's configuration from the RLCK extraction UI.

The drop down on each Input/Ouput cell, or the context menu on a set of selected ports allows you to update the Input/Output configurations for one or more pins.

Output

The Output section allows you to specify the output of the RLCK extraction tool. You can choose the following options:

- Data Display with the RLCK fit for all frequencies of the original dataset.
- Create a Sub Circuit for a specific extraction frequency available in the Sparameter data.

Displaying Results in the Data Display Window

A dataset is created that is shown in a Data Display with a specific template matching the output data.

RLCk Dataset Naming Conventions

The RLCk dataset contains 4 blocks: data, inputs, outputs and ports.

Search	Hierarchy 🔻
 Equations Predefined Equations Datasets QFN_package QFN_package_rlck data inputs InputPortIndex Inputs outputs OutputPortInd Outputs index ports Ports TermType index 	ex >>Add >>
Dataset Content Block	

Dataset Block	Content
data	Contains R, L, G and C matrices versus frequency. The dimension of the R and L matrices is sweep_size(Inputs). The dimension of the G and C matrices is sweep_size(Outputs).
inputs	Contains a string array, Inputs, and an integer array, InputPortIndex. The former contains the port names whose TermType is input. The latter contains the port number or index.
outputs	Contains a string array, Outputs, and an integer array, OutputPortIndex. The former contains the port names whose TermType is output. The latter contains the port number or index.
ports	

Dataset Content Block

Contains two string arrays: Ports contains the port names and TermType the term type, both with the port number as index.

RLCk Data Display Template

A Data Display template dedicated to the RLCk extraction is available and loaded when the Data Display window opens.

The first page provides the matrices with equivalent circuit data at a frequency determined by the marker on the slider bar. The second page is used to automatically export the RLCK data at the selected frequency to 4 CSV files in the ADS workspace directory.



The Data Display window consists of two tabs:

You can manually insert that template as follows:

- 1. Open a new data display window.
- 2. Select Insert > Template. The DDS Template Libraries browser is opened.
- 3. Select the RLCk template under the Em_Product category and click OK.

Creating a Sub Circuit

You can generate a schematic at the extraction frequency by selecting the Sub Circuit option in the Output section. You specify the library and cell for the output and the schematic format that needs to be used. By default the schematic uses components from the ads_rflib for simulation within ADS only. When the Interoperable option is selected the generation builds a schematic using components from Cadence Virtuoso analog and basic libraries that can be used immediately in the Virtuoso schematic environment.

Interoperable operation requires an RFIC Interoperable License in ADS and analog and basic lib must be available in your ADS workspace.

The Input ports of the schematic are placed on the left of the design. They are followed by the RL block consisting of the series R, L and mutual inductances and mutual resistances between the branches. When connections have fan out from input to output an W interconnect structure is place after the RL block. On the right side the Ouput ports are places connected with the GC block providing the parasitic capacitances and admittances to the ground (Ref) and parasitic capacitance and admittance between the Ouput ports.

When the number of ports on the design is large this densely coupled circuit with fully populated matrices becomes very large. This makes it slow to create. But much more importantly circuit simulations with this complex circuit can take up a lot of simulation resources. In general however most of the mutual couplings elements are very small and have little effect on the simulation results. They can be neglected without ill effects on the simulation results. The Options section allows to control the dropping of the smallest elements by keeping only a the largest n mutual couplings and/or ignoring elements below the certain absolute threshold values for the mutual resistance, mutual inductance, admittance and capacitance.

Options

You can specify advanced options for extraction by clicking the Options button. The thresholding value is only applied when enabled in the UI. The N setting is applied on individual R, L, G and C elements. When the absolute value threshold is also active the smallest number elements is kept. When the N value is set to 0 all off diagonal elements of the RL and GC matrices are ignored.

The series (self) R and L value of each input branch is always maintained but the G and C to ground (Ref) on the outputs nodes can be dropped when below the threshold.

Prerequisites for a sub circuit cell that is interoperable with Virtuoso

Before you can use the Package Model Extraction functionality in the RFIC Interoperability mode you need to ensure that Cadence Virtuoso libraries analogLib and basic library are available in your ADS workspace. These libraries can be found on any Unix installation of Cadence Virtuoso:

- <Cadence Virtuoso IC tools install directory>/tools
 /dfII/etc/cdslib/basic
- <Cadence Virtuoso IC tools install directory>/tools
 /dfII/etc/cdslib/artist/analogLib

To update the analogLib library:

1. Select Tools > Build ADS analogLib.

- 2. Click Yes to build analogLib.
- 3. Select the analog library folder.
- 4. Click Select Folder.

After updating the library, you can add the analogLib and basic library in the Readonly mode to your ADS workspace.

To add these two libraries to your ADS workspace:

- 1. Select DesignKits > Manage Libraries.
- 2. Click Add Library in the Manage Library window.
- 3. Specify the path of the required library or click Browse to select a library.

	Manage Libraries		[23]
	Libraries and library definition file	s used by this workspace.	
	Name	Path	
	# Wib.defs	\$HOME\NXP_RLCK\RLCK_demo_wrk\lib	defs
	b 🎉 analog_rf.defs	\$HPEESOF_DIR\oalibs\analog_rf.defs	
	RLCK_demo_lib	RLCK_demo_lib	
	RLCK_Hb	RLCK_Rb	
	ads_pkg_if_extrac	t SHPEESOF_DIR\ael_addons\package_mo	IR Add Burn
4	analogLib	_\analogLib	Add Library
4	Dasic	-\basic SUOME ADC2015 DK TOK Library for A	Path
/	v 🔉 ilo.dets	showe(Abs2015_0K(TDK_Dbiary_for_A	C:\Users\ads\WD52015\Y0.0P_RLCK\basic Browse
			Name
			(The name specified when the library was created)
			basic
DesignKits ADS Board Link Library	•		Mode
Unzip Design Kit	Add Library Definition File Add I	Design Kit from Favorites Add Library	Read-only
Manage Favorite Design Kits			
Marca Director			OK Cancel Help
Manage Libraries			

4. Click OK.

The interoperable operation requires an RFIC Interoperable License in ADS.

Example

Following example illustrates the extraction capability.

- 1. Launch ADS
- 2. Select File > Open > Example and browse to the Momentum/RF directory.
- 3. Select the Package_RLCk_Extraction_wrk.7zads archive file and click Open.
- 4. Unarchive the workspace.

The cell named QFN_package contains a package with two ports (Ports 1 and 2) at the board side and 3 ports on the DIE side (Ports 3, 4, and 5) as shown below.



Port 3 and 4 at the die side are both connected to port 1 at the board side. This is a first net of interest. Port 5 at the die side is connected to port 2 at the board side. This is a second net of interest.

The equivalent circuit topology for this package is illustrated above.

An s-parameter model was generated by running Momentum RF. Open the Package Model Extraction addon, select the QFN_package.ds dataset, partition the ports as shown below and run the extraction.

Package Model Extraction [RLCk_Extraction_lib:QFN_package:layout]							
File							
<u> </u>							
S-Parameter Sum	mary						
File:	C:/ADS/ADS2017/AAA_	Examples/RLCk_Extraction_wrk/data/QFN_package.ds					
Frequencies: [Hz]	1e+08 to 1e+09, 10pt						
Port Input/Outpu	t Partitioning						
a							
Number	Name	Input/Output					
Filter	Filter	Filter					
1	P1 1	input					
- ²	P2 1 P3 1	Input					
1 1 A	P4 (Output					
10 5	P5 (Output					
Output							
Output Data Display 	Dataset Name	QFN_package_rlck.ds					
Output Data Display Sub Circuit 	Dataset Name Extraction Frequency [H	QFN_package_rlck.ds z] 100.0000 M					
Output Data Display Sub Circuit 	Dataset Name Extraction Frequency [H Library	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib					
Output Data Display Sub Circuit	Dataset Name Extraction Frequency [H Library Cell	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib QFN_package_rlck					
Output Data Display Sub Circuit 	Dataset Name Extraction Frequency [H Library Cell Schematic & Symbol View	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib QFN_package_rlck v schematic					
Output Data Display Sub Circuit	Dataset Name Extraction Frequency [H Library Cell Schematic & Symbol View	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib QFN_package_rlck QFN_package_rlck schematic symbol Interoperable					
Output Data Display Sub Circuit	Dataset Name Extraction Frequency [H Library Cell Schematic & Symbol View	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib QFN_package_rlck QFN_package_rlck schematic Image: Schematic Image: ADS					
Output Data Display Sub Circuit Show Options	Dataset Name Extraction Frequency [H Library Cell Schematic & Symbol View	QFN_package_rlck.ds z] 100.0000 M RLCk_Extraction_lib QFN_package_rlck QFN_package_rlck schematic symbol Image: ADS					

CoilSys

CoilSys is a new capability in ADS that is useful for RF/uWave IC designers. It is an Add-On to create a DRC clean inductors (single-ended/differential), balun, or transformer and transmission lines layout with layers mapped to a Momentum substrate. These layouts can be tuned/parameterized, and EM simulated. It can further create Pcells for both ADS and Virtuoso flow. CoilSys is used in automating:

- generation of scalable EM model through Advanced Model Composer.
- creation of required Pcells (ItemDef based and CDF based) in PDK which are not always available.
- creation of libraries of inductors, baluns, transformers, and transmission lines.
- generation of lookup table for synthesis and Inductor Finder for singleended and differential type of inductors.

Supported Platforms

CoilSys is an add-on utility that works on all ADS supported platforms. See Supported Platform.

Licensing

CoilSys required an ADS Advanced Layout license.

Using CoilSys

To use CoilSys, create a workspace or open an existing one, enable CoilSys, and then setup CoilSys components.

Creating Workspace for CoilSys

- 1. Launch ADS 2017.
- 2. Select File > New > Workspace to create a new workspace.
- 3. Specify the workspace name and browse to the desired workspace location.
- 4. Select Setup layout technology immediately after creating the library option.
- 5. Click Change Libraries.
- 6. Select Add User Favorite Library/PDK... option and browse to the lib.defs file of a foundry PDK.
- 7. Click OK.
- 8. Click Create Workspace and then click Finish. If the PDK is correctly selected in step 6, the PDK technology is selected by default in the dialogue shown below.

pdk180, 0.0005 micron layout resolution			
Don't setup layout tech until creating a layout			
reate PCB Technology (wizard)			
tandard ADS Layers, 0.0001 mil layout resolution			
tandard ADS Layers, 0.0001 millimeter layout resolution			
tandard ADS Layers, 0.001 micron layout resolution			
Custom (Opens Technology dialog)			
	 	 	_
Save selected technology as default for this workspace.			

Enabling CoilSys

CoilSys is an ADS add-on utility and can be enabled from the ADS main window by selecting Tools > App Manager... > CoilSys option after creating or opening a workspace.

lame	Enabled	Addon File Location
ADS Application Features		
Layout Command Line Editor		
ADS Board Link (ABL)		
PCB Library Import Tools - Mentor		
PCB Library Import Tools - Cadence		
PCB Library Import Tools - All Vendor	s 🗌	
PCB Library User Tools - All Vendors		
ADFI Import Tools		
EM Circuit Excitation		
Via Drawing Utility		
PDK Builder		
Bondwire Utility Tools		
Electrothermal Floorplanner	\checkmark	
SnP Utilities	\checkmark	
Si RFIC - Dummy Metal Fill Utility		
Package RLCK Model Extraction		
CoilSys	\checkmark	
User Addons		
d User ADS Addon Remove User ADS Addon		

After enabling CoilSys under App Manager, the "User_Input.ael" is copied to the current local workspace. Update Via Dimensions and Via Layer list as per foundry PDK technology specifications. This is to ensure generation of DRC clean PCells.

NOTE The variable viaLayerList must follow a specific order. It must always starts from the bottom of the stack, with via defined between the top metal and bottom metal, that is, the via close to the substrate must be defined first.

For example: For the following substrate the viaLayerList variable should be declared as shown below:



decl viaLayerList =list("Metal1", "Poly","Cont",

"Metal2", "Metal1", "Via1",

"Metal3", "Metal2", "Via2",

"Metal4", "Metal3", "Via3",

"Metal5", "Metal4", "Via4",

"Metal6", "Metal5", "Via5");

The packaging of Library from CoilSys depends on the mode that was set to either Legacy mode or Interoperable mode. To set the mode, follow these steps:

- 1. Select Options > Preferences from the ADS main window. The Main Preference window is displayed.
- 2. Click Interoperability tab and set the desired mode before using CoilSys.
- 3. Restart ADS.
- 4. Select Add-Ons > CoilSys option to launch CoilSys.
- 5. Setting Up CoilSys Components

Setting up CoilSys involves the following:

- 1. Create Pcells (Inductor/Transmission)
- 2. Generate Model
- 3. Package into Library

Create Inductor Pcell

This option enables the user to create Pcells for the different type of inductors.

- 1. Select Add-Ons > CoilSys > Create Inductor Pcell option.
- 2. Select the desired Inductor from the available list: Single Ended; Differential, Balun, and Solenoid.
- **3.** Select the desired type of Geometry from the available list: Square (with and without Center Tap), Octagonal (with and without Center Tap) and Circular.

🛅 CoilSys Cock	pit - Inductor								×
Library MyLibrar	y1_lib	•		Cell name	Enter	a valid cell na	ame to access of	ther options	
Inductor Single-Er	nded	•		Geometry	Squar	e			•
Line width 3	um	Line space	3			um	Solenoid width	32	um
Turns 3		Primary turns	3				Secondary turn	s 3	
Dimension Xouter	•	X/Y	200	/ 200		um	Miter length	8	um
Substrate	Substrate file no	ot found 💌							
Coil layers	None							Select Laye	'S
Underpass layers	None							Select Laye	rs
Center tap layers	None							Select Laye	rs
Underpass width	3 VES	um		UPassOrien CoilShupt	Nia	pendicular	•		
Coil2Eab Add Pin				Select	Vici				
				oureeur					
						Preview	Create Pcell	Cance	I
0									

- 0
- 4. Preview button helps the user to inspect the component layout instantly.
- 5. Create Pcell button will make the component part of the library.

Create Transmission Pcell

This option enables the user to create Pcells for the different type of Transmission lines.

- 1. Select Add-Ons > CoilSys > Create Transmission Pcell option.
- 2. Select the desired Transmission line from the available list: MicroStrip Line, Strip Line, Coupled Waveguide, and Coupled Waveguide with ground.
- 3. Preview button helps the user to inspect the component layout instantly.
- 4. Create Pcell button will make the component part of the library.

Generate Model

Once the Pcell is created, then it is available for generation of EM model through Advanced Model Composer. To create an EM-based model, follow these steps:

1. Select CoilSys > Generate Model.

The following dialog box opens up with the available PCells created through CoilSys.

del		×
Component Type	Geometry Type	Has Model
Single-Ended	Square	No
	Generate Model	Cancel
	del Component Type Single-Ended	del Component Type Geometry Type Single-Ended Square Generate Model

2. Select the Pcell from the above CoilSys Build Model dialog box, and select OK.

The Advanced Model Components window is displayed with the list of parameters for generation of the scalable EM-based model.

mSetup	
Ayout Parameters Select Parameter Width = constant(3 um) Space = constant(5 um) turns = constant(3.25) xDim = constant(200 um) yDim = constant(200 um) UPassWidth = constant(3 um)	Define Sweep Plan for Parameter Name UPassWidth Sweep Type Constant Value Value 3.0000000 um
	Update Parameter Sweep Plan

Package into Library

This enables the user to generate the add-on library. To create Library, follow these steps:

1. Select CoilSys > Package Library option and the following dialog box open up.

It lists all Cells (Pcells / Model) created through CoilSys.

Select all Cells or selected ones to package the library.
 While creating the package, provide a package name and location details.

Select Cell Name Component Type Geometry Type Has Mode MyLibrary1_lib:cell_1 Single-Ended Square No	Library na Create in 🖌 Select	C:\MyWorkspace1_wrk	:		Browse
	Select	Cell Name MyLibrary1_lib:cell_1	Component Type Single-Ended	Geometry Type Square	Has Model No

To use the packaged library, add Base PDK to the workspace before adding the library generated from CoilSys.

Inductor	Geometry	Lin Wi	e So dth W	olenoid 'idth	Line Space	Turns	Primary Turns	Sec. Turns	Х	Y
Single-	Square	Yes	s No	D	Yes	Yes	No	No	Yes	Ye
ended	Octagonal	Yes	s No	D	Yes	Yes	No	No	Yes	Ye
	Circular	Yes	s No	D	Yes	Yes	No	No	Yes	Ye
	Mitered Rectangular	Yes	s No	0	Yes	Yes	No	No	Yes	Ye
Differential	Square	Yes	s No	C	Yes	Yes	No	No	Yes	Ye
	Octagonal	Yes	s No	C	Yes	Yes	No	No	Yes	Ye
	Square_CT	Yes	s No	D	Yes	Yes	No	No	Yes	Ye
	Octagonal_CT	Yes	s No	D	Yes	Yes	No	No	Yes	Ye
Balun	Square	Yes	s No	D	Yes	No	Yes	Yes	Yes	Ye
	Octagonal	Yes	s No	C	Yes	No	Yes	Yes	Yes	Ye
	Rectangular_Vert_Coup	led Yes	s No	C	Yes	Yes	No	No	Yes	Ye
	Octagonal_Vert_Couple	d Yes	s No	C	Yes	Yes	No	No	Yes	Ye
	Octagonal_Vert_Couple	d_Sym Yes	s No	D	Yes	Yes	No	No	Yes	Ye
	Square_CT		s No	No	Yes	No	Yes	Yes	Yes	Ye
	Octagonal_CT	Yes	s No	C	Yes	No	Yes	Yes	Yes	Ye
Solenoid	Single-layer	Yes	s Ye	9S	Yes	Yes	No	No	No	Nc
Transmissio	on Line Con wid	ductor th	Condu Lengt	uctor h	Gro Wid	und th	Ground Space		Via Rows	
MicroStrip	Yes		Yes		Yes		No		No	_

Parameters Supported for Various Topologies

Strip Line	Yes	Yes	Yes	No	No
Coupled Waveguide	Yes	Yes	Yes	Yes	No
Coupled Waveguide with Ground	Yes	Yes	Yes	Yes	Yes

CoilShunt Parameter

- Via, Via Corners, Slot and Slot Broken are the supported CoilShunt types.
 - Via: Vias will be drawn all over the coil if there is an overlap between Mx and Mx-1 layers over the coil.



- Via Corners: Vias will be drawn at the corners of the coil if there is an overlap between Mx and Mx-1 layers over the coil.

İ۳.							-11			
	н					- 11				
						ĿЩ		· · · ·		
									·	
						1				
K		11	•							

- Slot: Slotted vias will be drawn all over the coil continuously if there is an overlap between Mx and Mx-1 layers over the coil.



- Slot Broken: Slotted vias will be drawn all over the coil discontinuously, at corners the slot will be broken, if there is an overlap between Mx and Mx-1 layers over the coil.



Coil2Fab Parameter

Add_Pins and Gnd_Shield are supported. By default Add_Pins is selected. Gnd_Shield draws ground shield for inductors as shown below:



NOTE Gnd_Shield will only be drawn for cases when the two bottommost metal layers in the selected stack-up are connected through a via.

Therefore, in few cases, you might observe no Gnd_Shield is drawn, even after enabling Gnd_Shield. In such cases, the symbol has to be updated manually by the user to remove the ports generated for Gnd_Shield.

Using Package in ADS Flow

- 1. Invoke ADS.
- 2. Create a new workspace with Base PDK added to it. See Using Coilsys for details.

- 3. Add the library from the package created using CoilSys into ADS workspace.
- 4. Select DesignKits > Manage Libraries...
- 5. Click on Add Library Definition File.

Browse the lib.defs file in the package and click Close.

NOTE If the package name is abc, the lib.defs file would exist directly in the package if it is created in legacy mode, whereas it would exist in abc/abc_ADS if it is created in interoperable mode.

Now, the library created using CoilSys can be used in ADS flow.

Using Package in Virtuoso Flow

- 1. Use the cds.lib file in the PDK directory to load the Base PDK in Virtuoso. If the file does not exist, then create it.
- Update the cds.lib file so that the library from the package created using CoilSys is included in Virtuoso. For example: if abc is the package created at XYZ location, then add the following line in the cds.lib file. DEFINE abc XYZ/abc/abc.
- 3. Invoke Virtuoso.
- 4. Select Tools > Technology File Manager from the CIW window.
- 5. Attach the technology of the base PDK to the library created using CoilSys.

NOTE Base PDK must be the same PDK which was used while creating the Package.

👻 Technology Tool Box	X
File <u>H</u> elp	cādence
~ Editor	
DEFT	
Managor	
New	Attach
Load	Dump
Save	Discard
litilities	
Install Device	Edit Layers
Graph	Guard Ring
▼ Attach Technolo	gy Library to D
	<u> </u>
Design Library	abc
Technology Library	gpdk180
	0.
ОК	Cancel Defa

Here, the reference PDK is gpdk180. So, the technology of gpdk180 is being added to the library 'abc' which is created using CoilSys.

Now, the library created using CoilSys can be used in Virtuoso flow.

Synthesis and Inductor Finder

Synthesis and Inductor Finder helps synthesizing single-ended or differential inductors. The inductor must have an EM model generated using CoilSys.

- Select Add-Ons → CoilSys → Synthesis menu option. The CoilSys Inductor Synthesis window is displayed.
- 2. Select an inductor from the list that has a model.
 (OK button is enabled only after a 'cell name' is entered and selected inductor has a model)

CoilSys Inductor Synthesis								
Enter a cell name to generate	e schematic templa	ate s1						
Select a c	ell to generate lo	okup table						
Cell Name	Inductor Type	Geometry Type	las Mode					
SYN_FLOW_CK_lib:de_sq	Differential	Square	Yes					
SYN_FLOW_CK_lib:se_sq	Single-Ended	Square	Yes					
	1	ок	Cancel					

3. Specify a cell name to generate schematic template and click OK. The CoilSys Inductor Parameters window is displayed.

📇 CoilSys Synthesis Parameters							
Enter parameter values to sweep for the component 'se1' (type 'Single-Ended' & geometry 'Square')							
	Min Limit	Max Limit	Step Size				
Line width (um)	3	3	1				
Line space (um)	3	3	1				
Turns	3	3	1				
X dimension (um)	200	200	20				
Y dimension (um)	200	200	20				
Underpass width (um)	3	3	1				
Miter length (um)	14	14	1				
Operating frequency (GHz)	0	10	.1				
Validate	ОК		Cancel				

4. Specify the parameter values except for the grayed-out parameters and click OK.

NOTE Specifying large values may result in longer simulation run time.

The schematic is displayed with information message.

d [SYN_FLOW_CK_lib:d:schematic] * (Schematic):10	- 🗆 ×
File Edit Select View Insert Options Tools Layout Simulate Window DynamicLink Help	
D 📁 🖬 👜 🗞 🗡 🍠 🥂 🗵 🕂 🖲 🧟 🧟 🥝 🥝 🔶 🗔 🖬 🎘 😹	
I Type Component N 🔽 🏥 🕼 🕼 🖓 🕂 🐨 🖓 🖓 🏦 🎆 🖓 🕂 🖏 💦 📩	V į I į OP į 🚆 🔹 »
0 👗 🖏 🗶 🟴 🚰 📴 📾 🖾 😵 🕺 🤋	
Palette 🗗 🗙	^
Demokit, Non_Linear	
	erm2 lum=2
Form Formation Message:10 X space um 22 INO RES Simulation will begin now as par data provided unsign um space um 24	=50 Ohm
Sinuadon win degin now as per data provide meson um	VAR VAR1 Line_Width=4
Poteo BatchSimCerrenoier Control PAD BitAltSimCerrenoier Starts VarrUne_Width Starts Stop=6 Step=1 Line VarrUne_Width Starts Stop=6 Okpc/2 Step=1 Line Stop=10 Ok/z Stop=10 Ok/z	X_Dimension=400 Y_Dimension=400 Line_space=5
Poteo Poteo Var=Tune_space* Statts StopPs StopPs Tune StopPs Tune_space StopPs Tune_space Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= StopPs Tune_space StopPs Tune_space Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= StopPs Tune_space* Statt=A00 Stop=StopPs Tune= StopPs Tune=space* Statt=A00 Stop=StopPs Tune= Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= StopPs Tune=space* StopPs Tune=StopPs Tune= Var=T_Vine=space* Statt=A00 Stop=StopPs Tune= Var=T_Vine=space* StopPs Tune=StopPs Tune= StopPs Tune=StopPs Tune=	
Poteo Analyset[1 = 51*] Control UseSverpModule=no SverpModule=no MH42 SverpModule=ne SverpModule=ne	
Poteon UseSeparateProcess=no Imma Mérgo Latasets=no M0 D0 FET 0	
	, .
Add Wire: Select points using the mouse or the coordinate entry windc 5 items ads_device:drawing	in

5. Click OK to start the simulation.

The parameters like L, Q, Reff are extracted automatically and displayed in the DDS window. The simulation data is saved in the current workspace.



- 6. Select Add-Ons \rightarrow CoilSys \rightarrow Find Inductor menu option.
- **7.** The Inductor Finder window has two drop down lists : Select Library & Select simulation template.

When a particular library is selected, the respective simulation templates in that library are listed in the simulation template drop down menu; select one.

The simulation data of selected template is displayed in form of a lookup table.

Parameters								Selec	t simulation templ	ate gpdk_lib_SE1s	ch			
inductance (H) Enter value for inductance							ce tolerance(i	n percentage)						
uality factor	lity factor Enter value for quality factor					Quality fa	actor toleranc	e(in percentage)						
requency (GHz)	Enter value for frequency					Frequenc	y tolerance(ir	percentage)						
Line Width(um)	Underpass Width(um)	Turns	Line Space(um)	Y Dimension(um)	X Dimensi	ion(um)	Frequency	Eff. Inductance(H)	Quality factor	i. Resistance(Ohr				
3	3	3	3	200	200		100.0 MHz	4.046e-09	0.081	31.576				
3	3	3	3	200	200		200.0 MHz	4.047e-09	0.161	31.603				
3	3	3	3	200	200		300.0 MHz	4.049e-09	0.241	31.642				
3	3	3	3	200	200		400.0 MHz	4.051e-09	0.321	31.69				
3	3	3	3	200	200		500.0 MHz	4.054e-09	0.401	31.747				
}	3	3	3	200	200		600.0 MHz	4.056e-09	0.481	31.811				
3	3	3	3	200	200		700.0 MHz	4.059e-09	0.56	31.883				

For additional filtering through the results, specify the parameters (along with tolerance) to separate out desired inductor from the total simulated results.

Parameters														
inductance (H)	6e-9					5								
Quality factor	Enter value for quality fact	or				Quality fac	tor tolerance	(în percentage)						
Frequency (GHz)	9					10								
Line Width(um)	Underpass Width(um)	Turns	Line Space(um)	Y Dimension(um)	X Dime	nsion(um)	Frequency	Eff. Inductance(H)	Quality factor	f. Resistance(Ohr				
3	3	3	3	200	200		8.400 GHz	5.728e-09	3.102	97.444				
3	3	3	3	200	200		8.500 GHz	5.783e-09	3.078	100.352				
3	3	3	3	200	200		8.600 GHz	5.84e-09	3.052	103.403				
3	3	3	3	200	200		8.700 GHz	5.898e-09	3.024	106.606				
3	3	3	3	200	200		8.800 GHz	5.958e-09	2.996	109.969				
3	3	3	3	200	200		8.900 GHz	6.02e-09	2.966	113.5				
3	3	3	3	200	200		9.000 GHz	6.084e-09	2.935	117.211				
3	3	3	3	200	200		9.100 GHz	6.15e-09	2.903	121.112				
3	3	3	3	200	200		9.200 GHz	6.217e-09	2.87	125.215				
3	3	3	3	200	200		9.300 GHz	6.287e-09	2.836	129.531				

- 8. Select an inductor from the lookup table.
- 9. Click Create Inductor.
- **10.** Specify a cell name to create a pcell of the synthesized inductor.

📇 CoilSys Inductor Finder	—
Enter a cell name to create pcell	OK Cancel

- 11. Click OK.
- **12.** The desired Inductor Pcell is displayed in the Layout window.



Preview Layouts

Preview Layouts

Before performing a simulation, you can validate your three dimensional design in the 3D Preview window. The representation seen in the 3D viewer consists of the same definition that will be used in a simulator. If a view is incorrect in the 3D viewer, it is essential to redesign before attempting a simulation. Using the 3D View window, you can now preview a layout design.

In the 3D Viewer, you can view the following components of a layout design:

- View all the nets, component instances, pins and substrate objects.
- View material definitions.
- Customize the view of conductors and dielectrics.

Displaying the 3D Viewer Window

Select EM > 3D EM Preview > Without EM Setup Preprocessing to generate a 3D view of the layout without performing any pre-processing task. This is the fastest way to generate a 3D view. You do not need to create an EM Setup view for generating the 3D view from layout.



If you select **EM > 3D EM Preview > With EM Setup Preprocessing** to generate a 3D view of the selected components, pins are not displayed in the Preview window.

You can also click \coprod in the layout window to display the layout preview:



Viewing the Components of a Layout Design

In the layout preview window, the Project panel provides a tree-structured representation of a design. You can view the following components of a design:

- Nets
- Components
- Substrates
- Pins

Searching and Viewing Nets

All nets in the layout view are listed in the project panel.

To find a specific net:

- 1. Click the Filter icon (\uparrow) next to Nets.
- **2.** Type the required net name.
- 3. Click to display a list of matching algorithms, as shown in the following figure:



4. Select the string matching algorithm: Simple, Regular Expression, or Hierarchical.



Viewing Components

All component instances in the layout view are listed and grouped per component.

How to Find a Component Instance

To find a specific component instance:

- 1. Click the Filter icon (\mathbf{T}) next to Components in the Parts tree.
- **2.** Type the component instance name.
- 3. Click * to display a list of matching algorithms, as shown in the following figure:



4. Select the string matching algorithm: Simple, Regular Expression, or Hierarchical.

Viewing Substrate

You can view substrate objects in the layer grouped by Conductor layer, Via layer or Dielectric layer.

How to View the Geometry by Layer

First, switch off the Parts Visibility toggle. This toggle can be found on the Geometry Window's View toolbar. Then, select the appropriate layer in the Substrate tree. The view displays the objects on the selected layer only.

Pins

All top level pins in the layout are listed.

Definitions

The *Definitions* branch stores definitions that can be applied to or shared with other objects within the project. You can apply a definition to other objects in the Project panel by clicking and dragging the definition object on the required object.

Materials

To view a material definition object, select a material from the Materials list.



Customizing the View

Using the View panel, you can change the view of conductors and dielectrics in the preview window.

Conductors	Solid	Wireframe	Hide
Dielectrics	Solid	Wireframe	Hide

In the View panel:

- Click Solid to apply make the objects shaded
- Click Wireframe to make the display only the edges
- Click Hide to make all the objects invisible.

The translucency of the objects, which are solid, are controlled by the translucency slide.

Layout Display Settings

The Layout Display Settings panel enables you to show, hide, shade, or unshade layers in the 3D Viewer window. You can also import or export the required file. To open this panel, select Tools > Layout Display Settings.

Layer Display Settings	X
All Layers	> 🔷 🔳 🖾
ETCH_TOP (1000)	🐼 🗊 📒
ETCHS2 (1001)	👁 🧰 📕
ETCH_SPLIT3 (1002)	👁 🧰 📒
ETCHSPLIT4 (1003)	۹ 🧰 📃
ETCHS5 (1004)	
ETCHVDD6 (1005)	Image: A state of the state
ETCHS7 (1006)	
ETCH_SPLIT8 (1007)	👁 🗐 🔳
ETCHS9 (1008)	👁 🗐 📕
ETCH_BOTTOM (1009)	
DRILLTOP_S2 (1010)	👁 🗐 📕
DRILL_TOP_BOTTOM (1011)) 💿 🧰 📕
DRILLS9_BOTTOM (1012)	👁 🗐 📒
4	h
F Im	port 🖨 Export

Customizing the Layout View

The following table describes options available in the View menu and View Tools toolbar:

Option	Icon	Description
Toolbars		- View Tools

Option	Icon	Description
View Manipulation	geometry_view_manipulation.	- Select: The Select tool is used to select objects as well as manipulate the view of the simulation space.
	git	 Orbit: The Orbit tool is selected to perform rotation of the simulation space through left- clicking-and-dragging.
		 Pan: The Pan tool is selected to perform translation of the simulation space through right-clicking-and-dragging.
		 Zoom: Zoom-in or zoom-out of simulation space by scrolling up or down the mouse wheel, respectively.
		- Zoom to Window: Zoom into a rectangular shaped area of the geometry as specified by the user. To use, select the tool, then left- click and drag the mouse to designate the rectangular zoom area.
Zoom to Extents	geometry_zoomextents.png	Enable you to zoom automatically as can view the entire geometry in the workspace window.
Zoom to Selection	geometry_zoomselection.png	Enables you to automatically zoom selected object only.
Standard Views	geometry_standardviews.gif	The Standard View button function is to automatically change the perspective of the objects in the workspace window.
		- Front (-Y)
		- Back (+Y)
		– Top (-Z)
		- Bottom (+Z)
		 Right (-X)
		- Left (+X)
lsometric Views	geometry_isometricviews.gif	The Isometric Views button function is to automatically change the perspective of the objects in the workspace window.
		 Front/Right/Top

Option	Icon	Description
		- Front/Left/Top
		 Front/Right/Bottom
		 Front/Left/Bottom
		 Back/Right/Top
		- Back/Left/Top
		 Back/Right/Bottom
		- Back/Left/Bottom
Custom Views	geometry_customviews.gif	Enable you to define any perspective than the Standard Views and Isometric Views, and display at that perspective.
		- Add View
Cutting Planes	geometry_customplanes.gif	Cutting Planes button feature allows you to define an arbitrary cross-section of the object and display it.
		 Toggle Cutting Plane
		- Edit Cutting Plane
		 Save Cutting Plane
Connectivity Tool	Ω	Toggle the connectivity tool.
Toggle Bounding Box Visibility		Toggles the visibility of the bounding box of a design when the design is selected.

Connectivity Tool

The Connectivity Tool allows you to verify the accuracy of large designs or unexpected behavior after imports. The connectivity tool is multi-threaded and after an initial preparation phase gives the real-time electrical connectivity of objects inside the design.

To check the connectivity of a design:

- 1. Click the Connectivity 🔟 icon present on the View Tools toolbar.
- 2. Position the mouse over the required part:
 - **a.** Wait for the connected parts to be highlighted (if highlighting on hover is enabled).

b. Click the part; you can add/remove parts using the key modifiers Ctrl /Shft; Note that by clicking on a part the highlighting on hover is temporarily disabled until all parts are removed from the selection.

Settings

In the Connectivity Tool Settings tab, you can specify the Electric Conductivity Threshold, and Evaluation Frequency, which allows you to control which materials are considered as electric conductors.

The Highlight Connected Parts while Hovering over a Part check box allows you to enable/disable the highlighting on hover.

	Geometry - Connectivity Tool	_ # X
⊻iew		
Connectivity Tool Settings	Neighborhood of Connected Parts	
Electric Conductivity Threshold: 100000 S/m I Highlight Connected Parts while Hovering over a Part		
Evaluation Freq	uency: 1 GHz	

Neighborhood

In the Neighborhood of Connected Parts tab, you can control which neighbors of the selected part (see 2.b) are highlighted by using the slider; Note that this can also be changed using the Ctrl + Mouse Wheel.

You can also change the colors of the selected parts and highlighted parts.

The incremental highlighting of the neighborhood allows you to find where unwanted connections, e.g., short circuits are made.



This information is subject to change without notice. www.keysight.com

