AWR Analyst
Getting Started Guide

Product Version 22.1 ISR3

AWR Analyst Getting Started Guide

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Table of Contents

Chapter 1. Introduction

The following AWR Design Environment Getting Started Guides are available:

- The *Microwave Office Getting Started Guide* provides step-by-step examples that show you how to use Microwave Office software to create circuit designs.
- The *AWR Analyst Getting Started Guide* provides step-by-step examples that show you how to use Analyst software to create and simulate 3D EM structures from the Microwave Office program.
- *Microwave Office MMIC Getting Started Guide* provides step-by-step examples that show you Monolithic Microwave Integrated Circuit (MMIC) features and designs.
- *AWR Visual System Simulator Getting Started Guide* provides step-by-step examples that show you how to use VSS software to create system simulations and to incorporate Microwave Office software circuit designs.

To set up the AWR Design Environment software for PCB style design, choose **Tools > Create New Process** to display the Create New Process dialog box, then click the **Help** button for details on using this tool.

Introducing the AWR Design Environment Platform

This platform comprises two powerful tools that can be used together to create an integrated system or RF design environment: VSS and Microwave Office software. These powerful tools are fully integrated in the AWR Design Environment platform and allow you to incorporate circuit designs into system designs without leaving the design environment.

VSS software enables you to design and analyze end-to-end communication systems. You can design systems composed of modulated signals, encoding schemes, channel blocks and system level performance measurements. You can perform simulations using the VSS software predefined transmitters and receivers, or you can build customized transmitters and receivers from basic blocks. Based on your analysis needs, you can display BER curves, ACPR measurements, constellations, and power spectrums, to name a few. VSS software provides a real-time tuner that allows you to tune the designs and then see your changes immediately in the data display.

Microwave Office software enables you to design circuits composed of schematics and electromagnetic (EM) structures from an extensive electrical model database, and then generate layout representations of these designs. You can perform simulations using any of the Cadence AWR simulation engines, such as a linear simulator; the Cadence APLAC® HB simulator for nonlinear frequency-domain simulation and analysis; the AXIEM 3D-planar EM simulator; the Analyst 3D-FEM simulator; or transient circuit simulators (the APLAC transient simulator or an optional Spectre simulator), and display the output in a wide variety of graphical forms based on your analysis needs. You can then tune or optimize the designs and your changes are automatically and immediately reflected in the layout. Statistical analysis allows you to analyze responses based on statistically varying design components.

The tool set spans the entire IC design flow, from system-level to circuit-level design and verification, including design entry and schematic capture, time- and frequency-domain simulation and analysis, physical layout with automated device-level place and route and integrated design rule checker (DRC), and a comprehensive set of waveform display and analysis capabilities supporting complex RF measurements.

OBJECT ORIENTED TECHNOLOGY

At the core of the AWR Design Environment platform capability is advanced object-oriented technology. This technology results in software that is compact, fast, reliable, and easily enhanced with new technology as it becomes available.

About This Guide

Through working examples, this Getting Started Guide is designed to familiarize you with Microwave Office, VSS, and Analyst software; and MMIC capabilities.

Prerequisites

You should be familiar with Microsoft® Windows® and have a working knowledge of basic circuit and/or system design and analysis.

This document is available as a download from the Cadence AWR Knowledge Base.

If you are viewing this guide as online Help and intend to work through the examples, you can download and print out the PDF version for ease of use.

Contents of this Guide

Chapter 2 provides an overview of the AWR Design Environment platform including the basic menus, windows, components and commands.

In the *Microwave Office Getting Started Guide* the subsequent chapters take you through hands-on examples that show you how to use Microwave Office software to create circuit designs including layout and AXIEM 3D planar EM layout and simulation.

In the *Analyst Getting Started Guide* the subsequent chapters take you through hands-on examples that show use of the Analyst 3D Electromagnetic simulator for 3D EM simulation within Microwave Office software. Use of 3D parametric layout cells and a 3D Layout Editor is included.

In the *Microwave Office MMIC Getting Started Guide* the subsequent chapters take you through hands-on examples that allow you to work with Monolithic Microwave Integrated Circuit (MMIC) features and designs.

In the *AWR Visual System Simulator Getting Started Guide* the subsequent chapters take you through hands-on examples that show you how to use VSS software to create system simulations and to incorporate Microwave Office software circuit designs.

Conventions Used in This Guide

This guide uses the following typographical conventions:

Item	Convention
Anything that you select (or click on) in the AWR Design Environment program, such as menus, nested submenus, menu options, dialog box options, buttons, and tab names	Shown in a bold alternate font. Nested menu selections are shown with a "**>**" to indicate that you select the first menu item and then select the submenu item: Choose **File > New Project**.
Text that you enter using the keyboard	Shown in a bold within quotation marks: Enter "**my_project**" in **Project Name**.
Keys or key combinations that you press	Shown in a bold alternate font with initial capitals. Key combinations using a "+" indicate that you press and hold the first key while pressing the second key:

Item	Convention
	Press **Alt+F1**.
File names and directory paths	Shown in italics: See the *DEFAULTS.LPF file*.

Getting Additional Information

There are multiple resources available for additional information and technical support for Cadence products.

Cadence AWR Knowledge Base

The Cadence AWR Knowledge Base includes these and other resources:

- Application Notes - Technical papers on various topics written by Cadence or our partners.
- Examples - Pages explaining project examples in the installed software or available for download.
- Licensing - A step-by-step guide to resolving most licensing problems.
- Questions - Frequently Asked Questions (FAQs) and answers for common customer issues.
- Scripts - Scripted utilities to help solve specific problems.
- Documentation - Downloadable copies of the latest released documentation.
- Videos - Short technical videos on how to accomplish specific tasks.

Documentation

Documentation for the AWR Design Environment platform includes:

- *What's New in AWR Design Environment v22.1?* presents the new or enhanced features, elements, system blocks, and measurements for the current release. This document is available in the Help by clicking the Windows **Start** button and choosing **AWRDE 22.1 > AWR Design Environment Help** and then expanding the **Cadence AWR Design Environment** node on the **Contents** tab, or by choosing **Help > What's New** while in the program.
- The *AWR Design Environment Installation Guide* describes how to install the AWR Design Environment platform and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips. This document is downloadable from the Cadence AWR Knowledge Base.
- The AWR Design Environment User Guide provides an overview of the AWR Design Environment platform including chapters on the user interface; using schematics/system diagrams, data files, netlists, graphs, measurements, and output files; using variables and equations in projects, and more. In addition, an appendix providing guidelines for starting a new design is included.
- The AWR Design Environment Simulation and Analysis Guide discusses simulation basics such as swept parameter analysis, tuning/optimizing/yield, and simulation filters; and provides simulation details for DC, linear, AC, harmonic balance, transient, and EM simulation/extraction theory and methods.
- The AWR Design Environment Dialog Box Reference provides a reference of many program dialog boxes with dialog box graphics, overviews, option details, and information on how to access each dialog box.
- The AWR API Scripting Guide explains the basic concepts of AWR Design Environment scripting and includes coding examples. It also provides information on the most useful objects, properties, and methods for creating scripts in the AWR Script Development Environment (AWR SDE). In addition, this guide contains the AWR Design Environment Component API list.

- The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the AWR Design Environment platform. This document is available within the program by choosing **Help > Quick Reference**. This is an excellent document to print and keep handy at your desk.
- Context sensitive Help is available for most operations or phases of design creation. To view an associated Help topic, press the **F1** key during design creation.

Documentation for Microwave Office software includes:

- The AWR Microwave Office Layout Guide, which contains information on creating and viewing layouts for schematics and EM structures, including use of the Layout Manager, Layout Process File, artwork cell creation/editing/properties, Design Rule Checking, and other topics.
- The AWR Microwave Office Element Catalog, which provides complete reference information on all of the electrical elements that you use to build schematics.
- The AWR Microwave Office Measurement Catalog, which provides complete reference information on the "measurements" (for example, computed data such as gain, noise, power, or voltage) that you can choose as output for your simulations.

Documentation for VSS software includes:

- The AWR Visual System Simulator System Block Catalog, which provides complete reference information on all of the system blocks that you use to build systems.
- The AWR Visual System Simulator Measurement Catalog, which provides complete reference information on the measurements you can choose as output for your simulations.
- The AWR Visual System Simulator Modeling Guide, which contains information on simulation basics, RF modeling capabilities, and noise modeling.

Documentation for the 3D Editor and Cadence Analyst™-MP multi-physics simulator (stand-alone product for multi-physics types of EM problems) includes:

- The *What's New in Analyst-MP v22.1* (*Analyst_Whats_New.pdf*), which presents the new or enhanced features for both the 3D Layout Editor and Analyst-MP simulator software.
- The *Analyst-MP Getting Started Guide* (*Analyst_Getting_Started.pdf*), which provides step-by-step examples that show you how to use Analyst-MP simulator software.
- The *Analyst User Guide* (*Analyst_User_Guide.pdf*), which provides an overview of the 3D Editor and Analyst-MP simulator software; including chapters on the user interface, structures, simulations, post-processing, variables, data files, and scripting.

Online Help

All AWR Design Environment documentation is available as on-line Help.

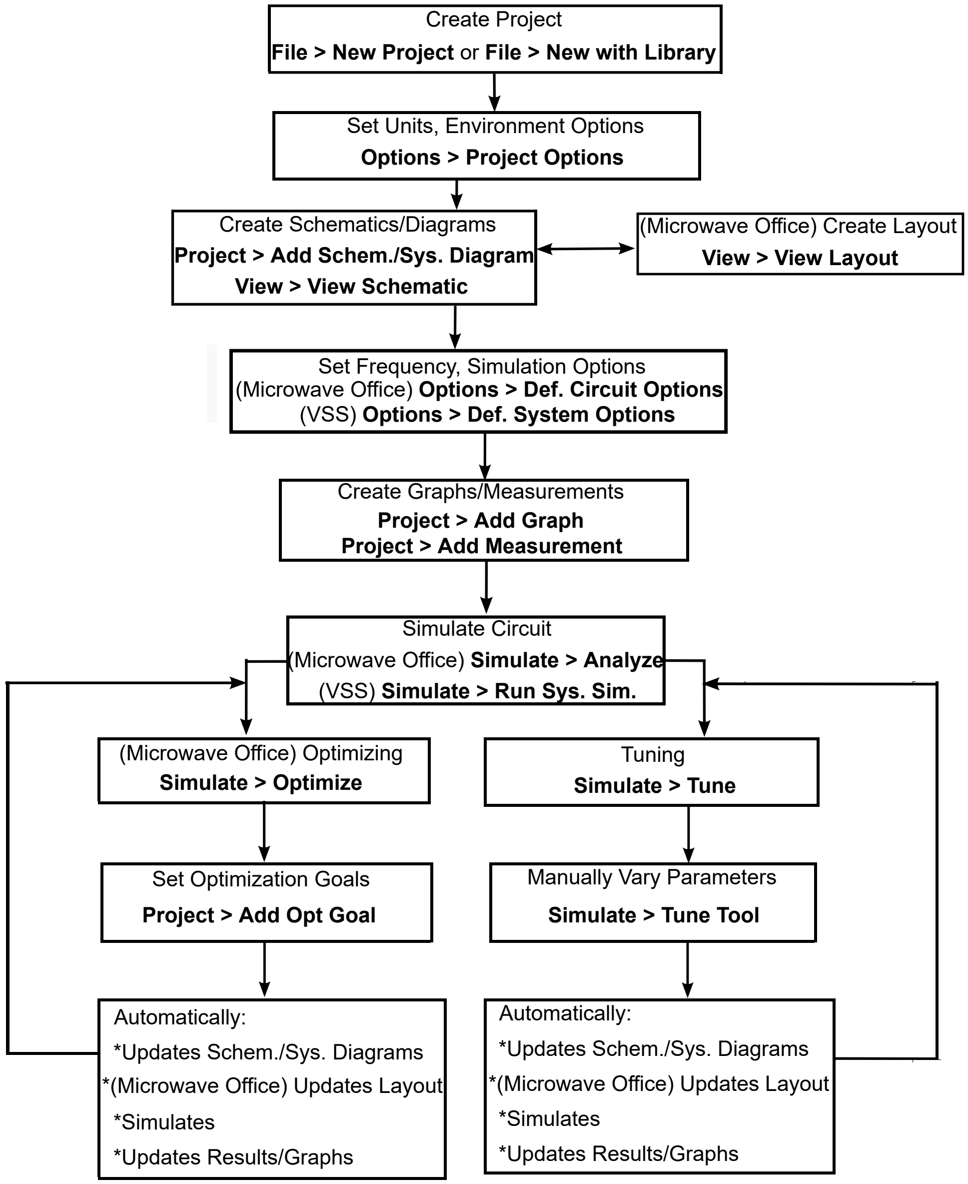
To access online Help, choose **Help** from the menu bar or press **F1** anywhere in the program. Context sensitive help is available for elements and system blocks in the Elements Browser and within schematics or system diagrams, and for measurements from the Add/Modify Measurement dialog box.

Online Support

The Cadence Learning and Support System is available from the Cadence Support website. You can navigate to this site from the AWR Design Environment platform by choosing **Help > Get Technical Support**.

Chapter 2. AWR Design Environment Platform

The basic design flow in the Cadence® AWR Design Environment® platform is shown in the following flow chart.



This chapter describes the windows, menus and basic operations for performing the following tasks in the AWR Design Environment platform:

- Creating projects to organize and save your designs
- Creating system diagrams, circuit schematics, and EM structures
- Placing circuit elements into schematics
- Placing system blocks into system diagrams

- Incorporating subcircuits into system diagrams and schematics
- Creating layouts
- Creating and displaying output graphs
- Running simulations for schematics and system diagrams
- Tuning simulations

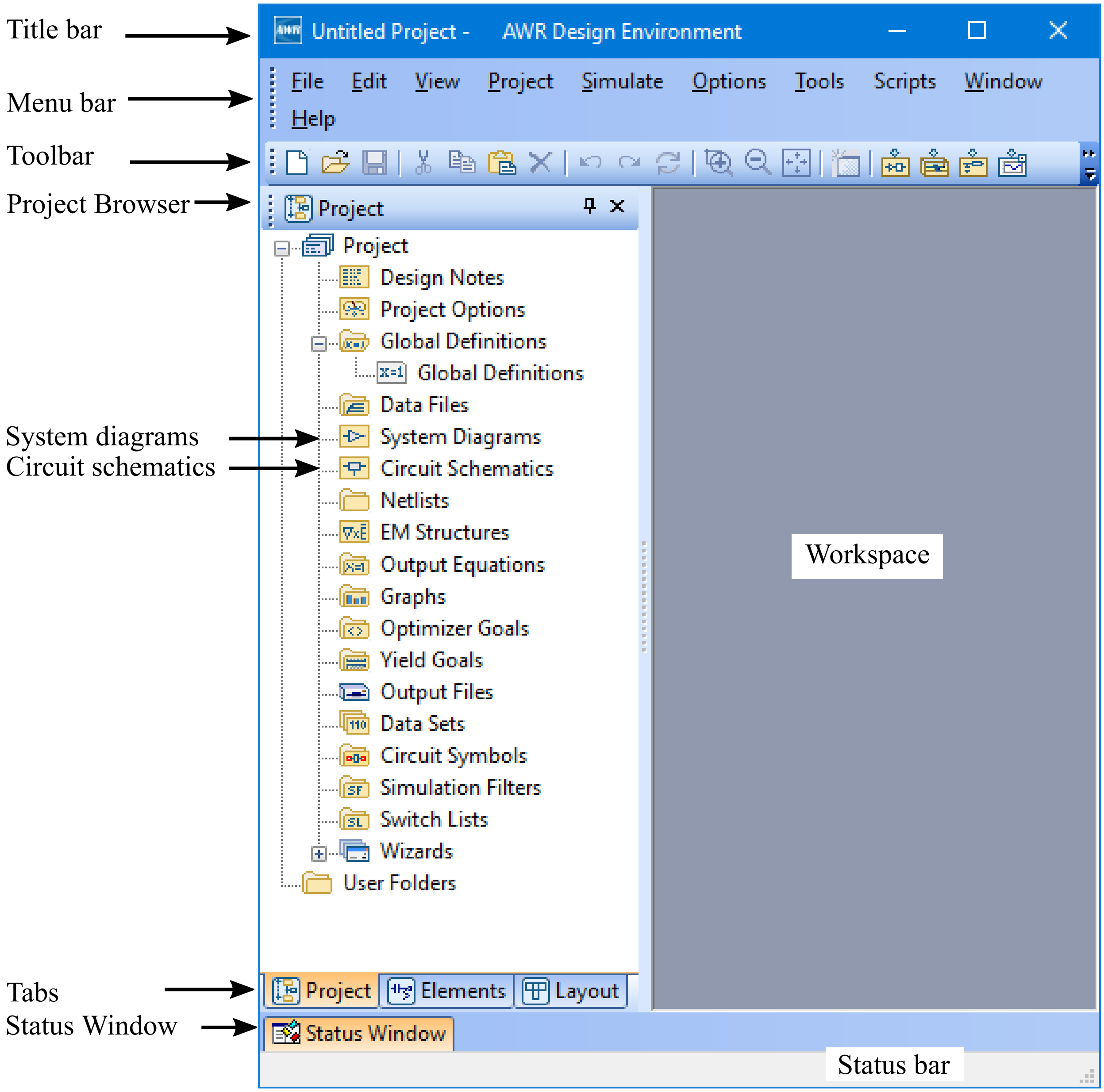
NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the AWR Design Environment platform. Choose **Help > Quick Reference** to access this document.

Starting AWR Software Programs

To start the AWR Design Environment platform:

1. Click the Windows **Start** button.
2. Choose **All Programs > AWRDE 22.1 > AWR Design Environment 22.1**.

The following main window displays.



If the AWR Design Environment platform was not configured during installation to display in your **Start** menu, start the application by double-clicking the **This PC** icon on your desktop, opening the drive and folder where you installed the program, and double-clicking on *MWOffice.exe*, the AWR Design Environment platform application.

AWR Design Environment Platform Components

The AWR Design Environment platform contains the windows, components, menu selections and tools you need to create linear and nonlinear schematics, set up EM structures, generate circuit layouts, create system diagrams, perform simulations, and display graphs. Most of the basic procedures apply to Cadence Microwave Office® software, Cadence Visual System Simulator™ and (VSS) communications and radar systems design software. The major components of the AWR Design Environment platform are:

Component	Description
Title bar	The title bar displays the name of the open project and any Process Design Kit (PDK) used with the project.
Menu bar	The menu bar comprises the set of menus located along the top of the window for performing a variety of Microwave Office and VSS tasks.
Toolbar	The toolbar is the row of buttons located just below the menu bar that provides shortcuts to frequently used commands such as creating new schematics, performing simulations, or tuning parameter values or variables. The buttons available depend on the functions in use and the active window within the design environment (as well as any customization of toolbar button groups). Position the cursor over a button to view the button name/function.
Workspace	The workspace is the area in which you design schematics and diagrams, draw EM structures, view and edit layouts, and view graphs. You can use the scrollbars to move around the workspace. You can also use the zoom in and zoom out options from the **View** menu.
Project Browser (**Project** tab)	Located by default in the left column of the window, this is the complete collection of data and components that define the currently active project. Items are organized into a tree-like structure of nodes and include schematics, system diagrams and EM structures, simulation frequency settings, output graphs, user folders and more. The Project Browser is active when the AWR Design Environment platform first opens, or when you click the **Project** tab. Right-click a node in the Project Browser to access menus of relevant commands.
Elements Browser (**Elements** tab)	The Elements Browser contains a comprehensive inventory of circuit elements for building your schematics, and system blocks for building system diagrams for simulations. The Elements Browser displays by default in the left column in place of the Project Browser when you click the **Elements** tab.
Layout Manager (**Layout** tab)	The Layout Manager contains options for viewing and drawing layout representations, creating new layout cells, and working with artwork cell libraries. The Layout Manager displays by default in the left column in place of the Project Browser when you click the **Layout** tab.
Status Window (**Status Window** tab)	The Status Window displays error, warning, and informational messages about the current operation or simulation. The Status Window displays by default at the bottom of the workspace when you click the **Status Window** tab.
Status bar	The bar along the very bottom of the design environment window that displays information dependent on what is highlighted. For example, when an element in a schematic is selected, the element name and ID displays. When a polygon is selected, layer and size information displays, and when a trace on a graph is selected, the value of a swept parameter displays.

You can invoke many of the functions and commands from the menus and on the toolbar, and in some cases by right-clicking a node in the Project Browser. This guide may not describe all of the ways to invoke a specific task.

Basic Operations

This section highlights the windows, menu choices, and commands available for creating simulation designs and projects in the AWR Design Environment platform. Detailed use information is provided in the chapters that follow.

Working with Projects

The first step in building and simulating a design is to create a project. You use a project to organize and manage your designs and everything associated with them in a tree-like structure.

Project Contents

Because Microwave Office software and VSS software are fully integrated in the AWR Design Environment platform, you can start a project based on a system design using VSS software, or on a circuit design using Microwave Office software. The project may ultimately combine all elements. You can view all of the components and elements in the project in the Project Browser. Modifications are automatically reflected in the relevant elements.

A project can include any set of designs and one or more linear schematics, nonlinear schematics, EM structures, or system level blocks. A project can include anything associated with the designs, such as global parameter values, imported files, layout views, and output graphs.

Creating, Opening, and Saving Projects

When you first start the AWR Design Environment platform, a default empty project titled "Untitled Project" is loaded. Only one project can be active at a time. The name of the active project displays in the main window title bar.

After you create (name) a project, you can create your designs. You can perform simulations to analyze the designs and see the results on a variety of graphical forms. Then, you can tune or optimize parameter values and variables as needed to achieve the desired response. You can generate layout representations of the designs, and output the layout to a DXF, GDSII, or Gerber file. See Appendix B, *New Design Considerations* in *AWR Design Environment User Guide* in the *AWR Design Environment User Guide* for advanced guidelines on starting a new design. You can also transfer technology and design information with Virtuoso and DE-HDL/Allegro platforms through a Cadence Unified Library. See Appendix E, *AWR Design Environment Interoperability with Virtuoso and Allegro* in *AWR Design Environment User Guide* in the *AWR Design Environment User Guide* for details.

To create a project choose **File > New Project**. Name the new project and the directory you want to write it to by choosing **File > Save Project As**. The project name displays in the title bar.

To open an existing project, choose **File > Open Project**. To save the current project, choose **File > Save Project**. When you save a project, everything associated with it is automatically saved. Cadence AWR® projects are saved as **.emp* files.

Opening Example Projects

Cadence provides a number of project examples (**.emp* files) in the installation directory to demonstrate key concepts, program functions and features, and show use of specific elements.

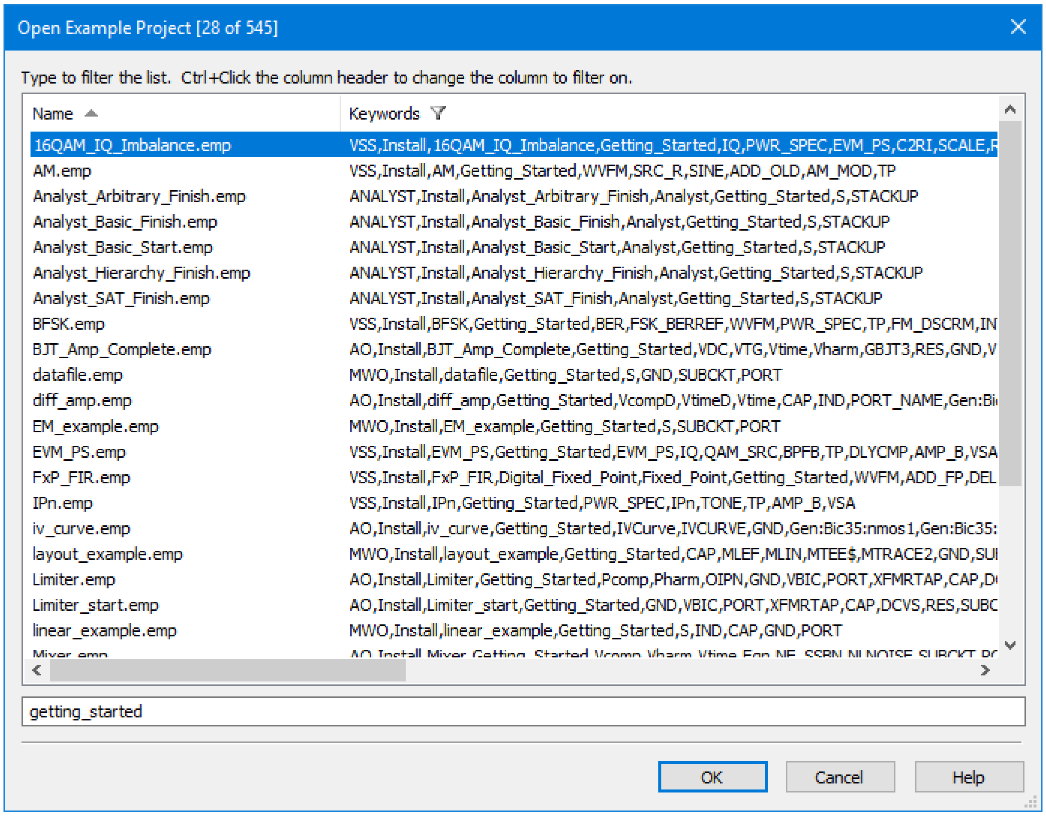
To search for and open example projects referenced in this guide:

1. Choose **File > Open Example**.

   The Open Example Project dialog box displays with columns for the project name and keywords associated with each example project.

2. Filter the list using "getting_started" as a keyword by **Ctrl**-clicking the **Keywords** column header and typing "**getting_started**" in the text box at the bottom of the dialog box.

   As shown in the following figure, the example list is filtered to display only those projects that have the "getting_started" keyword associated with them.



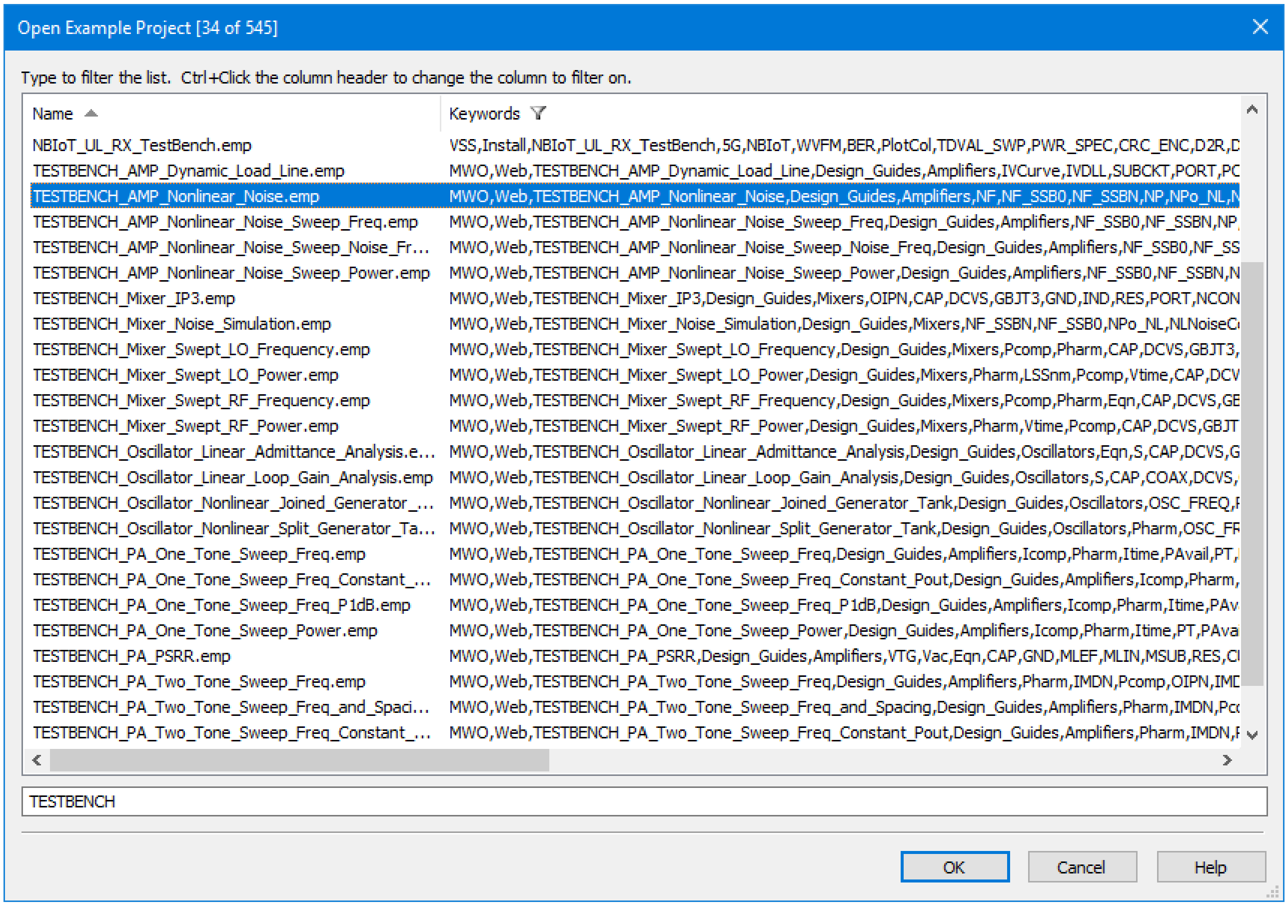
NOTE: You can filter examples by keyword or by file name. An inverted triangle in the column header indicates the column on which your search is filtered. Press the **Ctrl** key while clicking a column header to change which column is used to filter.

Importing Test Benches

Cadence provides several test bench examples that can serve as design guides for various applications such as mixers, amplifiers, and oscillators. These test benches are set up for import into your working project.

To import a test bench into your project:

1. Choose **File > Import Project**.
2. Browse to *C:\Program Files\AWR\AWRDE\22.1\Examples* or *C:\Program Files (x86)\AWR\AWRDE\22.1\Examples* and import the desired test bench. The test bench project file names are prefaced with *"TESTBENCH"* as shown in the following figure.



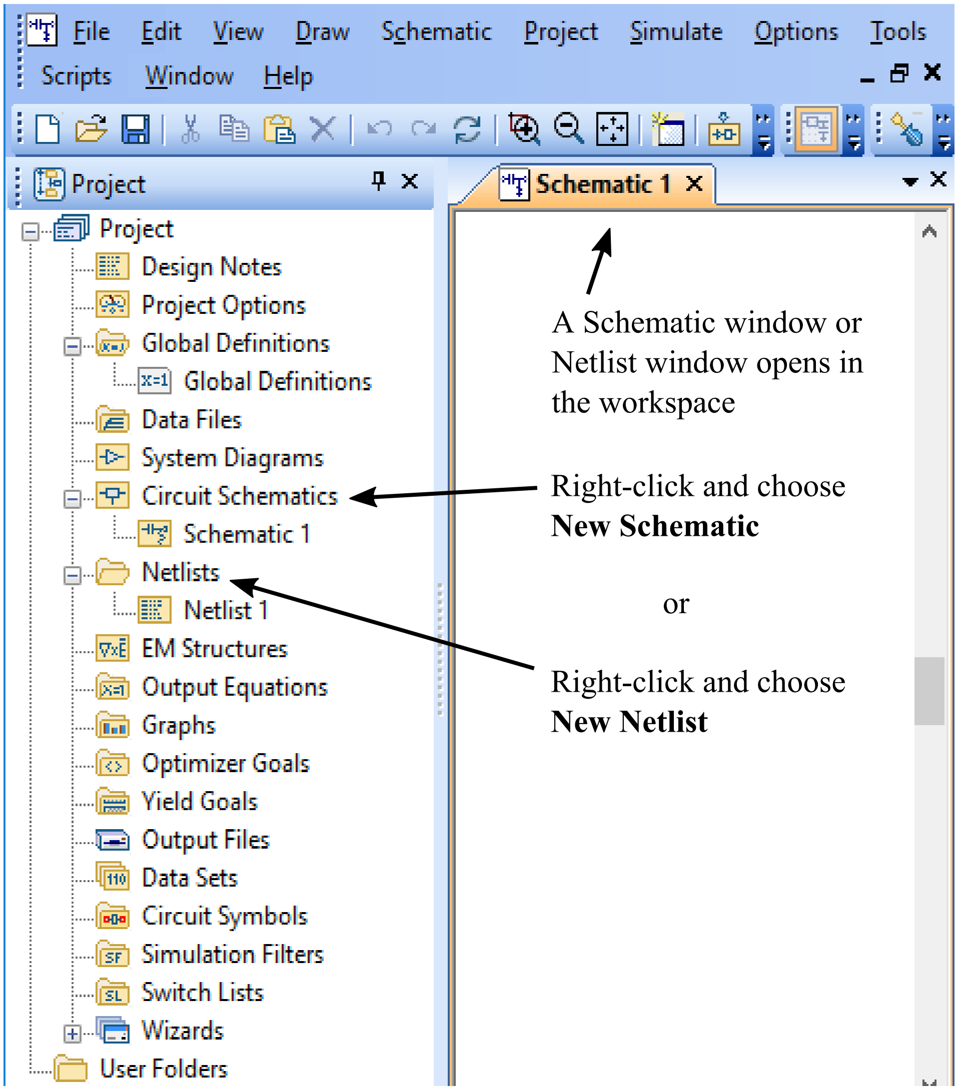
Working with Schematics and Netlists in Microwave Office

A schematic is a graphical representation of a circuit, while a netlist is a text-based description.

To create a schematic, right-click **Circuit Schematics** in the Project Browser, choose **New Schematic**, and then specify a schematic name.

To create a netlist, right-click **Netlists** in the Project Browser, choose **New Netlist**, and then specify a netlist name and type.

After you name the schematic or netlist, a window for it opens in the workspace and the Project Browser displays the new item as a subnode under **Circuit Schematics** or **Netlists**. In addition, the menu bar and toolbar display new command choices and buttons particular to building and simulating schematics or netlists.

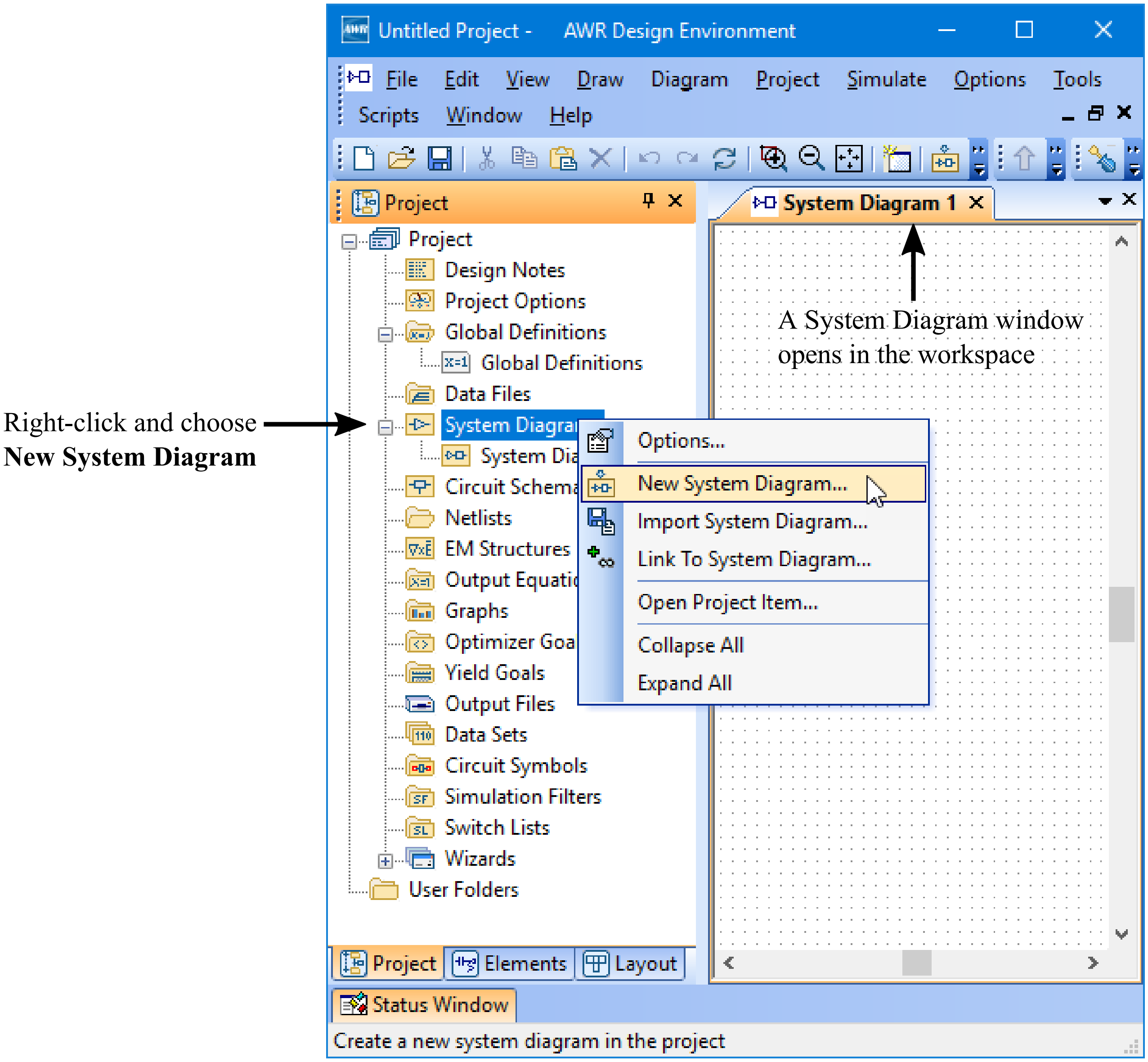


Adding Data to Netlists

When you create a netlist, an empty netlist window opens into which you type a text-based description of a schematic. Netlist data is arranged in blocks in a particular order, where each block defines a different attribute of an element such as units, equations, or element connections. For more information about creating netlists, see “Creating a Netlist” in *AWR Design Environment User Guide*.

Working with System Diagrams in VSS

To create a system diagram, right-click **System Diagrams** in the Project Browser and choose **New System Diagram**, and then specify a system diagram name.



After you name the system diagram, a window for it opens in the workspace and the Project Browser displays the new item as a subnode under **System Diagrams**. In addition, the menu bar and toolbar display new command choices and buttons particular to building and simulating systems.

Connecting Element and System Block Nodes

You can connect elements or system blocks directly by positioning them so their nodes touch. Small green boxes display to indicate the connection. You can also connect elements with wires.

- To connect element or system block nodes with a wire, position the cursor over a node. The cursor displays as a wire coil symbol. Click at this position to mark the beginning of the wire and drag the mouse to a location where a bend is needed. Click again to mark the bend point. You can make multiple bends.
- Right-click to undo the last wire segment added.

- To start a wire from another wire, select the wire, right-click and choose **Add wire**, then click to mark the beginning of the wire.
- To terminate a wire, click on another element node or on top of another wire.
- To cancel a wire, press the **Esc** key.
- When placing or positioning an element, alignment guidelines automatically display when the element nodes align with another element. To automatically add a wire between the nodes, press the **Shift** key when placing the element.

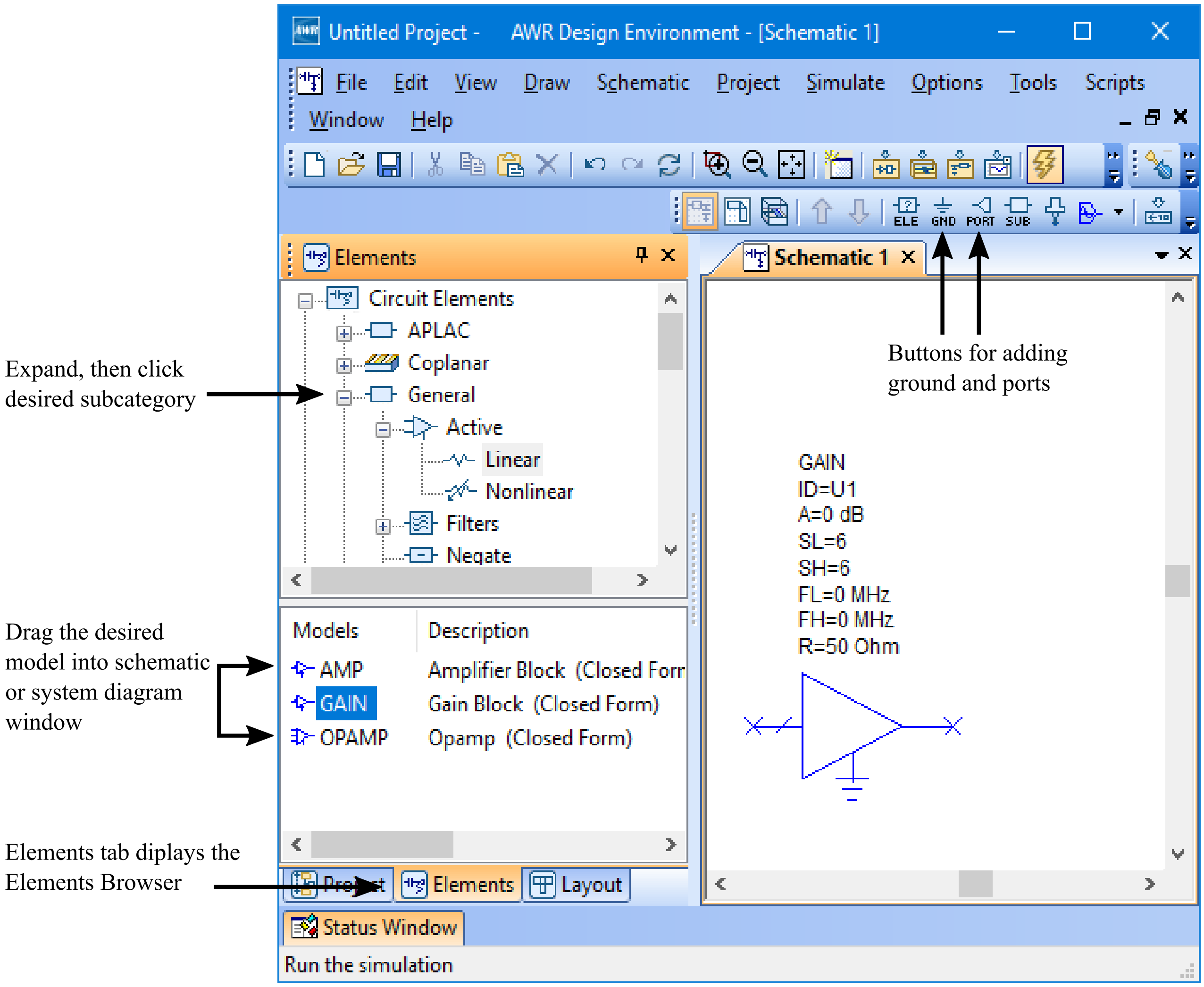
Using the Elements Browser

The Elements Browser gives you access to a comprehensive database of hierarchical groups of circuit elements for schematics and system blocks for system diagrams. The Libraries folder in the Elements Browser provides a wide range of electrical models and S-parameter files from manufacturers.

Circuit elements include models, sources, ports, probes, measurement devices, data libraries, and model libraries that can be placed in a circuit schematic for linear and non-linear simulations.

System blocks include channels, math tools, meters, subcircuits, and other models for system simulations.

- To view elements or system blocks, click the **Elements** tab. The Elements Browser replaces the Project Browser window.
- To expand and collapse the model categories, click the **+** or **-** symbol to the left of the category name to view or hide its subcategories. When you click on a category/subcategory, the available models display in the lower window pane. If there are more models than the window can show, a vertical scroll bar displays to allow you to scroll down to see all of the models.
- To place a model into a schematic or system diagram, simply click and drag it into the window, release the mouse button, right-click to rotate it if needed, position it, and click to place it.
- To edit model parameters, double-click the element graphic in the schematic or system diagram window. An Element Options dialog box displays for you to specify new parameter values. You can also edit individual parameter values by double-clicking the value in the schematic or system diagram and entering a new value in the text box that displays. Press the **Tab** key to move to the next parameter when editing.



NOTE: Choose **Draw > More Elements** to display the Add Circuit Element or Add System Block dialog box to search for elements. Press the **Ctrl** key while clicking a column header to change which column is used to filter.

Adding Subcircuits to Schematics

Subcircuits allow you to construct hierarchical circuits by including a subcircuit block in a schematic (insert a schematic inside of another schematic). The circuit block can be a schematic, a netlist, an EM structure, or a data file.

- To add a subcircuit to a schematic, click **Subcircuits** in the Elements Browser. The available subcircuits display in the lower window pane. These include all of the schematics, netlists, and EM structures associated with the project, as well as any imported data files defined for the project.
- To use a data file as a subcircuit, you must first create or add it to the project. To create a new data file, choose **Project > Add Data File > New Data File**. To import an existing data file, choose **Project > Add Data File > Import Data File**. Any new or imported data files automatically display in the list of available subcircuits in the Elements Browser.
- To place the desired subcircuit, simply click it and drag it into the schematic window, release the mouse button, position it, and click to place it.

- To edit subcircuit parameters, select the subcircuit in the schematic window, right-click, and choose **Edit Subcircuit**. Either a schematic, netlist, EM structure, or data file opens in the workspace. You can edit it in the same way that you would edit the individual circuit block types.

Adding Subcircuits to System Diagrams

Subcircuits allow you to construct hierarchical systems and to import results of circuit simulation directly into the system block diagram.

- To create a subcircuit to a system diagram, choose **Project > Add System Diagram > New System Diagram** or **Import System Diagram** and then click **Subcircuits** under **System Blocks** in the Element Browser. The available subcircuits display in the lower window pane.
- To place the desired subcircuit, simply click and drag it into the system diagram window, release the mouse button, position it, and click to place it.
- To edit subcircuit parameters, select the subcircuit in the system diagram window, right-click, and choose **Edit Subcircuit.**
- To add a system diagram as a subcircuit to another system diagram, you must first add ports to the system that is designated as a subcircuit.

Adding Ports to Schematics and System Diagrams

To add ports to a schematic or system diagram, expand the **Ports** category in the Elements Browser. Under **Circuit Elements** or **System Blocks**, click **Ports** or one of its subgroups, for example, **Harmonic Balance**. The available models display in the lower window pane.

Drag the port into the schematic or system diagram window, right-click to rotate it if needed, position it, and click to place it.

For a shortcut when placing ports and ground, click the **Ground** or **Port** buttons on the toolbar, position the ground or port, and click to place it.

To edit port parameters, double-click the port in the schematic or system diagram window to display an Element Options dialog box.

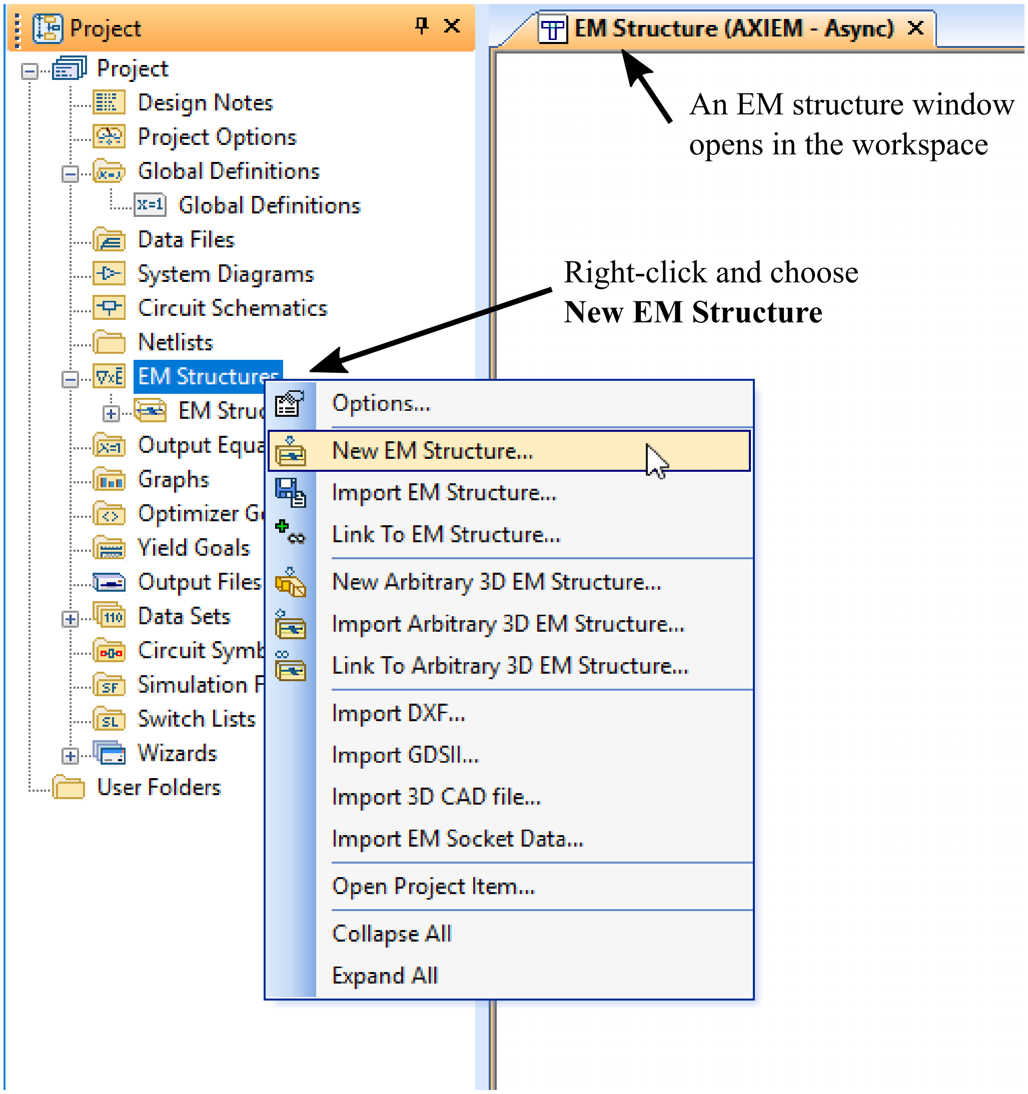
NOTE: You can change the port type after placing it by double-clicking the port and selecting a **Port type** on the **Port** tab of the dialog box.

Creating EM Structures

EM structures are arbitrary multi-layered electrical structures such as spiral inductors with air bridges.

To create an EM structure, right-click the **EM Structures** node in the Project Browser, and choose **New EM Structure**.

After you specify an EM structure name and select a simulator, an EM structure window opens in the workspace and the Project Browser displays the new EM structure under **EM Structures**. In addition, the menu and toolbar display new choices particular to drawing and simulating EM structures.



NOTE: The EM structure examples presented in this guide use Cadence AXIEM® 3D planar EM analysis.

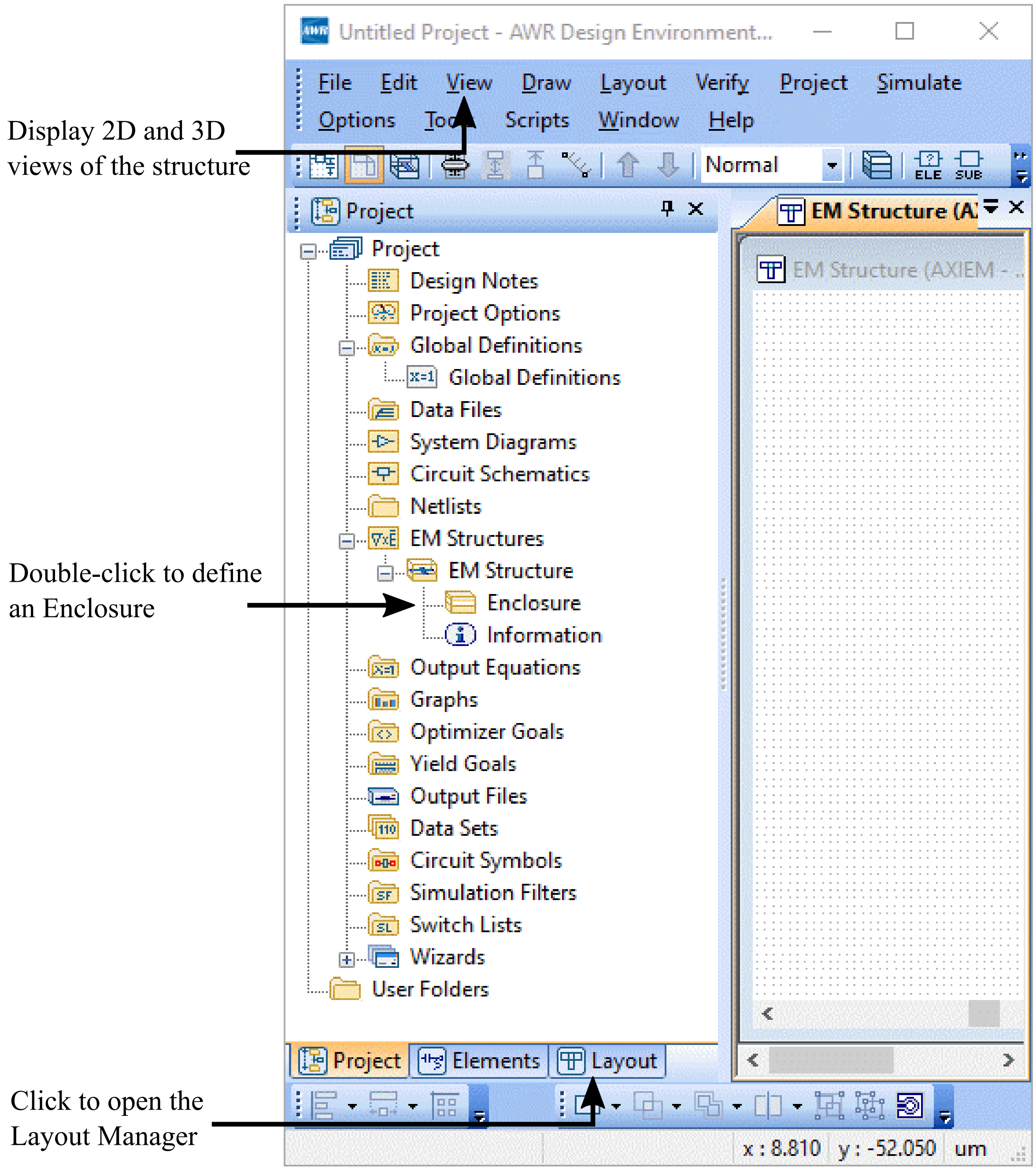
Adding EM Structure Drawings

Before you draw an EM structure, you must define an enclosure. The enclosure specifies things such as boundary conditions and dielectric materials for each layer of the structure.

To define an enclosure, double-click **Enclosure** under your new EM structure in the Project Browser to display a dialog box in which you can specify the required information.

After you define the enclosure, you can draw components such as rectangular conductors, vias, and edge ports in the Layout Manager.

You can view EM structures in 2D (double-click the EM structure node in the Project Browser) and 3D (right-click the EM structure node in the Project Browser and choose **View 3D EM Layout**), and you can view currents and electrical fields using the **Animate** buttons on the EM 3D Layout toolbar.

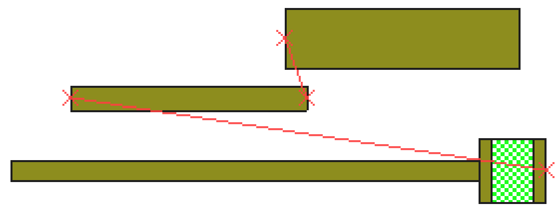


Creating a Layout with Microwave Office

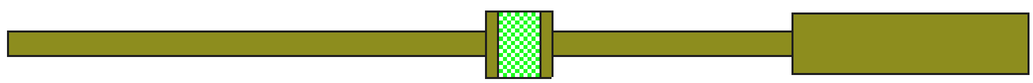
A layout is a view of the physical representation of a circuit, in which each component of the schematic is represented by a layout cell. In the object-oriented AWR Design Environment platform software, layouts are tightly integrated with the schematics and EM structures that they represent, and are simply another view of the same circuits. Any modifications to a schematic or EM structure are automatically and instantly reflected in their corresponding layouts.

To create a layout representation of a schematic, click the schematic window to make it active, then choose **View > Layout**. A layout window tab opens with an automatically-generated layout view of the schematic.

With a schematic window active, you can also click the **View Layout** button on the toolbar to view the layout of a schematic.



The resulting layout contains layout cells representing electrical components floating in the layout window. Choose **Edit > Select All** then choose **Edit > Snap Objects > Snap Together** to snap the faces of the layout cells together. The following figure shows the layout view from the previous figure after a snap together operation.

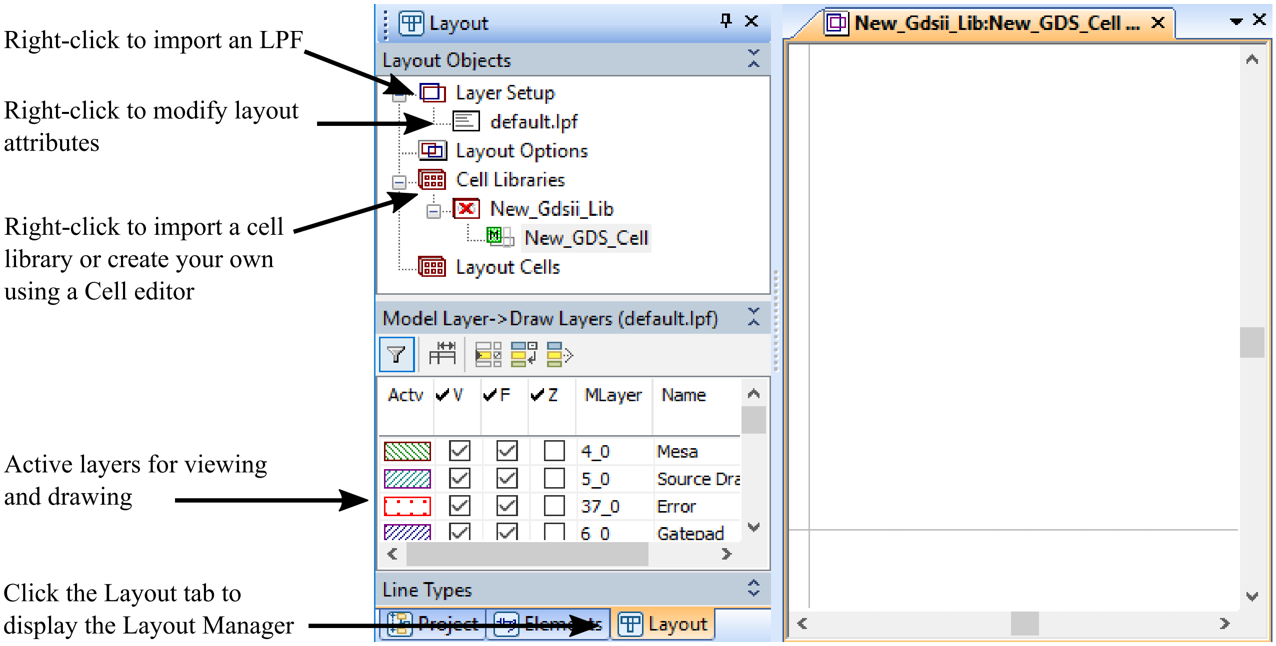


When you choose **View > View Layout**, corresponding schematic components with default layout cells are automatically generated for common electrical components such as microstrip, coplanar waveguide, and stripline elements. After the layout is generated, the schematic window displays in blue the components that do not map to default layout cells, and displays in magenta the components that do have default layout cells. You must use the Layout Manager to create or import layout cells for components without them. For more information see "Using the Layout Manager".

You can draw in the schematic layout window using the Draw tools to build substrate outlines, draw DC pads for biasing, or to add other details to the layout. In this mode, the layout is not part of a schematic element and therefore does not move as part of the snapping process.

Modifying Layout Attributes and Drawing Properties

To modify layout attributes and drawing properties, and to create new layout cells for elements without default cells, click the **Layout** tab to open the Layout Manager.

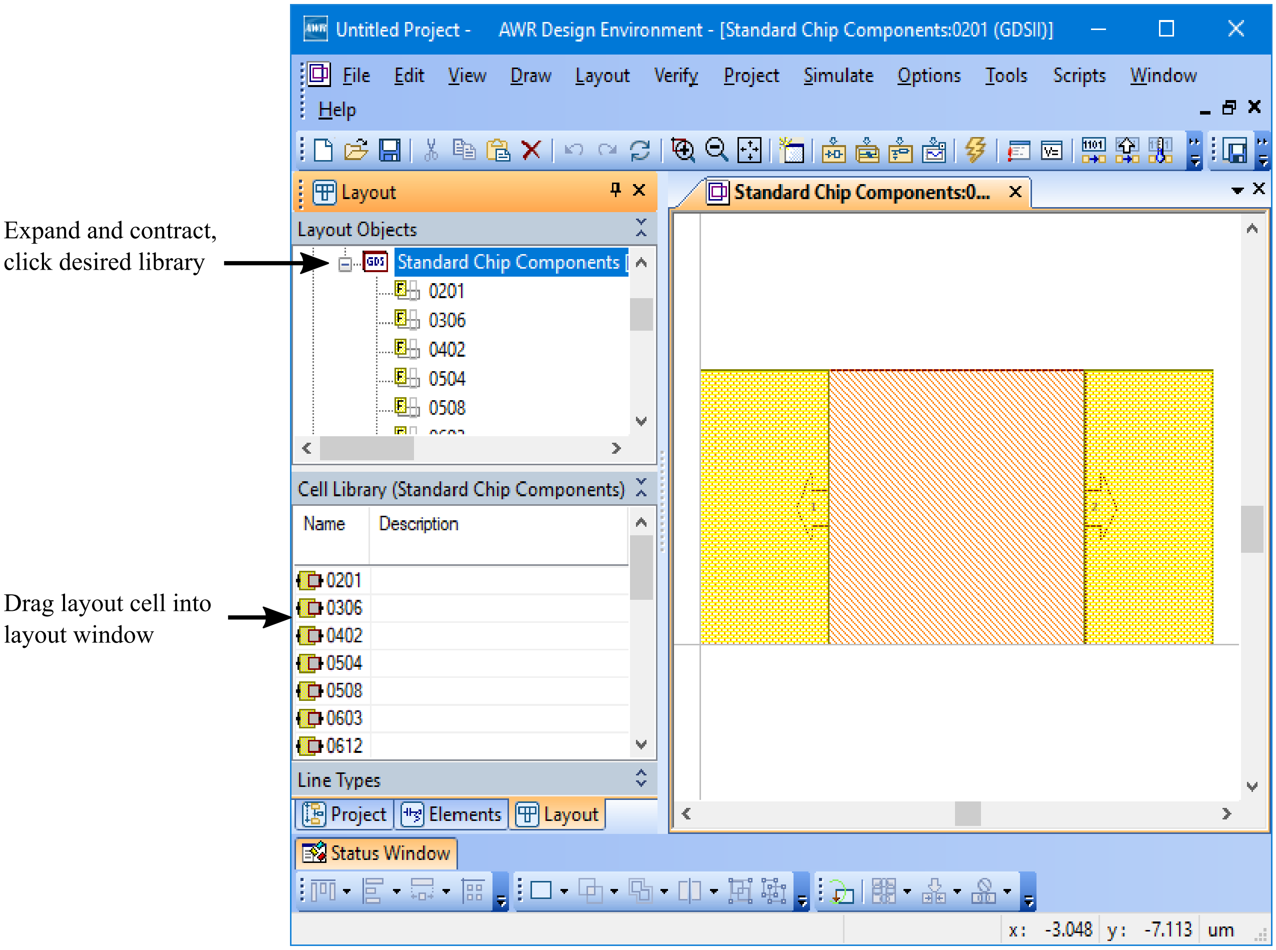


Using the Layout Manager

The **Layer Setup** node in the Layout Manager defines layout attributes such as drawing properties (for example, line color or layer pattern), 3D properties such as thickness, and layer mappings. To modify layer attributes, double-click the node (named "default.lpf" in the previous figure) below the **Layer Setup** node. You can also import a layer process file (LPF) to define these attributes by right-clicking **Layer Setup** and choosing **Import Process Definition**.

The **Cell Libraries** node in the Layout Manager allows you to create artwork cells for elements that do not have default layout cells. The powerful Cell Editor includes such features as Boolean operations for subtracting and uniting shapes, coordinate entry, array copy, arbitrary rotation, grouping, and alignment tools. You can also import artwork cell libraries such as GDSII or DXF into the AWR Design Environment platform by right-clicking the **Cell Libraries** node and choosing **Import Cell Library**.

After creating or importing cell libraries, you can browse through the libraries and select the desired layout cells to include in your layout. Click the **+** and **-** symbols to expand and contract the cell libraries, and click the desired library. The available layout cells display in the lower window pane.



After you define a cell library, you can assign cells to schematic elements. You can also use a cell directly in a schematic layout by clicking and dragging the cell into an open schematic layout window, releasing the mouse button, positioning it, and clicking to place it.

To export a schematic layout to GDSII, DXF, or Gerber formats, click the layout window to make it active, and choose **Layout > Export Layout**. To export a layout cell from the cell libraries, select the cell node in the Layout Manager, right-click and choose **Export Layout Cell**.

Creating Output Graphs and Measurements

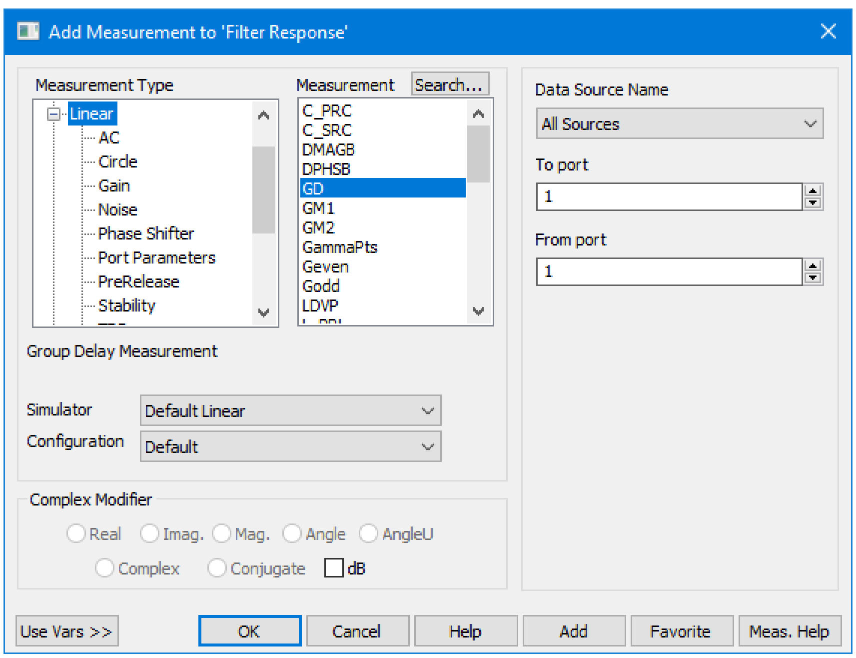
You can view the results of your circuit and system simulations in various graphical forms. Before you perform a simulation, you can create a graph, specifying the data or measurements (for example, gain, noise or scattering coefficients) that you want to plot.

To create a graph, right-click **Graphs** in the Project Browser and choose **New Graph** to display a dialog box in which to specify a graph name and graph type. An empty graph displays in the workspace and the graph name displays under **Graphs** in the Project Browser. The following graph types are available:

Graph Type	Description
Rectangular	Displays the measurement on an x-y axis, usually over frequency.

Graph Type	Description
Rectangular - Real/Imag	Displays real versus imaginary components of complex data on a rectangular graph.
Smith Chart	Displays passive impedance or admittances in a reflection coefficient chart of unit radius.
Polar	Displays the magnitude and angle of the measurement.
Histogram	Displays the measurement as a histogram.
Antenna Plot	Displays the sweep dimension of the measurement as the angle and the data dimension of the measurement as the magnitude.
Tabular	Displays the measurement in columns of numbers, usually against frequency.
Constellation	Displays the in-phase (real) versus the quadrature (imaginary) component of a complex signal.
3D Plot	Displays the measurement in a 3D graph.

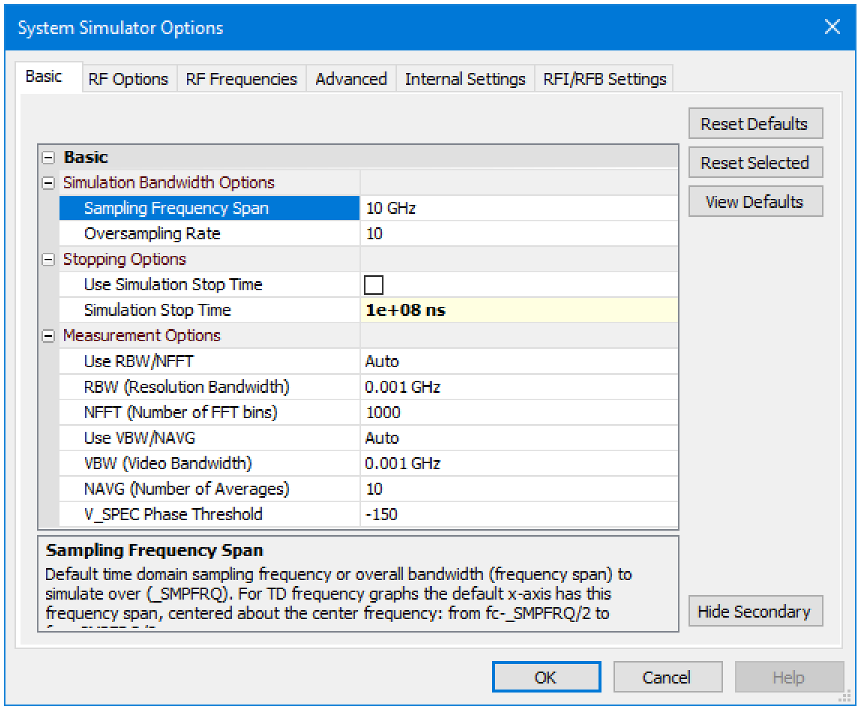
To specify the data that you want to plot, right-click the new graph name in the Project Browser and choose **Add Measurement**. An Add Measurement dialog box similar to the following displays to allow you to choose from a comprehensive list of measurements.



Setting Simulation Frequency and Performing Simulations

To set the Microwave Office simulation frequency, double-click the **Project Options** node in the Project Browser, or choose **Options > Project Options** and then specify frequency values on the **Frequencies** tab in the Project Options dialog box. By default, all the schematics use this frequency for simulation. You can overwrite this frequency with an individual schematic frequency by right-clicking the schematic name under **Circuit Schematics** in the Project Browser and choosing **Options**. Click the **Frequencies** tab, clear the **Use project defaults** check box and then specify frequency values.

To set VSS system simulation frequency, double-click the **System Diagrams** node in the Project Browser or choose **Options > Default System Options**, and then specify frequency values on the **Basic** tab in the System Simulator Options dialog box.



To run a simulation on the active project, choose **Simulate > Analyze**. The simulation runs automatically on the entire project, using the appropriate simulator (for example, linear simulator, harmonic balance nonlinear simulator, or 3D-planar EM simulator) for the different documents of the project.

When the simulation is complete, you can view the measurement output on the graphs and easily tune and/or optimize as needed.

You can perform limited simulations by right-clicking the **Graphs** node or its subnodes to simulate only the graphs that are open, only a specific graph, or simulate for just one measurement on a graph.

Tuning and Optimizing Simulations

The real-time tuner lets you see the effect on the simulation as you tune. The optimizer lets you see circuit parameter values and variables change in real-time as it works to meet the optimization goals that you specified. These features are shown in detail in the linear simulator chapter.

You can also click the **Tune Tool** button on the toolbar. Select the parameters you want to tune and then click the **Tune** button to tune the values. *As you tune or optimize, the schematics and associated layouts are automatically updated!* When you re-run the simulation, only the modified portions of the project are recalculated.

Using Command Shortcuts

The use of keyboard command shortcuts (or hotkeys) can greatly increase efficiency within the AWR Design Environment platform. Default menu command shortcuts are available for many common actions such as simulation, optimization, and navigating between the Project Browser, Elements Browser and Layout Manager. Default shortcuts display on menus or by choosing **Tools > Hotkeys** to display the Customize dialog box where you can also create custom hotkeys.

Using Scripts and Wizards

Scripts and wizards allow you to automate and extend AWR Design Environment platform functions through customization. These features are implemented via the Microwave Office API, a COM automation-compliant server that can be programmed in any non-proprietary language such as C, Visual Basic™, or Java.

Scripts are Visual Basic programs that you can write to do things such as automate schematic-building tasks within the AWR Design Environment platform software. To access scripts, choose **Tools > Scripting Editor** or any of the options on the **Scripts** menu.

Wizards are Dynamic Link Library (DLL) files that you can author to create add-on tools for the AWR Design Environment platform; for example, a filter synthesis tool or load pull tool. Wizards display under the **Wizards** node in the Project Browser.

Using Online Help

Online Help provides information on the windows, menu choices, and dialog boxes in the AWR Design Environment platform, as well as for design concepts.

To access online Help, choose **Help** from the main menu bar or press the **F1** key anytime during design creation. The Help topic that displays is context sensitive-- it depends on the active window and/or type of object selected. The following are examples:

- Active window = graph, Help topic = "Working with Graphs".
- Active window = schematic (with nothing selected), Help topic = "Schematics and System Diagrams in the Project Browser".
- Active window = schematic (with an element selected), Help topic = the Help page for that element.
- Active window = schematic (with an equation selected), Help topic = "Equation Syntax".
- Active window = schematic layout (with nothing selected), Help topic = "Layout Editing".

Context sensitive Help is also available by:

- clicking the Help button in most dialog boxes
- right-clicking a model or system block in the Elements Browser and choosing **Element Help**, or selecting an element in a schematic or a system block in a system diagram and pressing **F1**, or clicking the **Element Help** button in the Element Options dialog box.
- clicking the **Meas Help** button in the Add/Modify Measurement dialog box

Chapter 3. ANA: Using the Analyst 3D Electromagnetic Simulator

The Cadence® Analyst™ 3D FEM EM analysis solver is based on the finite element method. As with any FEM field solver, the basic steps include constructing the geometry, setting up boundary and source (excitation) conditions, configuring the frequency and other solver options, and finally examining the mesh and simulation results. Before performing the detailed procedures in this and subsequent chapters, it is important to acquire a general understanding of how and when to use the Analyst software instead of the Cadence AXIEM® 3D planar EM analysis software.

AXIEM analysis is based on the method of moment (MoM). The algorithm implemented in AXIEM software limits its application to 3D planar shapes. The term "3D planar" simply means that the shapes (geometries) are constructed in a series of planes (parallel to the horizontal x-y plane) and extruded along the third dimension (vertical z-axis). The boundary faces of all geometries have the normal direction either parallel or perpendicular to the z-axis. If the metal geometry you want to model has a boundary face that is tilted against the z-axis (for example: bond wires, ball grid arrays, solder bumps, connectors, waveguides, cavity resonators, and RF packages), then you should use Analyst software. In addition, AXIEM analysis also assumes that the dielectric is uniform and extends to infinity along the x-y directions and piecewise uniform along the z-direction. If you want to model a finite dielectric substrate (like a patch antenna), a finite dielectric brick (like a dielectric resonator), a small air "void" (as beneath an inductor), or a multi-technology design (like a GaAs chip on a PCB), then AXIEM analysis is not applicable. In addition, one of the reasons that the AXIEM solver is usually more efficient than 3D FEM solvers is that it meshes and models only the conductors, not the dielectrics. However, if the geometry contains many ground planes and conductor vias, the volume occupied by conductor is similar to that of dielectric, the number of unknowns that AXIEM software needs to solve for might be similar to that of the FEM solver, then Analyst software might be faster. Lastly, with the exception that AXIEM software offers certain port types that are not directly supported in Analyst software (though conversion is possible), any AXIEM EM structure can be easily converted to use the Analyst software simulator and you can compare the answers from MoM and FEM.

Another important consideration when comparing Analyst and AXIEM software is how to construct the geometry for simulation in Analyst software. The Cadence AWR Design Environment® platform offers many convenient and flexible ways that have attracted increasing interest from designers:

1. 3D planar shapes can be constructed as a Cadence Microwave Office® software layout. You start with defining a dielectric stackup, and then draw planar shapes on a series of planes ("drawing layers"). These planar shapes are extruded along the z-direction through the definition of conductor thickness or via extension depth. The details of constructing 3D planar layout are covered in the *Microwave Office Getting Started Guide* chapters. Assuming that you have constructed a 3D planar geometry in AXIEM (also called "AXIEM structure" or "AXIEM document"), you can convert it into an "Analyst structure" (or "Analyst document") in a few steps, as covered later in this chapter.
2. Geometries of a mixed technology design can be constructed by "lumping" together 3D planar shapes defined with different underlying stackups (and more precisely, layer processing files (LPFs)) through "hierarchy". Basically, you place one structure (EM document) as the "subcircuit" or "child structure" of another structure (EM document). You then align them by specifying the z-position of the bottom face of the "subcircuit", as discussed in *"ANA: Hierarchy and 3D Parts in Analyst "*.
3. Several commonly used three-dimensional shapes can be placed into planar layout directly from within Microwave Office. These are called 3D parameterized cells ("3D pCells") that include bondwire, ball grid array (BGA), ribbon, tapered via, QFN package, and others. Details for adding a bondwire p-cell are discussed in *"ANA: Hierarchy and 3D Parts in Analyst "*.
4. You can use the Analyst 3D editor to draw 3D geometry. You can draw commonly seen primitive geometries (such as spheres, cylinders, and cubes) directly, and Boolean operation of 3D shapes can be performed using the Analyst 3D Editor. See "Encapsulating the Chip and Bond Wires" for details.
5. You can place arbitrary 3D EM elements from user-customizable libraries into an Analyst structure. Sample elements such as coaxial lines, SMA connectors and helical coil inductors are included in the AWR Design Environment

platform installation. *"ANA: Using Arbitrary 3D Structures in Analyst"* discusses the steps for including SMA in a model. You can construct and add your own elements to the library; see the *AWR Design Environment User Guide* for details. (The 3D pCells are different because they are constructed by calling codes compiled within Microwave Office. The basic shape of a 3D pCell is fixed, while you can change values of certain dimensions (parameters). Arbitrary 3D EM elements are constructed by executing recordable, user-modifiable scripts from Analyst 3D Editor.)

6. Three-dimensional geometries constructed in other software can be imported into an Analyst structure through intermediate files (ACIS SAT file or STEP file). You need to define the material for individual shapes after importing. Importing an example SAT file is discussed in *"ANA: Importing SAT Files in Analyst"*.
7. The **Layout > Copy to Arbitrary 3D EM Structure** command provides an easy way to convert an EM structure constructed in the Microwave Office EM Layout Editor into an arbitrary 3D structure that you can open and modify in the Analyst 3D Editor. Material, port, and boundary definitions are preserved. The 3D structure shows the effects of layer processing and shape modifier rules. This feature is useful in various scenarios. For example, it is easier to examine in the Analyst 3D Editor whether there is any gap between the starting point of a bondwire and the contacting metal trace.

With these key points in mind, you are better prepared for the detailed steps in the following procedures. As you perform the steps, consider why you are doing them and how you might design the user interface differently. This mindset allows you to drive the software rather than simply following instructions. After getting your first result, you will start to ask important questions such as how to obtain the best accuracy or fastest simulation speed. The best way to answer these questions is to examine the examples and experiment with different port settings, boundary condition types, and solver options. To get started, you need to know where these controlling parameters are set in the user interface.

Creating a Simulation for a Simple EM Structure

This example demonstrates how to set up a simulation for an Analyst 3D EM document. Basic setup concepts already covered in previous chapters are not included, so you should familiarize yourself with these before starting.

This example includes the following main steps:

- Installing the Analyst 3D EM simulator
- Converting an AXIEM structure to an Analyst structure
- Properly configuring ports and the enclosure
- Viewing the initial mesh
- Running a simulation

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the AWR Design Environment platform. Choose **Help > Quick Reference** to access this document.

Opening an Existing Project

The example you create in this chapter is available in its complete form as *Analyst_Basic_Finish.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl**-click the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box.

To create a project:

1. Choose **File > Open Example** to locate and open the *Analyst_Basic_Start.emp* file.
2. Choose **File > Save Project As**. The Save As dialog box displays.

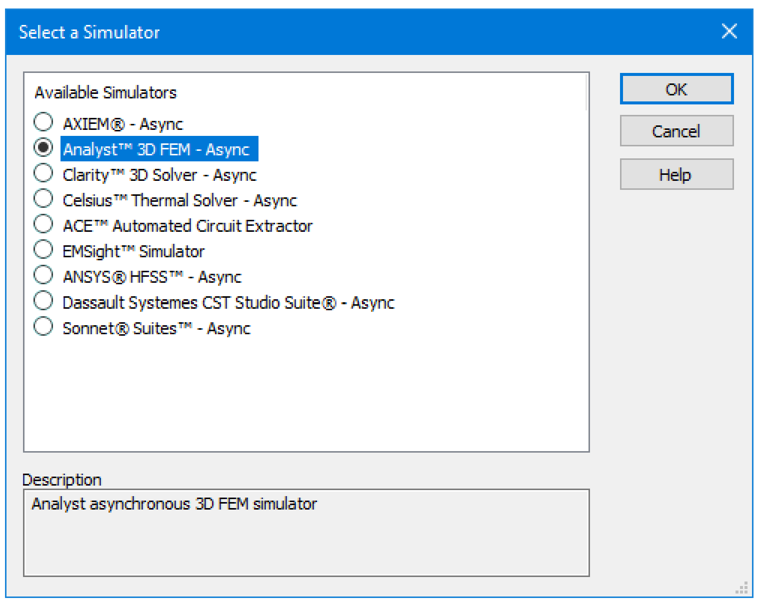
3. Navigate to the directory in which you want to save the project, type "**Analyst_GS_basic**" as the project name, and then click **Save**.

NOTE: Simulation results may vary slightly from the images in this guide. Finite Element Method (FEM) simulations require a convergence based on a mesh refinement sequence. Slight changes in the mesh refinement between versions of the solver can cause results to vary slightly. While the default convergence tolerance is sufficient for most geometries, if you find results shift you can decrease the convergence tolerance to ensure that the results are accurate.

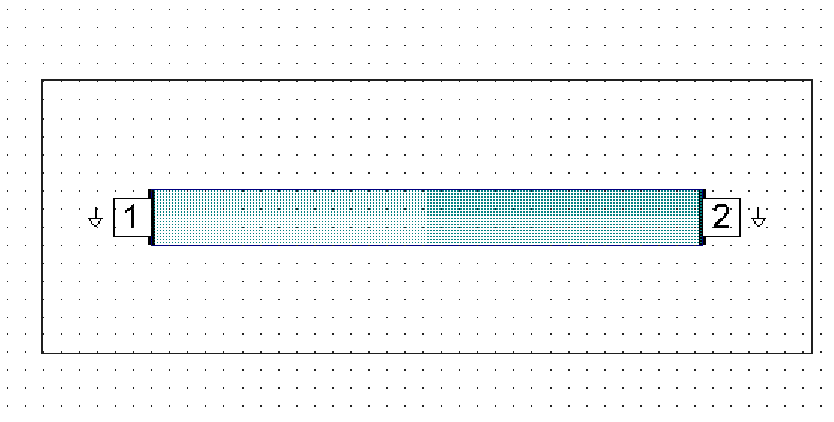
Converting the AXIEM Structure to Analyst

To convert the AXIEM structure to an Analyst structure:

1. Simulate the project to see the "AXIEM_Line" structure simulate. After simulation, view the data on the "S Parameters" Smith Chart.
2. Select the "AXIEM_Line" EM structure in the Project Browser and choose **Edit > Copy** and **Edit > Paste** or press **Ctrl + D** to duplicate the structure. The new structure displays in a new window. Make sure you close this window.
3. In the Project Browser, right-click the copied EM document and choose **Rename**. Type "**Analyst_Line**" as the new name and then click the **Rename** button.
4. In the Project Browser, right-click "Analyst_Line" and choose **Set Simulator**. In the Select a Simulator dialog box select **Analyst™ 3D EM - Async** and then click **OK**.



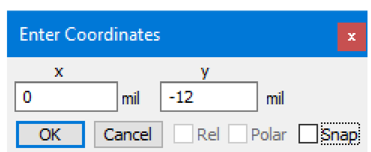
Double-click the "Analyst_Line" EM structure. The Analyst structure displays as shown in the following figure.



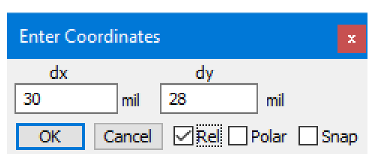
Like most finite element simulators, Analyst software requires well-defined boundary conditions and a 3D bounding box. Analyst software also requires that you configure the top, bottom, *and* side enclosure boundary conditions for the structure. A shape in the 2D layout defines the boundary size, then you edit the shape properties to specify the boundary condition for each side of the boundary. Each Analyst structure has a default 2D rectangular boundary shape drawn with the approximate open boundary conditions on each edge; you can edit the existing shape or add a new shape of any size. This example draws a new boundary shape to replace the default boundary shape. The default height of this boundary shape is specified in the enclosure and is defined in the stackup for this structure.

To draw a shape as the boundary:

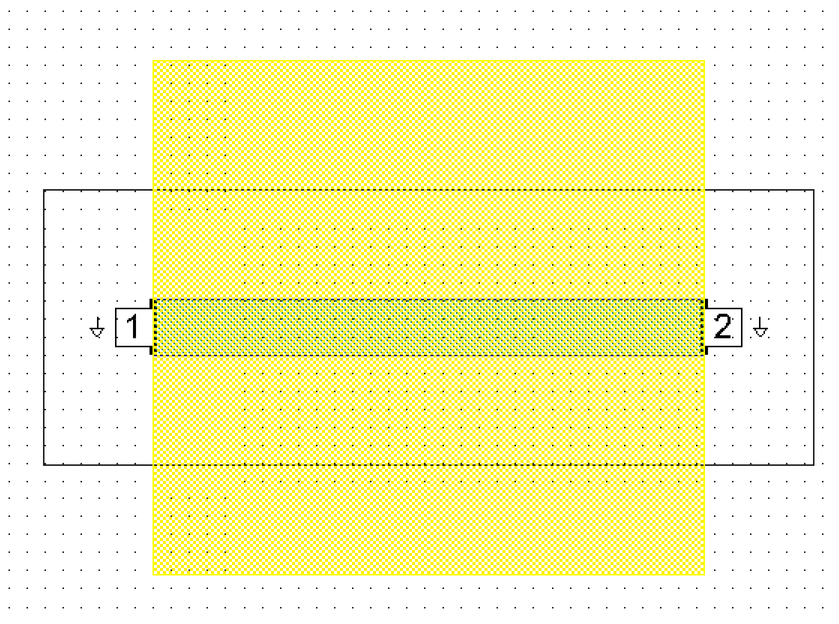
1. With the "Analyst_Line" document window active, choose **Draw > Rectangle**.
2. With the cursor in the window, press the **Tab** key to display the Enter Coordinates dialog box, then type the values show in the following figure and click **OK**.



3. Press the **Tab** key again and type the following values, then click **OK**. Note the **Rel** setting.

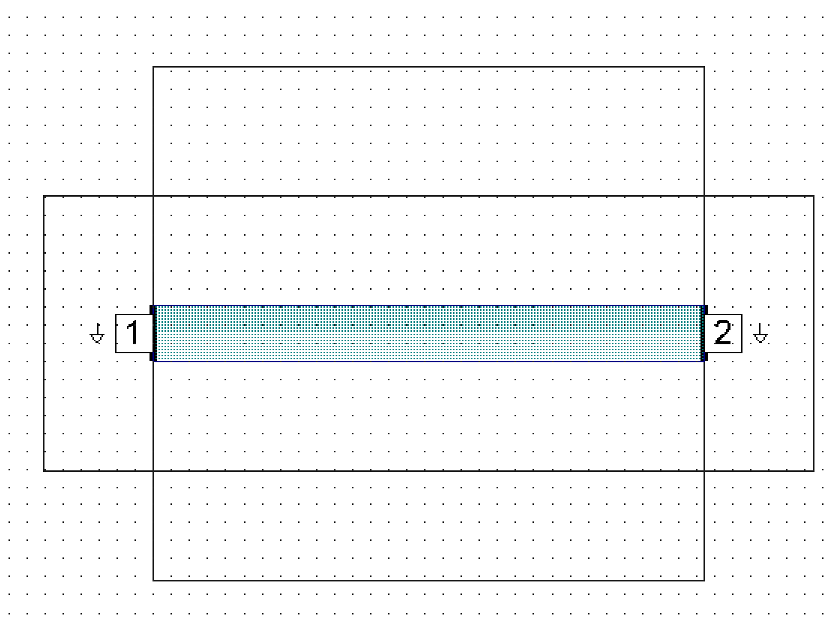


The layout displays as shown in the following figure.



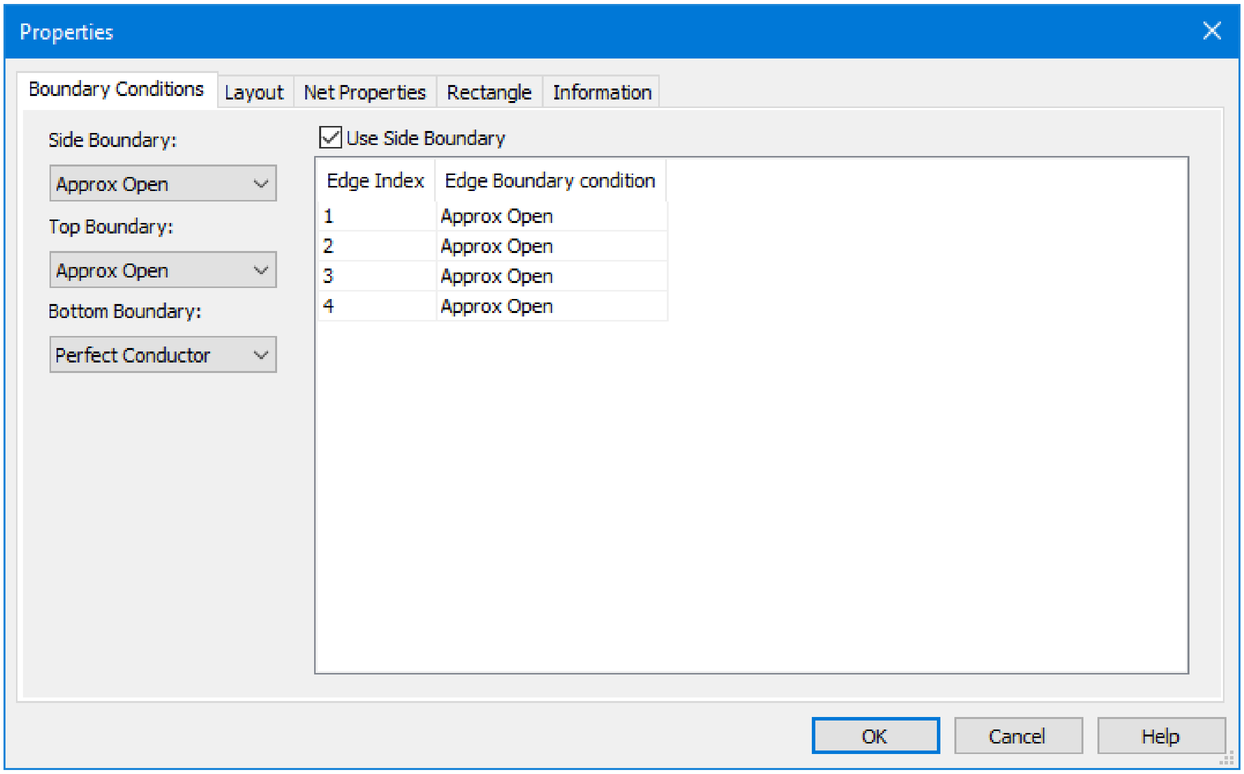
The new shape is automatically selected.

4. Choose **Draw > Create 3D EM Simulation Boundary** or click the **Create Simulation Boundary** button on the toolbar to create a new boundary shape in the layout.
5. The layout displays as shown in the following figure.

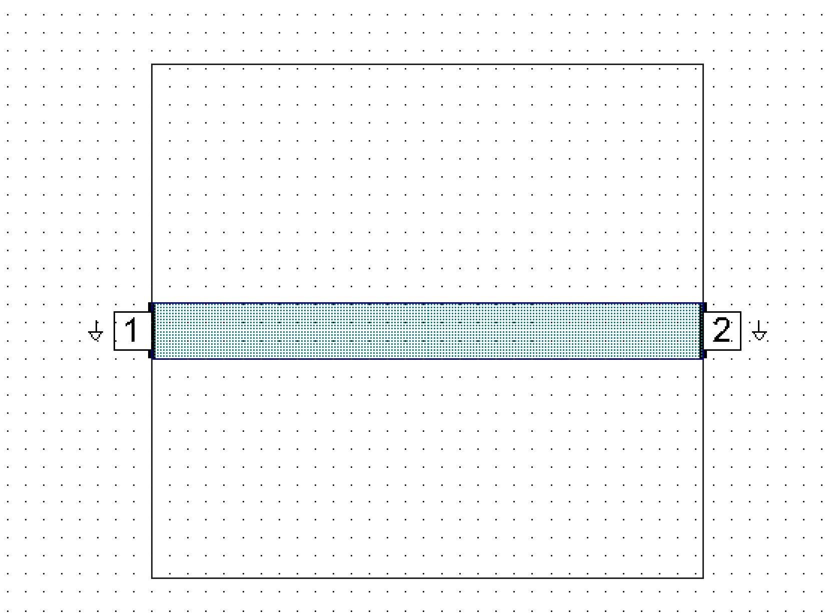


Notice that there are two boundary shapes. Select the original boundary shape (which extends in the x-direction beyond the end of the line) and press the **Delete** key.

6. Select the remaining boundary shape, right-click and choose **Shape Properties** to open the Properties dialog box. Ensure that the boundary conditions match this figure.

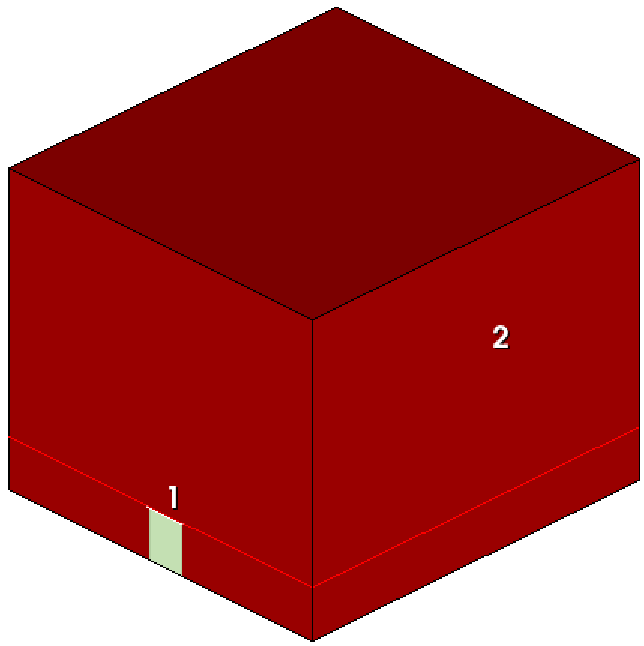


The layout displays as shown in the following figure.

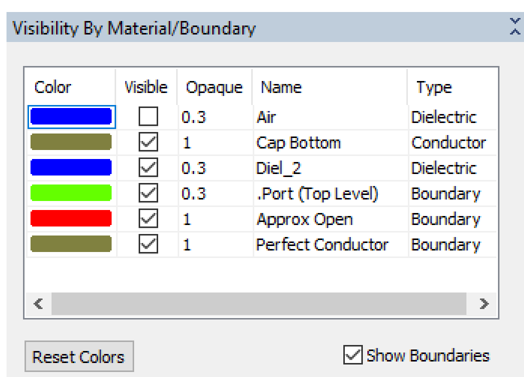


You can view the boundary conditions of the structure in the 3D view of the structure to verify that they are correct.

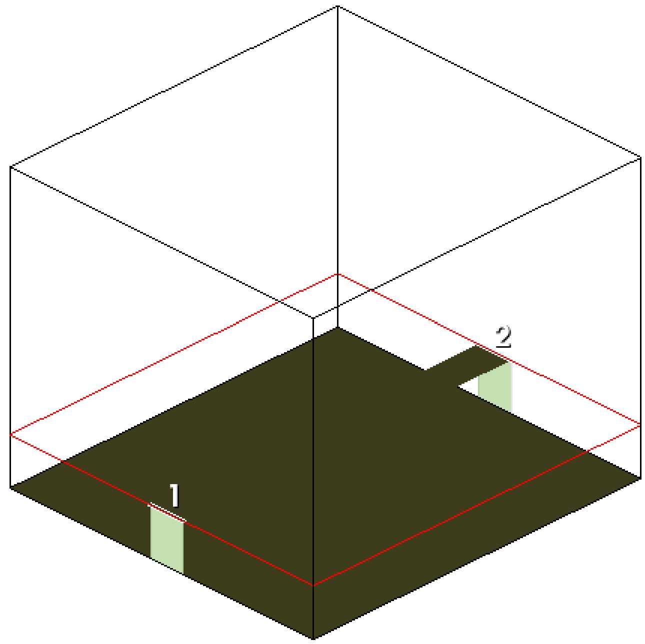
1. Open the 3D layout of the structure by choosing **View > View 3D EM Layout** or by clicking the **View EM 3D Layout** button on the EM 3D Layout toolbar.
2. Click the **Show Boundary Conditions** button on the EM 3D Layout toolbar. The layout displays the boundary conditions as shown in the following figure.



3. Click the **Layout** tab to open the Layout Manager, then click the arrows on the right of the **Visibility By Material/Boundary** pane to open it.



You can turn off the visibility for specific materials. For example, when turning off the **Approx Open** material, the 3D layout displays without those boundary conditions visible.

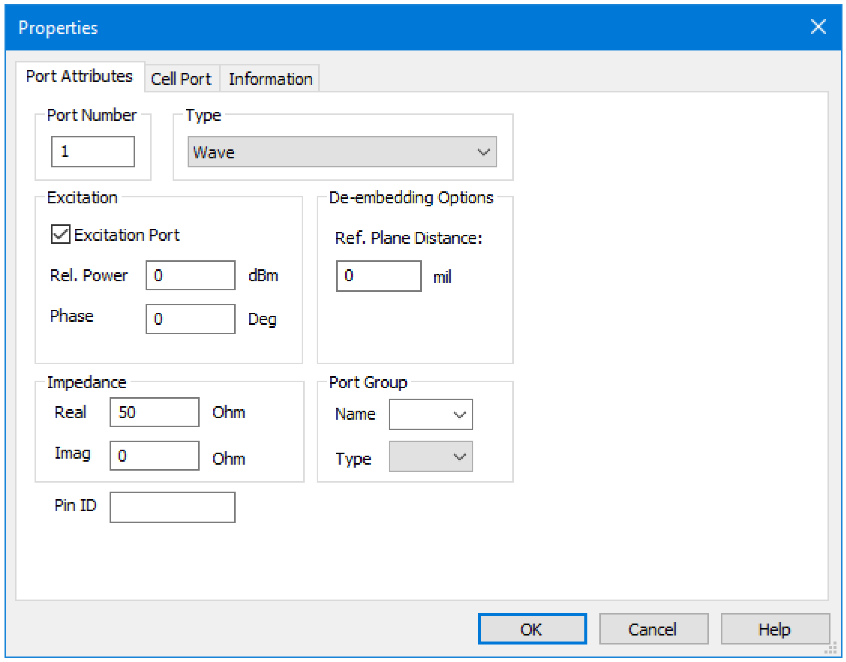


4. Click the **Show Boundary Conditions** button on the EM 3D Layout toolbar to turn off this display for the next several steps.

Analyst software supports wave ports and lumped ports. Wave ports are generally preferred. You can attach wave and lumped ports to shapes on or inside the simulation boundary.

To configure wave ports for this structure:

1. With the 2D Layout View window active, double-click a port.
2. In the Properties dialog box on the **Port Attributes** tab, make sure that **Type** is set to **Wave** and then click **OK**.



3. Repeat these steps for the other port.

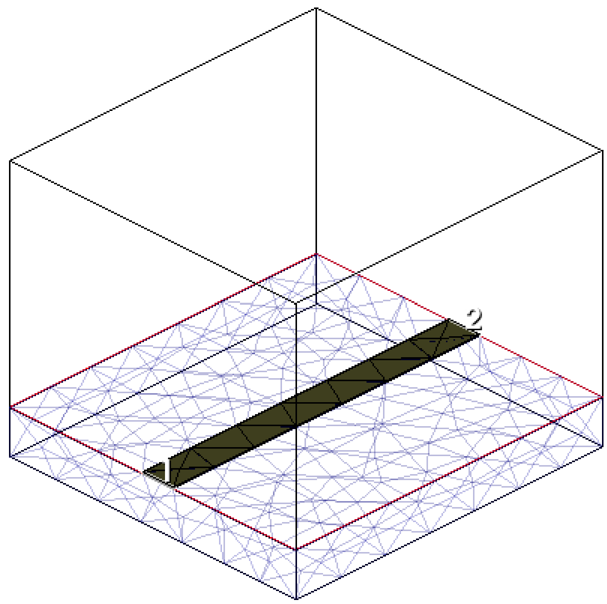
Running the Simulation

The Analyst structure is ready for simulation from 0.1 to 10 GHz in 1 GHz steps.

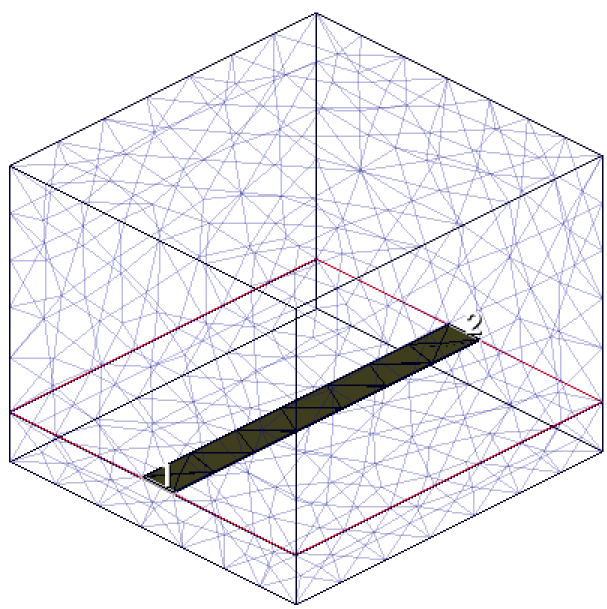
Before simulating, you can view the mesh in the 3D view of the structure. The first mesh you see is the initial solver's mesh. As each additional Adaptive Mesh Refinement (AMR) step continues, the view of the mesh updates.

To view the mesh:

1. Open the 3D layout of the structure by choosing **View > View 3D EM Layout** or by clicking the **View EM 3D Layout** button on the EM 3D Layout toolbar.
2. Click the **Show 3D Mesh** button on the EM 3D Layout toolbar. The layout displays as shown in the following figure. This is the initial mesh of the structure with **Perfect Conductor** material turned off.



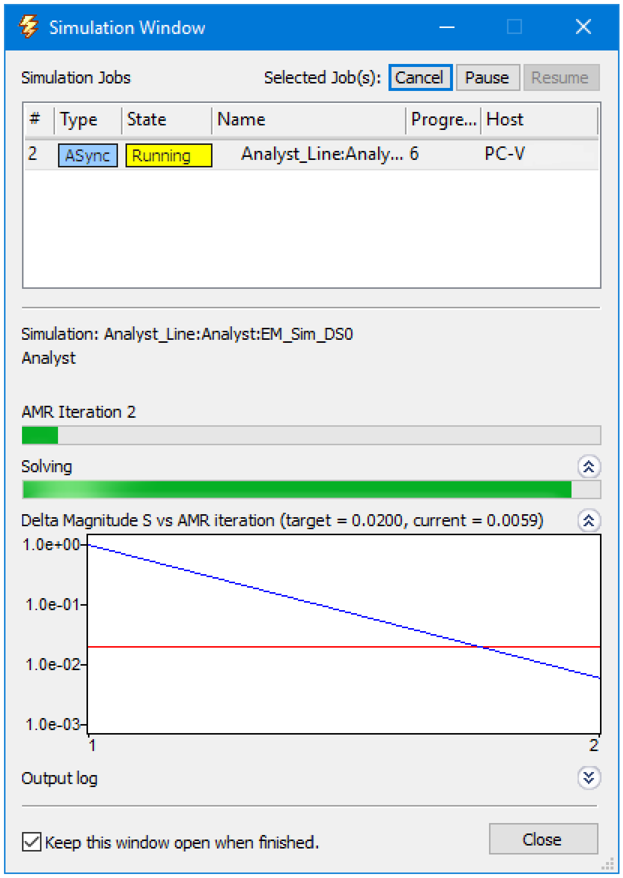
Additionally, the display for the Air layer is turned off by default. You can turn it on in the Layout Manager by opening the **Visibility By Material/Boundary** pane. The mesh is visible in the Air layer as shown in the following figure.



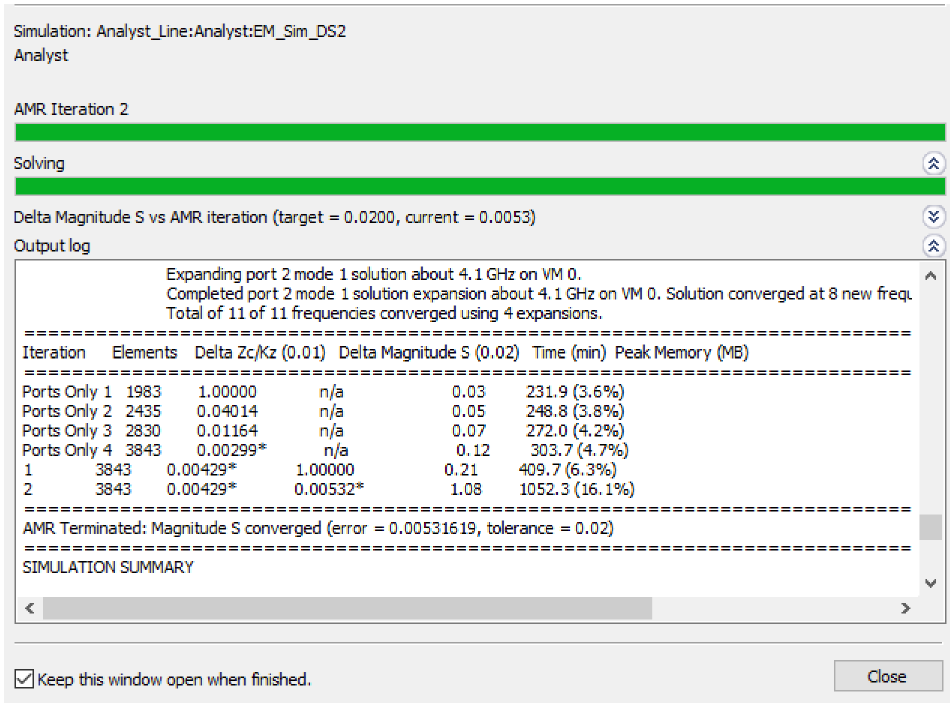
To simulate the structure:

1. Choose **Simulate > Analyze** or click the **Analyze** button on the toolbar. **NOTE:** Typically this simulation takes a few minutes to run with no other programs competing for resources.

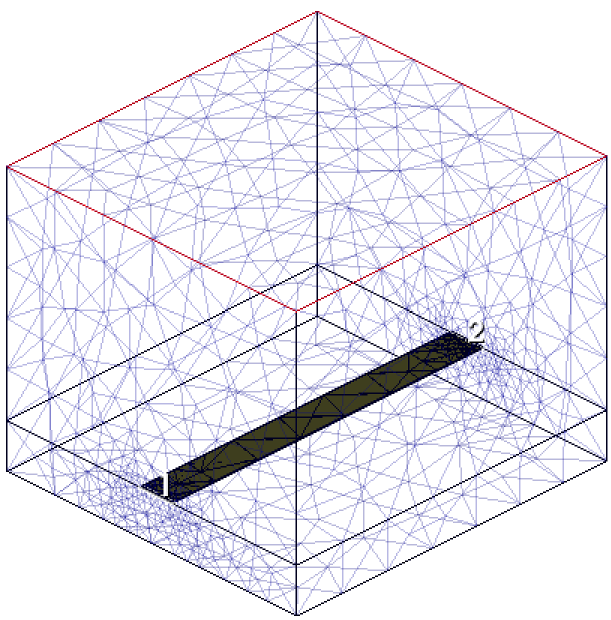
2. The Simulation dialog box displays the status during simulation.



Click the down arrow icon on the right of the dialog box to display details about the simulation progress. The following figure shows the output log from this simulation. You can watch each AMR sequence and see how close the simulation is to converging. Simulation times and memory use vary by computer.



3. As the simulation progresses, the mesh is refined. If the 3D layout is visible after viewing the mesh, you see the mesh update after each AMR sequence. The following figure shows the mesh after the AMR sequence converges with the Air layer visibility turned off.

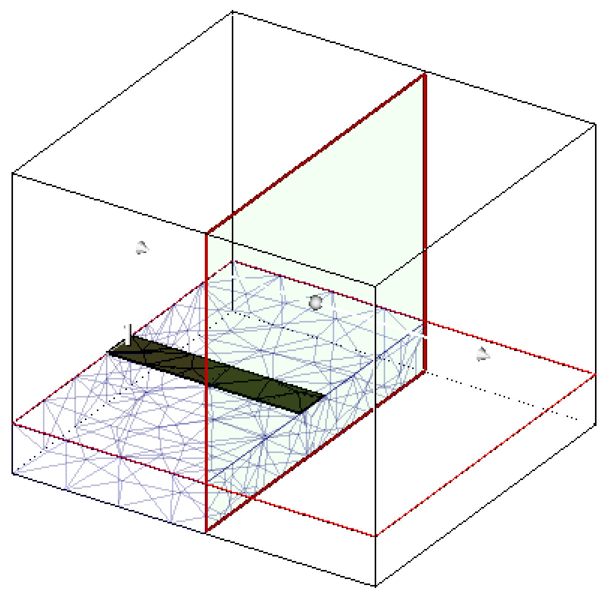


NOTE: While the simulation is running you can continue this exercise as the Analyst simulator is asynchronous, like the AXIEM simulator.

It is useful to know how to use cut planes when viewing information such as mesh, boundary conditions, and E-fields in the EM structure 3D view.

To use a cut plane:

1. Open the 3D layout of the structure by choosing **View > View 3D EM Layout** or by clicking the **View EM 3D Layout** button on the EM 3D Layout toolbar.
2. Click the **Use Cut Plane** button on the toolbar and ensure that the **Show Cut Plane** button is also pressed. The following figure shows a plane that cuts the structure where the information is drawn on one side and removed from the other side.

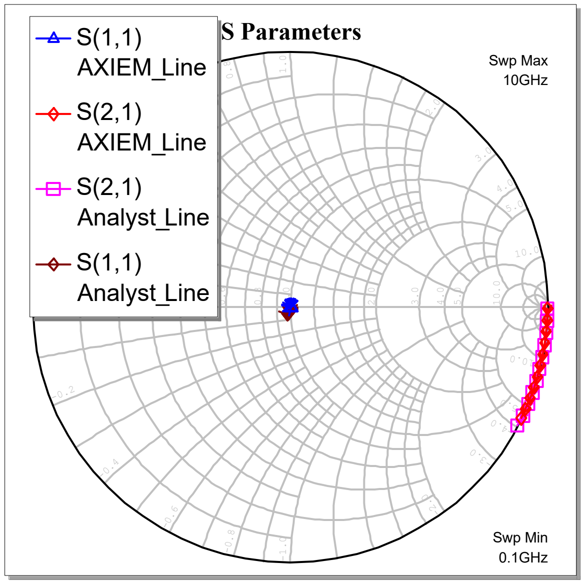


3. Click and drag on the cut plane to move its position. Click and drag on the arrows at the end of the line through the cut plane to change the orientation. Use the **x**, **y**, or **z** keys to move the cut plane in that orientation and make the plane orthogonal to that plane. If this is not obvious, move the cut plane to an odd angle by moving the arrow that extends beyond the plane, and then use these keys to observe the cut plane.

To view simulation results:

1. In the Project Browser, right-click the "AXIEM_Line:S(2,1)" measurement on the "S Parameters" graph and choose **Duplicate** to display the Modify Measurement dialog box.
2. Change the **Data Source Name** to "Analyst_Line" and then click **OK**.
3. Repeat these steps for the "AXIEM_Line:(S(1,1)" measurement.
4. After the simulation is complete you might need to resimulate the project to update the graph data. Note that the structure does not resimulate because nothing has changed.

Double-click the "S Parameters" graph in the Project Browser. When the graph data is updated, the graph displays similar to the following figure.



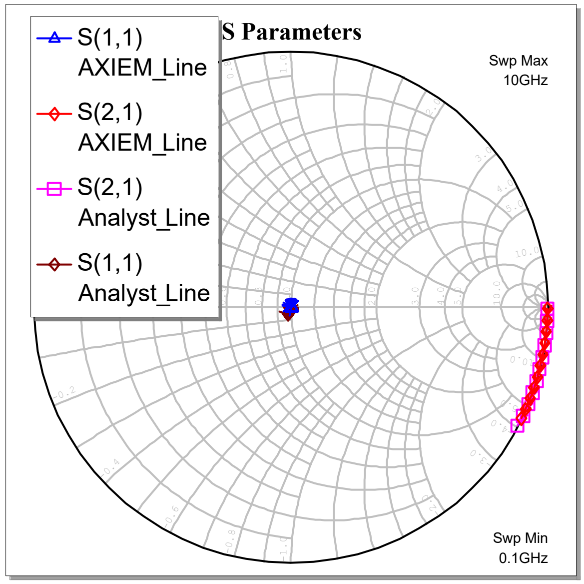
Some structures with iterative solvers (like FEM solvers) might have two sets of S-parameters between AMR iterations that can be close enough to one another to satisfy the convergence criteria. In actuality, the mesh may not have refined enough to catch certain behaviors. If this occurs, you can lower the convergence criteria to force the simulator to take more AMR iterations to try to capture this behavior, although this restarts the simulation from AMR iteration 1.

Another possibly more efficient way than lowering the convergence tolerance and simulating the entire structure again, is to force the simulator to run one more AMR iteration to ensure that the simulation did converge. This simulation starts where the last simulation ended and does not need to simulate the prior AMR iterations.

To run one more AMR iteration, right-click the "Analyst_Line" EM structure and choose **Refine Solution**. A simulation starts on the next AMR iteration.

You can refine the solution multiple times if the results shift. This potentially saves a lot of time re-running full simulations after lowering the convergence tolerance.

For this particular structure the results don't really change so you know the solution has converged.


S Parameters
S(1,1)
AXIEM_Line
S(2,1)
AXIEM_Line
S(2,1)
Analyst_Line
S(1,1)
Analyst_Line
Swp Max
10GHz
Swp Min
0.1GHz

Chapter 4. ANA: Hierarchy and 3D Parts in Analyst

Hierarchy is a powerful concept when using Cadence® Analyst™ 3D FEM EM analysis software in the Cadence AWR Design Environment® platform. By using hierarchy, you can greatly simplify a dielectric stackup setup. Each EM document can have its own dielectric stackup. When you combine them using hierarchy (one is added as a subcircuit to the other), you only need to specify the appropriate location in the Z direction, and the program properly simulates the combined dielectric stackup of both structures, effectively combining the stackups. This approach offers these advantages:

- You do not have to define one complete (and sometimes complicated) dielectric stackup for your entire structure, typically before you add any geometry.
- You can easily add more dielectric layers at later stages in the structure simulation process.

Building a Hierarchical Simulation

This chapter is a continuation of *"ANA: Using the Analyst 3D Electromagnetic Simulator"*. If you have not completed that example please do so before continuing here, as it includes concepts that may not be discussed in this example.

This chapter demonstrates how to build hierarchical simulations in the Analyst simulator as well as using predefined parameterized cells in the geometry.

The example in this chapter includes the following main steps:

- Setting up a hierarchical design
- Adding bond wire 3D parameterized cells (pCells)

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the AWR Design Environment platform. Choose **Help > Quick Reference** to access this document.

Opening an Existing Project

The example you create in this chapter is available in its complete form as *Analyst_Hierarchy_Finish.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl**-click the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. This example is a continuation of the previous Analyst chapter.

To open an existing project:

1. Choose **File > Open Example** to locate and open the *Analyst_Basic_Finish.emp* file.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**Analyst_GS_hierarchy**" as the project name, and then click **Save**.

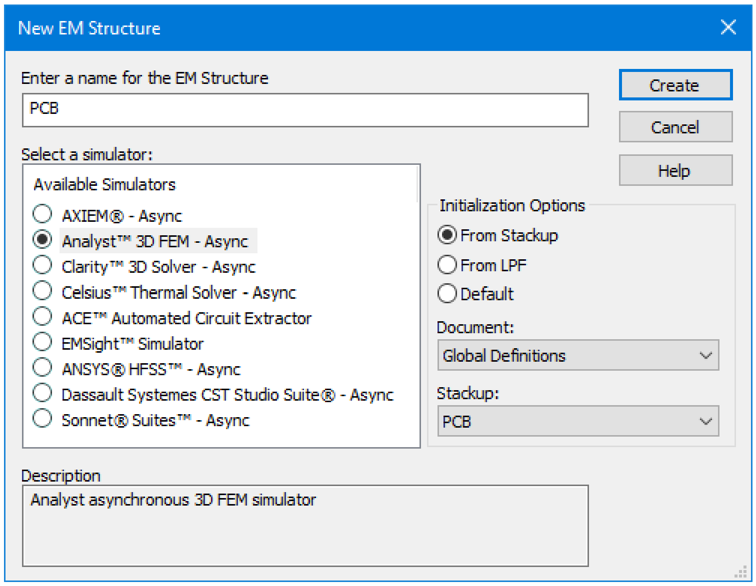
NOTE: Simulation results may vary slightly from the images in this guide. Finite Element Method (FEM) simulations require a convergence based on a mesh refinement sequence. Slight changes in the mesh refinement between versions of the solver can cause results to vary slightly. While the default convergence tolerance is sufficient for most geometries, if you find results shift you can decrease the convergence tolerance to ensure that the results are accurate.

Creating an Analyst EM Structure

To create an Analyst EM structure:

1. Right-click the **EM Structures** node in the Project Browser and choose **New EM Structure**. (Alternatively, choose **Project > Add EM Structure > New EM Structure** or click the **Add New EM Structure** button on the toolbar).
2. In the New EM Structure dialog box, type "**PCB**" as the structure name, select **Analyst™ 3D FEM - Async** as the simulator, select **From Stackup** under **Initialization Options**, select **PCB** as the **Stackup**, and then click **Create**.

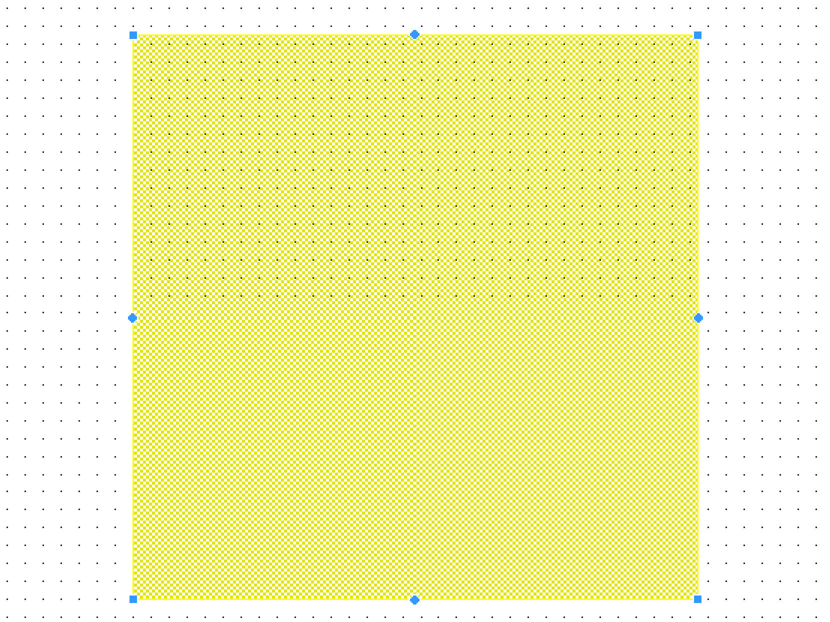
   NOTE: The STACKUP element named "PCB" in the Global Definitions contains all of the setup for this EM structure so this example can focus on Analyst-specific topics.

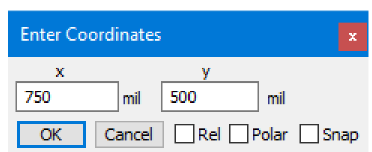
   An EM structure layout window displays.
3. In the Project Browser, double-click the **Enclosure** node under the "PCB" EM structure to display the Element Options - ENCLOSURE Properties dialog box. Set the **Grid_X** and **Grid_Y** to "**1**" mil to set the drawing grid for the EM document (it has no impact on the Analyst software simulation).

By default, a square boundary shape is added when the structure is created. To configure the boundary for this example:

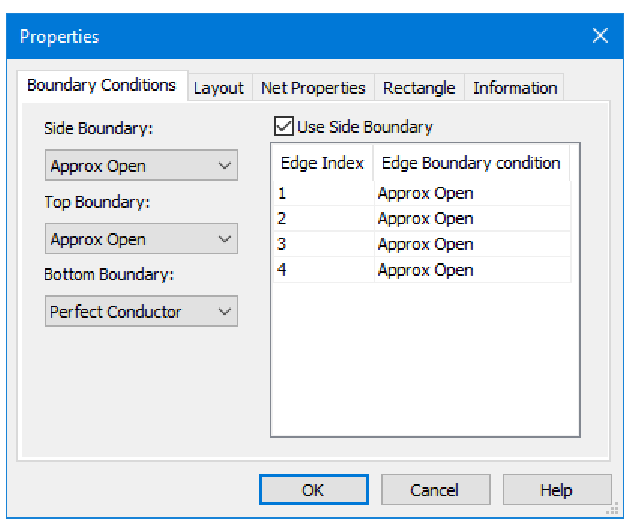
1. Make the 2D Layout View window active (it should be the active window after creating the EM document).
2. Double-click the boundary shape to enter edit mode. Diamonds/squares display at the vertices and mid-points of each boundary edge as shown in the following figure.



3. While clicking and dragging the upper right corner of the square, press the **Tab** key. In the Enter Coordinates dialog box, type the values shown in the following figure to make a boundary shape that is 750 x 500 mil. Note that the **Rel** check box is **NOT** selected.



4. Right-click the 2D layout and choose **View All** to see the entire boundary shape.
5. Select the boundary shape, right-click and choose **Shape Properties** to display the Properties dialog box. Click the **Boundary Conditions** tab and ensure that the settings match those in the following figure.

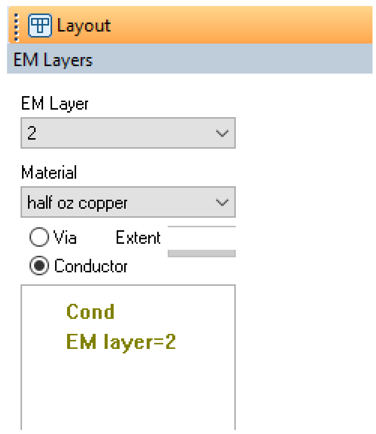


The **Use Side Boundary** check box forces each side of the shape to use the same setting as the **Side Boundary** setting. If you clear this check box, you can specify the boundary condition for each side of the boundary shape. **NOTE:** You can also set boundary information in the Element Options - ENCLOSURE Properties dialog box by double-clicking the **Enclosure** node under the EM document name in the Project Browser and clicking the **Boundary Conditions** tab.

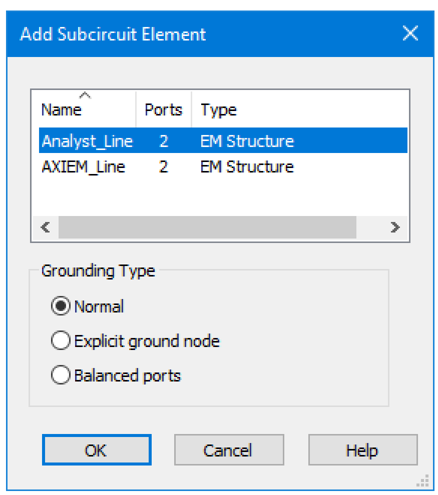
6. Right-click the "PCB" EM structure in the Project Browser and choose **Options** to display the Options dialog box, then click the **Frequencies** tab. Notice that the frequency range is already set with the values copied from the project frequencies. The **Mesh** and **Analyst** tabs have options for changing mesh or Analyst solver options if needed.

You now need to add the "Analyst_Line" EM document to the "PCB" EM document. To build hierarchy for this example:

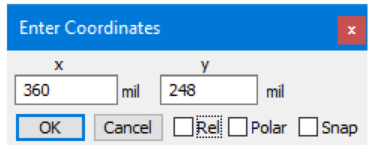
1. Click the **Layout** tab to open the Layout Manager, then click to open the **EM Layers** pane.



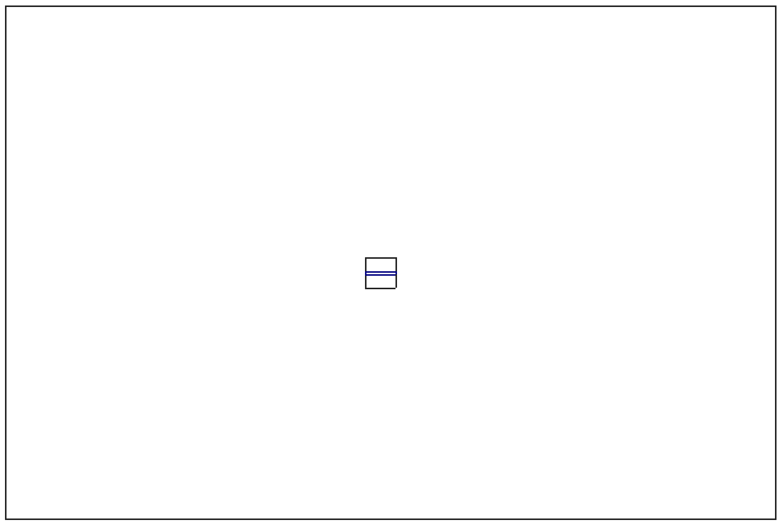
2. To set the initial Z location of the subcircuit, ensure that **EM Layer** is set to "2".
3. With the "PCB" EM structure layout window active, choose **Draw > Add Subcircuit** to open the Add Subcircuit Element dialog box. Select "Analyst_Line" and then click **OK**.



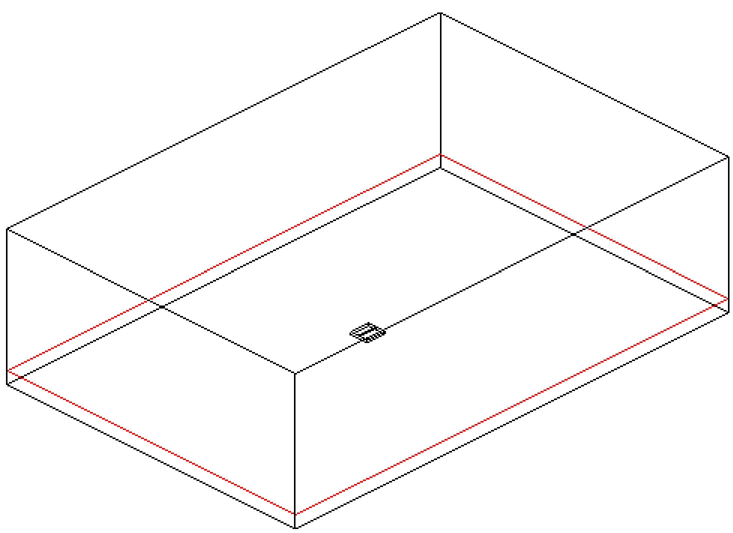
4. You can now place the subcircuit anywhere in the layout. Press the **Tab** key to display the Enter Coordinates dialog box and type the following values, then click **OK**. Note the **Rel** setting.



Click **OK** to place the subcircuit in the middle of the current boundary condition area.

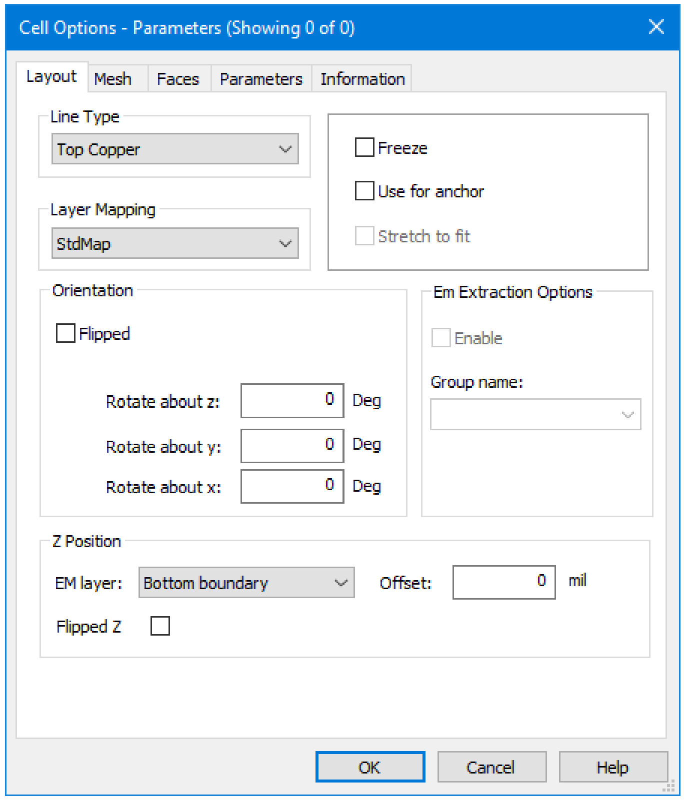


5. View the 3D layout of the "PCB" structure by right-clicking the structure in the Project Browser and choosing **View EM 3D Layout**.

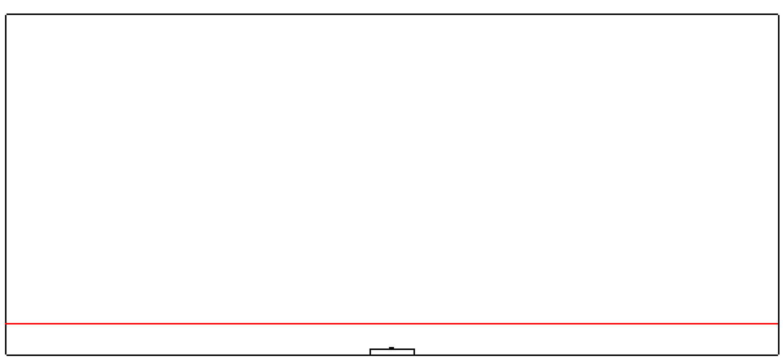


Note that the line layout is sitting on the top of the substrate layer in the new structure.

6. You can easily control the Z location of the subcircuit. To move the line to the bottom of the substrate, select the line in the 2D layout, right-click and choose **Shape Properties** to display the Cell Options dialog box.
7. On the **Layout** tab, in the **Z Position** section, change the **EM layer** to **Bottom boundary**.



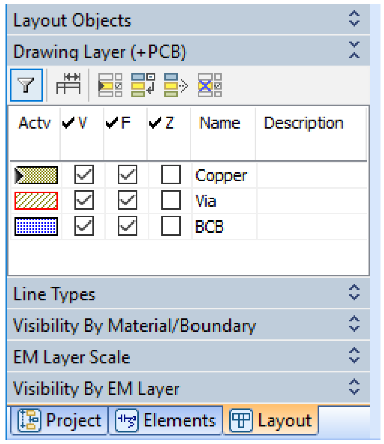
The "Analyst_Line" subcircuit now displays on the bottom of the substrate as shown in the following figure.



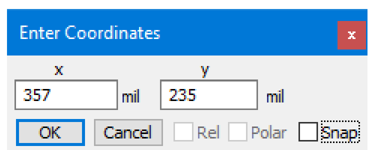
8. Reset the subcircuits layer back to "2". You can edit the properties or just choose **Edit > Undo**.

You can now draw the ground pad for the chip to sit on, and the lines feeding up to the chip.

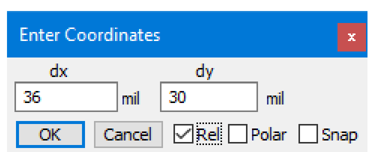
1. With the "PCB" structure 2D Layout View window active, click the **Layout** tab to open the Layout Manager, then click to open the **Drawing Layers** pane.



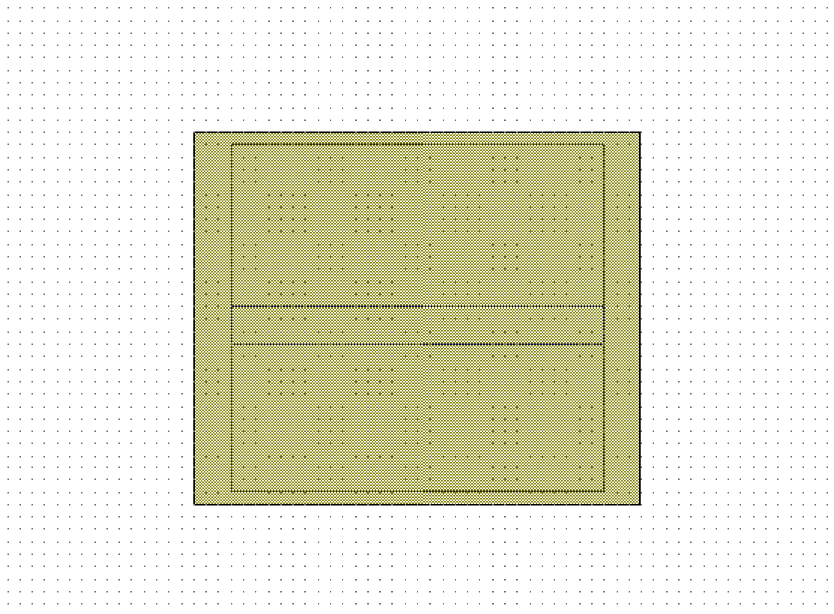
2. With the **Copper** layer selected, choose **Draw > Rectangle**. Press the **Tab** key to display the Enter Coordinates dialog box and type the following values, then click **OK**.



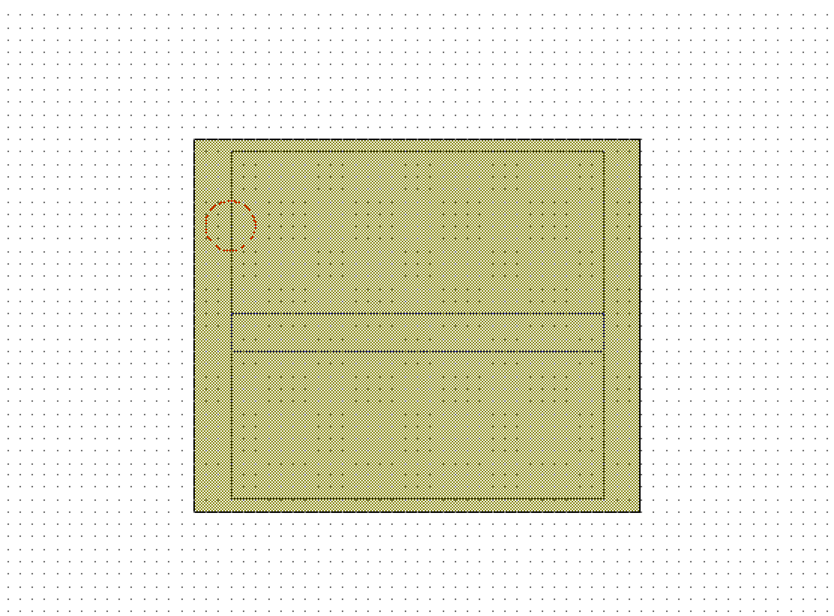
Press the **Tab** key again and type the following values, then click **OK**. Note the **Rel** setting.



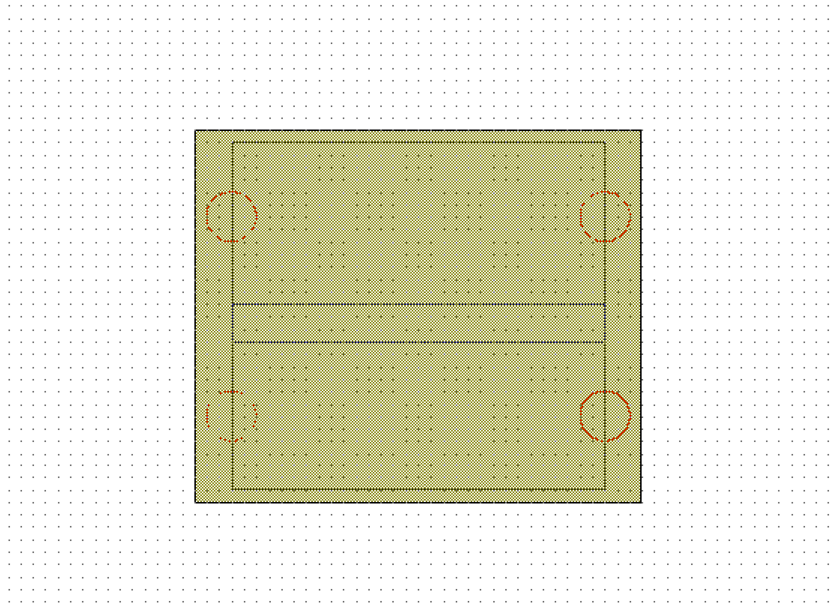
A shape displays under the line as shown in the following figure.



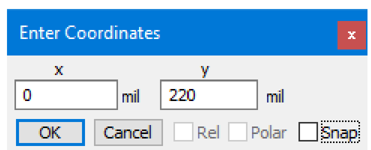
3. Select the **Via** layer and then choose **Draw > Ellipse**. Click anywhere in the 2D Layout View Window and press the **Tab** key to display the Enter Coordinates dialog box. Enter **4** as the **dx** coordinate and **4** as the **dy** coordinate, select **Rel**, and then click **OK** to create a 4 x 4 via. Next, drag the via under the line grounding pad as shown in the following figure. The exact placement is not critical for this example, but the via should be some distance from the edge.



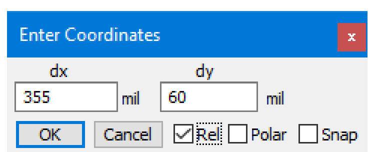
4. Select the via and type **Ctrl + C** and then **Ctrl + V** to copy and paste the via. Paste three vias under the line ground pad as shown in the following figure.



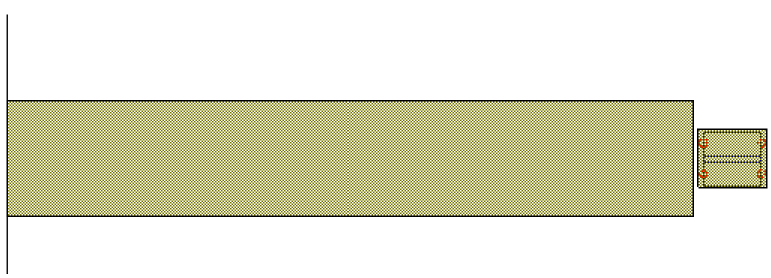
5. To construct the microstrip feed lines, with the **Copper** layer selected, choose **Draw > Rectangle**. Press the **Tab** key to display the Enter Coordinates dialog box and type the following values, then click **OK**.



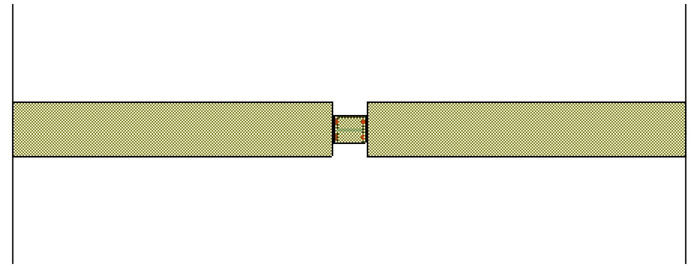
Press the **Tab** key again and type the following values, then click **OK**. Note the **Rel** setting.



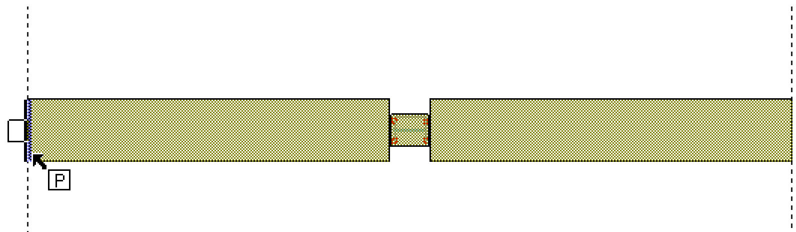
A shape now leads up to the line as shown in the following figure.



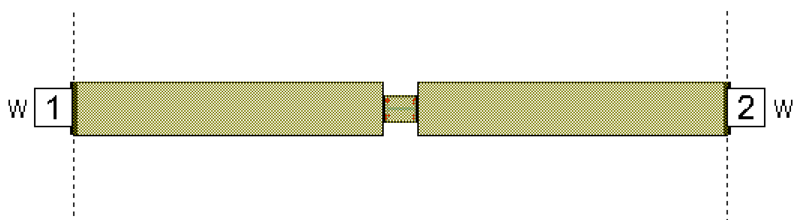
6. Copy and paste this shape to the other side of the line as shown in the following figure. Ensure that the line is perfectly aligned with the right edge of the boundary shape. This is necessary to enable using a wave port on this shape.



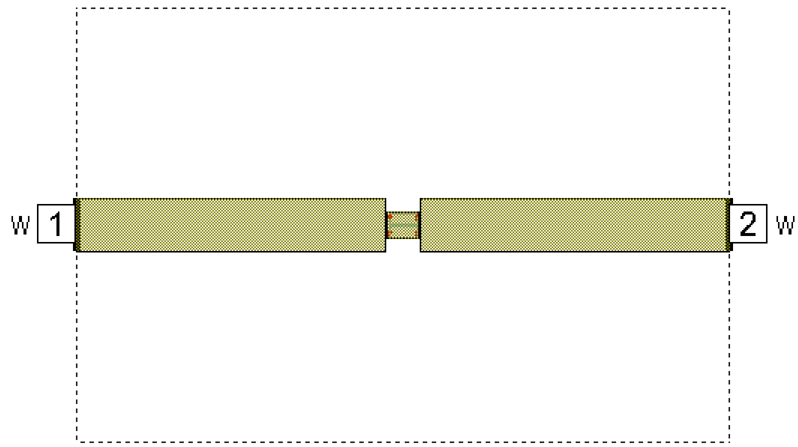
7. Choose **Edit > Auto Repeat** and then choose **Draw > Add Edge Port**. Move the cursor to the left edge of the left-most shape until the port symbol displays, as shown in the following figure, then click to place the port. Move the cursor to the right edge of the right-most shape until the port symbol displays, then click to place the port. Press **Esc** to exit Auto Repeat mode.

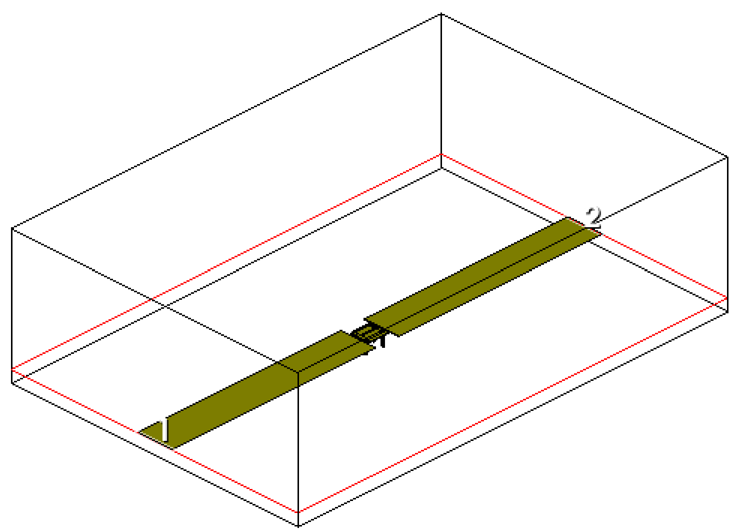


8. Double-click each port and in the Properties dialog box change the port **Type** to **Wave**. You can also choose **Edit > Port Properties** to edit both ports at once. See the *Analyst User Guide* for information about the differences in port types.



The 2D and 3D layout of the structure should display as shown in the following figures.



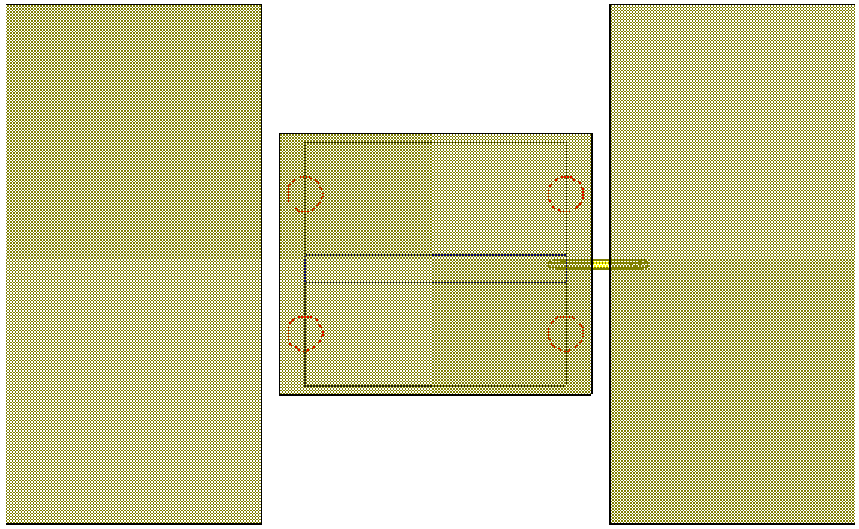


Adding a 3D Parameterized Cell (Bond Wires)

The next step is to add bond wires between the chip and the PCB. To do so, you can use a bond wire model from the 3D EM Elements libraries.

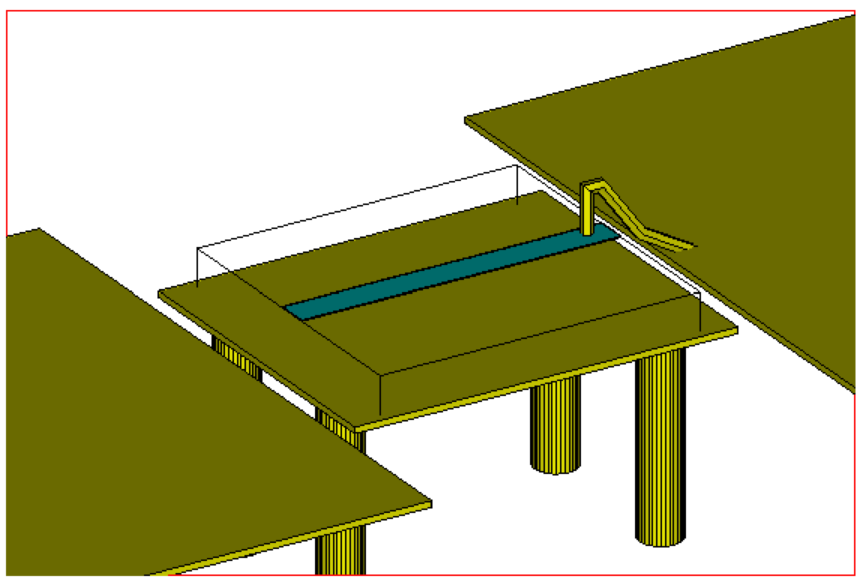
To add the bond wires:

1. Click the "PCB" EM structure 2D layout window to make it active.
2. In the Elements Browser, under **3D EM Elements**, expand the **Libraries** category and then click the **AWR web site > Interconnects** subgroup. Select the Bondwire model and place it in the "PCB" EM structure as shown in the following figure.

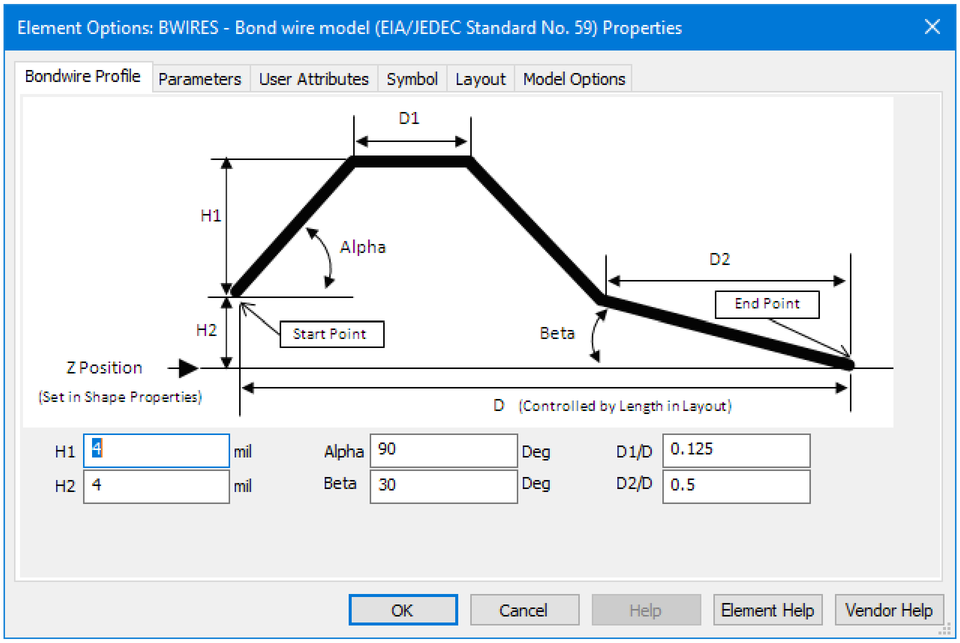


As with the subcircuit, the bond wire is placed at the Z location specified by the current EM layer. You can change the shape properties of the item to move up and down in the Z direction.

The following figure shows the 3D view of the bond wire on the chip.



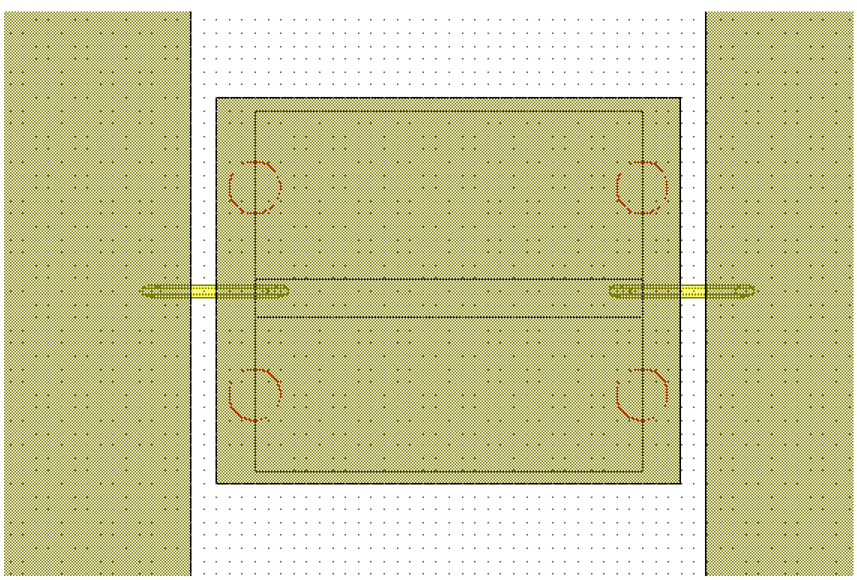
3. To configure the loop characteristics of the wire, select the wire in the 2D layout, right-click and choose **Element Properties** to display the Element Options - BWIRES dialog box. Verify that your settings match those in the following figure.



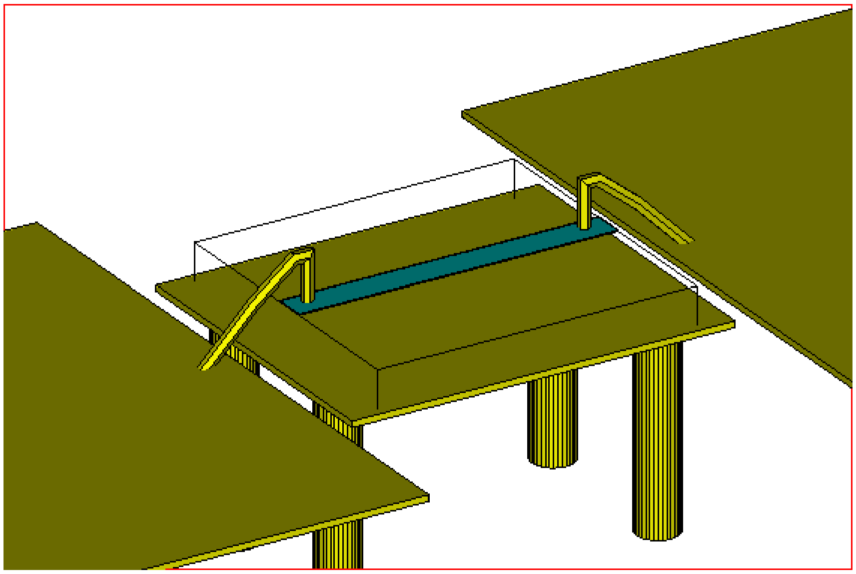
The heights set for the bond wire parameters are relative from the layer set for the 3D pCell. In a previous step you set the wire on EM layer 2, so the heights are set as heights above layer 2 in the "PCB" stackup.

4. To place an identical wire on the other side of the structure, in the 2D layout, select the bond wire and choose **Edit > Copy** and then **Edit > Paste**. **Ctrl** + right-click twice to flip the wire about the y-axis so that the high end is on the line.

5. Move the cursor over the other end of the line and click to place the line as shown in the following figure. It helps to have both the 2D and 3D layouts visible to ensure that you have the correct end of the pasted bond wire on the chip.

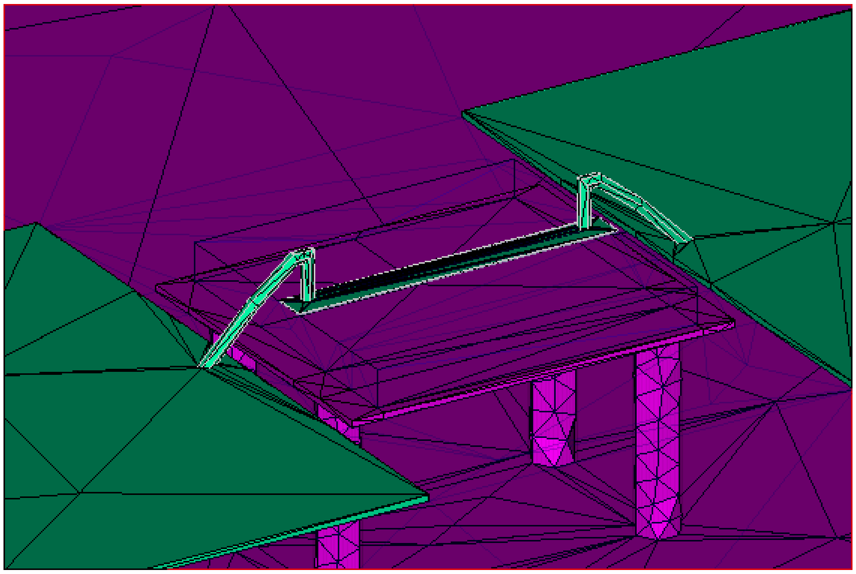


The 3D view now shows the wires on both sides of the chip.



6. To ensure that all elements are connected properly and there are no gaps in between, you can view the colored mesh. To do so, with the "PCB" EM structure 3D view active, click the **Show Mesh Connectivity** button on the EM 3D Layout toolbar. The shapes with the same color are electrically connected.

   The following figure shows the 3D view after the color meshing is turned on. Note that everything is connected properly.



Chapter 5. ANA: Using Arbitrary 3D Structures in Analyst

There are many ways to use 3D geometries in the Cadence® Analyst™ 3D FEM EM analysis software. As shown in the previous chapter, you can use built-in parameterized cells like a bond wire. You can also use components that are vendor-defined and hosted on the web via XML. It is also possible to create your own static or parameterized 3D components in the Cadence AWR Design Environment® platform 3D Editor. With the number of options available for creation/use, a 3D geometry structure set up becomes very easy.

Creating and Importing 3D Structures

This chapter is a continuation of *"ANA: Hierarchy and 3D Parts in Analyst"*. If you have not completed that example please do so before continuing here, as it includes concepts that may not be discussed in this example.

In this example you attach an SMA connector to the board and also cover the bond wires and chip area with a dielectric material. The SMA connector is a predefined structure designed in the AWR Design Environment platform 3D Editor. As part of this example you create a structure in the 3D Editor for the dielectric encapsulant.

This example includes the following main steps:

- Connecting the SMA connector to the board
- Using dielectric bricks to encapsulate the bond wire
- Running the full structure simulation
- Viewing results and E-fields
- Simplifying the structure to simulate just the SMA connector

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the program. Choose **Help > Quick Reference** to access this document.

Opening an Existing Project

The example you create in this chapter is available in its complete form as *Analyst_Arbitrary_Finish.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl**-click the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. This example is a continuation of the previous Analyst chapter.

To open an existing project:

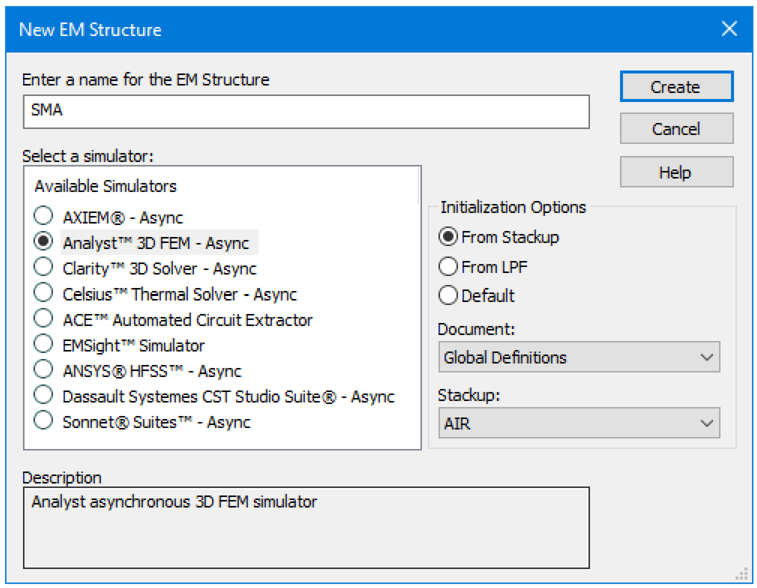
1. Choose **File > Open Example** to locate and open the *Analyst_Hierarchy_Finish.emp* file.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**Analyst_GS_arbitrary**" as the project name, and then click **Save**.

NOTE: Simulation results may vary slightly from the images in this guide. Finite Element Method (FEM) simulations require a convergence based on a mesh refinement sequence. Slight changes in the mesh refinement between versions of the solver can cause results to vary slightly. While the default convergence tolerance is sufficient for most geometries, if you find results shift you can decrease the convergence tolerance to ensure the results are accurate.

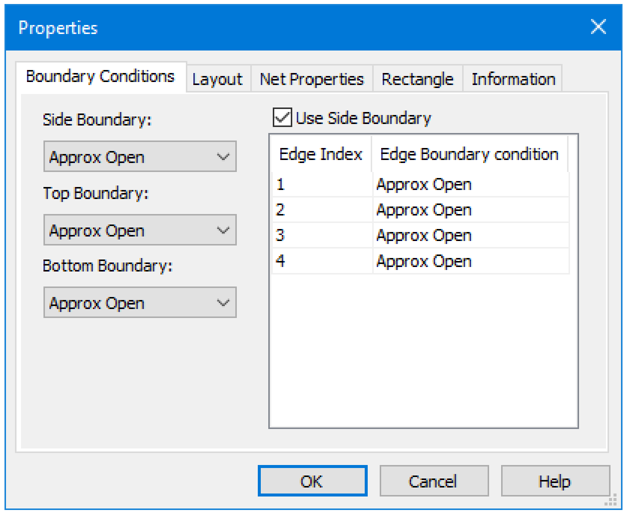
Adding the SMA Connector

You now add the SMA connectors to the "PCB" EM structure. Another level of hierarchy is used so you can simulate the chip and board itself, and then simulate the same structure with the SMA connectors to see the impact the connectors have on the overall circuit performance. To add the SMA connectors to the board:

1. Right-click the **EM Structures** node in the Project Browser and choose **New EM Structure**. (Alternatively, choose **Project > Add EM Structure > New EM Structure** or click the **Add New EM Structure** button on the toolbar) to display the New EM Structure dialog box.
2. Type "**SMA**" as the structure name, select **Analyst™ 3D EM - Async** as the simulator, select **From Stackup** as the **Initialization Options**, select **AIR** as the **Stackup**, and then click **Create**.

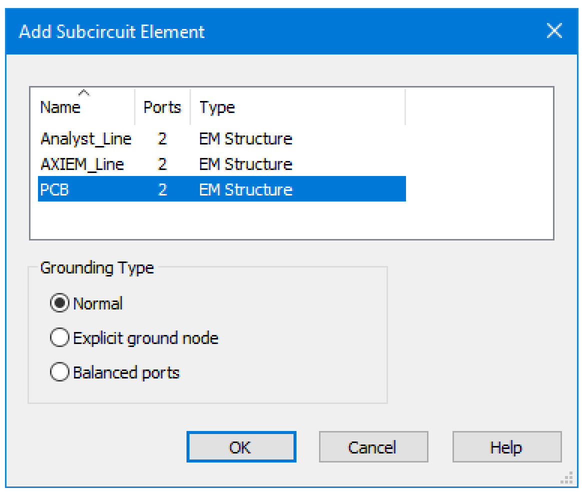


3. With the 2D Layout View window active, double-click the boundary shape to enter edit mode. Diamonds/squares display at the vertices and mid-points of each boundary edge.
4. Click and drag the upper right corner of the square and press the **Tab** key. In the Enter Coordinates dialog box, clear the **Rel** check box, and enter the values to make a boundary shape that is 750 x 500 mil, then click **OK**.
5. Right-click the 2D layout and choose **View All** to see the entire boundary shape.
6. Select the boundary shape, right-click and choose **Shape Properties** to display the Properties dialog box. Click the **Boundary Conditions** tab and ensure that the settings match those in the following figure.

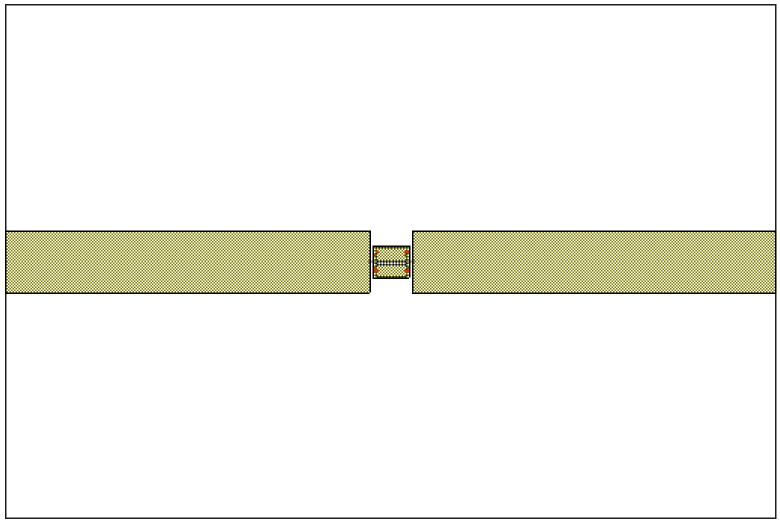


7. You now add the "PCB" EM document to the "SMA" EM document.

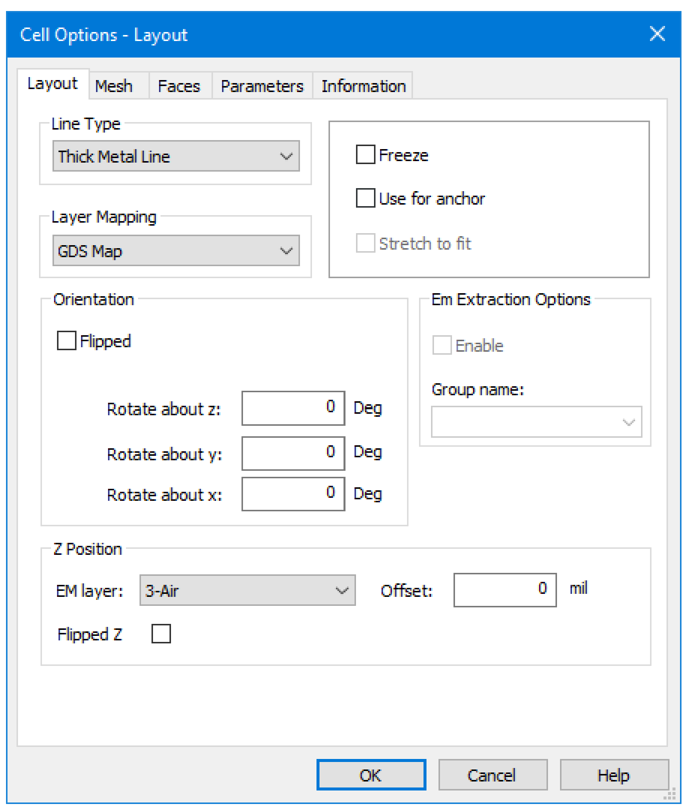
   With the "SMA" EM structure layout window active, choose **Draw > Add Subcircuit** to open the Add Subcircuit Element dialog box. Select "PCB" and then click **OK**.



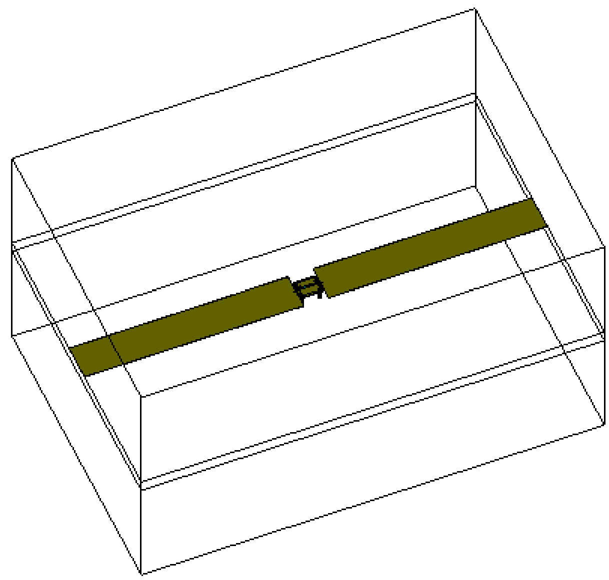
8. Place the subcircuit inside the boundary. You can press the **Ctrl** key while doing so to align the subcircuit with the boundary.



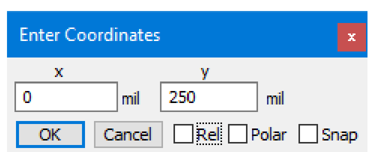
9. Select the "PCB" subcircuit, right-click, and choose **Shape Properties** to display the Cell Options dialog box. Under **Z Position**, change the **EM layer** to **3-Air**.



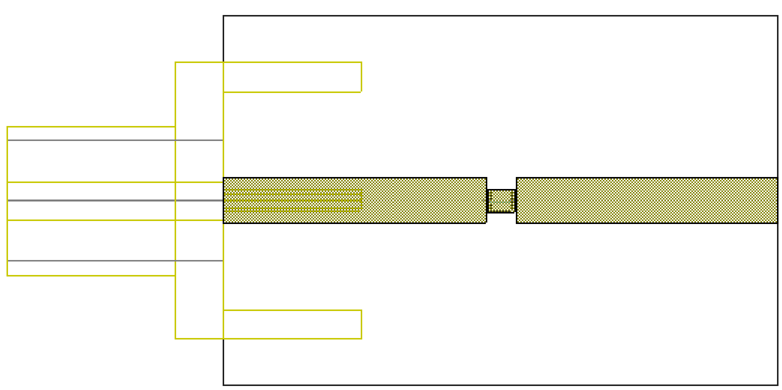
10. Choose **View > View 3D EM Layout** to see a 3D view of the "SMA" EM structure as follows.



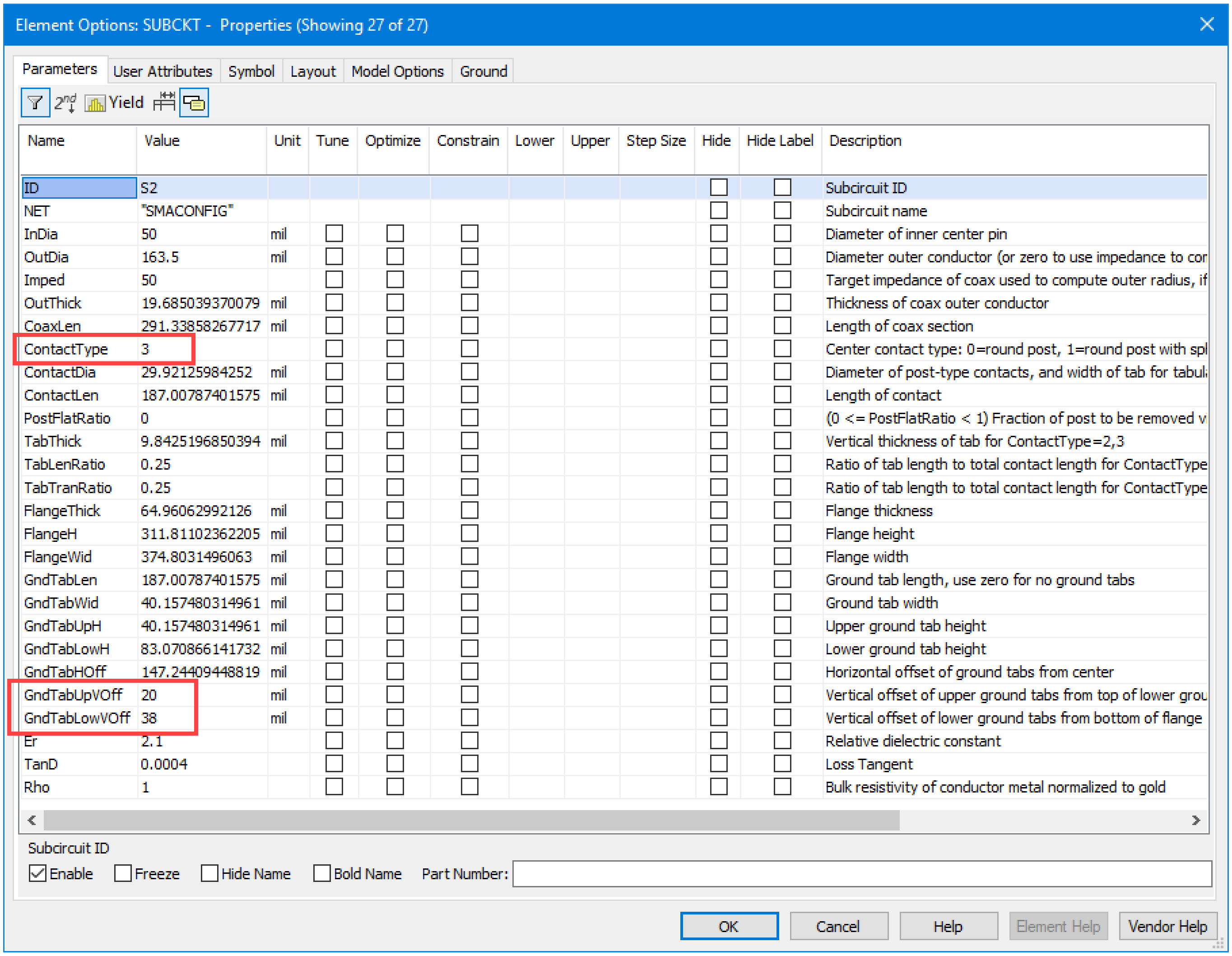
11. With the 2D view of the "SMA" EM structure active, in the Elements Browser under **3D EM Elements**, expand the **Libraries** category and then click the **AWR web site > Connectors > SMA** subgroup. Select the SMACONFIG model and place it in the "SMA" EM structure.
12. Press the **Tab** key while dragging SMACONFIG into the "SMA" EM structure, enter the following values in the Enter Coordinates dialog box, then click **OK**.



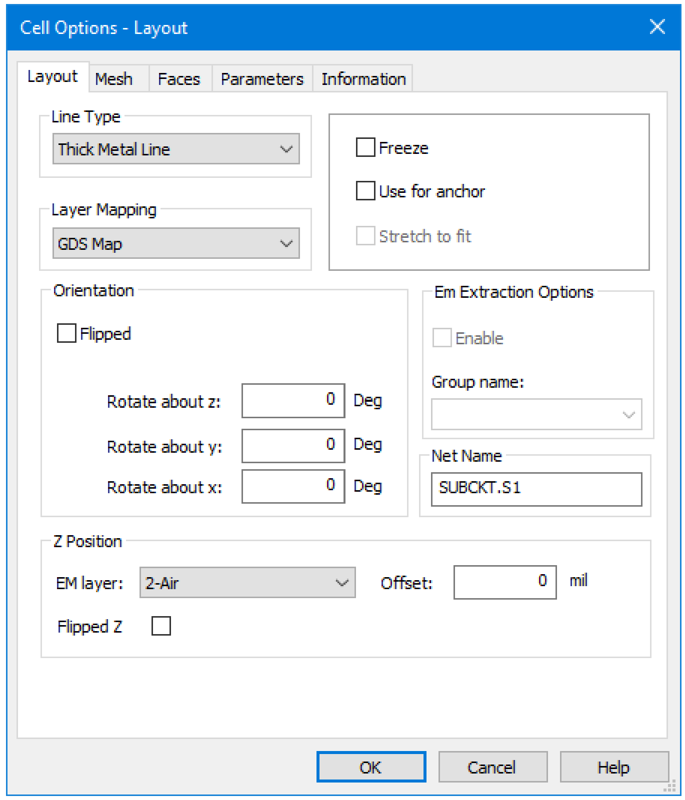
The following figure shows the 2D view of the "SMA" EM structure.



13. Select the SMACONFIG model, then right-click and choose **Element Properties** to display the Element Options: SUBCKT - Properties dialog box. Verify that your settings for ContactType, GndTabUpVoff, and GndTabLowVoff match those in the following figure and then click **OK**. All the other values remain at defaults.

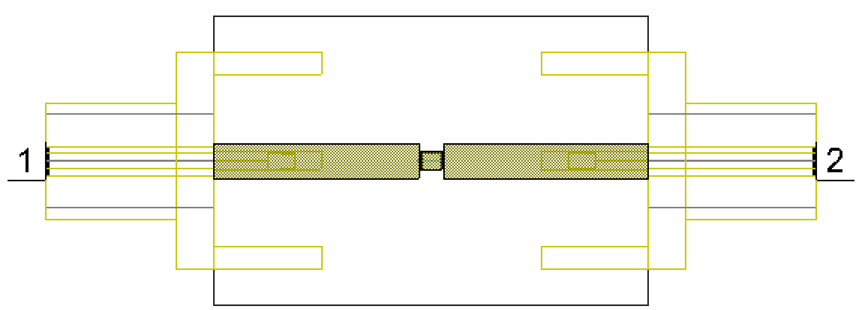


14. Right-click the 2D SMACONFIG layout again and choose **Shape Properties** to display the Cell Options dialog box. Under **Z Position**, make sure the **EM layer** is set to **2-Air**.

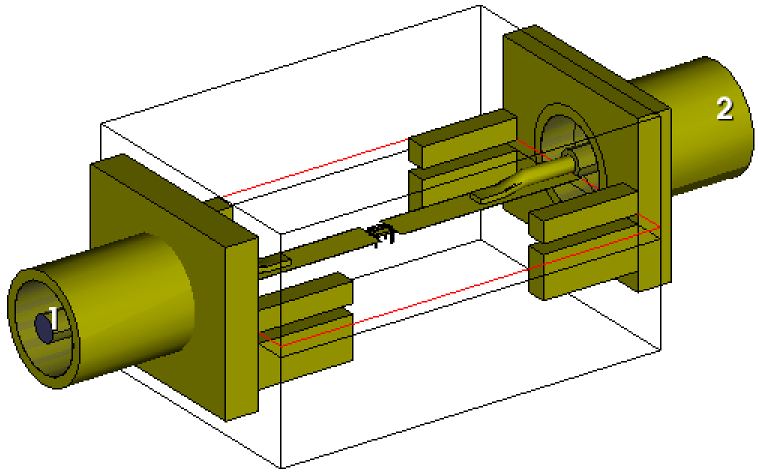


15. Copy the SMACONFIG element and paste it on the other side of the EM structure. After rotating the structure press **Tab**, clear the **Rel** check box and enter "**802**" for **X** and "**250**" for **Y**.

16. Choose **Draw > Add Edge Port** and hover the cursor near the edge of the center conductor of the left SMA until the port outline displays. Click to place the port. Repeat this step for the right port. When you are done the 2D view should look like the following figure:

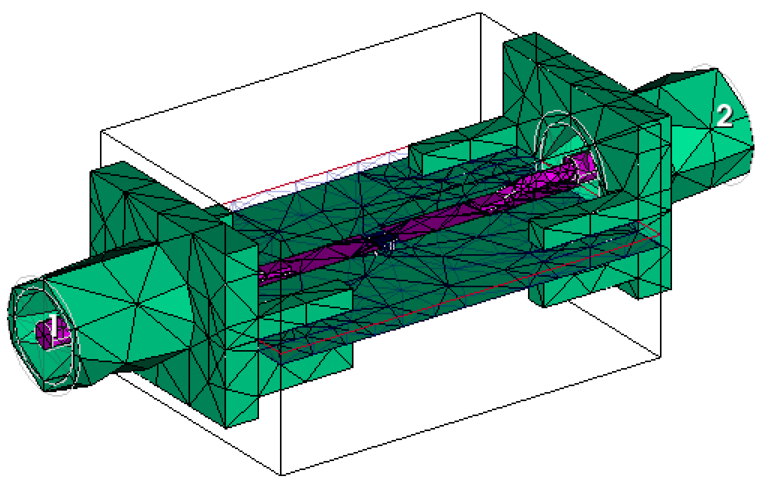


17. The following figure shows the 3D view of the "SMA" EM structure.



18. To ensure that everything is connected properly, in the 3D view of the "SMA" EM structure, click the **Show Mesh Connectivity** button on the toolbar. The following figure shows the 3D view after the mesh connectivity is turned on.

    NOTE: Conducting boundary conditions propagate through hierarchy, so the ground plane from the PCB EM structure is present in the SMA EM structure as seen with the mesh connectivity.



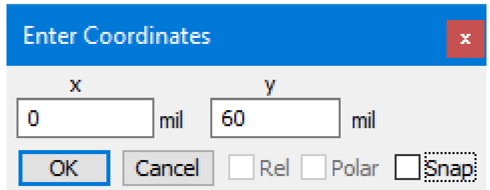
Adding Solder Pads

Before simulating, a more realistic design of this structure would include the solder pads for the SMA connection. You can add four copper shapes and vias at the same level of hierarchy as the SMA connectors. The final project includes these solder pads as shown in the following figure.

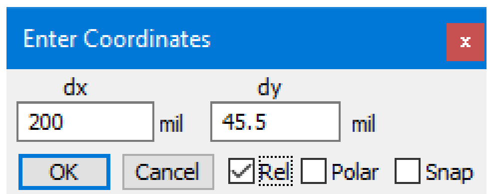
To add the solder pads and vias:

1. Open the 2D Layout View of the "SMA" EM structure and click the **Layout** tab to open the Layout Manager.

2. Select the **Copper** drawing layer and start to draw a rectangle by pressing **CTRL + B** or by choosing **Draw > Rectangle**.
3. Press the **Tab** key to display the Enter Coordinates dialog box, then enter the values shown in the following figure and click **OK**.



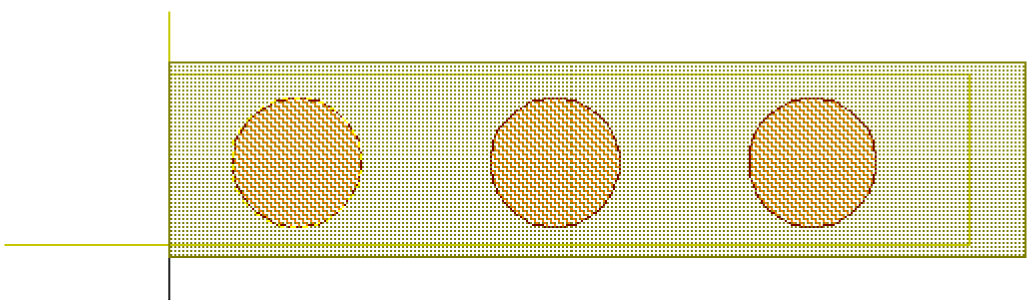
4. Press the **Tab** key again and enter the size of the rectangle. Note the **Rel** setting.



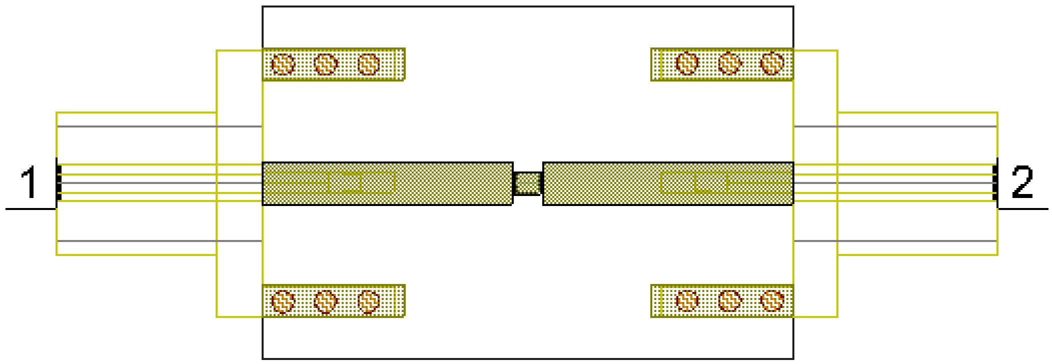
5. Select the **Via** drawing layer and choose **Draw > Circle**, then press the **Tab** key to display the Enter Coordinates dialog box. Enter "**30**" for **x** and "**82**" for **y** and then click **OK**.

   Press the **Tab** key again and enter "**15**" for **R**.

   Copy the via and paste two more vias 60 mils apart. When pasting you can press the **Tab** key to enter coordinates and space the vias evenly. When you are done the layout should display as shown in the following figure.

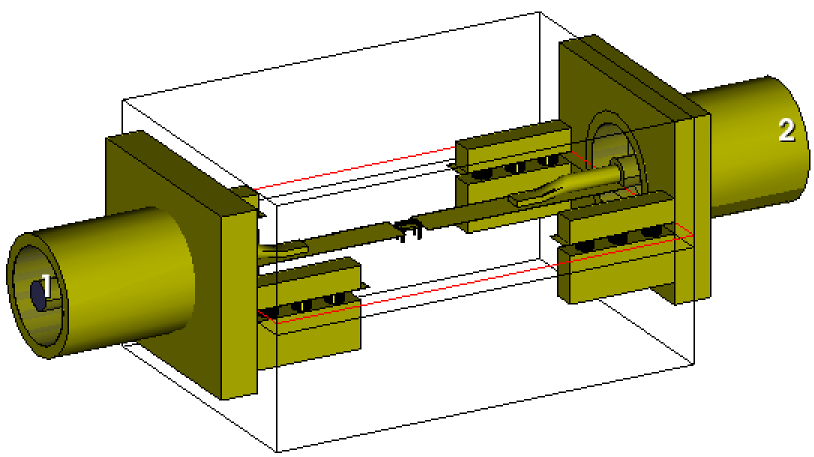


6. Copy the solder pad and all three vias and paste them in the appropriate positions under the SMA flanges. When you are done the 2D layout should display as shown in the following figure.



Simulating the Entire Structure

Now that the solder pads and vias are added, it is time to simulate the structure.

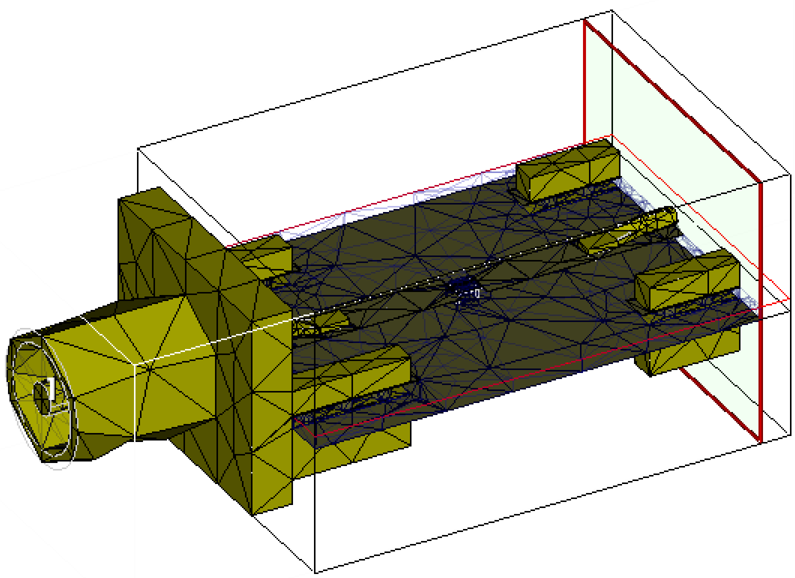


To visualize the E-fields for this structure after the simulation is complete you need to set certain options. To set up the structure to save E-field information:

1. Right-click the "SMA" EM structure in the Project Browser and choose **Options** to display the Options - SMA dialog box, then click the **Analyst** tab.
2. Change the **Field Output Frequency** to **AMR Frequencies Only**, then click **OK**.

3. Click the **Show Currents/Fields** button on the EM 3D Layout toolbar to display the fields on a cut plane. As each AMR sequence finishes, the E-field display updates.
4. With the SMA EM structure 3D view window active, click the **Show 3D Mesh** button on the toolbar to view the mesh of the structure and watch it update at each AMR step. (**NOTE:** To ensure that your project matches this example, your toolbar should display as shown in the following figure, specifically the activation of the **Show Cut Plane** and **Use Cut Plane** buttons.

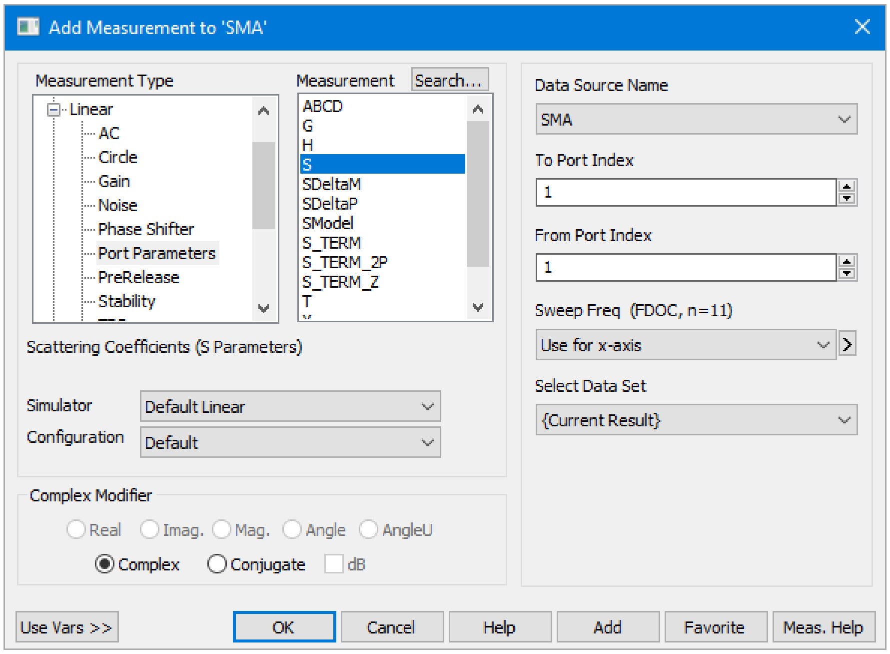
The following figure shows the initial mesh with the cut plane off to the side.



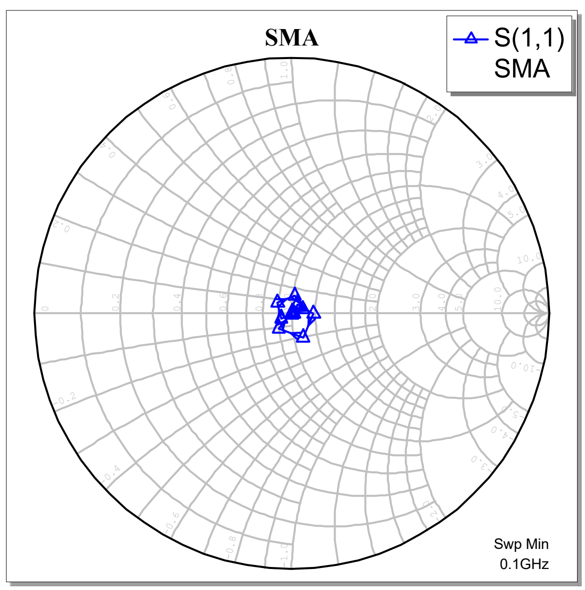
5. Choose **Simulate > Analyze** or click the **Analyze** button on the toolbar to start the simulation. **NOTE:** Typically, this simulation takes about 10 minutes to run with no other programs competing for resources. While the simulation runs you can configure a graph to display results from the new simulation.

To create a graph:

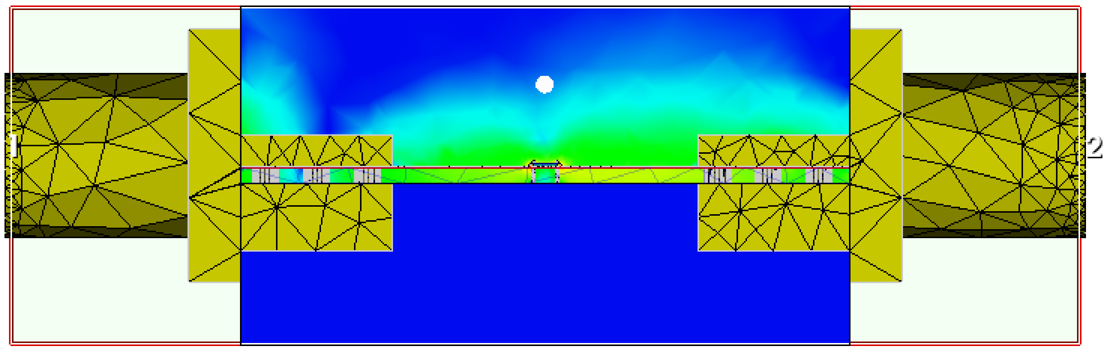
1. Right-click the **Graphs** node in the Project Browser and choose **New Graph**.
2. Name the graph "**SMA**", select **Smith Chart** as the graph type, and click **Create**.
3. Right-click the new graph and choose **Add New Measurement** to add "S11" to the graph. Verify that your settings match those in the following figure.



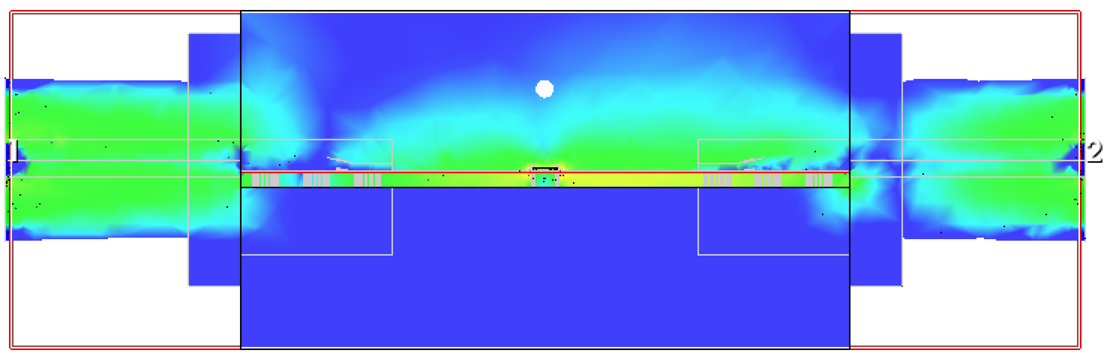
4. When the simulation is complete, the graph data displays as shown in the following figure.



You can see the finished solution E-field display in the 3D layout. With the 3D Layout View active, press the **y** key to move the cut plane to the y plane. The annotations display similar to the following figure.

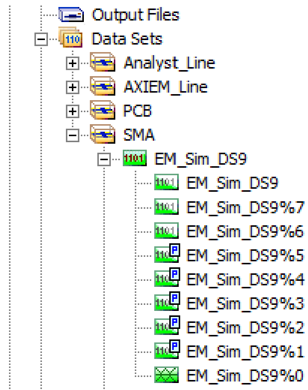


To see the fields in the structure behind the metal, click and hold on the cut plane and it displays the fields hidden behind the metal.



You can view the mesh and E-field data for each AMR sequence during simulation or after the simulation is complete. To view each AMR sequence result:

1. With the "SMA" EM structure 3D layout window active, expand the **Data Sets** node in the Project Browser and then expand the **SMA** subnode. The current data set for this EM structure displays with a green icon and you can also expand it. The data sets below it are for each AMR sequence run in the simulation.



2. **Shift**-click a sub data set to show the mesh and E-fields for that AMR sequence. Note that the E-field does not display for the port-only AMR sequences. These display a small "P" on the data set icon and are the lowest in the list.

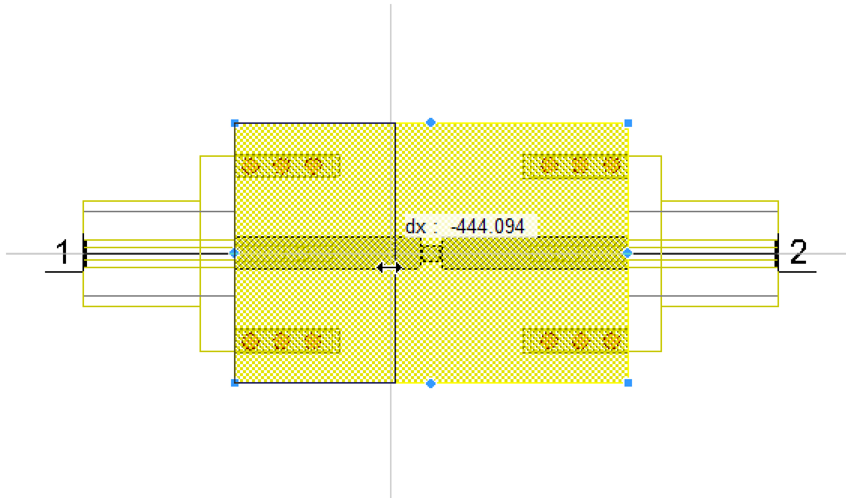
Configuring for Transition Simulation Only

In the previous example, you simulate the SMA, PCB, and the chip as an entire structure. You may, however, want to simulate only a subset of the geometry drawn. In this example, you look at the response of the SMA to see how it affects when adding it to the network. You can do so by moving the boundary shape and configuring ports; you do not need to alter the original geometry.

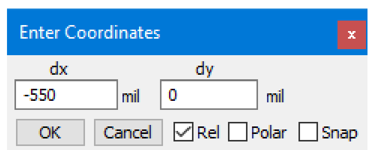
For Analyst software simulation, only shapes inside the boundary are simulated. so you can simulate only parts of your geometry without needing to edit much on the full geometry of your structure. In the following step you only simulate one of the SMAs using the current geometry setup.

The first step is to duplicate the structure already created. To duplicate this structure:

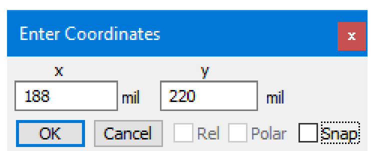
1. Right-click the "SMA" EM structure and choose **Duplicate**. A new EM structure named "SMA 1" displays in the Project Browser.
2. Right-click "SMA 1" and choose **Rename EM Structure**. In the Rename EM Structure dialog box, name the structure "SMA_Simplified", then click **Rename**.
3. With the "SMA_Simplified" window active, double-click the boundary shape to enter edit mode (diamonds/squares display on each vertex and edge mid-point).
4. Click and hold the mid-point of the right boundary shape and drag it towards port 1 as shown in the following figure.



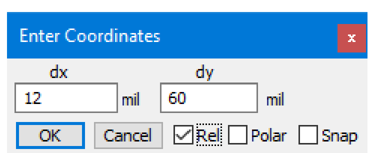
5. As you get closer to port 1, press the **Tab** key and enter the following values. Make sure the **Rel** box is selected.



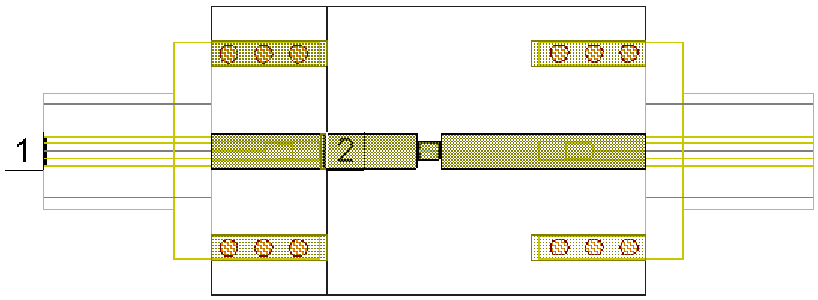
6. Select the **Copper** drawing layer and choose **Draw > Rectangle**. Press the **Tab** key to display the Enter Coordinates dialog box and type the following values, then click **OK**.



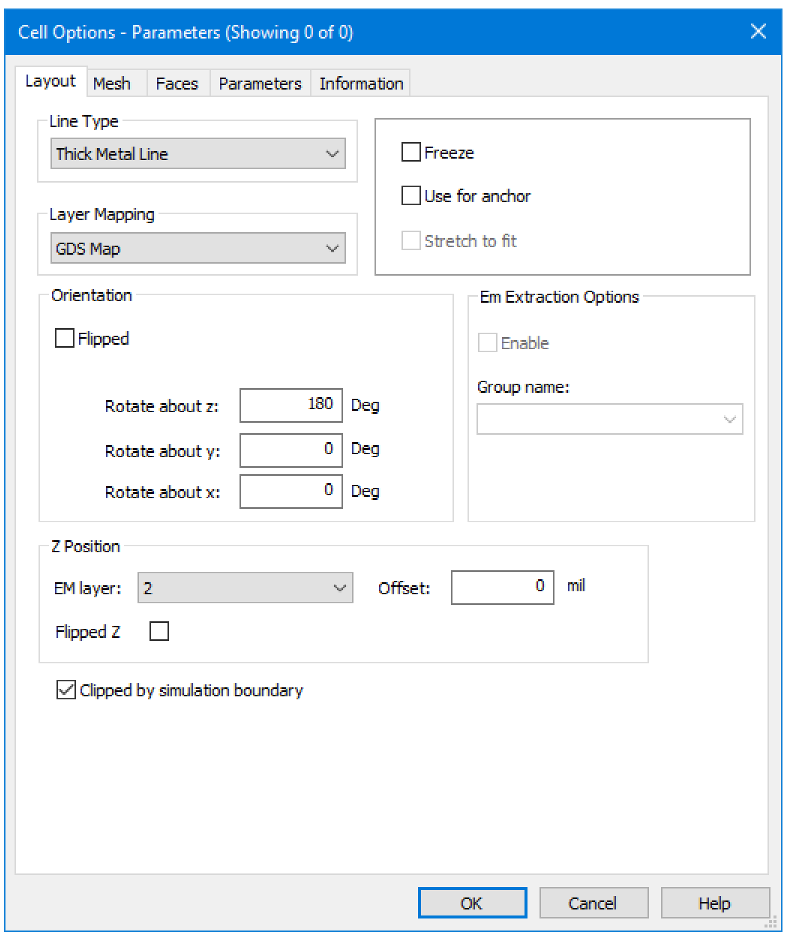
7. Press the **Tab** key again and type the following values, then click **OK**. Note the **Rel** setting.



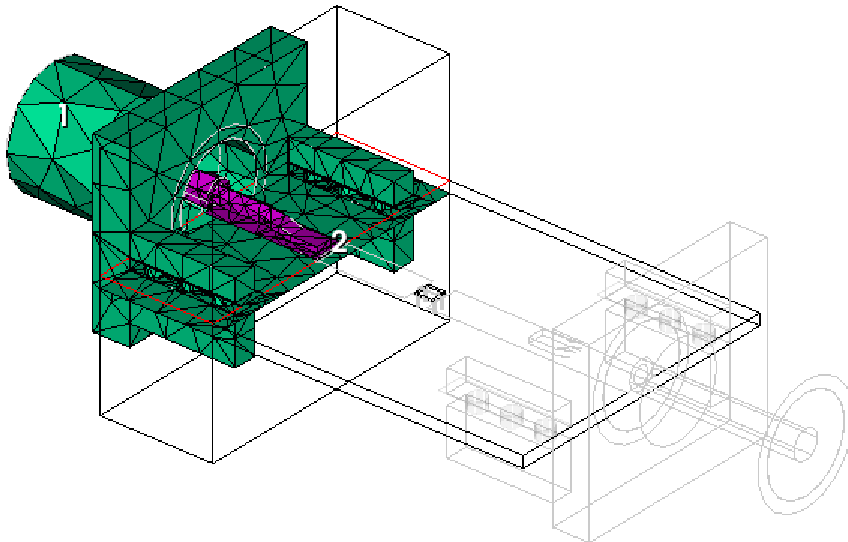
8. Delete port 2 on the right side. Choose **Draw > Add Edge Port** to add a port to the right edge of the rectangle created in the previous step. After you add the port, double-click it and change the **Type** from **Lumped Down** to **Wave**.



9. Even though the boundary shape is not touching the SMA on the right, that SMA is simulated with the current setup. You can prevent this by right-clicking that SMA and choosing **Shape Properties**, then selecting the **Clipped by simulation boundary** check box in the Cell Options dialog box.

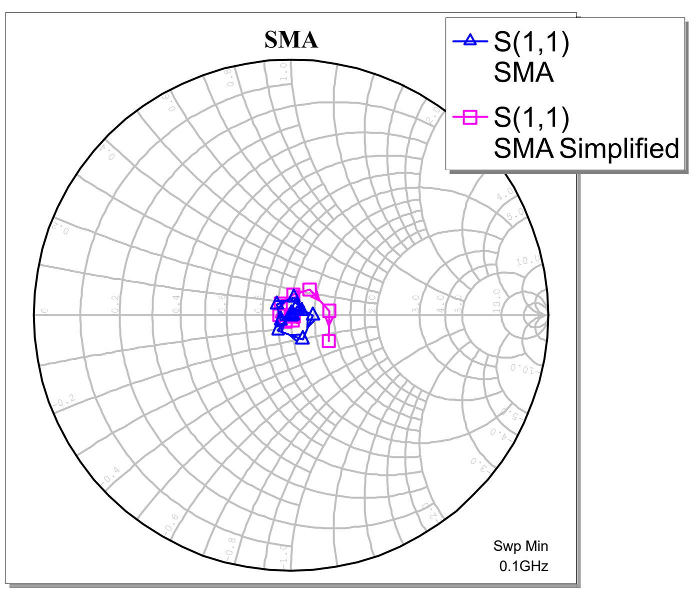


10. Observe the connectivity mesh before simulating.



11. Run the simulation. **NOTE:**: Typically, this simulation takes several minutes to run with no other programs competing for resources.

    While the simulation runs, add the S11 measurement to the "SMA" graph to see the response of this circuit when the simulation completes. The "SMA" graph should display similar to the following figure.



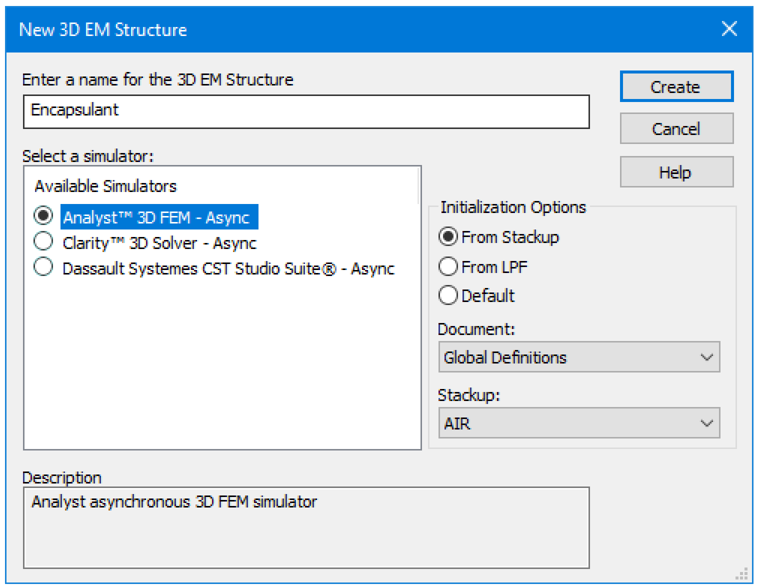
Encapsulating the Chip and Bond Wires

The next step is to encapsulate the chip and bond wires with a dielectric material. In previous steps you imported an arbitrary 3D EM structure, the SMA connectors. Now you are going to create an arbitrary 3D EM structure and use it in this design. The encapsulant is parameterized such that after creation you can specify the parameters on the subcircuit of the EM document rather than modifying the geometry in the AWR Design Environment platform 3D Editor.

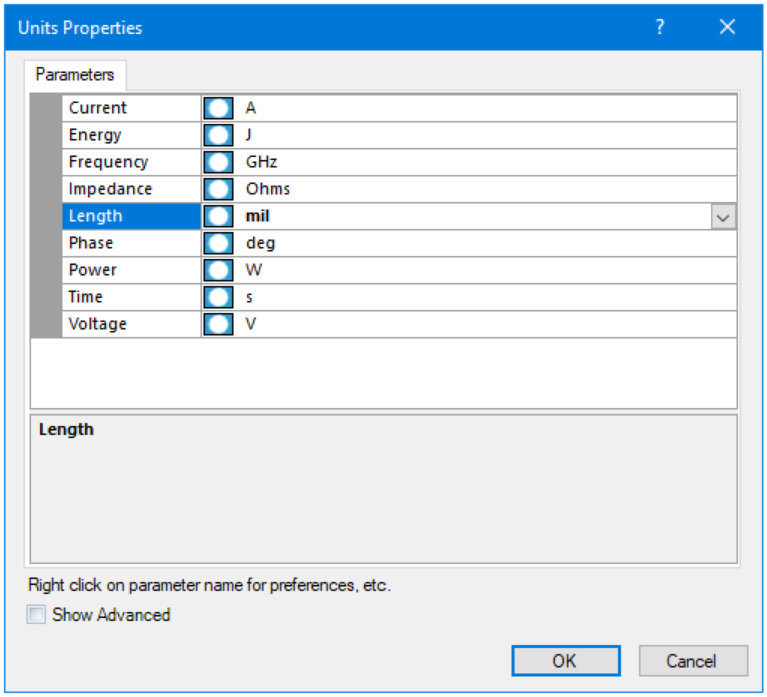
The encapsulant is an extremely simple parameterized 3D EM structure but it is possible to create fully parameterized EM structures with complex geometries such as the SMA connector used in this example. To see different examples of parameterized EM structures, browse the 3D EM Elements in the Elements Browser.

To create the encapsulant:

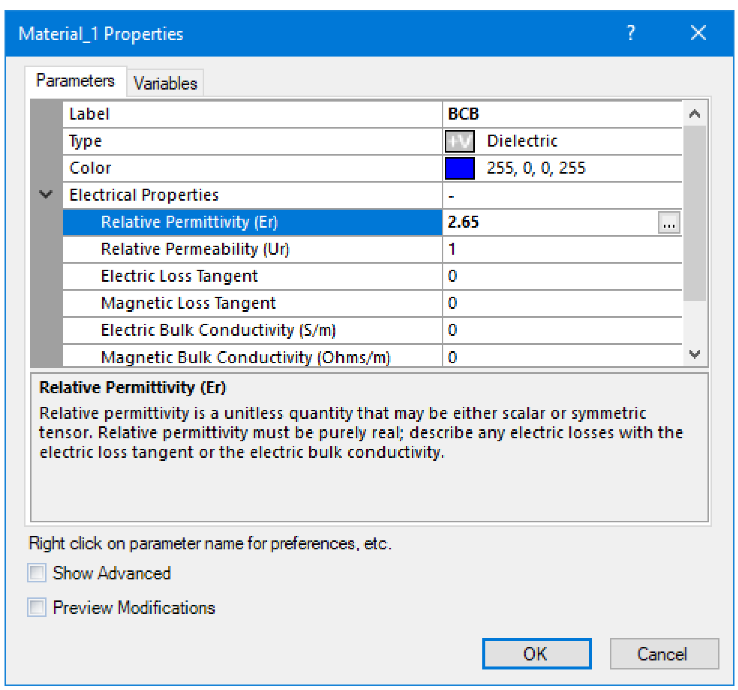
1. Right-click the **EM Structures** node and choose **New Arbitrary 3D EM Structure** to display the New 3D EM Structure dialog box.
2. Type "**Encapsulant**" as the structure name, select **Analyst™ 3D EM - Async** as the simulator, select **From Stackup** under **Initialization Options**, select **AIR** as the **Stackup**, and then click **Create**.



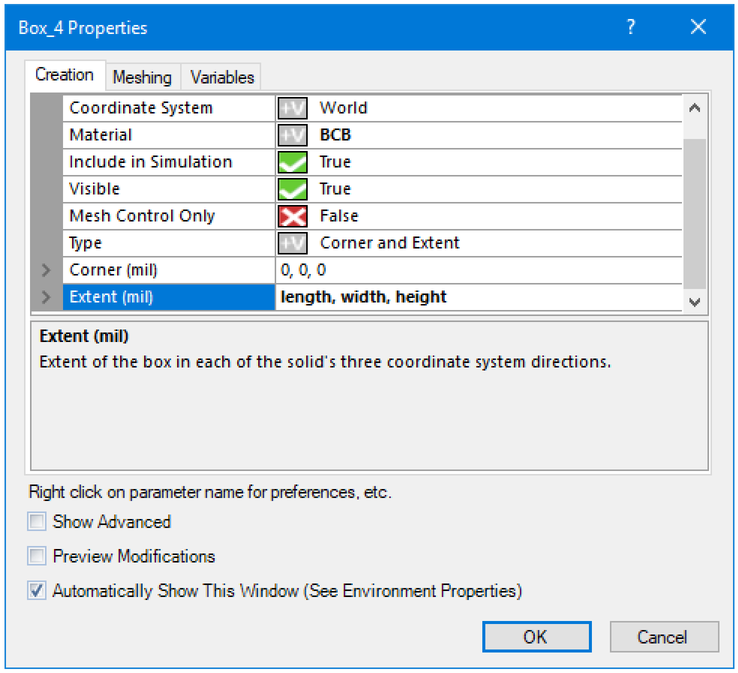
3. Right-click the "Encapsulant" EM structure and choose **Open in 3D Editor**. The AWR Design Environment platform 3D Editor opens.
4. On the **Home** ribbon in the **Settings** group, click **Units** to display the Working Units dialog box.
5. Change the **Length** units to **mil** and click **OK**.



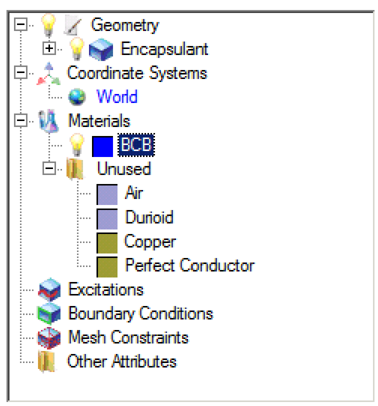
6. On the **Structure** ribbon, in the **Entity Creation** group, click **Draw Entity** and choose **Box**.
7. Press the **Space bar** to display the Edit Properties dialog box. Set **Material** to **<New>**.
8. In the Edit Properties dialog box for the new material, specify the **Label** as "**BCB**", and set the **Relative Permittivity (Er)** to "**2.65**", then click **OK**.



9. In the Edit Properties dialog box for the new box, specify the **Label** as "**Encapsulant**", set **Corner** to "**0, 0, 0**", and then for **Extent**, type "**length, width, height**", then click **OK**.

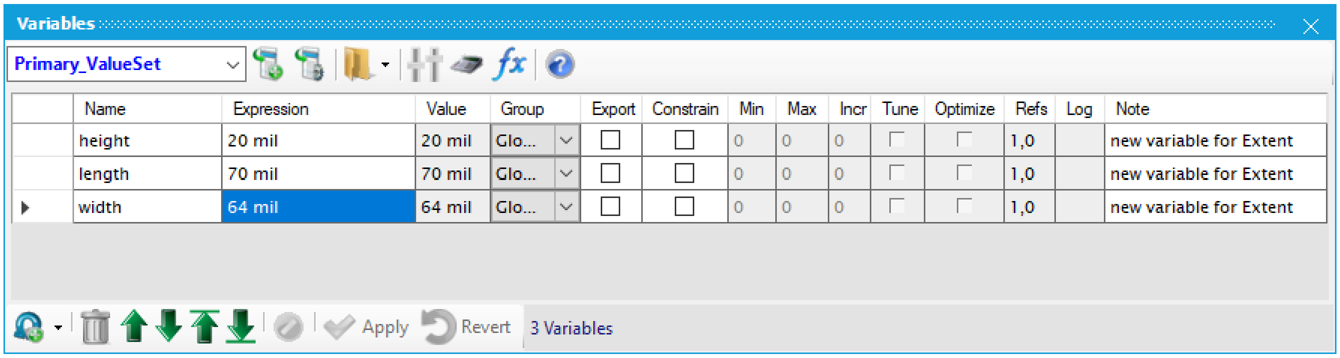


The BCB material is included in the list of materials that you can use on other shapes.



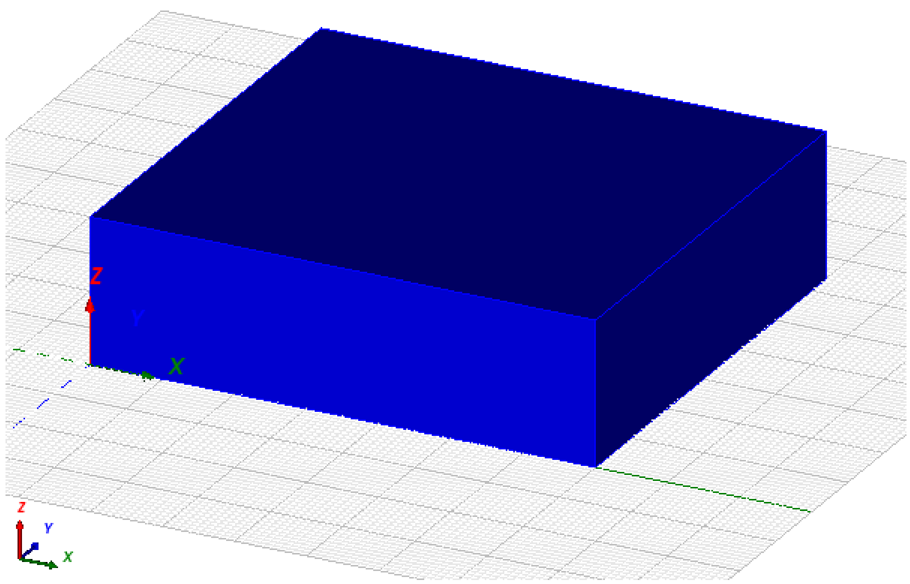
10. Set the values of the variables in the Variables window (click the **Variables** tab at the bottom left of the main window by default, or on the **Window** ribbon in the **Controls** group, click **Control Visibility** and choose **Variables** to display the Variables window). Set **length** to "**70 mil**", **width** to "**64 mil**", and **height** to "**20 mil**", then click the **Apply** button to accept the changes. Note that you must type "mil" in the expression. This flexibility allows you to enter values in units besides the current drawing unit.

    When complete, the Variables window should match the following figure.



11. Press the **Fit** button on the toolbar. Alternatively, press **F**, the default hotkey for the Fit command.

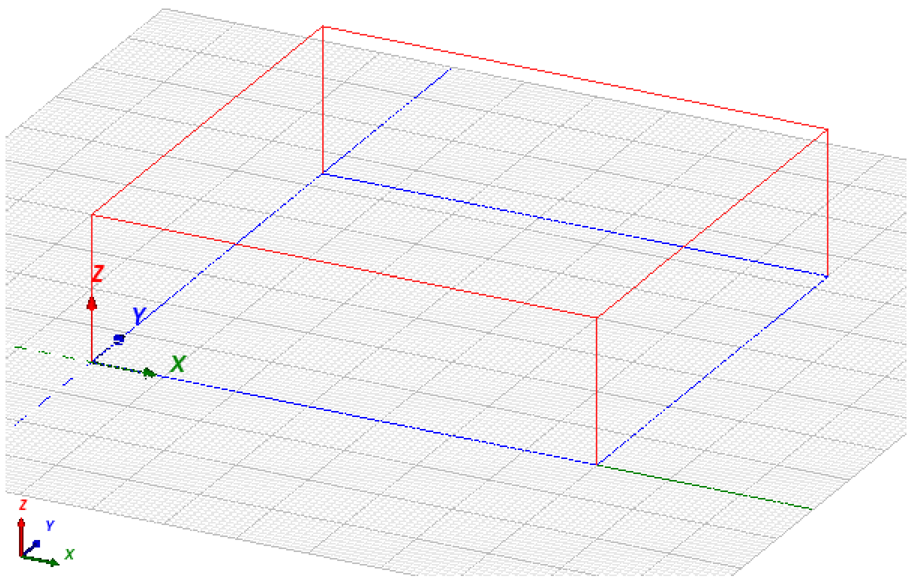
The dielectric brick should display as shown in the following figure:



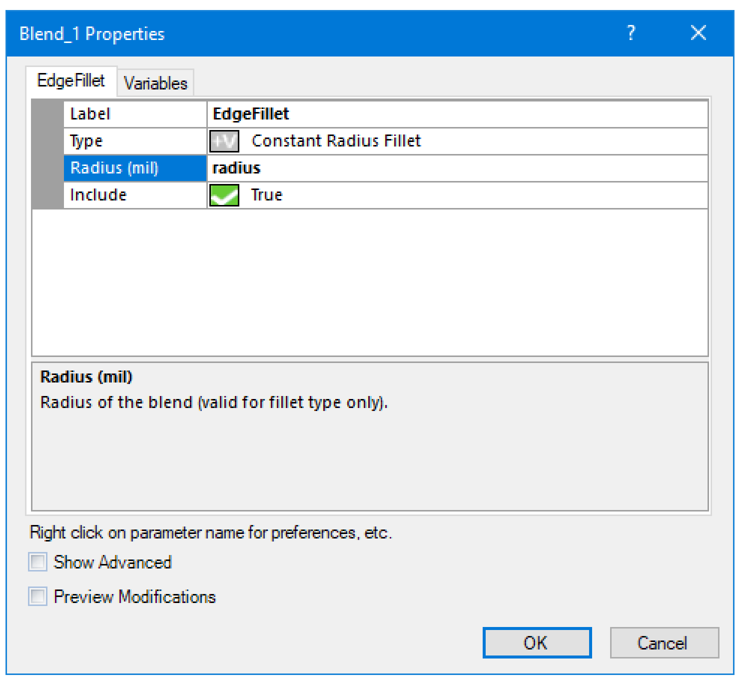
12. To make the dielectric a more realistic shape, you need to fillet the box to soften the corners. To soften the shape, on the **Structure** ribbon in the **Entity Modification** group, click **Modify Solids** and choose **Fillet Edges**.

    The box becomes transparent, except for the edges, to allow you to select which edges you want to soften.

13. Click an edge to add it to the group of edges to alter. When selected, the edge displays in red.
14. Select the following edges and then press **Enter** to display the Edit Properties dialog box.



15. Specify the **Label** as "**EdgeFillet**" and the **Radius** as "**radius**", then click **OK**.



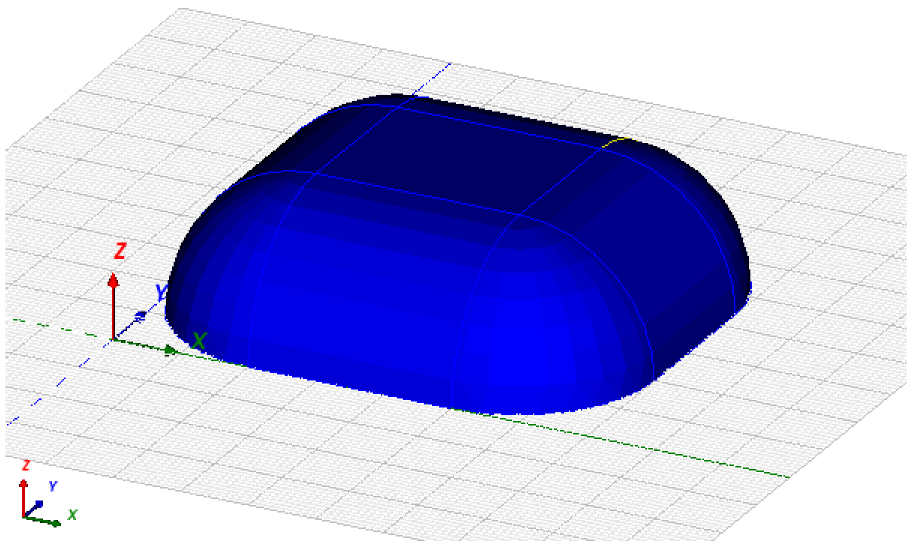
16. In the Variables window, set the **Expression** of the new radius variable to "**20 mil**" and click **Apply**.

17. The final step in creating a parameterized arbitrary 3D EM structure is to set the variables to export from the AWR Design Environment platform 3D Editor into the AWR Design Environment software. In the Variables window, click the **Export** check box for each of the four variables that define this structure.

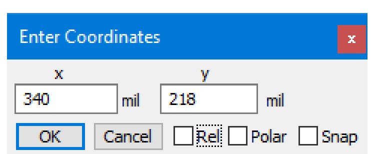
18. When defining a parameterized EM structure you should constrain the variables that define the structure. Enable constraints for each variable and set the values to match those in the following figure.

Name	Expression	Value	Group	Export	Constrain	Min	Max	Incr	Tune	Optimize	Refs	Log	Note
height	20 mil	20 mil	Global	☑	☑	0 mil	20 mil	1 mil	☐	☐	1,0		new variable for Extent
length	70 mil	70 mil	Global	☑	☑	0 mil	70 mil	1 mil	☐	☐	1,0		new variable for Extent
width	64 mil	64 mil	Global	☑	☑	0 mil	64 mil	1 mil	☐	☐	1,0		new variable for Extent
radius	20 mil	20 mil	Global	☑	☑	0 mil	20 mil	1 mil	☐	☐	1,0		new variable for Radius

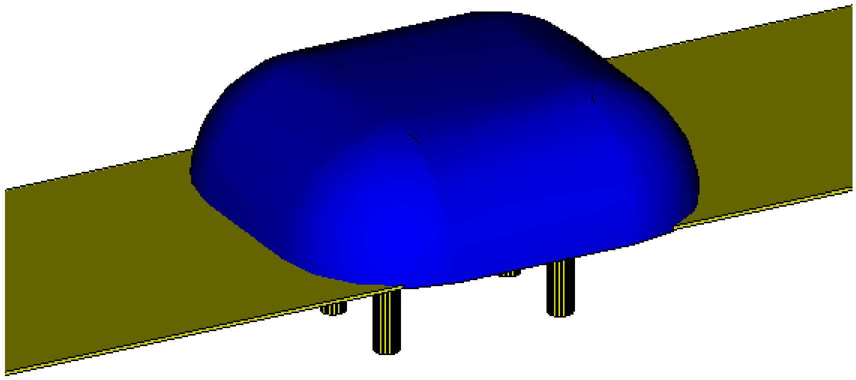
When complete, the final structure should display as shown in the following figure.



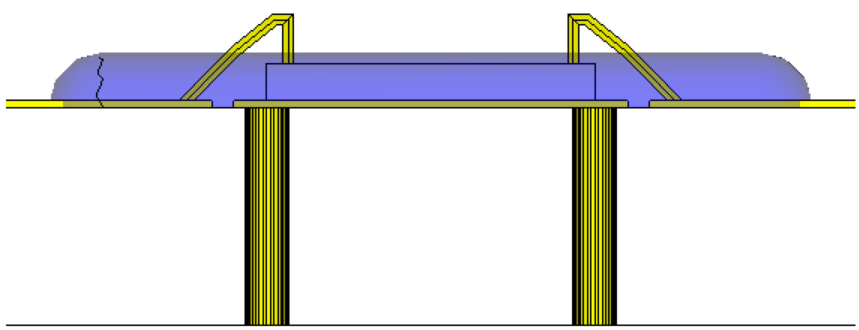
19. Click the **Save** button on the toolbar and then exit the AWR Design Environment platform 3D Editor. Upon closing the 3D editor, the changes to the "Encapsulant" EM structure are sent to the AWR Design Environment software.
20. Open the 2D Layout View of the PCB EM structure and add the encapsulant by choosing **Draw > Add Subcircuit** and selecting **Encapsulant**. Press the **Tab** key to enter the coordinates as shown in the following figure.



21. Right-click the encapsulant in the 2D layout and choose **Shape Properties**.
22. In the Cell Options dialog box under **Z Position**, set the **EM layer** to "**2**". The encapsulant covers the chip and the bond wires.



23. Select the **Clipped by simulation boundary** check box and then click **OK**.
24. To see what is in the encapsulant, in the Layout Manager **Visibility By Material/Boundary** pane, set the **Opaque** value for the BCB material to "**0.5**".
25. Change the height of the encapsulant. From the 2D layout of the PCB EM structure, right-click the encapsulant and choose **Element Properties** to display the Element Options dialog box. Change the height to "**5**" and the radius to "**5**", then click **OK**. The bond wires extrude from the dielectric with these settings.



26. Change both the height and radius to "**10**" so the bond wires do not extrude from the dielectric.

Chapter 6. ANA: Importing SAT Files in Analyst

In addition to the geometry creation mechanisms covered in previous Cadence® Analyst™ 3D FEM EM analysis chapters, there is another mechanism, importing Standard ACIS Text (SAT) files, that you can take advantage of to create a structure. SAT files are a commonly used file format to transfer 3D geometries between tools. Product vendors often provide SAT files of their products for convenience.

Using a Custom Housing to Enclose the PCB

This chapter is a continuation of "*ANA: Using Arbitrary 3D Structures in Analyst*". If you have not completed that example please do so before continuing here, as it includes concepts that may not be discussed in this example.

In this example you replace the SMA connectors that are attached to the PCB with a custom housing. The custom housing is a three-dimensional geometry that is pre-defined in a SAT file. SAT files define the geometry of a structure but have no concept of the materials that comprise the structure. Materials and ports must be assigned to the structure to make it ready for simulation.

This example includes the following main steps:

- Importing the SAT housing
- Assigning materials to the structure
- Running the full structure simulation

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the Cadence AWR Design Environment® platform. Choose **Help > Quick Reference** to access this document.

Opening an Existing Project

The example you create in this chapter is available in its complete form as *Analyst_SAT_Finish.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl**-click the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. This example is a continuation of the previous Analyst chapter.

To open an existing project:

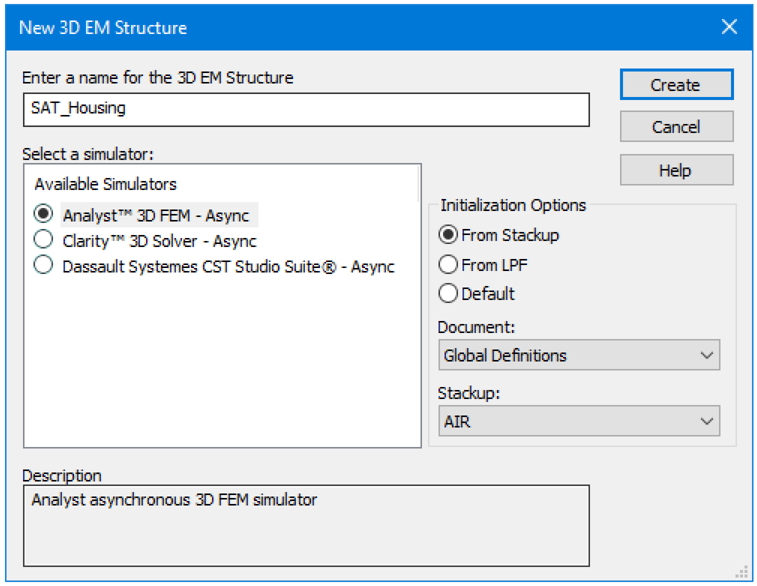
1. Choose **File > Open Example** to locate and open the *Analyst_Arbitrary_Finish.emp* file.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**Analyst_GS_SAT**" as the project name, and then click **Save**.

NOTE: Simulation results may vary slightly from the images in this guide. Finite Element Method (FEM) simulations require a convergence based on a mesh refinement sequence. Slight changes in the mesh refinement between versions of the solver can cause results to vary slightly. While the default convergence tolerance is sufficient for most geometries, if you find results shift you can decrease the convergence tolerance to ensure the results are accurate.

Importing the Housing

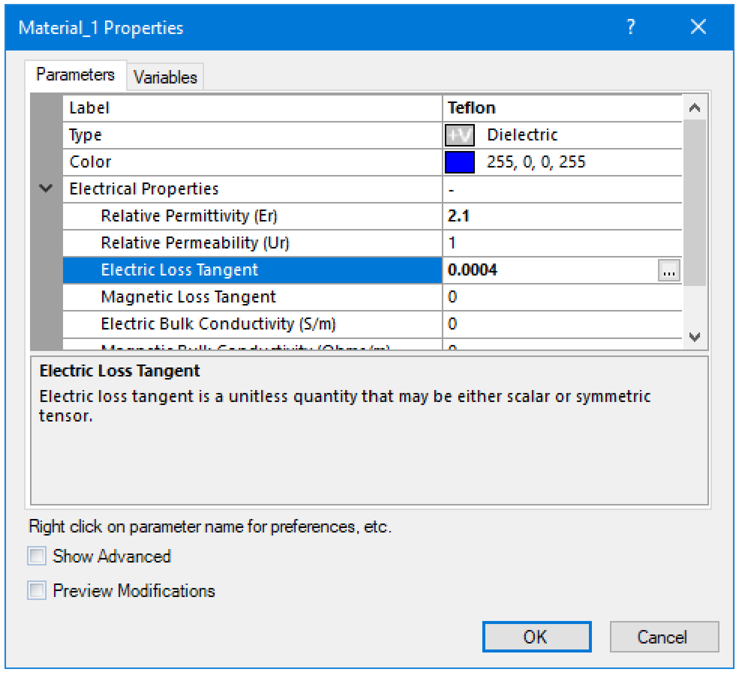
You can only import or modify SAT files in the AWR Design Environment platform 3D Editor. To import the SAT housing:

1. Right-click the **EM Structures** node in the Project Browser and choose **New Arbitrary 3D EM Structure** (alternatively, choose **Project > Add Arbitrary 3D EM Structure > New Arbitrary 3D EM Structure**) to display the New 3D EM Structure dialog box.
2. Type **SAT_Housing** as the structure name, select **Analyst™ 3D EM - Async** as the simulator, select **From Stackup** as the **Initialization Options**, select **AIR** as the **Stackup**, and then click **Create**.

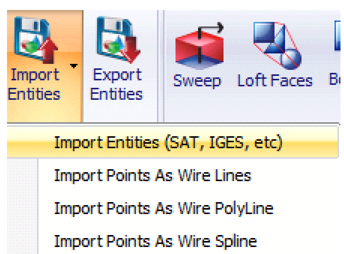


3. Right-click the “SAT_Housing” EM Structure and choose **Open in 3D Editor**.

   Next you define the materials that comprise the housing. Copper is a default material in the list and is used for the metal of the housing. The dielectric of the coaxial inputs is Teflon.
4. In the AWR Design Environment platform 3D Editor, right-click the **Materials** node in the Browser and choose **New Material**. Type "**Teflon**" as the **Label**, "**2.1**" as the **Relative Permittivity**, and "**0.0004** as the **Electric Loss Tangent**. Verify that your settings match those in the following figure.



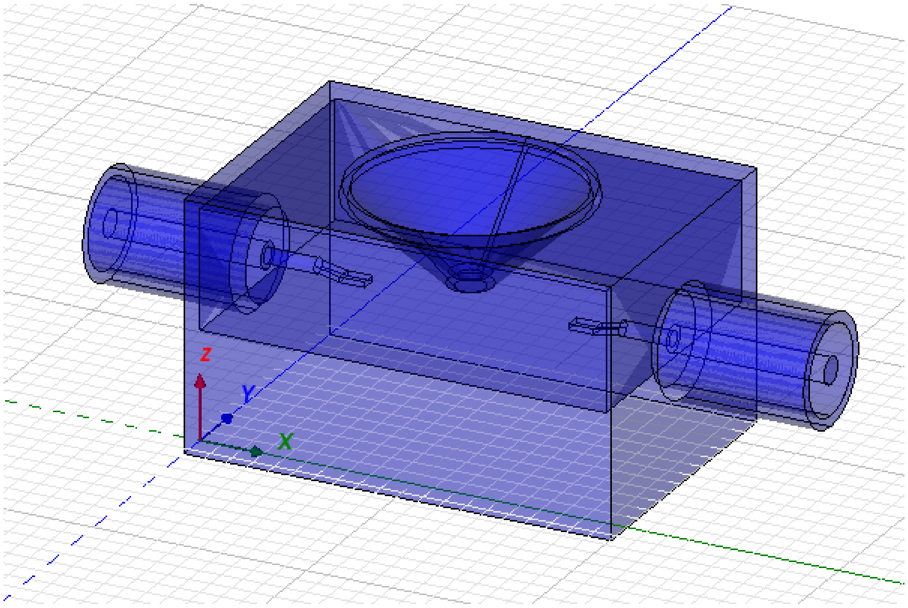
5. On the **Structure** ribbon in the **Entity Creation** group, click **Import Entities** and choose **Import Entities (SAT, IGES, STEP, etc)**.



6. In the File Open dialog box, browse to the *Analyst_SAT_Finish_Housing.sat* file in the \Examples directory. If you installed in the default location this path is similar to *C:\Program Files (x86)\AWR\AWRDE\22.1\examples*.

   If you used a different installation path, you can locate the \Examples directory by choosing **Help > Show Files/Directories** in the program and then double-clicking the **Examples** folder.

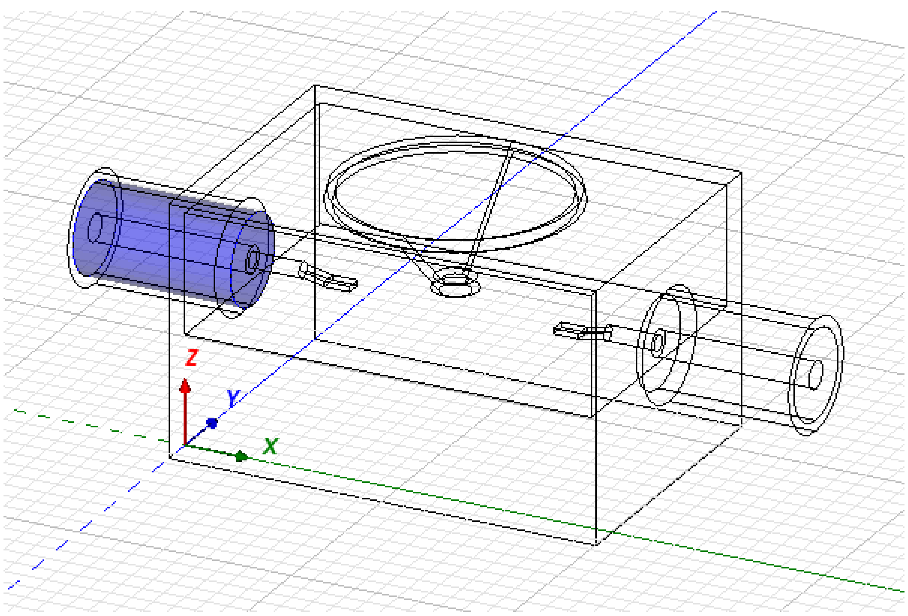
   The imported housing displays as follows:



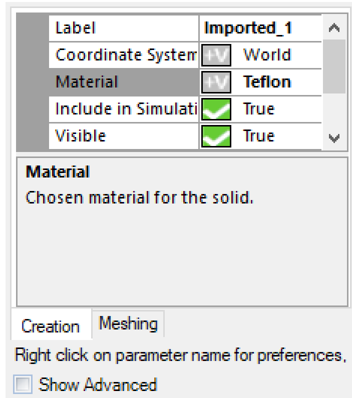
Assigning Materials to the Structure

You now assign materials to the housing structure. In the previous step you imported the geometry of the housing into the 3D editor. When imported, the material for each section of the geometry is by default set to “Air”. In this example you set the coax dielectric to “Teflon”, and both the housing and center conductors to “Copper”. You also add the ports.

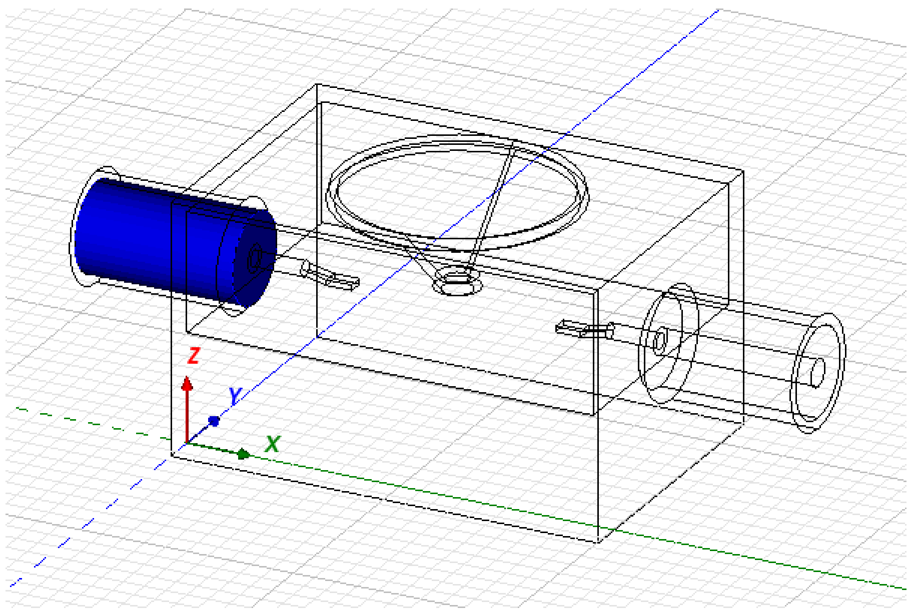
1. In the **Geometry** node of the Browser, select **Imported_1**. The left dielectric of the coax is highlighted.



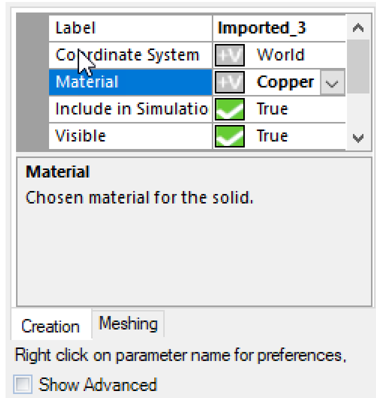
2. Change the **Material** from **Air** to **Teflon**.



The 3D view displays as follows.

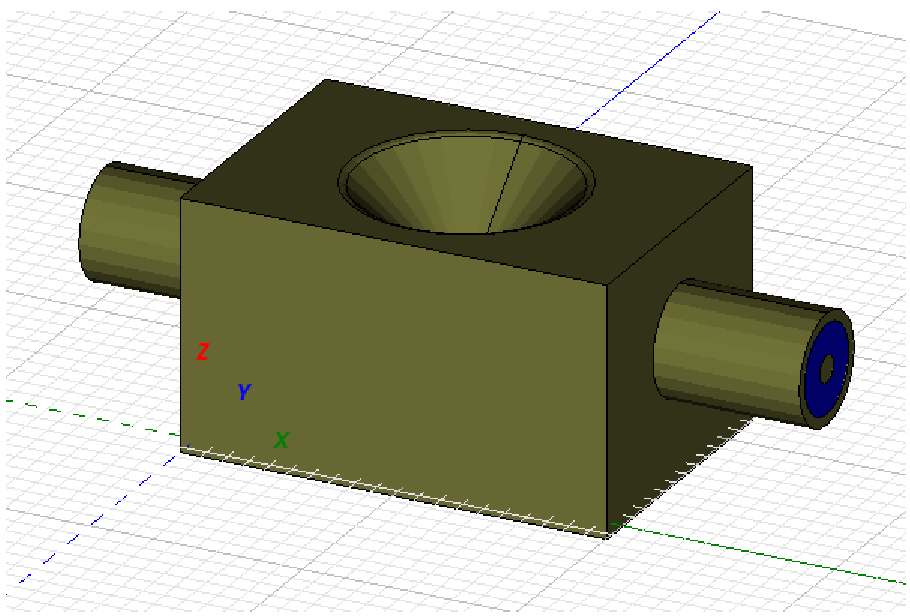


3. Repeat these steps for the right dielectric by selecting **Imported_2** and changing the **Material** to **Teflon**.
4. Select **Imported_3**. The right center conductor of the coax is highlighted.
5. Change the **Material** from **Air** to **Copper**.

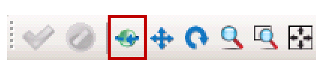


6. Repeat this step for the left conductor and the housing by selecting **Imported_4** and **Imported_5** and changing the **Material** to **Copper**.
7. Click the **Geometry** node to display the full structure, and then click the **Fit** button on the toolbar or press the **F** key.

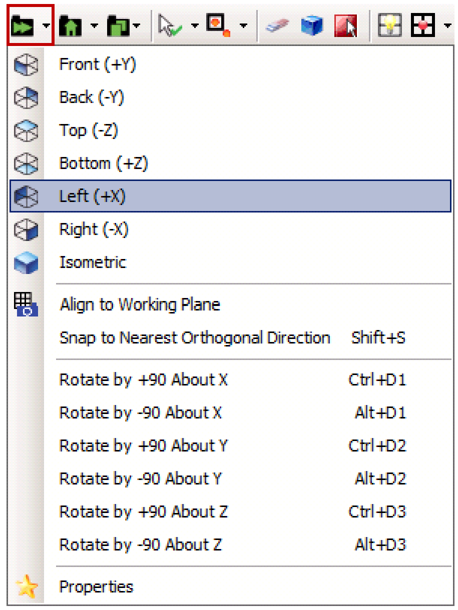
   The materials for the housing are now set and the 3D view should display as shown in the following figure.



8. To add the ports to the structure, view the structure from the left side. You can rotate the structure by clicking the middle mouse button and moving the cursor (alternatively, click the **Rotate** button on the toolbar or press the **R** key).

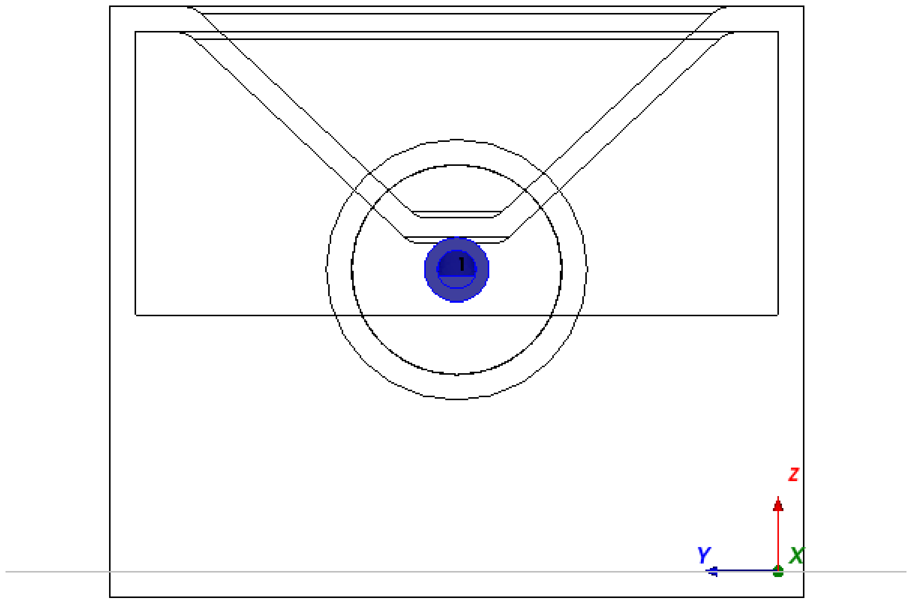


   You can also change the camera view by clicking the **Next Camera View** button on the toolbar and choosing **Left** from the drop-down menu.

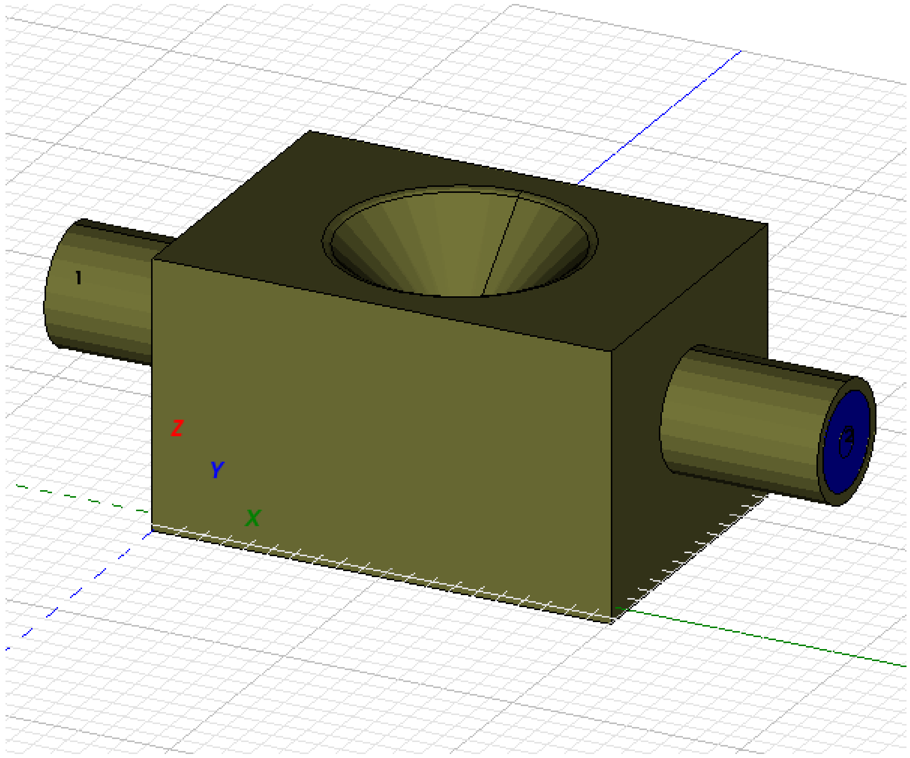


9. In the **Geometry** node of the Browser, select **Imported_4** to highlight the left conductor.
10. On the **Structure** ribbon in the **Attributes** group, click **Apply New** and choose **Port**.
11. Click the face of the conductor to apply the port to it, then press **Enter**. The Port Properties dialog box displays. The default settings for the port are sufficient so just click **OK**.

    The 3D view of the structure from the left should display as shown in the following figure.



12. Repeat these steps for the right side to add the second port. When complete, the housing should display as shown in the following figure.

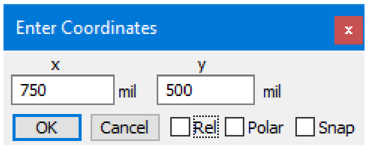


13. Click **Save** and close the AWR Design Environment platform 3D Editor to transfer the Geometry to the program.

Simulating the Structure

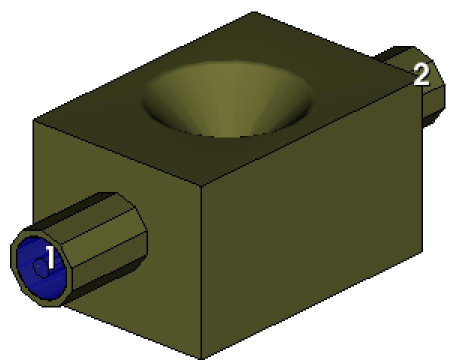
The next step places the PCB structure in the housing and set up the structure for simulation. You add the housing and the PCB as subcircuits in another top level EM structure. The hierarchical design style that is used in the *Analyst Getting Started Guide* examples is a powerful way to compare variations of structures that have portions of the structure that do not vary. In this example you previously simulated the PCB structure with edge launch SMA connectors and now you can compare the results with the PCB structure in a custom housing. While this example is simple, it is meant to illustrate the concept of a hierarchical 3D EM design style.

1. In the Project Browser, right-click the **EM Structures** node and choose **New EM Structure** to display the New EM Structure dialog box.
2. Type "**Housing**" as the EM Structure name, select **Analyst™ 3D EM - Async** as the simulator, select **AIR** as the **Stackup**, and then click **Create**.
3. In the 2D layout of the "Housing" EM structure, double-click the Enclosure. The edit diamonds display.
4. Click the top right edit diamond and begin to drag it. Press the **Tab** key to display the Enter Coordinates dialog. Clear the **Rel** check box and enter "**750**" for **x** and "**500**" for **y** then click **OK**.

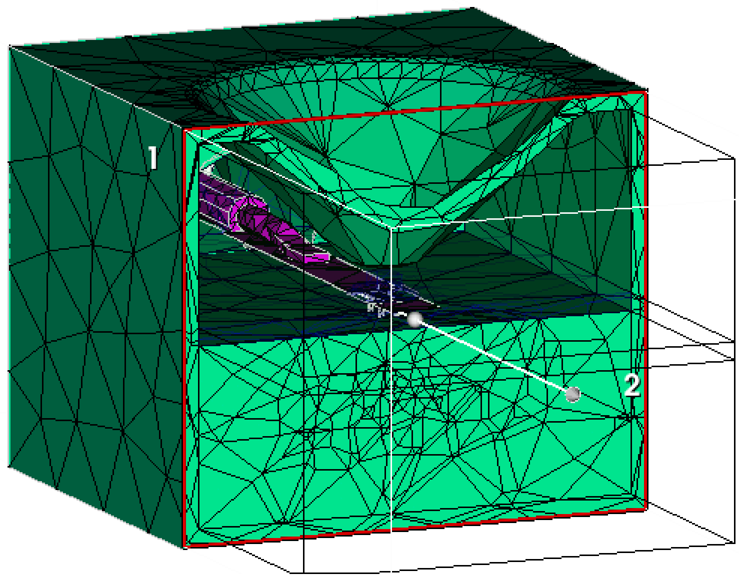


5. Choose **Draw > Add Subcircuit** and select **PCB**.
6. Press the **Ctrl** key to help align the PCB with the enclosure boundary. Right-click the subcircuit of the PCB and choose **Shape Properties**. Make sure that the PCB **Z Position** has an **EM layer** of **3**, then click **OK**.
7. Repeat the previous step, but choose the "SAT_Housing" instead of the PCB and set the **EM layer** to **Bottom Boundary** instead of **3**.
8. In the 2D layout view of “Housing”, choose **Draw > Add Edge Port** to add a port to the left end of the SAT_Housing coax. Repeat this for the right end as well.

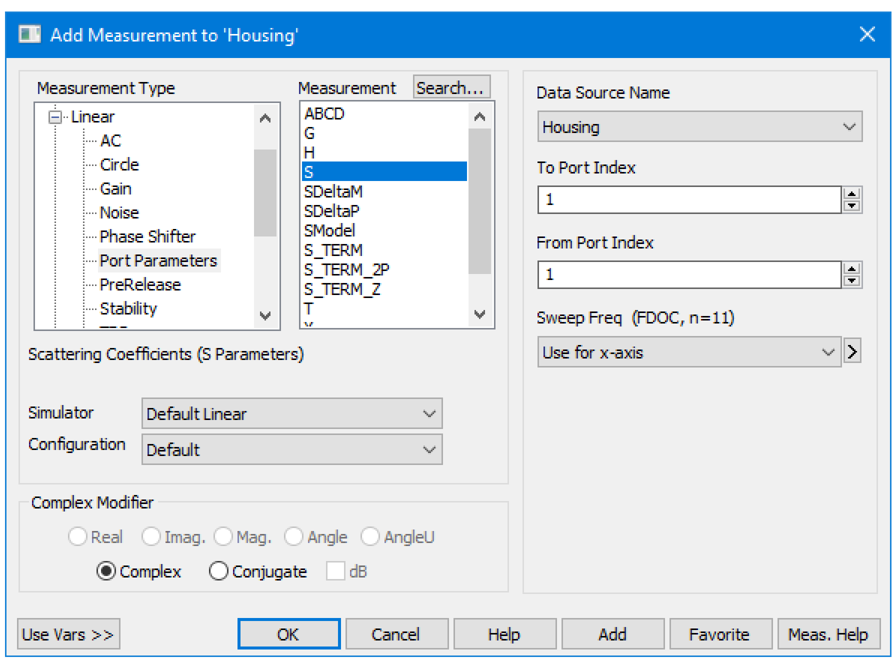
   The 3D view should display as shown in the following figure.



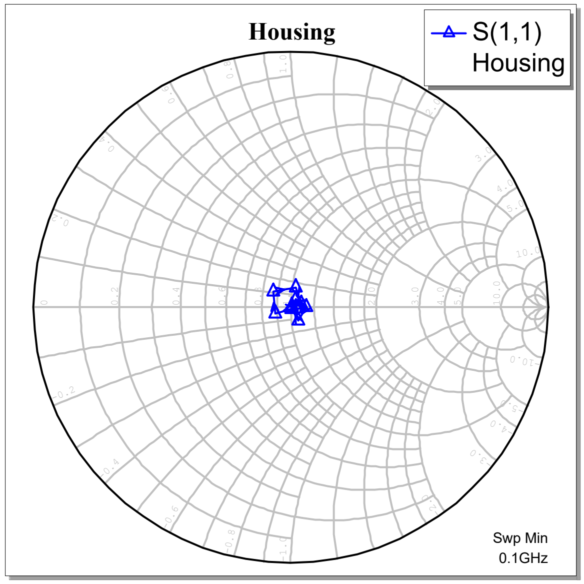
9. With the 3D layout view active, add a cut plane by clicking the **Use Cut Plane** button on the toolbar and then mesh the structure by clicking the **Show 3D Mesh** button on the toolbar (alternatively, right-click the EM structure and choose **Mesh**).
10. Click the **Show Mesh Connectivity** button on the toolbar. The 3D view should display as shown in the following figure. The colors, representing electrical connectivity, may vary from the figure.



11. Right-click the **Graphs** node and choose **New Graph**. In the New Graph dialog box, type **Housing** as the graph name, select **Smith Chart** as the graph type, and then click **Create**.
12. Right-click the new graph and choose **Add New Measurement** to display the Add/Modify Measurement dialog box.
13. Add "S11" to the graph. Verify that your settings match those in the following figure.



14. Right-click the "Housing" EM structure and choose **Simulate**. When the simulation is complete, the "Housing" graph should display as shown in the following figure.


Housing
S(1,1)
Housing
Swp Min
0.1GHz

Index

Q

R

S

T

W