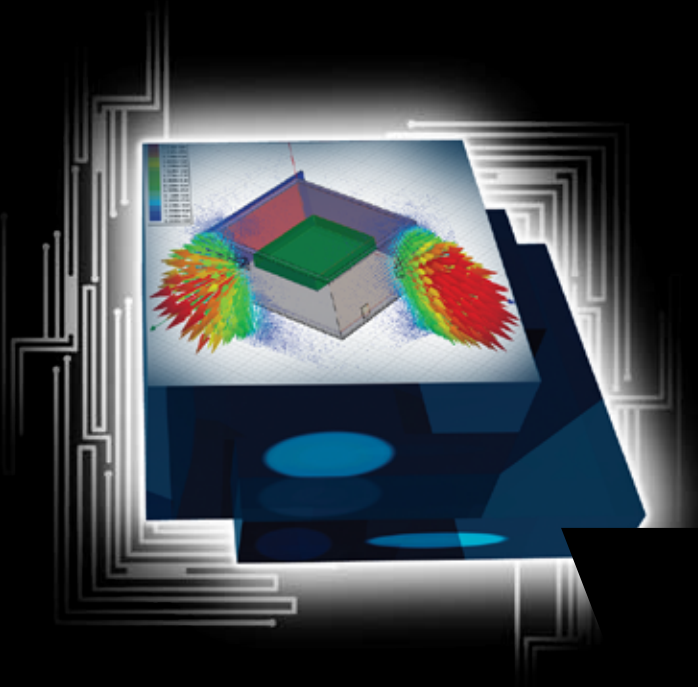


HFSS



**An Introduction to HFSS:
Fundamental Principles, Concepts, and Use**

An Introduction to HFSS:

**Fundamental Principles,
Concepts, and Use**

TABLE OF CONTENTS

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

ANSYS makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. ANSYS shall not be liable for errors contained herein or for incidental or consequential damage in connection with the furnishing, performance, or use of this material.

This document contains proprietary information that is protected by copyright. All rights are reserved.

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
USA

© 2013

Fundamentals of HFSS

The mathematical method used by HFSS.....	8
The adaptive solution process and its importance to HFSS.....	11
The computational volume and its parts.....	13
The six general steps in an HFSS simulation.....	14
The three solution types.....	16

HFSS Boundaries

Boundaries in HFSS and the need for them.....	21
The available boundaries within HFSS.....	24
Boundaries Examples – Perfect Electric Conductor.....	26
Boundaries Examples – Radiation Boundary.....	28
Boundaries Examples – Perfectly Matched Layer (PML).....	30
Boundaries Examples – Finite Conductivity.....	32
Boundaries Examples – Layered Impedance.....	34
Boundaries Examples – Impedance.....	36
Boundaries Examples – Symmetry.....	38
Boundaries Examples – Lumped RLC.....	40
Boundaries Examples – Master/Slave.....	42
Boundaries Examples – Screening Impedance.....	44
Boundaries Examples – Perfect H.....	45
Applying boundaries.....	46

HFSS Excitations

Excitations in HFSS.....	48
Excitations Example – Wave Ports.....	50
Excitations Example – Lumped Ports.....	54
The difference between lumped ports and wave ports.....	57

HFSS Solution Setup

The solution frequency.....	58
The Delta S setting.....	60
The maximum refinement per pass and maximum number of passes settings.....	62
The different frequency sweeps.....	64
The differences between local, remote and DSO solutions.....	67

HFSS Modeling GUI Basics

The modeling GUI windows.....	69
The various hotkeys.....	71
Snapping on to a point.....	73
Assigning boundaries in the GUI.....	75
Assigning How do I assign a Driven Modal Solution excitations in the GUI.....	76
Assigning Driven Terminal Solution excitations in the GUI.....	77
Assigning and creating materials.....	78
Creating variables.....	81

HFSS Post-Processing

Plotting S-parameter results.....	82
Exporting touchstone files.....	83
Advanced plotting of results.....	85
Plotting antenna results.....	87
Plotting field results.....	90
Creating animations.....	92

Appendix

Additional reading materials.....	93
-----------------------------------	----

PREFACE

Since the inception of ANSYS High Frequency Structure Simulator (HFSS™) in the late 1980's, thousands of engineers have used HFSS in the analysis of electromagnetic components. Initially used to model waveguide transitions, HFSS was quickly utilized for other engineering design challenges. HFSS is now used by designers in all segments of the electronics industry. Whether it is an airborne antenna system, an integrated circuit, a high-speed interconnect or any other type of electronic component, HFSS often is used during the design stage, and is an integral part of the design process.

It is the intent of this book to serve those engineers who are just now starting to tap into the power of electromagnetic design using simulation. It is also meant for those users that regularly rely on HFSS but do not use it every day.

The information in this book is a consolidation and review of materials from various internally and externally published ANSYS, INC sources, as well as rules of thumb developed by engineers within the company. The material is presented in a format that has been used and honed during many years of training novice HFSS users.

This book is not meant to serve as an exhaustive text on the underlying mathematics or all the capabilities of HFSS. It is also not intended to be the only reference an HFSS user should access. It is, however, meant to serve as a primer and a convenient quick reference guide that should answer eighty percent of a user's questions.

I would like to thank all those individuals who in my years at ANSYS, INC have shared with me their knowledge and understanding of HFSS. I also wish to give my appreciation to those individuals who created the original materials that serve as the basis for many of the items in this book. And finally, thanks go to the management at ANSYS, INC that encouraged me to create this text and the internal reviewers that helped shape its final content.

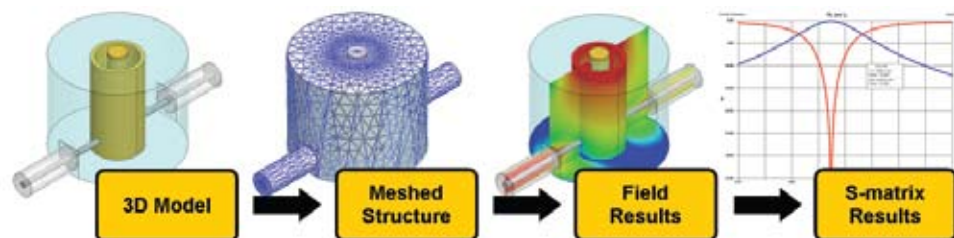
Markus Kopp

The Mathematical Method Used By HFSS

In Brief

HFSS™ uses a numerical technique called the Finite Element Method (FEM). This is a procedure where a structure is subdivided into many smaller subsections called finite elements. The finite elements used by HFSS are tetrahedra, and the entire collection of tetrahedra is called a mesh. A solution is found for the fields within the finite elements, and these fields are interrelated so that Maxwell's equations are satisfied across inter-element boundaries. Yielding a field solution for the entire, original, structure. Once the field solution has been found, the generalized S-matrix solution is determined.

In Pictures



In Detail

Mathematically, HFSS solves for the electric field \mathbf{E} using equation (1), subject to excitations and boundary conditions.

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \mathbf{E} \right) - k_0^2 \epsilon_r \mathbf{E} = 0 \quad (1)$$

where $\mu_r = \frac{\mu}{\mu_0}$, $\epsilon_r = \frac{\epsilon}{\epsilon_0}$, $k_0^2 = \omega^2 \epsilon_0 \mu_0 = \frac{\omega^2}{c^2}$

HFSS calculates the magnetic field \mathbf{H} using equation (2),

$$\mathbf{H} = \frac{1}{\omega \mu} \nabla \times \mathbf{E} \quad (2)$$

The remaining electromagnetic quantities are derived using the constitutive relations.

The above clearly implies that HFSS “thinks” in terms of electric and magnetic fields and not the more common concepts of voltages and currents. As a result, it is very important that an HFSS simulation encompasses a volume within which electric and magnetic fields exist. These volumes generally include conducting materials as well as the dielectric materials, including air, that surround the conductors.

In practice, to calculate the fields and S-matrix associated with a structure with ports, HFSS derives a finite element matrix using the above field equations. The following shows, in principle, the procedure that HFSS follows.

1. Divide the structure into a finite element mesh using tetrahedral elements.
2. Define testing functions W_n , for each tetrahedron, resulting in thousands of basis functions
3. Multiply field equation (1) by a W_n and integrate over the solution volume

$$\int_V \left(W_n \cdot \nabla \times \left(\frac{1}{\mu_r} \nabla \times \mathbf{E} \right) - k_0^2 \epsilon_r W_n \cdot \mathbf{E} \right) dV = 0 \quad (3)$$

This procedure yields thousands of equations for $n=1,2,\dots,N$

Manipulating the N equations, using Green's theorem and the divergence theorem yields:

$$\int_V \left[(\nabla \times W_n) \cdot \frac{1}{\mu_r} \nabla \times \mathbf{E} - k_0^2 \epsilon_r W_n \cdot \mathbf{E} \right] dV = \int_S (\text{boundary terms}) dS, \quad (3a)$$

for $n=1,2,\dots,N$ writing,

$$\mathbf{E} = \sum_m^N x_m W_m, \quad n = 1, 2, \dots, M \quad (4)$$

rewrites (3a) as,

$$\sum x_m \int_V \left[(\nabla \times W_n) \cdot \left(\frac{1}{\mu_r} \nabla \times W_m \right) - k_0^2 W_n \cdot W_m \right] dV = \int_S (\text{boundary terms}) dS \quad (5)$$

for $n=1,2,\dots,N$

Equation (5) then has the form

$$\sum x_m A_{n,m} = b_n, \quad n = 1, 2, \dots, N \quad (6)$$

or

$$\mathbf{Ax} = \mathbf{b} \quad (7)$$

In the matrix equation, A is a known $N \times N$ matrix that includes any applied boundary condition terms, while b contains the port excitations, voltage and current sources and incident waves.

Once you have solved for x , from equation 7 above, you know \mathbf{E} .

The above procedure implies that the solution process used by HFSS is straightforward and reasonably simple. However, this is not the case, and it is very important to note that the field solution process utilized by HFSS is actually an iterative process. HFSS uses the above process repeatedly, changing the mesh in a very deliberate manner, until the correct field solution is found. This repetitive process is known as the adaptive iterative solution process and is a key to the highly accurate results that HFSS provides. This process will be described in the next section.

As an example of this, consider a simple waveguide structure. Initially, HFSS will calculate the modes that can exist in the cross-section that serves as the excitation port of the waveguide. These modes depend on the port cross-section size, and port composition. HFSS uses a two dimensional FEM solver to calculate these modes. This initial calculation is referred to as the "port solution". Once the port modes are known, they are used to specify the b matrix.

Once the right hand side of equation 7. has been determined, HFSS computes the full three-dimensional electromagnetic fields within the solution volume using the adaptive solution process. When the final fields have been determined, HFSS derives the generalized S- matrix for the entire model.

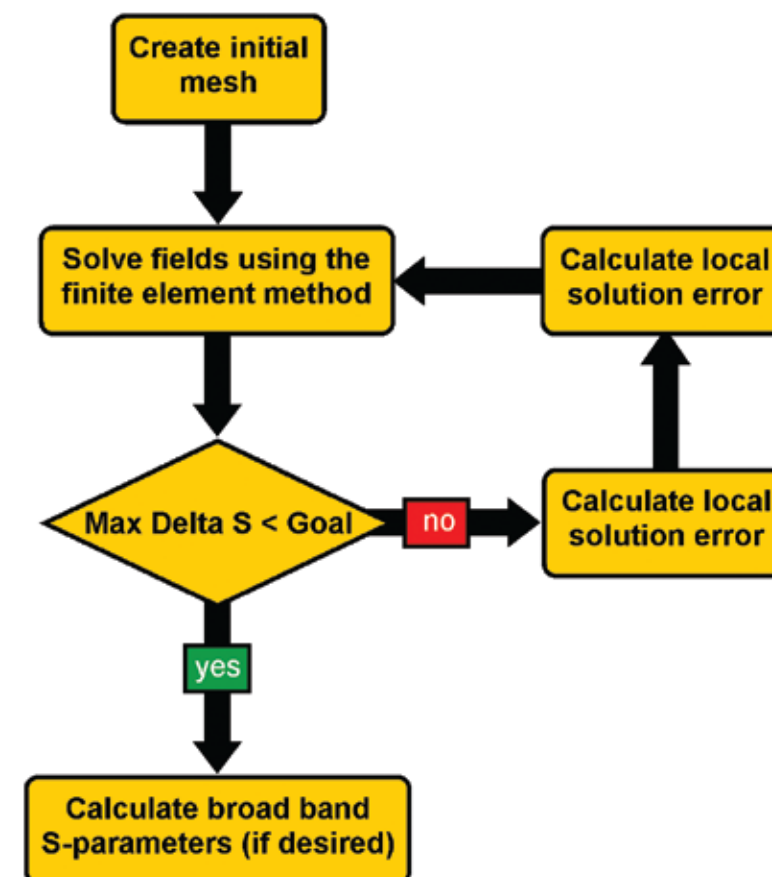
Noteworthy is the fact that the gamma results, and characteristic wave impedance Z_0 , that HFSS produces for a given simulation are actually determined by the port solution, and apply in a strict sense only to the transmission line that is the wave port.

The Adaptive Solution Process and Its Importance to HFSS

In Brief

The adaptive solution process is the method by which HFSS guarantees that a final answer to a given EM problem is the correct answer. It is a necessary part of the overall solution process and is the key reason why a user can have extreme confidence in HFSS's accuracy.

In Pictures



In Detail

The adaptive analysis is a solution process in which the mesh is refined iteratively. Refinement of the mesh is localized to regions where the electric field solution error is high. This adaptive refinement increases the solution's accuracy with each adaptive solution. The user sets the criteria that control mesh refinement during an adaptive field solution. Most HFSS problems can only be accurately solved by using the adaptive refinement process.

Following is the general process followed during an adaptive analysis:

1. HFSS generates an initial, geometrically conformal, mesh.
2. Using the initial mesh, HFSS computes the electromagnetic fields that exist inside the structure when it is excited at the solution frequency. (If you are running a frequency sweep, an adaptive solution is performed only at the specified solution frequency.)
3. Based on the current finite element solution, HFSS determines the regions of the problem domain where the exact solution has a high degree of error. A pre-defined percentage of tetrahedra in these regions is refined. Tetrahedra are refined by creating a number of smaller tetrahedra that replace the original larger element.
4. HFSS generates another solution using the refined mesh.
5. HFSS recomputes the error, and the iterative process (solve -> error analysis -> refine) repeats until the convergence criteria are satisfied or the requested number of adaptive passes is completed.
6. If a frequency sweep is being performed, HFSS then solves the problem at the other frequency points without further refining the mesh.

The above process will create a geometrically conformal, and electromagnetically appropriate, mesh for any arbitrary HFSS simulation. This ensures that HFSS will provide the correct result to a given simulation.

Mathematically, the error is computed along the following lines.

Let E^{approx} be the solution to step 2. above. This value is inserted back into

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \mathbf{E} \right) - k_0^2 \epsilon_r \mathbf{E} = 0 \quad (8)$$

yielding

$$\nabla \times \left(\frac{1}{\mu_r} \nabla \times \mathbf{E}^{approx} \right) - k_0^2 \epsilon_r \mathbf{E}^{approx} = \text{residue} \quad (9)$$

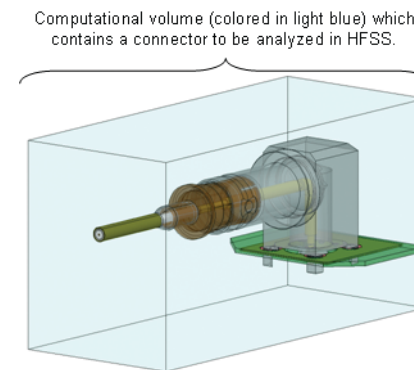
For each tetrahedron in the mesh, the residue function is evaluated. A percentage of the tetrahedra with high residue values are selected and refined.

The Computational Volume and Its Parts

In Brief

The computational volume, or solution space, is the volume within which HFSS explicitly calculates all EM fields. Any field quantities that are outside this volume can be derived from the fields within it.

In Pictures



In Detail

The computational volume consists of all the regions and objects within which a user wants to determine the electromagnetic fields. The outermost surfaces, or faces, of the computational volume are generally referred to as outer faces or outside boundaries. These faces are the outermost boundaries of the model, and HFSS explicitly calculates all fields within the solution space they define. All field quantities that are calculated by HFSS outside this volume are derived from the fields within the solution space.

The Six General Steps in an HFSS Simulation

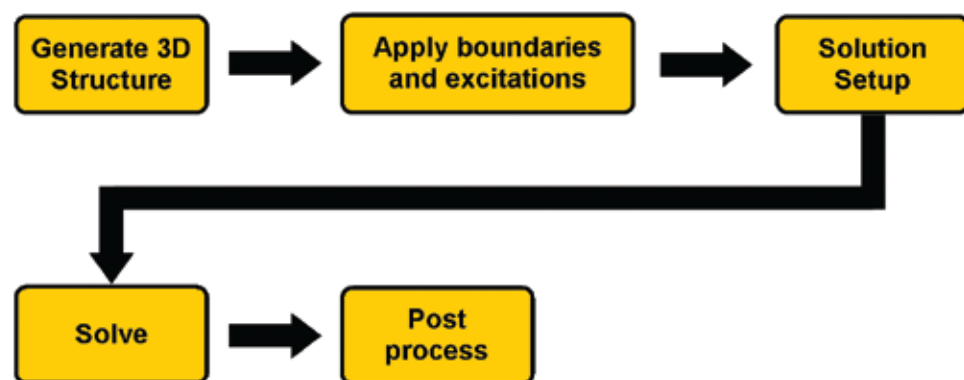
In Brief

There are six main steps to creating and solving a proper HFSS simulation.

They are:

1. Create model/geometry
2. Assign boundaries
3. Assign excitations
4. Set up the solution
5. Solve
6. Post-process the results

In Pictures



In Detail

Every HFSS simulation will involve, to some degree, all six of the above steps. While it is not necessary to follow these steps in exact order, it is good modeling practice to follow them in a consistent model-to-model manner.

Step One: The initial task in creating an HFSS model consists of the creation of the physical model that a user wishes to analyze. This model creation can be done within HFSS using the 3D modeller. The 3D modeller is fully parametric and will allow a user to create a structure that is variable with regard to geometric dimensions and material properties. A parametric structure, therefore, is very useful when final dimensions are not known or design is to be “tuned.” Alternatively, a user can import 3D structures from mechanical drawing packages, such as SolidWorks®, Pro/E® or AutoCAD®. However, imported structures do not retain any “history” of how they were created, so they will not be parameterizable upon import. If parameterization of the structure is desired, a user will need to manually modify the imported geometry so that parameterization is possible.

Step Two: The assignment of “boundaries” generally is done next. Boundaries are applied to specifically created 2D (sheet) objects or specific surfaces of 3D objects. Boundaries have a direct impact on the solutions that HFSS provides; therefore, users are encouraged to closely review the section on Boundaries in this document.

Step Three: After the boundaries have been assigned, the excitations (or ports) should be applied. As with boundaries, the excitations have a direct impact on the quality of the results that HFSS will yield for a given model. Because of this, users are again encouraged to closely review the section on excitations in this document. While the proper creation and use of excitations is important to obtaining the most accurate HFSS results, there are several convenient rules of thumb that a user can follow. These rules are described in the excitations section.

Step Four: Once boundaries and excitations have been created, the next step is to create a solution setup. During this step, a user will select a solution frequency, the desired convergence criteria, the maximum number of adaptive steps to perform, a frequency band over which solutions are desired, and what particular solution and frequency sweep methodology to use.

Step Five: When the initial four steps have been completed by an HFSS user, the model is now ready to be analyzed. The time required for an analysis is highly dependent upon the model geometry, the solution frequency, and available computer resources. A solution can take from a few seconds, to the time needed to get a coffee, to an overnight run. It is often beneficial to use the remote solve capability of HFSS to send a particular simulation run to another computer that is local to the user’s site. This will free up the user’s PC so it can be used to perform other work.

Step Six: Once the solution has finished, a user can post-process the results. Post-processing of results can be as simple as examining the S-parameters of the device modelled or plotting the fields in and around the structure. Users can also examine the far fields created by an antenna. In essence, any field quantity or S,Y,Z parameter can be plotted in the post-processor. Additionally, if a parameterized model has been analyzed, families of curves can be created.

All of the above topics will be discussed in greater detail in the following sections of this document.

The Three Solution Types

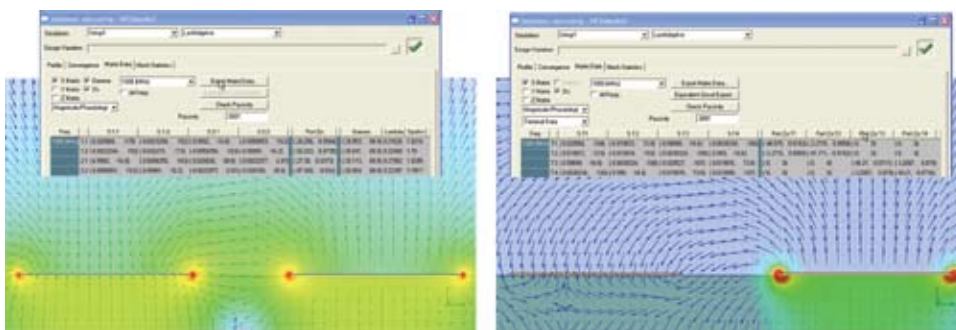
In Brief

When using HFSS, a user must initially specify what type of solution HFSS needs to calculate. There are three types of solutions available:

1. Driven Modal
2. Driven Terminal
3. Eigenmode

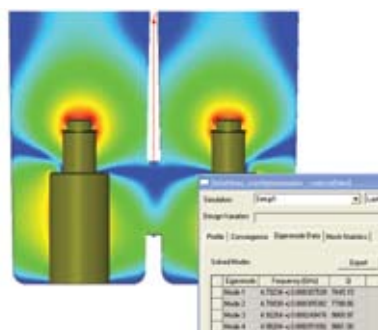
The solution type can be selected by clicking on HFSS in the main menu bar, selecting Solution type, and selecting the desired type from the menu.

In Pictures



Coplanar Microstrip structure analyzed using Driven Modal Solution Type showing odd mode of propagation E-field plot and S-matrix data.

Coplanar Microstrip structure analyzed using Driven Terminal Solution Type showing differential mode E-field plot and S-matrix data.



Coupled Cavity resonator analyzed using Eigenmode solver showing first Eigen frequency field plot and first four Eigen frequency data.

In Detail

HFSS has three solution types. Traditionally, the Driven Modal solution type is used for most HFSS simulations, especially those that include passive, high-frequency structures such as microstrips, waveguides, and transmission lines.

For simulations that deal with Signal Integrity, the newest solution type, Driven Terminal Mode, is used. These simulations generally include models that have multi-conductor transmission lines.

The driven modal and driven terminal solution types are similar and rely on the same mathematical calculations when solving a given problem. The difference in the two solution types is in the types of results that are available to a user. Simulations that use the driven modal solution type yield S-matrix solutions that will be expressed in terms of the incident and reflected powers of waveguide modes.

The S-matrix that is given by the driven terminal type, however, is expressed in terms of terminal voltages and currents.

As a practical example, if HFSS is used to model a pair of coplanar, parallel microstrip transmission lines, then the driven modal solution will yield results in terms of the even and odd modes that propagate on the structure. By contrast, the driven terminal mode solution will give the common and differential mode results.

The eigenmode solver will provide results in terms of eigenmodes or resonances of a given structure. This solver will provide the frequency of the resonances as well as the fields at a particular resonance.

Boundaries In HFSS and the Need For Them

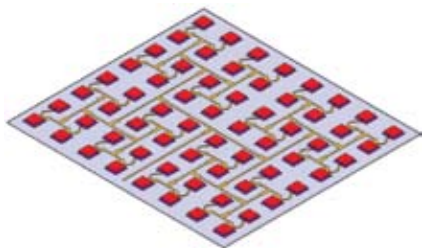
In Brief

Within the context of HFSS, boundaries exist for two main purposes:

1. to either create an open or a closed electromagnetic model or,
2. to simplify the electromagnetic or geometric complexity of the electromagnetic model.

In Pictures

Electromagnetically “Open” Structures

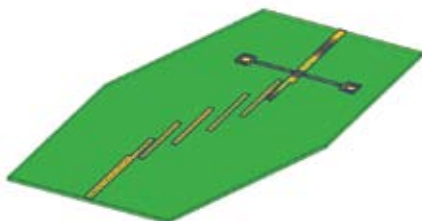


Antenna Array



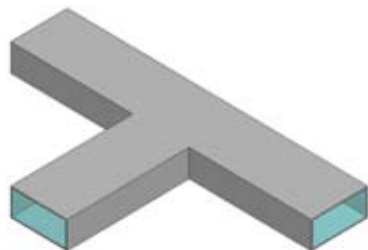
Printed Circuit Board

Electromagnetically Simplified Structure



Filter where microstrip traces are modeled using 2D objects with boundary applied.

Electromagnetically “Closed” Structures



Waveguide



Cavity Resonator

Geometrically Simplified Structure



Coaxial structure where “shield” is replaced by appropriate boundary.

In Detail

While the concept of boundaries can be confusing to an HFSS user, they can be simply thought of serving two main purposes. The first of these is to create either an open or a closed model.

A closed model simply represents a structure, or a solution volume, where no energy can escape except through an applied port. For an Eigenmode simulation, this could be a cavity resonator. For a driven modal or terminal solution, this could be a waveguide or some other fully enclosed structure.

An open model represents an electromagnetic model that allows electromagnetic energy to emanate or radiate away. Common examples would be an antenna, a PCB, or any structure that is not enclosed within a closed cavity.

While most HFSS simulations deal with models that are open, by default, HFSS initially assumes that any given model is closed. HFSS assumes all outer surfaces of the solution space are covered, or coated, by a perfect electric conductor boundary. In order to create an open model, a user will need to specify a boundary on the outer surfaces that will overwrite the default perfect electric conductor boundary.

The second reason why boundaries are used within HFSS is to decrease the geometric/electromagnetic complexity of a given structure or model. These boundaries should only be used internally to a model or possibly on a symmetry plane. They should be applied to specifically created 2D sheet objects or to specific surfaces of 3D objects. While boundaries can be very useful, a user should exercise caution when using them as they can create unintended results if applied incorrectly. Every HFSS model a user creates will use boundaries on the outer surfaces of the solution space. This is a direct result of the fact that a user must specify whether a given model is open or closed. As a result, any given HFSS model will always either have Conducting, Radiation, or Perfectly Matched Layer Boundary on all outer surfaces.

Conducting boundaries are the perfect electric conductor, finite conductivity, or impedance boundary.

Not every HFSS model, however, will use simplifying boundaries. When using boundaries to create simpler models, users should take care to not create a model that has unreasonable or inappropriate boundaries applied.

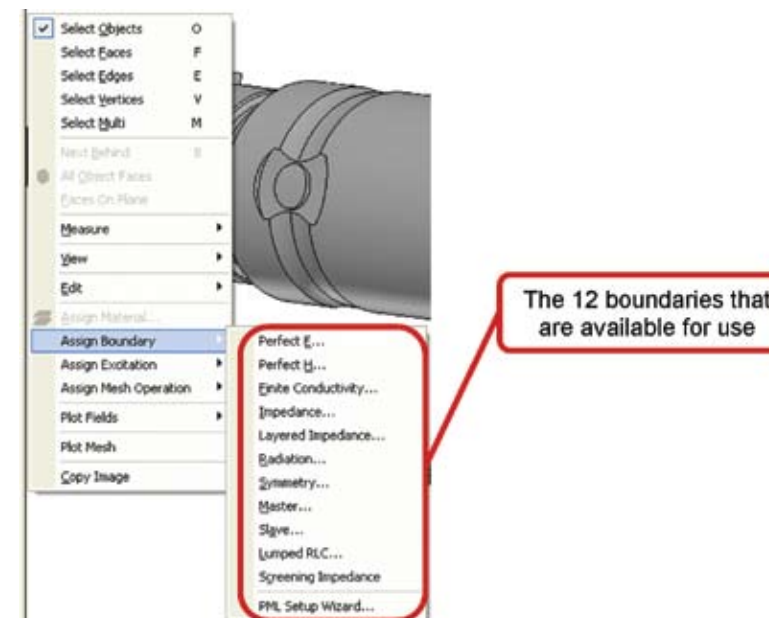
The Available Boundaries Within HFSS

In Brief

There are twelve boundaries available within HFSS. Boundaries are applied to specifically created 2D sheet objects, or surfaces of 3D objects. The twelve boundaries are:

1. Perfect Electric Conductor (PEC): default HFSS boundary fully encloses the solution space and creates a closed model
2. Radiation: used to create an open model
3. Perfectly Matched layer (PML): used to create an open model and preferred for antenna simulations
4. Finite Conductivity: allows creation of single layer conductors
5. Layered Impedance: allows creation of multilayer conductors and thin dielectrics
6. Impedance: allows creation of ohm per square material layers
7. Lumped RLC: allows creation of ideal lumped components
8. Symmetry: used to enforce a symmetry boundary
9. Master: used in conjunction with Slave Boundary to model infinitely large repeating array structures
10. Slave: used in conjunction with Master Boundary to model large infinitely repeating array structures
11. Screening Impedance: allows creation of large screens or grids
12. Perfect H: allows creation of a symmetry plane

In Pictures



In Detail

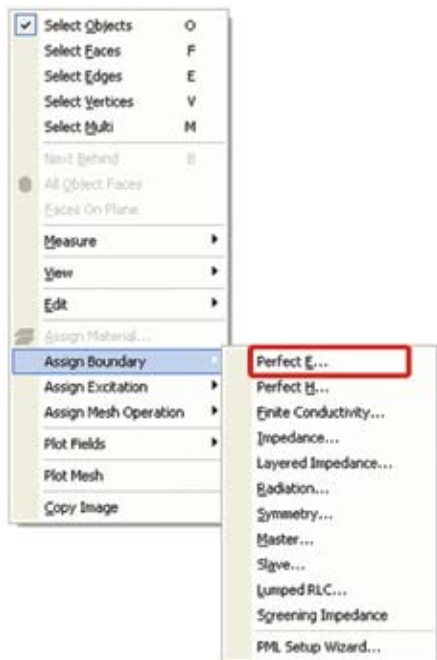
The twelve boundaries that are available in HFSS will each be discussed in detail in the following sections.

Boundaries Examples – Perfect Electric Conductor

In Brief

The Perfect Electric Conductor or PEC Boundary is the HFSS default boundary that is applied to all outer faces of the solution space. It represents a lossless perfect conductor. This default boundary creates a closed model. This boundary can also be used to create a symmetry plane if it is placed on an outer face of the solution space.

In Pictures



Cavity resonator showing the default Perfect Electric Boundary on all outer solution space surfaces

In Detail

The HFSS default boundary on all outer surfaces is a perfect electric conductor boundary. This is a result of the fact that the solution space is enveloped by an automatically created object called the background. There are no fields calculated within this background object as it is defined as a perfect conductor. As a result, all solution space faces that are outer faces will initially have a PEC boundary applied to them. Additionally, this background object fills any “voids” that exist within the solution volume. The implication is that any regions that are empty within a solution volume are filled by HFSS with PEC material.

The PEC boundary can also be used to create a symmetry plane in a model. The type of symmetry that this boundary will enforce is one where the E field components are normal to a given surface.

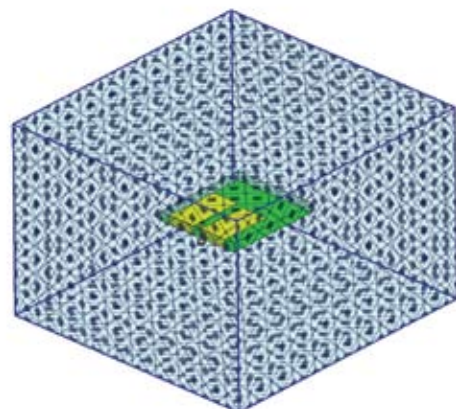
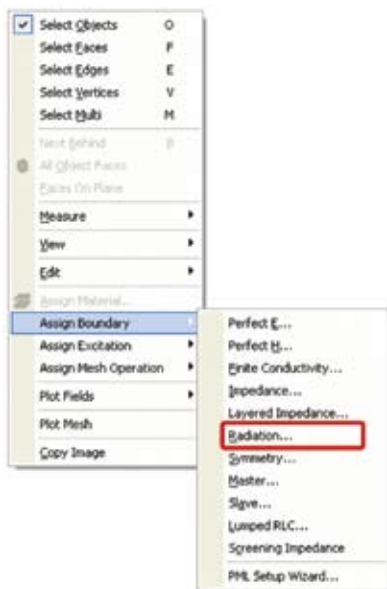
Additionally, the PEC boundary can be applied to 2D sheet objects to represent lossless conductors such as transmission line traces or patch antenna elements. Lastly, the PEC boundary when applied to a plane that represents the “ground” plane of a radiating structure has the option of being an Infinite Ground Plane. This is useful when modeling antennas with infinite ground planes.

Boundaries Examples – Radiation Boundary

In Brief

The Radiation Boundary is used to create an open model in HFSS. It should only be applied to outer faces of the solution space. If simulating an antenna, the radiation boundary should be placed a quarter wavelength away from any radiating surface.

In Pictures



Patch Antenna model showing the Radiation Boundary on all outer solution space surfaces

In Detail

A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space, such as antenna designs. HFSS absorbs the wave at the radiation boundary.

Radiation boundaries must be applied to outer faces of a model. If modeling an antenna, these faces must be at least a quarter wavelength away from any radiating surface.

At the radiation boundary surfaces, the second-order radiation boundary condition is used:

$$(\nabla \times \mathbf{E})_{tan} = jk_0 \mathbf{E}_{tan} - \frac{j}{k_0} \nabla_{tan} \times (\nabla_{tan} \times \mathbf{E}_{tan}) + \frac{j}{k_0} \nabla_{tan} (\nabla_{tan} \cdot \mathbf{E}_{tan}) \quad (10)$$

where

E_{tan} is the component of the E-field that is tangential to the surface,

k_0 is the free space phase constant $\omega\sqrt{\mu_0\epsilon_0}$,

and j is $\sqrt{-1}$

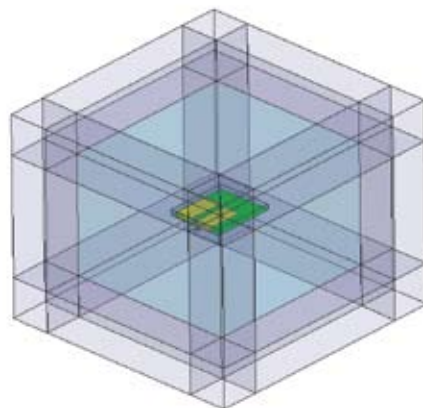
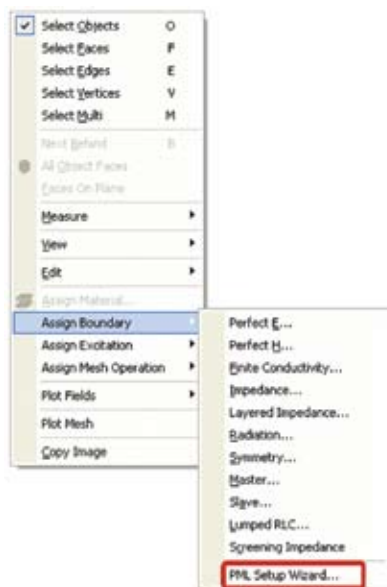
The second-order radiation boundary condition is an approximation of free space. The accuracy of the approximation depends on the distance between the boundary and the object from which the radiation emanates.

Boundaries Examples – Perfectly Matched Layer (PML)

In Brief

The Perfectly Matched Layer or PML Boundary is also used to create an open model. Like the Radiation Boundary, it should be applied only to outer faces of the solution space. It is also the preferred boundary to use when simulating antennas.

In Pictures



Patch Antenna model showing the *Perfectly Matched Layer Boundary* objects on all outer solution space surfaces

In Detail

Perfectly matched layers, while not boundaries in a strict sense, are fictitious materials that absorb the electromagnetic fields impinging upon them. These materials have complex and anisotropic material properties.

Perfectly matched layers are the preferred boundary to use when simulating antenna models. These antenna simulations can be single element, small antenna array, or infinite array.

Perfectly matched layers are more appropriate than radiation boundaries for antenna simulations. They can be used to reduce the solution volume if desired. For antenna modeling, however, it is best to keep the PML a quarter of a wavelength away from any radiating structure. Any homogenous isotropic material, including lossy materials such as ocean water, can surround the design.

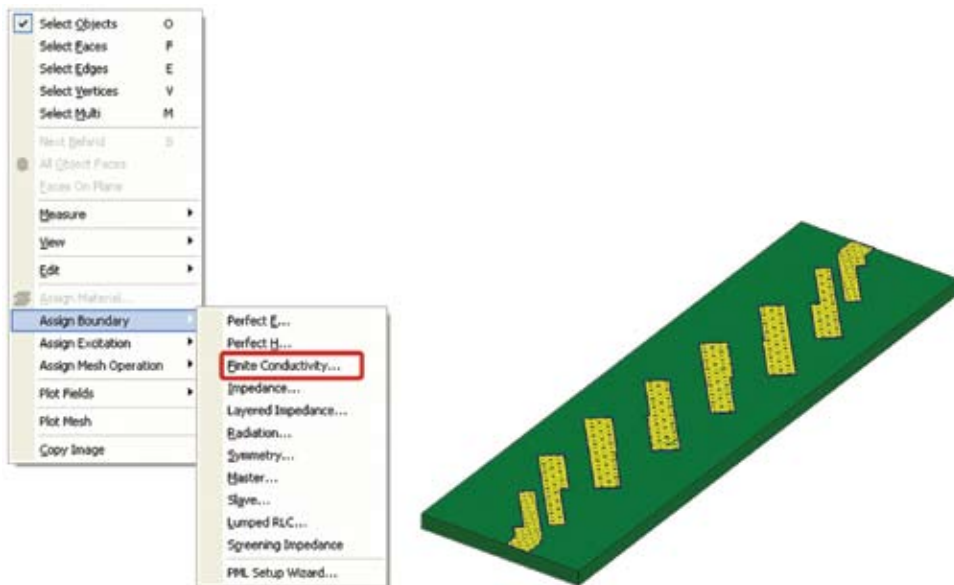
Perfectly matched boundaries are automatically generated with the aid of the PML Wizard. This wizard will guide a user through the creation of the PML objects/materials.

Boundaries Examples – Finite Conductivity

In Brief

The Finite Conductivity Boundary is used when it is desired to model conductors as 2D sheet objects. This can be useful when modeling traces on a PCB or planar antennas.

In Pictures



Filter model showing the Finite Conductivity Boundary applied to 2D sheet objects on top surface of a substrate

In Detail

The finite conductivity boundary is useful when modeling metal structures that are very thin but still thicker than a skin depth. This boundary can therefore be used to model signal traces, ground planes, or a radiating element. It also allows a user to specify the surface roughness of the conducting object that it is applied to. This boundary is applied to a sheet object that will represent the 3D conductor but has been drawn as a 2D object.

In HFSS, finite conductivity boundaries represent imperfect conductors. At the finite conductivity boundary, the following condition holds:

$$\mathbf{E}_{tan} = Z_s (\hat{n} \times \mathbf{H}_{tan}) \quad (11)$$

where

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

H_{tan} is the component of the H-field that is tangential to the surface.

Z_s is the surface impedance of the boundary, $\frac{1+j}{\delta\sigma}$ where,

δ is the skin depth, $\delta = \sqrt{\frac{2}{\omega\mu\epsilon}}$, of the conductor being modeled.

ω is the frequency of the excitation wave.

σ is the conductivity of the conductor.

μ is the permeability of the conductor.

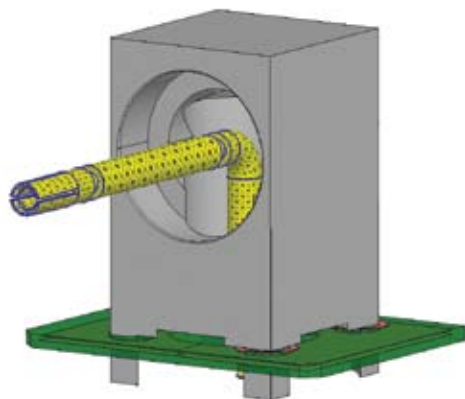
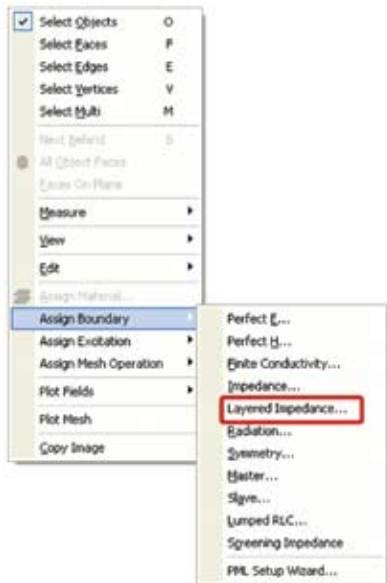
The finite conductivity boundary condition is valid only if the conductor being modeled is a good conductor; that is, if the conductor's thickness is much larger than the skin depth in the given frequency range. If the conductor's thickness is on the order of or less than the skin depth in the given frequency range, HFSS's layered impedance boundary condition should be used.

Boundaries Examples – Layered Impedance

In Brief

The Layered Impedance Boundary is used to model conducting objects that are composed of layers of conducting material. It is commonly used to model thin multiple plating layers.

In Pictures



Connector model (some parts hidden for clarity) showing the Layered Impedance Boundary applied to surfaces of the connector female center pin

In Detail

A layered impedance boundary is used to model multiple layers in a structure as one equivalent impedance surface. If desired, it can also take into account the conductor surface roughness. This boundary is extremely useful as it allows a user to simulate structures that would otherwise need layers of very small tetrahedra. This boundary then allows a user to quickly simulate a structure that would otherwise be very large and computationally intensive.

The layered impedance boundary can be specified as either internal or external depending on where they are applied by a user. If a user applies a layered impedance boundary to an outer face of the solution space, it is of the external type. If a user applies the layered impedance to an internal sheet object or face of a 3D object, then it is of the internal type.

The impedance of the layered structure is calculated by recursively calling the impedance calculation formulation, known from transmission line theory:

$Z_{input,k}$ is the input impedance for the k^{th} layer and

$$Z_{input,k} = Z_{w,k} \frac{Z_{input,k+1} \cosh(\gamma_k d_k) + Z_{w,k} \sinh(\gamma_k d_k)}{Z_{input,k+1} \sinh(\gamma_k d_k) + Z_{w,k} \cosh(\gamma_k d_k)} \quad (12)$$

where

$$Z_{w,k} = \sqrt{\frac{\mu_0 \mu_{rk}}{\epsilon_0 \epsilon_{rk}}}$$

$$\gamma_k = k_0 \sqrt{\mu_{rk} \epsilon_{rk}}$$

ϵ_{rk} is the relative complex permittivity of the k^{th} layer.

μ_{rk} is the relative complex permeability of the k^{th} layer

where

$$\epsilon_{rk} = epsr_k - j \left(\frac{\sigma_k}{\omega \epsilon_0} + epsr_k \cdot tandek \right)$$

$$\mu_{rk} = mur_k - j (mur_k \cdot tandmk)$$

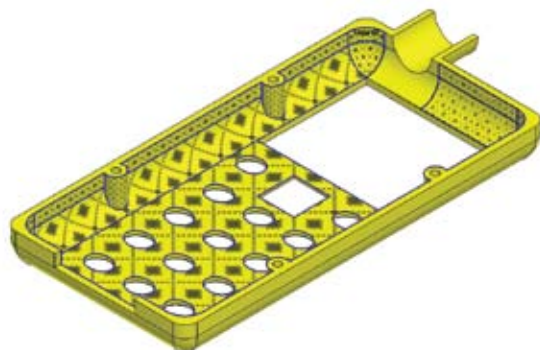
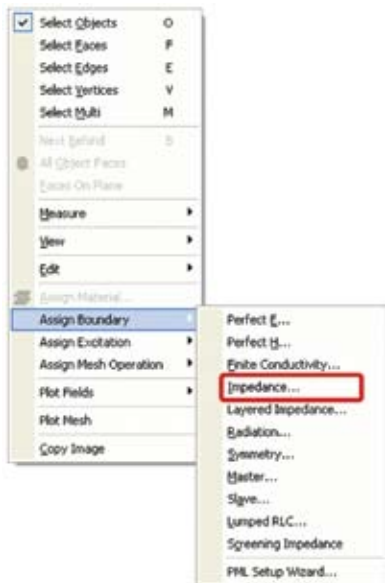
and where mur_k , $epsr_k$, σ_k , $tandek$, $tandmk$, d_k are all user input data specifying the k^{th} layer.

Boundaries Examples – Impedance

In Brief

The Impedance Boundary represents a resistive surface and is commonly used to simulate thin materials that have an ohms per square characterization or for thin film resistors.

In Pictures



Cell phone housing showing the *Impedance Boundary* applied to specific inner surfaces of housing in order to model conductive paint

In Detail

In HFSS, impedance boundaries represent surfaces of known impedance. The behaviorbehaviour of the field at the surface and the losses generated by the currents flowing on the surface are computed using analytical formulas.

Similar to finite conductivity boundaries, the following condition applies at impedance boundaries:

$$\mathbf{E}_{tan} = Z_s (\hat{n} \times \mathbf{H}_{tan}) \quad (13)$$

where \hat{n} is the is the unit vector that is normal to the surface.

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

H_{tan} the component of the H-field that is tangential to the surface.

Z_{sis} the surface impedance of the boundary, $R_s + jX_s$, where

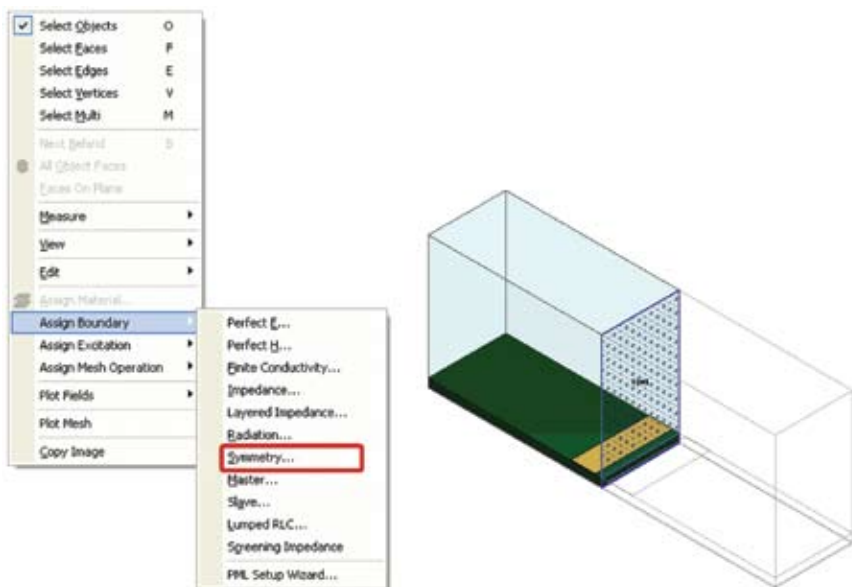
R_s and X_s are the user- defined resistance and reactance in ohms per square, respectively.

Boundaries Examples – Symmetry

In Brief

The Symmetry Boundary can be used to reduce the overall size of a model by applying it along a plane of geometric and/or electrical symmetry. The symmetry boundary has two variants: the E symmetry and H symmetry. When using a symmetry boundary, an impedance multiplier must be specified. This boundary is not available for models using the Driven Terminal Solution type.

In Pictures



Coplanar microstrip model showing the *Symmetry Boundary* applied to symmetry plane. (Grey ghost image only shown for clarity.)

In Detail

In HFSS, symmetry boundaries represent perfect E or perfect H planes of symmetry. Symmetry boundaries enable the user to model only part of a structure, which reduces the size or complexity of their design, thereby shortening the solution time. This boundary, however, is not available for simulations that use the driven terminal solution type.

When applying a symmetry boundary, the electric field is forced to either be tangential or normal to the symmetry plane. If an E-symmetry plane is used, the electric field is forced to be normal to the symmetry plane. Conversely, an H-symmetry plane forces the electric field to be tangential to the plane of symmetry.

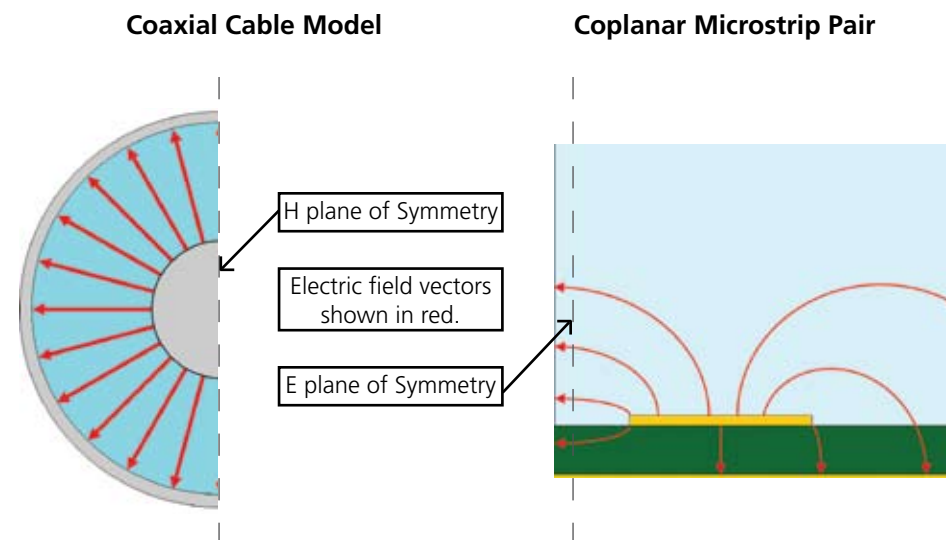
Symmetry planes can be applied only to outer faces of the solution space. The symmetry plane must be planar, and in an HFSS simulation, a maximum of three orthogonal symmetry planes can be used.

When using a symmetry boundary, an impedance multiplier must be set. The value of this impedance multiplier is determined by the type and number of symmetry planes used in a simulation. The impedance multiplier for models that use an E plane of symmetry is two because these models have one-half of the voltage differential and one-half of the power flow of the full structure. This results in an impedance that is one-half of that of the full structure.

Models that utilize an H plane symmetry boundary have the same voltage differential but half the power flow of the full structure, resulting in impedances that are twice those for the full structure. As a result, an impedance multiplier of one half must be used for models that use an H symmetry boundary.

If multiple symmetry planes are used or if only a wedge of a structure is modeled, you must multiply the individual impedance multipliers together.

Two common examples of a symmetry boundary are shown below. (Only the modeled halves are shown.)



The H symmetry is used since that boundary enforces a field condition that matches field behavior if no symmetry boundary where to be used.

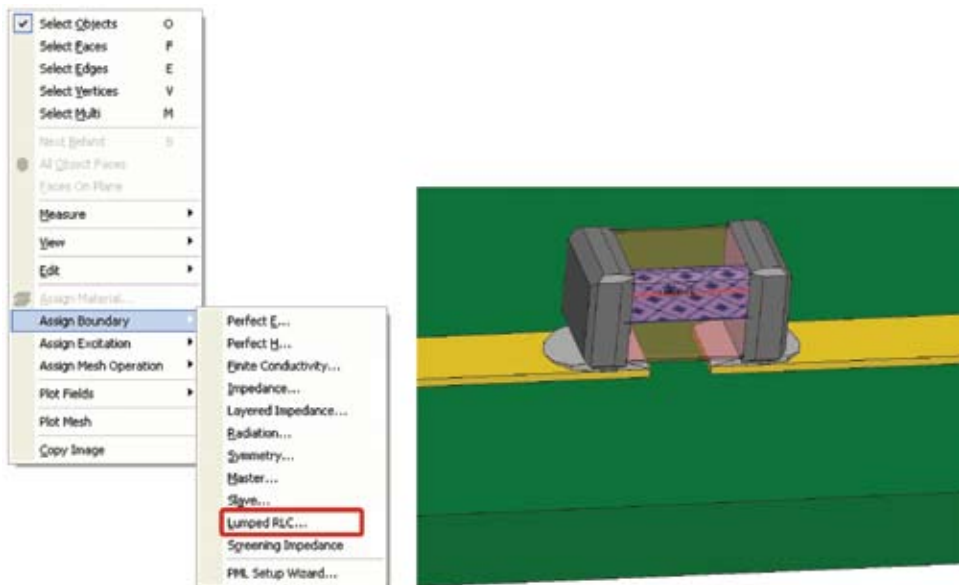
The E symmetry is used since that boundary enforces a field condition that matches the field behavior for the odd mode of propagation in the microstrip pair.

Boundaries Examples – Lumped RLC

In Brief

The Lumped RLC Boundary is used to model ideal lumped resistors, inductors or capacitors. This boundary can be used to model single elements or multiple R, L, or C in a parallel combination.

In Pictures

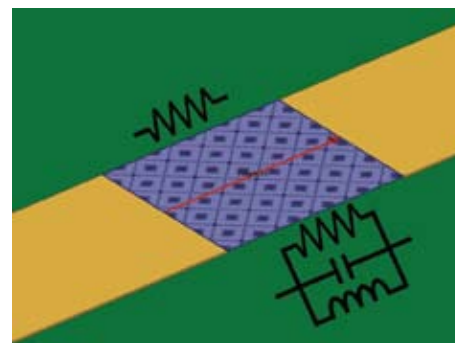


Capacitor on microstrip model showing the *RLC Boundary* applied to a plane internally to the capacitor body.

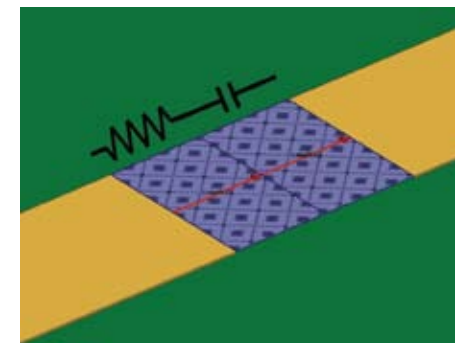
In Detail

The Lumped RLC boundary is a modified impedance boundary. Unlike the impedance boundary, this boundary can be used to directly specify a resistor, inductor, or capacitor value in an HFSS simulation. Once a user has specified the values of R, and/or L, and/or C, HFSS determines the impedance per square of the lumped RLC boundary at each frequency, effectively converting the RLC boundary to an impedance boundary.

A parallel combination of ideal passive components is achieved by simply specifying values for two or all three lumped component values within the same boundary dialog. To create series circuits, two separate RLC boundaries must be applied to two separate 2D objects which are arranged in an end-to-end fashion. This is shown in the graphic below.



Microstrip model showing the *RLC Boundary* applied to a 2D sheet object in order to model a single inline resistor, or inline parallel combination of lumped components.



Microstrip model showing the *RLC Boundary* applied to two end-to-end 2D sheet objects in order to model a series combination of inline resistor and capacitor.

Since HFSS converts the user input to an impedance, the impedance boundary is used; and as such, the following applies lumped RLC boundaries:

$$\mathbf{E}_{tan} = Z_s (\hat{n} \times \mathbf{H}_{tan}) \tag{14}$$

where \hat{n} is the unit vector that is normal to the surface.

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

H_{tan} the component of the H-field that is tangential to the surface.

Z_s is the surface impedance of the boundary, $R_s + jX_s$, where

R_s is the resistance in ohms per square.

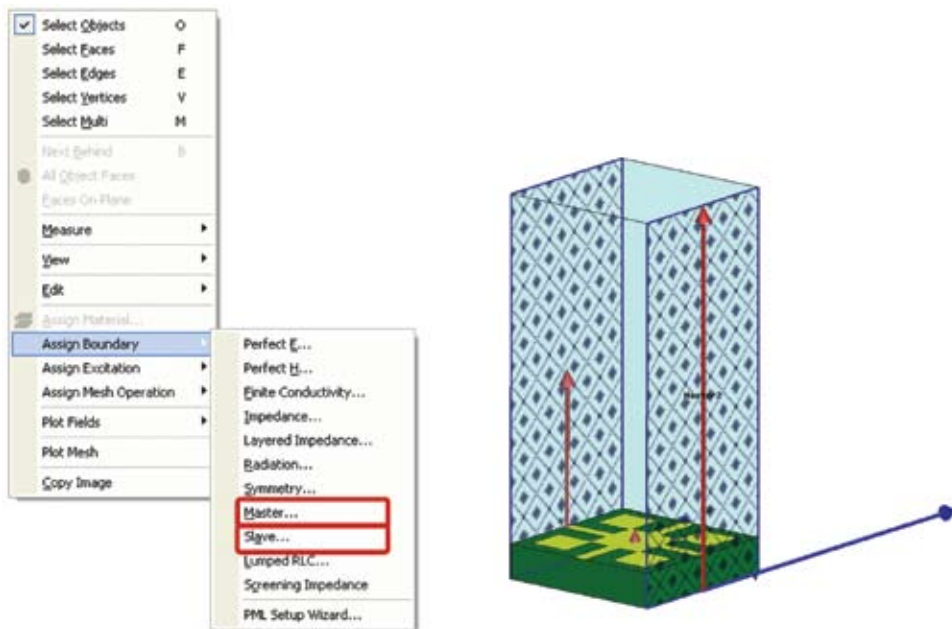
X_s is the reactance in ohms per square.

Boundaries – Examples Master/Slave

In Brief

The combination of Master and Slave boundaries is useful when modeling large repeating or periodic structures. Commonly these boundaries are used to model infinite antenna arrays or frequency selective surfaces.

In Pictures



Photonic bandgap model showing a *Master/Slave Boundary* pair applied to opposite faces of the solution space.

In Detail

The use of master/slave pairs allows a user to model periodic structures such as antenna arrays or frequency selective surfaces (FSS). When using a master and slave pair, the E-field on one surface, the slave surface is forced to match the E-field on another, the master surface, to within a phase difference.

Master/slave boundary pairs can only be assigned to outer faces of the solution space. These faces should be planar. Additionally, the geometry that contacts the master boundary must be identical to the geometry that contacts the slave boundary. If the geometries on the master/slave pair are not identical, the solution will fail.

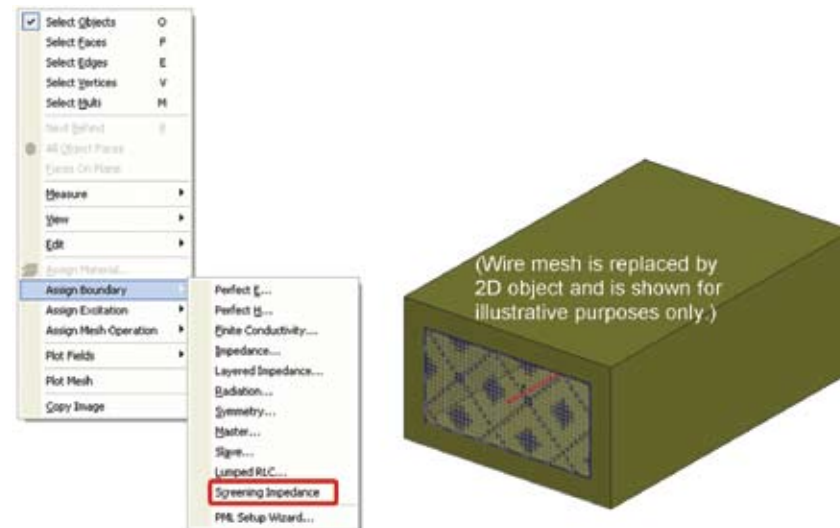
Each master and slave boundary needs a coordinate system that is specified by the user, which defines the plane on which the boundary exists. These coordinate systems must match each other. If they do not, HFSS will automatically transpose the slave boundary to match the master boundary. If the resulting master/slave surfaces do not have the same relative position, an error will occur.

Boundaries Examples – Screening Impedance

In Brief

The Screening Impedance Boundary is useful when simulating a large planar screen or grid that has a periodic structure.

In Pictures



Shielding enclosure model showing a *Screening Impedance Boundary* applied to a face that represents a wire mesh screen.

In Detail

Large planar metal screens or grids that are periodic can be replaced by a screening impedance boundary. This boundary applies a homogeneous characteristic impedance to the surface to create an equivalent electrical representation of the geometric grid pattern.

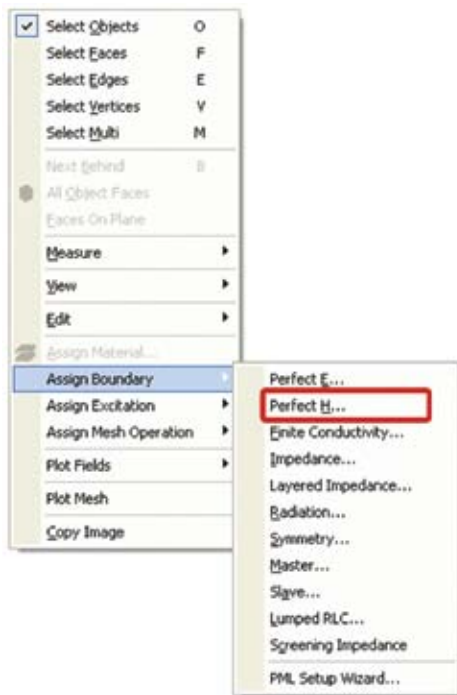
Use of the screening impedance necessitates use of a secondary HFSS simulation. In this secondary simulation, a user creates the repeating screening structure and solves it using Master/Slave boundaries. The resulting transmission and reflection coefficients are then used in the original HFSS project as input to the screening impedance boundary.

Boundaries Examples – Perfect H

In Brief

The Perfect H Boundary can be used to create a natural boundary through which fields propagate, or it can be used to model a perfect magnetic conductor.

In Pictures



In Detail

The perfect H boundary can be applied either internally to a model or at an outer face of the solution space. If it is applied internally, this boundary will force the tangential components of the H field to be identical on both sides of the surface to which it was applied. If this surface is a conducting body, then the perfect H boundary creates an aperture through which energy can propagate.

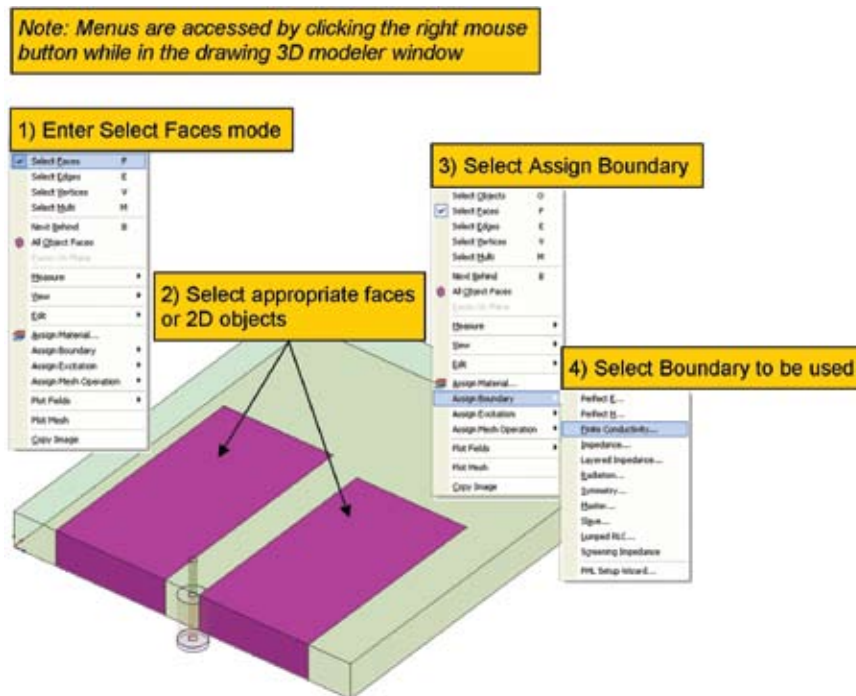
If this boundary is applied to an outer face of the solution space, it is equivalent to a perfect magnetic conductor, where the tangential H field is zero.

Applying boundaries

In Brief

Boundaries are applied either to specifically created 2D sheet objects or to and individual face or faces of one or more 3D objects.

In Pictures



In Detail

Boundaries are always applied to either specifically created 2D sheet objects or surfaces of 3D objects.

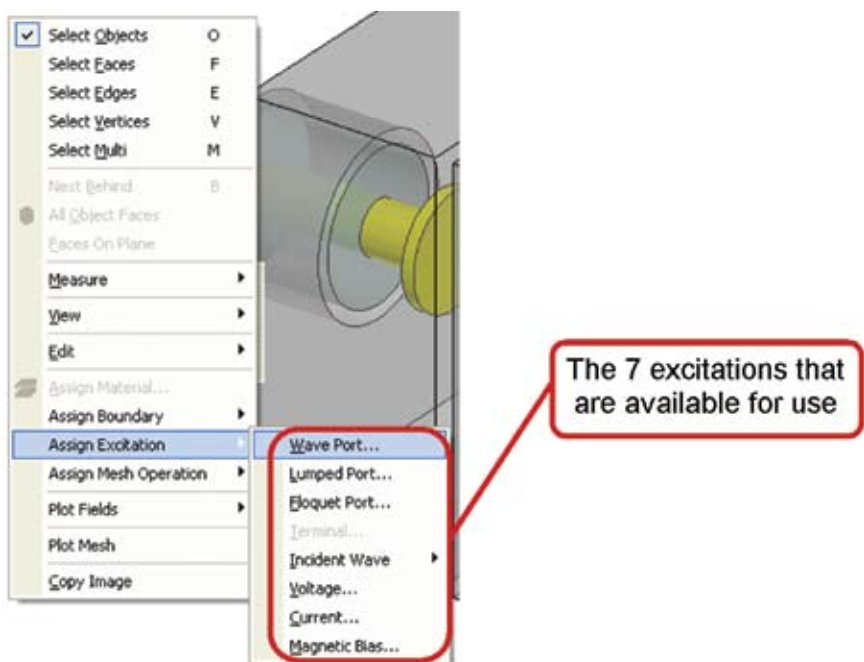
Boundaries are applied in the HFSS modeling window by selecting a face of a 3D object or a 2D sheet object and selecting the boundaries command. The subsequent menu will allow a user to select which boundary to apply to the selected face(s) or surface(s). If additional information is needed, a user will have to specify the appropriate information in the wizard dialogs that appear.

Excitations in HFSS

In Brief

There are seven types of excitations in HFSS: Wave Ports, Lumped Ports, Floquet Ports, Incident Fields, Current Sources, Voltage Sources and Magnetic Bias Source. All excitation types provide field information, but only the Wave port, Lumped Port, and Floquet port provide S-parameters. The use of the Magnetic Bias Source allows a user model a magnetic bias acting on a ferrite material.

In Pictures



In Detail

In HFSS, it is with the various excitations that a user can specify the sources of fields, voltages, charges or currents for a given simulation.

The most commonly used excitation types, or ports, are the wave port and the lumped port. These ports provide field information as well as S,Y,Z parameters and, in the case of the wave port, a port wave impedance and gamma, the propagation constant. The wave impedance and gamma values are related to the transmission line structure that is represented by the wave port.

For models where a magnetic bias is present, such as a circulator, the magnetic bias source can be used in conjunction with wave or lumped ports to create a model.

For simulations of large planar and periodic structures such as infinite antenna arrays, frequency selective surfaces or photonic bandgap structures, the Floquet port can be used.

If an ideal current or voltage source is desired, the current and voltage sources can be used. However, these sources will only provide field information and therefore are of limited use in an RF design environment.

Only the wave port and the lumped port types will be discussed in detail in the following sections.

Both the Wave Port and Lumped Port are available for use in both the Driven Modal Solution type and the Driven Terminal Solution Type. There is, however, a small difference in how the ports are set up.

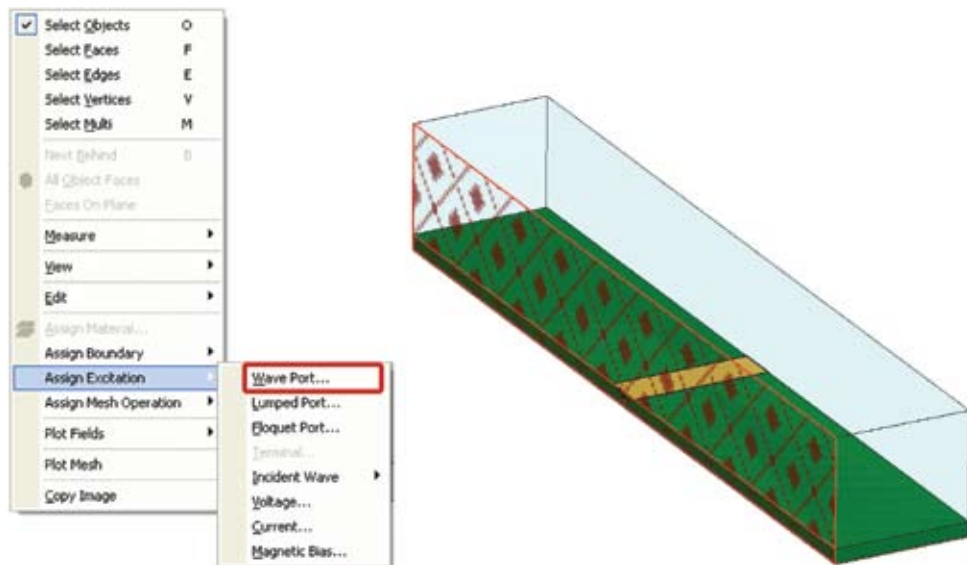
Excitations Example – Wave Ports

In Brief

A Wave Port is the most commonly used type of excitation used in HFSS. This port type is very useful for exciting microstrip, stripline, coaxial, or waveguide transmission lines. It should be applied only to an outer face of the solution space.

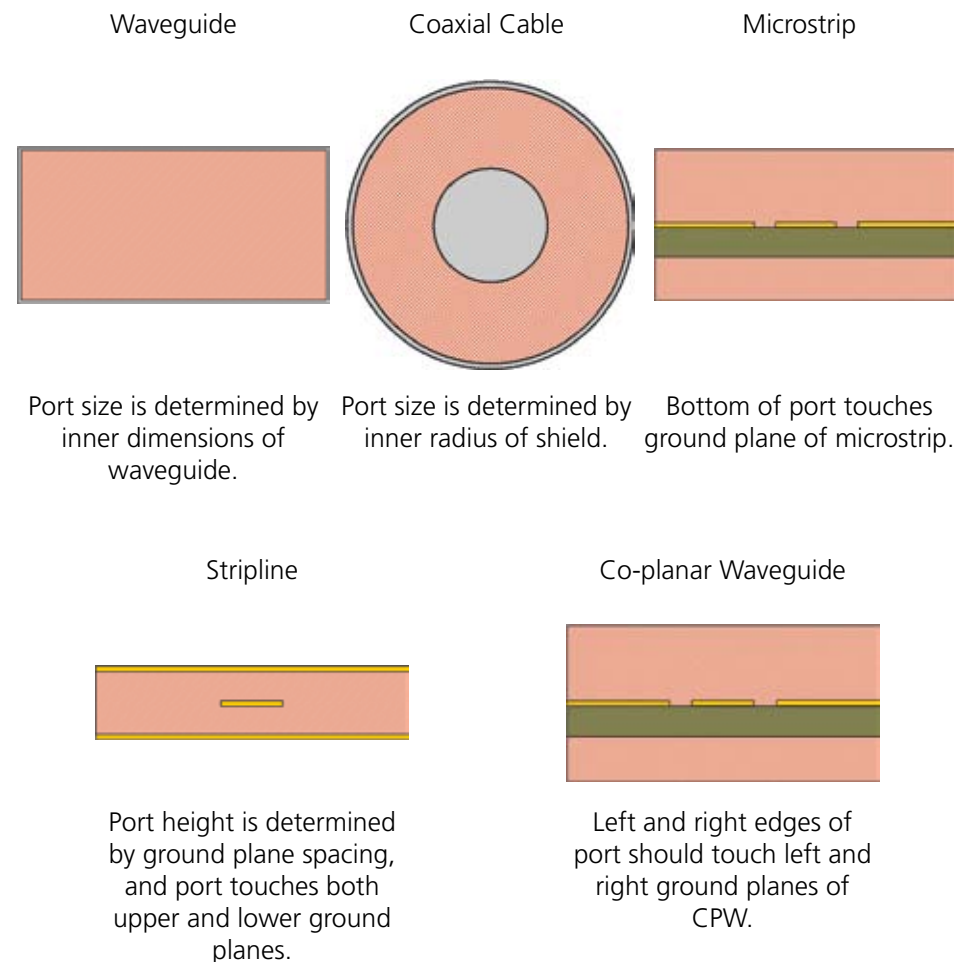
Shown below are examples of commonly used wave ports with proper size dimensions.

In Pictures



Microstrip model showing a *Wave Port* applied to all faces that form the front of the model.

Below are some additional typical wave port examples with appropriate dimensions.



Port size is determined by inner dimensions of waveguide. Port size is determined by inner radius of shield. Bottom of port touches ground plane of microstrip.

Port height is determined by ground plane spacing, and port touches both upper and lower ground planes. Left and right edges of port should touch left and right ground planes of CPW.

In the examples shown above, the rules of thumb for the port sizes are applicable for both driven modal and driven terminal solution types.

In Detail

A wave port represents the region or area where energy enters, or sources, the solution space. This port, therefore, is ideally suited to sourcing structures that are good transmission lines and should be applied only at outer faces of the solution volume. The wave port yields S,Y,Z parameters, characteristic wave impedance, and gamma, the propagation and attenuation constant. The S-parameters that are produced by a wave port are generalized and can be viewed as S-parameters that use the frequency-dependent characteristic wave impedance of the port as their normalization constant.

Since HFSS calculates gamma during the port solution, results can be de-embedded into or out of a port. This operation then will subtract or add transmission line length to the model changing the S-matrix accordingly.

Results can also be normalized to any constant complex impedance.

The solution to a wave port also represents the initial calculation that is performed by HFSS during the solution process. HFSS calculates a 2D solution for the wave port first and subsequently uses that solution as the source for the 3D model. When HFSS performs its solution sequence, HFSS assumes that each defined wave port is connected to a semi-infinite waveguide. HFSS also assumes that this semi-infinite waveguide has the exact cross-section and material properties as the port. It is the 2D fields in this semi-infinite waveguide that are solved initially. Those same fields are impressed onto the port region of the 3D model to obtain a solution to the 3D model.

A key consideration when making a wave port is port size. As stated, wave ports are regions where energy enters the solution space. Wave ports are also “attached” to virtual waveguides, and this is the fundamental reason why the exact size of a wave port is important. Ideally, the fields that are impressed onto the 3D model are an exact representation of how the fields behave in nature on the structure that is being simulated. But the impressed fields are actually the solution of the semi-infinite waveguide. This solution is affected by the walls of the waveguide and therefore can lead to the impressed port fields being incorrect. Consequently, the wave port should be made as large as needed to properly represent the fields, but no larger. In other words, the wave port is essentially bounded by PEC boundaries. Therefore, the port needs to be made large enough so that the PEC edges do not perturb the fields in the port area to a large degree.

Luckily, there are a number of rules of thumb that work well for common transmission lines. The dimensions and rules of thumb are given above in pictorial form. They are applicable for both the driven modal solution type or the driven terminal type.

HFSS generates a solution by exciting each wave port individually, where each desired incident mode contains one watt of time-averaged power. To find a solution to a given port, the desired port is energized with 1 watt of power while all other ports in the simulation are set to zero watts incident power.

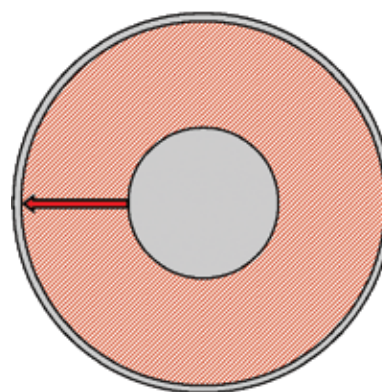
Wave ports can be used with the driven modal solution type or the driven terminal solution type. However, depending on the solution type chosen, the wave port setup is slightly different.

When creating a wave port in the driven modal solution type, a user must specify the number of modes desired. For waveguide simulations, the determination of needed, or wanted, modes is straightforward. But when a wave port is used to source a transmission line such as a microstripline, the number of modes should be set equal to the number of signal traces that are enclosed within the given wave port. For instance, for a co-planar pair of microstriplines enclosed within a single wave port, two modes should be specified. In the final solution, these two modes represent the even and odd modes of propagation.

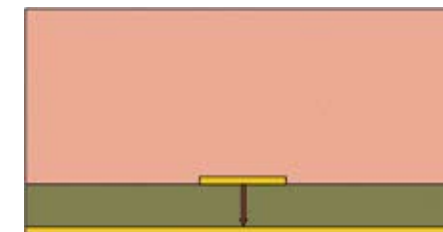
A user should also set an integration line in the port. While the integration line allows the calculation of the voltage-based wave impedance that is traveling on the transmission line, it also serves as a phase reference. As a result, integration lines should be drawn in a consistent manner in all ports. If integration lines are drawn in a non-consistent manner on different ports, additional 180-degree phase shifts will occur between the model ports. It is good modeling practice to draw integration lines between points of maximum potential difference in a wave port.

When creating a wave port in the driven terminal solution type, the location and number of modes needed is automatically determined by HFSS. Additionally, the proper number of integration lines needed is also automatically created by HFSS.

Some examples of integration lines are shown below.



Coaxial cable wave port showing integration line (in red). Line is drawn between points of maximum potential difference. In this case, it is drawn in the space between the center conductor and the shield.



Microstrip wave port showing integration line (in red). Line is drawn between points of maximum potential difference. In this case, it is drawn in the space between the signal line conductor and the “ground” plane.

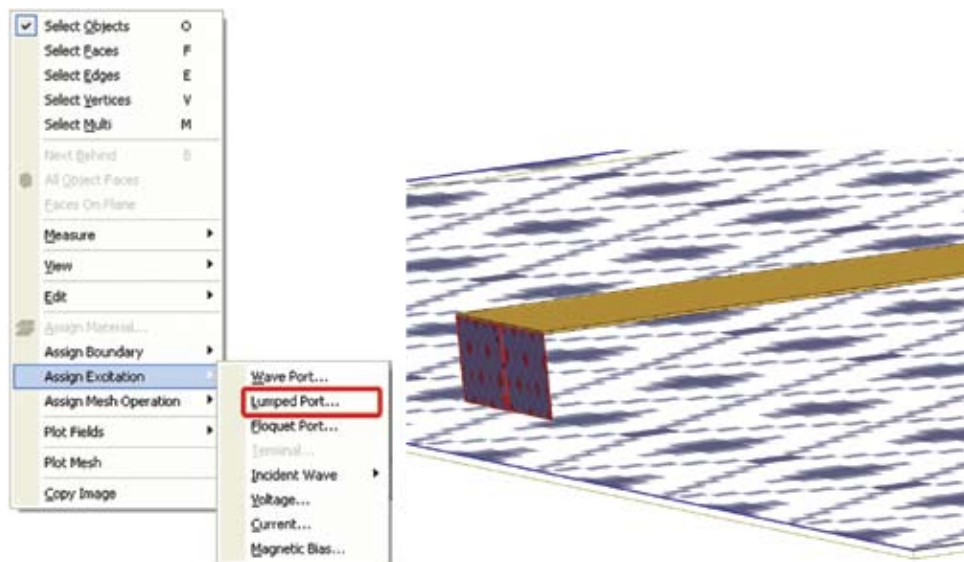
Excitations Example – Lumped Ports

In Brief

Lumped Ports are the other commonly used excitation type in HFSS. This port type is analogous to a current sheet source and can also be used to excite commonly used transmission lines. Lumped ports are also useful to excite voltage gaps or other instances where wave ports are not applicable. They should only be applied internally to the solution space.

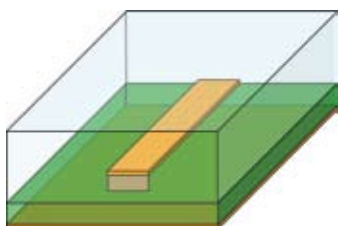
Shown below are examples of commonly used wave ports with proper size dimensions.

In Pictures

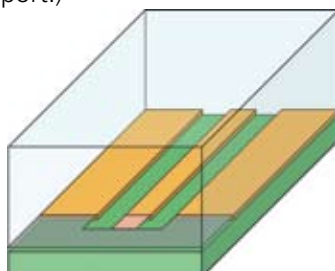


Microstrip model showing a *Lumped Port* applied between the signal trace and ground plane (textured fill).

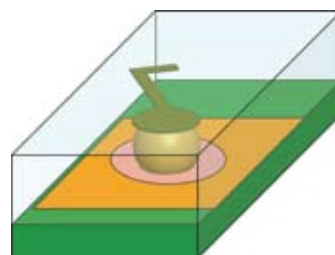
( region represents lumped port.)



Port is internal to the solution Space. The 2D port rectangle touches the signal trace with one edge and the opposite edge touches the ground plane.



Port is internal to Solution Space. The 2D port rectangle touches the signal trace with one edge, and the opposite edge touches user-drawn PEC objects (grey).



Port is internal to Solution Space. Port is an annular ring around BGA Ball.

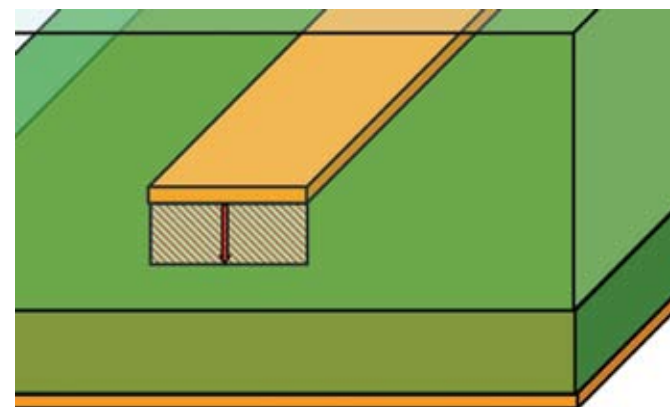
In Detail

Lumped ports are ports that can be used in simulations where energy needs to be sourced internally to a model. Lumped ports are simpler to create than wave ports but do not yield as much information as a wave port. Lumped ports yield S,Y,Z parameters and fields, but they do not yield any gamma or wave impedance information. The results of a lumped port cannot be de-embedded but can be renormalized. Unlike wave ports, lumped ports can support only a single mode. A lumped port can be defined on any 2D object that has edges which contact two conducting objects. The boundary that is applied to all edges that do not touch a conductor is a perfect H, which ensures that the normal electric field is equal to zero on those edges.

When creating a lumped port, it is necessary that a user draw an integration line for each port. This integration line should be drawn between the center points of the edges that contact metal objects. For an example of this, see the graphic at the end of this section.

The complex impedance Z_s , defined when the port was created, serves as the reference impedance of the S-matrix of the lumped port. The impedance Z_s , has the characteristics of a wave impedance; it is used to determine the strength of a source, such as the modal voltage V and modal current I , through complex power normalization.

It should also be noted that when the reference impedance is a complex value, the magnitude of the S-matrix is not always less than or equal to 1, even for a passive device.



Microstrip lumped port showing integration line (in red). Line is drawn between along the centerline of the port between the edges that contact metal objects.

The difference between lumped ports and wave ports

In Brief

Wave ports are applied at outer faces, yield S,Y,Z parameters, fields, wave impedance, gamma, and can be de-embedded.

Lumped ports are applied internally, yield S,Y,Z parameters and fields.

Both can be renormalized to a specific real impedance.

In Pictures

Port	applied	Gamma	Yields S,Y,Z	Renormalize	De-embed
Wave	externally	yes	yes	possible	possible
Lumped	internally	no	yes	possible	not possible

In Detail

The main differentiator between lumped ports and wave ports is the location of where they are applied to the model. Wave ports should only be applied at outer faces of the solution volume, whereas the lumped port should only be used internally to the solution volume.

Another key difference is that wave ports are specifically suited to sourcing good transmission lines, while lumped ports are well suited to sourcing structures that are not good transmission lines such as BGA balls, bondwires, etc.

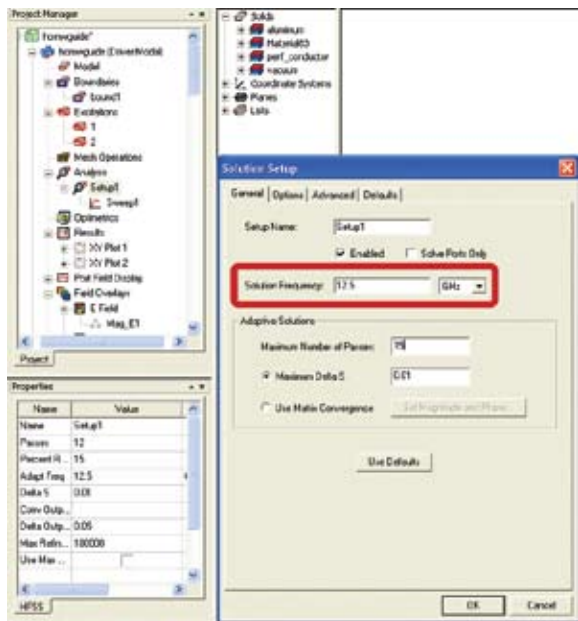
Wave ports also yield more information than a lumped port. While both ports yield fields and S-parameters, wave ports also yield gamma, the attenuation and propagation constants, as well as the wave impedance of the transmission line that is enclosed within the wave port. This information can be useful when designing transmission lines.

The solution frequency setting

In Brief

The solution frequency is used by HFSS to determine the maximum initial tetrahedra size and is the frequency at which HFSS explicitly solves the given model.

In Pictures



In Detail

The solution frequency is the frequency at which HFSS explicitly solves a given simulation. It is also at this frequency that the adaptive solution operates, and it is the fields at this frequency that are used to determine whether a model has converged or not.

The solution frequency should be set to the operating frequency of the device being simulated.

If a frequency sweep result is desired in a simulation, the solution frequency should be set to either the device operating frequency, the center frequency of the sweep, or to a frequency that is between 60 and 80 percent of the maximum frequency desired. The frequency that is used depends on what type of frequency sweep will be used.

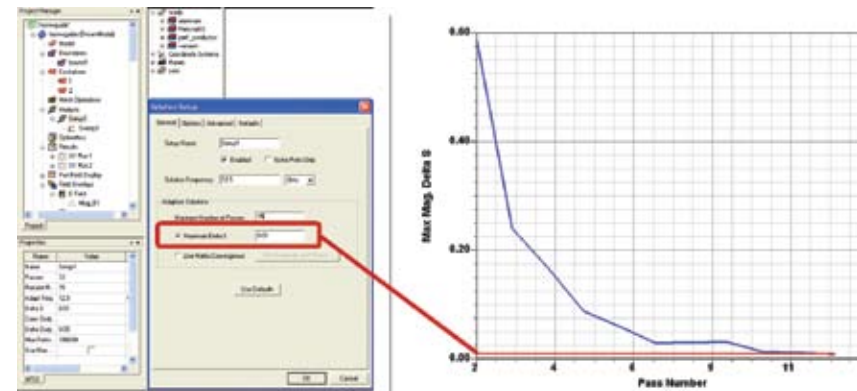
On a practical note, for most antenna simulations, the solution frequency should be set to the operating frequency of the antenna. For simulations of filters, the solution frequency should be set to the center of the band pass frequency. The solution frequency is also the frequency that should be used for any calculations the user performs when creating a model that depend on a frequency. Examples of these types of calculations are air region size for antenna problems, skin depth calculations, PML wizard input, etc.

The delta S setting

In Brief

The Delta-S parameter is the main convergence criterion used by HFSS when determining whether a model has converged or not.

In Pictures



In Detail

As mentioned, the adaptive process is a key element to ensuring that HFSS yields the correct answer. Because of the direct relationship between the electric fields in a simulation and the calculated S- matrix for that simulation, the convergence of the simulation is presented to a user via the delta-S value. The value of delta-S is the change in the magnitude of the S-parameters between two consecutive passes. Or, in electric field terms, the change in the electric field distribution between successive solutions. Once the magnitude and phases of all S- parameters change by less than the user-specified delta-S value, the analysis stops and is considered converged. Or conversely, again in electric field terms, once the electric fields are no longer changing in the given model, the field solution has converged and is correct.

If the desired delta-S parameter is never reached, HFSS will continue until the requested number of passes is completed.

The maximum delta-S is defined as

$$Max_{ij} [mag\{S_{ij}^N - S_{ij}^{N-1}\}] \quad (15)$$

where:

- i and j cover all matrix entries.
- N represents the pass number.

The delta-S number should be set between 0.005 and 0.01 for the majority of HFSS simulations.

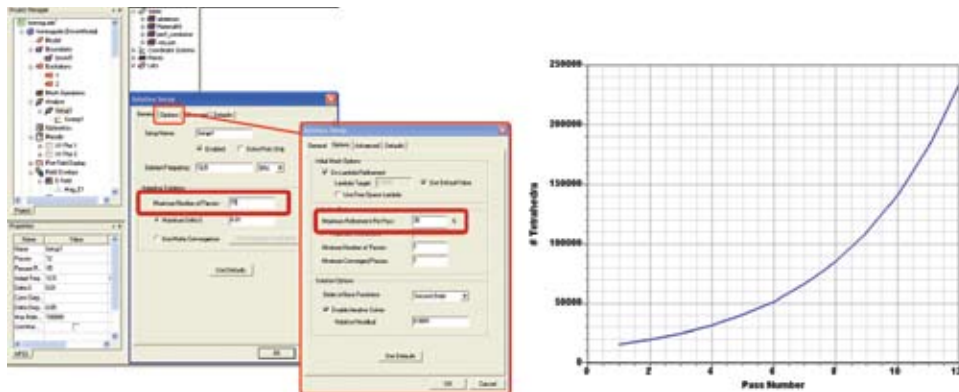
The Maximum Refinement Per Pass and Maximum Number of Passes Settings

In Brief

The Maximum number of passes is the maximum number of adaptive iterations HFSS performs in order to reach convergence.

The Maximum refinement per pass is the percentage of tetrahedral elements that are subdivided with each adaptive pass.

In Pictures



Plot showing number of tetrahedra increase versus adaptive pass. (Maximum refinement per pass set to 30%.)

In Detail

Refinement percentage and number of adaptive passes are both used in the adaptive solution process. The refinement percentage specifies the largest number of tetrahedra that can be subdivided per adaptive pass.

The maximum number of adaptive passes is the maximum number of times HFSS will refine the mesh in order to try and converge to an answer.

The adaptive solution process uses the delta-S, maximum refinement per pass, and maximum number of passes to converge to the correct answer. The delta-S and maximum number of passes determine when HFSS will stop the adaptive solution process. If convergence is reached before the maximum number of passes has been performed, the solution process will stop. HFSS will stop if convergence is not reached, but the maximum number of passes has been reached. In such cases, it is recommended to increase the number of passes so that HFSS can reach convergence.

The Different Frequency Sweeps

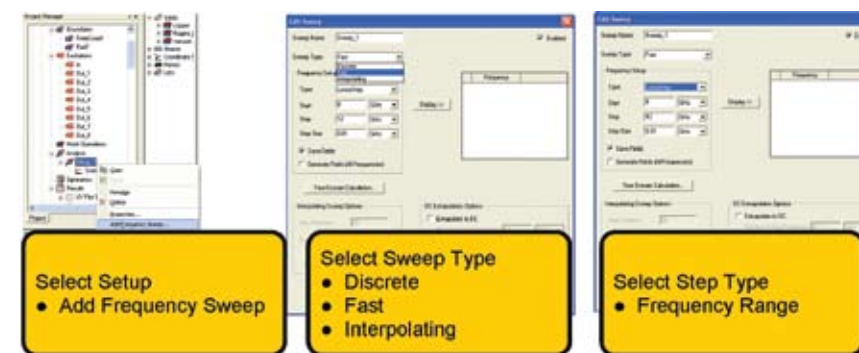
In Brief

HFSS has three distinct sweep types: the discrete sweep, the fast sweep, and the interpolating sweep. Depending on the needs of a user, a particular sweep type may be preferred. Generally, the solution times required for a frequency sweep type increase in the following order: fast, interpolating, and discrete.

But, for solutions that require field information at only a few (less than five) discrete frequency points, the discrete sweep can be faster than either of the other two. The fast sweep is useful when many frequency points are desired over a limited frequency range. The interpolating sweep is most useful when solving problems from DC to a high frequency.

For both the interpolating and fast sweeps, the number of desired frequency points is not related to the time it takes to generate the frequency sweep results. Both of these sweeps, in essence, generate a pole-zero transfer function, and it is the generation of this function that requires the majority of the solution time. Once the "transfer" function has been generated, S-parameter data is rapidly calculated.

In Pictures



In Detail

HFSS has three sweep types available: discrete, fast, and interpolating.

The fast sweep generates a full-field solution within the specified frequency range. The fast sweep is best suited for simulations that have a number of sharp resonances. A fast sweep is highly accurate in determining the behavior of a structure near a resonance.

The fast sweep works by using the center frequency of the sweep to create an Eigen value problem that will be used in an Adaptive Lanczos-Padé Sweep (ALPS)- procedure to determine all the field solutions in the requested frequency range.

Because the fast sweep uses the results of the adaptive process to generate the Eigen value problem, it is efficient to set the solution frequency to be equal to the center sweep frequency when using the fast sweep.

A key benefit of the fast sweep is that it allows a user to post-process and display fields at any frequency and at any location within the frequency sweep.

The interpolating sweep estimates a solution for the S-matrix over an entire frequency range. HFSS does this by choosing appropriate frequency points at which to solve for the field solution. HFSS continues to choose frequency points until the full sweep solution lies within a given error tolerance.

The interpolating sweep is best suited for very broadband frequency sweeps. The interpolating sweep uses less RAM than a fast sweep. A key benefit of the interpolating sweep is that it can easily determine the frequency sweep response from DC to any desired high frequency. The interpolating sweep, however, only has the solution frequency field data available for post-processing. Field data for other frequencies within the interpolating sweep range are therefore not available.

The discrete sweep generates explicit field solutions at specific frequency points in the desired frequency sweep. The discrete sweep solution time is directly dependent on the number of frequency points desired. The more frequency steps a user requests, the longer HFSS will need to complete the frequency sweep. The explicit field solution is obtained by substituting the desired frequencies into the matrix equation that was created during the adaptive solution process. Each frequency solution is therefore explicitly based on the adaptive solution, and not interpolated via a numerical method like the fast and interpolating sweeps. Arguably, therefore, the discrete sweep is the most accurate sweep available. It, however, is also the sweep that requires the most time to generate frequency sweep results when many frequency steps are desired.

The differences between local, remote and DSO solutions

In Brief

A Local Simulation is performed on the local computer (i.e., the user's computer).

A Remote Simulation is one where a user does not solve a given HFSS simulation locally.

Rather, the user sends the simulation to be performed on another computer somewhere on the local network.

A DSO, or Distributed Solution Option, is a simulation where a user sends a given HFSS simulation to be solved in parallel on a number of different computers. DSO currently works only for single simulations that have discrete or interpolating sweeps or simulations that use one of the Optimetrics™ features such as parametric sweeps.

In Pictures



In Detail

Historically, all HFSS simulations have been local simulations. However, in many enterprise environments, there are select computers that are optimized for maximum RAM, speed, etc. The remote solve capability allows a user to pre- and post-process a given HFSS simulation on the local machine but also have the computationally intensive solution performed on a different, possibly more powerful, computer. The remote solve capability will allow an engineer to still be productive on his local machine doing other tasks, while the intensive number crunching is done on another machine.

The Distributed Solve Option is extremely beneficial for simulations that involve a large number of discrete frequency steps, an interpolating sweep, or for performing a parametric, optimization, statistical, or sensitivity analysis. For the cases of the discrete or interpolating sweep, required frequency steps are solved in parallel on a number of different computers. If, for instance, a total of 100 discrete frequency steps are desired in a given simulation, DSO can solve all 100 frequency points in parallel provided 100 separate processors are available. This reduces the total simulation time by almost a factor of 100.

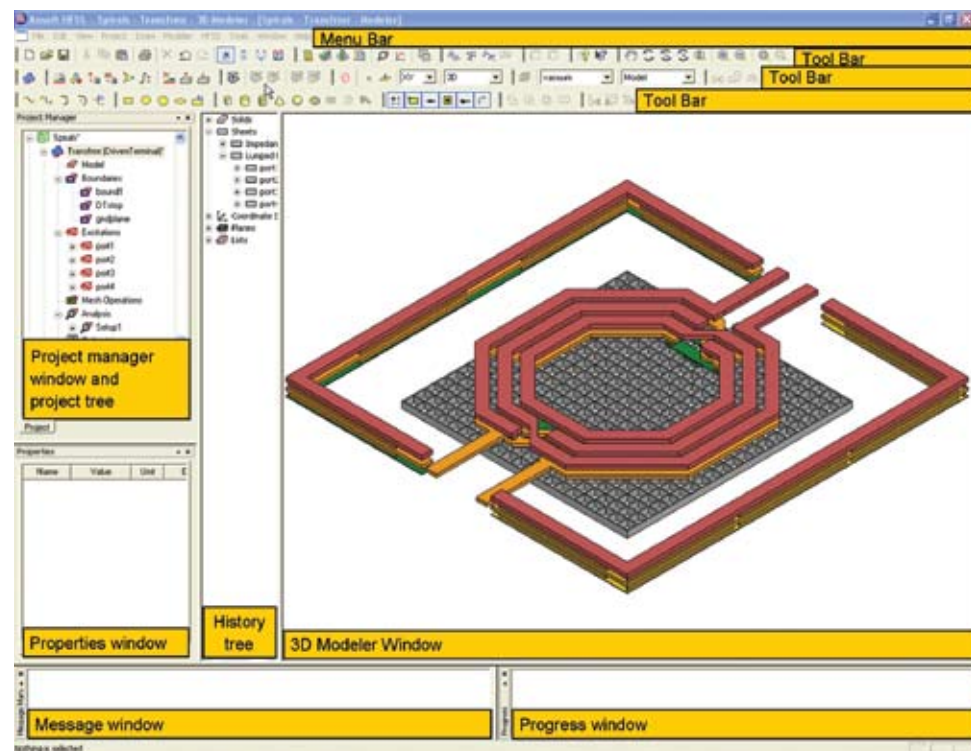
For the simulations involving the Optimetrics™ capabilities, the same general concept applies. For instance, if a parametric sweep with 20 variations is desired, the DSO option can take this parametric simulation and solve it in parallel on 20 different computers. Employing the DSO will greatly reduce the total computation time required for a parametric simulation.

The modeling GUI windows

In Brief

There are six distinct windows that can be active within the HFSS GUI: the project, history tree, drawing, properties, message, and progress.

In Pictures



In Detail

The HFSS GUI is broken down into a number of distinct windows, each of which serve a unique purpose during the creation and solution of a given HFSS model. The menu bar at the top of the screen is the toolbar window. From this toolbar, all operations that are possible within HFSS are accessible. Often, however, it is also possible to perform a desired operation by simply performing a right mouse button click in an appropriate window. For example, boundaries can be assigned to a surface by selecting a surface, clicking on HFSS in the menu bar, selecting boundaries and specifying the boundary in the menu. Alternatively, the same boundary can be applied to the same object by clicking on the right mouse button in the 3D modeller window, selecting boundaries and choosing the boundary to apply. The largest window is the 3D modeller window. Within this window, a user can create the objects that will be simulated. Objects are created by initially selecting a basic solid, sheet, or line type from the menu bar or drawing icons. The objects are then created by clicking and dragging the mouse. By clicking on the right mouse button while in this window, users can get access to specific menu items related to model generation or setup.

The project manager window serves as the central command window for a given simulation. Within this window, all pertinent simulation setup information is displayed. All pertinent pre-processing information is shown, such as the boundaries, ports, solution setup, frequency sweep, and mesh operations. Also shown in the project tree are the various optometric setups. All post-processing items, such as results, field plots, and antenna setup are also contained in the project tree. The geometry tree window contains the history of every geometric part drawn in HFSS. For each drawn part, the steps taken to create that particular object, or object history, are recorded. This model object history can be modified by clicking on an individual object-specific entry in the history tree.

The properties window displays the properties of a selected object in the modeling window, geometry tree, or project tree.

The message window displays all messages pertinent to a simulation. The progress window displays the progress of a given simulation being performed on the local machine, on a remote machine, or on a distributed network of machines using the Distributed Solve Option available with HFSS.

The various hotkeys

In Brief

Hotkeys are specific keys or a combination of keys that have a specific purpose. The most common hot keys are for pan, rotate, and zoom. Additionally, hotkeys can be used to produce planar XY, YZ, XZ, and the standard isometric views of objects in the modeling window.

SHIFT + Left Mouse Button: Drag

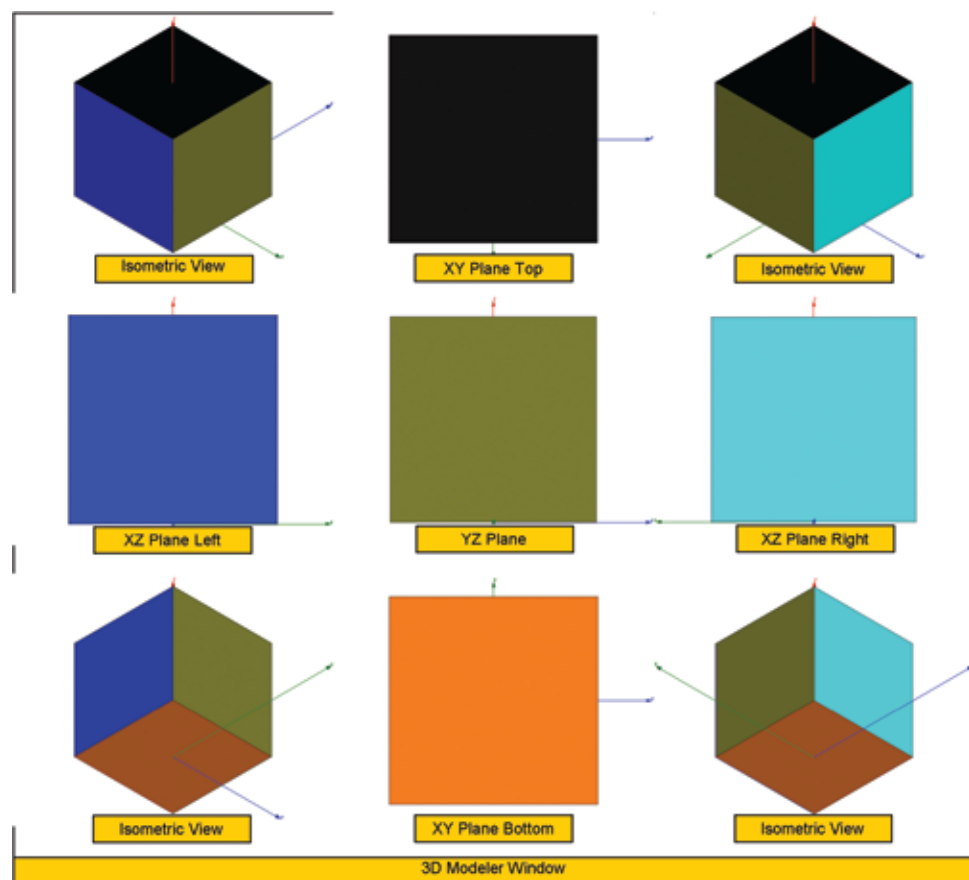
Alt + Left Mouse Button: Rotate model

Alt + SHIFT + Left Mouse Button: Zoom in / out

In Pictures

Holding the <ALT> key and double-clicking the left mouse button will orient objects in drawing window according to the figure below.

Colored  box shows double-click location



In Detail

There are a number of additional hotkeys. They are broken down into two groups: general hotkeys and 3D modeller hotkeys.

General Hotkeys

- F1: Help
- F1 + Shift: Context help
- F4 + CTRL: Close Program
- CTRL + C: Copy
- CTRL + N: New Project
- CTRL + O: Open
- CTRL + S: Save
- CTRL + P: Print
- CTRL + V: Paste
- CTRL + X: Cut
- CTRL + Y: Redo
- CTRL + Z: Undo
- CTRL + 0: Cascade windows
- CTRL + 1: Tile windows horizontally
- CTRL + 2: Tile windows vertically

3D Modeller Hotkeys

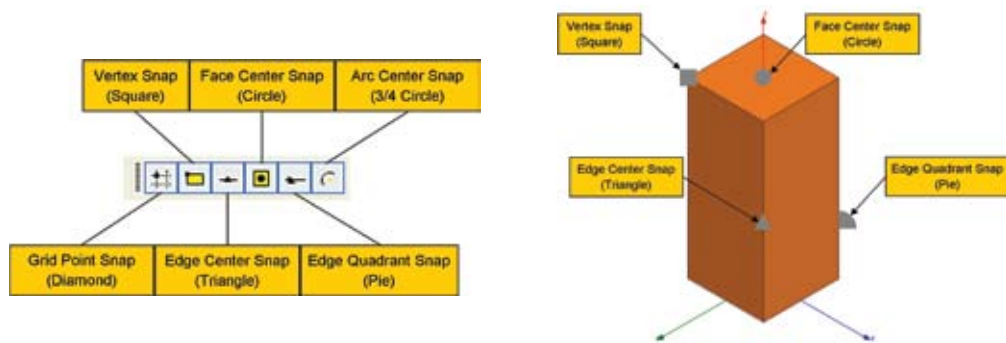
- B: Select face/object behind current selection
- F: Face select mode
- O: Object select mode
- CTRL + A: Select all visible objects
- CTRL + SHIFT + A: Deselect all objects
- CTRL + D: Fit view
- CTRL + E: Zoom in, screen center
- CTRL + F: Zoom out, screen center
- CTRL + Enter: Shifts the local coordinate system temporarily
- SHIFT + Left Mouse Button: Drag
- Alt + Left Mouse Button: Rotate model
- Alt + SHIFT + Left Mouse Button: Zoom in/out
- F3: Switch to point entry mode (i.e., draw objects by mouse)
- F4: Switch to dialogue entry mode (i.e., draw object solely by entry in command and attributes box)
- F6: Render model wire frame
- F7: Render model smooth shaded

Snapping on to a point

In Brief

The HFSS modeling UI employs a visual feedback system that allows the user to “snap” to a particular location on an object. The cursor changes shape when it is moved over a specific location, thus indicating that any drawing object created will be snapped to that specific location.

In Pictures



Simple 3D object showing various “snap” locations and the location-specific icon.

In Detail

By default, the selection point and graphical objects are set to “snap to,” or adhere to, a point on the grid when the cursor hovers over it. The coordinates of this point are used, rather than the exact location of the mouse. The cursor changes to the shape of the snap mode when it is being snapped.

The cursor in the HFSS modeling UI is usually a small diamond. However, the diamond shape will change into a circle, triangle, pie slice, or rectangle, depending on whether the cursor is moved into close proximity with a face center, mid edge, quarter edge, or corner vertex, respectively. Once the cursor has changed shape, a drawing object will be “snapped” to the location that corresponds to the cursor shape. For example, if a user wants to draw a cylinder that is centered on a face of a cube, simply move the cursor over the center of the face until the cursor changes to a circle. Once it has changed shape to a circle, click to set the start location of the cylinder and the cylinder center will be snapped to the center of the cube face.

You can follow the above procedure and snap to any convenient point when creating any model objects.

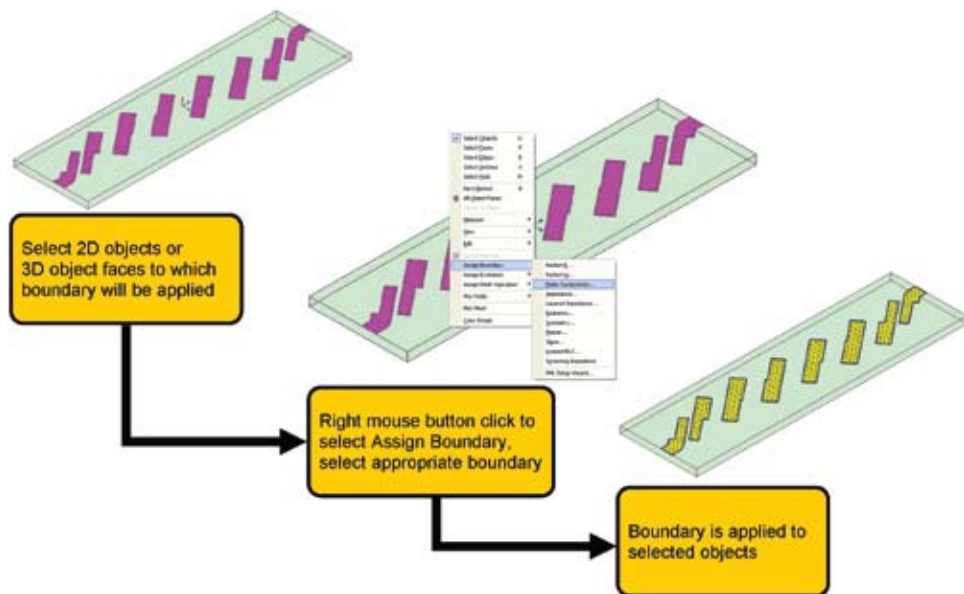
Snap locations can be activated and de-activated at a user’s discretion. To do this, a user can simply select which snap to have active by selecting the appropriate icon. Alternatively, a user can vary the snap selection by selecting Modeller in the tool bar and choosing snap mode.

Assigning boundaries in the GUI

In Brief

Boundaries are assigned to specifically created 2D object in an HFSS model or to specific faces of 3D objects.

In Pictures



In Detail

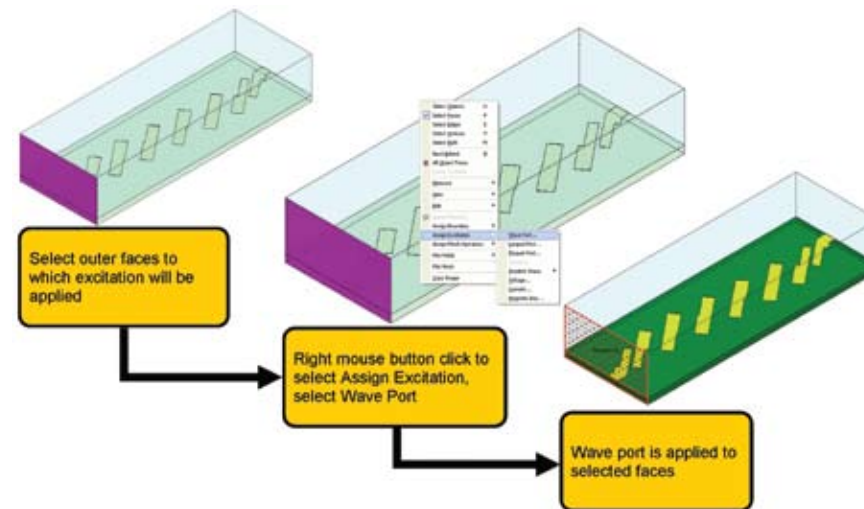
To assign a boundary to a 2D object or 3D face, simply change to the select faces mode and select the appropriate 2D object or 3D face. If a common boundary is to be applied to multiple faces, the multiple faces can be selected by holding the CTRL key. Once all the desired faces have been selected, simply perform a right mouse button click and select Assign Boundary. Finally, select the desired boundary. Alternatively, once all the faces have been selected, a user can click on HFSS in the top-level menu bar, select boundaries, choose assign, and select the desired boundary.

Assigning Driven Modal Solution excitations in the GUI

In Brief

Excitations are assigned to specifically created 2D object in an HFSS model or to specific faces of 3D objects. The solution type selected dictates the steps a user needs to follow in order to create a port excitation. Shown Below are the steps for a driven modal solution.

In Pictures



In Detail

To assign an excitation to a 2D object or 3D face, simply change to the select faces mode and select the appropriate 2D object or 3D face. Multiple faces can be selected if a common excitation is to be applied to them. Once all the desired faces have been selected, simply perform a right mouse button click, select assign excitations, and choose the desired excitation. Alternatively, once all the faces have been selected, a user can click on HFSS in the top-level menu bar, select excitations, choose assign, and select the desired excitation.

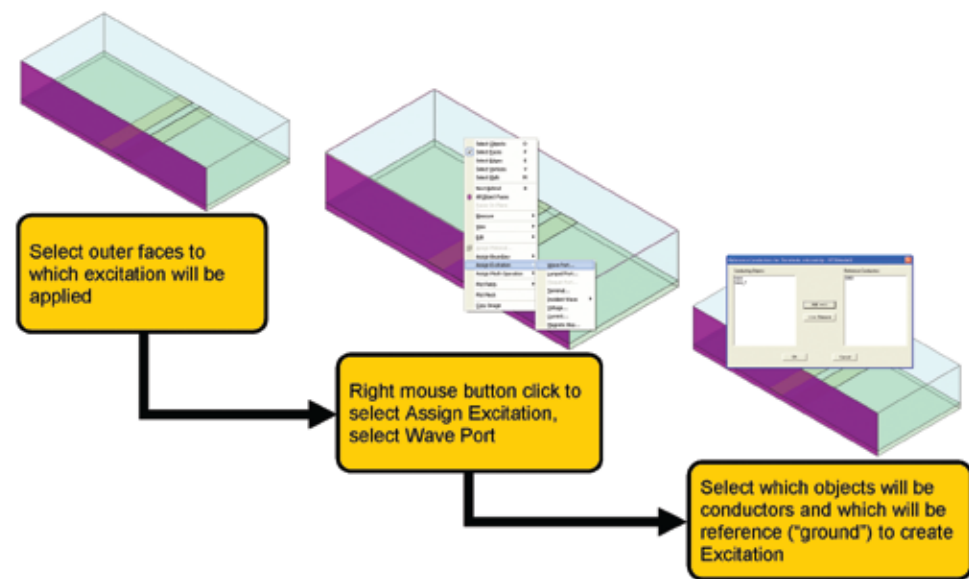
A user should ensure that the port area is of the proper dimension. For reference, see the section on ports. While it is not necessary to create an integration line when creating a wave port, it is good modelling practice and is, therefore, strongly encouraged.

Assigning Driven Terminal Solution excitations in the GUI

In Brief

Excitations are assigned to specifically created 2D object in an HFSS model or to specific faces of 3D objects.

In Pictures



In Detail

To assign an excitation to a 2D object or 3D face, simply change to the select faces mode and select the appropriate 2D object or 3D face. Multiple faces can be selected if a common excitation is to be applied to them. Once all the desired faces have been selected, simply perform a right mouse button click, select assign excitations, and choose the desired excitation. Alternatively, once all the faces have been selected, a user can click on HFSS in the top-level menu bar, select excitations, choose assign, and select the desired excitation.

Users should ensure that ports are properly sized. Also, when creating ports for a driven terminal solution, it is not necessary to create an integration line. This is done automatically by HFSS during the port generation process.

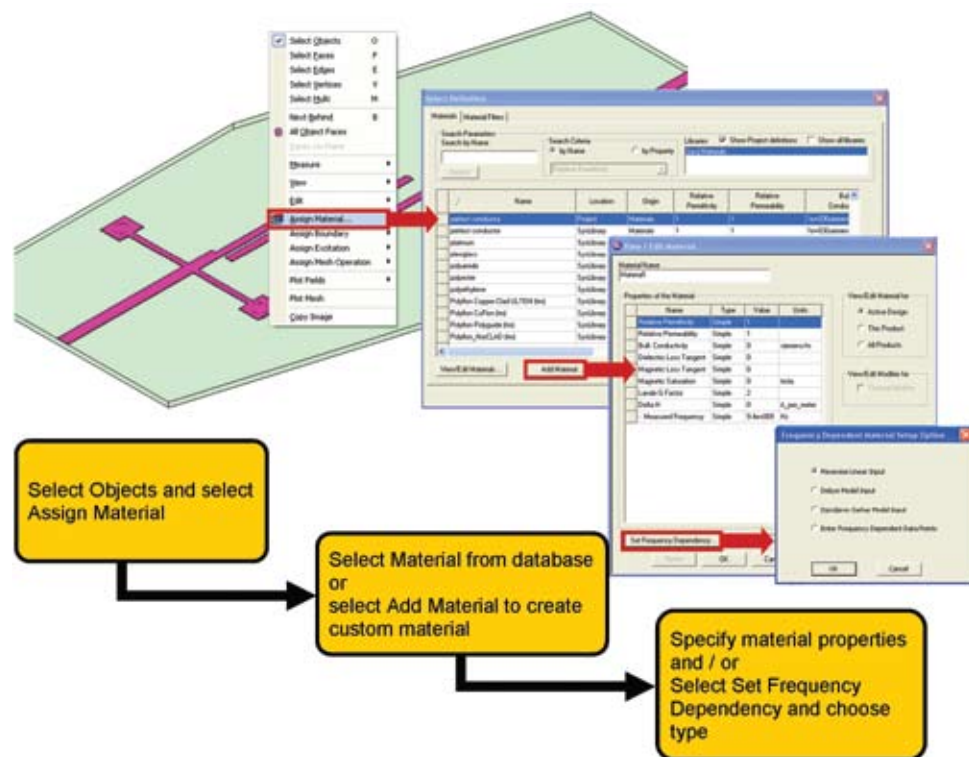
Assigning and creating materials

In Brief

All 3D objects in HFSS must have a material property assigned to them. Objects are assigned a default material during the 3D object creation process. The material assigned to a given object can be changed at any time after the object is created.

If a particular material is not found in the default HFSS material database, a custom material can be created. Frequency-dependent material can also be created if needed.

In Pictures



In Detail

To change the material property of an object to something other than the default material, simply select the object and select Edit and Properties. Select the attributes tab and choose the material desired from the materials database. Alternatively, you can select the object and edit the material in the properties window that appears after the object has been selected. Lastly, you can directly double-click on the object name in the geometry tree and change the material in the dialog that appears.

To add a material to the database, simply access the materials database and click on the Add Material button at the bottom of the dialog. Simply enter a name along with the desired material properties and close the dialog by clicking OK. The created material will automatically be assigned to the 3D object.

Frequency-dependent materials can be based on four distinct definitions: Piecewise-linear, Debye, Djordjevic-Sarkar, or as a collection of data points. Each method creates a material that will have specific material properties as a function of frequency.

The Piecewise Linear and Frequency-Dependent Data Points models apply to both the electric and magnetic properties of the material. However, they do not guarantee that the material satisfies causality conditions, and so they should be used only for frequency-domain applications.

The Debye and Djordjevic-Sarkar models apply only to the electrical properties of dielectric materials. These models satisfy the Kramers-Kronig conditions for causality, and so are preferred for applications where HFSS results, both S-parameter and equivalent circuit, will be used in a time-domain simulation.

The Piecewise linear model defines a material that has three distinct regions. The first region has a flat constant dielectric property, the second has a linear slope, and the last section is flat again. A user specifies the break frequencies between sections one and two and sections two and three.

The Debye model is a single pole model for the frequency dependency of a lossy dielectric. A user specifies the two frequencies along with the dielectric constant and loss tangent at those frequencies. If desired, a user can also specify the permittivity at an optical frequency and DC conductivity and constant relative permeability.

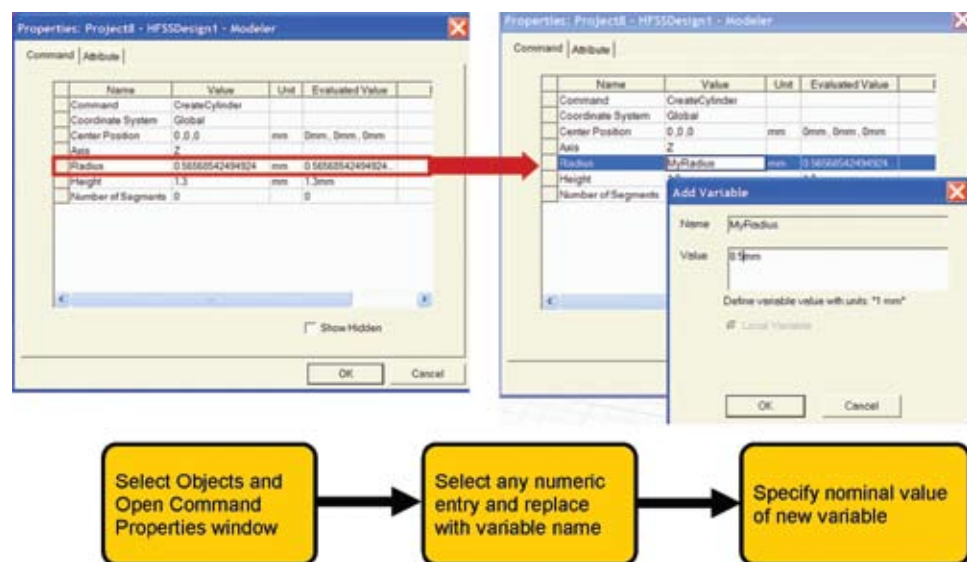
For the Djordjevic-Sarkar model, a user specifies the permittivity and loss tangent at a single frequency. Additionally, a user may enter the conductivity and permittivity at DC. This model was specifically developed for materials that are commonly used in printed circuit board and package designs.

Creating variables

In Brief

A variable can be created for any dimensional or material property, or output value. Variables can be Design variables or Project variables. Once a variable has been created, parametric sweeps, optimization, sensitivity and statistical analysis are possible.

In Pictures



In Detail

There are two types of variables in HFSS: design variables and project variables. Project variables can be assigned to any parameter value within the HFSS project where it was created. By contrast, design variables can only be assigned to a parameter value within the design where it was created. A project may contain many designs, so depending on how often a variable is to be used and where may determine what type of variable to use.

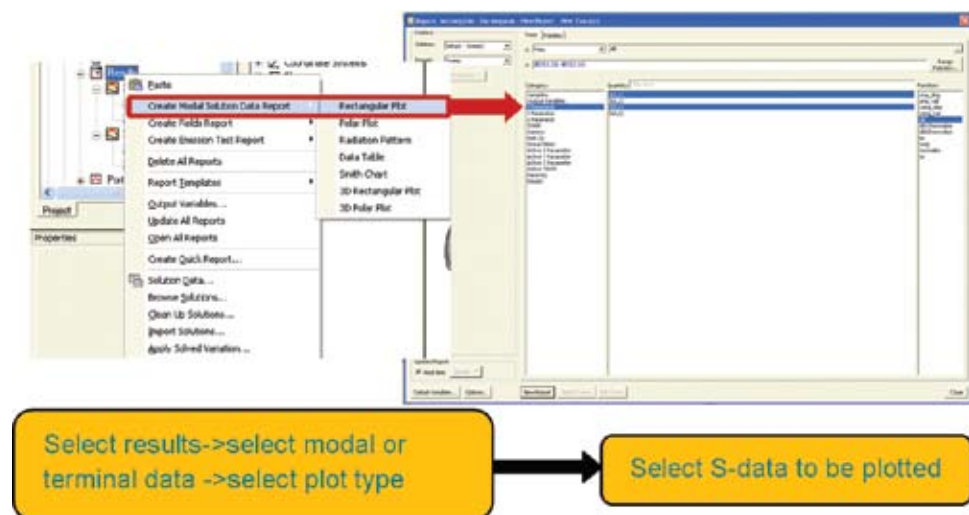
Regardless of what type of variable is created, they always represent a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in HFSS. Variables are very useful in situations where a parameter value is changed often, if it is desired to perform a parametric analysis, perform an optimization, or intend to create an output variable to which HFSS is desired to converge to.

Plotting S-parameter results

In Brief

One of the most useful outputs from HFSS is the S-parameter. Once a simulation has finished, S-parameters at a single frequency or over a frequency sweep can be plotted.

In Pictures



In Detail

S-parameters are easily plotted using the HFSS results editor. However, depending on the port type, S-parameters are either generalized or normalized.

The S-parameters for a wave port are, by default, generalized. Generalized S-parameters do not have a normalization constant but rather are normalized to the characteristic impedance of the corresponding wave port. As a result, when comparing HFSS wave port results to measured data, it is important to re-normalize the HFSS results to the normalization constant used when the measurement was performed.

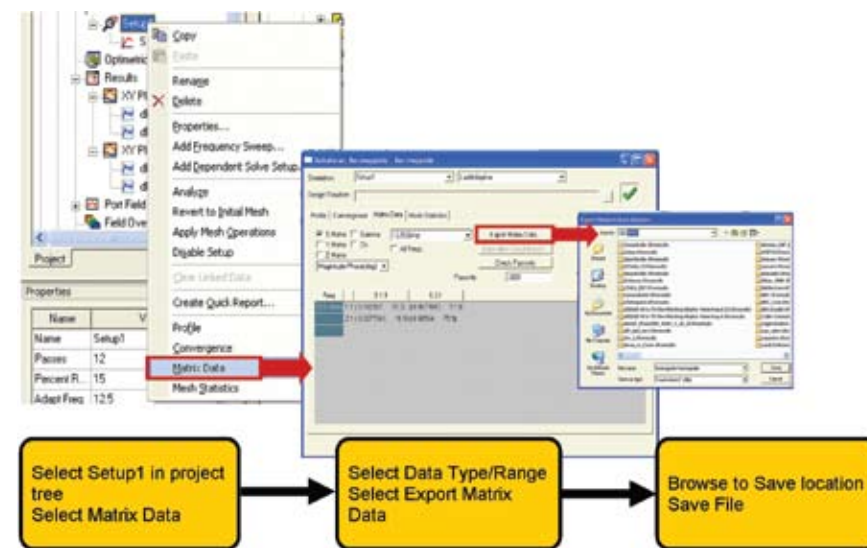
If a given S-parameter is based on a lumped port, the S-parameters are normalized to the value of Z_0 specified when the port was created. When comparing measured and simulated data, it is again necessary to re-normalize the HFSS data if the user-specified Z_0 is not equivalent to the normalization constant used in the measurement.

Exporting touchstone files

In Brief

HFSS allows exportation of any of the quantities that HFSS calculates. Touchstone files are easily exported by following the steps below. A common use of exported touchstone files is their use in circuit simulators.

In Pictures



In Detail

Depending on the data desired, results are exported directly from the results editor or by using the field calculator. The export of touchstone files can be accomplished in a number of ways.

In addition to the procedure shown above, a user can export touchstone files by clicking on HFSS in the menu bar, selecting Results, and selecting Solution Data. In the dialog that is displayed, select Export Matrix Data. Browse to the proper file location and save the file.

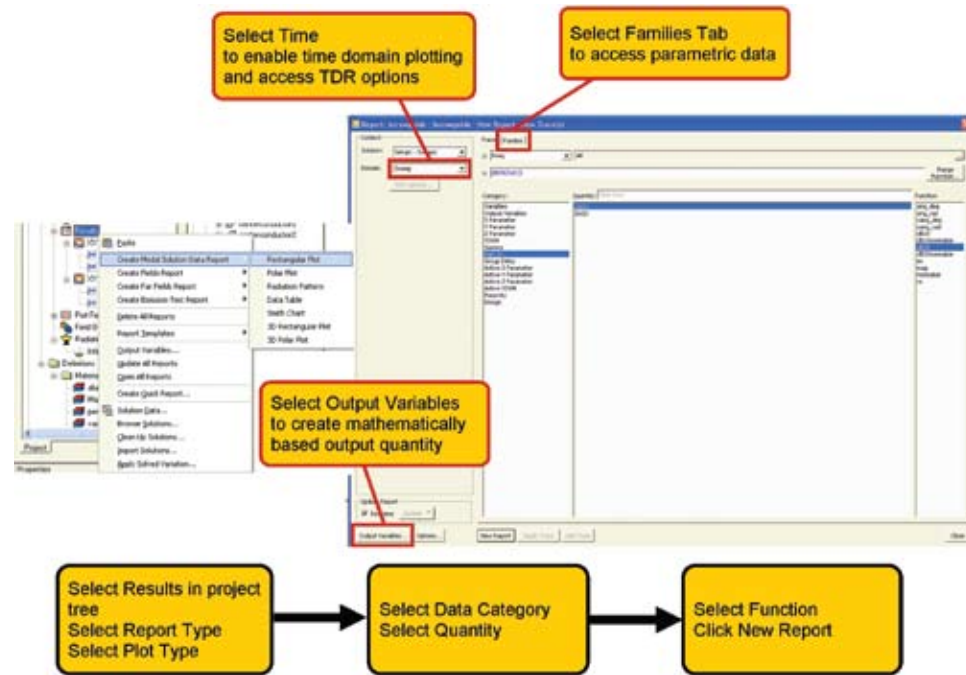
Alternatively, a user can click on results in the project tree and select solution data. In the dialog that is displayed, select export matrix data. Browse to the proper file location and save the file.

Advanced plotting of results

In Brief

In addition to S-parameters, HFSS can plot a number of additional quantities of interest in RF/microwave/SI design using the Results Editor. A partial list of these plotable quantities include gamma information, Y and Z parameters, TDR results, VSWR data, and Group Delay.

In Pictures



In Detail

Virtually any quantity of interest to an electrical engineer can be plotted in HFSS. The standard output quantities can be enhanced by creating output variables based on mathematical expressions. These output variables can be created by clicking on the Output Variables button in the lower right-hand corner of the New Report Dialog.

Additionally, users can plot time-domain responses for a given simulation model by selecting TIME as the sweep variable in the drop-down dialog in the upper left of the New Report Dialog. Once the TIME variable has been selected, the TDR options button becomes active and a user can choose to plot a time-based system response such as a TDR plot.

Also, if a user has performed a parametric sweep, the data families will be available to be plotted if a user selects the Families tab in the top center of the New Report Dialog.

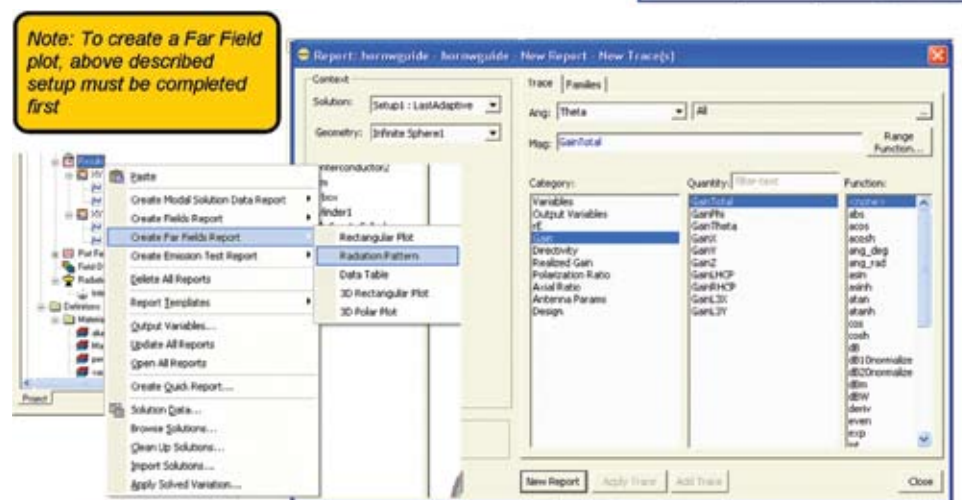
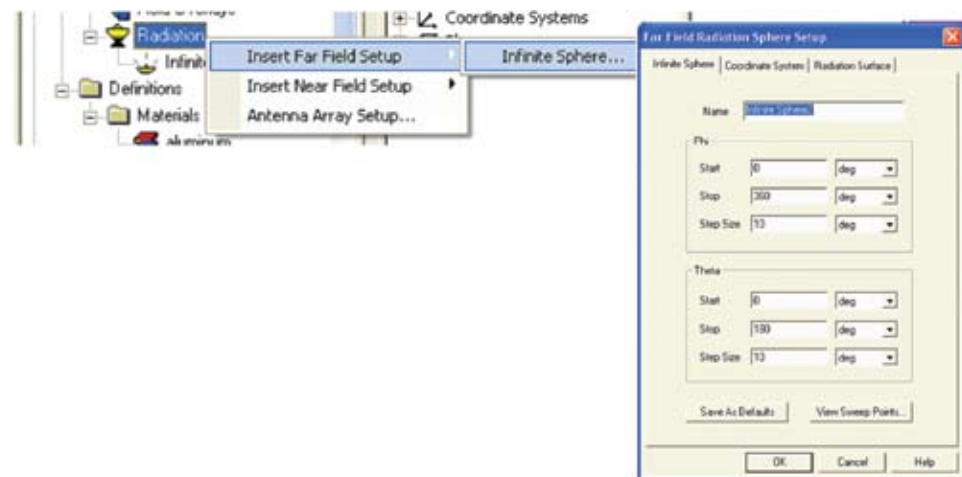
Lastly, all reports can also be generated by clicking on HFSS in the menu bar, selecting results, and selecting create modal (or terminal) reports. In the new report dialog, a user can specify what data is to be plotted (as shown above).

Plotting antenna results

In Brief

Far field antenna patterns are easily generated by HFSS by again using the Reports Editor. The procedure is similar to plotting the standard circuit parameters. But the model should have included either Radiation or PML boundaries, and a Far Field Setup must be defined before Far Field quantities can be plotted.

In Pictures



In Detail

While the plotting of far fields is straightforward, there are some key items a user should know regarding how HFSS generates far field data.

When HFSS generates far field data, the field values on the radiation surface(s) are used to compute the fields in the space surrounding the modeled structure, outside of the solution volume. This space is broken down into the near field and far field regions, where the near field is the region close to the solution volume. In general, the electric field in this external region can be written as

$$E = \left(\int_S (j\omega\mu_0 H_{tan}) G + (E_{tan} \times \nabla G) + (E_{normal} \nabla G) \right) dS. \quad (16)$$

Where

S represents the radiation boundary surfaces.

J is the imaginary unit.

ω is the angular frequency, $2\pi f$.

μ_0 is the relative permeability of free space.

H_{tan} is the component of the magnetic field that is tangential to the surface.

E_{normal} is the component of the electric field that is normal to the surface.

E_{tan} is the component of the electric field that is tangential to the surface.

G is the free space Green's function, given by

$$G = \frac{e^{-jk_0|r-r'|/\sqrt{\mu_r\epsilon_r}}}{|r-r'|} \quad (17)$$

where k_0 the free space wave number, $\omega\sqrt{\mu_0\epsilon_0} = \frac{\omega}{c}$

r, r' represent the field and source points, respectively,

ϵ_0 is the permittivity of free space, μ_r and ϵ_r are the relative permeability and permittivity of a dielectric, respectively.

The r dependence seen above is a key far fields characteristic of a spherical wave. The far field is a spherical TEM wave, which can be described by the following equation:

$$E = \eta H \times \hat{r} \quad (18)$$

where η is the intrinsic impedance of free space.

When calculating the far fields, the previously discussed far-field approximations are used, and the result is valid only for field points in the far-field region.

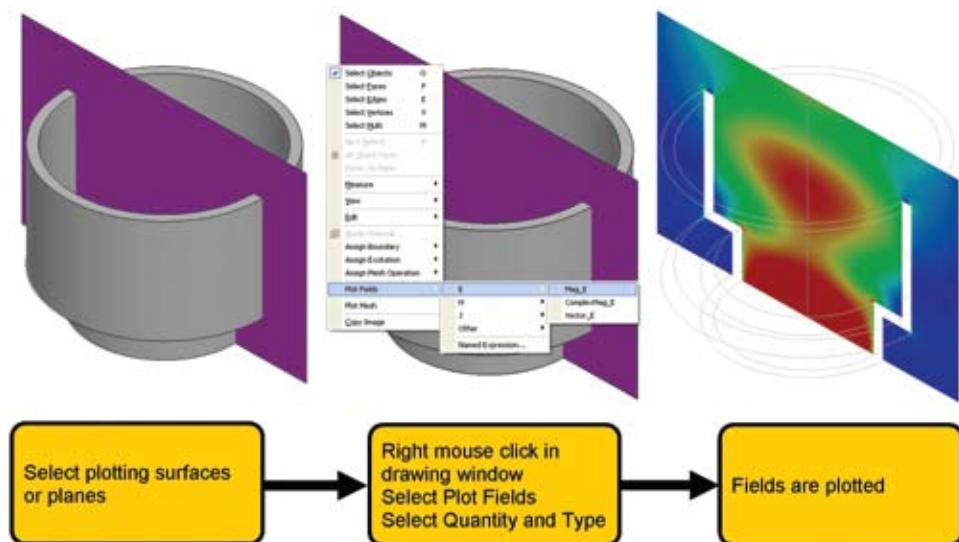
Plotting field results

In Brief

HFSS can produce a plot of any standard electromagnetic quantity, such as the electric field, magnetic field, Poynting vector, or current density. Generally, fields are displayed on specifically created 2D objects, faces of 3D objects, or on coordinate system planes. Plots can be scalar quantity plots or vector quantity plots.

Specific quantities based on mathematical operations on the basic field quantities can also be plotted by use of the fields calculator.

In Pictures



In Detail

Field plots, or, more specifically, field overlays, are representations of the basic or derived field quantities on specific surfaces of objects or within an object for the current design variation.

A field overlay's appearance can be changed by modifying the settings in the Plot attributes dialog. This dialog modifies a plot folder and all field overlays contained within that folder will use the same attributes.

Field overlays can also be created by the use of the field calculator. The field calculator allows a user to create mathematical operations on the basic field quantities. These results can be plotted or exported depending on the needs of the user. Once a mathematical operation has been created in the calculator, it can be added to the Named Expressions stack. After adding, this quantity can be plotted by selecting Plot Fields -> Named Expressions dialog.

Field overlay plots can be saved as HFSS-specific .dsp files. To save a file, simply click on HFSS in the menu bar, select Fields, and select Save As. A list of plots will be displayed and a user can select which plots to save. Once saved, they can be opened in HFSS at a later time.

Alternatively, a user can simply perform a right mouse button click in the modeller window and select Copy Image. This operation places the contents of the window into the "clipboard," and the user can paste the field overlay into a document such as Word or PowerPoint.

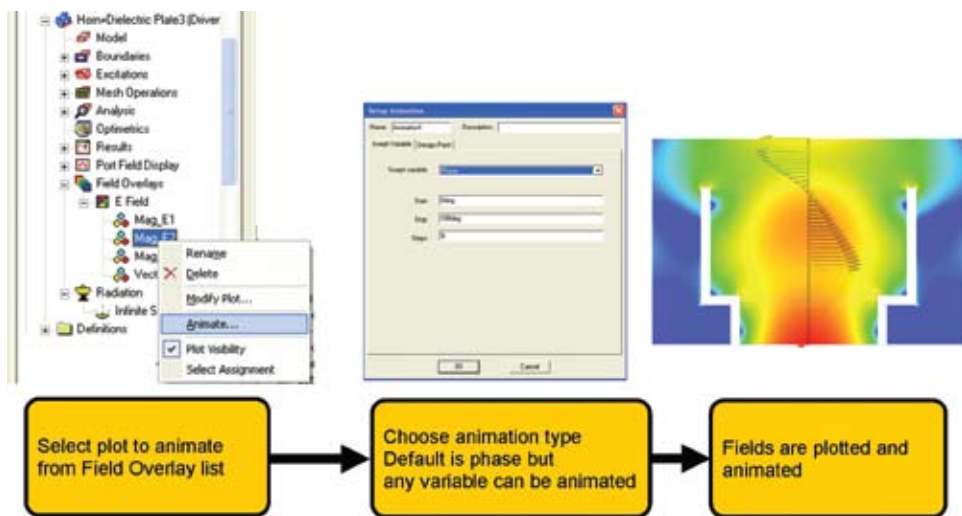
Creating animations

In Brief

HFSS allows the animation of any field overlay plot. Field overlays can be animated with respect to excitation phase or any other variable that is part of the HFSS design, including user-created variables.

Only existing field overlays can be animated.

In Pictures



In Detail

An animated plot is a series of frames that displays a field, mesh, or geometry at varying values. To create an animated plot, you specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation. The user specifies how many frames to include in the animation.

APPENDIX

Additional reading materials

Zoltan J. Cendes, P.P. Silvester, "Numerical solution of dielectric loaded waveguides: I-finite-element analysis," IEEE Transactions on Microwave Theory and Techniques, vol. 18, no. 12, pp.1124-31, 1970.

Shenton, D. N., Zoltan J. Cendes, "Three-dimensional finite element mesh generation using Delaunay Tesselation," IEEE Transactions on Magnetics, vol. 21, no. 6, pp. 2535-2538, 1985.

Jin-Fa Lee, Zoltan J. Cendes, "Transfinite elements: A highly efficient procedure for modeling open field problems," Journal of Applied Physics, vol. 61, pp. 3913-3915, 1987.

Zoltan J. Cendes, Jin-Fa Lee, "The Transfinite Element Method for modeling MMIC devices," IEEE Transactions on Microwave Theory and Techniques, vol. 36, pp. 1639- 1649, 1988.

Jin-Fa Lee, Din-Kow Sun, Zoltan J. Cendes, "Full-Wave Analysis of Dielectric Waveguides Using Tangential Vector Finite Elements," IEEE Trans. on Microwave Theory Tech., vol. 39, pp. 1262- 1271, 1991.

Lee J F , Sun D. K., and Cendes Z J , "Tangential vector finite elements for electromagnetic field computation," IEEE Transactions on Magnetics, vol. 27, no. 5, pp 4032-4035, 1991.

Zoltan J. Cendes, "Vector finite elements for electromagnetic field computation," IEEE Transactions on Magnetics, Vol. 27 No.5, pp.3953-66, 1991.

J. Eric Bracken, Din-Kow Sun, Zoltan J. Cendes, "S-Domain methods for simultaneous time and frequency characterization of electronic devices," IEEE Transactions on Microwave Theory and Techniques, vol. 46, pp. 1277- 1282, 1998.

Din-Kow Sun, Jin-Fa Lee, Zoltan J. Cendes, "ALPS - a new fast frequency-sweep procedure for microwave devices," IEEE Transactions on Microwave Theory and Techniques, vol. 49, pp. 398- 402, 2001.

Antonije R. Djordjevic', Radivoje M. Biljic', Vladana D. Likar-Smiljanic', Tapan K. Sarkar, "Wideband frequency-domain characterization of FR-4 and time-domain causality," IEEE Transactions on Electromagnetic Compatibility, vol. 43, pp. 662-667, 2001.

Din-Kow Sun, Jin-Fa Lee, and Zoltan J. Cendes, "Construction of nearly orthogonal Nedelec bases for rapid convergence with multilevel preconditioned solvers," SIAM Journal on Scientific Computing, vol. 23, no. 4, pp. 1053-1076, 2001.

I. Bardi, G. Peng and Zoltan J. Cendes, "Improvements in adaptive mesh refinement and multi-level methods in high frequency electromagnetics," ACES Symposium, 2002.

I. Bardi, Z. Badics, Zoltan J. Cendes, "Total and scattered field formulations in the transfinite element method", IEEE Transactions on Magnetics , vol. 44, pp. 778-781, 2008.

Bruno Bianco, Luigi Panini, Mauro Parodi, and Sandro Ridella, "Some Considerations about the Frequency Dependence of the Characteristic Impedance of Uniform Microstrips," IEEE Transactions on Microwave Theory and Techniques, vol. MTT-26 No. 3, March 1978.

Edward F. Kuester, David C. Chang, and Leonard Lewin, "Frequency-Dependent Definitions of Microstrip Characteristic Impedance," International URSI Symposium on Electromagnetic Waves, " Munich, 26-29 August 1980, pp. 335 B/1-3.

