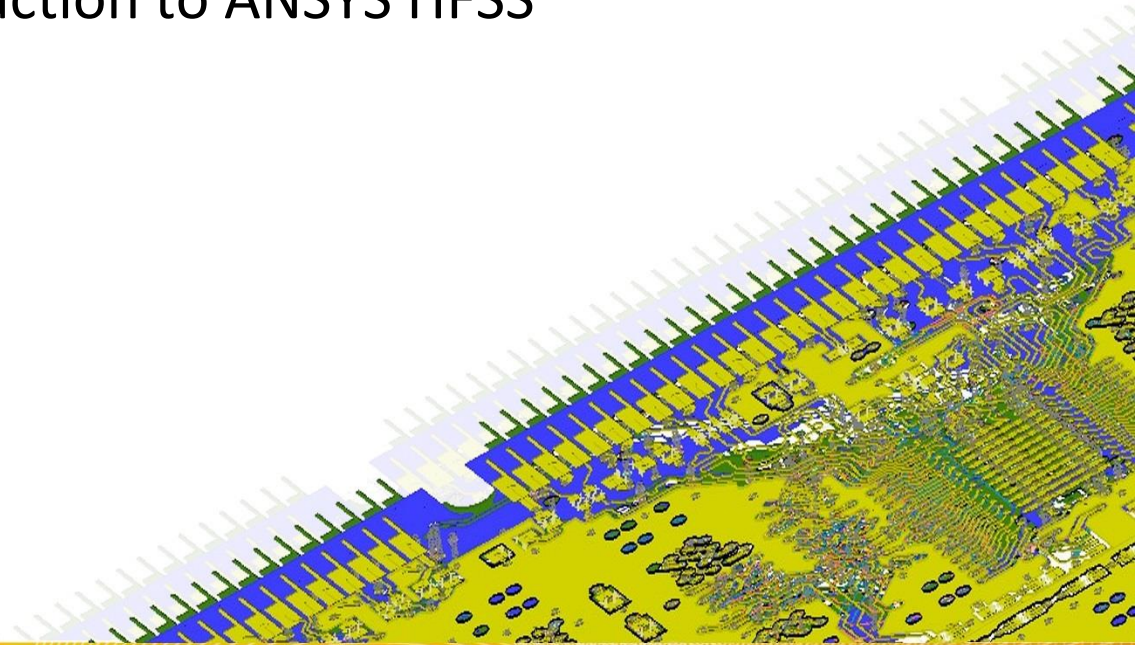


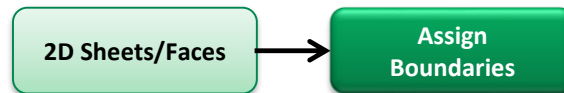


# Lecture 3-1: HFSS 3D Design Setup

Introduction to ANSYS HFSS



# Boundaries for 2D Sheets or Faces



# Boundary Conditions

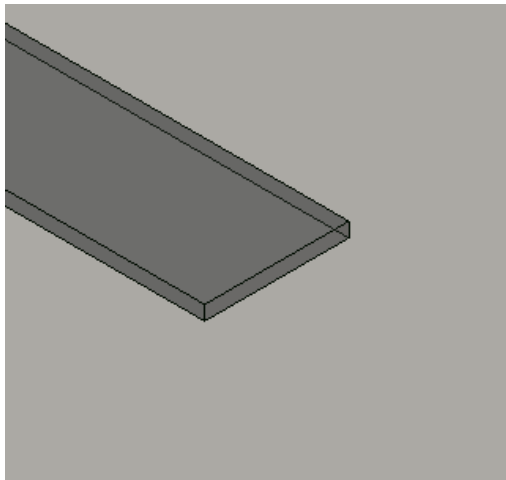
- **Boundary Conditions**

- Boundary conditions are used to simplify a model's geometry or make meshing more efficient
- For a complete discussion on all of the boundary conditions available within HFSS see Appendix 6.2 3D Boundary Conditions
- Boundary conditions can be grouped into three categories
  - **Material Properties for sheets/faces**
    - **Boundary Condition**
      - Finite Conductivity Surfaces
      - Perfect E (Electric) or Perfect H (Magnetic) Surfaces
    - **Surface Approximations for Manufacturing/Component Definitions:** Examples of this would be passive components such as resistors, inductors and capacitor components or Plating. These are important to the performance of the design, but in general explicitly modeling the details of the physically device or manufacturing process are beyond the scope of the simulation.
      - **Boundary Condition**
        - Lumped RLC Surface: a parallel combination of a lumped resistor, inductor, and/or capacitor
        - Layered Impedance Surface
    - **Radiation:** By default HFSS assumes everything outside of the user geometry (Background) is defined as a Perfect Electric Surfaces. In many cases, the default background boundary condition needs to be replaced by a radiation surface.
      - **Boundary Condition**
        - Radiation Surface
          - FE-BI: Finite Element Boundary Integral
        - Perfectly Matched Layers (PML)
        - Infinite Ground Plane

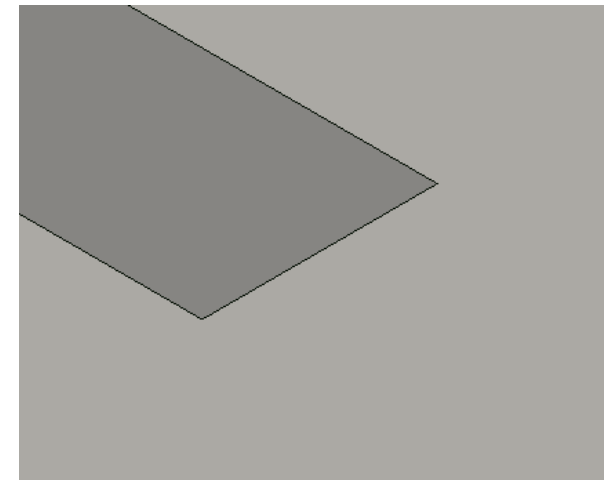
# How to model conductors (3D vs 2D)?

- **Conductor Thickness**

- In many applications, the thickness of a conductor on a PCB is not critical to the performance of a device. The efficiency of the solution problem can be increased by removing the thickness of conducting layer. The following slides compare examples of using a 3D and 2D conductor for a patch antenna and an edge coupled filter.
- In summary, the patch antenna shows no difference in performance for a 3D vs 2D conductor because the edge effect of the conductor does not impact the device performance. In the case of the coupled line filter, the edge effects of the trace thickness make a noticeable difference and needs to be included.



**Conductor as a 3D Solid**



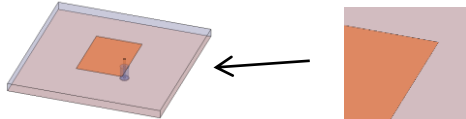
**Conductor as a 2D Sheet**

# Trace Thickness Effects

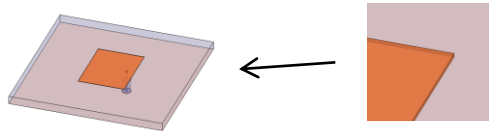
- **Conducting traces often modeled as 2D objects for electromagnetic simulations**

- More computationally efficient since fewer meshing surfaces required
- Good approximation for many structures operating in skin depth regime

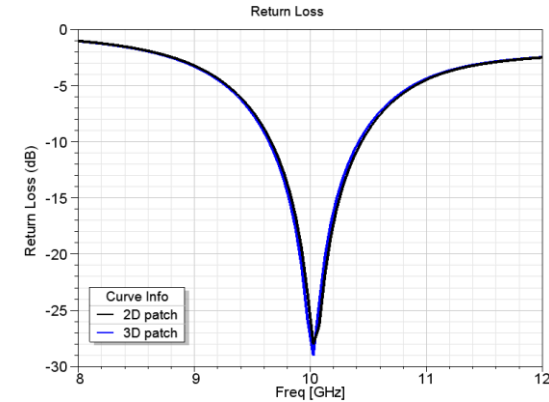
Patch antenna  
modeled with 2D sheet



Patch antenna  
modeled with 3D object



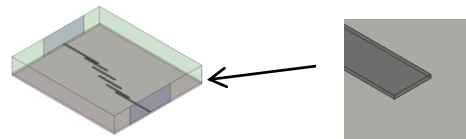
Frequency response of both models



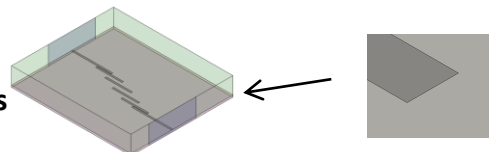
- **Planar filters which use edge coupling to operate require 3D objects (finite thickness) for modeling conducting traces**

- Applications whose performance depends upon closely-coupled traces
- End-coupled, parallel-coupled, hairpin filters, etc.

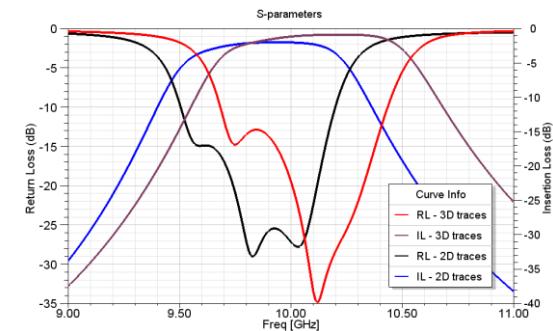
Edge-coupled filter  
modeled with 3D objects



Edge-coupled filter  
modeled with 2D sheets



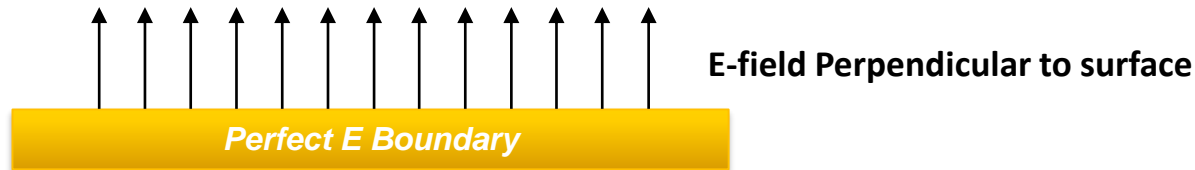
Frequency response of both models



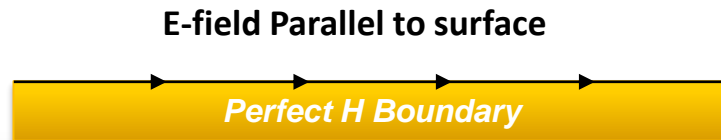
# Material Properties for 2D sheets/faces

- **Technical Definition of Boundary Conditions**

- **Perfect E** – Perfect E is a perfect electrical conductor (PEC), also referred to as a perfect conductor.
  - Forces E-field perpendicular to surface
  - Used to model lossless metal surfaces, ground planes, cavity walls, etc.
  - Can be assigned to 2D or 3D objects
  - There are also two automatic Perfect E assignments:
    - Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary condition name *outer*.
    - Any object that is assigned the material pec (Perfect Electric Conductor) is automatically assigned the boundary condition Perfect E to its surface and given the boundary condition name *smetal*.



- **Perfect H** – Perfect H is a perfect magnetic conductor.
  - Forces H-field perpendicular to surface and E-field tangential
  - Does not exist in real world
  - **Natural** – for a Perfect H boundary that overlaps with a perfect E boundary, this reverts the selected area to its original material, erasing the Perfect E boundary condition. It does not affect any material assignments. It can be used, for example, to model a cut-out in a ground plane for a coax feed.



# Material Properties for 2D sheets/faces

- **Surface Loss Modeling**

- **Finite Conductivity** –A Finite Conductivity boundary enables you to define the surface of an object as a lossy (imperfect) conductor. HFSS applies this boundary for lossy metal materials. To model a lossy surface, you provide loss in Siemens/meter and permeability parameters. Loss is calculated as a function of frequency. It is only valid for good conductors. Forces the tangential E-Field equal to  $Z_s(n \times H_{tan})$ . The surface impedance ( $Z_s$ ) is equal to,  $(1+j)/(\delta\sigma)$ , where:

- $\delta$  is the skin depth,  $(2/(\omega\sigma\mu))^{0.5}$  of the conductor being modeled,  $\omega$  is the frequency of the excitation wave,  $\sigma$  is the conductivity of the conductor,  $\mu$  is the permeability of the conductor

The screenshot shows the 'Finite Conductivity Boundary' dialog box. It has a title bar with a close button. The 'Name' field is 'FiniteCond1'. The 'Parameters' section includes 'Conductivity' (58000000 Siemens/m) and 'Relative Permeability' (1). There are checkboxes for 'Use Material' (set to 'vacuum') and 'Infinite Ground Plane'. The 'Advanced' section has 'Surface Roughness Model' with radio buttons for 'Grosse' (selected) and 'Huray'. It also has 'Surface Roughness' (0 um), 'Hall-Huray Surface Ratio' (empty), and 'Layer Thickness' (0 mm). A 'Use Defaults' button is at the bottom. Three arrows point to specific fields with text annotations: one to the Conductivity field labeled 'Manually define Conductivity (Default is Copper)', one to the Use Material dropdown labeled 'Use Material Definition to define Conductivity', and one to the Layer Thickness field labeled 'Adjust the Conductivity based on conductor thickness'.

Finite Conductivity Boundary

Name: FiniteCond1

Parameters

Conductivity: 58000000 Siemens/m

Relative Permeability: 1

☐ Use Material: vacuum

☐ Infinite Ground Plane

Advanced

Surface Roughness Model: ☒ Grosse ☐ Huray

Surface Roughness: 0 um

Hall-Huray Surface Ratio:

☐ Layer Thickness: 0 mm

Use Defaults

OK Cancel

Manually define Conductivity (Default is Copper)

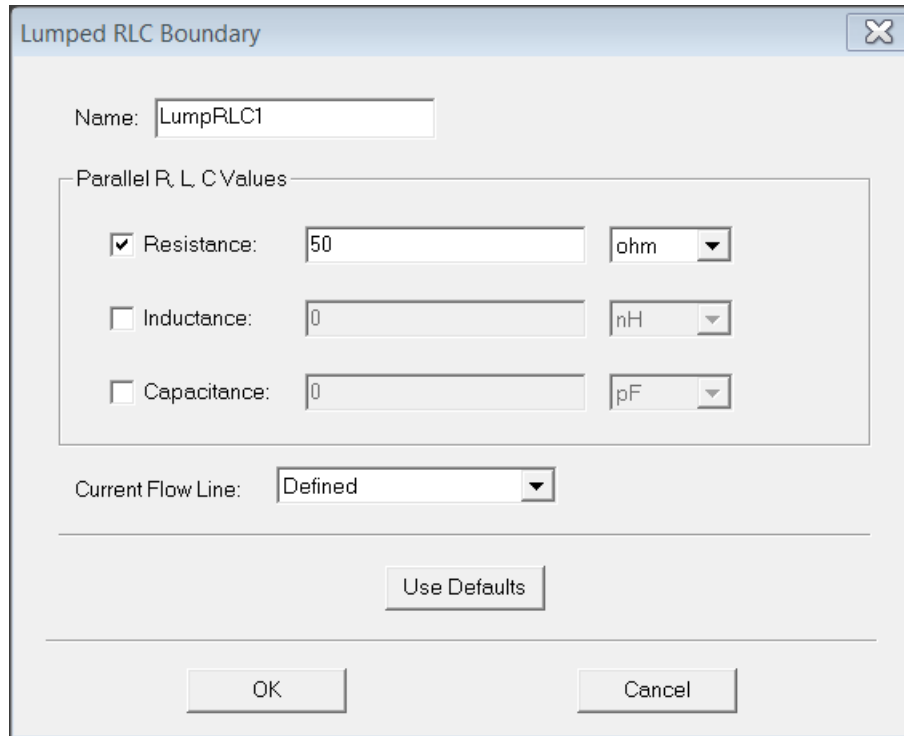
Use Material Definition to define Conductivity

Adjust the Conductivity based on conductor thickness

# Surface Approximations for Manufacturing/Component Definitions

- **Surface Loss Modeling**

- **Layered Impedance** – Multiple thin layers in a structure can be modeled as an impedance surface. See the Online Help for additional information on how to use the Layered Impedance boundary.
- **Lumped RLC** – a parallel combination of lumped resistor, inductor, and/or capacitor surface. The simulation is similar to the **Impedance** boundary, but the software calculate the ohms/square using the user supplied R, L, C values.



The screenshot shows the "Lumped RLC Boundary" dialog box. It has a title bar with a close button (X). The "Name" field is set to "LumpRLC1". Below it, the "Parallel R, L, C Values" section contains three rows: "Resistance" with a checked checkbox, a value of "50", and a unit dropdown set to "ohm"; "Inductance" with an unchecked checkbox, a value of "0", and a unit dropdown set to "nH"; and "Capacitance" with an unchecked checkbox, a value of "0", and a unit dropdown set to "pF". Below this section, the "Current Flow Line" dropdown is set to "Defined". At the bottom, there are "OK" and "Cancel" buttons, and a "Use Defaults" button is positioned above them.



# Radiation Surfaces

- **Radiating Boundary Conditions**

- **Radiation Surface**, also referred to as absorbing boundaries, enable you to model a surface as electrically open: waves can then radiate out of the structure and toward the radiation boundary. The system absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure and into space. Radiation boundaries may also be placed relatively close to a structure and can be arbitrarily shaped. This condition eliminates the need for a spherical boundary. For structures that include radiation boundaries, calculated S-parameters include the effects of radiation loss. When a radiation boundary is included in a structure, far-field calculations are performed as part of the simulation.
  - **Finite Element Boundary Integral (FEBI)** is an alternative to Radiation and PML boundaries for radiating designs. The FEBI boundary is a hybrid FEM (Volume) and IE solver (Radiating Surface). FEBI is a reflection-less boundary that can be applied to arbitrarily shaped volumes. Requires an HFSS-IE license.
- **Perfectly Matched Layers (PMLs)** are fictitious materials that fully absorb the electromagnetic fields acting upon them.
  - There are two types of PML applications: “free space termination” and “reflection-free termination” of guided waves.
    - In free space termination, all PML objects must be included in a surface that radiates into free space equally in every direction. PMLs can be superior to radiation boundaries in this case because PMLs enable radiation surfaces to be located closer to radiating objects, reducing the problem domain. Any homogenous isotropic material, including lossy materials such as ocean water, can surround the model.
    - In reflection-free termination of guided waves, the structure continues uniformly to infinity. The termination surface of the structure radiates in the direction in which the wave is guided. Reflection-free PMLs are superior to free space or radiation boundary terminations in this kind of application. Reflection-free PMLs are also superior for simulating phased array antennas because the antenna radiates in a certain direction.
- **Infinite Ground Plane** – While this is not a Radiating Boundary Condition, it has an impact on the far-field calculation. If radiation boundaries are used in a structure, the ground plane acts as a shield for far-field energy, preventing waves from propagating past the ground plane. To simulate the effect of an infinite ground plane, check the Infinite ground plane box when defining a Perfect E, Finite Conductivity, or Impedance boundary condition.
  - **NOTE:** Enabling the Infinite Ground Plane approximation ONLY affects post-processed far-field radiation patterns. It will not change the current flowing on the ground plane.

# Which Radiation Surface should I use?

- **Radiation Surface Selection**

- The following slides cover details about each boundary and are really intended for those doing Antenna, Array, or RCS simulations. Effects such as incident angle to the radiation surfaces are compared so a user can discern the differences in performance and select the appropriate boundary for the application.
- For everyone else, below is a simple selection table for choosing Radiation Surfaces. In general, a Radiation Boundary is the easiest to setup and is a good starting point.

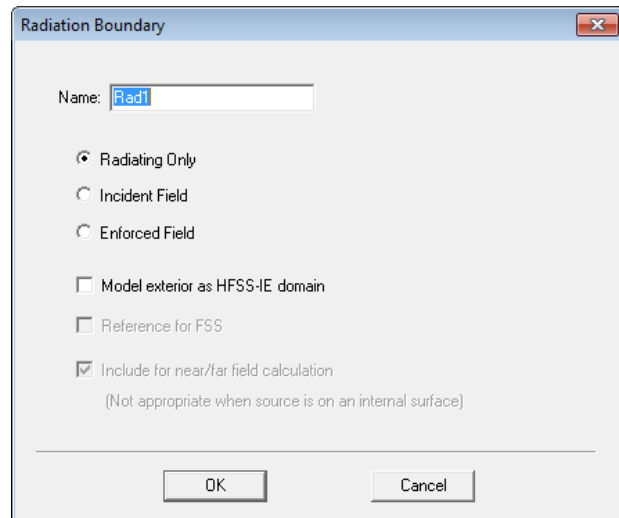
Antenna/EMI	Array/RCS	SI
Radiation		Radiation
PML	PML	
FE-BI	FE-BI	

# Radiation Boundary

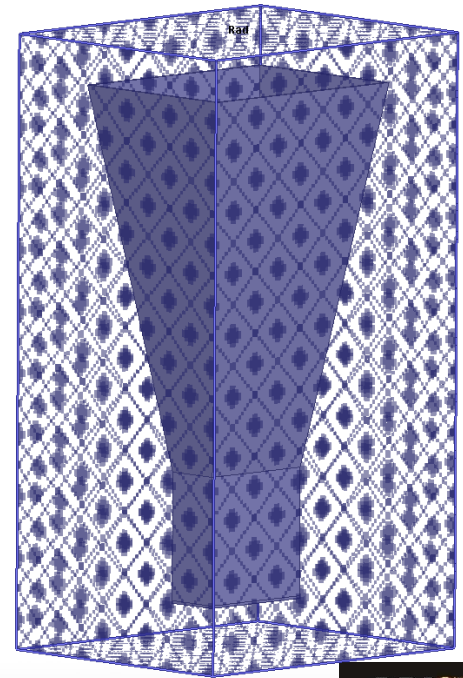
- **Mimics continued propagation beyond boundary plane**

- Absorption achieved via 2nd order radiation boundary
- Absorbs best when incident energy flow is normal to surface
- Distance from radiating structure
  - Place at least  $\lambda/4$  from strongly radiating structure
  - Place at least  $\lambda/10$  from weakly radiating structure
- Must be concave to all incident fields from within modeled space

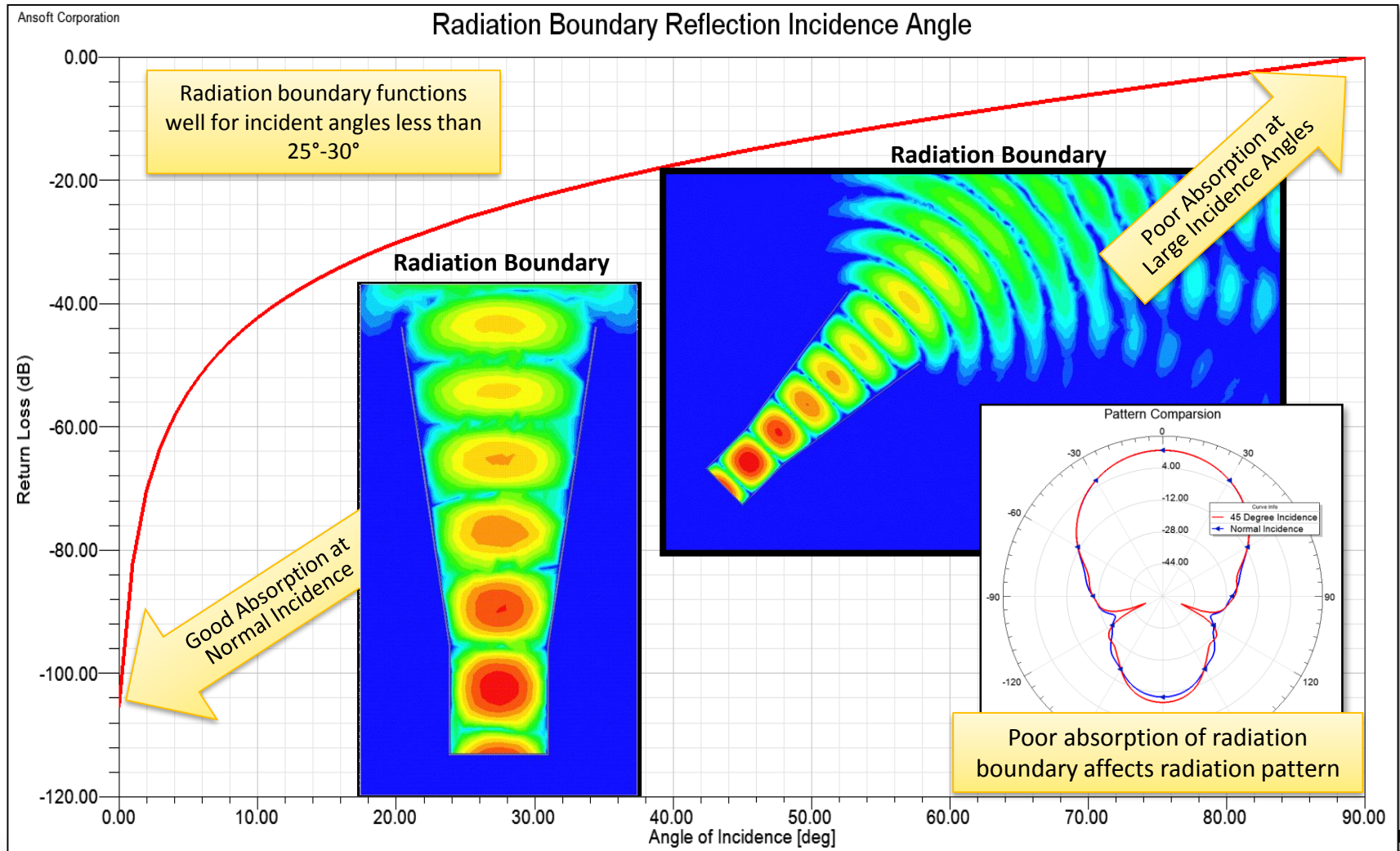
$$(\nabla \times \mathbf{E})_{tan} = jk_0 \mathbf{E}_{tan} - \frac{j}{k_0} \nabla_{tan} \times (\nabla_{tan} \times \mathbf{E}_{tan}) + \frac{j}{k_0} \nabla_{tan} (\nabla_{tan} \cdot \mathbf{E}_{tan})$$



Boundary is  $\lambda/4$  away from horn aperture in all directions



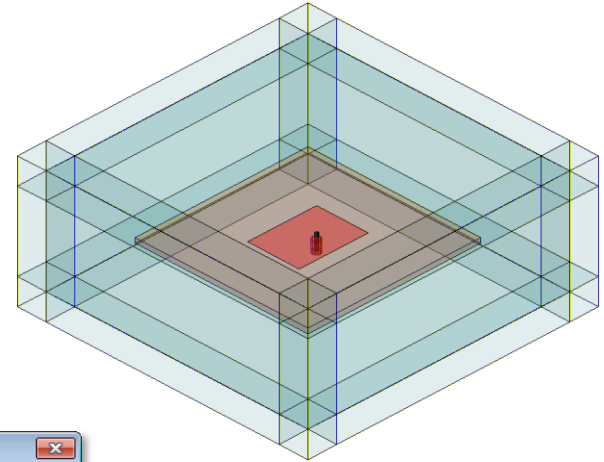
# Radiation Boundary: Incidence Angle Dependency



# Perfectly Matched Layer (PML)

- **Perfectly Matched Layer (PML)**

- Fictitious lossy anisotropic material which fully absorbs electromagnetic fields
- Two types of PML applications
  - “PML objects accept free radiation” if PML terminates free space
  - “PML objects continue guided waves” if PML terminates transmission line
- Guidelines for assigning PML boundaries
  - Use PML setup wizard for most cases
  - Manually create a PML when base object is curved or inhomogeneous



PML Setup Wizard : Cover Objects

☒ Create PML Cover Objects On Selected Faces

**1. Uniform Layer Thickness**

1.52211277464213 in

Use Default Formula

☒ Create joining corner and edge objects

☐ Use Selected Object as PML Cover  
(Use this option to cover non-planar faces)

Corresponding Base Object: AirBox

Uniform Layer Thickness: 1 in

Orientation in Local Coordinates: ☐ X Axis ☐ Y Axis ☐ Z Axis

☐ Do Not Create New Cover Objects

Base Face Radiation Properties

☒ Radiating Only

☐ Incident Field

< Back Next > Cancel

PML Setup Wizard : Material Parameters

**2. PML Objects Accept Free Radiation**

Min Frequency: 1 GHz

☐ PML Objects Continue Guided Waves

Propagation Constant at Min Frequency: 20 1/m

**3. Minimum Radiating Distance:**

3.1868 in

Use Default Formula

< Back Next > Cancel

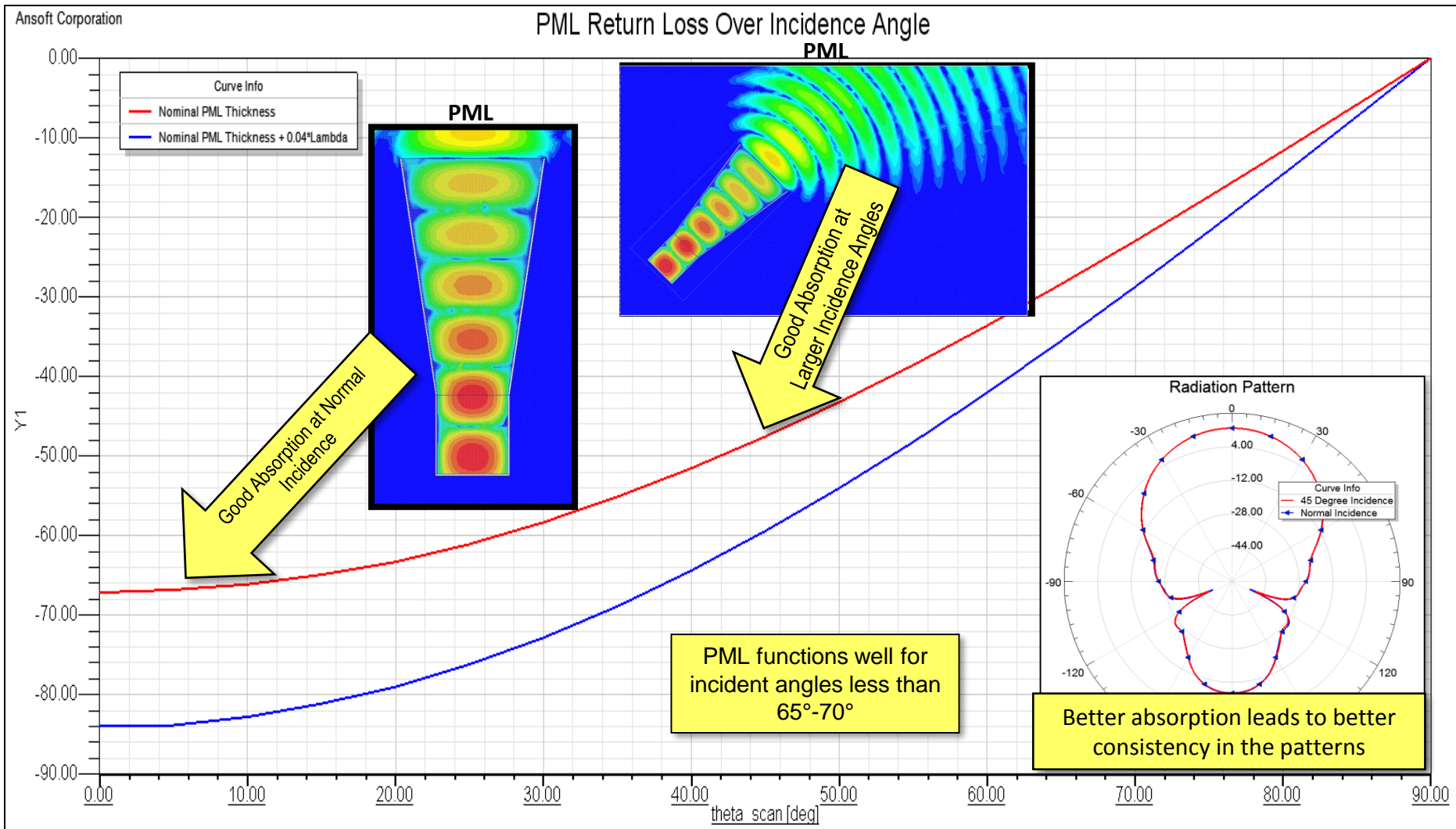
## PML Setup Wizard

### Parameters to Define:

1. Define PML Thickness
2. Specify Minimum Frequency (From Sweep)
3. Specify Minimum Distance from Antenna to PML (Measure in Model)



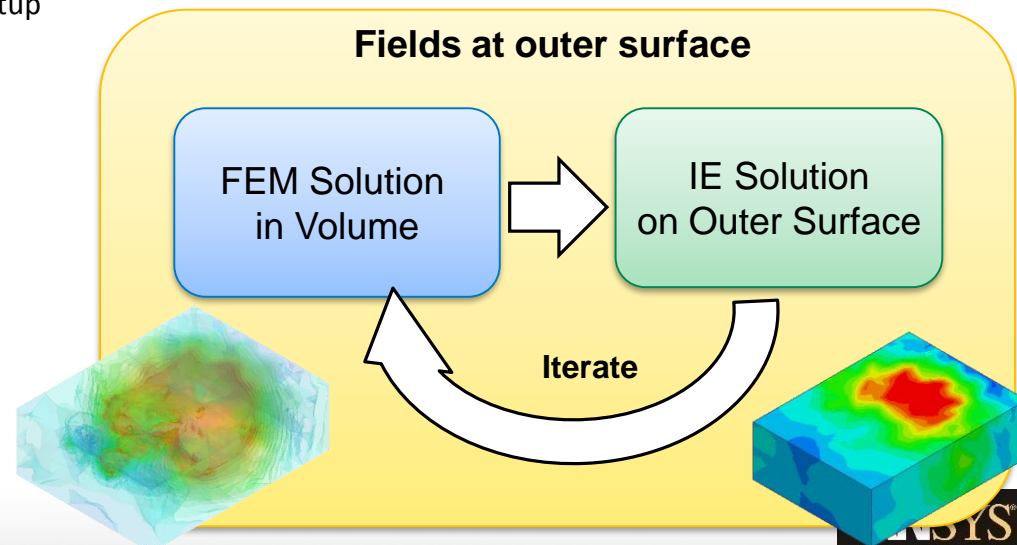
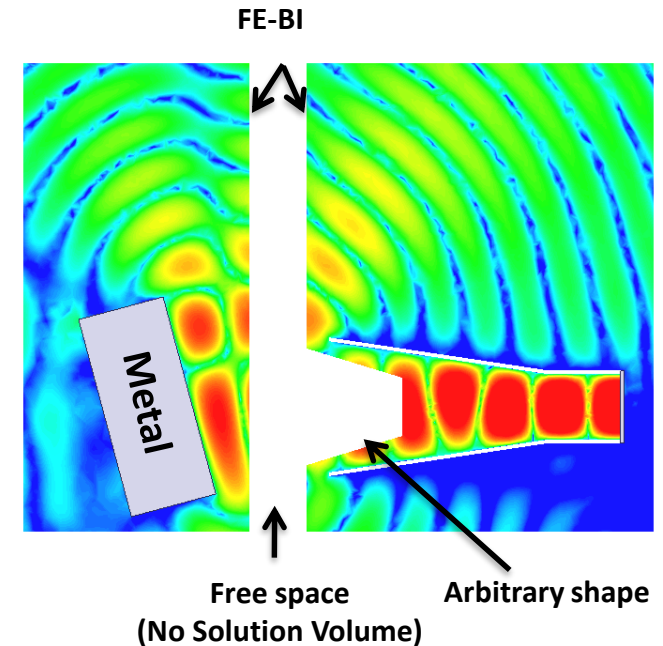
# PML: Incidence Angle Dependency



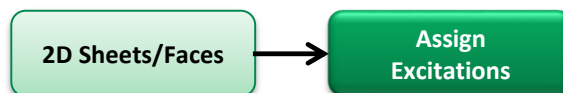
# Finite Element – Boundary Integral

- **FEBI**

- Mesh truncation of infinite free space into a finite computational domain
- Alternative to Radiation or PML
- Hybrid solution of FEM and IE
  - IE solution on outer faces
  - FEM solution inside of volume
- FE-BI Advantages
  - Arbitrary shaped boundary
    - Conformal and discontinuous to minimize solution volume
  - Reflection-less boundary condition
    - High accuracy for radiating and scattering problems
  - No theoretical minimum distance from radiator
    - Reduce simulation volume and simplify problem setup



# Excitations for 2D Sheets or Faces

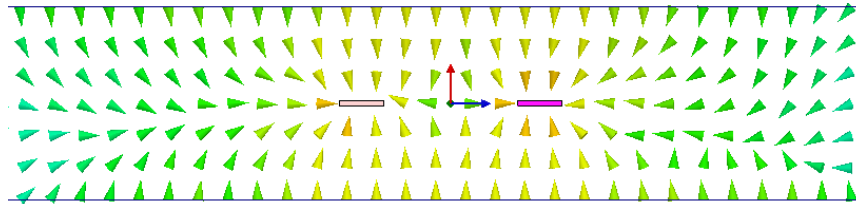




# Excitations

- **Excitations**

- There are many choices and important details regarding excitations that can make HFSS seem difficult to use. Most of the challenges faced with selecting and apply excitations result from the ability to create excitations that would not be possible in the physical world. Because this is just an introduction to HFSS, the following discussion will simplify the choices to establish a reliable and repeatable simulation process. If you need to solve designs outside of this introductory discussion, see Appendix 6.3 3D Excitations for an advanced discussion on the powerful excitation options available within HFSS.



- **Port Selection: When to select a Wave Port or Lumped Port?**

- **Wave Ports:** Use for closed port definitions. Examples of closed structures are Coax and Waveguides.
- **Lumped Ports:** Use for all other port definitions.
  - An exception to this guideline would be to use Wave Ports for Stripline

- **Solution Type: When to select Driven Modal or Driven Terminal?**

- **Driven Terminal** is typically faster to use because HFSS automates much of the setup. In general Driven Terminal should be used for any TEM/Quasi-TEM structures such as coax, microstrip, stripline, or co-planar waveguide. In the case of a design that only uses Lumped Ports, the resulting s-parameters will be identical in Driven Terminal and Driven Modal, therefore the ease of setup is usually the determining factor.
- **Driven Modal** is required for Waveguides.
  - In some cases, the fields post-processing might be easier when the excitations are defined in terms of Power (Driven Modal) instead of Voltage (Driven Terminal).

# Setting Solution Type

- **Solution type selection**

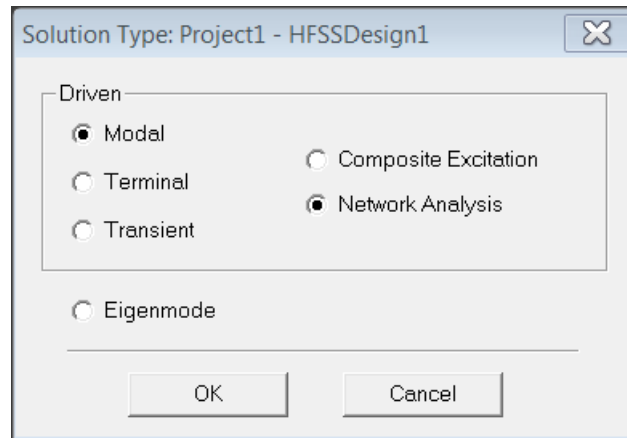
- As a general rule, one could choose the solution type based on the type of transmission line that is being analyzed

- **Driven Modal**

- Hollow waveguides (metallic rectangular, circular...etc)
- Any problem where a symmetry boundary condition is applied

- **Driven Terminal**

- Microstrip, stripline, coax, coplanar waveguide



# HFSS Excitation Methods

## Driven Modal

- Fields based transmission line interpretation
- Port's signal decomposed into incident and reflected waves
- Excitation's magnitude described as an incident power

### Modal Propagation

- Energy propagates in a set of orthogonal modes
- Modes can be TE, TM and TEM w.r.t. the port's normal
- Mode's field pattern determined from entire port geometry
- Each Mode has its own column and row in the S, Y, and Z parameters



## Driven Terminal

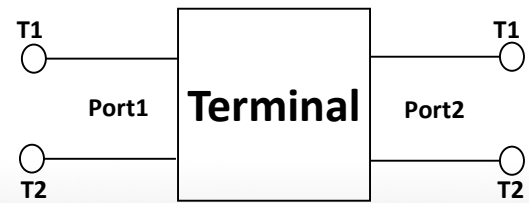
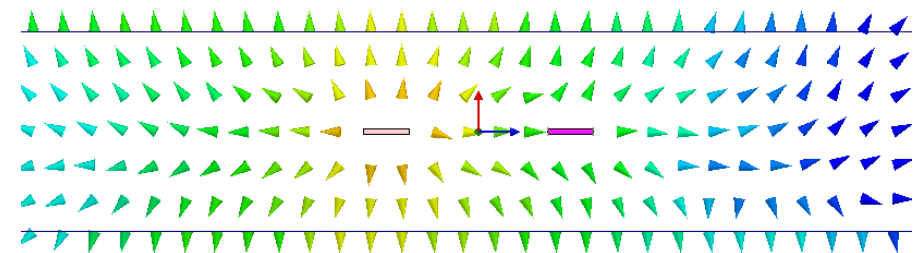
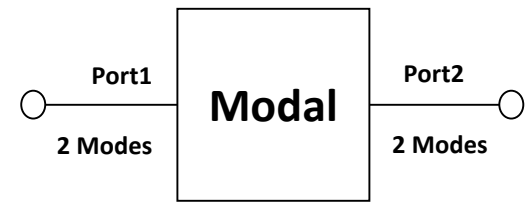
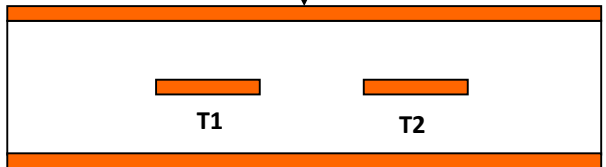
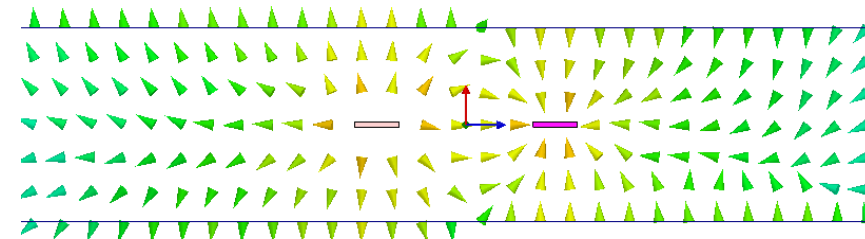
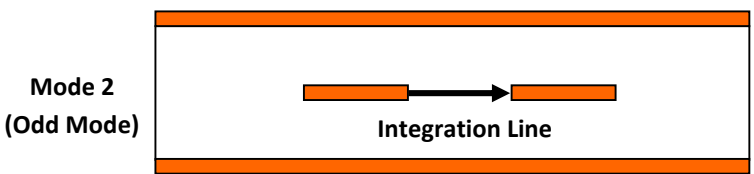
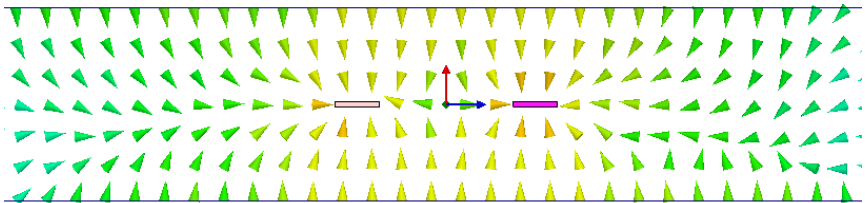
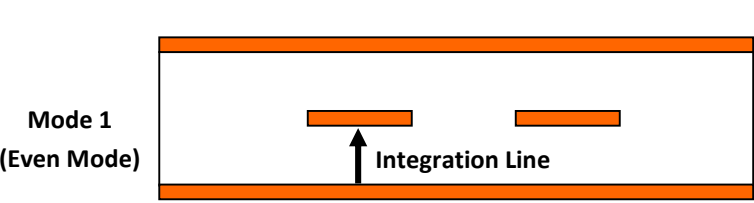
- Circuit Based transmission line interpretation
- Port's signal interpreted as a total voltage ( $V_{total} = V_{inc} + V_{ref}$ )
- Excitation's magnitude described as either a total voltage or an incident voltage
- Supports Differential S-Parameters

### Terminal Propagation

- Each conductor touching the port is considered a terminal or a ground
- Energy propagates along each terminal in a single TEM mode
- Each Terminal has its own column and row in the S, Y and Z parameters
- Does not support symmetry boundaries or Floquet Ports



# Excitations: Modal vs. Terminal

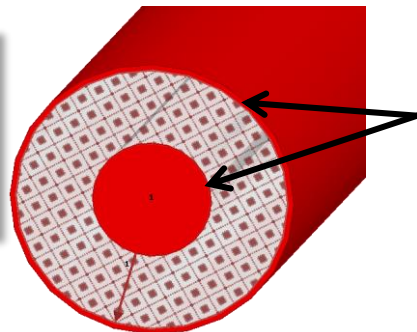


# Wave Ports

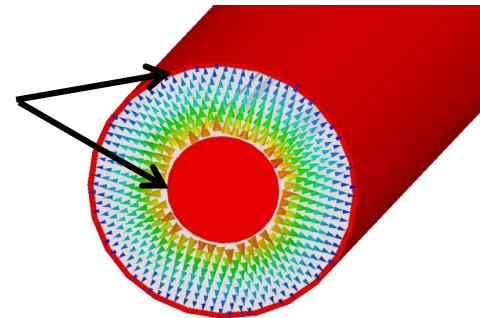
- **External port type**

- Arbitrary port solver calculates natural waveguide field patterns (modes)
  - Assumes semi-infinitely long waveguide with same cross-section and material properties as port surface
- Recommended only for surfaces exposed to background object
- Supports multiple modes, de-embedding, and re-normalization
- Computes generalized S-parameters
  - Frequency-dependent characteristic impedance
  - Perfectly matched at every frequency
- HFSS places a boundary condition on the Wave Port's edges to make sure the 2D eigenmode solution is finite sized
  - Typically the edge boundary takes the boundary condition defined in the 3D geometry.
    - **Exception:** When a Wave Port's edge touches a radiation boundary the edge's boundary is defined as PEC.
  - The Wave Port must encompass all relevant E-fields needed to describe the transmission line's behavior.

Freq	Port Zo	Gamma (Np/meter, rad/meter)	Lambda (meters)	Epsilon
10 (GHz)	1:1 ( 50.218, 0)	( 0, 209.58)	0.029979	1
	2:1 ( 50.553, 0)	( 0, 209.58)	0.029979	1



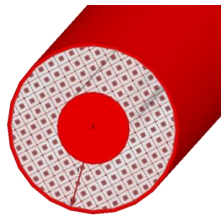
**Conducting  
Boundary  
Condition**



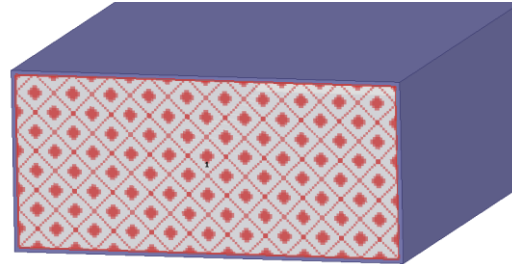
# Wave Port Sizing

- **Closed Transmission Line Structures**

- The boundary enforced on the port's edge implies the transmission line modeled by the Wave Port always sits inside a waveguide structure
- The enclosing material forms the port's edge boundary.



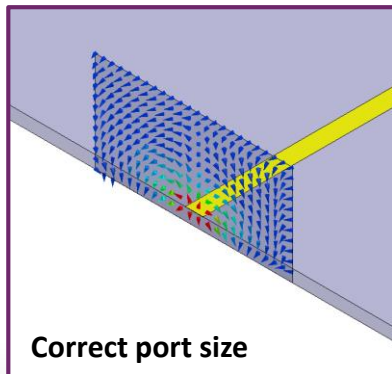
Coax



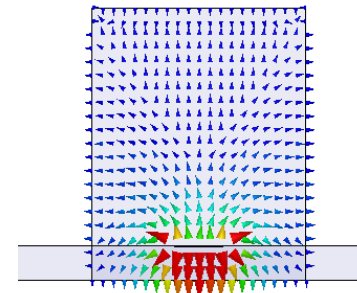
Waveguide

- **Open transmission line structures require additional consideration**

- Microstrip, Co-Planar Waveguide, Slotline (See Appendix for sizing recommendations)
- Wave Ports must be large enough to capture the entire transmission line's field structure
  - For open transmission line structures this mean the Wave Port must surround the entire structure
  - Make sure the transmission line fields are not interacting with the port's boundary condition.
  - Wave Ports that are too small can lead to incorrect characteristic impedances, and add addition reflection to the results



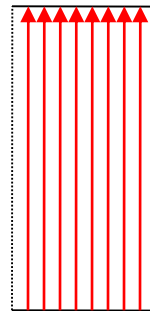
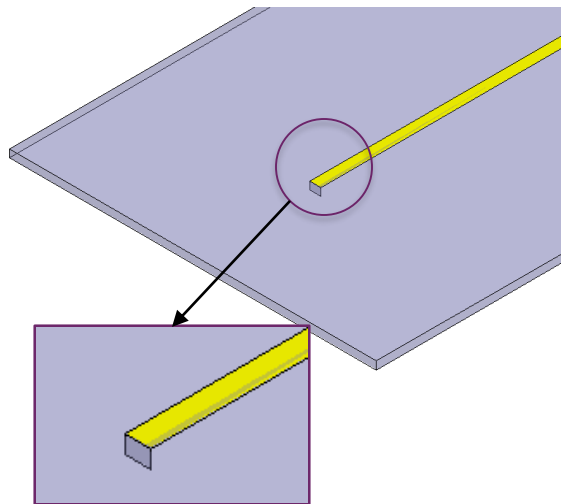
Correct port size



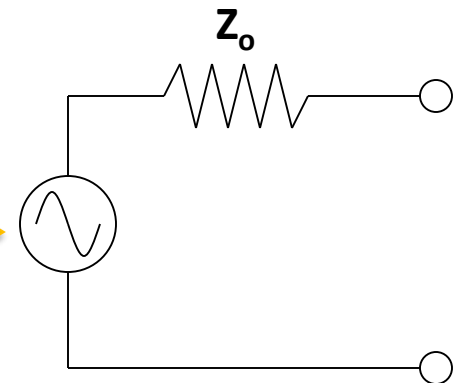
Port too narrow  
(fields coupled to sidewalls)

# Lumped Ports

- **Recommended only for surfaces internal to model**
  - Single TEM mode with no de-embedding
  - Uniform electric field on port surface
  - Normalized to constant user-defined  $Z_0$
- **Lumped port boundary conditions**
  - Perfect E or finite conductivity boundary for port edges which interface with conductor or another port edge
  - Perfect H for all remaining port edges



Uniform electric field  
User-defined  $Z_0$



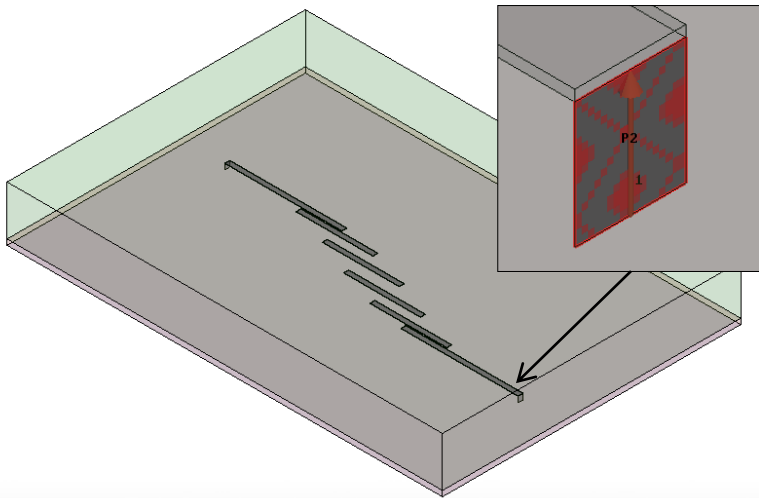
Dipole element  
with lumped port



# Lumped vs Wave Ports for Planar Filters

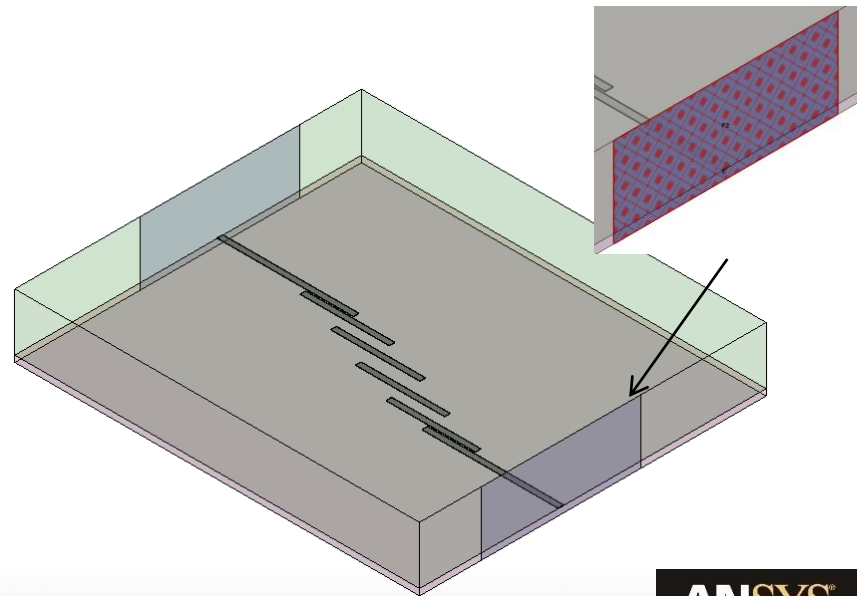
- **Lumped ports can be used to feed printed transmission lines**

- S-parameters normalized to user-specified characteristic impedance
- Single mode propagation
- No de-embedding operations available
- Must be located inside model



- **Wave ports can be used to feed printed transmission lines**

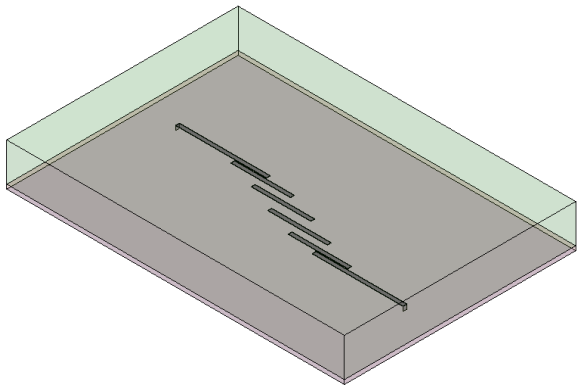
- S-parameters normalized to computed characteristic impedance (Generalized S-Parameters)
- Multiple propagating modes possible
- De-embedding available as post-processing operation
- Must touch background object (or be backed by conducting object)



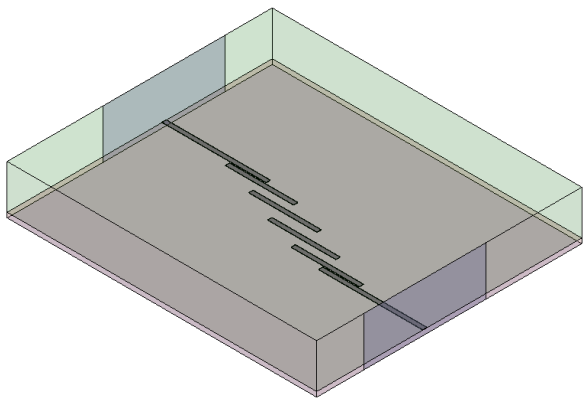


# Lumped vs Wave Ports for Planar Filters

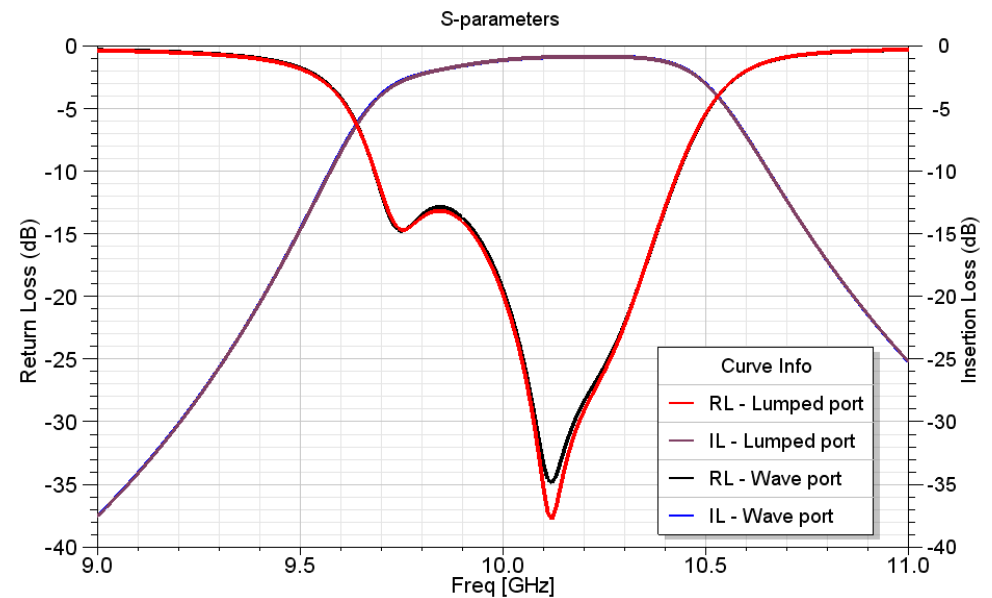
- Same results obtained from both port types



**Lumped Ports**



**Wave Ports**



# Wave Ports vs Lumped Ports

	Wave port	Lumped port
Accessibility	External Faces	Internal to Model
Higher order modes	Yes	No
De-embedding	Yes	No
Re-normalization	Yes	Yes
Setup complexity	Moderate	Low

# Analysis Setup

HFSS Analysis  
Setup

Adaptive Mesh  
Frequency

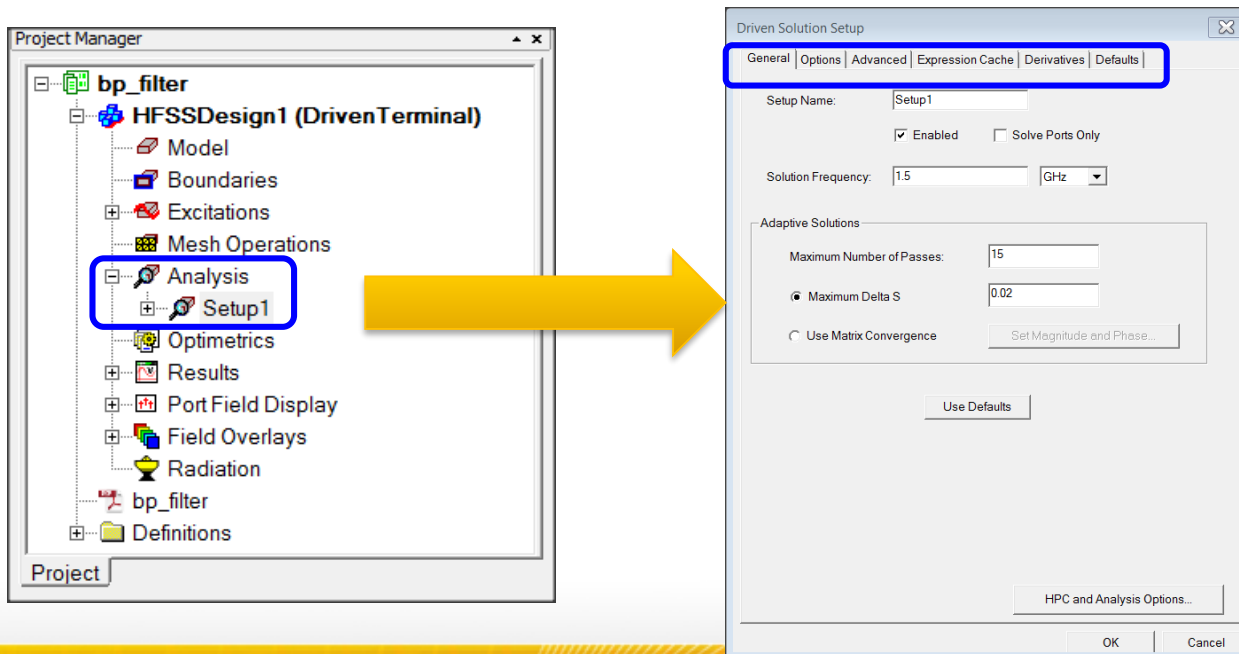
Solver  
Settings

Frequency  
Sweep

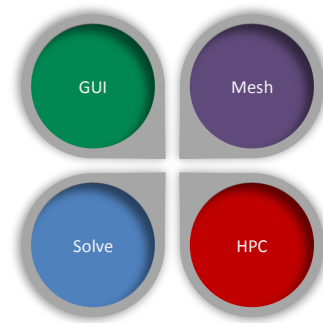
# Analysis Setup

- **The solution setup controls HFSS's solution process.**

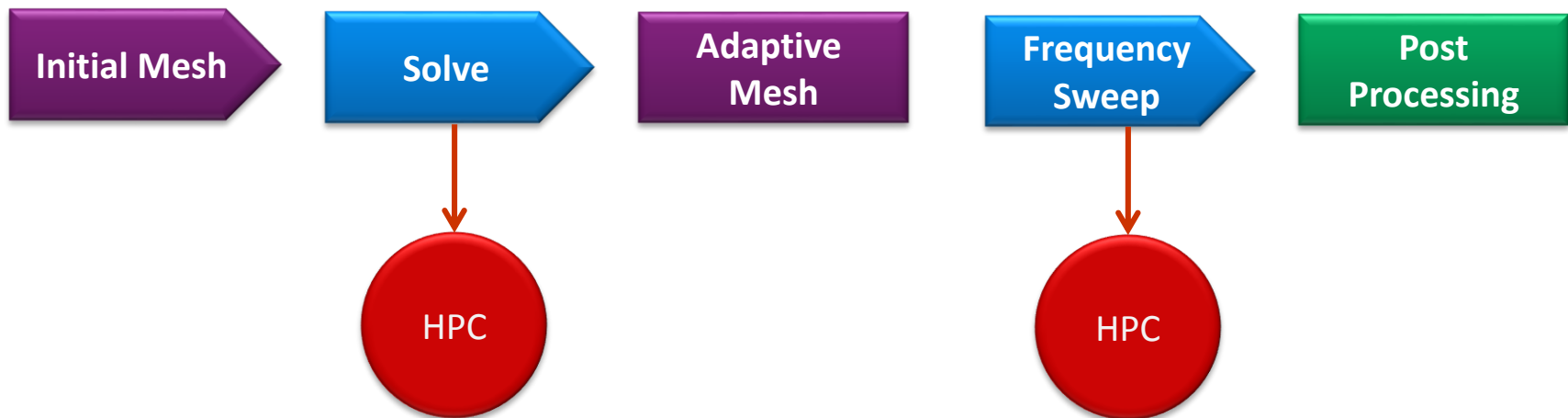
- Set the Solution Frequency
- Control the initial mesh
- Set the convergence criteria
- Control how the mesh is modified between passes
- Control the port's solution and convergence criteria
- Select the basis function used to describe the fields
- Choose the matrix solver
- Determine if Adjoint Derivatives will be calculated



# HFSS Solution Process

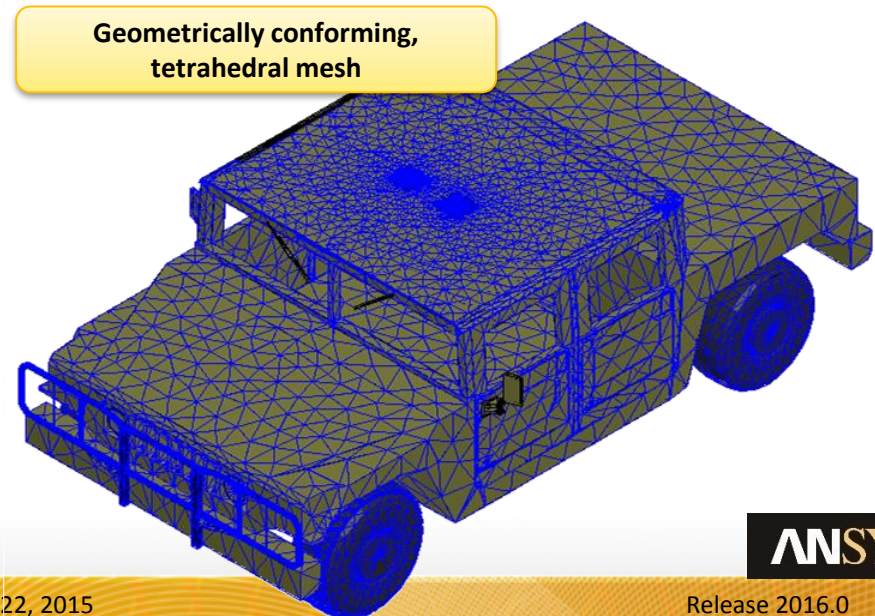
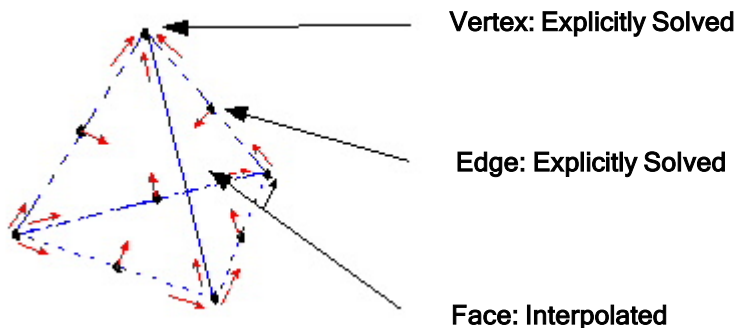


## Solution Process



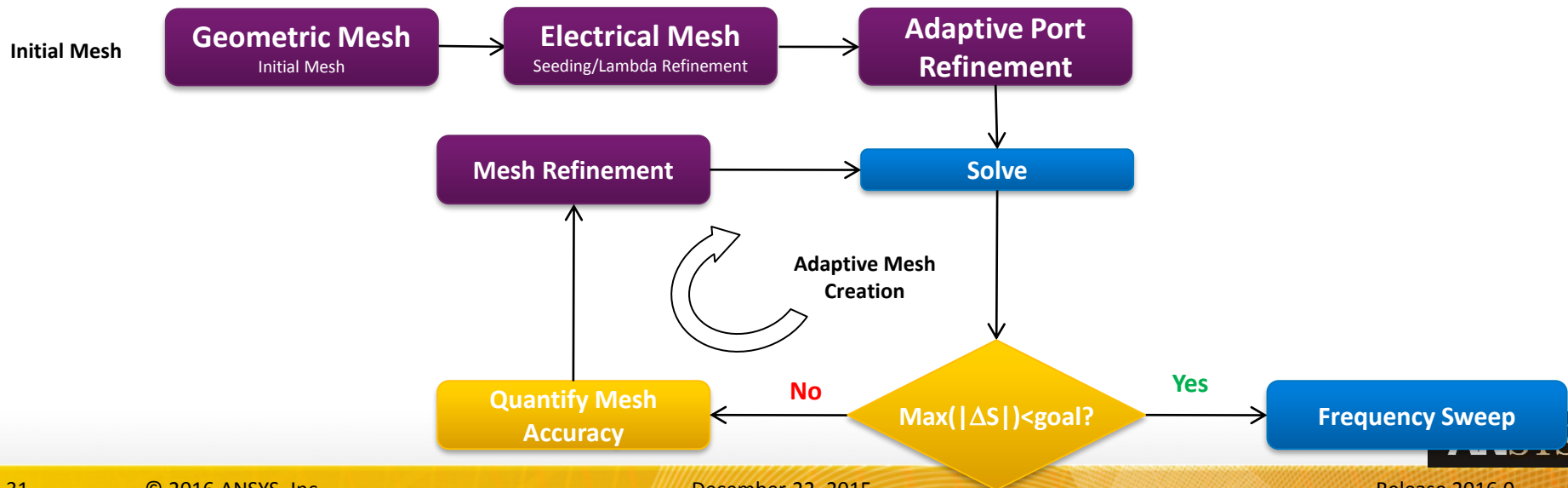
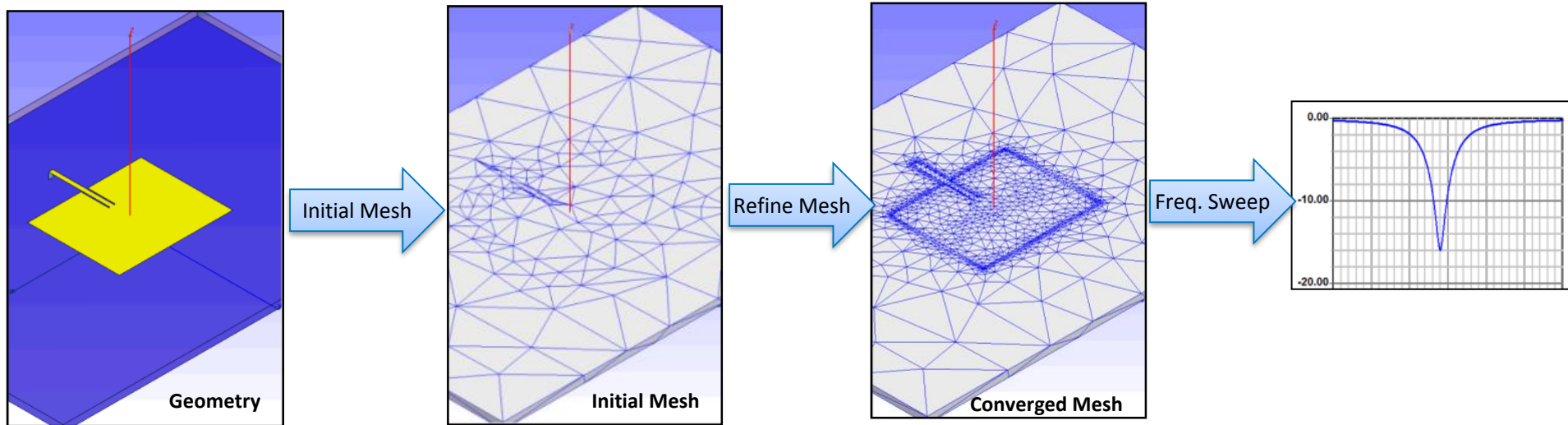
- **Automatic Adaptive Meshing for HFSS Simulation**

- Geometrically conforming, tetrahedral mesh automatically generated and refined below a user defined electrical length.
- Iterative algorithm solves the fields of the model and intelligently refines the mesh until S-parameters converge below a user defined threshold, Max Delta S.
  - User defines frequency or frequencies at which adaptive meshing is performed.
  - After each solution, tetrahedral elements are “graded” for their accuracy to Maxwell's Equations.
  - User defines percentage of “bad” tetrahedral elements to be refined after each pass (30% Default).



# HFSS – Automated solution process

Meshing





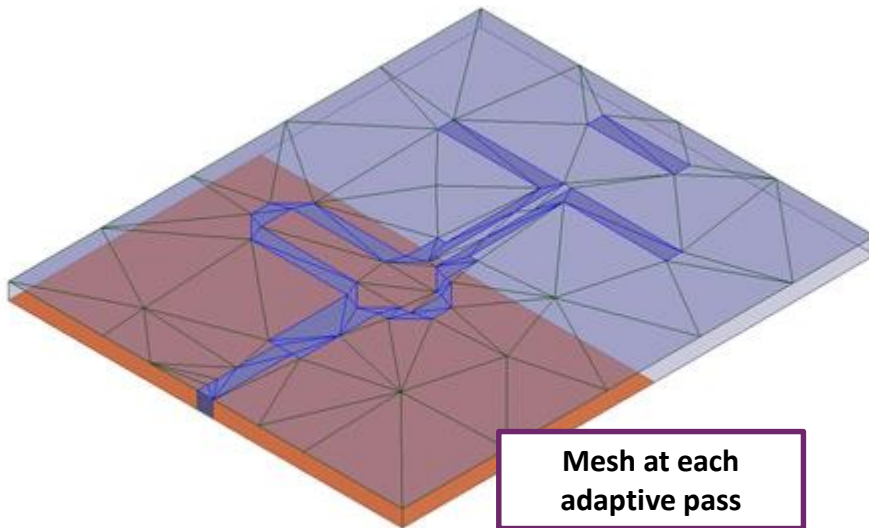
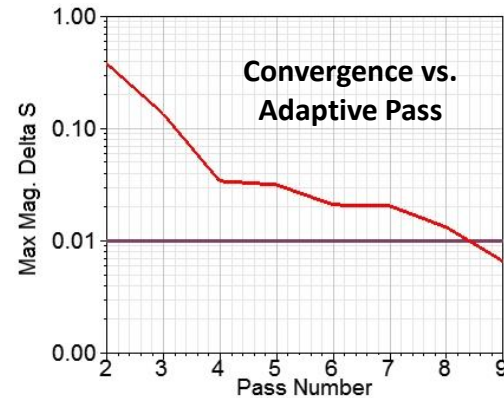
# Example: Adaptive Meshing

- **Automatic Adaptive Meshing**

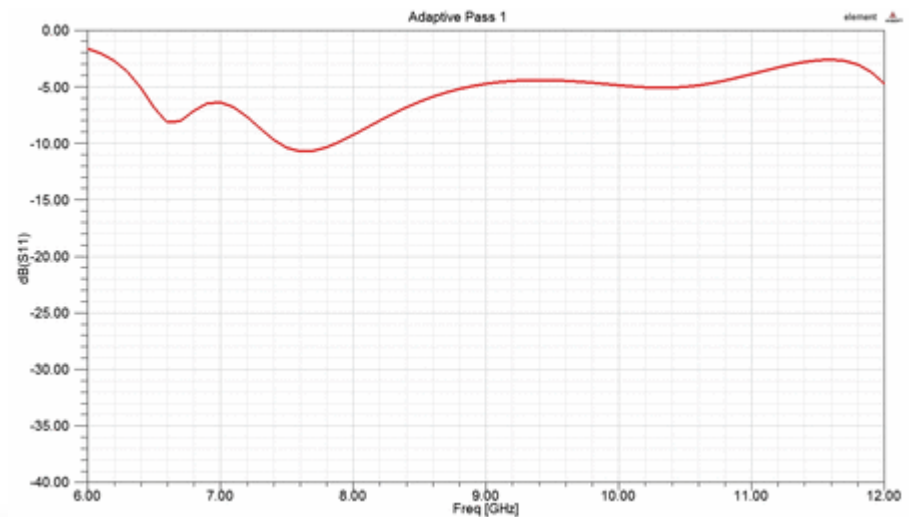
- Provides an Automatic, Accurate and Efficient solution
- Removes requirement for manual meshing expertise

- **Meshing Algorithm**

- Meshing algorithm adaptively refines mesh throughout geometry
- Iteratively adds mesh elements in areas where a finer mesh is needed to accurately represent field behavior
  - Resulting in an accurate and efficient mesh



Mesh at each adaptive pass





# Solution Frequency

- **The Solution Frequency sets:**

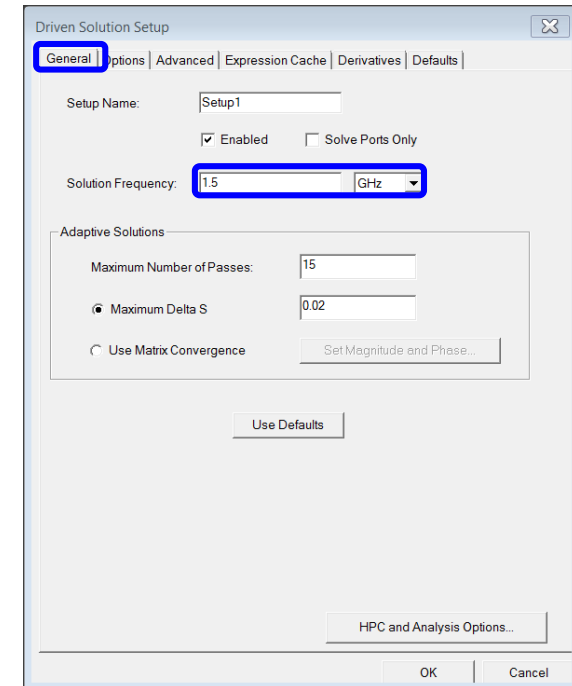
- The frequency used to create the adaptive mesh.
  - Defines the spatial resolution of the mesh through the Lambda Refinement step
  - Lambda Refinement is wavelength dependant.
- Determines the frequency used to evaluate the mesh's convergence.
- Typically choose the highest frequency of interest or for resonant antennas, the resonant frequency

- **A higher frequency mesh is generally valid at lower frequencies.**

- A mesh created at a higher frequency will be denser than a mesh at lower a frequency because the wavelength is smaller.
- The denser mesh is likely to pickup the field variations associated with lower frequencies behaviors.

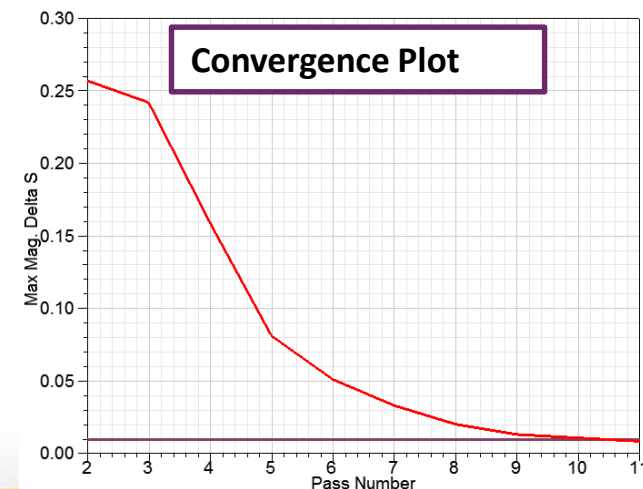
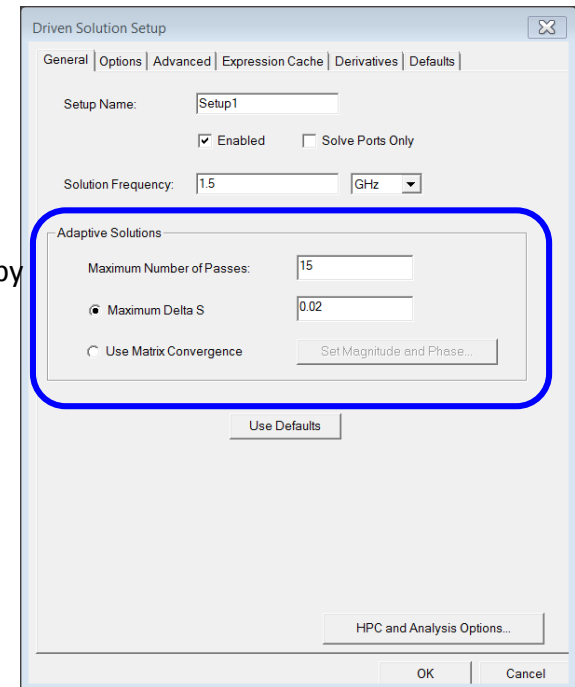
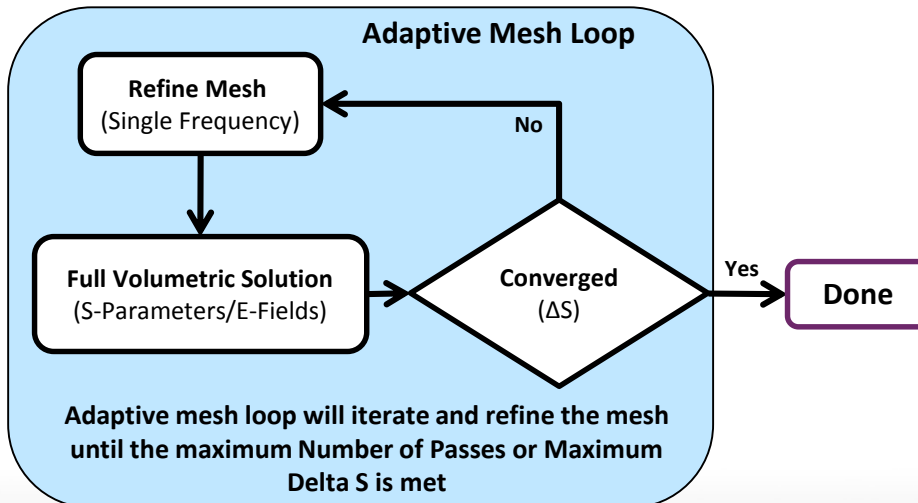
- **A low frequency mesh is generally NOT valid at higher frequencies.**

- A mesh created at a lower frequency will be coarser than a mesh created at a higher frequency because the wavelength is longer.
- The coarser mesh is less likely to pickup field variations associated with the higher frequencies



# Convergence Criteria

- **Adaptive mesh loop has two exit criteria:**
  - Maximum Number of Passes (Time)
    - Controls how many adaptive passes are allowed to be performed
    - 15 is a reasonable initial value
      - If value is too low, solution will complete before an accurate mesh is created by the adaptive solution process
      - Solution will likely take fewer than 15 passes to converge and will stop when convergence criteria is met
  - Convergence Per Pass (Accuracy)
    - Indicates the solution's sensitivity to mesh variations
    - Relates to the accuracy of the solution



# Converging on Maximum Delta S

- **Summarizes the S-Parameter's sensitivity**

- A single number for the entire S-Matrix
- Accounts for magnitude and phase variation for all S-parameters simultaneously.
- Default value of 0.02 is reasonable for most cases
- Reports the worst case violation

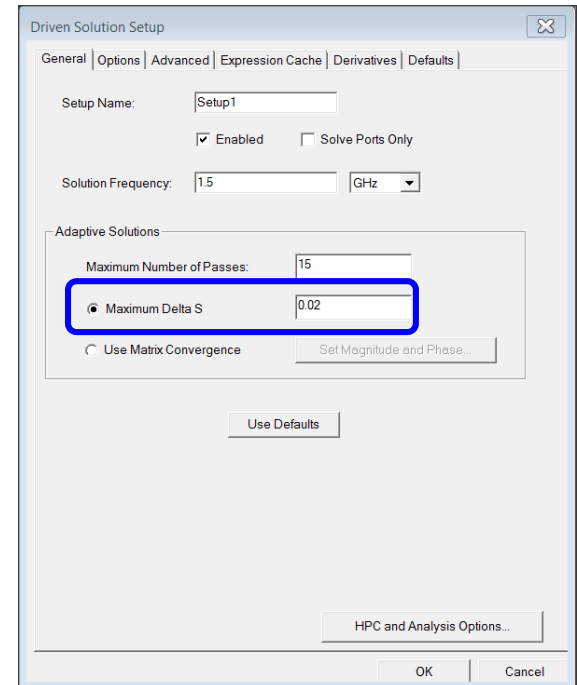
$$\text{Max}|\Delta S| = \text{Max}|[S]_N - [S]_{N-1}|$$

- **DON'T over-specify**

- Setting the Maximum Delta S too small wastes computer resources and time.

- **DON'T under-specify**

- Setting the Maximum Delta S Too large jeopardizes accuracy.



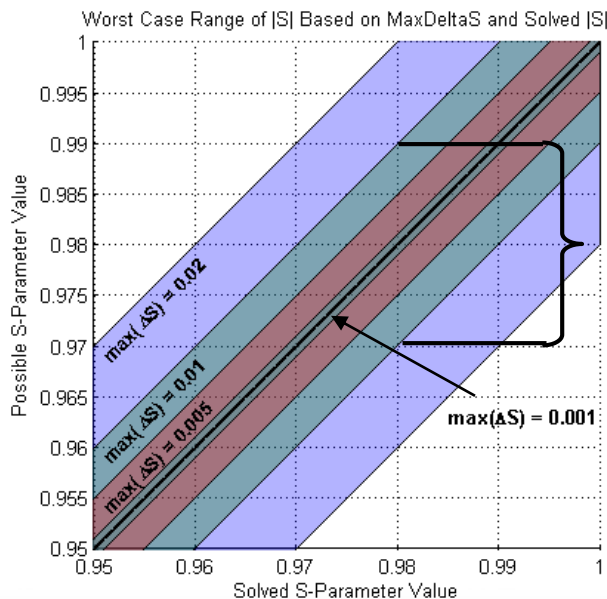
# Setting The Maximum Delta S Criteria

## • Maximum Delta S

- Always set the Delta S with your error tolerance in mind.
- Consider the worst case scenarios when setting the Maximum Delta S
  - A worst case magnitude error can be determined by assuming the S-parameter magnitude is the only source of error (phase is perfectly accurate) .
  - Most likely actual solution is much closer since the error will be split between the magnitude & phase

Expected Range for  $|S| = 0.98$  &  $\text{Max}(|\Delta S|) = 0.01$

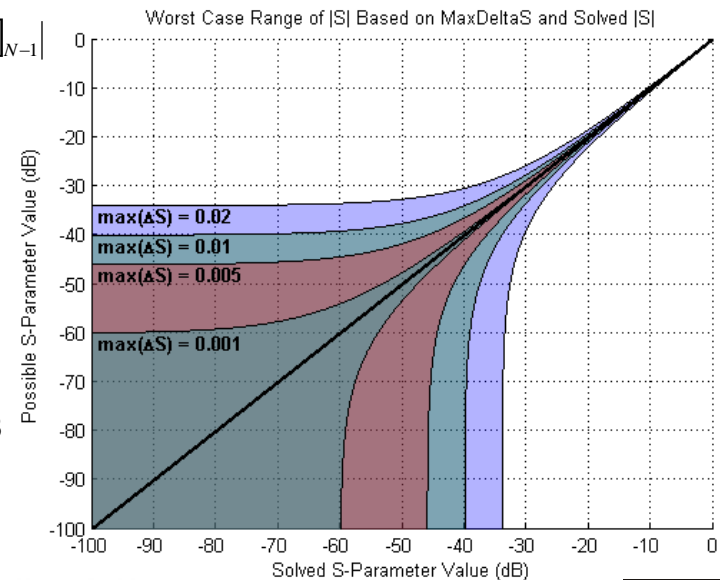
### Linear Units



$$\text{Max}|\Delta S| = \text{Max}[S]_N - [S]_{N-1}$$

A solve value of  $S = 0.98$   
could be between 0.99  
and 0.97 when using  
 $|\Delta S| = 0.01$

### dB Units



\* plots assume asymptotic convergence

# FEM Solver Technology Overview

- **Direct Solver**

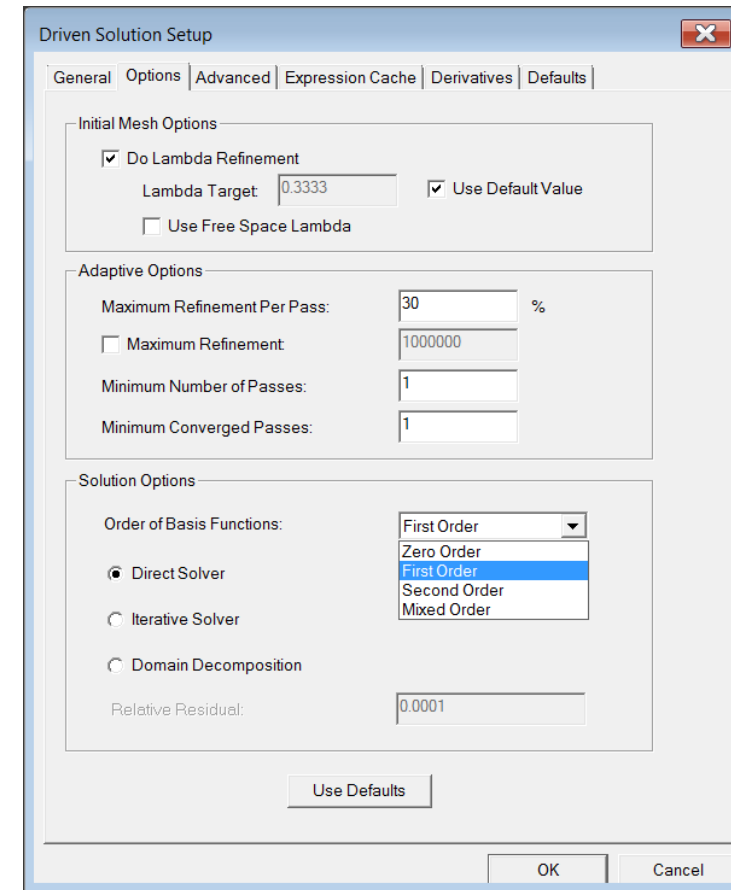
- Default technique
- Solves matrix equation  $Ax=b$ 
  - Multi-frontal Sparse Matrix Solver to find the inverse of A
  - Solves for all excitations(b) simultaneously

- **Iterative Solver**

- Reduces RAM and can improve simulation speed
- Solves matrix equation  $MAx=Mb$ 
  - M is a preconditioner – For HFSS this is a lower order basis function solution
  - Major computation is the matrix-vector multiplication:  $(MA)x$
  - Iterates for each excitation or simultaneously solve using HPC License
- Iterative Solver is more sensitive to mesh quality
  - Benefits from TAU Initial Mesh
  - Reverts to Direct solver if it fails to converge

- **Domain Decomposition**

- See High Performance Computing (HPC)



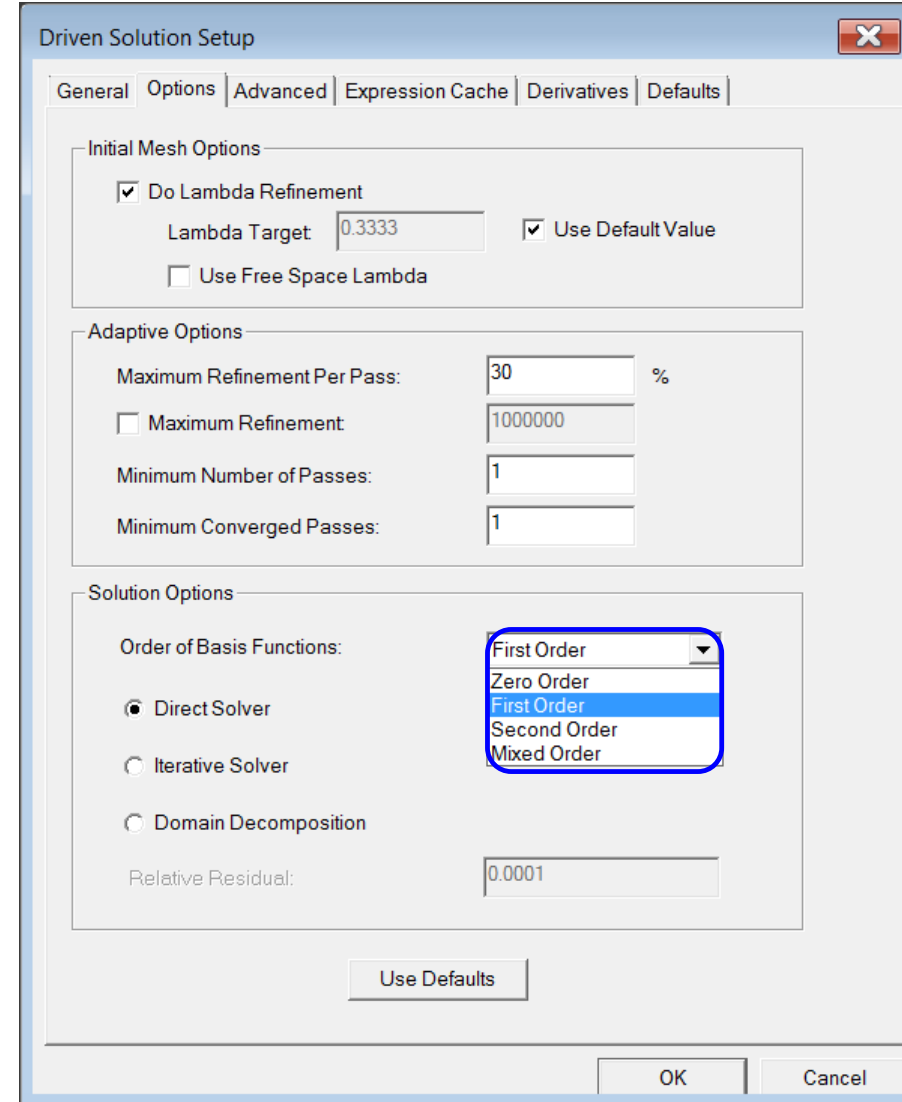
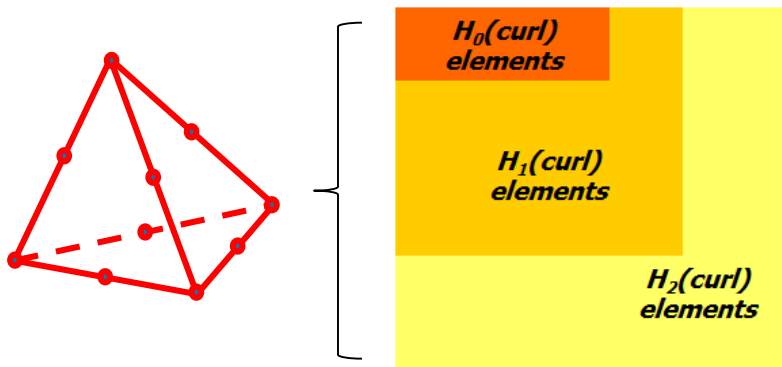
# FEM: Basis Order

## • Hierarchical basis functions

- Zero **or** First **or** Second order basis functions
- Higher-order elements have increased accuracy
- Convergence is a function of basis order

## • Mixed Order

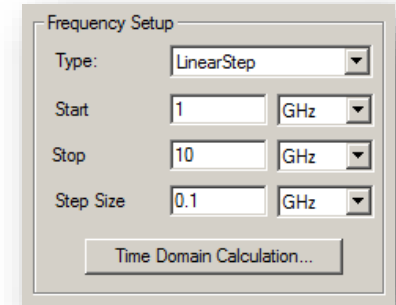
- hp-FEM Method
  - Refines element order( $p$ ) and element size( $h$ )
- Automatically distributes element order based on element size
- Generates optimum combination of hierarchical basis functions (Zero **and** First **and** Second)
- Efficient use of computing resources
- Default for HFSS 3D Layout



# Frequency Sweeps

- **Discrete Frequency Sweep**

- Solves using adaptive mesh at every frequency
- Matrix Data and Fields at every frequency in sweep

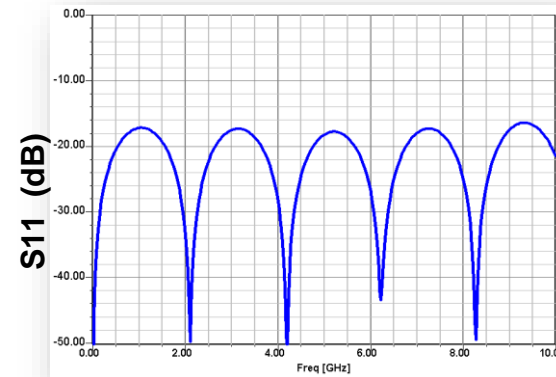


- **Interpolating Frequency Sweep**

- The calculation of wide-band s-parameters in HFSS is achieved using the interpolating sweep. This method fits s-parameter data to a rational polynomial transfer function using a minimum number of discrete finite element method (FEM) solutions.
- Matrix Data at every frequency in sweep

$$S = \frac{\beta_q (s - z_q)(s - z_{q-1}) \dots (s - z_1)}{\alpha_q (s - p_q)(s - p_{q-1}) \dots (s - p_1)}$$

See: IEEE Trans. Microwave Theory Tech., Vol. 46, No. 9, Sept. 1998



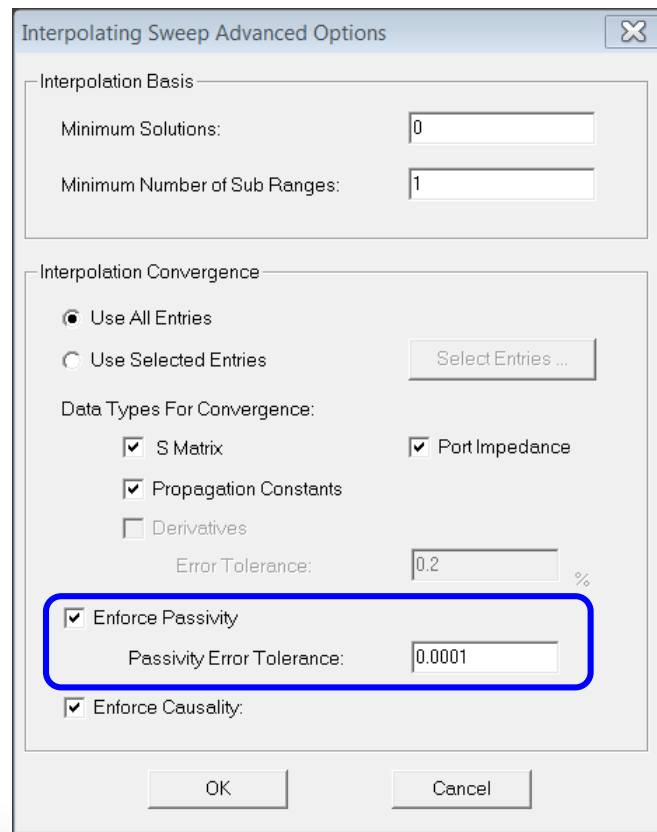
- **Fast Frequency Sweep**

- Uses an Adaptive Lanczos-Padé Sweep (ALPS)- based solver to extrapolate the field solution across the requested frequency range from the center frequency field solution
- The time and memory required for a Fast sweep may be significantly greater than the time and memory required for a single frequency solution.
- Matrix Data and Fields at every frequency in sweep

# HFSS Frequency Sweeps

- **Interpolation Convergence**

- The interpolating sweep yields the poles and zeros of the transfer function. This information can be directly used in the Laplace Element from which a Full-Wave SPICE™ model can be generated (HSPICE, Spectre RF, PSPICE).
- Enforce Passivity
- Enforce Causality



The image shows the 'Interpolating Sweep Advanced Options' dialog box. It has two main sections: 'Interpolation Basis' and 'Interpolation Convergence'. In the 'Interpolation Basis' section, 'Minimum Solutions' is set to 0 and 'Minimum Number of Sub Ranges' is set to 1. In the 'Interpolation Convergence' section, 'Use All Entries' is selected. Under 'Data Types For Convergence', 'S Matrix', 'Port Impedance', and 'Propagation Constants' are checked, while 'Derivatives' is unchecked. The 'Error Tolerance' is set to 0.2%. The 'Enforce Passivity' checkbox is checked and highlighted with a blue rectangle, with its 'Passivity Error Tolerance' set to 0.0001. The 'Enforce Causality' checkbox is also checked. At the bottom are 'OK' and 'Cancel' buttons.

Interpolating Sweep Advanced Options

Interpolation Basis

Minimum Solutions: 0

Minimum Number of Sub Ranges: 1

Interpolation Convergence

☒ Use All Entries

☐ Use Selected Entries [Select Entries ...](#)

Data Types For Convergence:

☒ S Matrix ☒ Port Impedance

☒ Propagation Constants

☐ Derivatives

Error Tolerance: 0.2 %

☒ Enforce Passivity

Passivity Error Tolerance: 0.0001

☒ Enforce Causality

OK Cancel