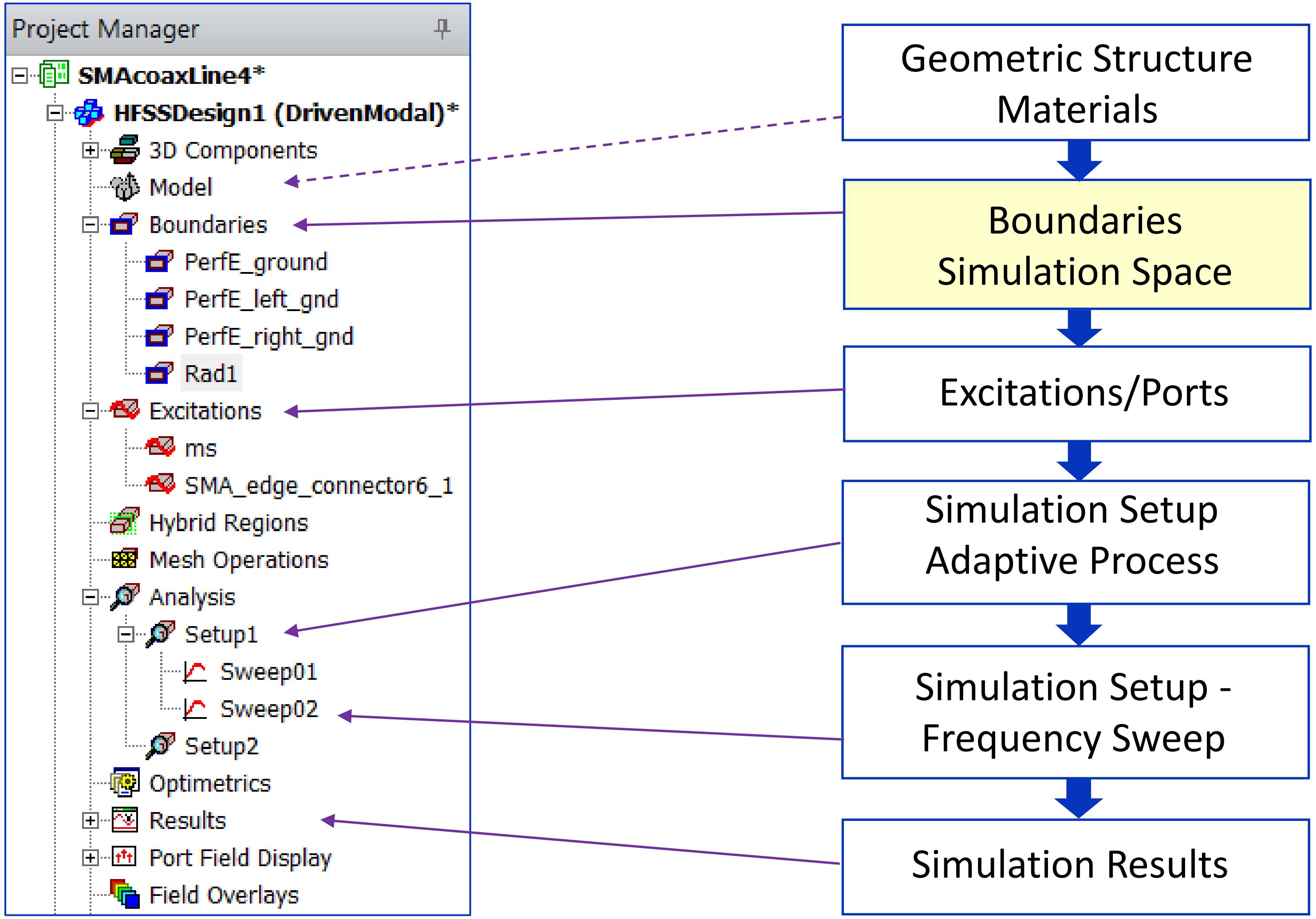
Module 2: Boundaries and Simulation Space

Release 2020 R2

Outline - HFSS Boundaries and FEM Simulation Space

- Boundaries Define HFSS Finite Element Method (FEM) Computational Volume
  - Boundaries define the simulation space (also called computational volume or solution space).
  - PEC boundaries can serve as the outer surface of the geometry of a closed model.
  - Air around a structure can be meshed in the computational volume (simulation space).
  - ***Radiation*** boundaries allow energy out of the solution space.
- Creating Boundaries and Simulation Space in HFSS FEM
  - ***Create Open Region*** makes a geometric region and assigns a boundary condition.
  - ***Draw > Region*** creates a ***Region*** in the model with padding.
  - ***PML*** (Perfectly Matched Layer) has a setup wizard.
  - Assigning boundary conditions to a radiating surface bounding box
- Types of Boundaries in HFSS
  - Boundary types: PEC, finite conductivity, ***Radiation***, ***PML***, and ***FE-BI*** are some
  - PEC - perfect electrical conductor
- Boundaries versus Materials
  - PEC boundaries can be used in place of ideal conductor material on a 3D solid
  - 2D sheets with boundary assignments can be inside of an HFSS simulation space

Boundaries in the HFSS *Project Manager* and Simulation Workflow



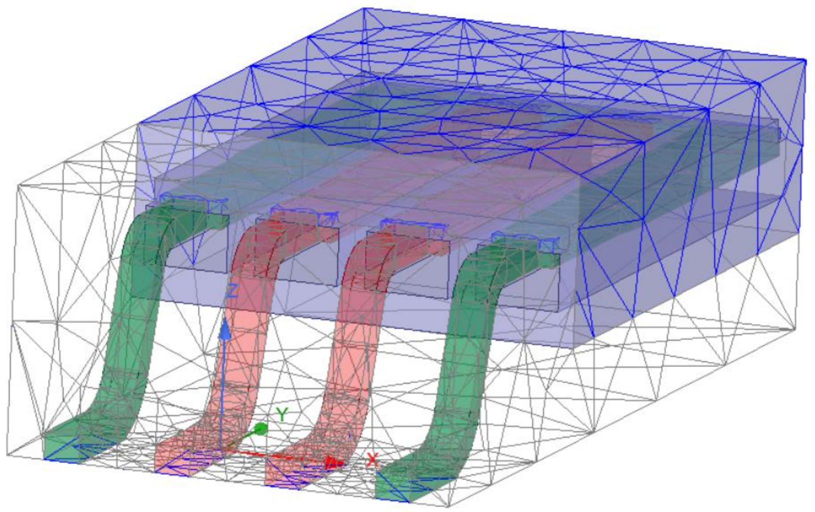
This presentation focuses on ***Boundaries*** in HFSS.

The HFSS Boundaries define what gets meshed and included in the FEM simulation.

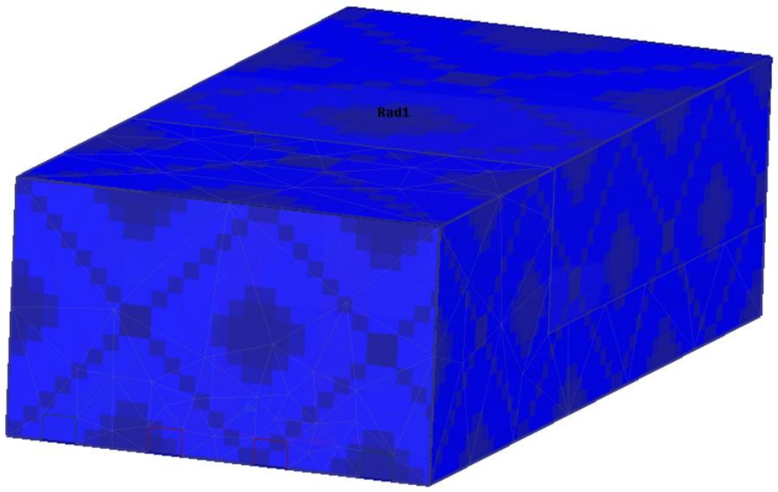
Boundaries relate to the HFSS model geometry and structure. Sometimes boundaries extend beyond the geometry and sometimes boundaries are defined on the outside of the model geometry.

The HFSS Online Help document ***HFSS.pdf***, chapter on ***Assigning Boundaries in HFSS and HFSS-IE*** is a good resource for ***Boundaries***.

HFSS FEM Mesh Gets Enclosed by Boundaries



- HFSS finite element method (FEM) first discretizes a simulation problem, creating a mesh.
- The mesh captures the electromagnetic character of the structure.
- The mesh is what gets simulated.
- A meshed connector is shown to the left.



- The mesh gets surrounded and enclosed by a boundary.
- The boundary defines the extents and boundary condition of the meshed simulation space.
- Boundaries and their properties provide the boundary conditions to the mathematics of the finite element method simulation.
- Boundaries and their properties are part of the accurate modeling of a given problem to be simulated.

The Purpose of Boundaries in HFSS

Closed and Open Models

The purpose of using boundary conditions in HFSS is to define the behavior of the electromagnetic field on the object interfaces and at the edges of a problem region. Defining boundary conditions reduces the electromagnetic or geometric complexity of the model.

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

An electromagnetically open model allows energy to emanate or radiate away. Common examples include an antenna, a printed circuit board, or any structure that is not enclosed within a closed cavity.

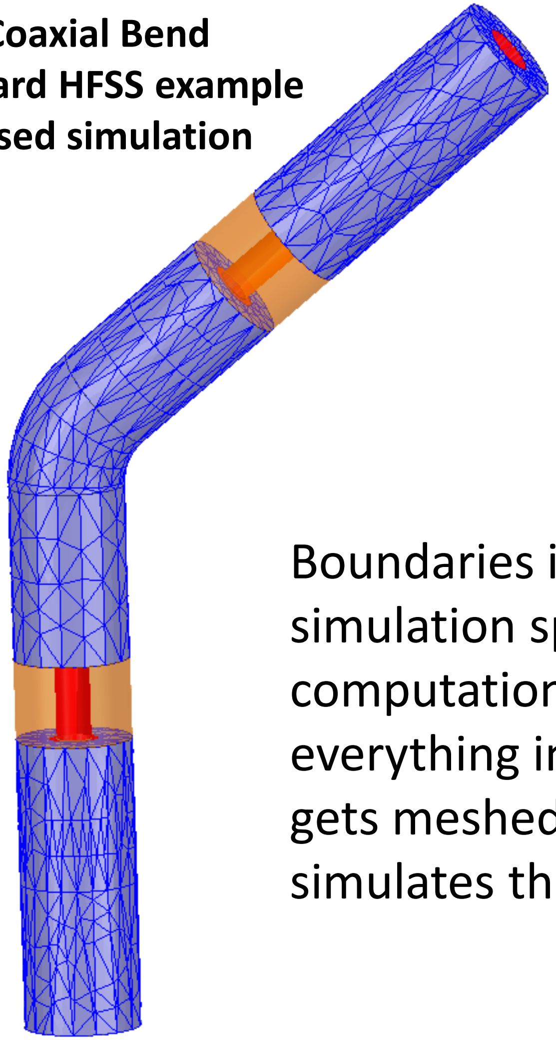
By default HFSS treats any given model as closed since all outer surfaces of the solution space are covered with a perfect electric conducting boundary. In order to create an open model, you must specify a boundary on the outer surfaces that overwrites the default perfect electric conducting boundary.

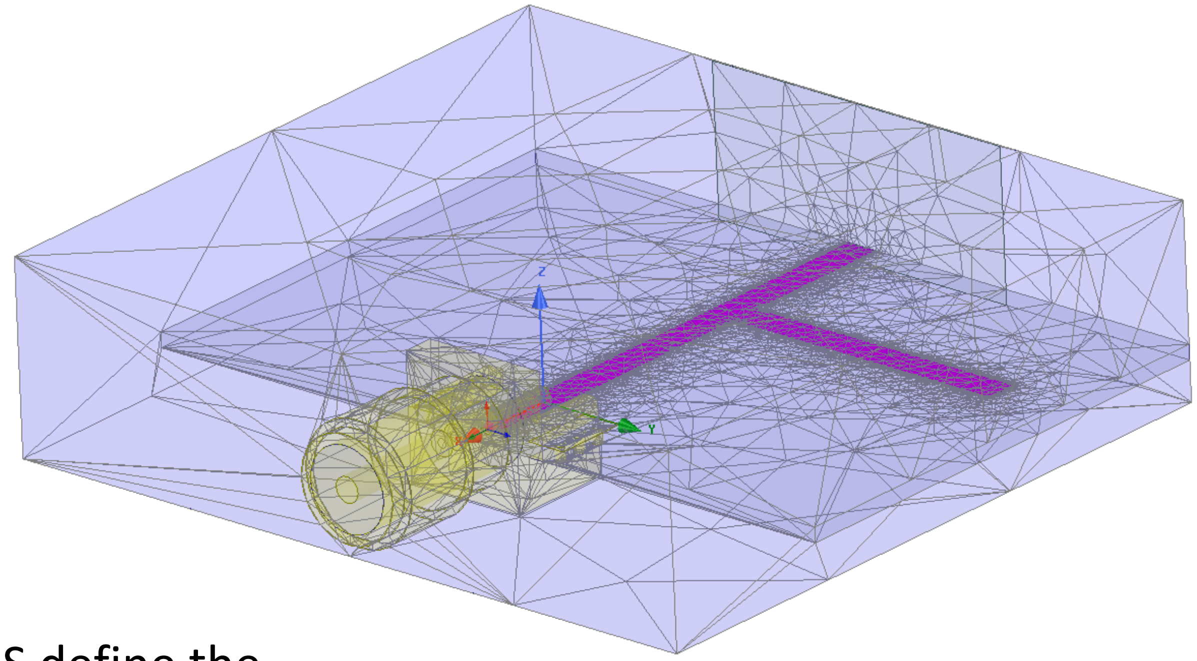
Boundary conditions are assigned on 2D sheet objects and surfaces of 3D objects.

This comes from ***An Introduction to HFSS.pdf***, in the HFSS install directories, in the chapter on ***HFSS Boundaries***. This is also available from the HFSS online ***Help***.

HFSS FEM Boundaries Define the Simulation Space

Coaxial Bend
Standard HFSS example
Closed simulation





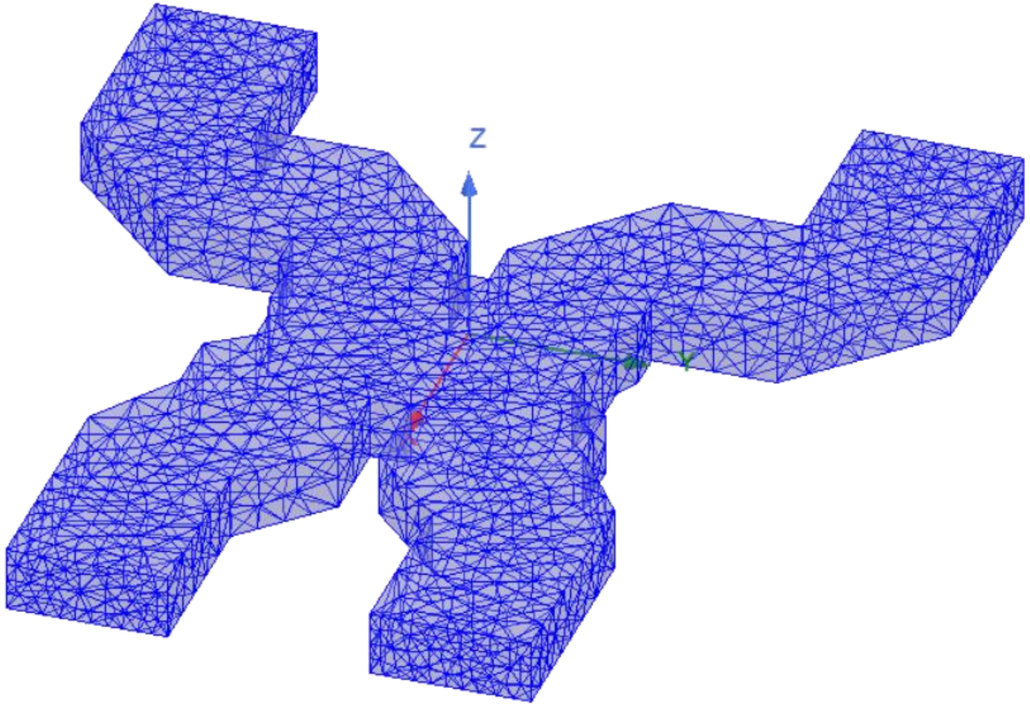
SMA coaxial to microstrip transition
Open simulation

Boundaries in HFSS define the simulation space, also called the computational volume, because everything inside the boundaries gets meshed and simulated. HFSS simulates the mesh.

The meshed region may be larger than the physical boundaries of the structure – or not. The coaxial connector boundary ***is*** the outer conductor. By contrast, the microstrip can have a boundary above and below the microstrip structure. Air above and below this microstrip gets meshed and simulated.

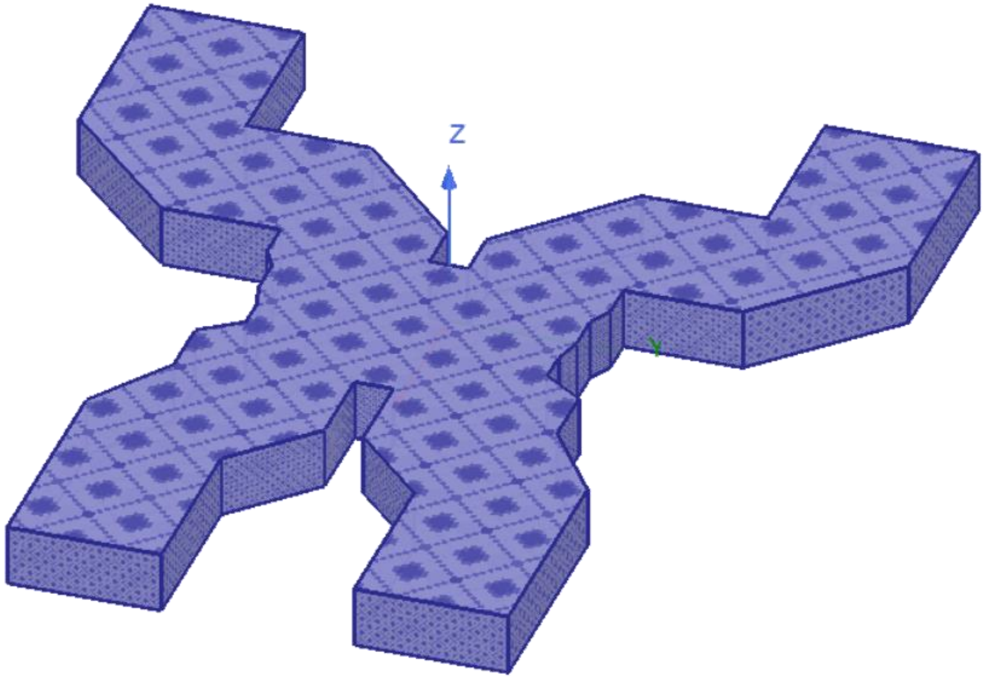
HFSS FEM Boundary Around the Simulation Space - Closed Model

The computational volume gets meshed.



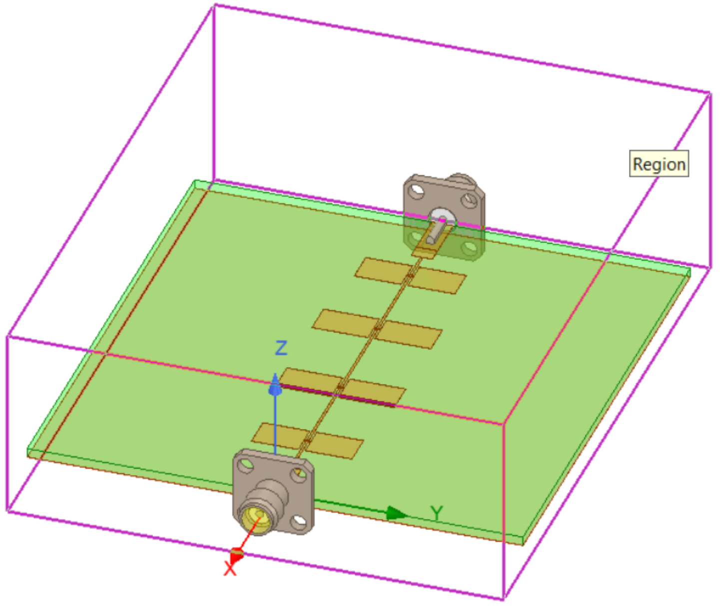
The entire volume, inside of this four-port waveguide combiner, gets meshed and simulated with finite element method (FEM).

A boundary surrounds the computational volume.

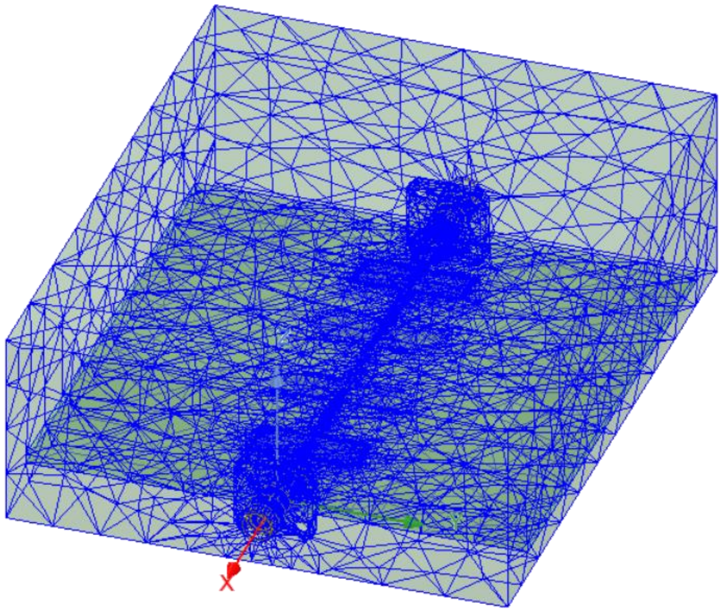


The outside faces of this waveguide combiner have a finite conductivity boundary that appears in the ***Project Manager*** under ***Boundaries***.

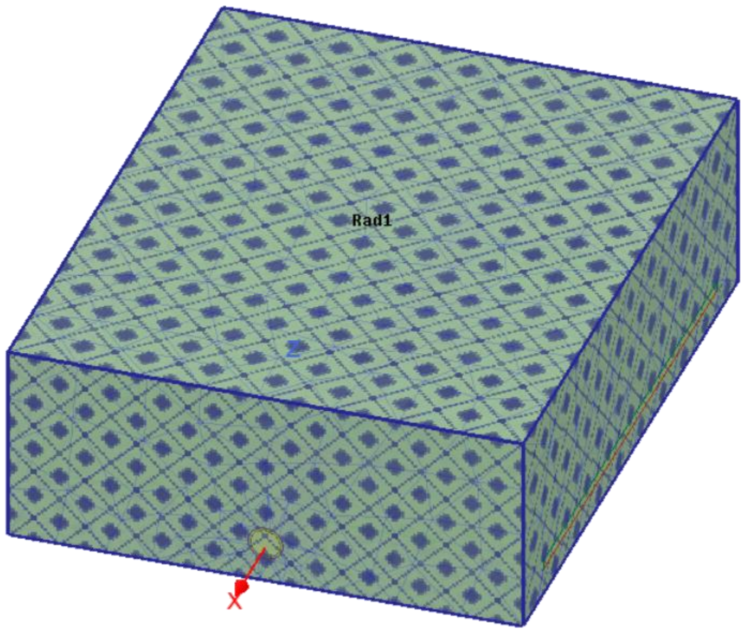
HFSS FEM Boundaries Around the Simulation Space - Open Model



The computational volume gets meshed.



A boundary surrounds the computational volume.

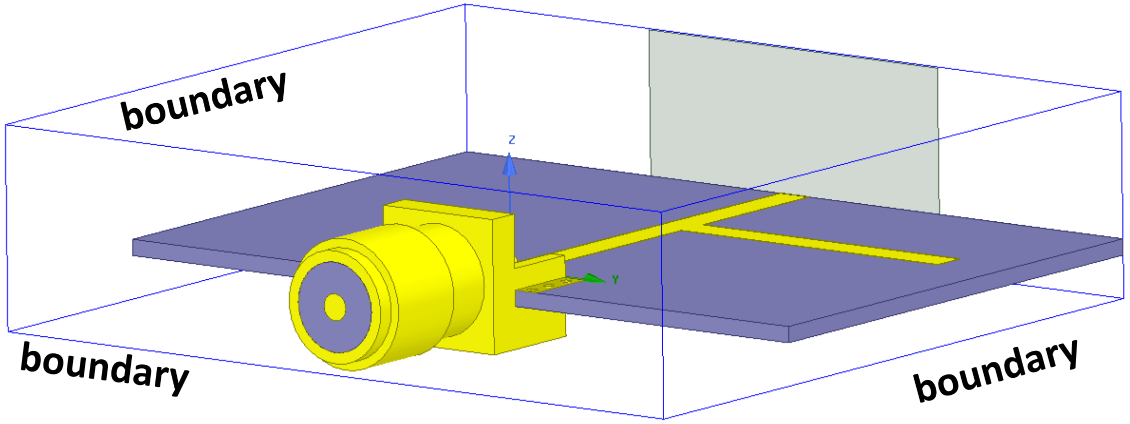


The structure is a 1.25 GHz low pass filter with SMA connectors. The ***magenta purple*** box is a ***Region*** created in the simulation specifically to define the boundary around the simulation space.

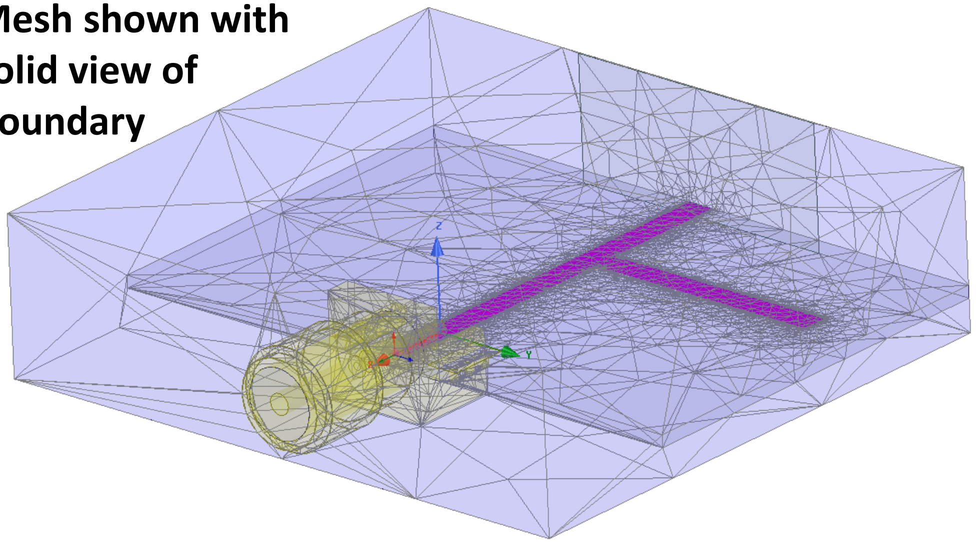
The entire volume inside of the ***Region*** box around the low pass filter gets meshed and simulated with finite element method (FEM). This includes substrate, air, and metal.

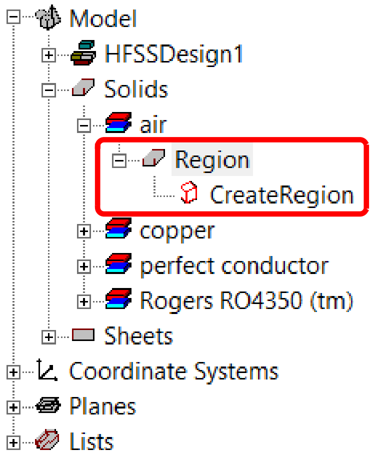
The entire surface of the ***Region*** box around the low pass filter has a ***Radiation*** boundary assigned.

Microstrip Boundary Includes Air in Computational Volume



Mesh shown with solid view of boundary





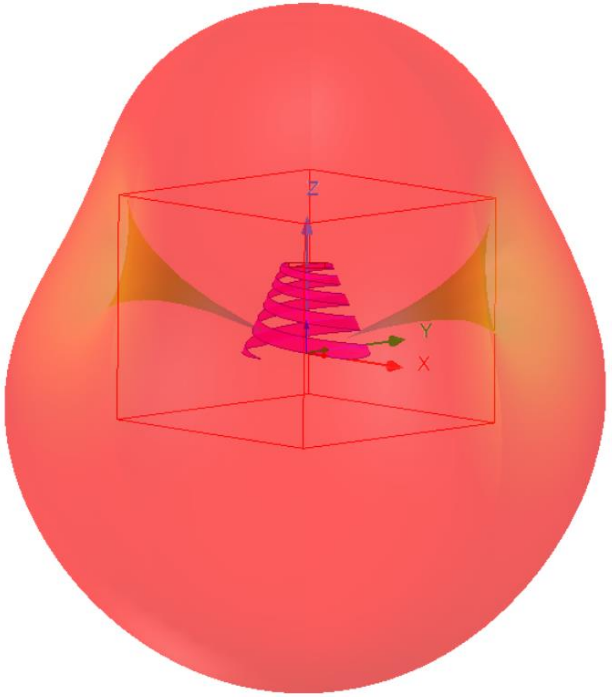
Microstrip includes an air region above the structure. This microstrip HFSS model has an air ***Region*** above and below the substrate (blue box in the picture). That entire air box is set to ***Radiation*** boundary. This entire simulation space, defined by the box, gets meshed in HFSS FEM.

HFSS FEM uses **3D mesh elements** - tetrahedra. This is sometimes referred to as ***volumetric meshing.***

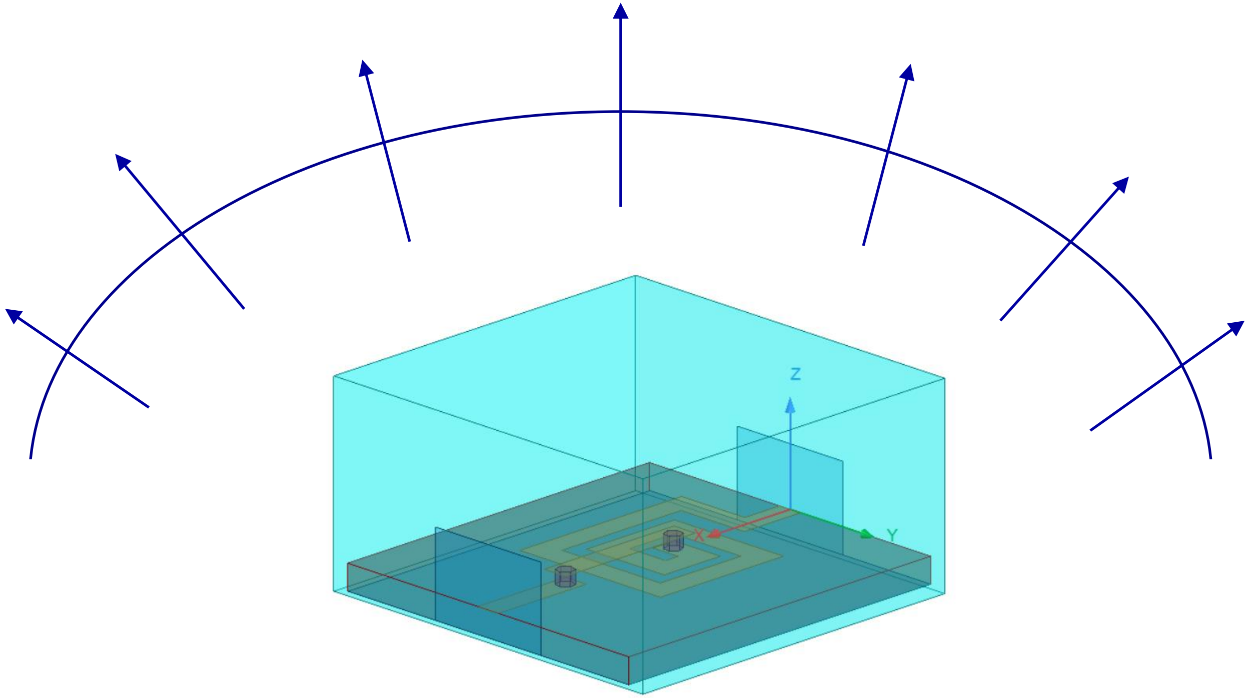
(In contrast, the HFSS integral (IE) solver uses surface meshing - 2D mesh elements - and doesn't have a computational volume.)

HFSS Absorbing Boundaries Model Radiation in Open Models

HFSS uses a defined FEM computational volume. In order to model physical devices with unbounded radiation into space, HFSS uses finite boundaries that absorb the radiation. By absorbing the radiation and removing it from the simulation volume, the absorbing boundaries model energy radiating away.



This spiral antenna is modeled inside of an airbox with a ***Radiation*** boundary. A far field pattern for one of the polarizations is overlaid on the model and Region box to illustrate how the real-world radiation emanates out of the box defined by the HFSS simulation space.



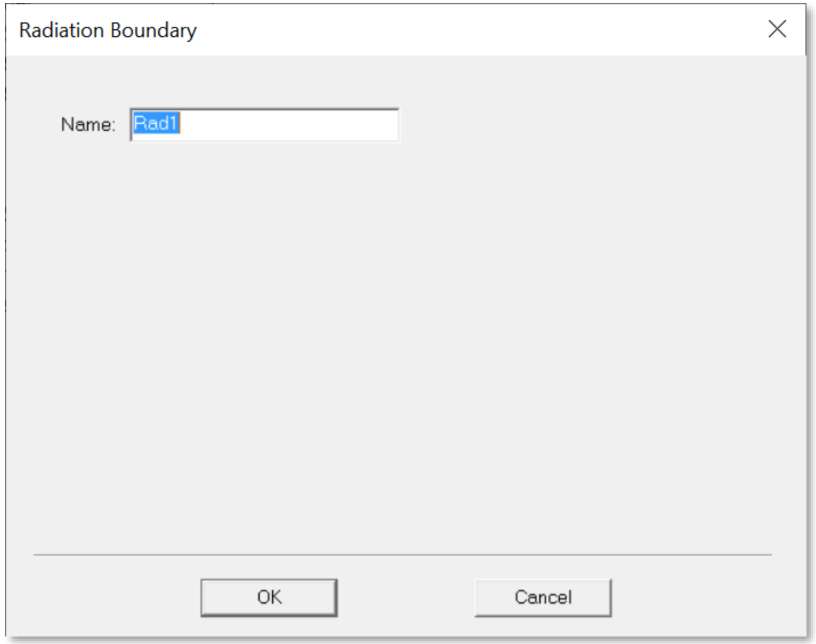
This spiral inductor is modeled inside of an airbox with a ***Radiation*** boundary.

Radiation Boundary

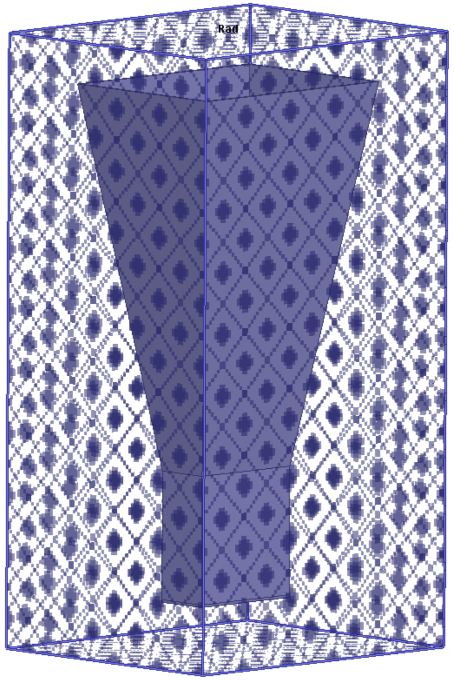
A ***Radiation*** boundary mimics continued propagation beyond boundary plane.

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

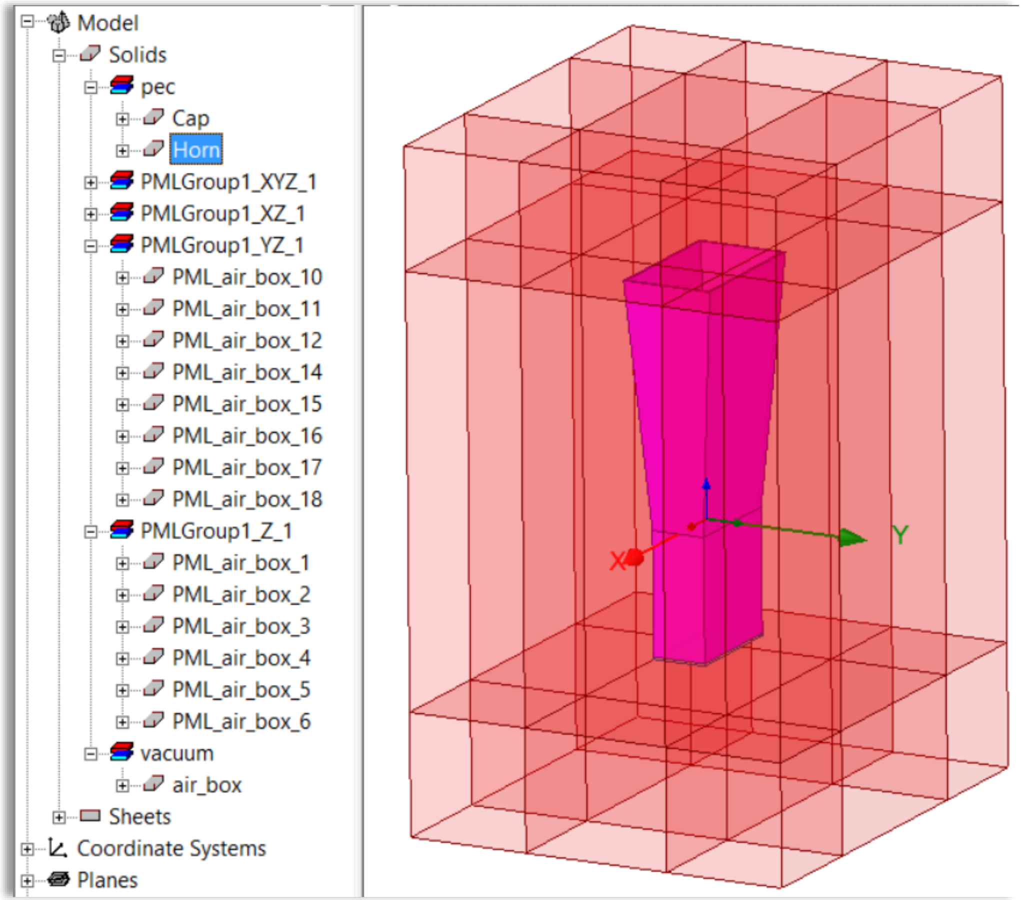
$$(\nabla\times \boldsymbol{E})_{tan} = jk_0\boldsymbol{E}_{tan} - \frac{j}{k_0}\nabla_{tan}\times(\nabla_{tan}\times \boldsymbol{E}_{tan}) + \frac{j}{k_0}\nabla_{tan}(\nabla_{tan}\bullet \boldsymbol{E}_{tan})$$



Boundary is λ/4 away from horn aperture in all directions



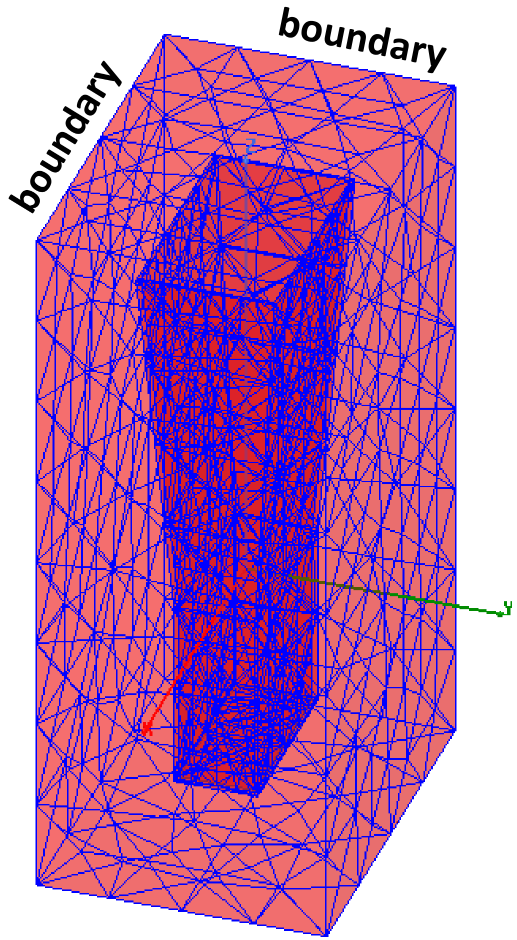
Open Models Allow Energy Out of the Computational Volume



Horn antenna with PML boundary

Two boundary choices for open models and antenna simulations are ***PML (perfectly matched layer)*** on the left, and ***Radiation (absorbing boundary condition)*** boundary on the right. Absorbing Boundary Condition (***ABC***) is good for broadside radiation.

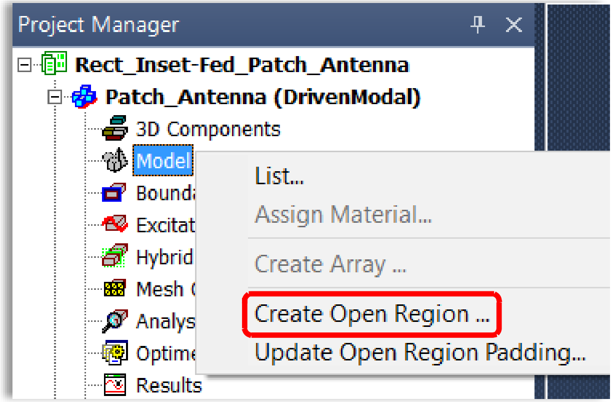
(Another good boundary choice ***FE-BI***, is not discussed here.)



Horn antenna with Radiation boundary

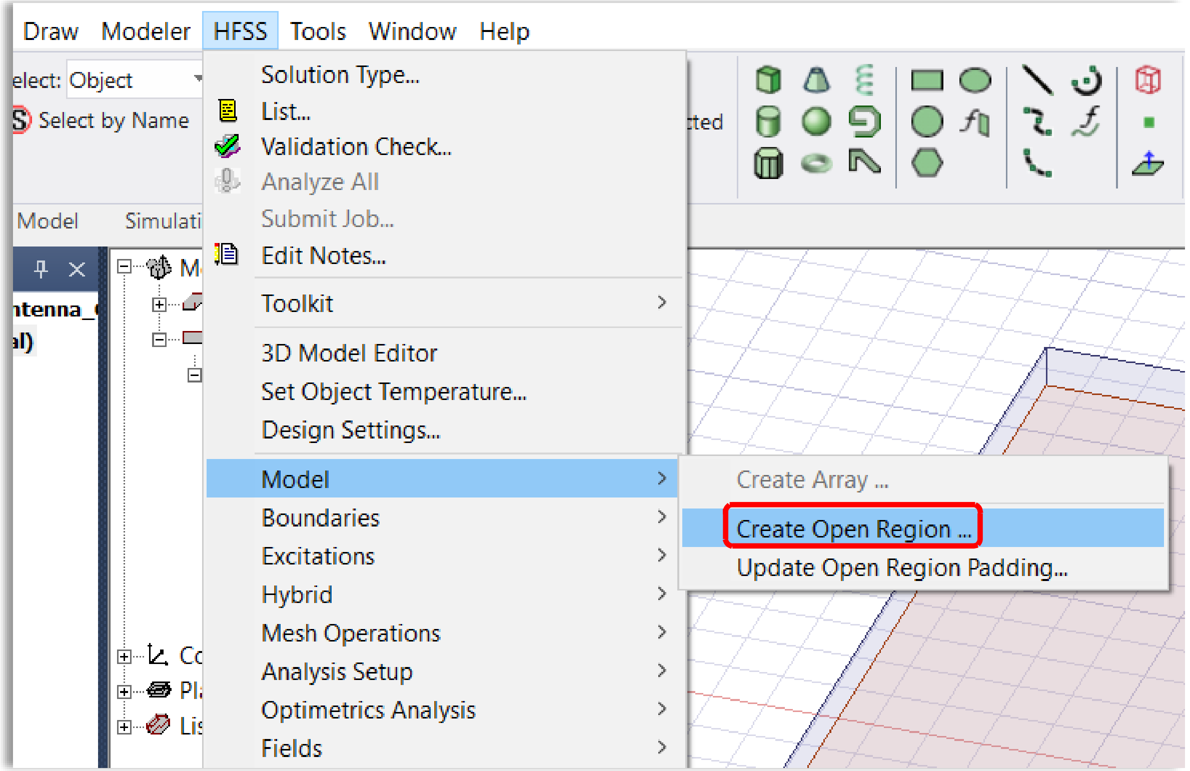
1. ***Create Open Region*** - an Automated Open Boundary Setup

In order to create a radiating boundary quickly in HFSS, use ***Create Open Region…***



In the ***Project Manager***, Right-click on ***Model*** and select ***Create Open Region…***

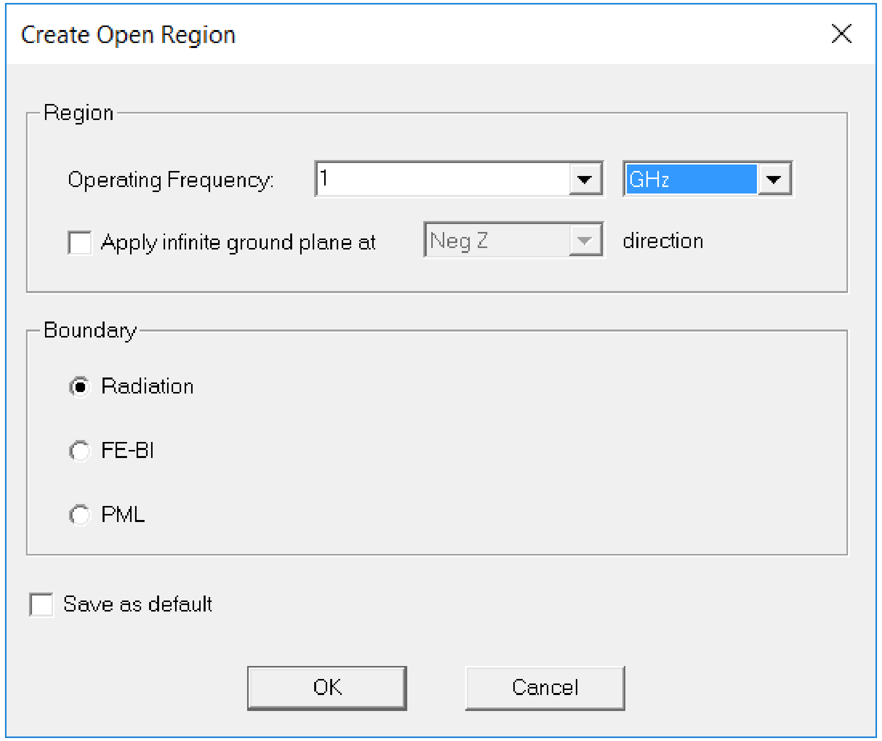
…OR…



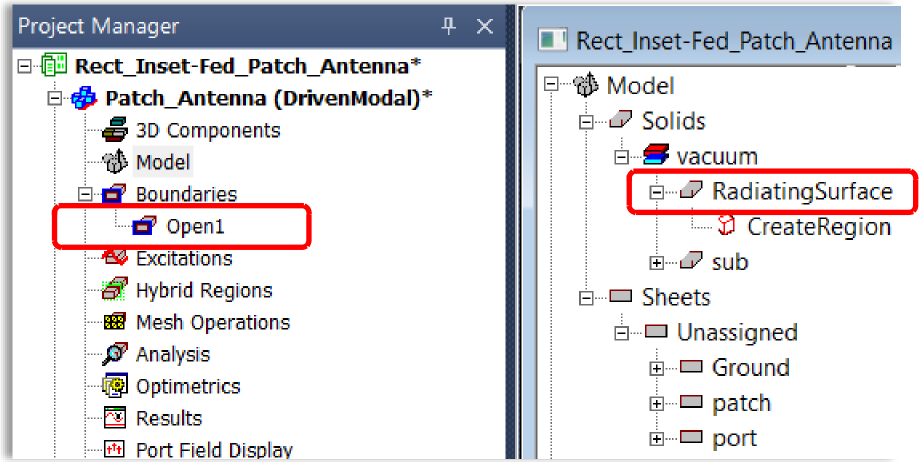
Select ***HFSS > Model > Create Open Region…***

…to bring up the ***Create Open Region…*** dialog box

Specify Frequency in the *Create Open Region* Dialog Box



The main specification for the ***Create Open Region…*** dialog box is the ***Operating Frequency***. This determines the size of the geometric box, the surface where the boundary is applied.

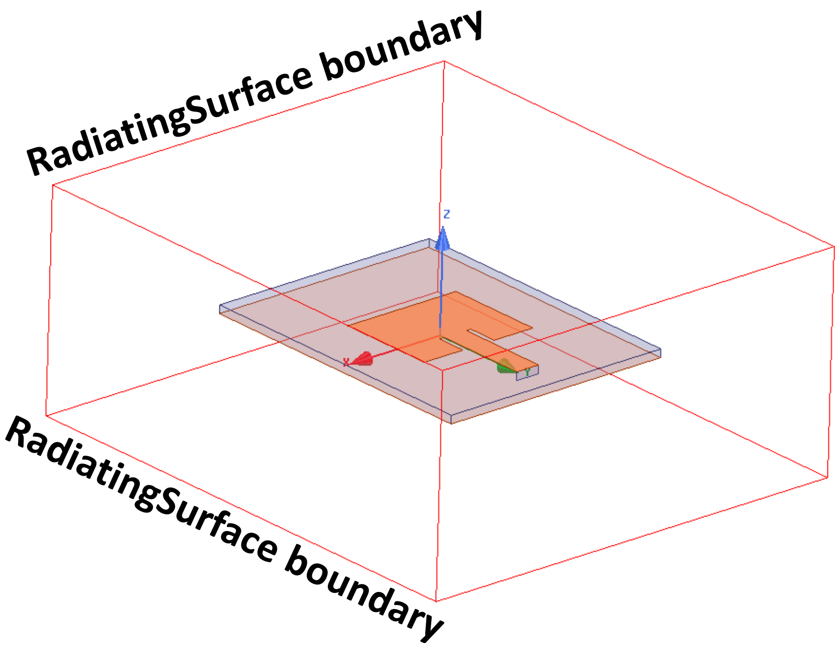


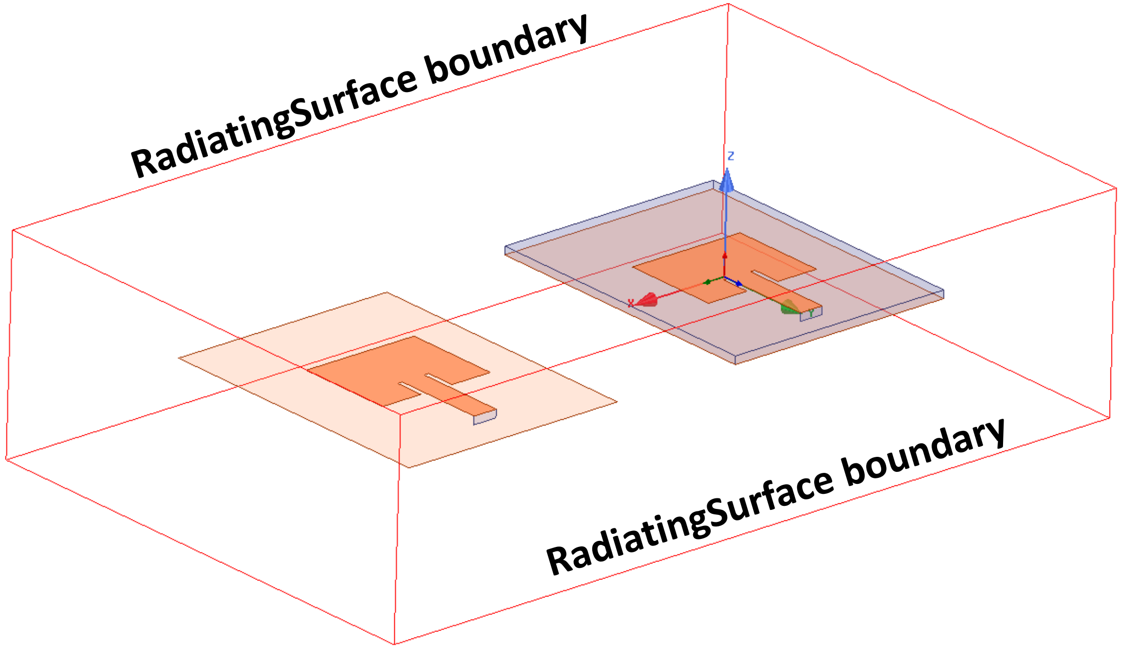
The ***Create Open Region…*** dialog box offers three choices for open boundaries, ***Radiation*** (ABC), ***PML*** (perfectly matched layer), and ***FE-BI*** (finite element - boundary integral) which is not discussed in this module. The lower the frequency, the smaller the box.

Create Open Region… places an entry ***Open1*** in the ***Project Manager*** under ***Boundaries***, and ***Create Open Region*** puts a ***RadiatingSurface*** in the ***3D Modeler Tree***.

Create Open Region **Automatically Adjusts Size**

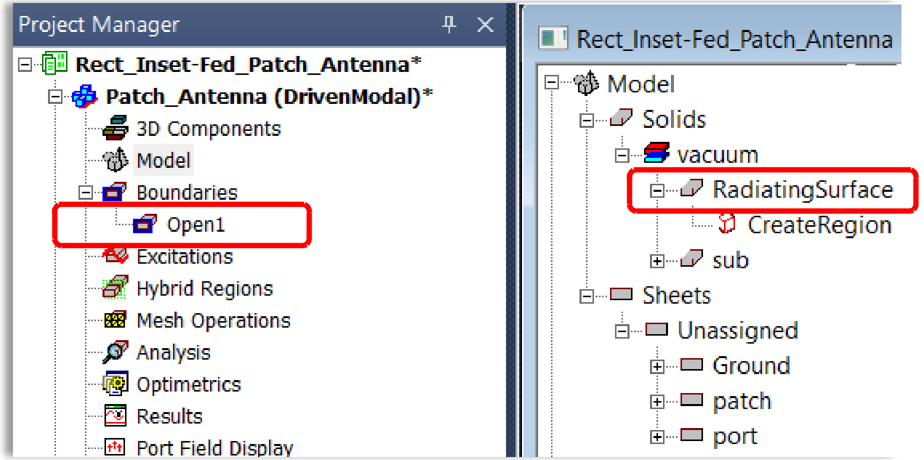
The size of the bounding box ***RadiatingSurface*** is automatically adjusted based on the size of the model structure. If the geometry gets bigger, or more objects get added, the boundary and the simulation space get larger.





The ***Operating Frequency*** specification determines the size of the geometric box (the ***Region…***the ***RadiatingSurface***) in the ***Create Open Region…*** operation. It automatically adjusts the size of the box.

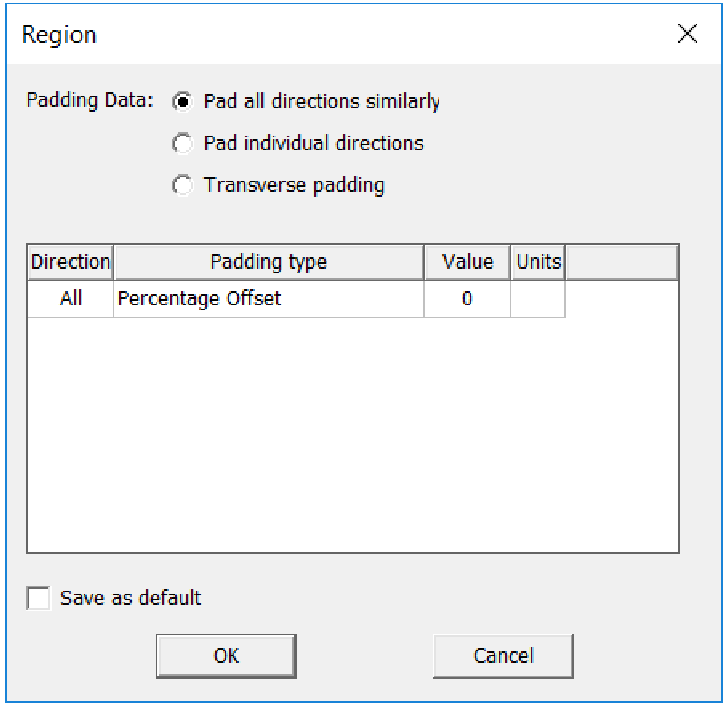
2. Create Just the Region in the Model with *Create Region*



There are two parts to an open boundary, the surface around the structure (the region) and the choice of boundary type. The ***Create Open Region…*** command specifies both.

Create Region… is an automated way to create the geometric region without assigning the boundary. The boundary gets assigned to the surface in a separate step. ***Region*** is found under ***Draw***.

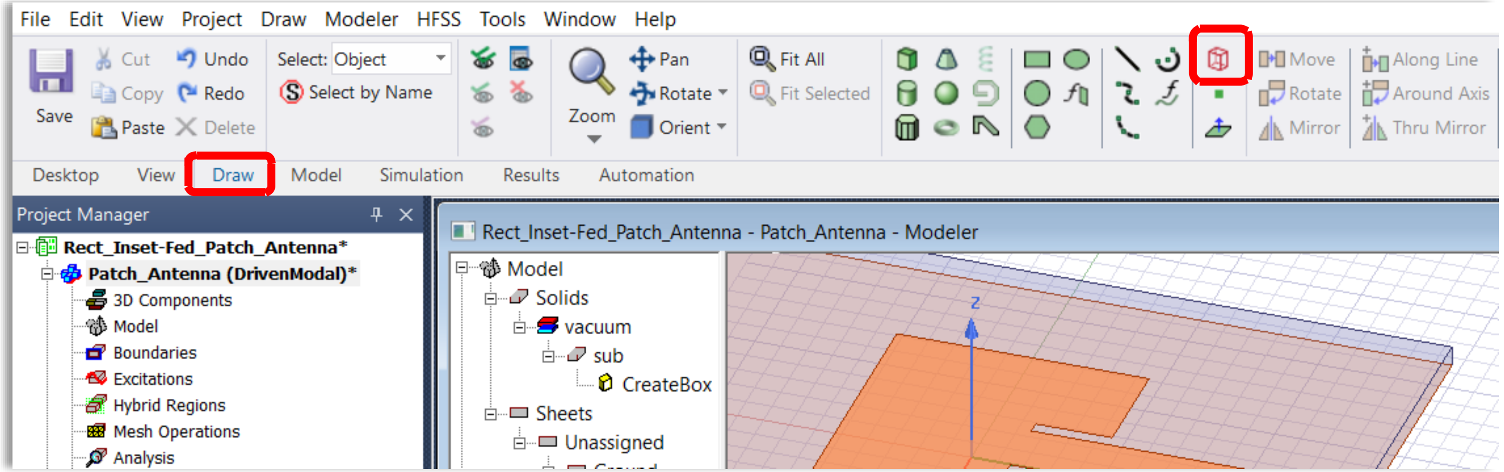
Draw > Region… specifies a ***Region*** geometrically, relative to the size of the structure, in terms of padding.

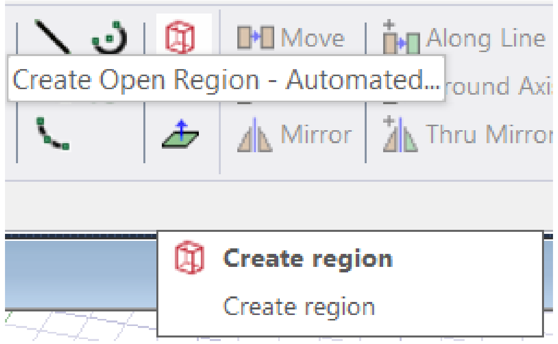


Create Region… brings up the Region dialog box.

How to Access *Create Region*

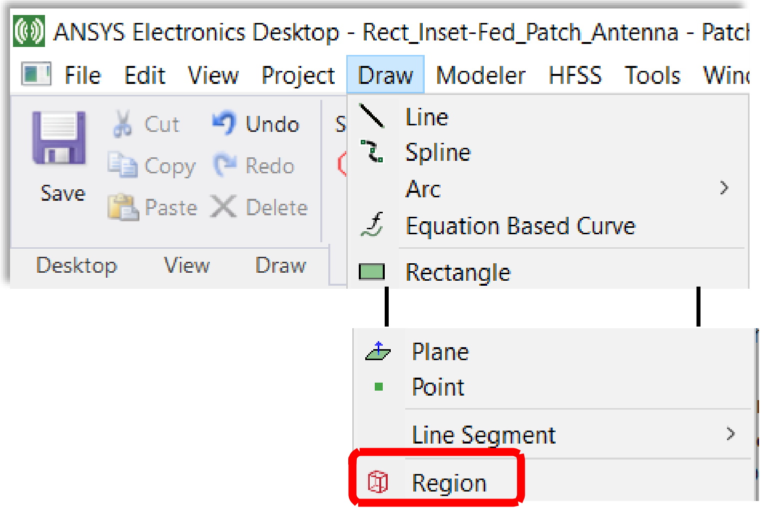
To select ***Create Region…*** select ***Draw*** in the ***Ribbon***, then click on the red box.



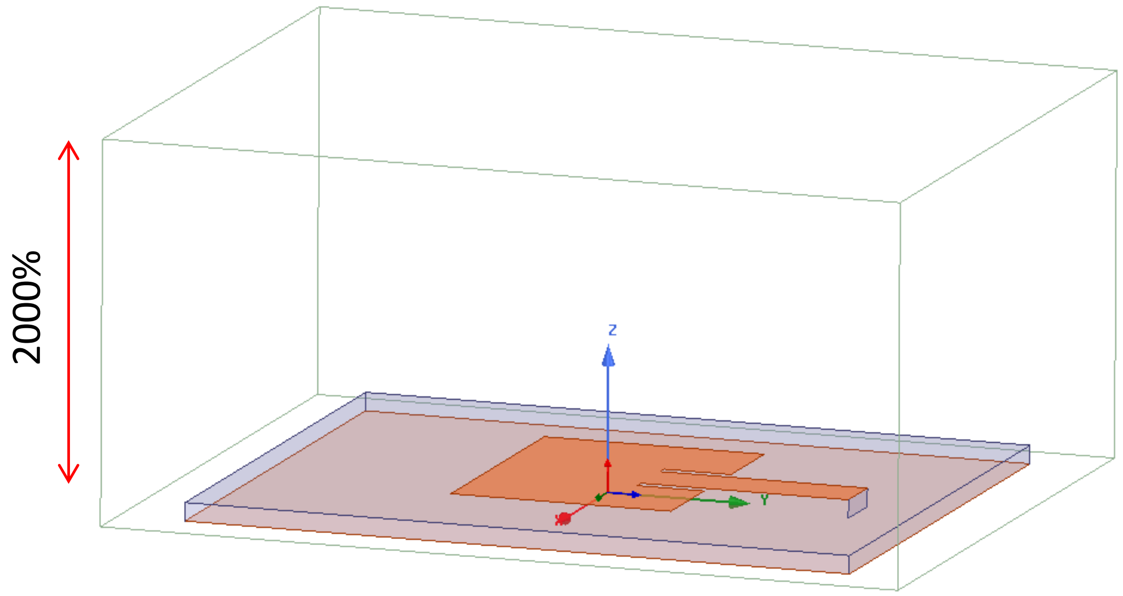


…OR…

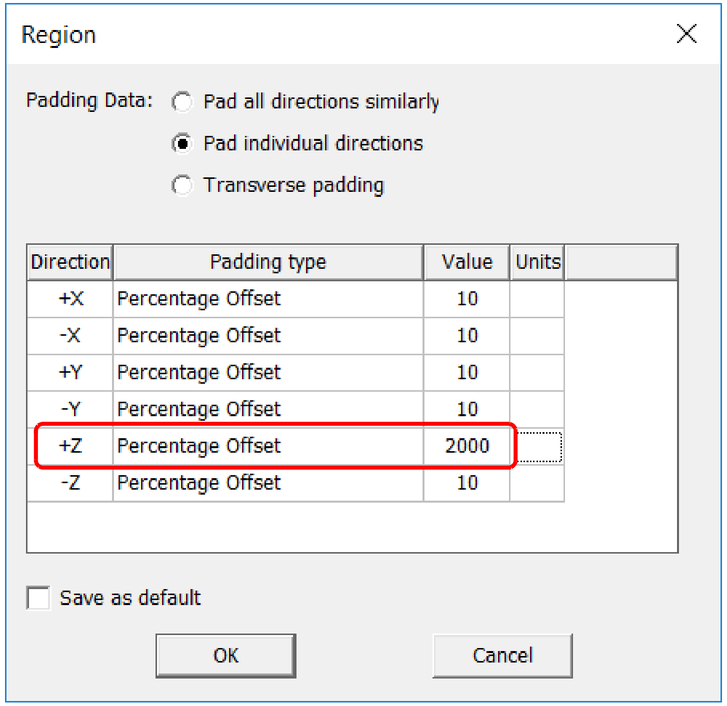
Draw > Region… selects ***Create Region*** from the top pull-down menus.



Percentage Offset in *Create Region*

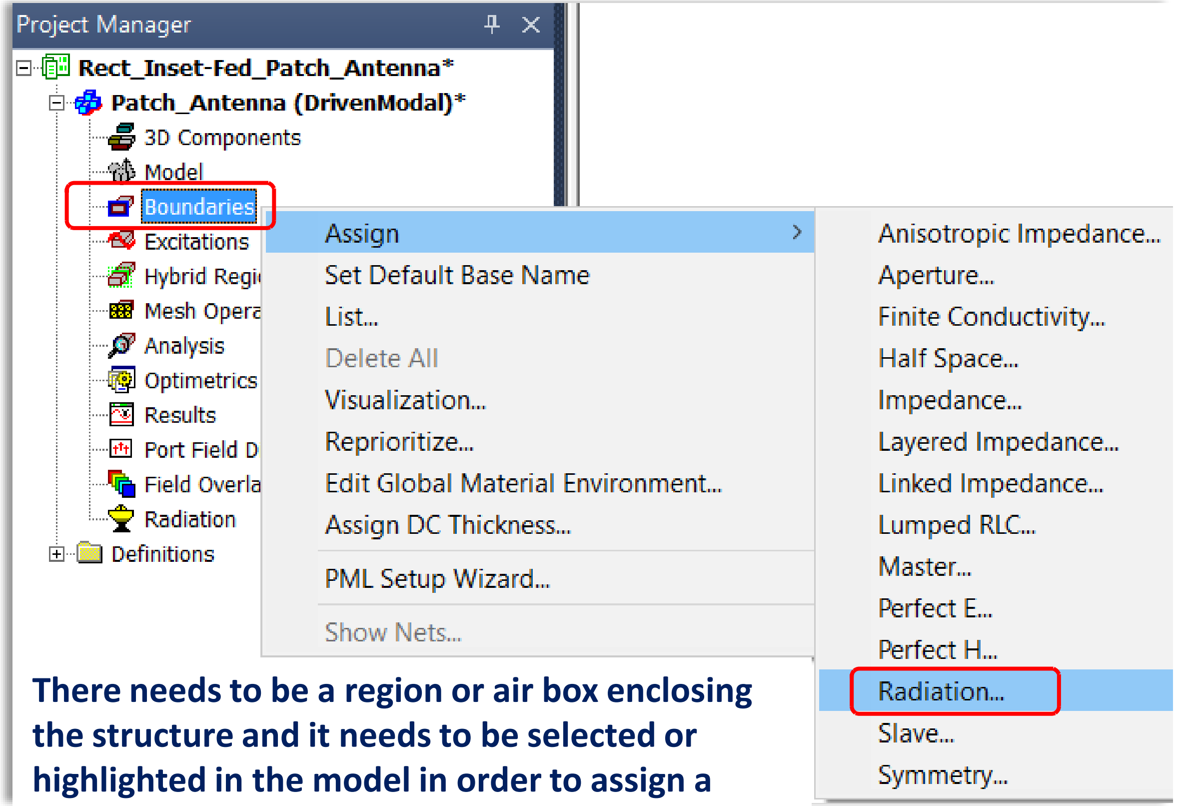


The height of the ***Region*** is set to 2000% of the structural dimension. All the other sides of the box are close around the structure; they are set at 10%.



Draw > Region… brings up the Region dialog box. Padding can be specified uniquely for six directions.

3. HFSS FEM Assigning Open Boundary Conditions (Manual Setup)



There needs to be a region or air box enclosing the structure and it needs to be selected or highlighted in the model in order to assign a boundary condition to it.

Once a ***Region*** or radiation surface is created and selected in the model, a boundary condition can be assigned to that surface.

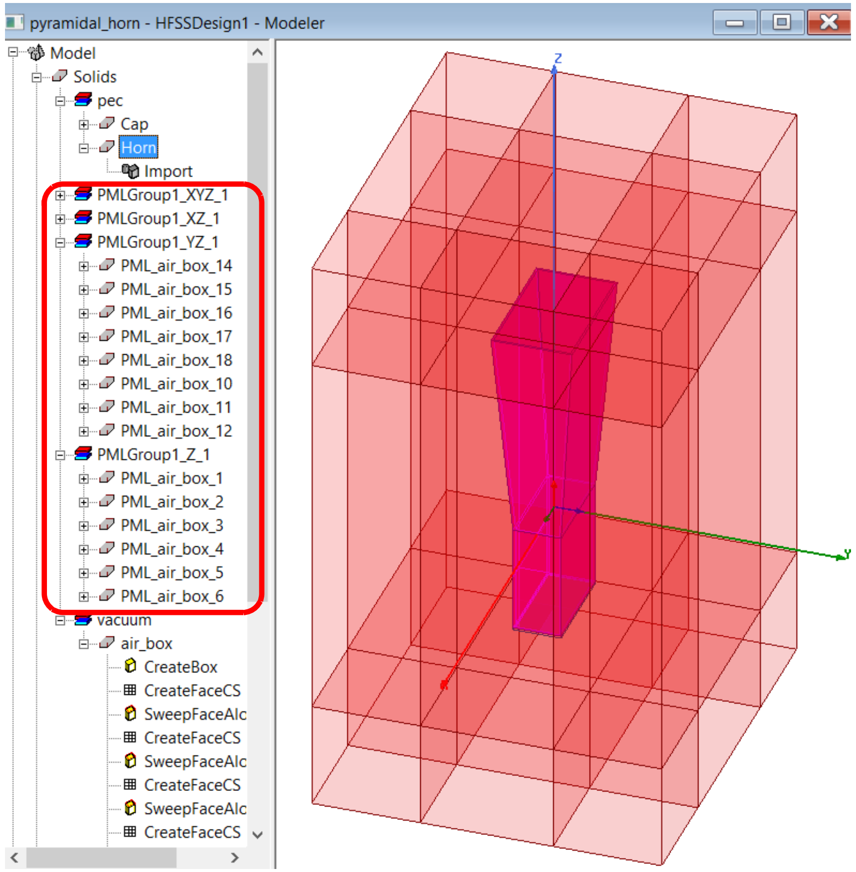
Boundary condition assignment can be accessed three ways in HFSS:

1. In the ***Project Manager***, right-click on ***Boundaries*** and select ***Assign*** and then choose a boundary condition.
2. Right-click anywhere in the model space and select ***Assign Boundary***.
3. From the top of the HFSS GUI, select ***HFSS > Boundaries > Assign.***

Radiation boundaries and perfectly matched layers only go on the outer most faces of the solution space.

Ansys

PML (Perfectly Matched Layer) is Not Strictly a Boundary Condition



A Perfectly Matched Layer (PML) creates structures, in the HFSS model, beyond just one bounding box. To set up a PML, look for the ***PML Setup Wizard***.

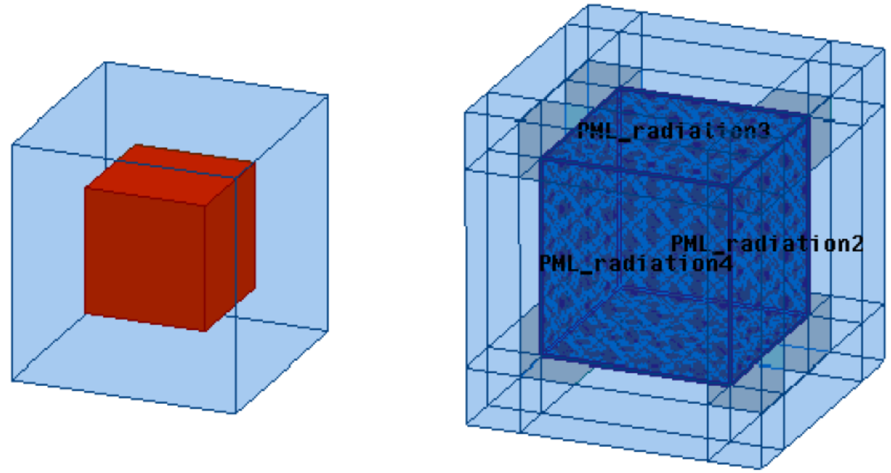


Figure 2-13 Radar cross section with *Perfectly Matched Layer Boundary*

Perfectly matched layers, though 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 (PMLs) are the preferred boundary when simulating antenna models and they are more appropriate than radiation boundaries for antenna simulations. PMLs can reduce the solution volume since they can be positioned at a distance 1/10 times the wavelength close to any radiating structure. For modeling antennas, it is recommend to keep the PML a quarter of a wavelength away from any radiating structure.

Perfectly matched boundaries are automatically generated with the aid of the PML Wizard. This wizard guides you through the creation of the PML objects/materials.

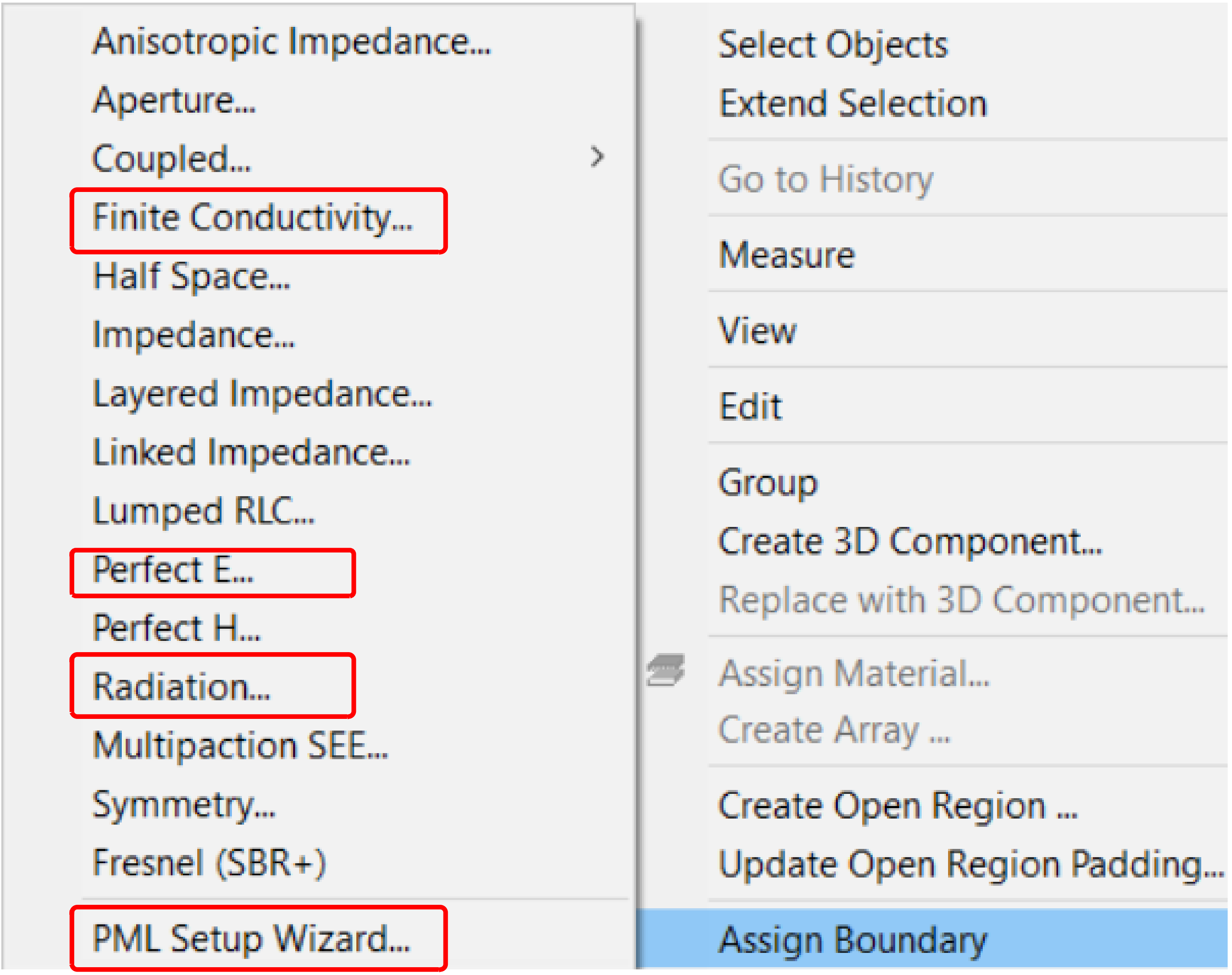
This information comes from HFSS Help ***Introduction to HFSS.pdf*** section on ***Perfectly Matched Layer***.

HFSS FEM Boundary Types Selection from ***Assign Boundary***

This HFSS FEM Getting Started course describes ***Perfect E*** ("***perfE***") also called perfect electrical conductor (PEC), ***Radiation (ABC),*** and ***Perfectly Matched Layer (PML).***

There are other boundary types available in HFSS FEM. This list is from the Assign Boundary command in the ***3D Modeler***.

The intermediate level course ***HFSS 3D Components, Boundary Conditions, Ports and Mesh*** also discusses boundary conditions.



Perfect Electric Conductor

Assigning a perfect electric boundary condition on a surface of an object causes the tangential component of the electric field to be zero. This type of boundary models a perfectly conducting surface in a structure that forces the electric field to be normal to the surface.

Since no electromagnetic field exists inside 3D objects with pec material, HFSS treats the surfaces of these objects as PEC boundaries. You can define PEC boundary on 2D sheet objects to represent lossless conductors such as transmission line traces or patch antenna elements. When assigning a PEC boundary on a surface that represents the “ground” plane of a radiating structure, select the Infinite Ground Plane option if you want to model an infinite ground plane.

In HFSS, PEC is a default boundary condition on all *outer* surfaces of the computational domain.

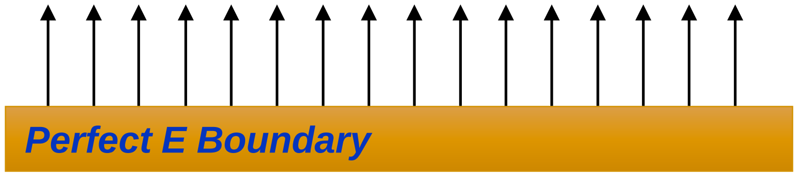
The PEC boundary can also be used to create a symmetry plane in a model. In this type of symmetry the E field components are normal to the plane. In the dielectric resonator antenna model shown below, the bottom face of the air volume object is defined as a perfect E boundary.

PEC (perfect electrical conductor) boundaries don't allow energy from the simulation model to get out of the simulation space.

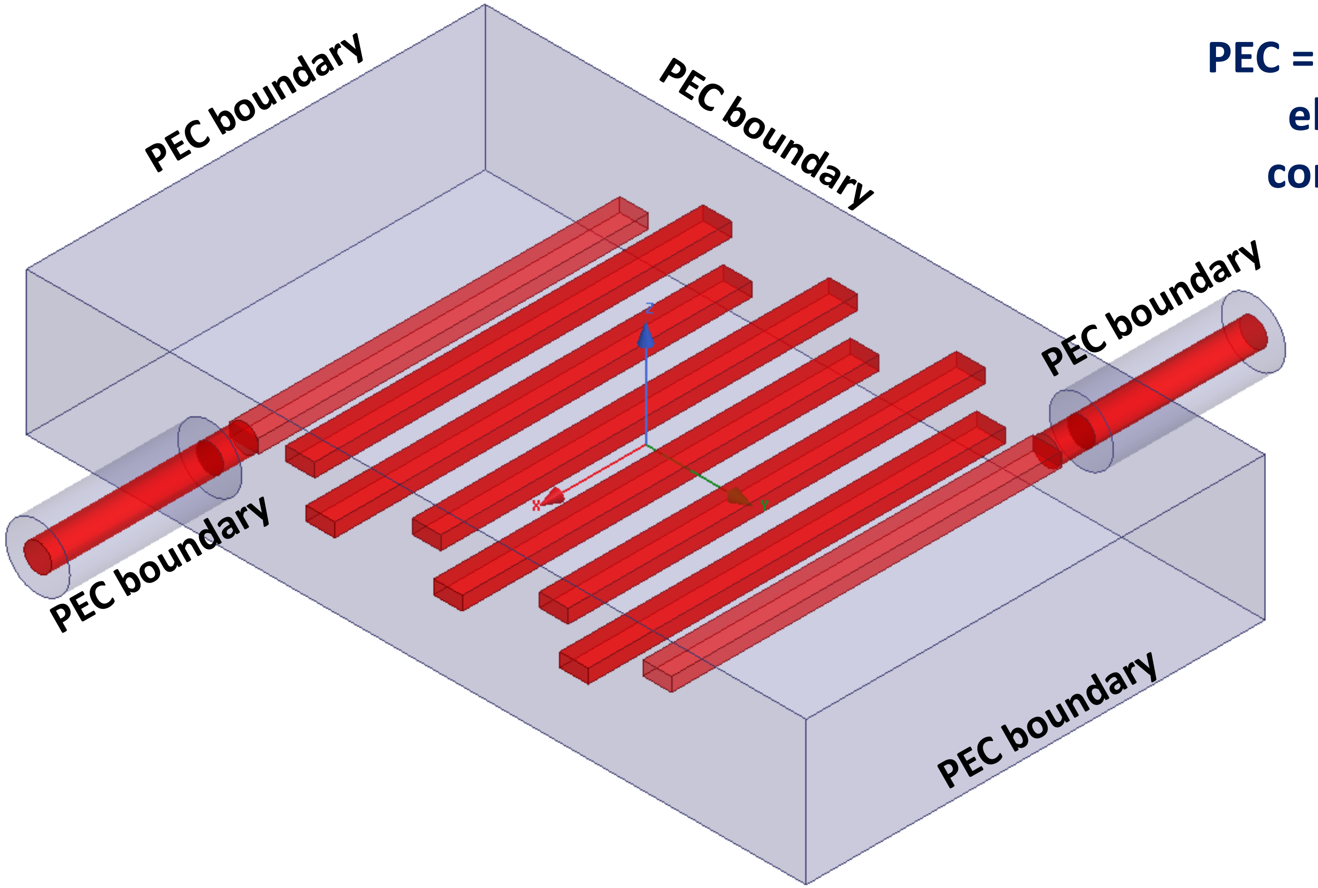
PEC is useful for creating closed models.

This description comes from the HFSS Help ***Introduction to HFSS.pdf*** section on ***HFSS Boundaries***.

PEC (perfect electrical conductor) only allows E-field perpendicular to the surface. The tangential electric field, parallel to the surface, must be zero.



Closed Model with Default PEC on the Outer Boundaries



PEC = perfect electrical conductor

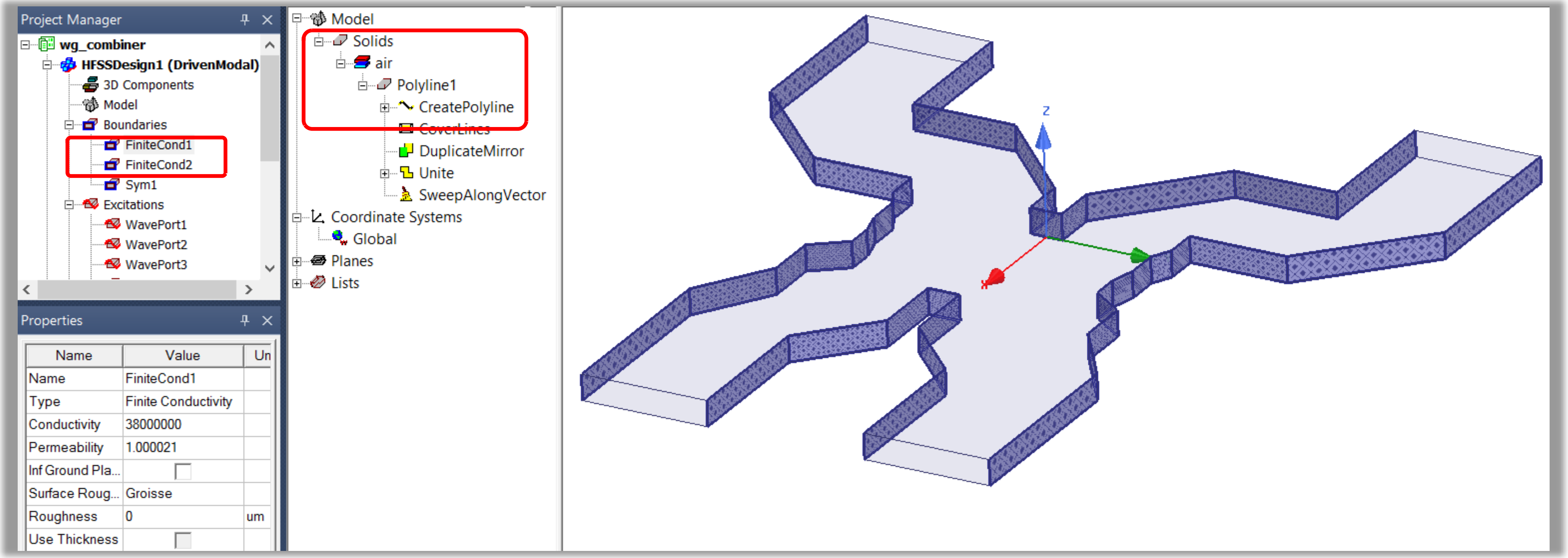
For the outer boundary of the simulation space, HFSS does not always require a boundary to be defined. If no boundary is defined, then HFSS applies perfect electric conductor (PEC) as a default boundary condition around the computational volume.

In the band pass filter example, and several others, there are vacuum objects which get a default PEC boundary applied to them.

In this band pass filter, the rectangular volume and the outer cylinders of the feeds are air or vacuum. By default, they are all covered with PEC boundary.

Boundaries Can Function Like Materials

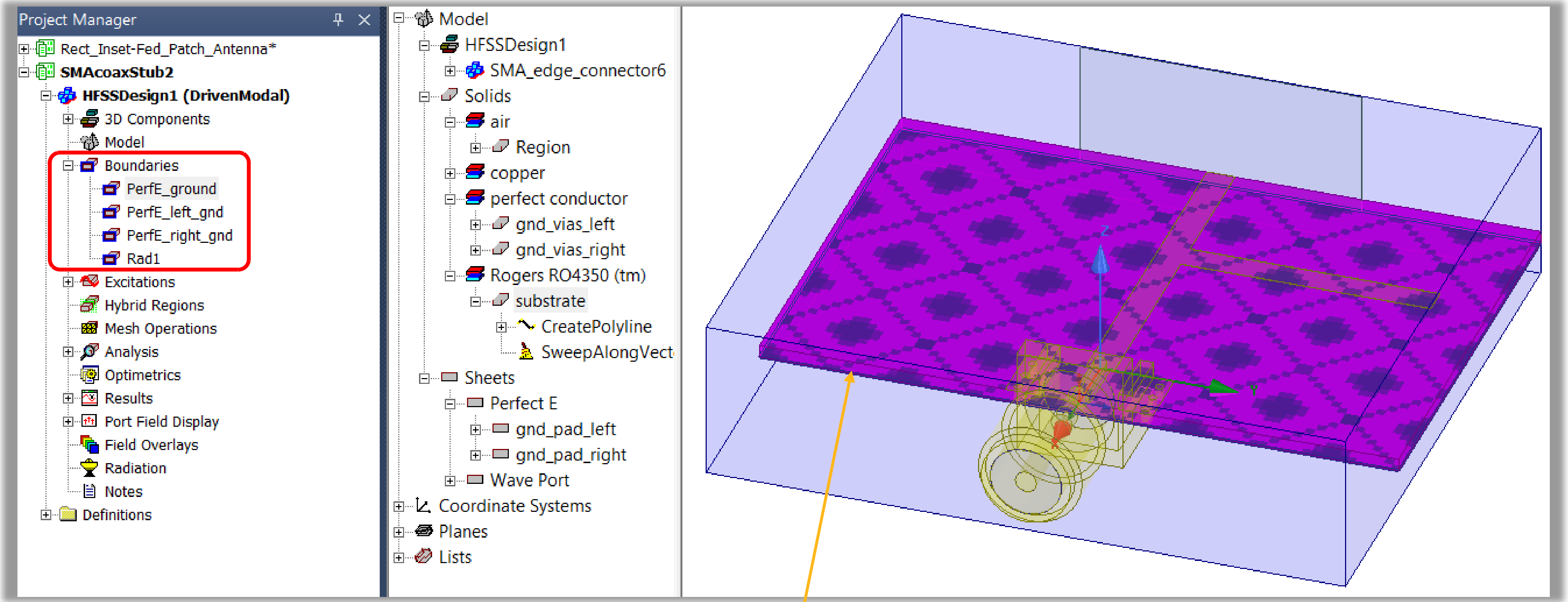
We can sometimes think of boundaries like materials for surfaces and sheets. In addition to PEC "perfE" (perfect electric conductor), we can use finite conductivity boundaries in HFSS FEM. There is no other material assigned for this waveguide combiner's metal surface.



In the ***Project Manager***, under ***Boundaries***, we see two different finite conductivity boundaries assigned. We can also see the conductivity in the ***Properties*** menu in the lower left.

perfE (PEC) Boundaries versus Good Conducting Materials

This SMA microstrip stub project has an open radiation boundary around the outside of the structure. In addition, this project uses ***perfE*** in three other places like a material.



There are four different boundary assignments.

The substrate has a PEC ground plane created by assigning perfE boundary condition to one face of the substrate surface.

Ansys

End of Presentation