Antenna Theory and Design

Associate Professor
WANG Junjun 王珺珺
School of Electronic and Information Engineering
Beihang University
wangjunjun@buaa.edu.cn
13426405497
New Main Building F1025

Chapter8

High Frequency Structure——HFSS

What is HFSS?

- HFSS is a high-performance full-wave electromagnetic(EM) field simulator for arbitrary 3D volumetric passive device modeling that takes advantage of the familiar Microsoft Windows graphical user interface.

- It integrates simulation , visualization, solid modeling, and automation in an easy-to-learn environment where solutions to your 3D EM problems are quickly and accurately obtained.

- Ansoft HFSS employs the Finite Element Method(FEM), adaptive meshing, and brilliant graphics to give you unparalleled performance and insight to all of your 3D EM problems.

- Ansoft HFSS can be used to calculate parameters such as S Parameters, Resonant Frequency, and Fields. Typical uses include:

Contents

We will discuss some basic concepts and terminology used throughout the Ansoft HFSS application. It provides an overview of the following topics:

- Introduction
  - Converting Older Files
  - Ansoft Terms
  - Getting Help
  - Overview
- Fundamentals
  - Opening a Design
  - Setting Model Type
- Parametric Model Creation
  - Boundary Conditions
  - Excitations
- Analysis Setup
- Data Reporting

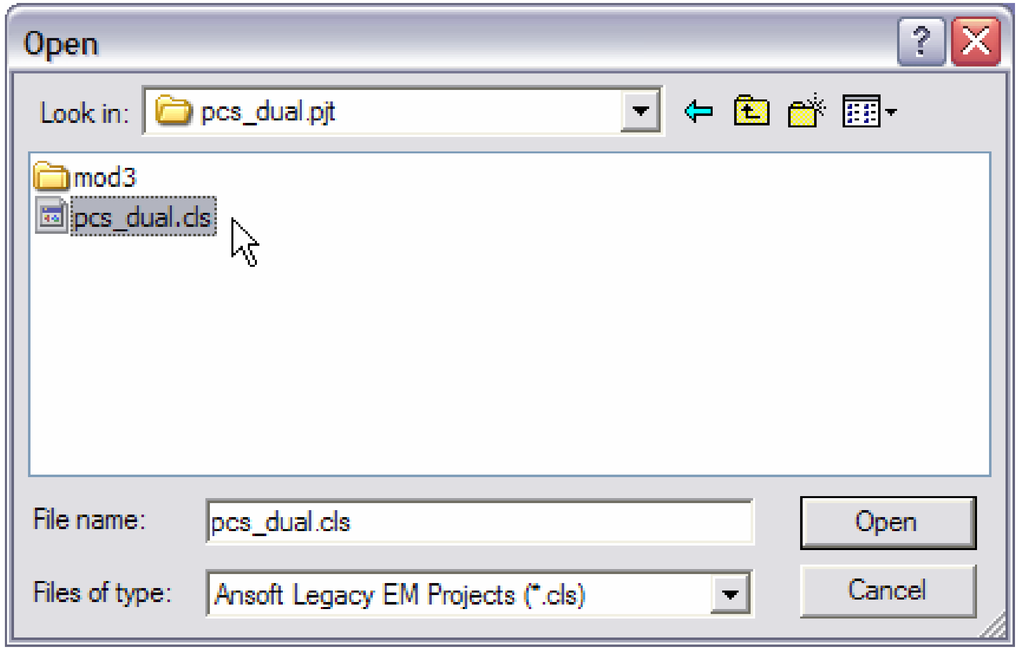
8.1 Introduction
8.1.1 converting older files

- ◆ Starting Ansoft HFSS
- ◆ Converting Older HFSS file to HFSS v12
- ➢ To access HFSS projects in an earlier version

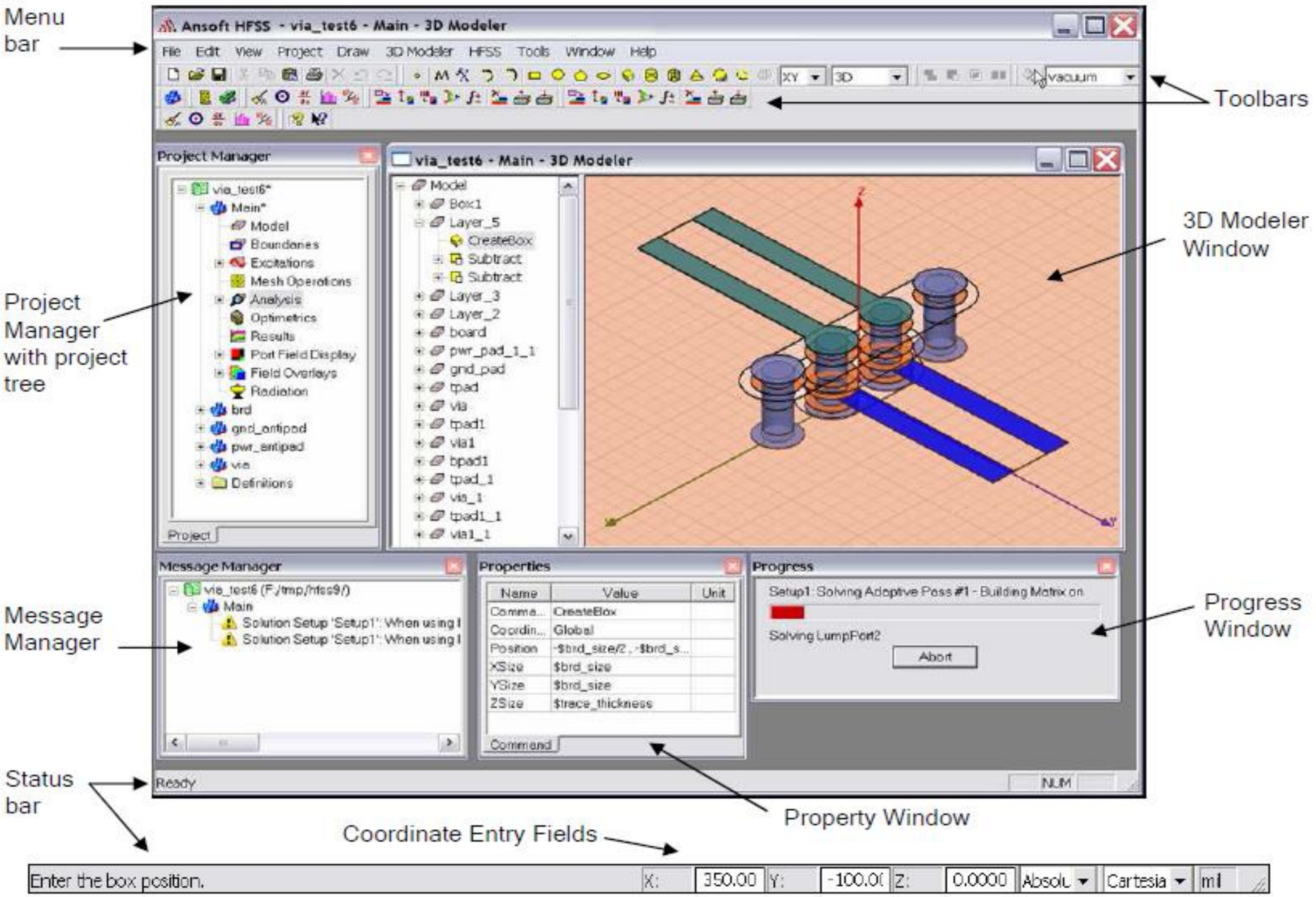
① Select the menu item File > Open

② Open dialog:

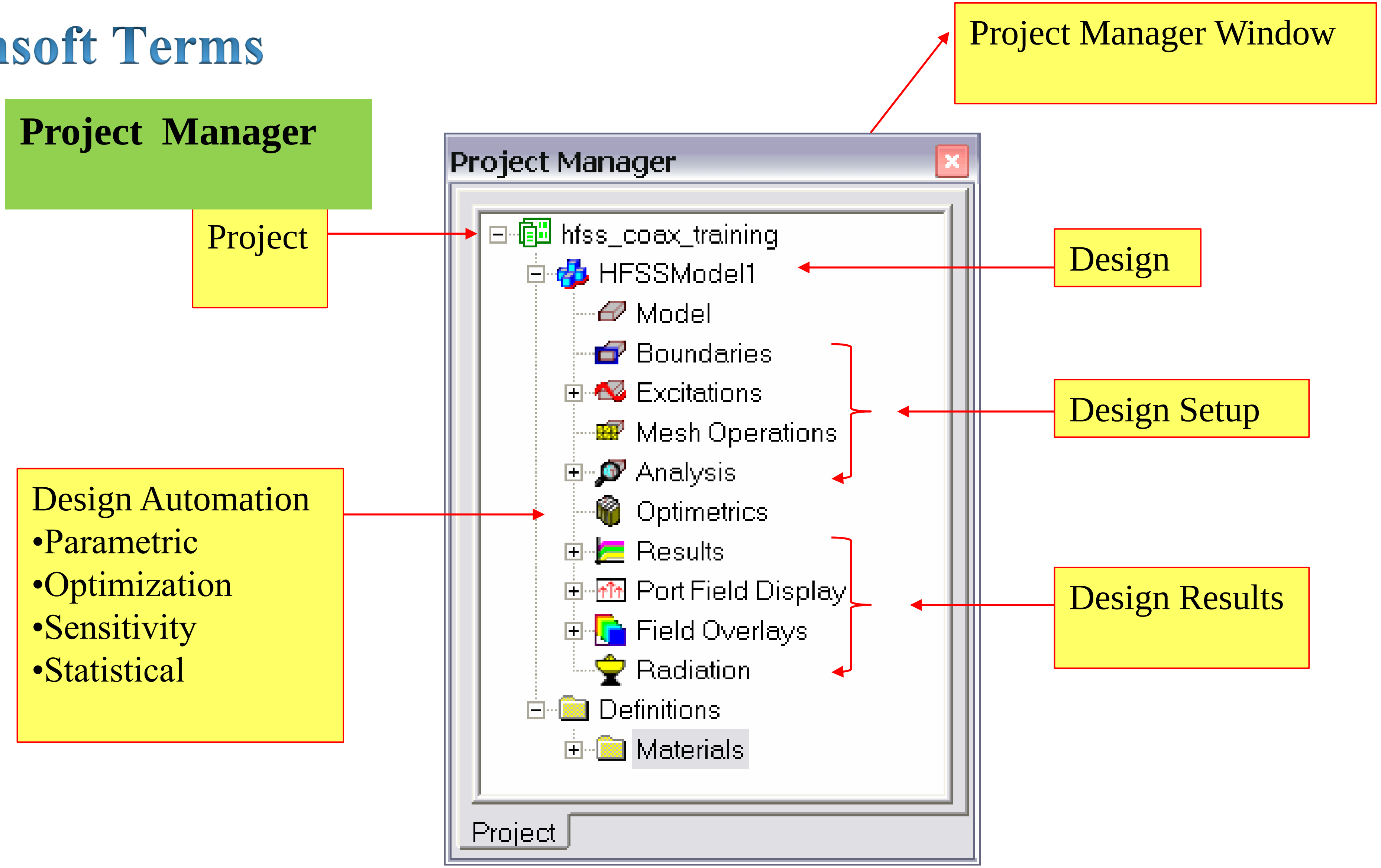
A. Files of Type: Ansoft Legacy EM Projects (.cls)
B. Browse to the existing project and select the .cls file
C. Click the Open button



8.1 Introduction
8.1.2 Ansoft Terms

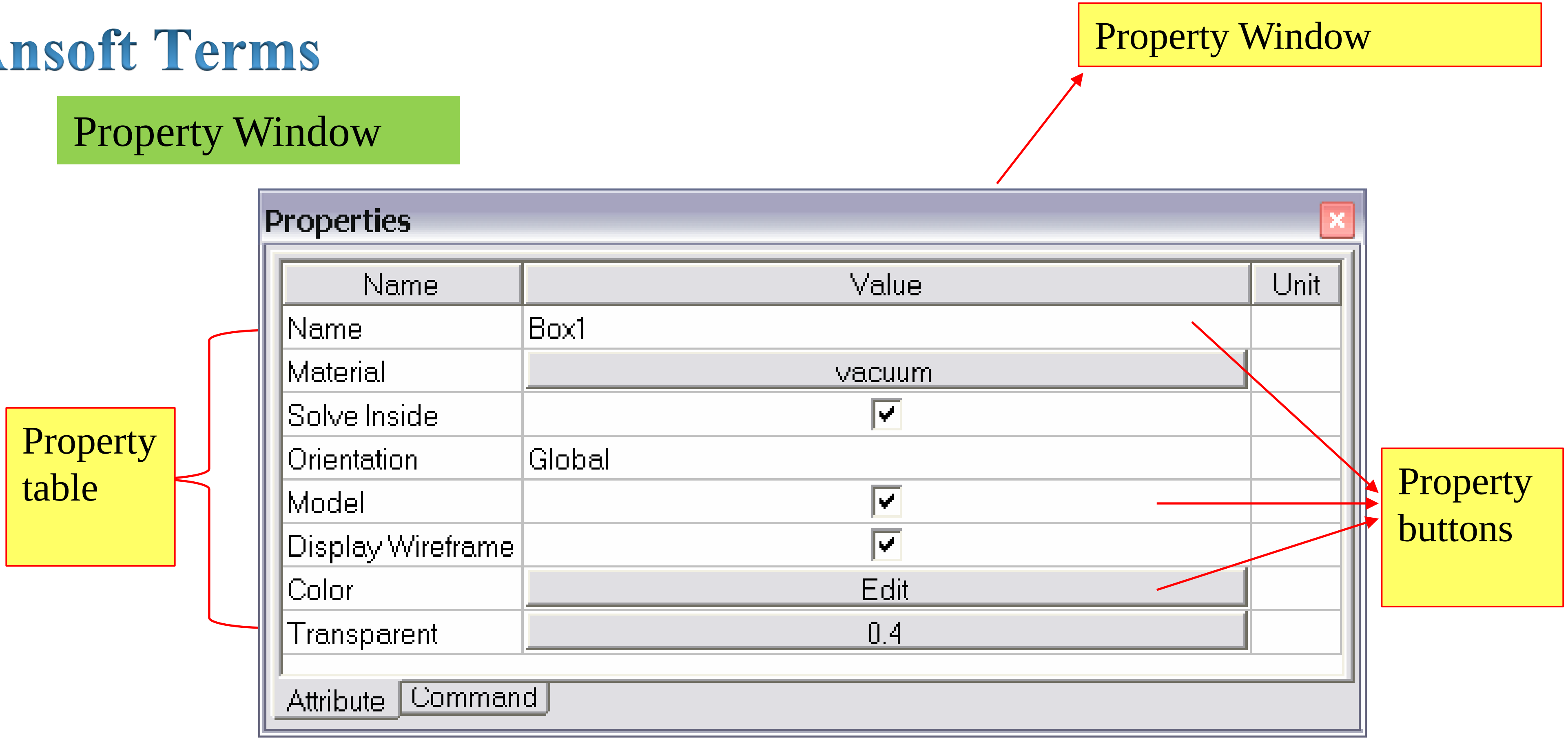


8.1Introduction
8.1.2Ansoft Terms

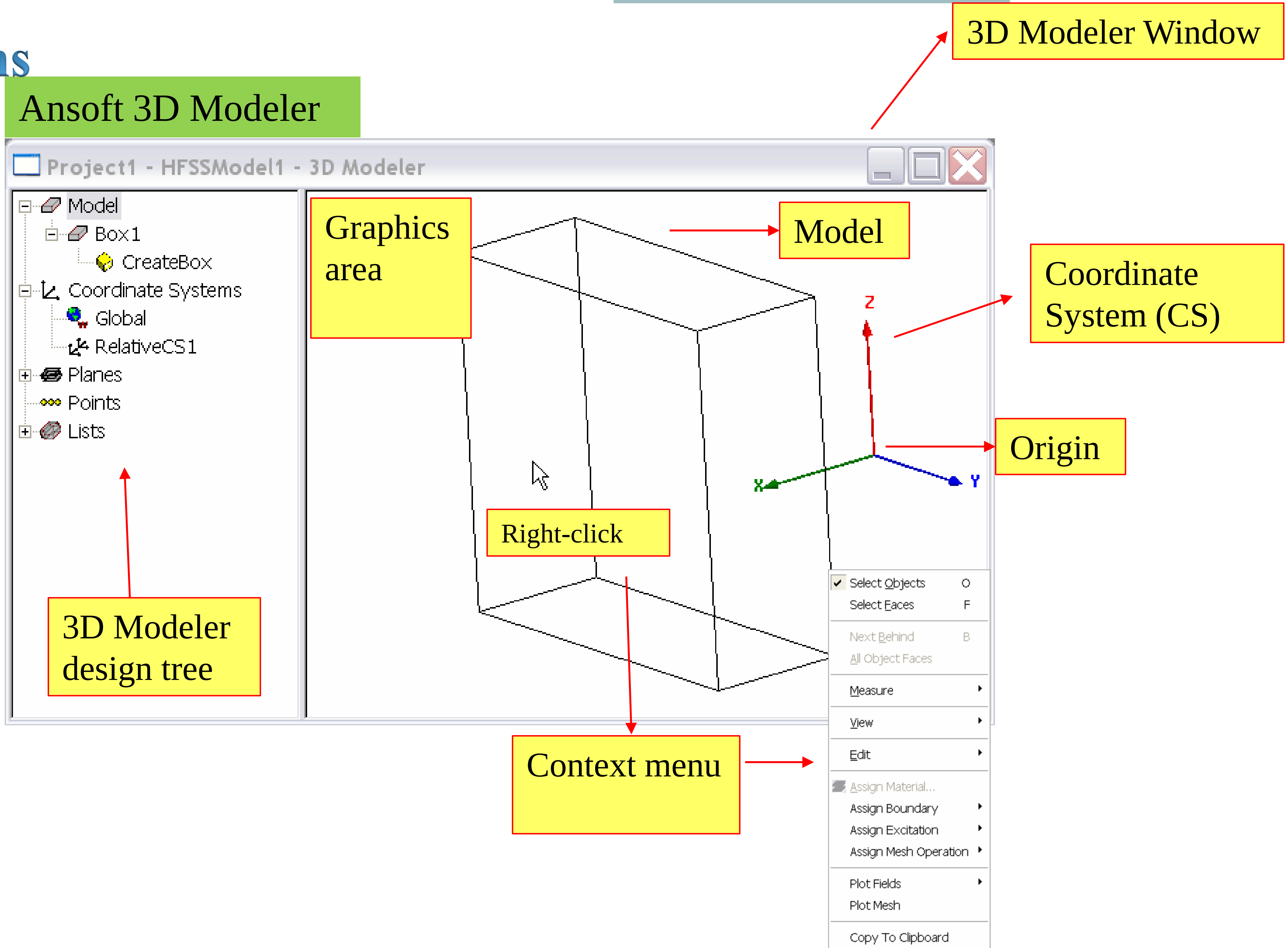


8.1Introduction
8.1.2Ansoft Terms

Property Window

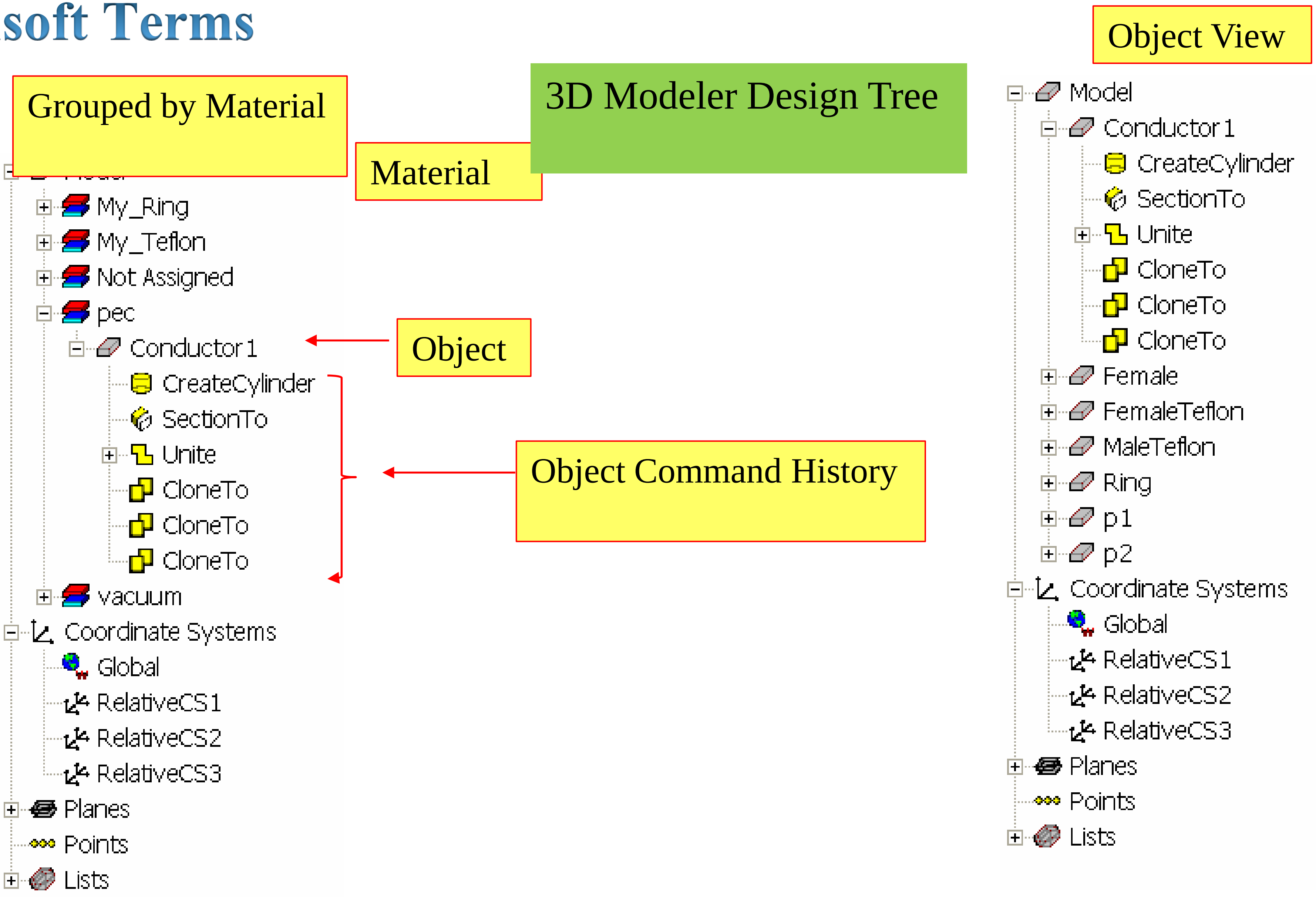


8.1Introduction
8.1.2Ansoft Terms



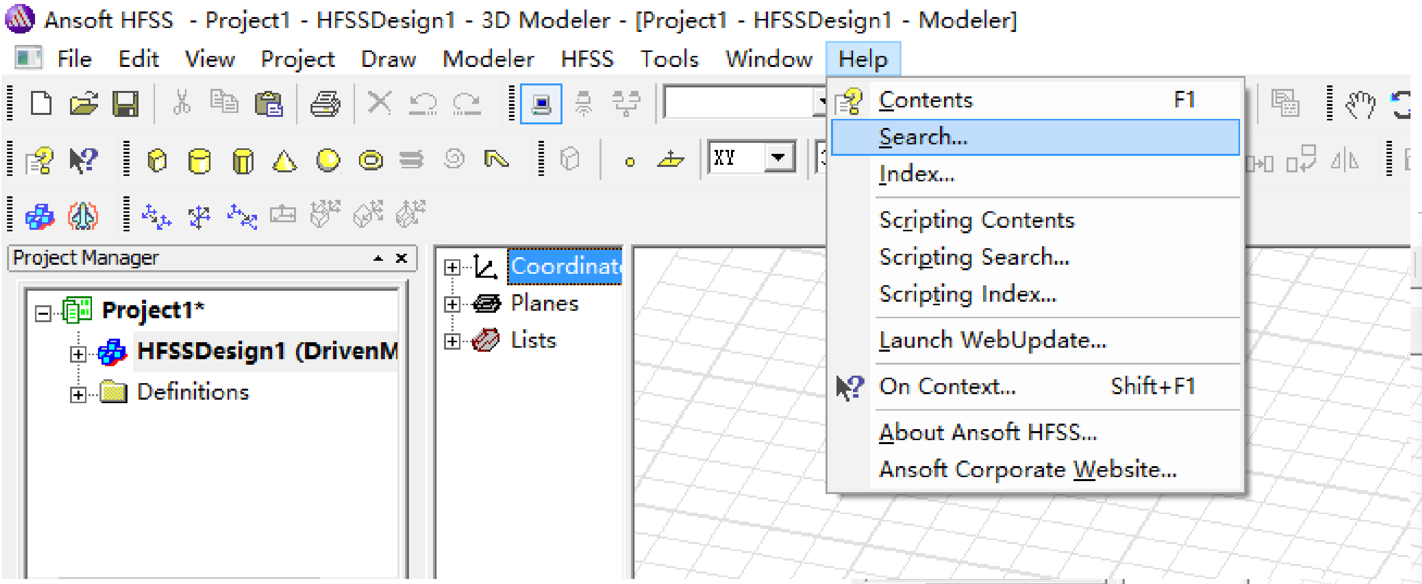
8.1Introduction

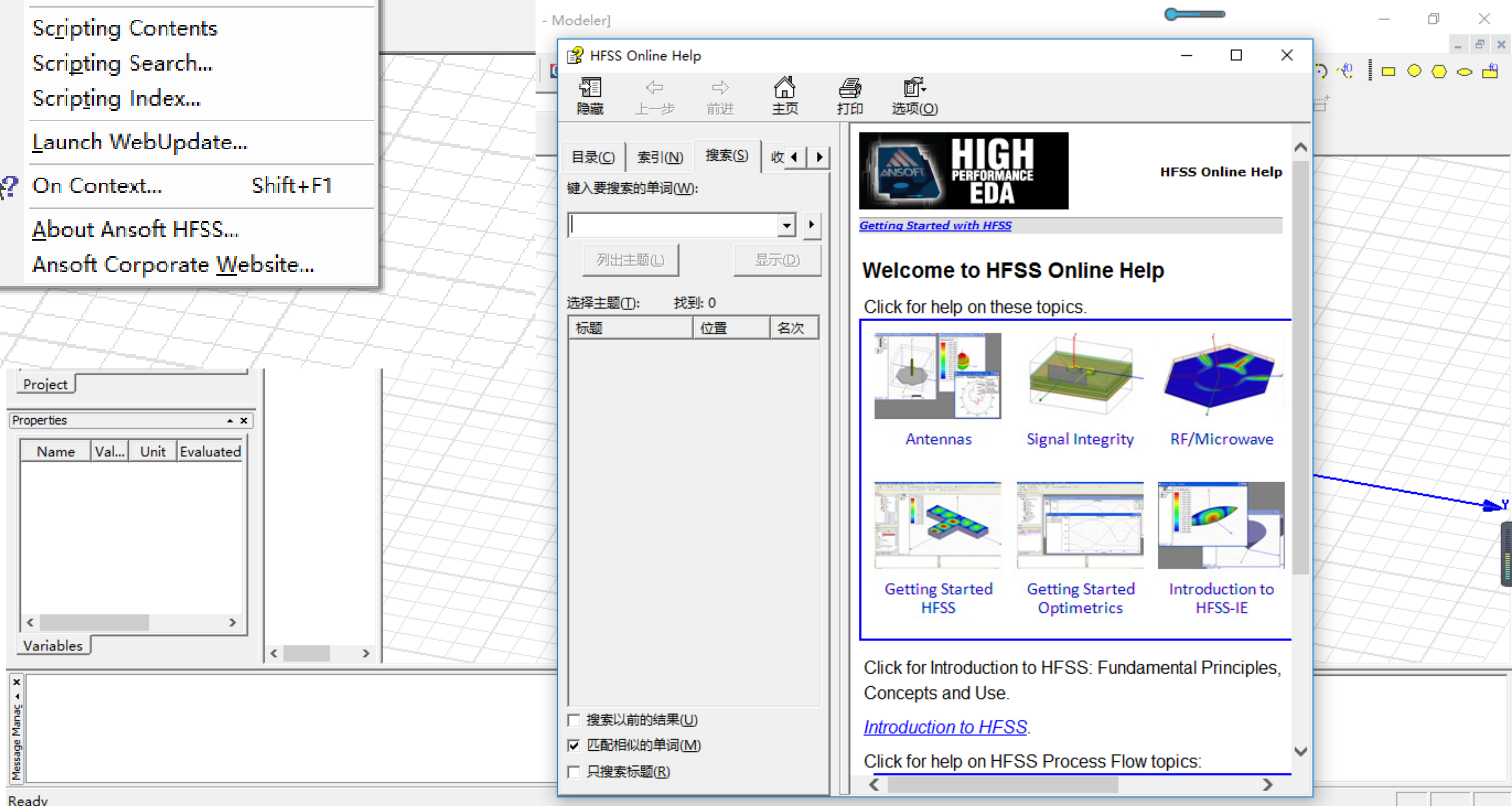
8.1.2Ansoft Terms



8.1 Introduction

8.1.3 Getting help



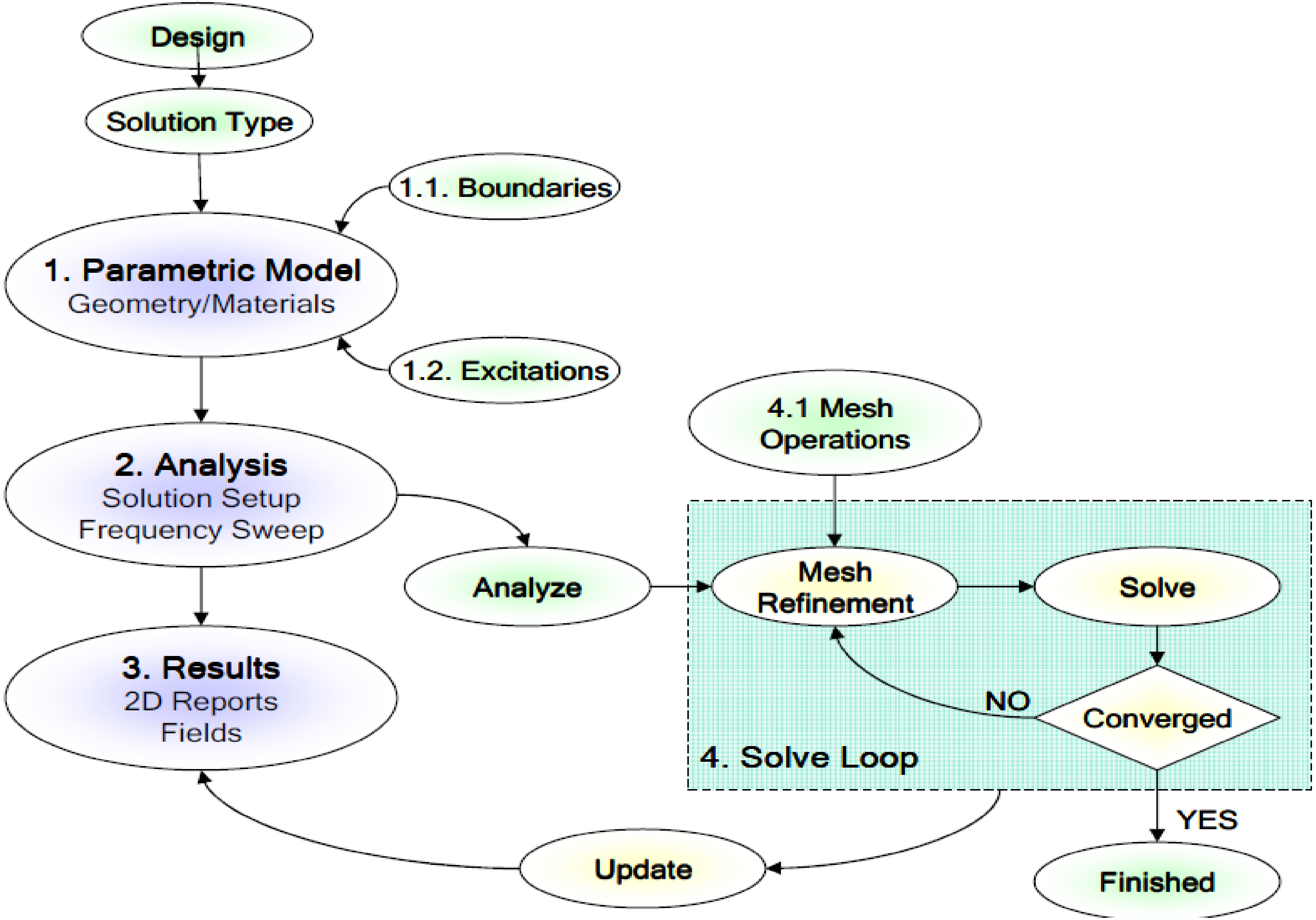


8.1 Introduction
8.1.4 Overview

◆Ansoft HFSS Desktop：

The Ansoft HFSS Desktop provides an intuitive, easy-to-use interface for developing passive RF device models. Creating designs, involves the following:

- 1. **Parametric Model Generation** – creating the geometry, boundaries and excitations
- 2. **Analysis Setup** – defining solution setup and frequency sweeps
- 3. **Results** – creating 2D reports and field plots
- 4. **Solve Loop -** the solution process is fully automated


Design
Solution Type
1.1. Boundaries
1. Parametric Model
Geometry/Materials
1.2. Excitations
4.1 Mesh Operations
2. Analysis
Solution Setup
Frequency Sweep
Analyze
Mesh Refinement
Solve
NO
Converged
4. Solve Loop
3. Results
2D Reports
Fields
YES
Update
Finished

8.2 Fundamentals

8.2.1 Opening a Design

◆ **Opening a New project**

➢ **To open a new project:**

1. In an Ansoft HFSS window, select the menu item File > New.
2. Select the menu Project > Insert HFSS Design.

◆ **Opening an Existing HFSS project**

➢ **To open an existing project**:

1. In an Ansoft HFSS window,select the menu File > Open.Use the Open dialog to select the project.
2. Click Open to open the project

8.2 Fundamentals
8.2.2 Set Solution Type

In the design of antenna,we often choose **Driven Modal** or **Driven Terminal**

- **Set Solution Type**

  - 1. **Driven Modal** - calculates the modal-based S-parameters. The S-matrix solutions will be expressed in terms of the incident and reflected powers of waveguide modes.

  - 2. **Driven Terminal** - calculates the terminal-based S-parameters of multiconductor transmission line ports. The S-matrix solutions will be expressed in terms of terminal voltages and currents.

  - 3. **Eignemode** – calculate the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies.

8.2 Fundamentals
8.2.2 Set Solution Type

◆ To set the solution type:

1. Select the menu item

   HFSS > Solution Type

2. Solution Type Window:

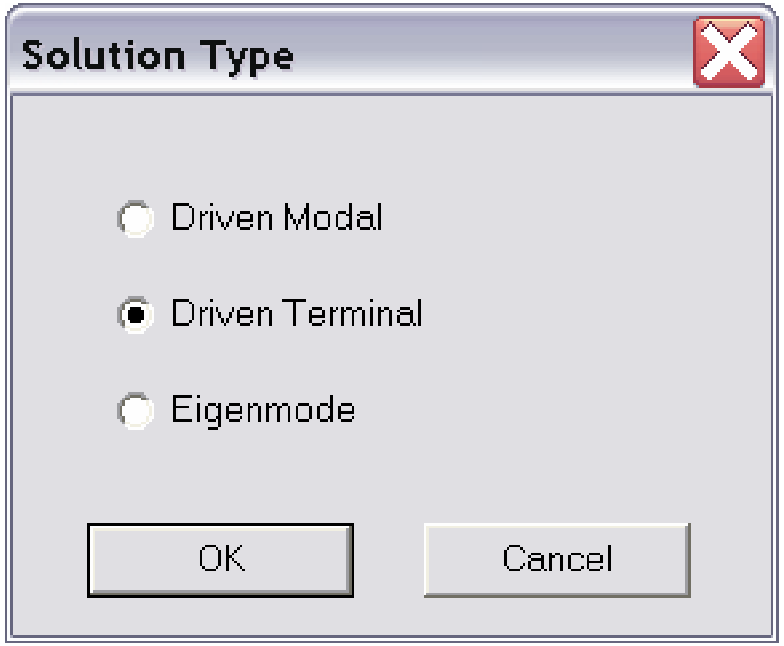
① Choose one of the following:

Driven Modal

Driven Terminal

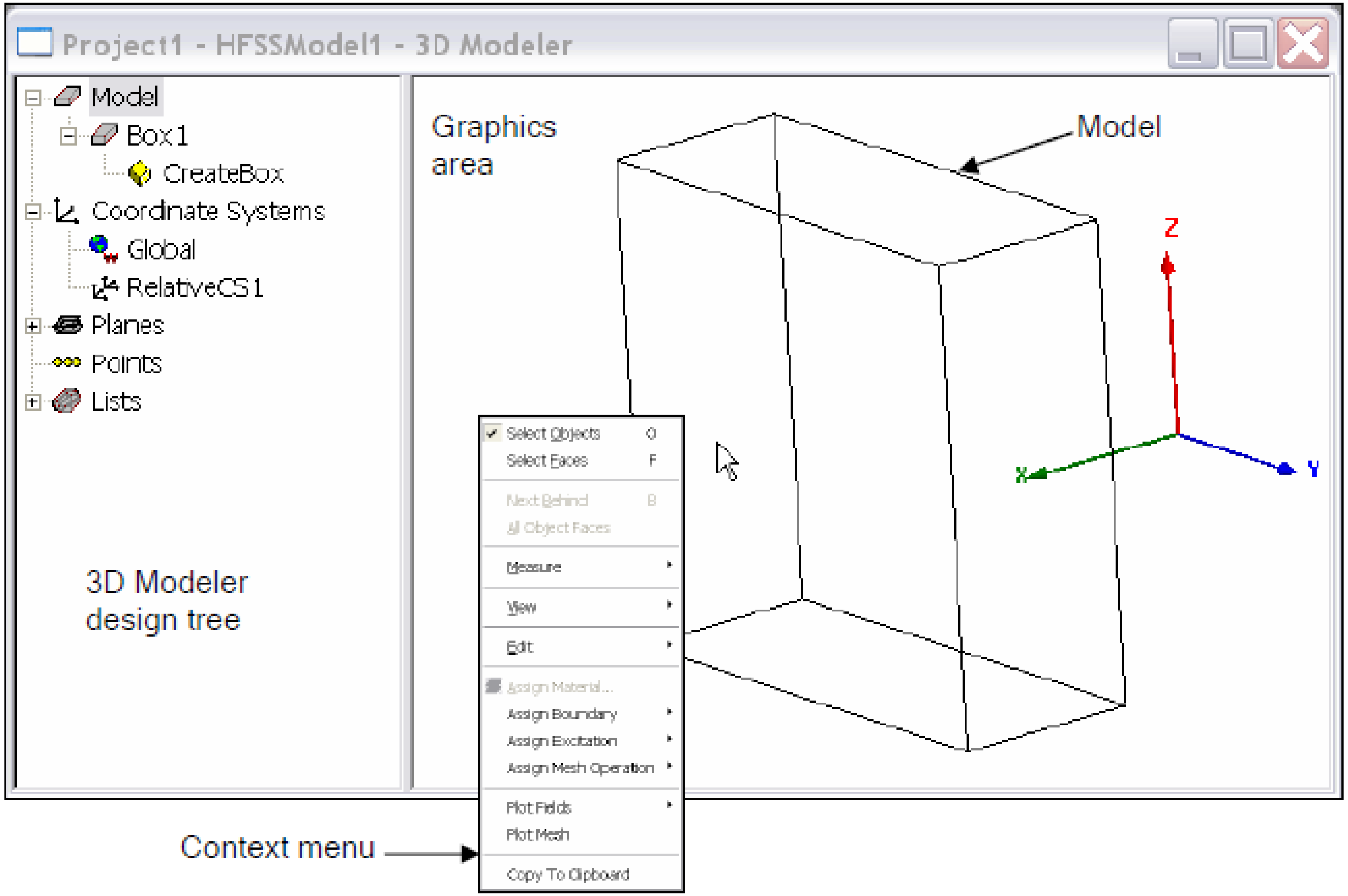
Eigenmode

② Click the **OK** button



8.3 Parametric Model Creation

- Overview of the 3D Modeler User Interface

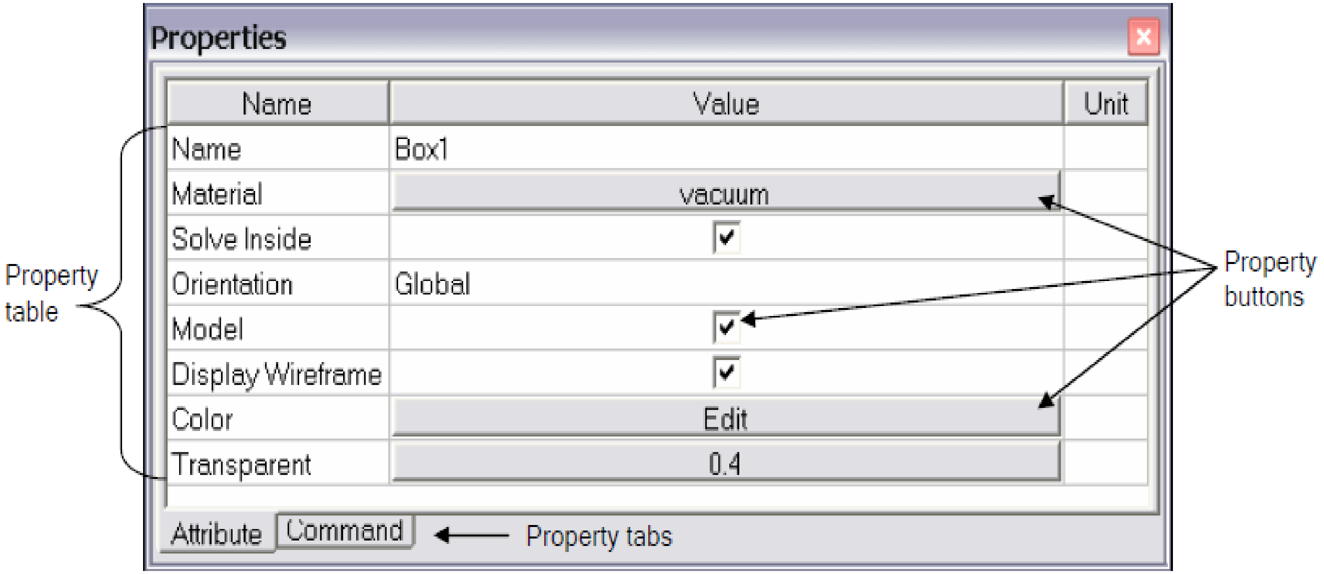


8.3 Parametric Model Creation

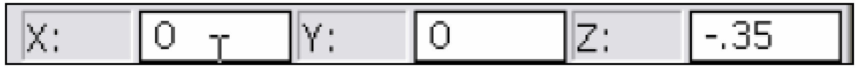
- When using the 3D Modeler interface you will also interact with two additional

Property Window

The Property Window is used to view or modify the attributes and dimensions of structural objects



Status Bar/Coordinate Entry



The Status Bar on the Ansoft HFSS Desktop Window displays the Coordinate Entry fields that can be used to define points or offsets during the creation of structural objects

8.3 Parametric Model Creation

- Creating and Viewing a Simple Structure
- Creating 3D structural objects is accomplished by performing the following steps:

1. Set the grid plane
2. Create the base shape of the object
3. Set the Height

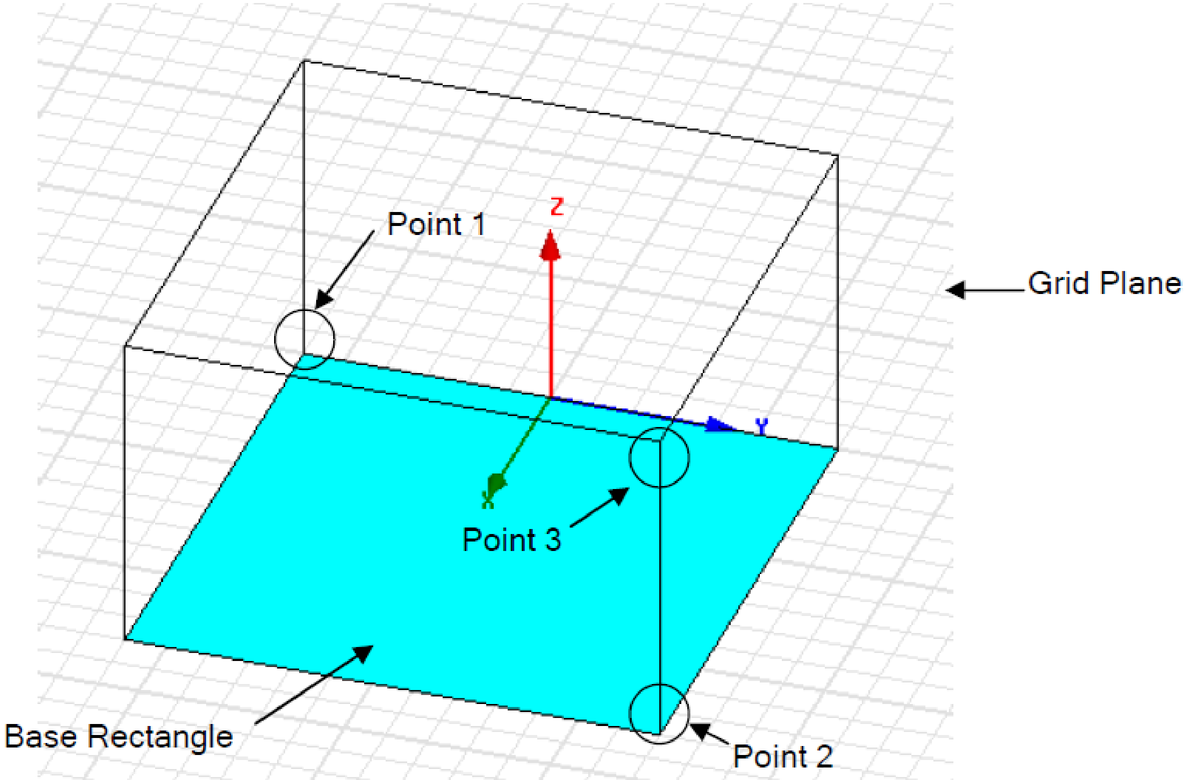
8.3 Parametric Model Creation

- Create a Box
- We will investigate creating a box to demonstrate these steps. These steps assume that project and a HFSS design have already been created. Three points are required to create the box. The first two form the base rectangle and the third sets the height

Point 1:
Defines the start point of the base rectangle

Point 2:
Defines the size of the base rectangle

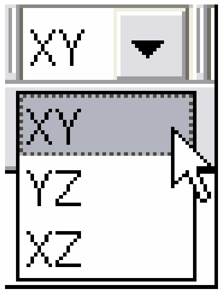
Point 3:
Defines the height of the Box



8.3 Parametric Model Creation

◆ Create a Box

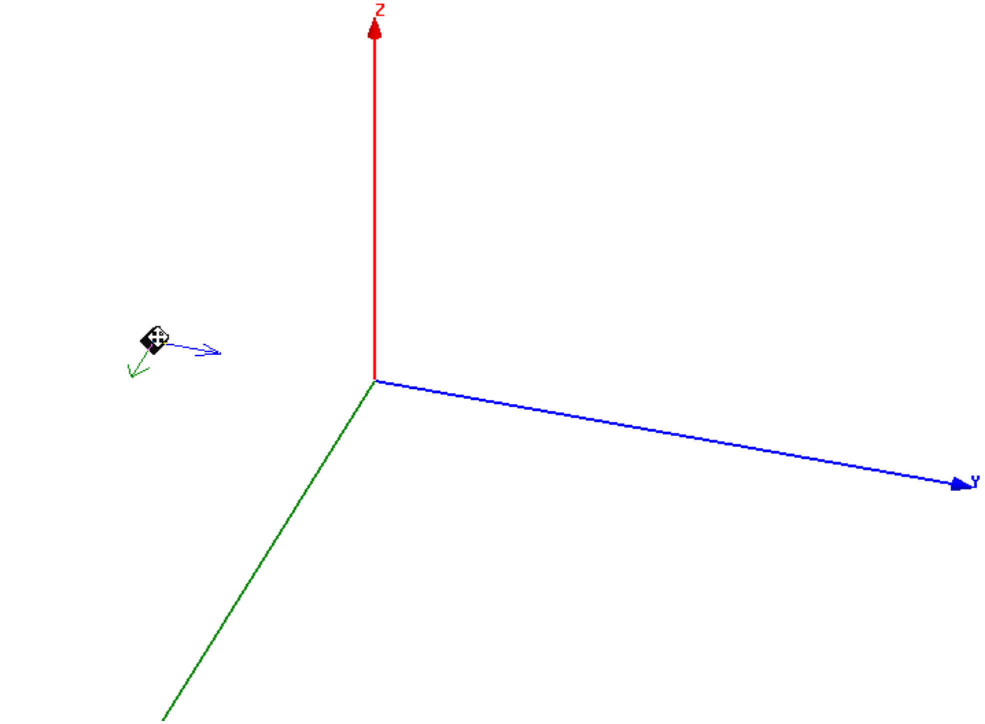
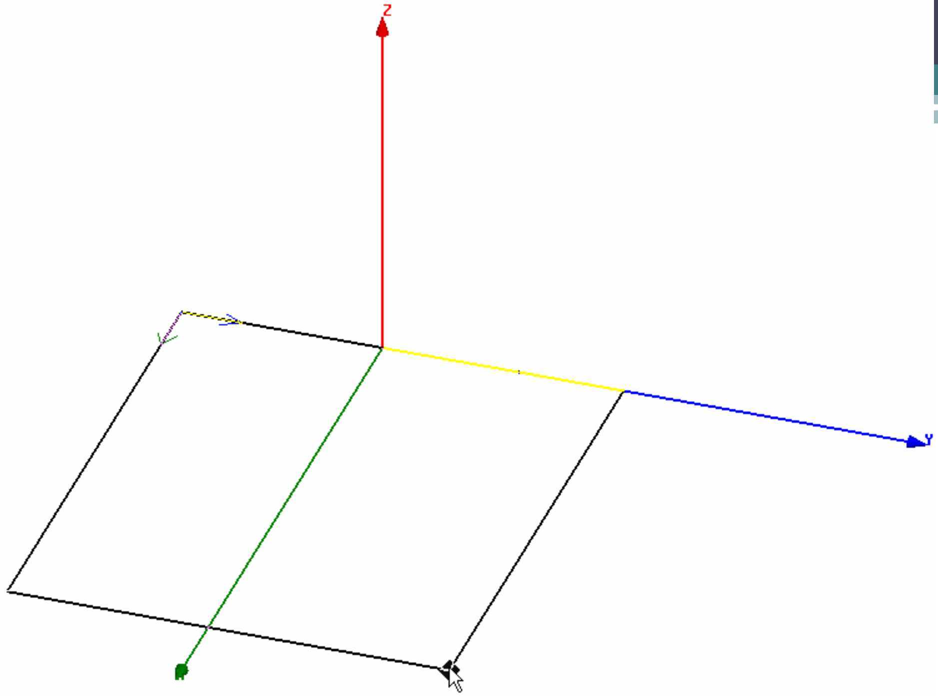
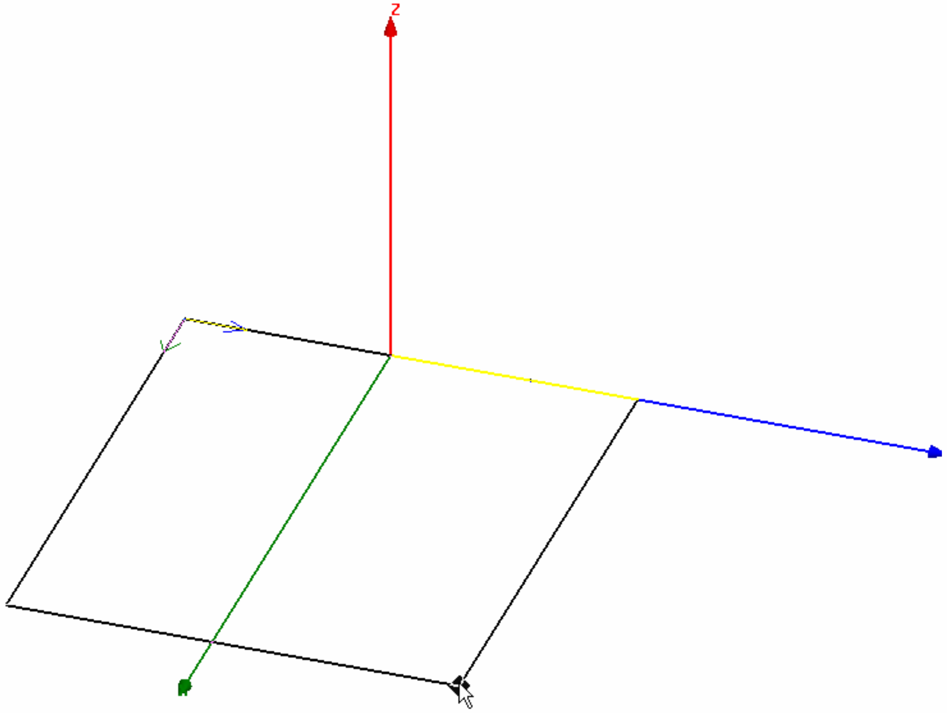
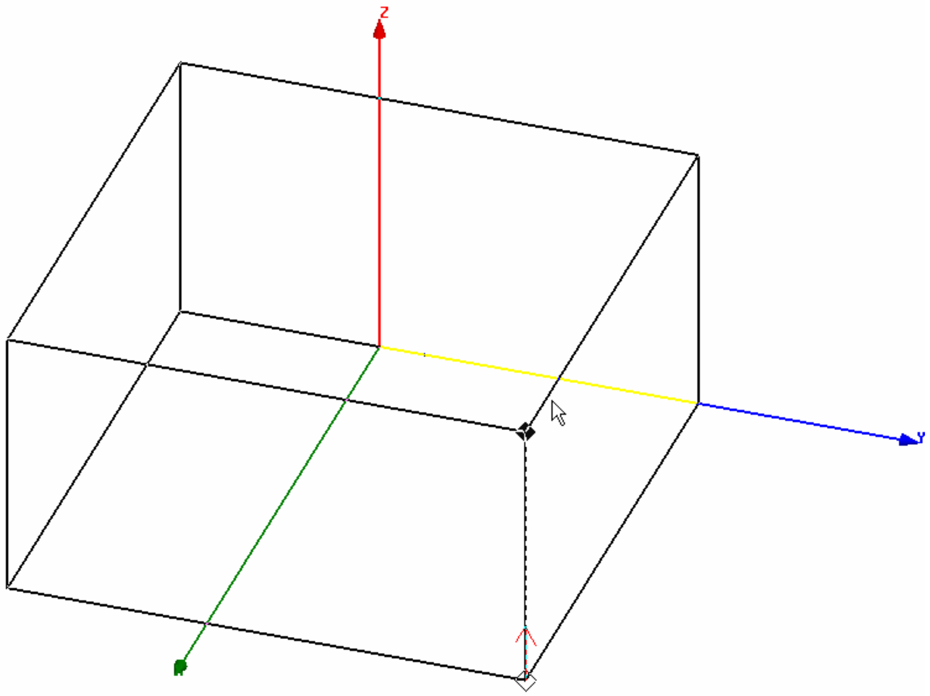
1. Select the menu item 3D Modeler > Grid Plane > XY
2. Use the mouse to create the base shape



① Set the start point by positioning the active cursor and click the left mouse button.

② Position the active cursor and click the left mouse button to set the second point that forms the base rectangle

③ Set the Height by positioning the active cursor and clicking left mouse button.

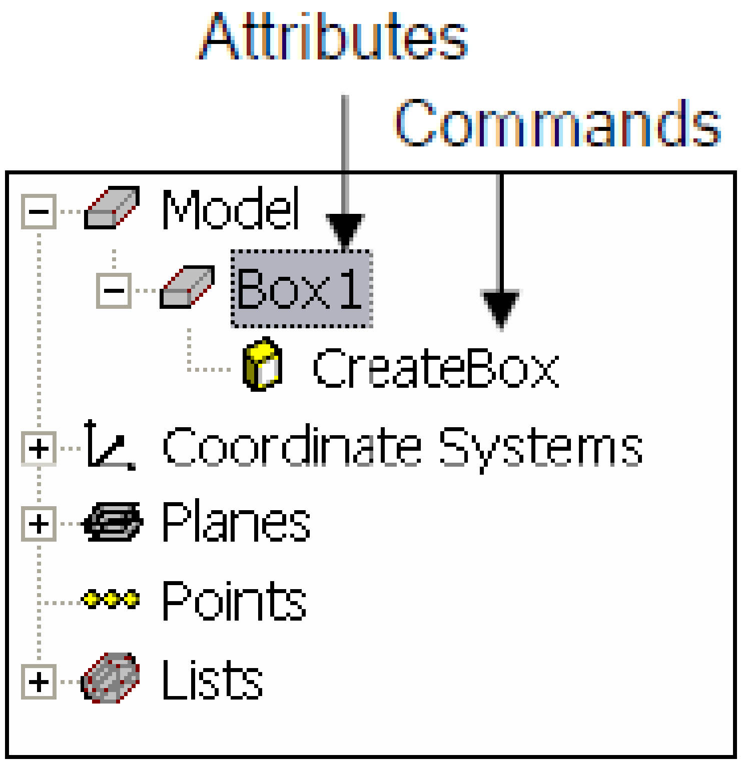

Z
Y

Z
Y

Z

Z
Y

8.3 Parametric Model Creation

- Specifying Points

- Object Properties

By default the Properties dialog will appear after you have finished sketching an object. The position and size of objects can be modified from the dialog. This methodallows you to create objects by clicking the estimated values using the mouse and then correcting the values in the final dialog.



Properties: Project22 - HFSSModel1 - 3D Modeler

Command | Attribute

	Name	Value	Unit	Description
	Command	CreateBox		
	Coordinate System	Global		
	Position	-1 , -2.2 , 0	mm	Point 1
	XSize	2	mm	
	YSize	2.4	mm	
	ZSize	0.6	mm	

☐ Show Hidden

Properties: Project22 - HFSSModel1 - 3D Modeler

Attribute

Defines the material, display, and solve properties

	Name	Value	Unit	Description	Read-only
	Name	Box1			☐
	Material	vacuum		Defined Project Material	
	Solve Inside	☑			☐
	Orientation	Global			☐
	Model	☑			☐
	Display Wireframe	☐			☐
	Color	Edit			☐
	Transparent	0			☐

☐ Show Hidden

OK | Cancel

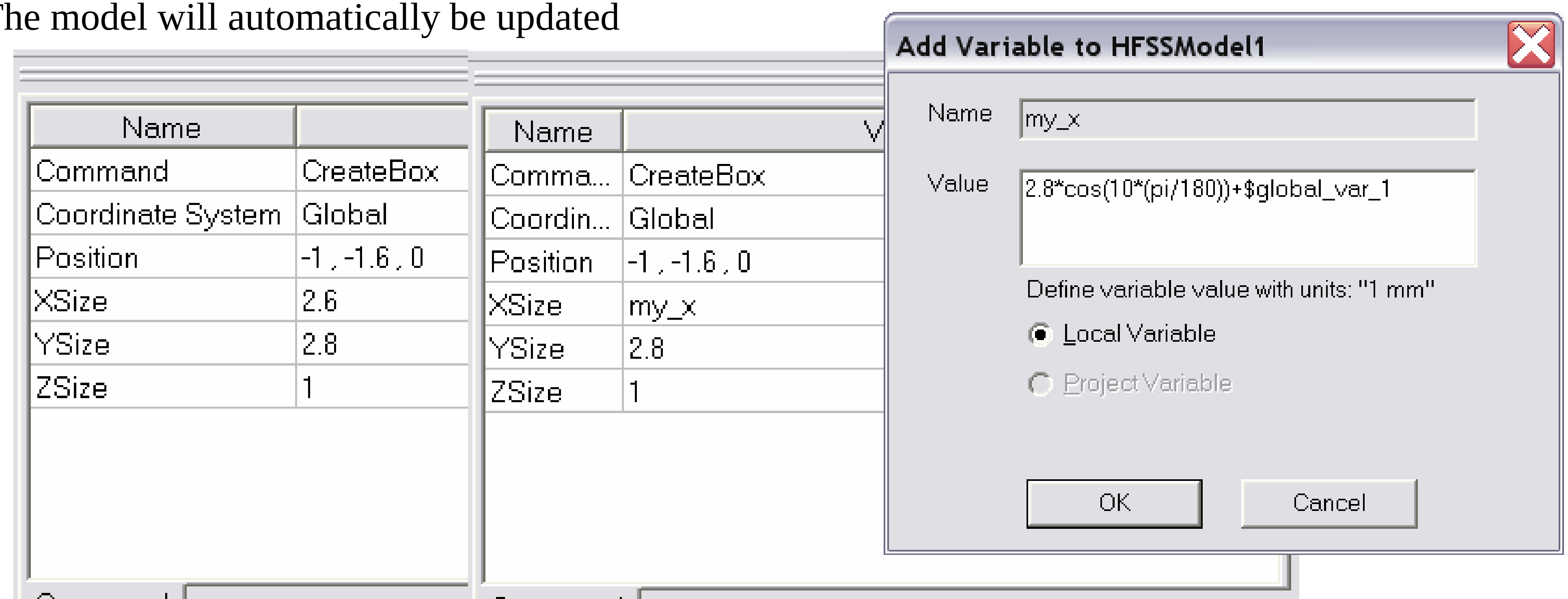
8.3 Parametric Model Creation

Defining Parameters

- **HFSS——Design Properties——.Add parameters**

OR

- Select the command to parameterized
- Choose the value to change
- Enter a variable in replace of the fixed value
- Define the variable using any combination of math functions or design variables.
- The model will automatically be updated



8.3 Parametric Model Creation

◆ Shortcuts

Since changing the view is a frequently used operation, some useful shortcut keys exist. Press the appropriate keys and drag the mouse with the left button pressed

ALT + Ctrl: Rotate
Shift + ALT: Dynamic Zoom
Shift + Ctrl: Move
Ctrl+D to fit your screen

8.3 Parametric Model Creation

◆ Combine Objects by Using Boolean Operations

Most complex structures can be reduced to combinations of simple primitives.Even the solid primitives can be reduced to simple 2D primitives that are swept along a vector or around an axis(Box is a square that is swept along a vector to give it thickness). The solid modeler supports the following Boolean operations:

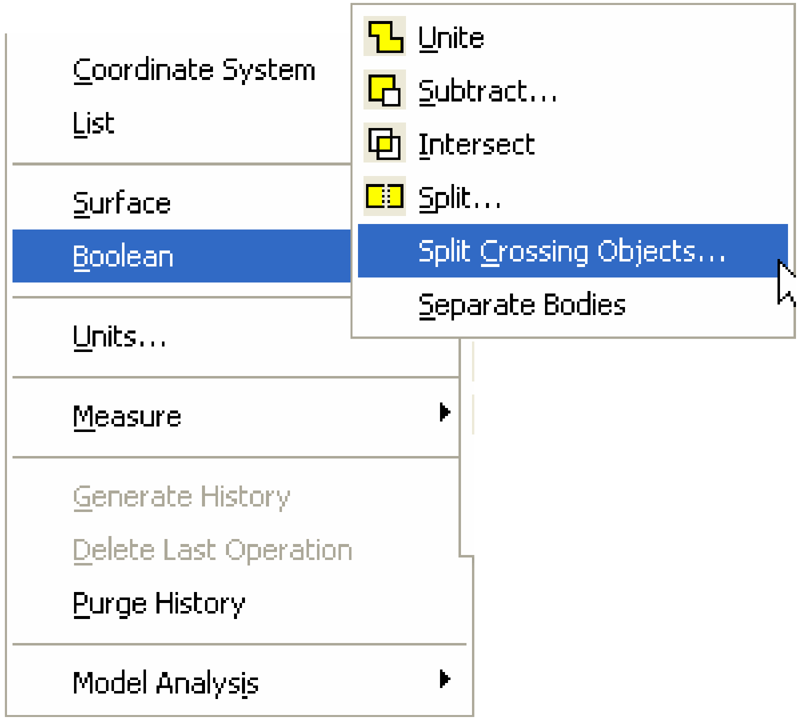
Unite – combine multiple primitives

Unite disjoint objects

Separate Bodies to separate

Subtract – remove part of a primitive from another

Split – break primitives into multiple parts



Boundary Conditions

◆ **Definition of Boundary Conditions**

➢ Select the object surface——
click the right ——Assign Boundary

◆ Perfect E

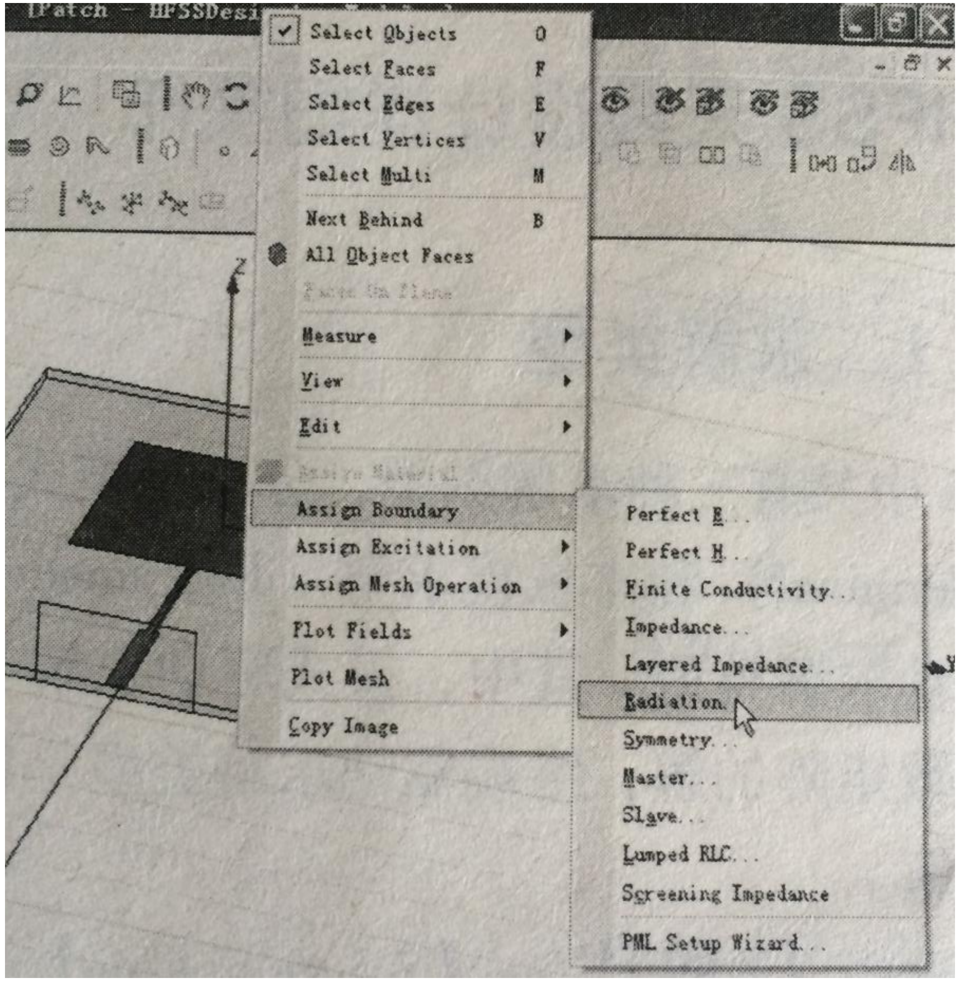
（Perfect H is a perfect magnetic conductor. Forces E-Field tangential to the surface）

◆ Perfect H

（Perfect H is a perfect magnetic conductor. Forces E-Field tangential to the surface.）

◆ Radiation

（Radiation boundaries, 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）



Excitations

- Excitations is defined as the excitation source in the three-dimensional or two-dimensional object surface .HFSS defines a variety of excitations, including Wave Port/Lumped Port/Floquet Port/Incident Wave/Voltage Source / Current Source/Magnetic Bias.

- The antenna transmit signal by means of a transmission line or waveguide, the connecting part between the antenna and a transmission line or waveguide is seen as a port plane . The excitation method of the Port plane in antenna design are **Wave Port** or **Lumped Port**.

Excitations

- Wave Port:

  If the port plane is touch the background plane, Wave Port is setted.

  （ The width and height of wave port are less than half wave length, otherwise It will Stimulate the waveguide mode)

- Lumped Port:

  If the port plane is inside of model , Lumped Port is setted.

- Wave Port

  - Driven Modal:

    when we set excitation , we must set the integration line to :

    1.confirm the direction of the electric field (integration line arrow represent the direction of positive electric filed)

    2.set integration path of the port voltage

  - Driven Terminal:

    we often set reference ground as the terminal line of reference conductor

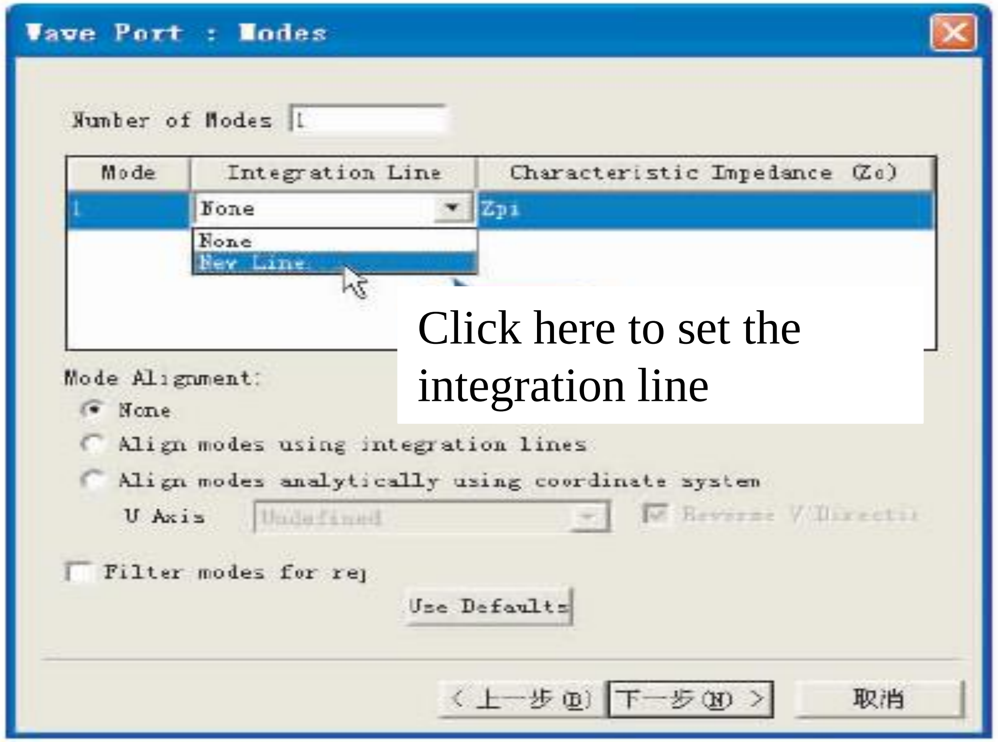
Excitations

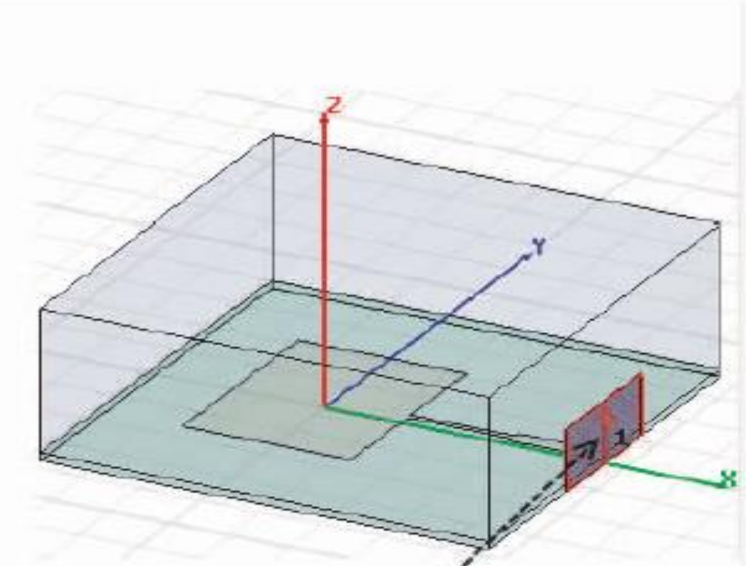
- Wave Port
- Driven Modal:

when we set excitation , we must set the integration line to :

1.confirm the direction of the electric field (integration line arrow represent the direction of positive electric filed)

2.set integration path of the port voltage



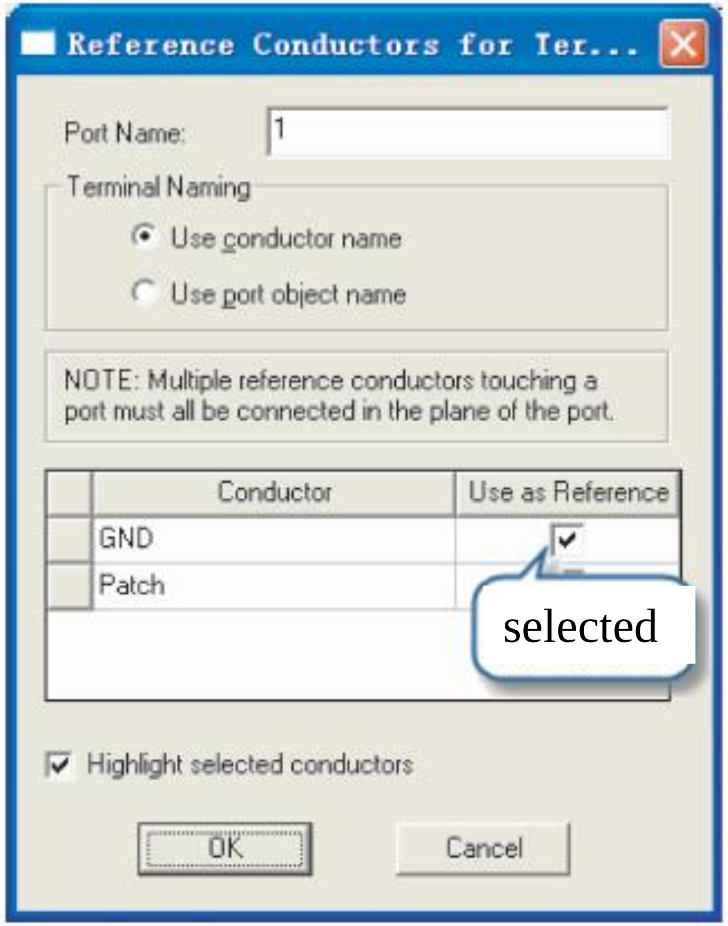


- Wave port integration line
- The arrow points the direction of positive electric filed

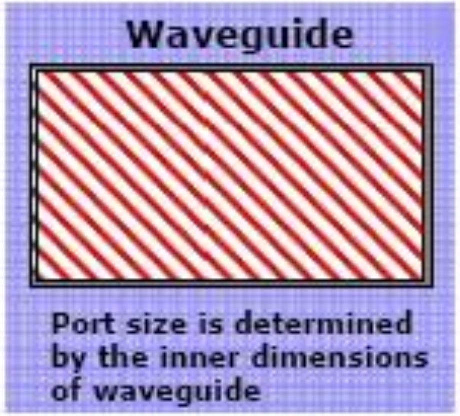
Excitations

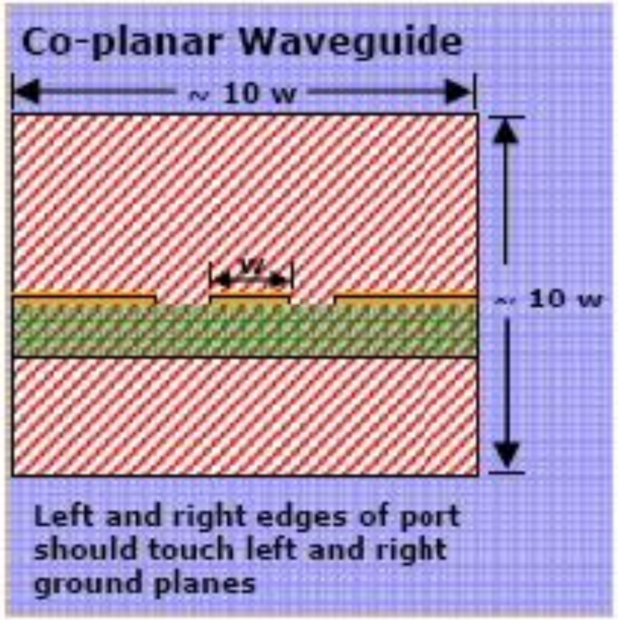
- Wave Port
- Driven Terminal:

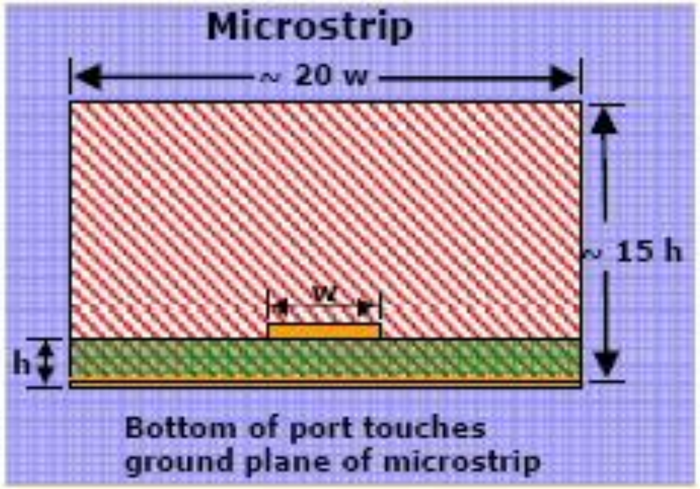
we often set reference ground as the terminal line of reference conductor



Excitations







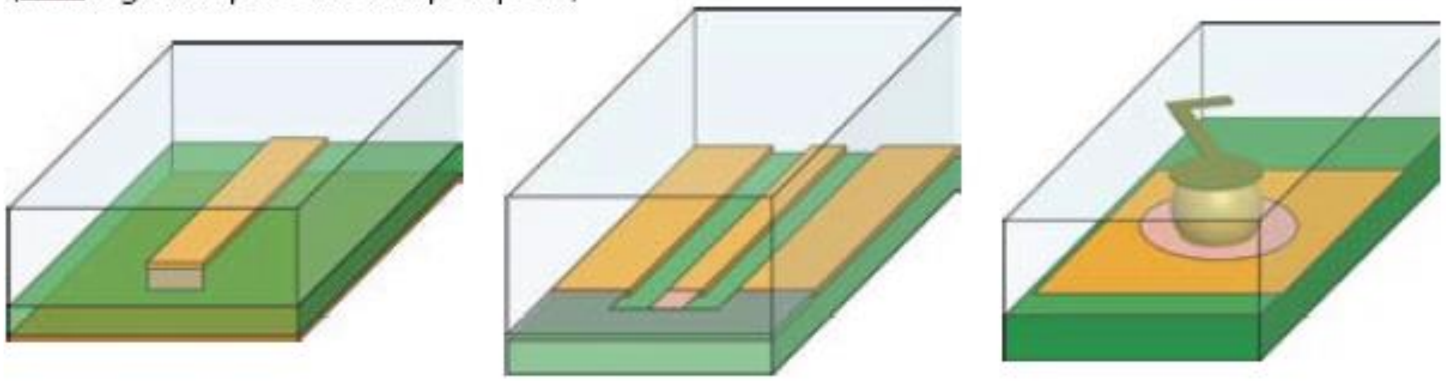
Wave ports represent places in the geometry through which excitation signals enter and leave the structure. They are used when modeling strip lines and other waveguide structures (for example, rectangular or circular)..

Excitations

Lumped ports are similar to traditional wave ports, but can be located internally and have a complex user-defined impedance. Lumped ports compute S-parameters directly at the port. An example use is modeling microstrip structures.

A lumped port can be defined as a rectangle from the edge of the trace to the ground or as a wave port. The default boundary is perfect H on all edges that do not come in contact with the metal or with another boundary condition.

(region represents lumped port.)



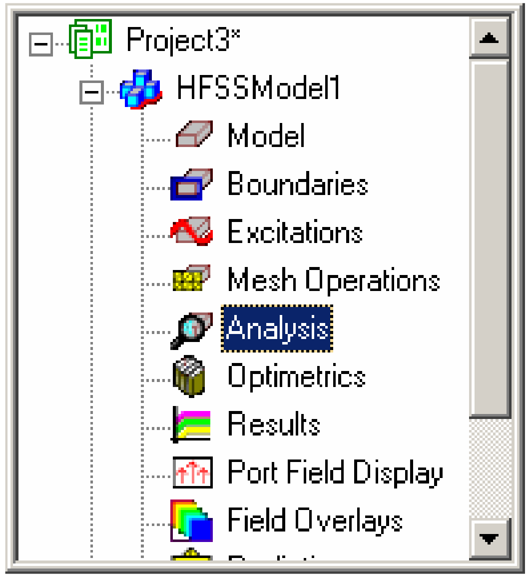
Port is internal to the solution Space. The 2D port rectangle touches the signal trace with one edge and the opposite edge touches the ground plane.

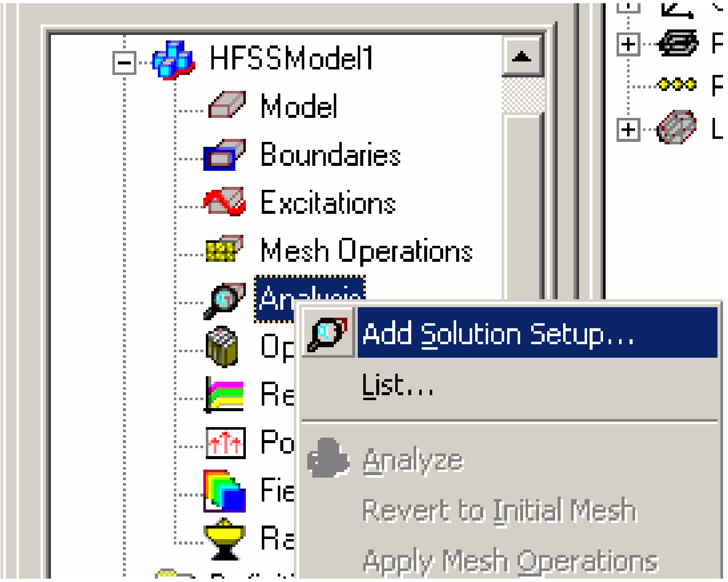
Port is internal to Solution Space. The 2D port rectangle touches the signal trace with one edge, and the opposite edge touches user-drawn PEC objects (grey).

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

8.4 Analysis Setup

- Adding a Solution Setup

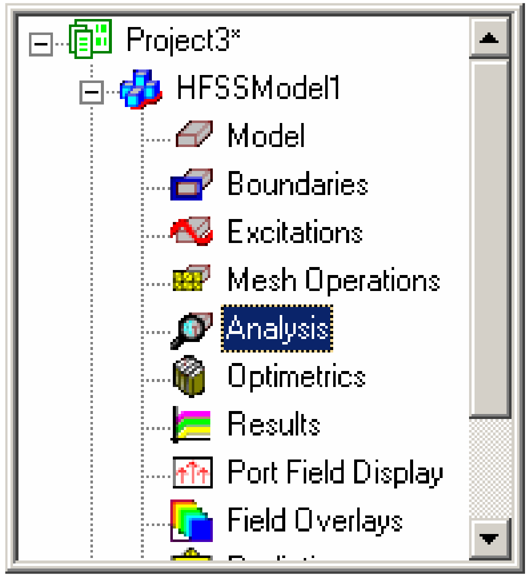


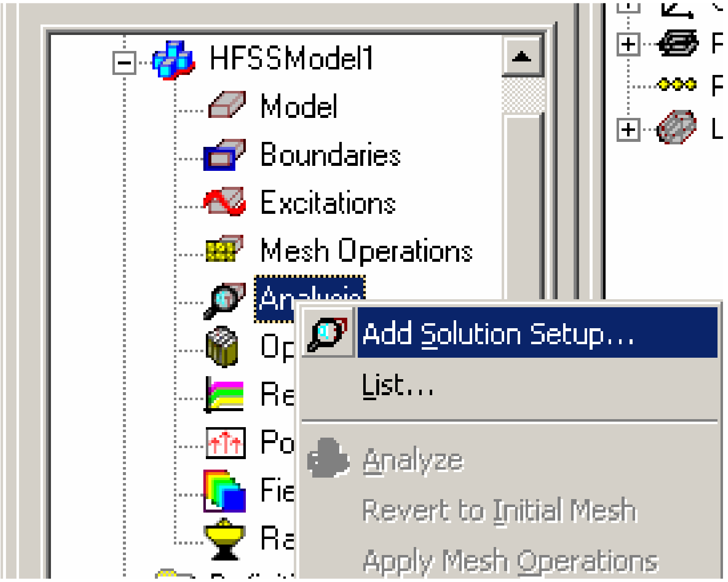


By default, the General Tab will be displayed. The Solution Frequency and the Convergence Criteria are set here.

8.4 Analysis Setup

- Enabling/Disabling a Solution Setup



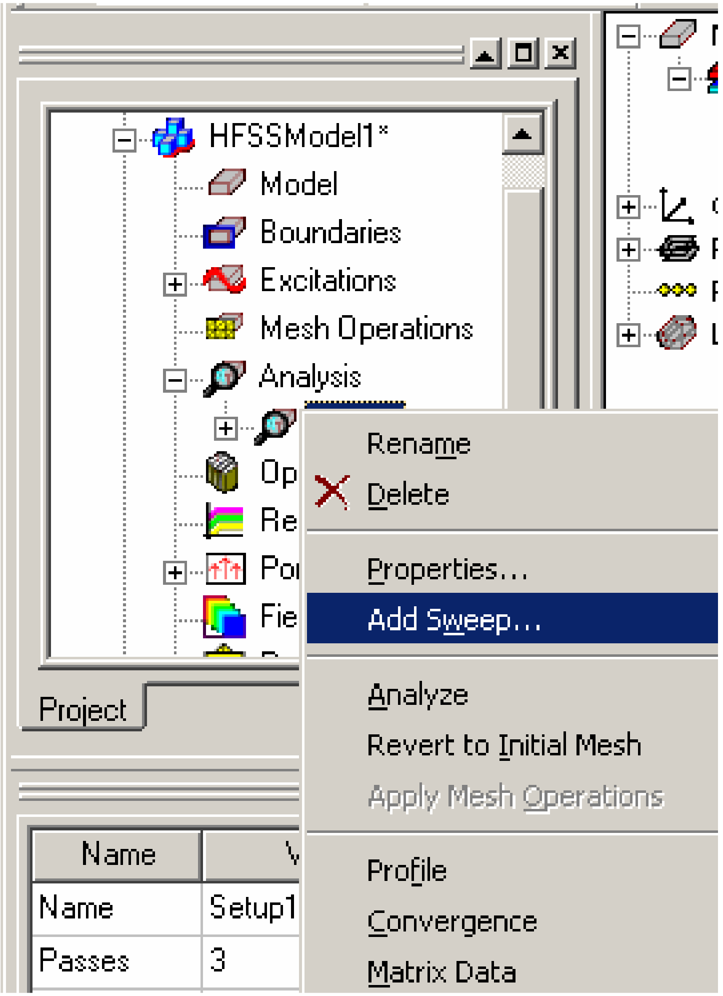


By default, the General Tab will be displayed. The Solution Frequency and the Convergence Criteria are set here.

8.4 Analysis Setup

- Add Sweep

After a Solution Setup has been added you can also add a Frequency Sweep .To do this, right-click on Setup in the HFSS Model Tree . The Edit Sweep window will appear



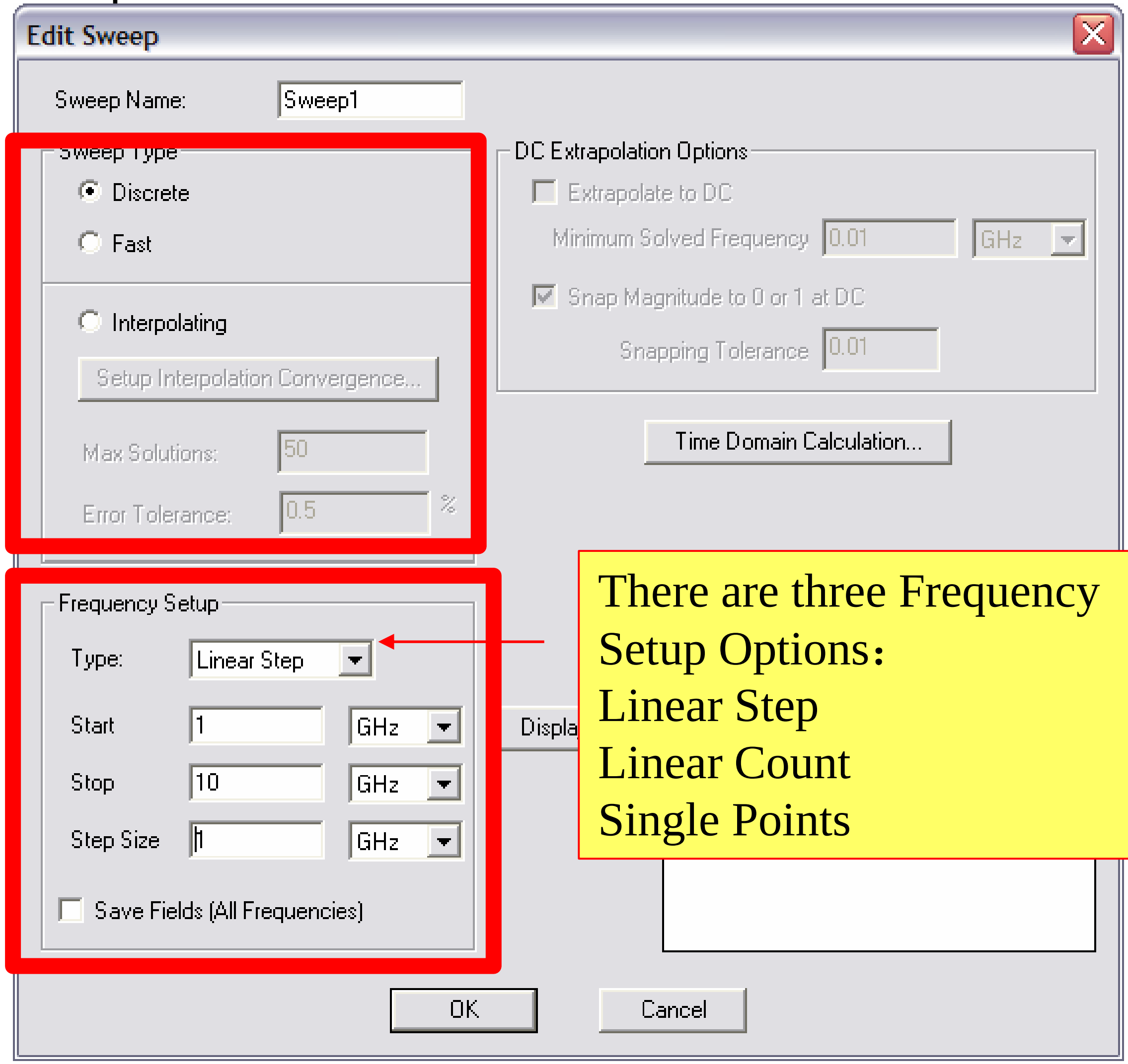
8.4 Analysis Setup

- Enabling/Disabling a Solution Setup

Sweep Type

After the sweep type has been chosen, the frequencies of interest must be specified.

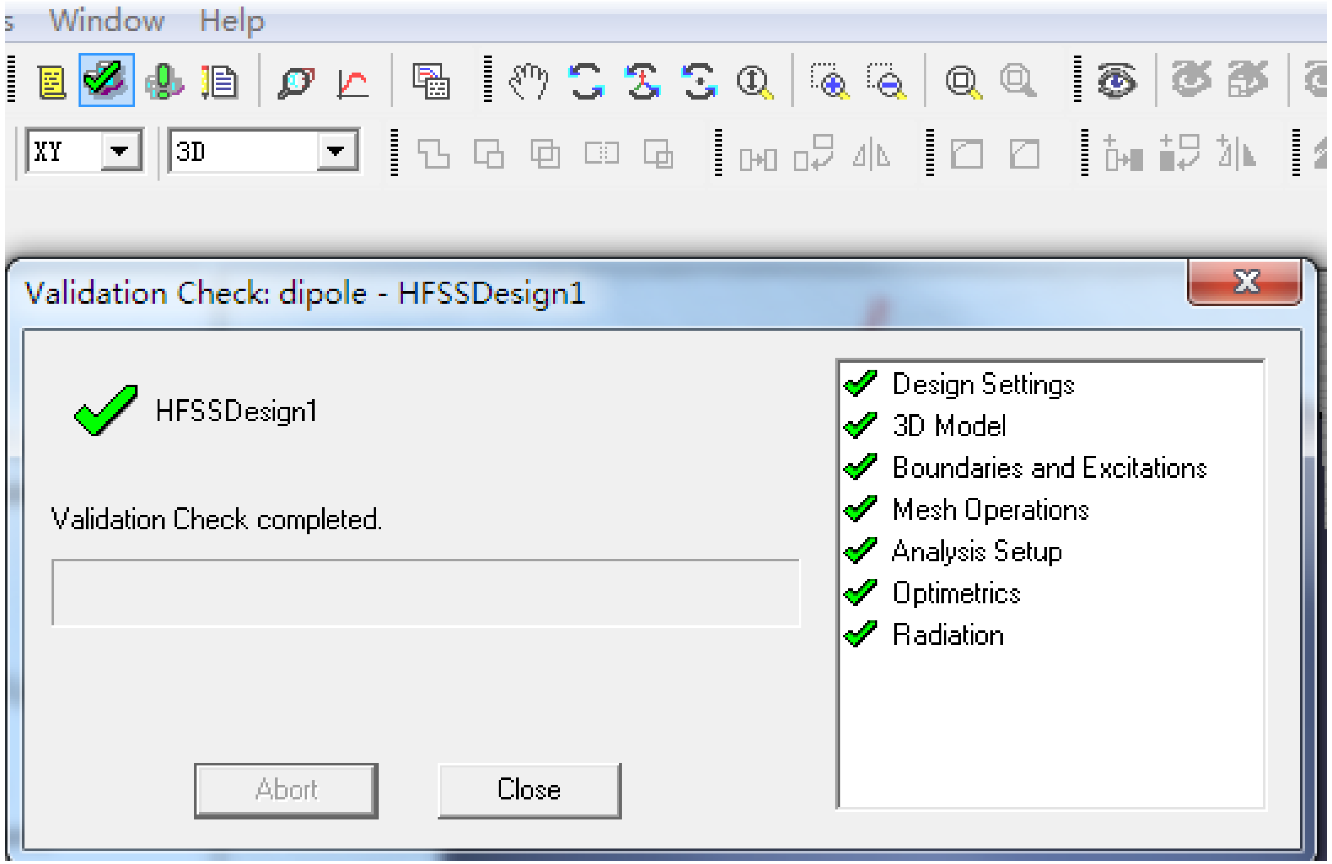
Frequency Setup



There are three Frequency Setup Options:
Linear Step
Linear Count
Single Points

8.5 Data plotting

- Validation check
- **HFSS—— Validation check**



8.5 Data plotting

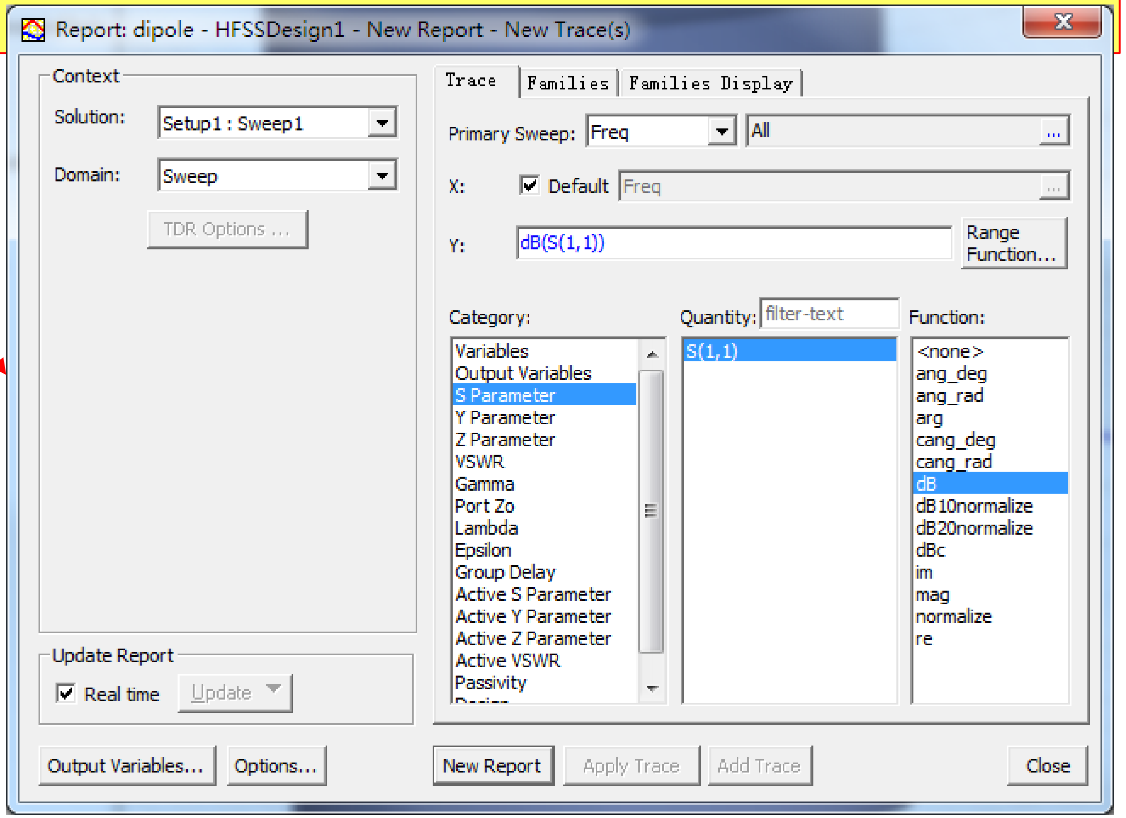
To Create a Plot:

- Plotting Data

- Types of Plots:
  Rectangular Plot
  Polar Plot
  3D Rectangular Plot
  3D Polar Plot
  Smith Chart
  Data Table
  Radiation Pattern

To Create a Plot:

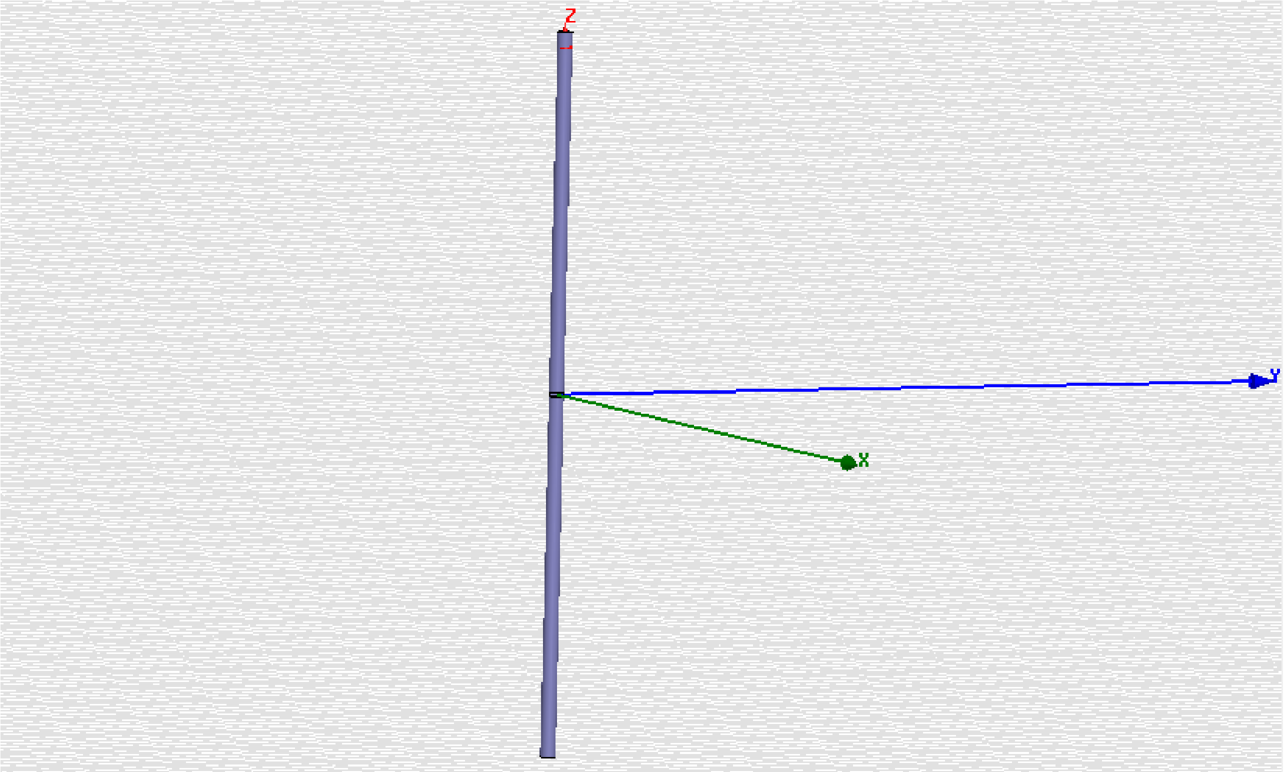
1. Select HFSS > Results > Create Report
2. Select Report Type and Display Type from the selections above
3. Click OK and the Report Editor will be displayed



Try——Antenna Simulation

1.Linear dipole antenna

- Arm of dipole： cylinder (0,0,0.12) r=0.5 h=23.88
- Feed port： Y-Z plane rectangle（0， -0.5， -0.12） 1， 0.24
- air： cylinder（0， 0， -34）r=25.5 h=68

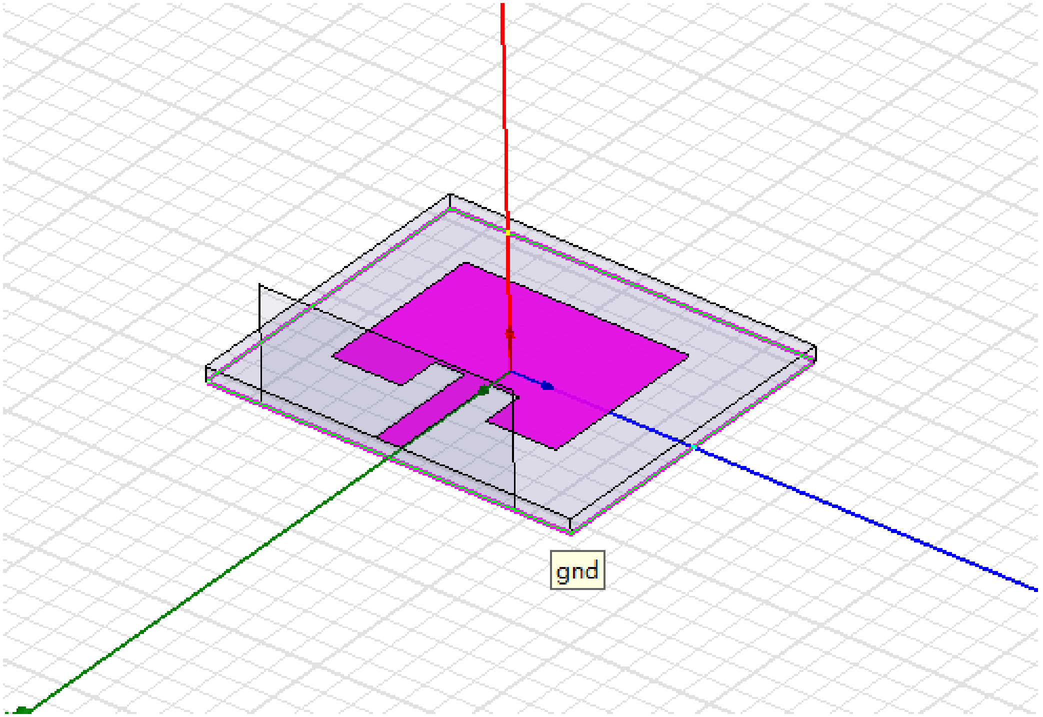


2. Microstrip antenna

- Substrate: box (11,-13,0) dx=22 dy=26 dz=1
- Ground: (11,-13,0) dx=22 dy=26 dz=0
- Patch: (6,-8,1) 12,16,0

  (6,-3,1) 3,6,0

  (11,-0.9 ,1) 8,1.8,0
- Feed(wave port): (11,-9, 0) 0,18,8



Air box (32,-22,0) 35,52,27

Thank you!