



Pacific Northwest
NATIONAL LABORATORY

*Proudly Operated by **Battelle** Since 1965*

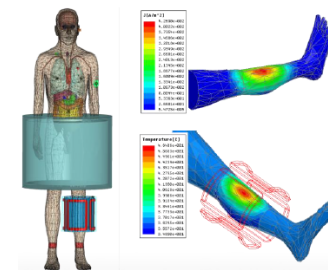
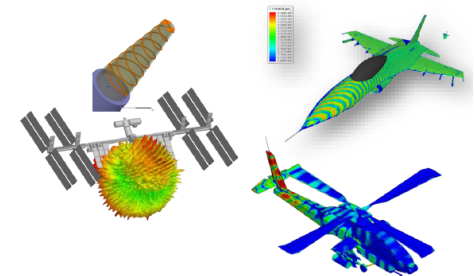
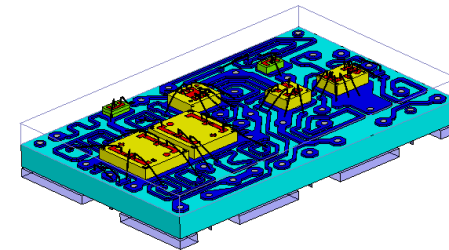
Ansys High Frequency Structure Simulator (HFSS) Tutorial

MARK JONES

PACIFIC NORTHWEST NATIONAL LABORATORY

1/10/17

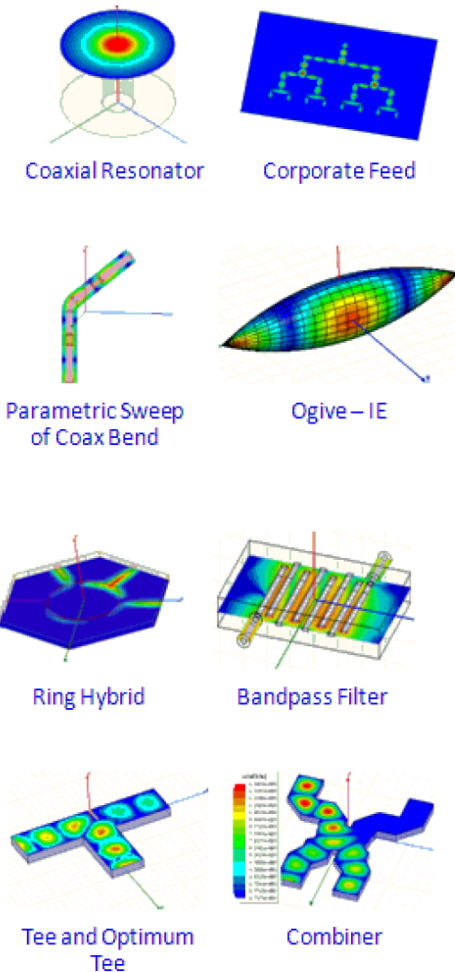
- ▶ Overview of HFSS
 - Capabilities and key features
 - Example measurement comparisons
- ▶ Cylindrical cavity tutorial
 - Eigenmode solver
 - Parametric geometry
 - Curvilinear elements
 - Modal frequencies, Q-factors, and fields
 - Field calculator
- ▶ Dipole antenna tutorial
 - Driven excitation solver
 - Radiation boundaries
 - Frequency sweep
 - S-parameters, near and far fields



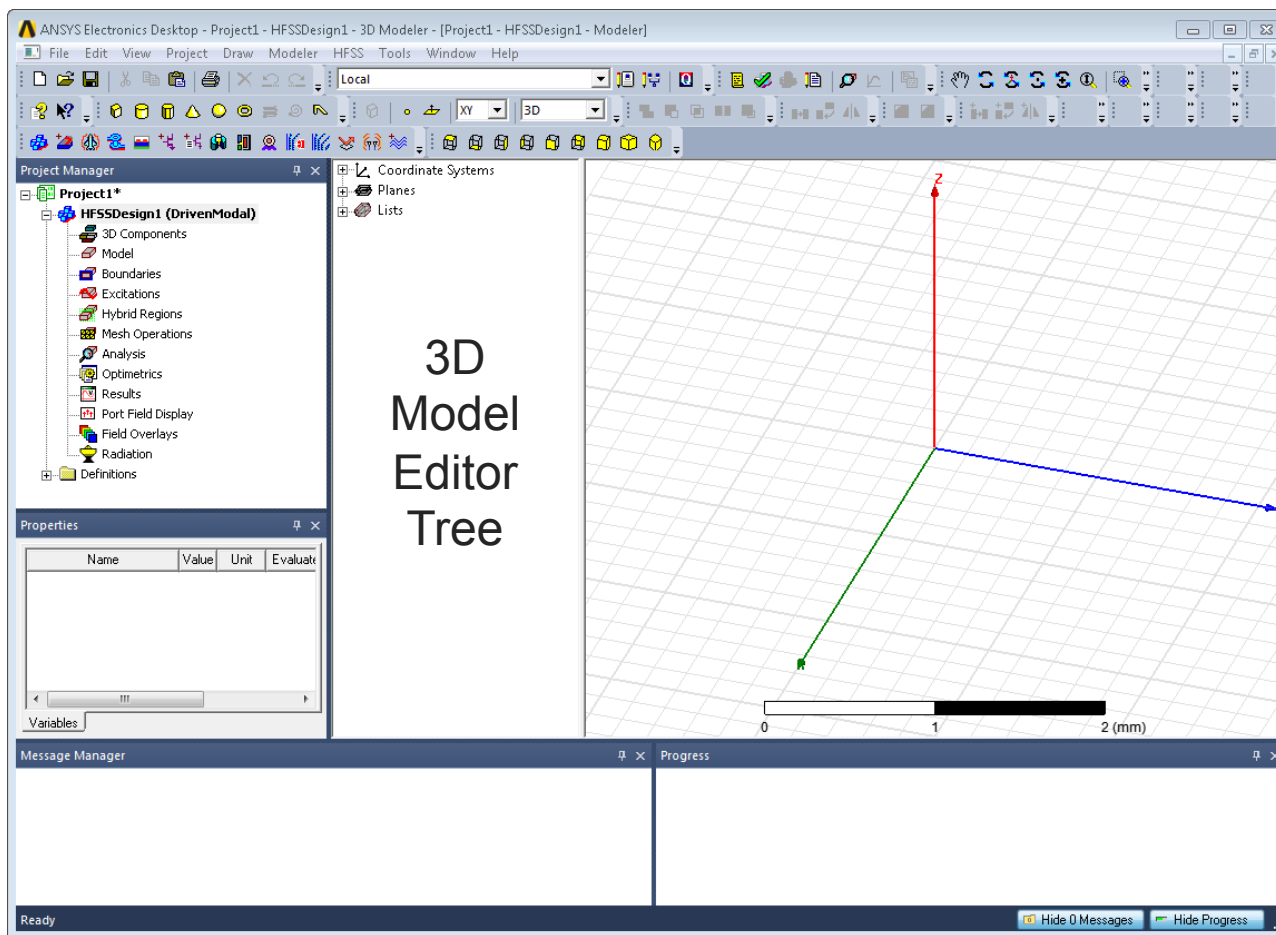


Introduction to HFSS

- ▶ Full-wave frequency-domain 3-D field solver based upon finite element method
 - Industry-standard accuracy
 - Adaptive meshing of arbitrary geometry
 - Fully parametric modeling
 - Optimization and HPC
 - Multi-physics via Ansys Workbench
- ▶ Widely used for RF/microwave design
 - Antenna design and platform integration
 - Filters and waveguide structures
 - Electronic packages and PCBs
 - Connectors and transitions
 - EMC/EMI
 - Radar cross-section
- ▶ Integrated into Ansys Electronics Desktop
 - Part of Ansys Electromagnetics Suite



HFSS User Interface



Project
Manager

3D
Model
Editor
Tree

Toolbars

3D Model
Editor
Graphics

Properties

Message
Manager

Progress
Window



Solution Types

- ▶ **Eigenmode solution**
 - Solves for natural resonances of structure based on geometry, materials, and boundaries
 - Provides modal frequencies, unloaded Q-factors, and fields

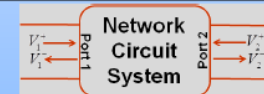
- ▶ **Driven solution**
 - Port or incident field used to excite the structure
 - *Driven modal* method commonly used for RF/microwave designs
 - *Driven terminal* method commonly used for multi-conductor transmission lines
 - Provides S-parameters and fields

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

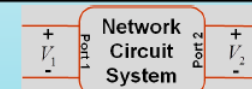


Driven Terminal

- 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

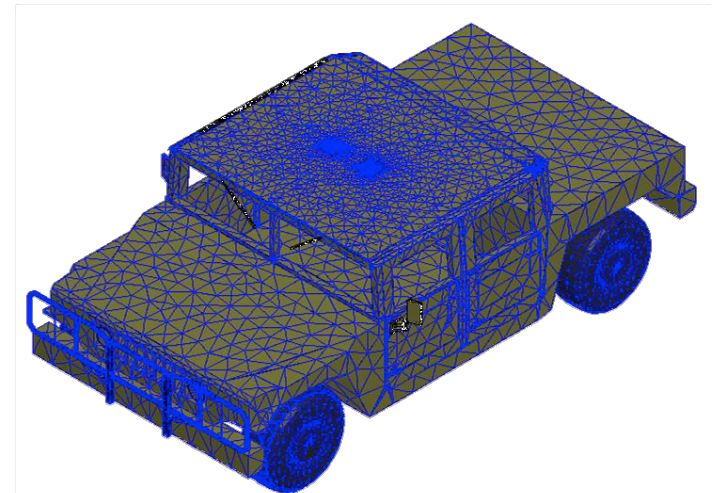
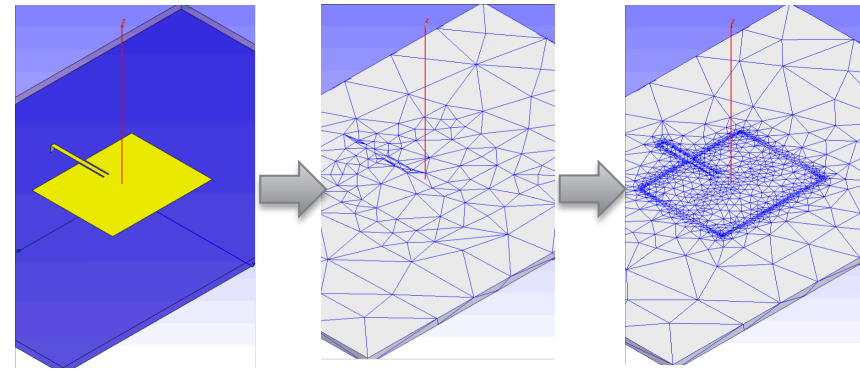
- Each conductor touching the port is considered a terminal or a ground
- Energy propagates along each terminal in a single TEM mode
- Each Terminal has its own column and row in the S, Y and Z parameters
- Does not support symmetry boundaries or Floquet Ports





Adaptive Mesh Algorithm

- ▶ Tetrahedral mesh automatically generated and refined below user-defined electrical length
 - Tetrahedral element shape conforms to arbitrary geometries
- ▶ Iterative algorithm solves fields and refines mesh until user-defined convergence threshold value is reached
 - Driven modal: S-parameter convergence
 - Eigenmode: Frequency convergence
- ▶ Produces graded mesh with fine discretization only where needed to accurately represent field behavior
 - Efficient use of computational resources
 - Tunes mesh to capture EM performance





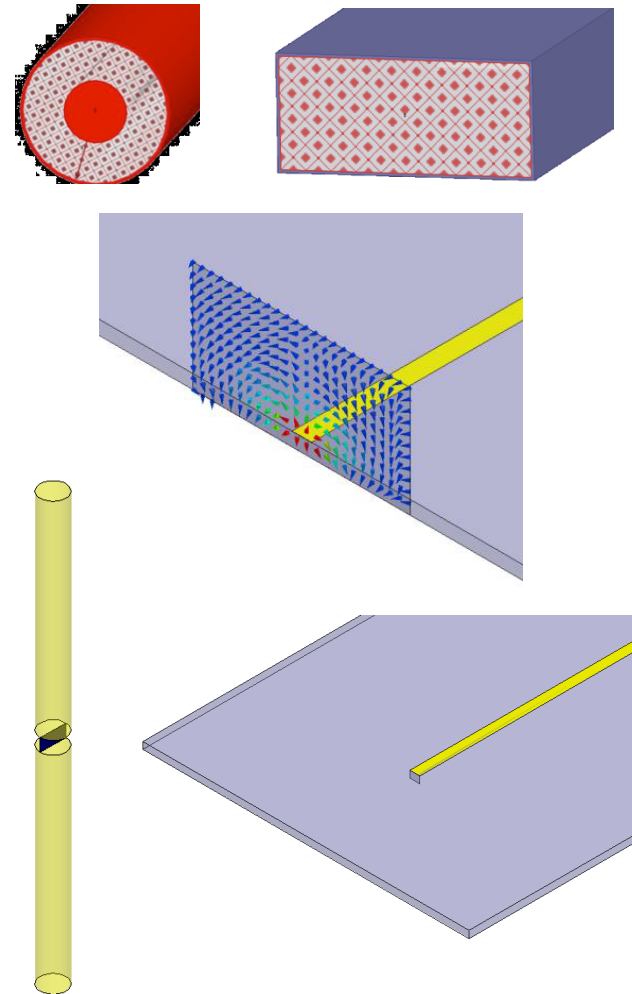
Port Excitations

▶ Wave ports

- 2D FEM solver calculates requested number of modes (treated as t-line cross-section)
- Solves for impedances and propagation constants
- Supports multiple modes and de-embedding
- Simple for closed t-lines
- Must allow room for fields of open t-lines
- Must touch external boundary or backed by conducting object

▶ Lumped ports

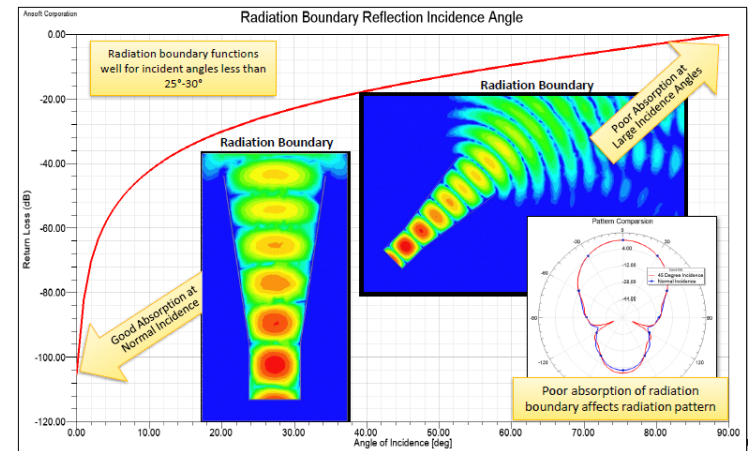
- User-assigned constant impedance
- Uniform electric field on surface
- Single TEM mode with no de-embedding
- Can be internal to model





Boundary Conditions

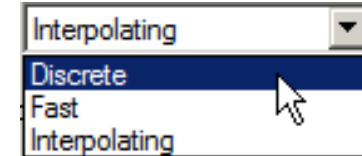
- ▶ Used to simplify geometry or make meshing more efficient
- ▶ Material properties for surfaces
 - Finite conductivity (imperfect conductor)
 - Perfect electric or magnetic conductor
- ▶ Surface approximations for components
 - Lumped RLC
 - Layered impedance
- ▶ Radiation
 - Absorbing boundary condition
 - Perfectly matched layers (PML)
- ▶ Any object surface that touches the background is automatically defined as Perfect E boundary



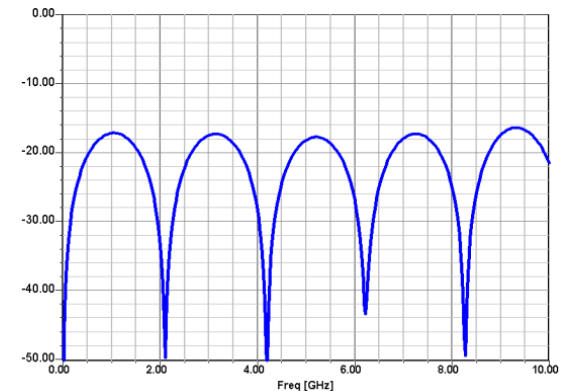


Frequency Sweeps

- ▶ Discrete sweep
 - Solves adapted mesh at every frequency
 - Matrix data and fields at every frequency
- ▶ Fast sweep
 - Extrapolates rational polynomial function for electric field over specified range
 - Usually valid over less than 10:1 BW
 - Matrix data and fields at every frequency
- ▶ Interpolating sweep
 - Solves minimum number of frequencies to create polynomial fit for S-parameters
 - Useful for very broadband S-parameters
 - Matrix data at every frequency



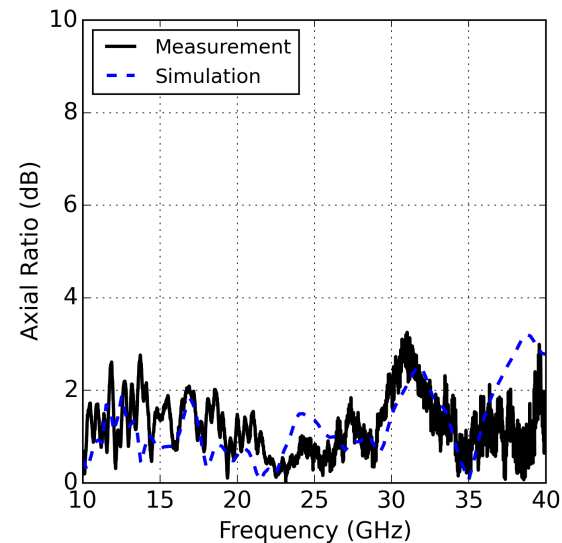
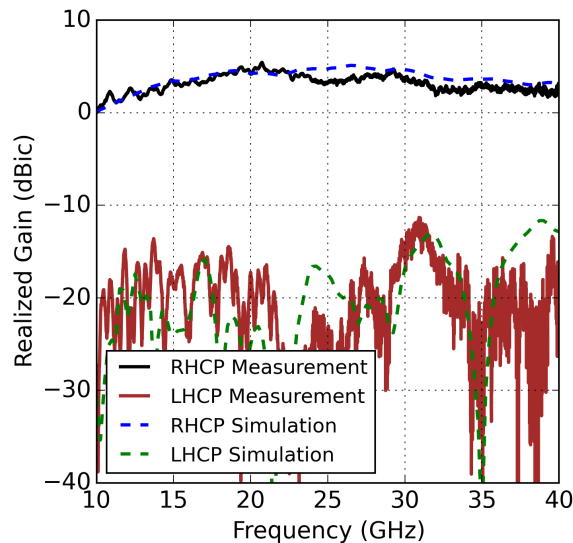
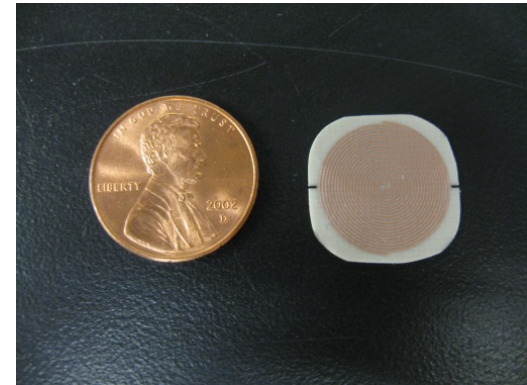
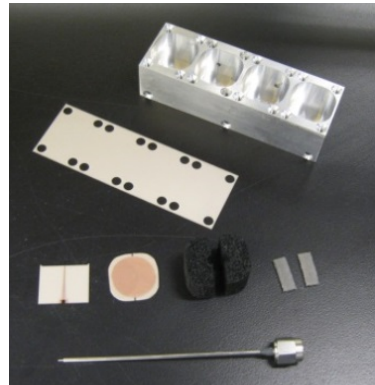
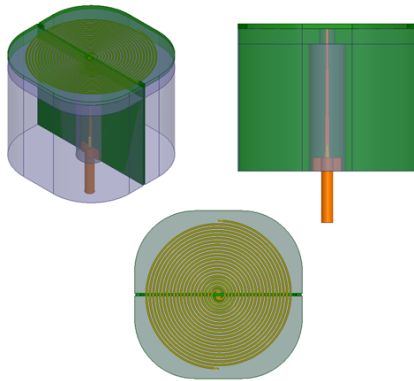
$$S = \frac{\beta_q (s - z_q)(s - z_{q-1}) \dots (s - z_1)}{\alpha_q (s - p_q)(s - p_{q-1}) \dots (s - p_1)}$$





Example Comparison with Measurement

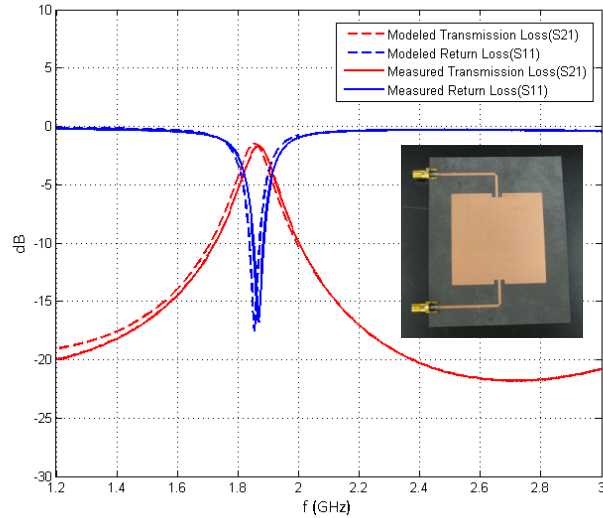
▶ Cavity-backed Archimedean spiral antenna with tapered line balun



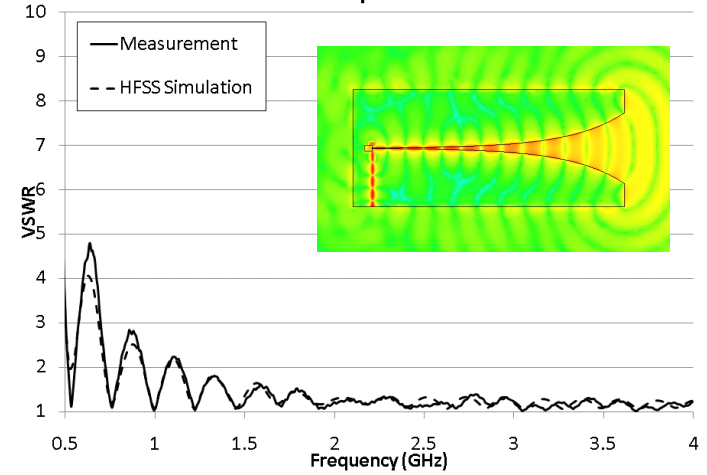


Example Comparisons with Measurement

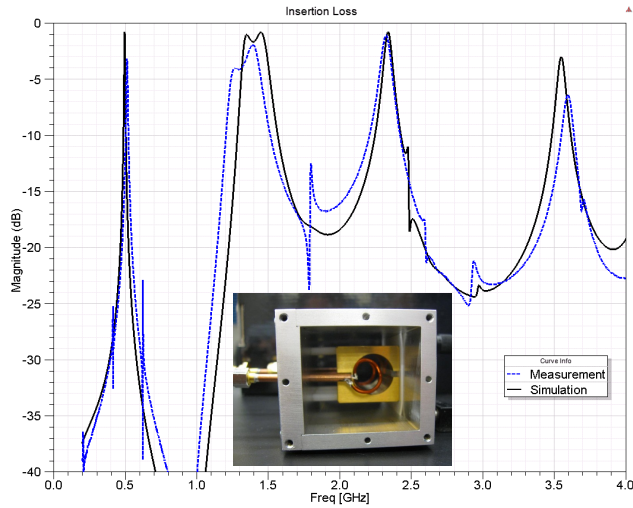
Resonant Patch Antenna



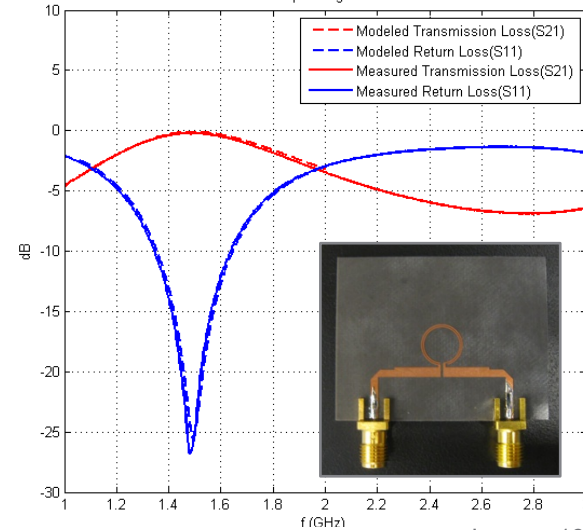
VSWR of Vivaldi Tapered Slot Antenna



Insertion Loss



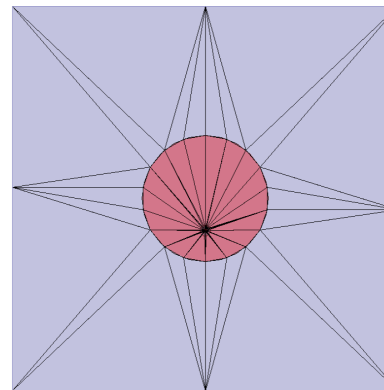
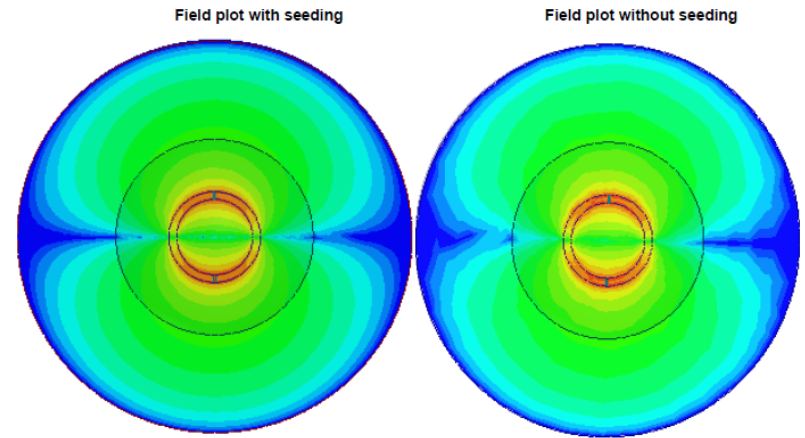
Planar Split Ring Resonator



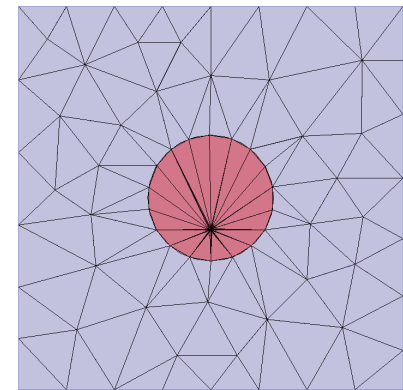


Mesh Controls

- ▶ Mesh seeding allows user to directly influence initial mesh
 - Reduce number of adaptive passes
 - Focus mesh elements in critical areas
 - Not required for accurate results
 - Can create better-looking field plots
 - Seeding radiation boundary can improve far-field data
- ▶ Lambda refinement
 - Ensures that initial mesh is refined to fraction of electrical wavelength
 - Electrical size depends on solver basis order
 - Zero: $\lambda/10$, First: $\lambda/3$, Second: $2\lambda/3$, Mixed: $2\lambda/3$



Initial geometric mesh

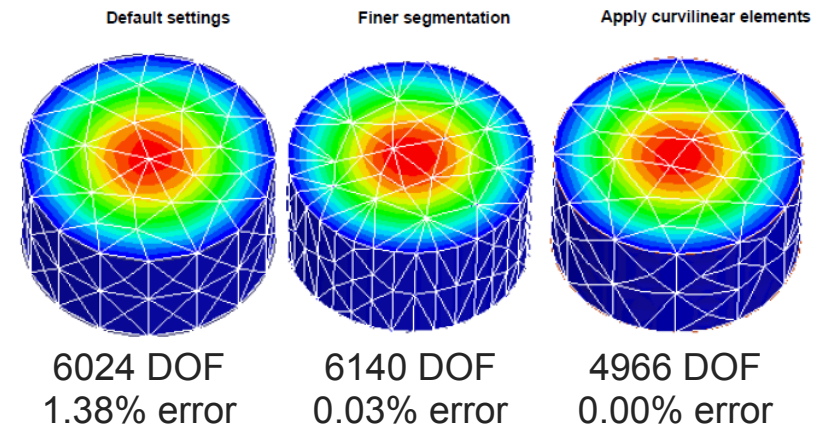
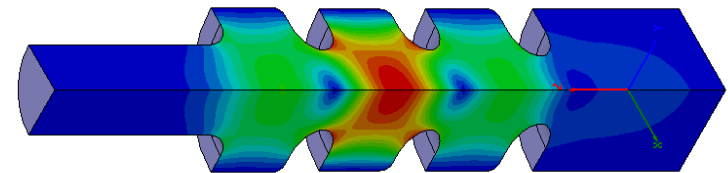
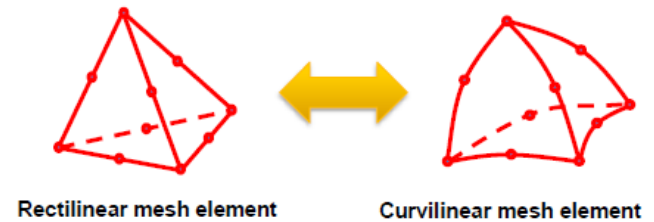


Electrical mesh after lambda refinement



Curvilinear Mesh Elements

- ▶ Global mesh approximation setting for all true surfaces in model
- ▶ Higher order (curvilinear) elements used to represent the geometry
 - Pulls midpoints of tetrahedra surfaces to true surface
- ▶ Pillbox resonator with analytical $f_R = 22.950$ GHz for TM_{010} mode
 - Default setting: 23.269 GHz
 - Finer segmentation: 23.012 GHz
 - Curvilinear elements: 22.950 GHz





FEM Solver

- ▶ Direct matrix solver is default technique
 - Exactly solves matrix equation $Ax = b$
 - Multi-frontal sparse matrix solver to find inverse of A (LU decomposition)
 - Solves for all excitations b simultaneously
- ▶ Iterative matrix solver is optional technique for driven solutions
 - Reduces RAM usage and often runtime
 - Solves matrix equation $Max = Mb$ where M is preconditioner
 - Begins with initial solution and recursively updates solution until tolerance is reached
 - Iterates for each excitation b
 - More sensitive to mesh quality, reverts to direct solver if it fails to converge

$$\nabla \times E = - \frac{\partial B}{\partial t}$$

$$\nabla \times H = J + \frac{\partial D}{\partial t}$$

$$\nabla \cdot D = \rho$$

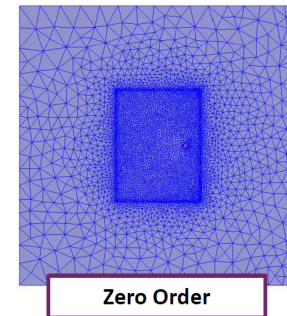
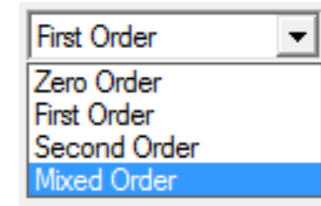
$$\nabla \cdot B = 0$$

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ 0 & a_{22} & a_{23} & a_{24} \\ 0 & 0 & a_{33} & a_{34} \\ 0 & 0 & 0 & a_{44} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \end{bmatrix}$$

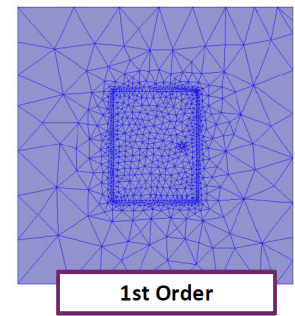


FEM Basis Functions

- ▶ Basis functions are n-order polynomials that describe how E-field varies along mesh elements edge, face, or volume
- ▶ Hierarchical basis functions
 - Zero **or** first **or** second order basis functions
 - Higher-order elements have increased accuracy but more unknowns (6, 20, 45)
- ▶ Mixed order basis functions
 - Zero **and** first **and** second order basis functions
 - hp-FEM method refines element order p and element size h
 - Automatically distributes element order based on element size to optimize use of resources
- ▶ Choice of ideal basis function is problem dependent
 - Mixed order efficiency is comparable to or better than best of single order basis functions



Zero Order



1st Order

Order	Tetrahedra	RAM	Solution Time	Adaptive Passes
0	449,445	2.9 GB	21 min	16
1 st	28,559	681 MB	2.5 min	11
2 nd	20,782	1.8 GB	9.5 min	9
Mixed	17,385	355MB	1.3 min	11



Fields Calculator

- ▶ Tool for performing math operations on saved fields
 - E, H, J, and Poynting data available
 - Geometric, complex, vector, and scalar data
 - Perform operations using model or non-model geometry
 - Generate numerical, graphical, geometrical, or exportable data
- ▶ Reverse Polish notation
- ▶ Frequently used expressions can be included in user library and loaded into any project
 - Eliminates need to re-create expressions used across projects

Named expressions
Context selection

Named Expressions

Name	Expression
Mag_E	Mag(AtPhase(Smooth(<Ex,Ez>, Phase)))
Mag_H	Mag(AtPhase(Smooth(<Hx,Hy,Hz>, Phase)))
Mag_Jvol	Mag(AtPhase(Smooth(<Jx,Jy,Jz>, Phase)))
Mag_Jsurf	Mag(AtPhase(<Jsurf,Jsurf,Jsurf>, Phase))
ComplexMag_E	Mag(CmplxMag(<Ex,Ez>))
ComplexMag_H	Mag(CmplxMag(<Hx,Hy,Hz>))
ComplexMag_Jvol	Mag(CmplxMag(<Jx,Jy,Jz>))
ComplexMag_Jsurf	Mag(CmplxMag(<Jsurf,Jsurf,Jsurf>))

Context Selection

Solution: Setup1: LastAdaptive
 Field Type: Fields
 Freq: 20GHz
 Phase: 0deg

Data Stack

Vol: Volume(Waveguide)
 ScI: Real(Mag(Poynting))
 ScISrf: SurfaceValue(Surface(GlobalXY), Dot(CmplxMag(<Ex,Ez>), SurfaceNormal))
 CVC: <Hx,Hy,Hz>

Stack operations

Push Pop RlUp RlDn Exch Clear Undo

Calculator functions

Quantity: +, -, 1/x, Pow, Mag, Value, Eval, Write..., Export...
 Geometry: -, 1/x, Pow, Mag, Value, Eval, Write..., Export...
 Constant: ^, Pow, Mag, Value, Eval, Write..., Export...
 Number: /, sqrt, Dot, Export...
 Function: Neg, Trig, Cross, Divg, Out, Tangent, Normal, Unit Vec
 Geom Settings: Abs, d/d?, Divg, Out, Tangent, Normal, Unit Vec
 Read: Smooth, Min, Max, Ln, Log

$$\frac{1}{2} \iint_S \text{Re} \{ \vec{E} \times \vec{H}^* \} \cdot \vec{ds}$$

$$\frac{1}{2} \sigma \iiint_V |E|^2 dv$$



Keyboard Shortcuts

General Shortcuts

- **F1:** Help
- **F1 + Shift:** Context help
- **F4 + CTRL:** Close window
- **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 + O:** Cascade windows
- **CTRL + 1:** Tile windows horizontally
- **CTRL + 2:** Tile windows vertically

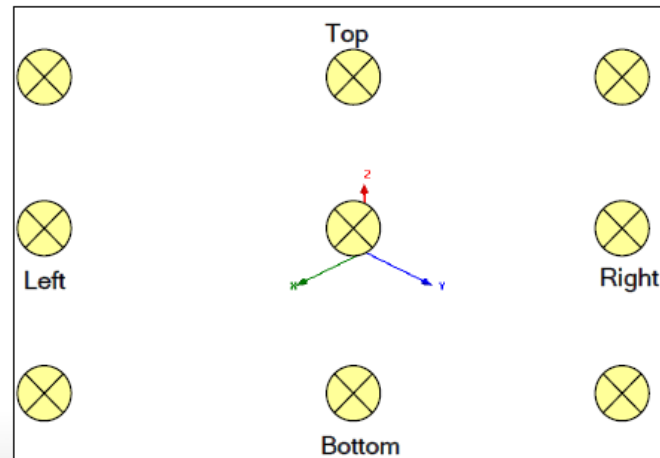
- **Alt + Double Click Left Mouse Button at points on screen:** Sets model projection to standard isometric projections (see diagram to the right).
- **Alt + Double Click Right Mouse Button at points on screen:** give the nine opposite projections.

3D Modeller Shortcuts

- **B:** Select face/object behind current selection
- **F:** Face select mode
- **O:** Object select mode
- **Hold X:** Curser movement restricted to x direction
- **Hold Y:** Curser movement restricted to y direction
- **Hold Z:** Curser movement restricted to z direction
- **CTRL + A:** Select all visible objects
- **CTRL + SHIFT + A:** Deselect all objects
- **CTRL + D:** Fit view

- **CTRL + Left Mouse Click:** 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

Predefined View Angles





Cylindrical Cavity Example



Cylindrical Cavity Example

▶ Empty copper cavity

- Radius = 21 cm
- Height = 100 cm

▶ Expected results for TM_{010} mode

- $f_R = 546.42$ MHz
- Q-factor = 61,391 (Li and Jiang, 2006)
- Form factor $C = 0.69$ (Peng *et al.*, 2000)
- Form factor $C = 0.692$ (Stern *et al.*, 2015)

$$f_{010} = \frac{c}{2\pi\sqrt{\mu_r\epsilon_r}} \sqrt{\left(\frac{p'_{01}}{r}\right)^2 + \left(\frac{0 \times \pi}{d}\right)^2}$$

$$f = \frac{c}{2.61r}$$

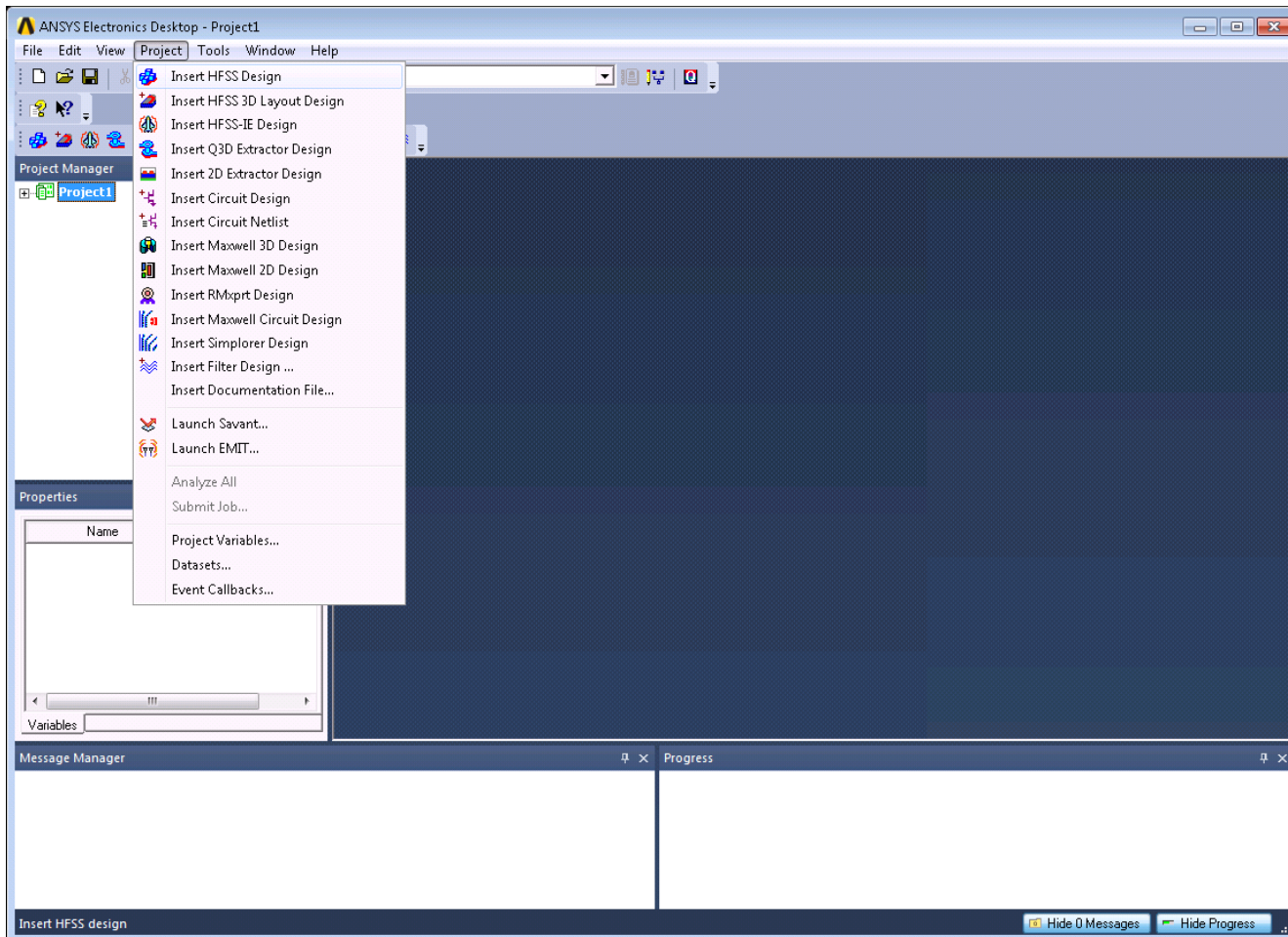
$$Q_u = \left(\frac{H}{R+H}\right)\left(\frac{R}{\delta}\right) \quad Q_0 = Q_c = \frac{2V}{S\sqrt{\frac{2}{\omega\mu\sigma}}}$$

$$C = \frac{\left| \int_V d^3x \vec{E}_\omega \cdot \vec{B}_0 \right|^2}{B_0^2 V \int_V d^3x \epsilon |\vec{E}_\omega|^2}$$



1: Create HFSS Project

- ▶ Insert into Electronics Desktop using **Project > Insert HFSS Design**





2: Set Eigenmode Solution Type

- ▶ Select **HFSS** > **Solution Type** > **Eigenmode**

ANSYS Electronics Desktop - Project1 - HFSSDesign1 - 3D Modeler - [Project1 - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Project Manager

- Project1*
- HFSSDesign1 (DrivenModal)
- Definitions

Properties

Name	Value	Unit	Evaluate
------	-------	------	----------

Variables

Message Manager

Progress

Solution Type: Project1 - HFSSDesign1

Solution Types

- Modal
- Terminal
- Transient
- Eigenmode

Driven Options

- Network Analysis Composite Excitation
- Auto-Open Region
- Save as default

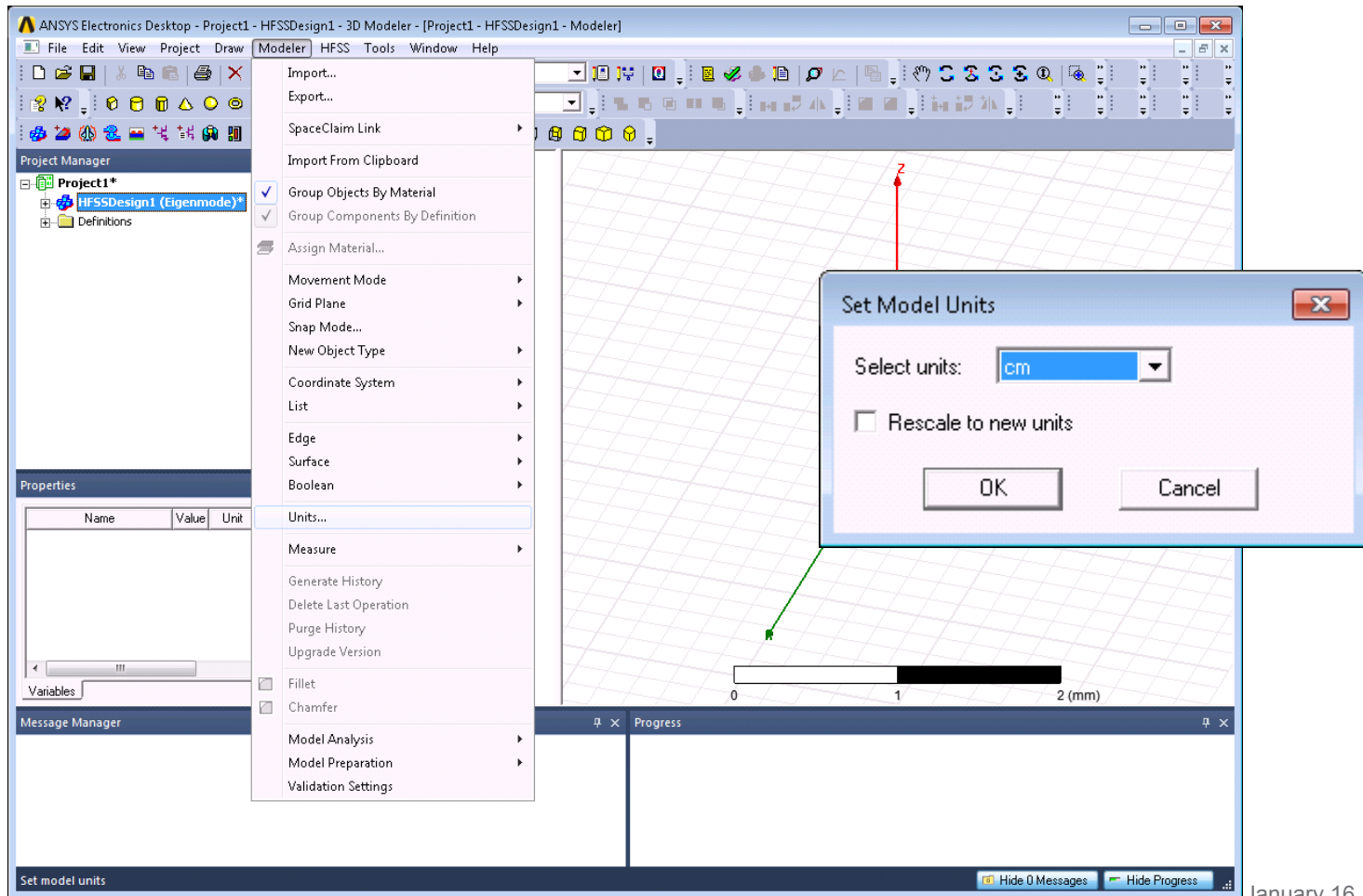
OK Cancel

Select solution type. Hide 0 Messages Hide Progress



3: Set Model Units

▶ Select **Modeler** > **Units** > **cm**





4: Set Dialog Data Entry Mode

▶ Select **Tools** > **Options** > **General Options** > **3D Modeler** > **Drawing** > **Dialog**

The screenshot displays the ANSYS Electronics Desktop interface. The **Tools** menu is open, and the path **Options** > **General Options** > **3D Modeler** > **Drawing** > **Dialog** is highlighted. The **Options** dialog box is open, showing the **Drawing Data Entry Mode** section with the **Dialog** radio button selected. The **Relative Coordinate System Creation Mode** section has **Axis/Position** selected. The **Polyline Creation** section has **Automatically cover closed polylines** checked. The **Show measure dialog during drawing** checkbox is also checked. The **Edit properties of new primitives** checkbox is unchecked. The **OK** and **Cancel** buttons are visible at the bottom of the dialog. The background shows the 3D Modeler workspace with a grid and a green line.



5: Set Default Transparency of 0.7

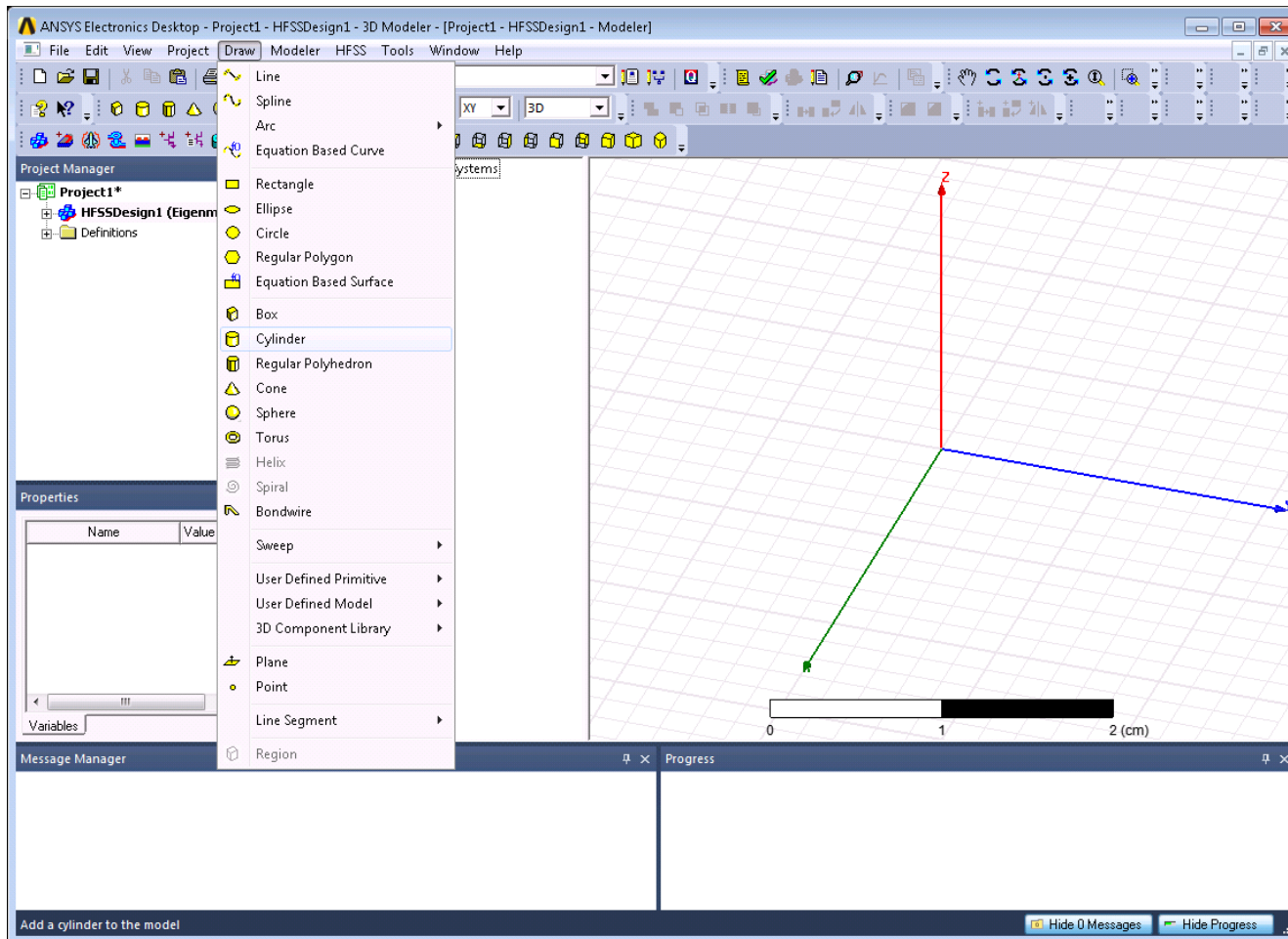
▶ Select **Tools > Options > General Options > Display > Rendering**

The screenshot displays the ANSYS Electronics Desktop interface. The **Tools** menu is open, and the **Options** sub-menu is selected. Within **Options**, the **General Options...** dialog box is open. In the **General Options** dialog, the **Display** section is expanded, and the **Rendering** sub-section is selected. The **Rendering Defaults** section shows the **Default transparency:** slider set to 0.7. The **Object Visualization** section shows the **Outline contrast:** slider set to 0.5. The **Message Manager** at the bottom shows the message "Set general options".



6: Create Parameterized Cavity

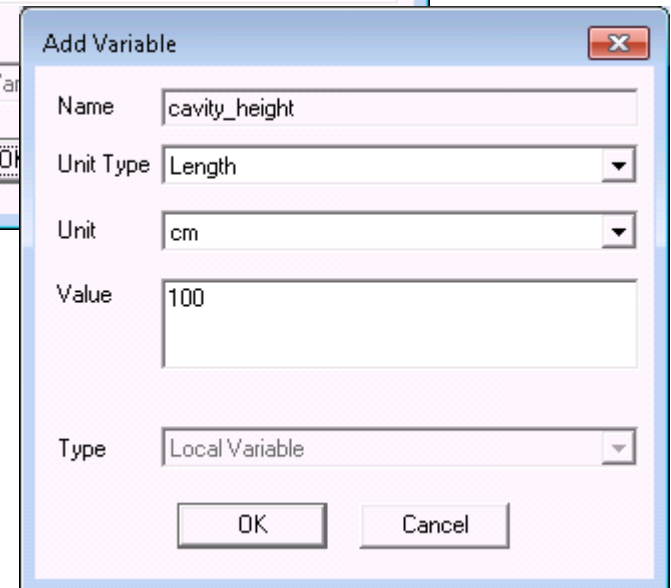
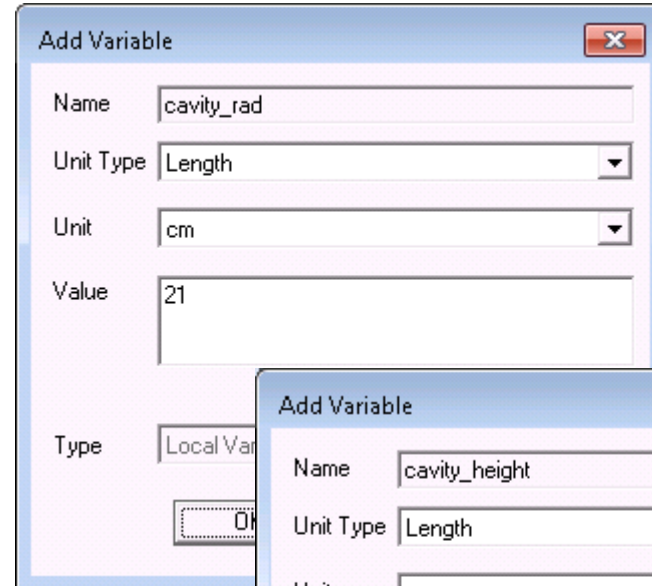
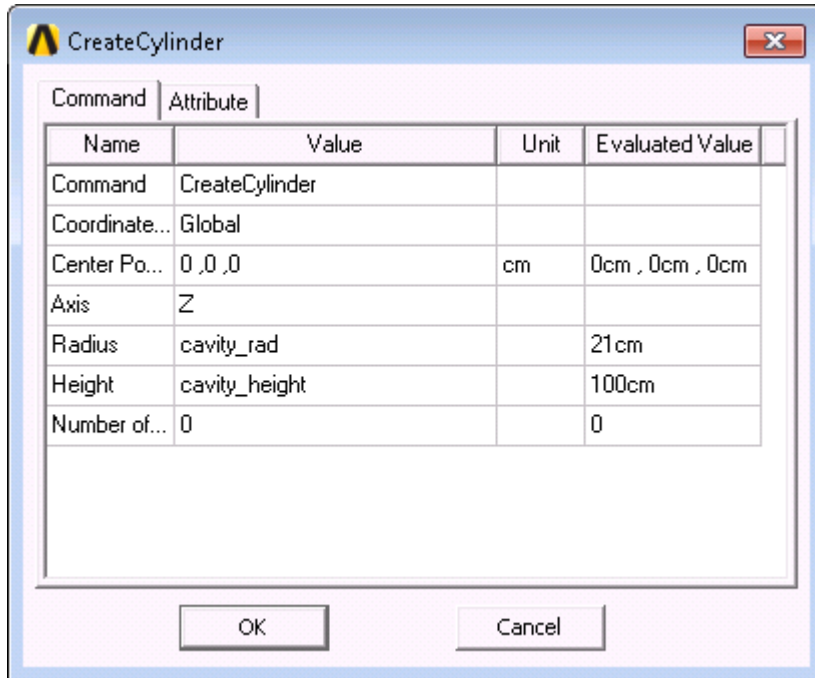
▶ Select **Draw** > **Cylinder**





6: Create Parameterized Cavity

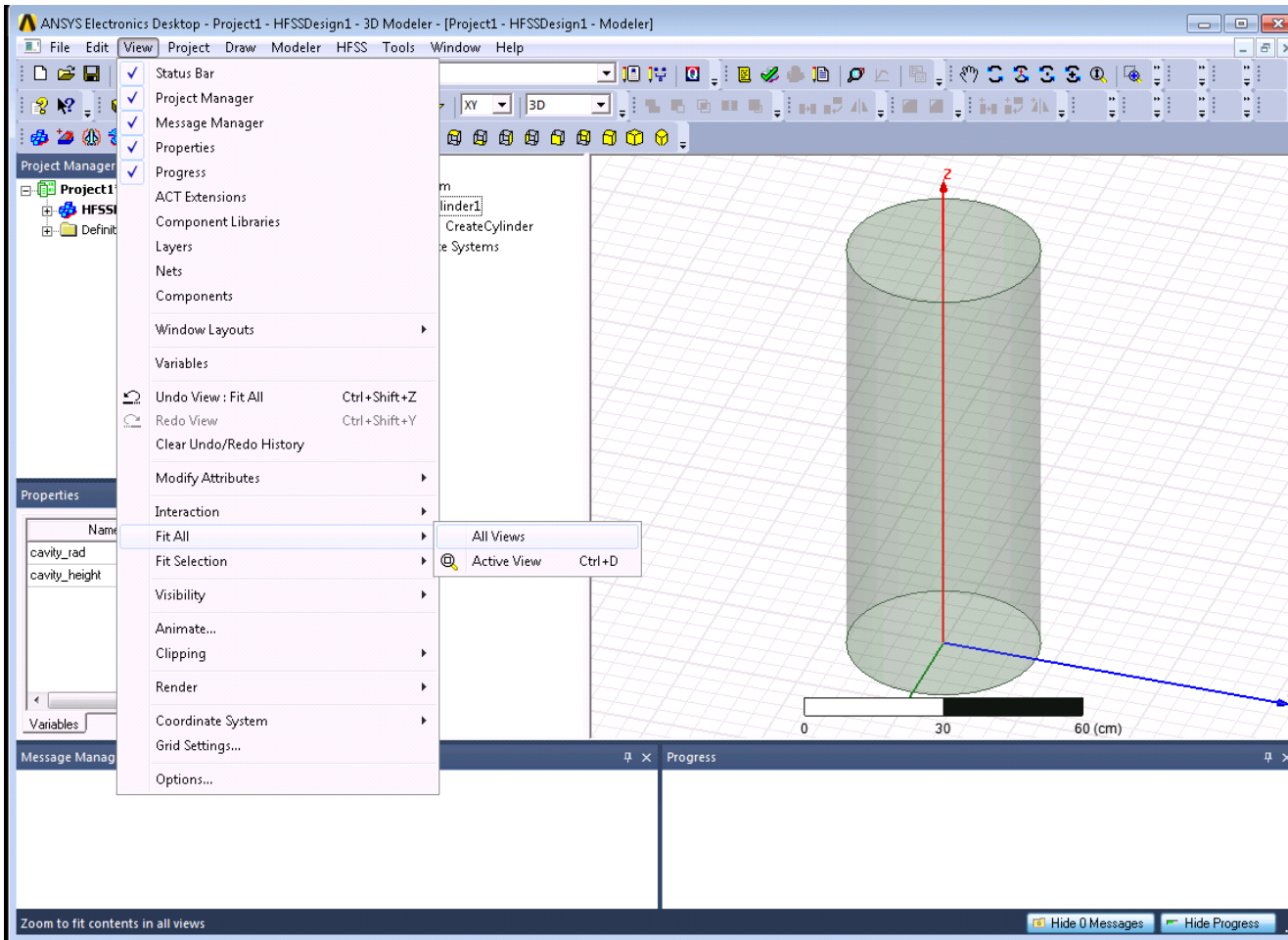
- ▶ Cavity_rad = 21 cm
- ▶ Cavity_height = 100 cm





6: Create Parameterized Cavity

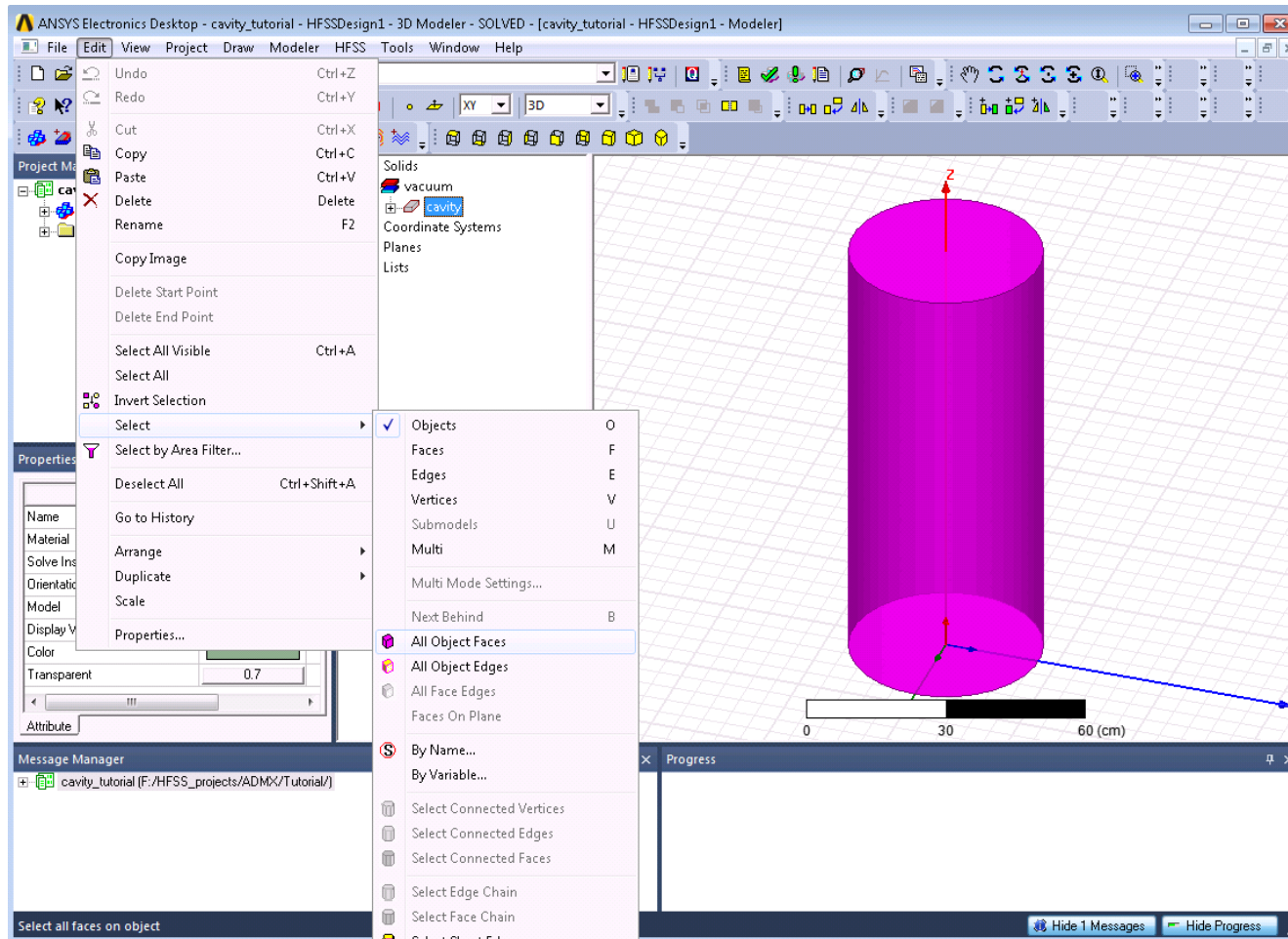
- ▶ Fit cavity to view using **View > Fit All > All Views**





7: Assign Cavity Wall Conductivity

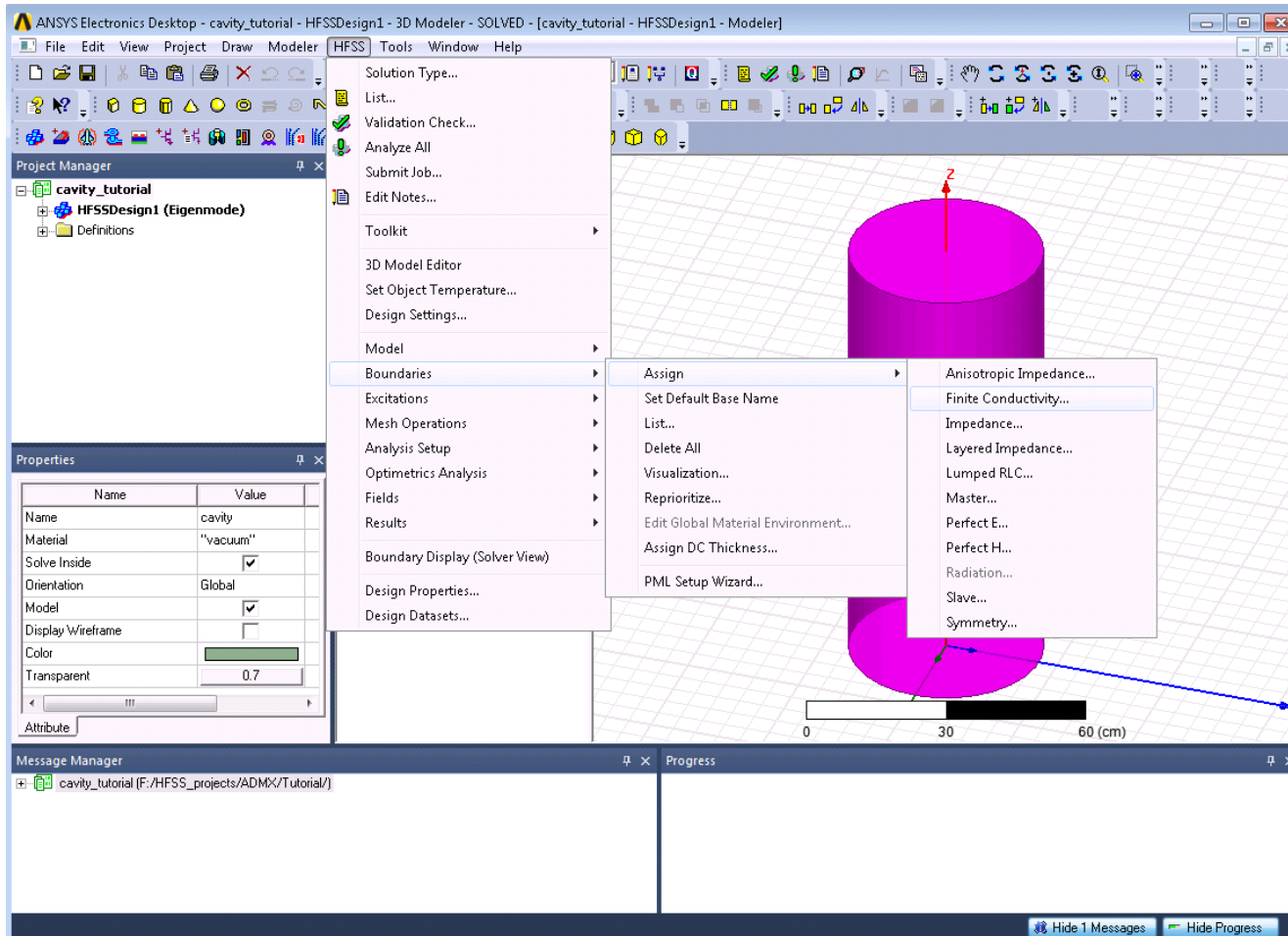
- ▶ Select cavity in 3D modeler tree and **Edit > Select > All Object Faces**





7: Assign Cavity Wall Conductivity

- ▶ Select **HFSS** > **Boundaries** > **Assign** > **Finite Conductivity**





7: Assign Cavity Wall Conductivity

- ▶ Enter name “cavity_walls” and use default 5.8E7 S/m

Finite Conductivity Boundary

Name:

Parameters

Conductivity: Siemens/m

Relative Permeability:

Use Material:

Infinite Ground Plane

Advanced

Surface Roughness Model: Grosjean Huray

Surface Roughness:

Hall-Huray Surface Ratio:

Set DC Thickness

One sided Object is on outer boundary

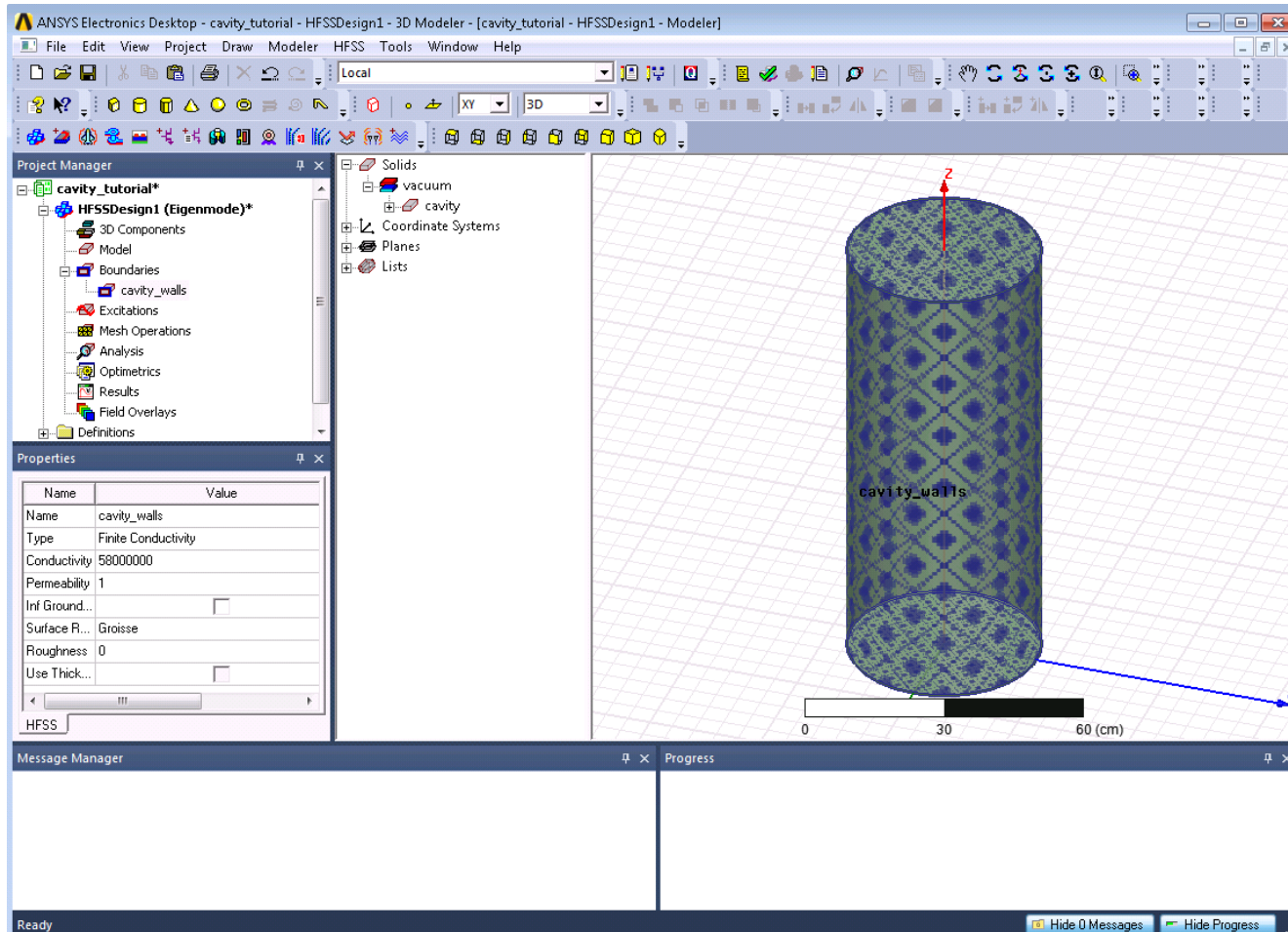
Two sided

Use classic infinite thickness model



7: Assign Cavity Wall Conductivity

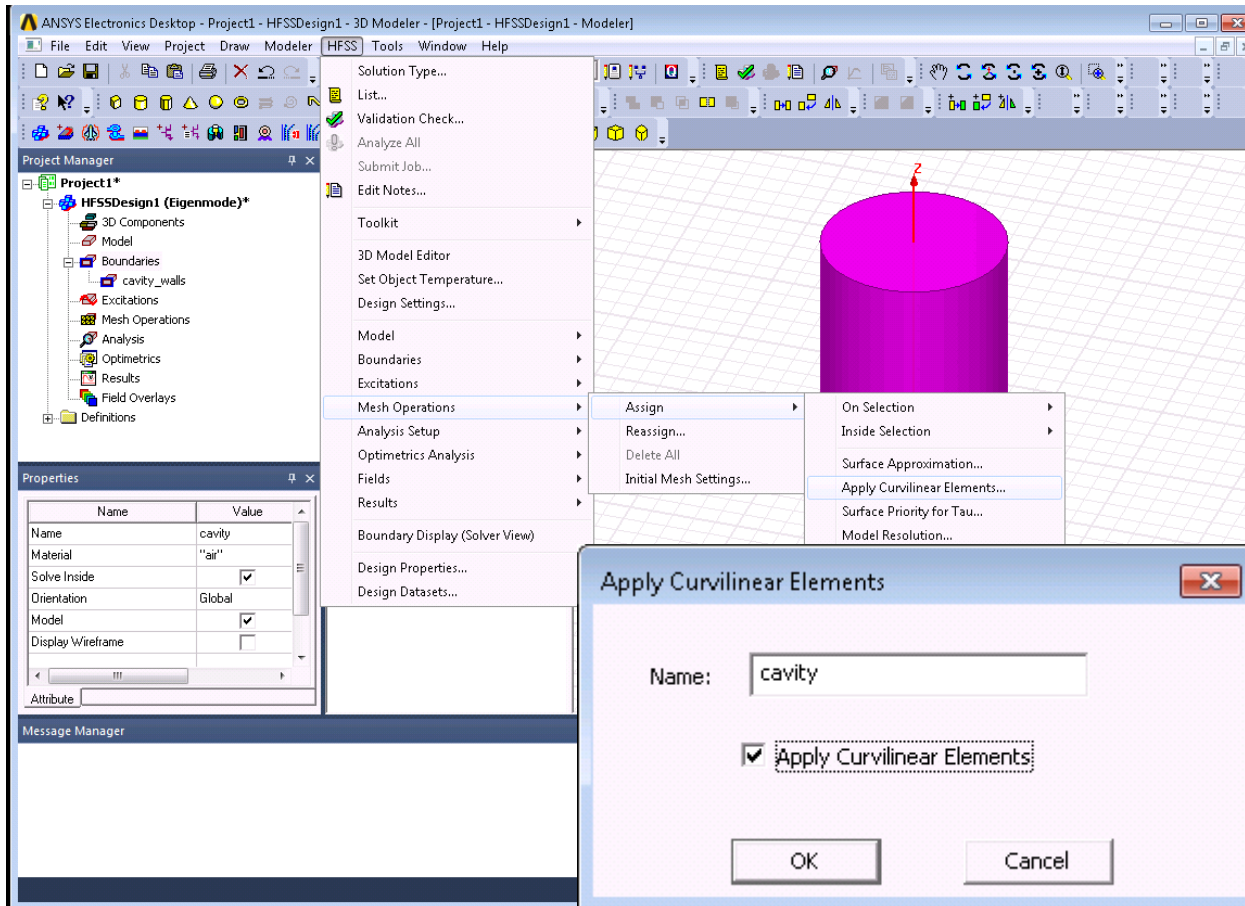
- ▶ Should have boundary condition as shown here





8: Apply Curvilinear Mesh Elements

- ▶ Select cavity in 3D modeler tree and apply curvilinear elements
- ▶ Select **HFSS** > **Mesh Operations** > **Assign** > **Apply Curvilinear Elements**





8: Apply Curvilinear Mesh Elements

- ▶ Can also apply curvilinear elements as global setting
- ▶ Right-click **Mesh Operations** > **Initial Mesh Settings**

ANSYS Electronics Desktop - cavity_tutorial - HFSSDesign1 - 3D Modeler - [cavity_tutorial - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Local

XY 3D

Project Manager

cavity_tutorial*

HFSSDesign1 (Eigenmode)*

3D Components

Model

Boundaries

cavity_walls

Excitations

Mesh Operations

Analysis

Optimize

Results

Field Overlays

Definitions

Solids

vacuum

cavity

Coordinate Systems

Planes

Lists

Assign

List...

Delete All

Initial Mesh Settings...

Properties

Message Manager

Progress

Initial Mesh Settings

General | Advanced

Mesh Method

Auto TAU Classic

Apply curvilinear elements to all curved surfaces

Curved Surface Meshing

Use Slider Manual Settings

Coarse Resolution fine

Small Mesh Size Large

Save as Default

OK Cancel

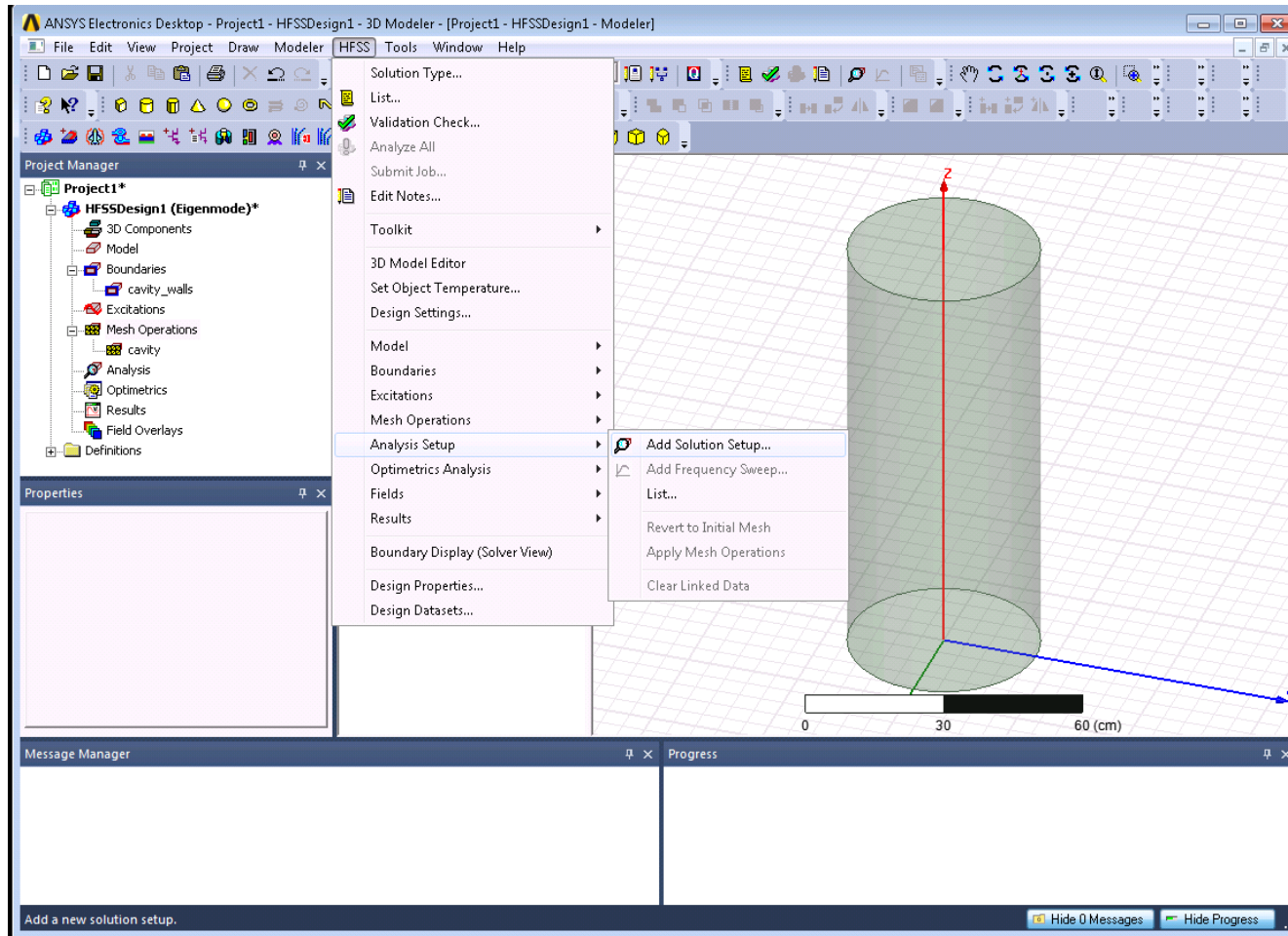
Hide 0 Messages Hide Progress

January 16, 2017 | 33



9: Add Solution Setup

▶ Select **HFSS** > **Analysis Setup** > **Add Solution Setup**





9: Add Solution Setup

- ▶ Enter Minimum frequency = 540 MHz, Number of Modes = 3, Maximum Number of Passes = 12, Max Delta Frequency / Pass = 2%, Minimum Passes = 4

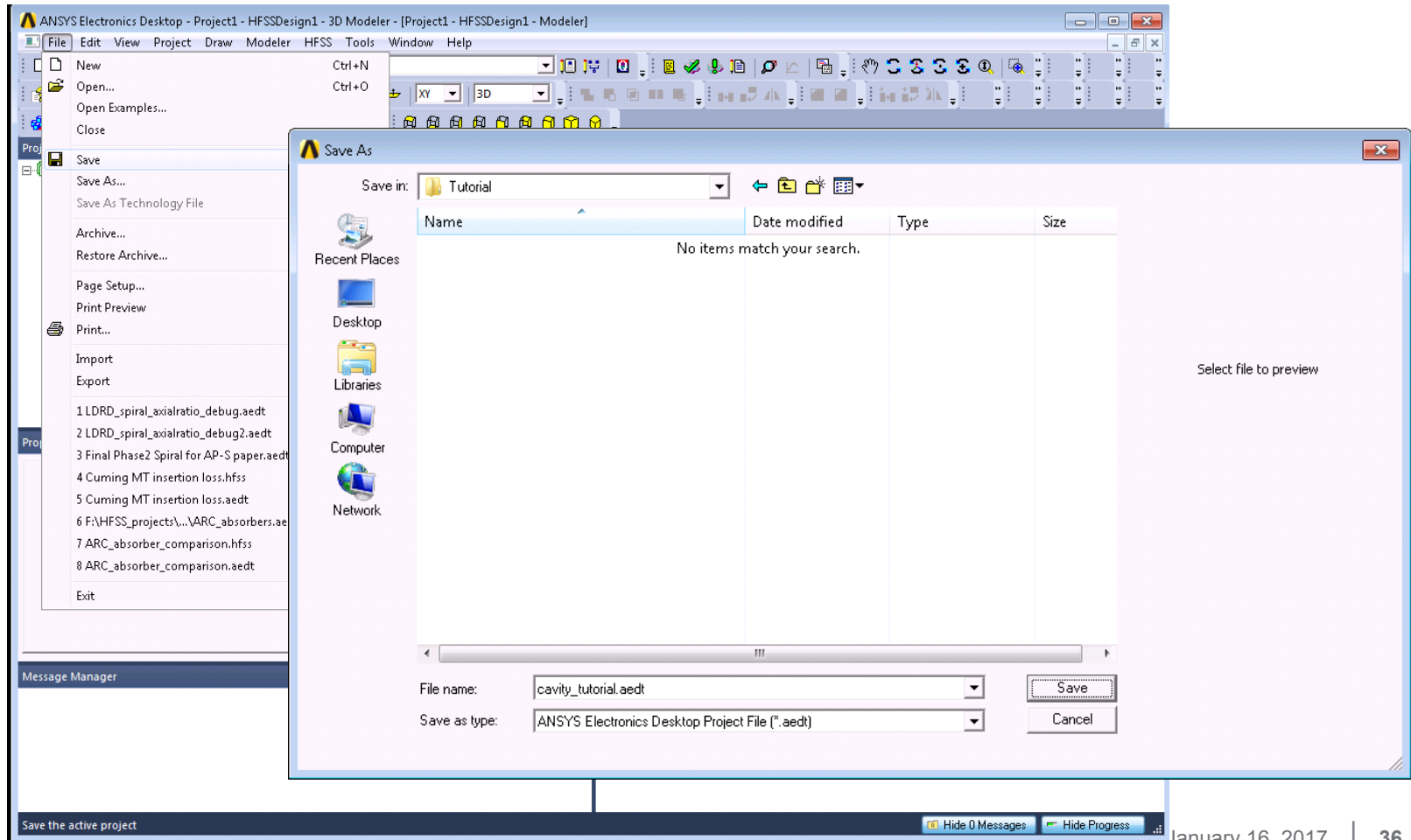
The dialog box 'Eigen Solution Setup' is shown with the 'General' tab selected. The 'Setup Name' is 'Setup1'. The 'Enabled' checkbox is checked. The 'Minimum Frequency' is set to 540 MHz. The 'Number of Modes' is 3. In the 'Adaptive Solutions' section, 'Maximum Number of Passes' is 12 and 'Maximum Delta Frequency Per Pass' is 2%. The 'Converge on Real Frequency Only' checkbox is unchecked. There are 'Use Defaults', 'HPC and Analysis Options...', 'OK', and 'Cancel' buttons.

The dialog box 'Eigen Solution Setup' is shown with the 'Options' tab selected. In the 'Initial Mesh Options' section, 'Do Lambda Refinement' is checked with a 'Lambda Target' of 0.2 and 'Use Default Value' is checked. 'Use Free Space Lambda' is unchecked. In the 'Adaptive Options' section, 'Maximum Refinement Per Pass' is 30%, 'Maximum Refinement' is 1000000, 'Minimum Number of Passes' is 4, and 'Minimum Converged Passes' is 1. In the 'Solution Options' section, 'Order of Basis Functions' is 'First Order'. There are 'Use Defaults', 'OK', and 'Cancel' buttons.



10: Save Project

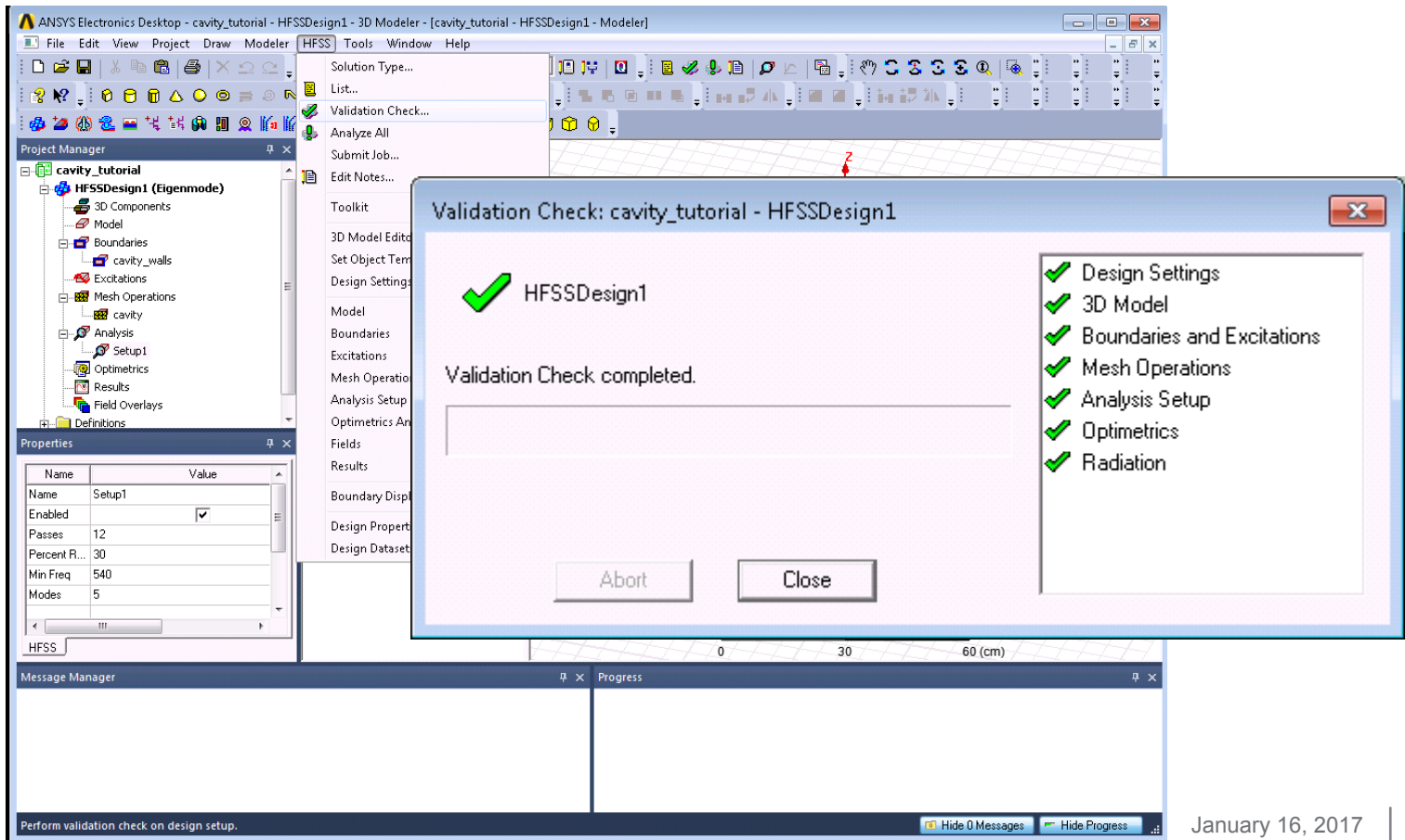
- ▶ Select **File > Save** and save project as “cavity_tutorial.aedt”





11: Perform Validation Check

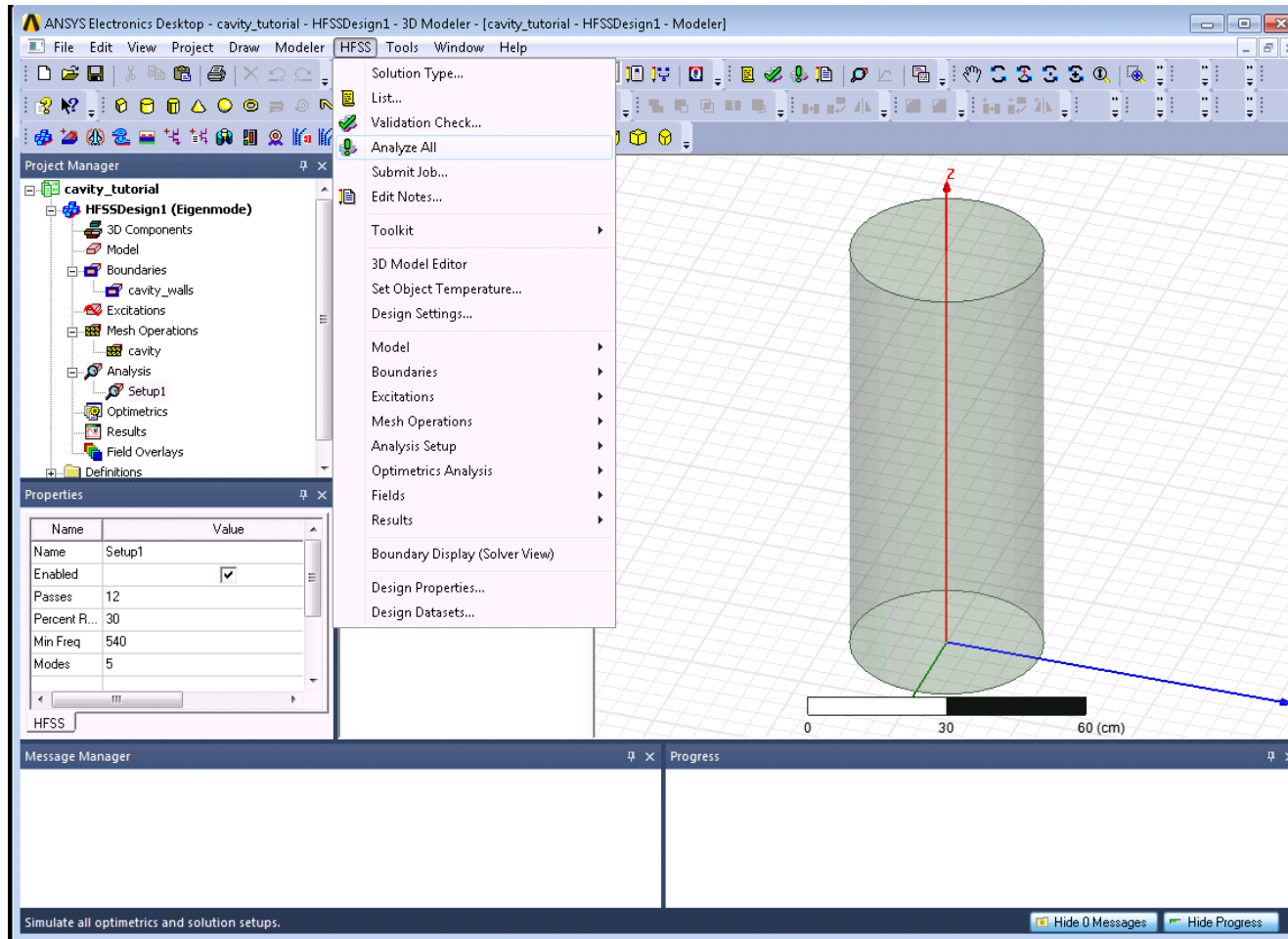
- ▶ Select **HFSS** > **Validation Check**
- ▶ Confirms that required steps to solve model have been performed





12: Solve Model

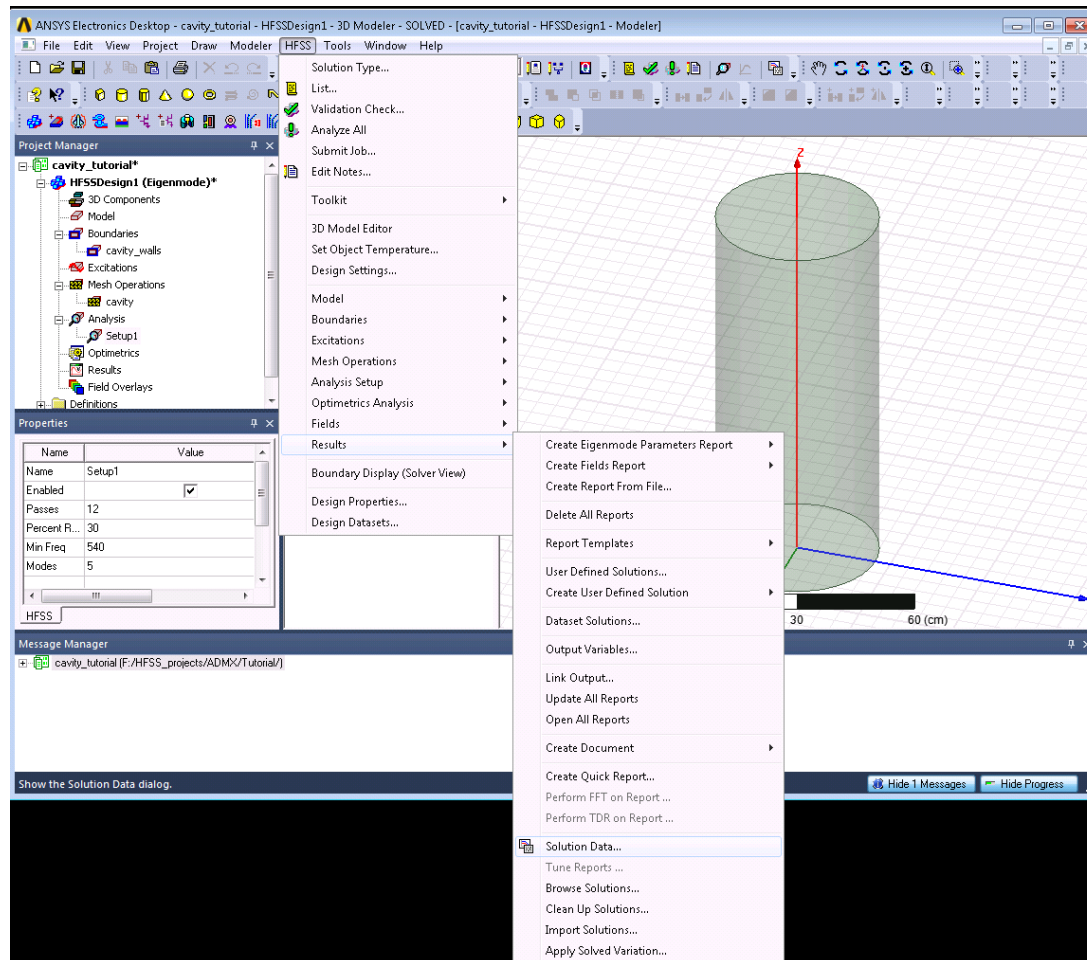
▶ Select **HFSS** > **Analyze All**





13: View Solution Data

▶ Select **HFSS** > **Results** > **Solution Data**





13: View Solution Data

- ▶ Select ***Eigenmode Data*** tab to view modal frequencies and Q-factors

TM₀₁₀
TM₀₁₁
TE₁₁₃

Solutions: cavity_tutorial - HFSSDesign1

Simulation: Setup1 LastAdaptive

Design Variation: cavity_height='100cm' cavity_rad='21cm'

Profile | Convergence | **Eigenmode Data** | Mesh Statistics

Solved Modes Export

Eigenmode	Frequency (GHz)	Q
Mode 1	0.546418 +j 4.45170e-006	61371.8
Mode 2	0.566601 +j 5.31945e-006	53257.4
Mode 3	0.614229 +j 4.32020e-006	71087.9

Close



13: View Solution Data

- ▶ Select **Convergence** tab to view adaptive pass information

Simulation: Setup1

Design Variation: cavity_height='100cm' cavity_rad='21cm'

Profile | **Convergence** | Eigenmode Data | Mesh Statistics

Number of Passes	Pass Number	Solved Elements	Max Delta Freq. %
Completed 4	1	2404	N/A
Maximum 12	2	3129	0.037594
Minimum 4	3	3963	0.018768
	4	4559	0.0047705

Max Delta Freq. %
Target 2
Current 0.0047705

View: Table Plot

Export..

CONVERGED

Consecutive Passes
Target 1
Current 3

Default Settings
Save Defaults Clear Defaults

Close



13: View Solution Data

- ▶ Select **Profile** tab to view run log file (21 sec runtime)

Solutions: cavity_tutorial - HFSSDesign1

Simulation: Setup1

Design Variation: cavity_height='100cm' cavity_rad='21cm'

Profile | Convergence | Eigenmode Data | Mesh Statistics

Task	Real Time	CPU Time	Memory	Information
Field Recovery	00:00:00	00:00:03	254 M	Disk = 3826 KBytes, 3 computed eigenmodes
Adaptive Pass 4				Eigenmode Solution
Mesh (volume, adapti...	00:00:01	00:00:02	38.8 M	4559 tetrahedra
Simulation Setup	00:00:00	00:00:00	45.5 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	102 M	Disk = 0 KBytes, 4559 tetrahedra
EigenSolver DCS8	00:00:02	00:00:13	290 M	Disk = 184 KBytes, matrix size 30744, matrix bandwidth 2
Field Recovery	00:00:00	00:00:03	290 M	Disk = 4334 KBytes, 3 computed eigenmodes
				Adaptive Passes converged
Solution Process				Elapsed time : 00:00:21 , Hfss ComEngine Memory : 55.6
Total	00:00:12	00:01:00		Time: 01/06/2017 10:59:51, Status: Normal Completion

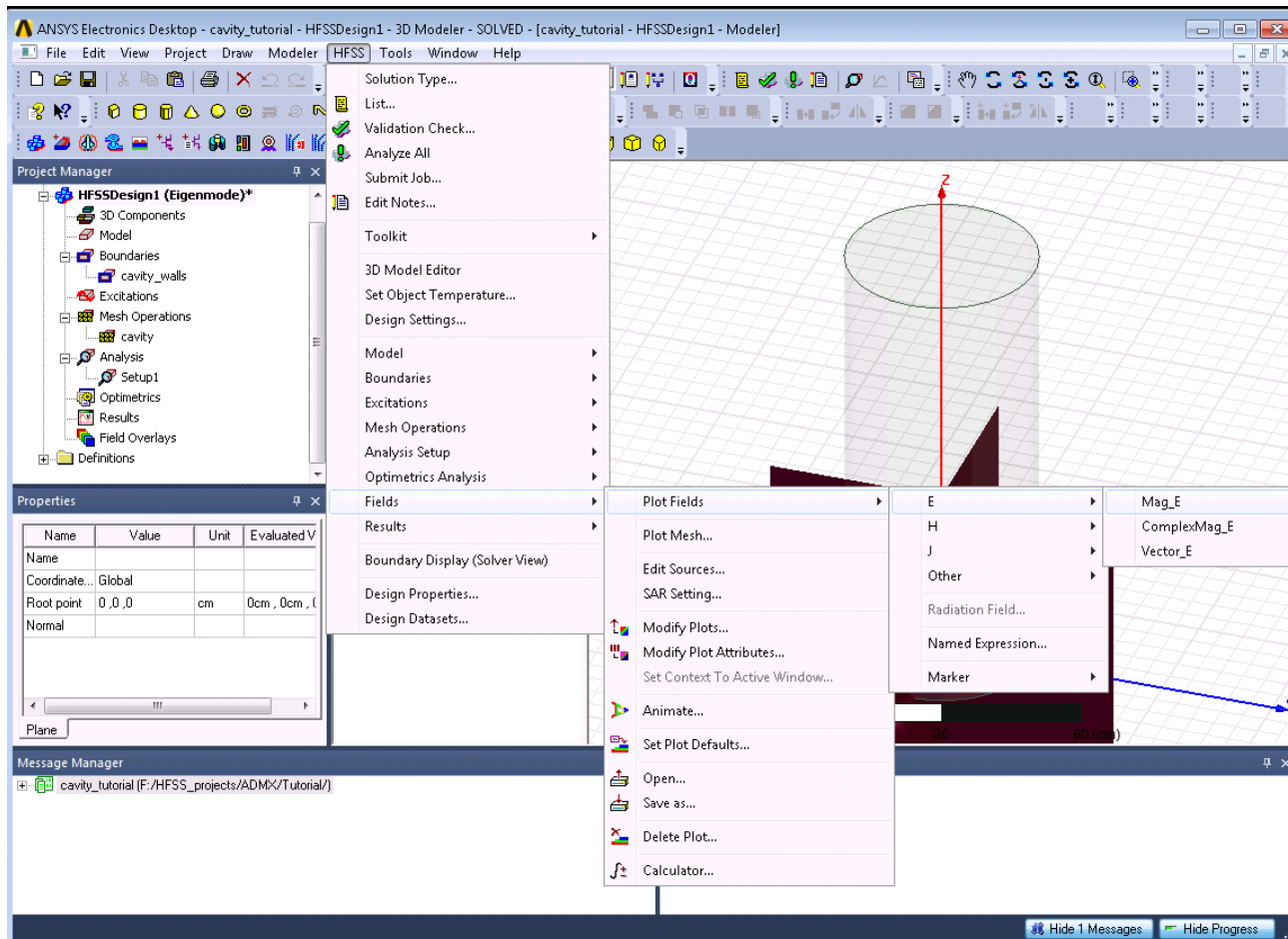
Export...

Close



14: View E-Field Phase Animation

- ▶ Select XZ and YZ planes in 3D modeler tree and select **HFSS** > **Fields** > **Plot Fields** > **E** > **Mag_E**





14: View E-Field Phase Animation

- ▶ Select Done to create plot of electric field magnitude

The screenshot displays the ANSYS Electronics Desktop interface. The 'Create Field Plot' dialog box is open, showing the following settings:

- Specify Name: (unchecked), Mag_E2
- Specify Folder: (unchecked), E Field
- Design: HFSSDesign1
- Context: Solution: Setup1: LastAdaptive, Field Type: Fields, Intrinsic Variables: Phase: 0deg
- Quantity: Mag_E (selected)
- In Volume: cavity, AllObjects
- Buttons: Done, Cancel, Save As Default
- Options: Plot on surface only, Streamline

The background shows a 3D plot of the electric field magnitude (E Field [V/m]) in a cylindrical cavity. The plot uses a color scale from blue (low magnitude) to red (high magnitude). A legend on the left of the plot shows the following values:

Color	Value [V/m]
Red	2.4732E+006
Orange	2.3083E+006
Yellow	2.1434E+006
Light Green	1.9785E+006
Green	1.8137E+006
Light Blue	1.6488E+006
Blue	1.4839E+006
Dark Blue	1.3190E+006
Very Dark Blue	1.1542E+006
Black	9.8929E+005
Dark Grey	8.2441E+005
Medium Grey	6.5953E+005
Light Grey	4.9466E+005
White	3.2978E+005
Very Light Grey	1.6491E+005
White	2.9691E+001

The plot also includes a scale bar at the bottom indicating 0, 30, and 60 cm.



14: View E-Field Phase Animation

- ▶ Right-click on Mag_E1 plot to animate phasor field

The screenshot shows the ANSYS Electronics Desktop interface. The 'Project Manager' tree on the left shows the 'E Field' plot selected. The 'Properties' panel shows the 'Mag_E1' plot. A context menu is open over the 'Mag_E1' plot, with the 'Animate...' option selected. The 'Setup Animation' dialog box is open, showing the following settings:

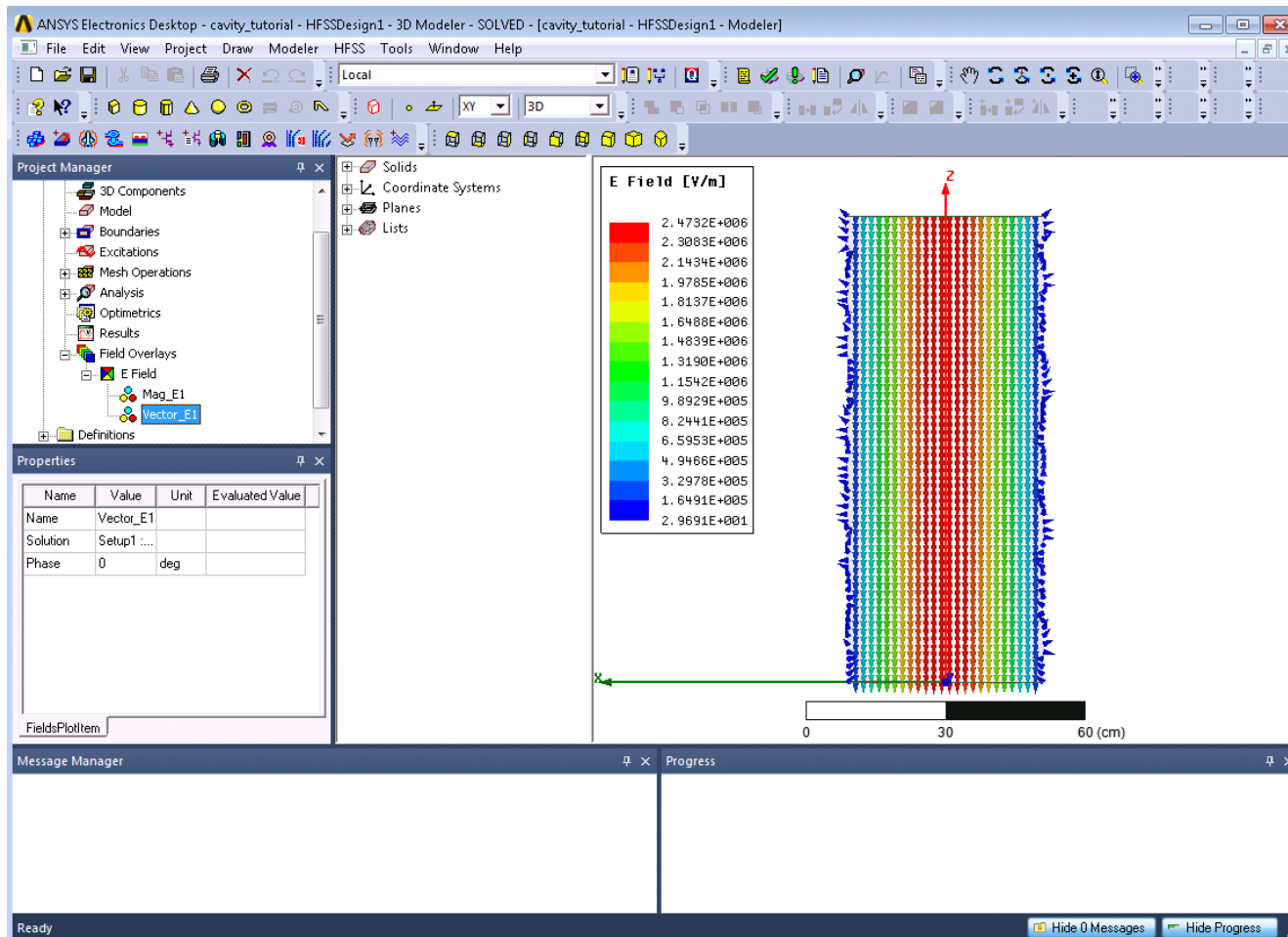
- Name: Animation1
- Description: (empty)
- Swept Variable: Design Point
- Swept variable: Phase
- Start: 0deg
- Stop: 170deg
- Steps: 17

The 'E Field [V/m]' plot shows a color scale from 2.4732E+0 to 2.9691E+0. The 'FieldsPlottItem' panel is visible at the bottom left.



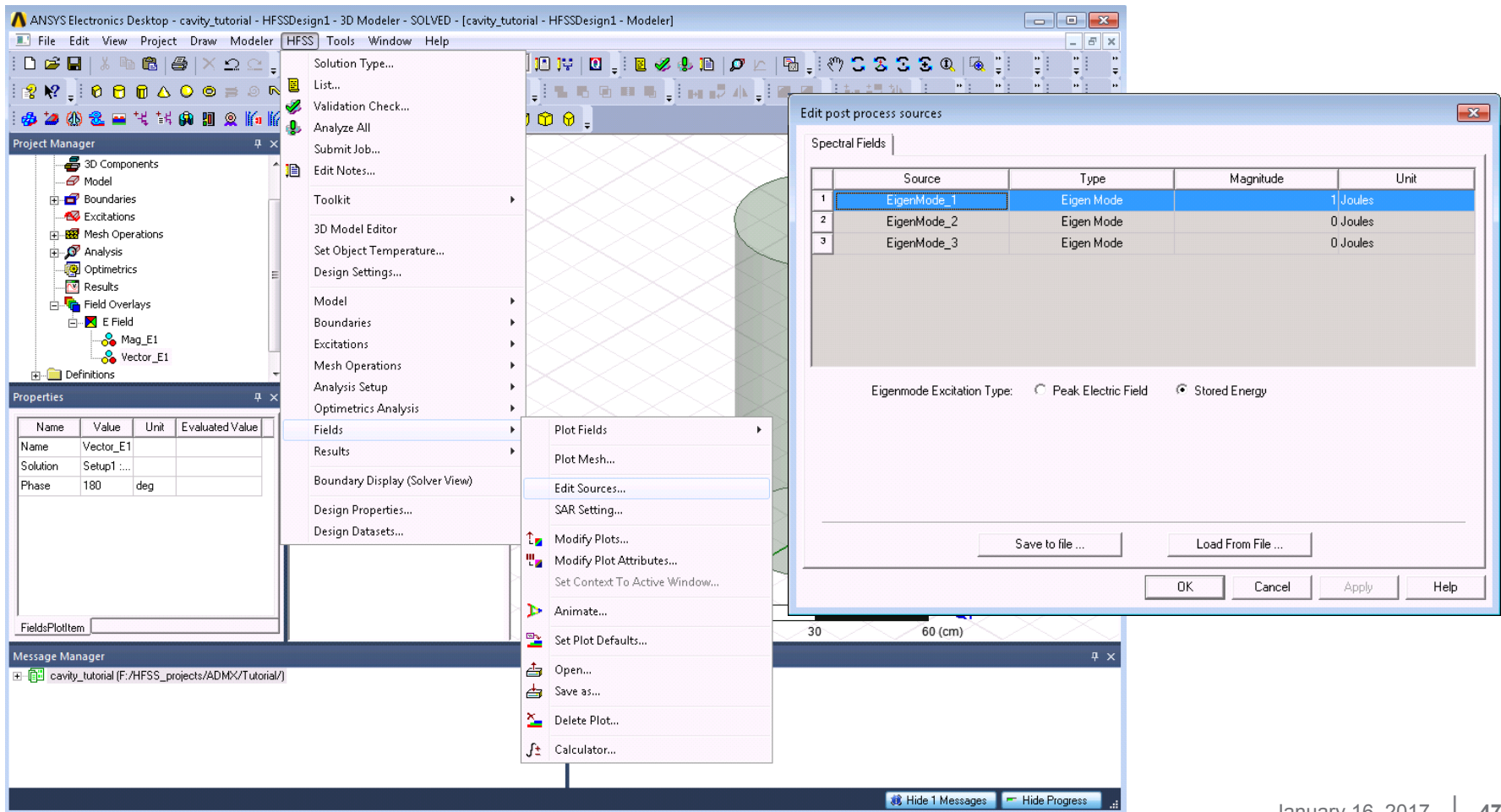
15: View E-Field Vector Animation

- ▶ Select XZ and YZ planes in 3D modeler tree and select **HFSS > Fields > Plot Fields > E > Vector_E**



How to Activate Mode of Interest for Field Plots and Calculations

► Select *HFSS* > *Fields* > *Edit Sources*



The screenshot shows the ANSYS Electronics Desktop interface with the HFSS application open. The 'Fields' menu is open, and the 'Edit Sources' option is selected. The 'Edit post process sources' dialog box is displayed, showing the 'Spectral Fields' table and the 'Eigenmode Excitation Type' options.

Source	Type	Magnitude	Unit
EigenMode_1	Eigen Mode	1	Joules
EigenMode_2	Eigen Mode	0	Joules
EigenMode_3	Eigen Mode	0	Joules

Eigenmode Excitation Type: Peak Electric Field Stored Energy

Buttons: Save to file ..., Load From File ..., OK, Cancel, Apply, Help



16: Calculate Form Factor

- ▶ Open field calculator using **HFSS > Fields > Calculator**

ANSYS Electronics Desktop - cavity_tutorial - HFSSDesign1 - 3D Modeler - SOLVED - [cavity_tutorial - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Solution Type...
List...
Validation Check...
Analyze All
Submit Job...
Edit Notes...
Toolkit
3D Model Editor
Set Object Temperature...
Design Settings...
Model
Boundaries
Excitations
Mesh Operations
Analysis Setup
Optimetrics Analysis

Project Manager

- cavity_tutorial*
- HFSSDesign1 (Eigenmode)*
 - 3D Components
 - Model
 - Boundaries
 - Excitations
 - Mesh Operations
 - Analysis
 - Optimetrics
 - Results
 - Field Overlays
 - E Field
 - Mag_E1

Properties

Name	Value	Unit	Evaluate
cavity_rad	21	cm	21cm
cavity_height	100	cm	100cm

Variables

Message Manager

- cavity_tutorial [F:\HFSS_projects\ADM\Tutorial\]

Fields Calculator

Named Expressions

Name	Expression
Mag_E	Mag(AIPhaseE)
Mag_H	Mag(AIPhaseH)
Mag_Jvol	Mag(AIPhaseJvol)
Mag_Jsurf	Mag(AIPhaseJsurf)

Context: HFSSDesign1
Solution: Setup1: LastAdaptive
Field Type: Fields
Phase: 0deg

Scal: 0.138539961037802
Scal: Integrate(Volume(cavity), 1)
Scal: 225741973236.581
Scal: Integrate(Volume(cavity), *(ComplexMag_E, ComplexMag_E))
Scal: -1.18070125088102
Scal: Integrate(Volume(cavity), Imag(ScalarZ(<Ex,Ey,Ez)))
Scal: -14.7122.81128389
Scal: Integrate(Volume(cavity), Real(ScalarZ(<Ex,Ey,Ez)))

Push Pop RIUp RIDn Exch Clear Undo

Input	General	Scalar	Vector	Output
Quantity	+	Vec?	Scal?	Value
Geometry...	-	1/x	Matl...	Eval
Constant	*	Pow	Mag	Write...
Number...	/	$\sqrt{\quad}$	Dot	Export...
Function...	Neg	Trig	Cross	
Geom Settings...	Abs	d/d?	Divg	
Read...	Smooth	\int	Curl	
Uoutput Vars	Complex	Min	Tangent	
	Domain	Max	Normal	
		Ln	Unit Vec	
		Log	X Form	

Done

Fields calculator

Hide 1 Messages Hide Progress

January 16, 2017 | 48



16: Calculate Form Factor

- ▶ Calculate integral of real(E_z)
 - Quantity > E
 - Scal? > ScalarZ
 - Complex > Real
 - Geometry > Volume > cavity
 - Integrate, Eval
- ▶ Calculate integral of imag(E_z)
 - Quantity > E
 - Scal? > ScalarZ
 - Complex > Imag
 - Geometry > Volume > cavity
 - Integrate, Eval
- ▶ Calculate integral of $|E|^2$
 - Copy ComplexMag_E to stack
 - Push
 - Multiply (*)
 - Geometry > Volume > cavity
 - Integrate, Eval

- ▶ Calculate cavity volume
 - Number -> 1
 - Geometry > Volume > cavity
 - Integrate, Eval
- ▶ Form factor = $(147123^2 + 1.1807^2) / (0.1385 * 225741973237) = 0.692$

$$C_E = \frac{\left| \int dV_c \vec{E}_c \cdot \vec{z} \right|^2}{V \int dV_c |E_c|^2}$$

The screenshot shows a 'Fields Calculator' window with a table of 'Named Expressions'.

Name	Expression
Surface_Force_Density	<SurfaceForceDensityx, SurfaceForceDensityy, SurfaceForceDensityz>
Ez	Abs(Integrate[Volume(Cylinder1), Real(ScalarZ(<Ex, Ey, Ez>))])
Etot	Integrate[Volume(Cylinder1), *(ComplexMag_E, ComplexMag_E)]
V	Integrate[Volume(Cylinder1), 1]
Form_Factor	/(*(Ez, Ez), *(Etot, V))



Cavity Simulation Results

- ▶ Good agreement between simulated and analytical results

Quantity	Calculation	Simulation	% Difference
Frequency	546.42 MHz	546.42 MHz	0.00%
Unloaded Q-factor	61,391	61,372	0.03%
Form Factor	0.692	0.692	0.00%

$$Q_0 = \frac{|\Gamma|}{2\Gamma} i$$

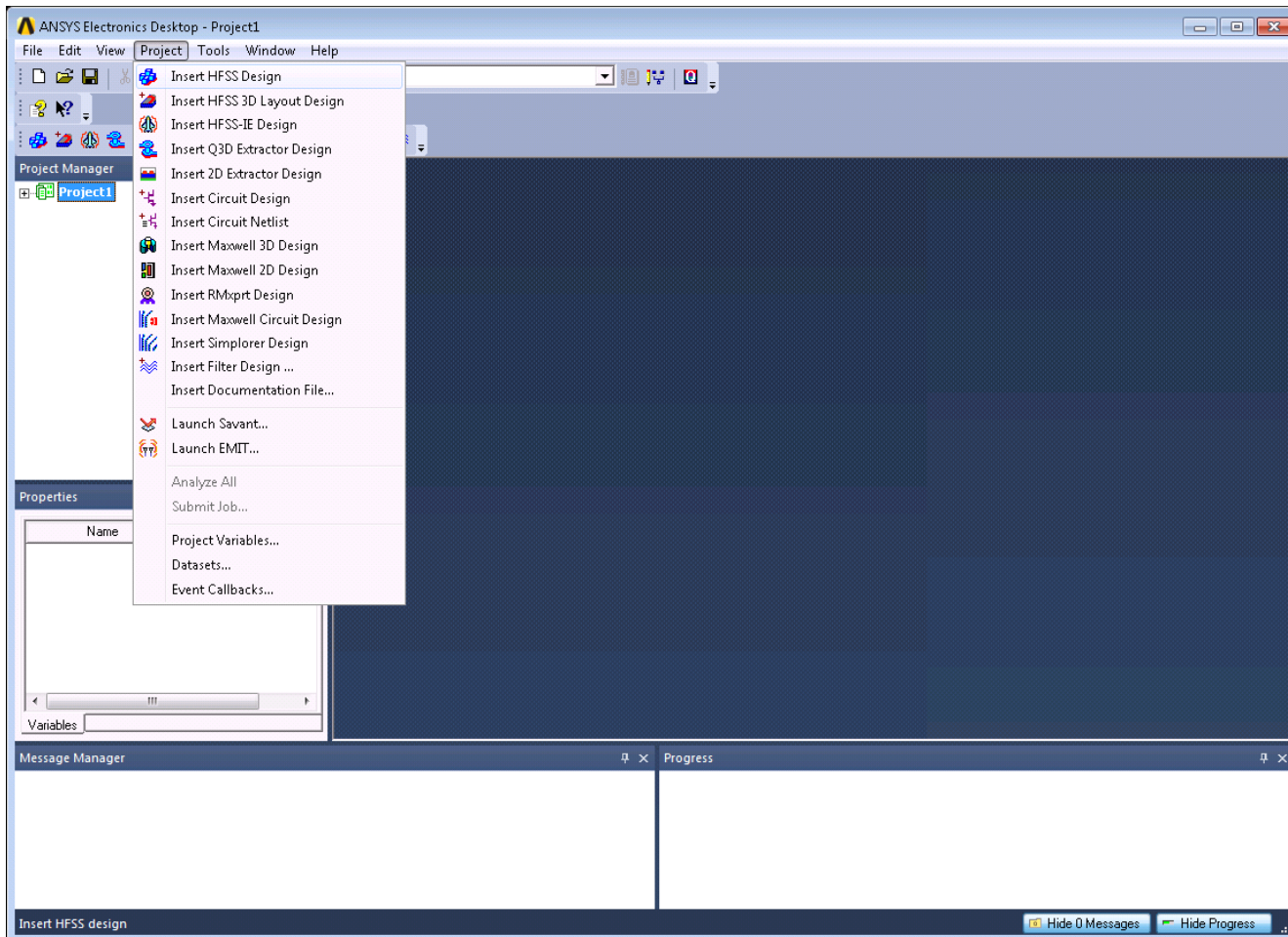


Dipole Antenna Example



1: Create HFSS Project

- ▶ Insert into Electronics Desktop using **Project > Insert HFSS Design**





2: Set Driven Modal Solution Type

- ▶ Select **HFSS** > **Solution Type** > **Modal**

ANSYS Electronics Desktop - Project1 - HFSSDesign1 - 3D Modeler - [Project1 - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Solution Type...
List...
Validation Check...
Analyze All
Submit Job...
Edit Notes...
Toolkit
3D Model Editor
Set Object Temperature...
Design Settings...
Model
Boundaries
Excitations
Hybrid
Mesh Operations
Analysis Setup
Optimetrics Analysis
Fields
Radiation
Results
Boundary Display (Solver View)
Design Properties...
Design Datasets...

Project Manager

Project1*
HFSSDesign1 (DrivenModal)
Definitions

Properties

Name	Value	Unit	Evaluate
------	-------	------	----------

Variables

Message Manager

Progress

Solution Type: Project1 - HFSSDesign1

Solution Types

- Modal
- Terminal
- Transient
- Eigenmode

Driven Options

- Network Analysis Composite Excitation
- Auto-Open Region

Save as default

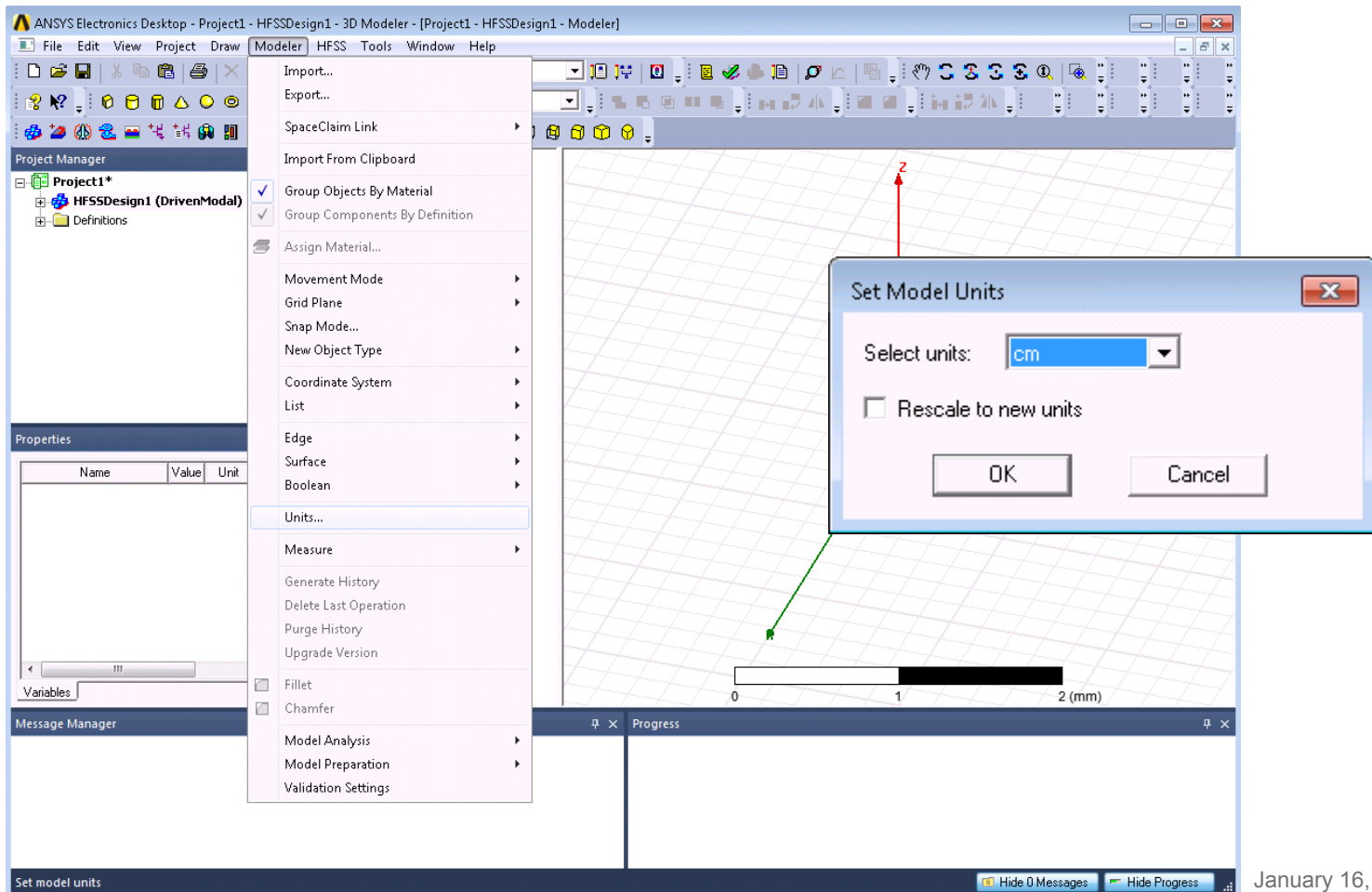
OK Cancel

Select solution type. Hide 0 Messages Hide Progress



3: Set Model Units

▶ Select **Modeler** > **Units** > **cm**





4: Set Dialog Data Entry Mode

▶ Select **Tools > Options > General Options > 3D Modeler > Drawing > Dialog**

The screenshot displays the ANSYS Electronics Desktop interface. The **Tools** menu is open, and the path **Options > General Options... > HPC and Analysis Options... > Export Options Files...** is highlighted. The **Options** dialog box is open, showing the **Drawing** section under **3D Modeler**. The **Drawing Data Entry Mode** is set to **Dialog**. The **Relative Coordinate System Creation Mode** is set to **Axis/Position**. The **Polyline Creation** section has **Automatically cover closed polylines** checked and **Show measure dialog during drawing** checked. The **Edit properties of new primitives** checkbox is unchecked. The **OK** and **Cancel** buttons are visible at the bottom of the dialog. The status bar at the bottom left shows "Set general options" and the bottom right shows "Hide 0 Messages" and "Hide Progress".



5: Set Default Transparency of 0.7

▶ Select **Tools** > **Options** > **General Options** > **Display** > **Rendering**

The screenshot displays the ANSYS Electronics Desktop interface. The **Tools** menu is open, and the navigation path **Options > General Options > Display > Rendering** is highlighted with a green line. The **Options** dialog box is open, showing the **Rendering Defaults** section where the **Default transparency** is set to **0.7**. The **Object Visualization** section shows **Outline contrast** set to **0.5**. The background shows a 3D model of a circuit board with a coordinate system (X, Y, Z) and a scale bar (0 to 2 mm). The **Project Manager** and **Properties** panels are also visible.

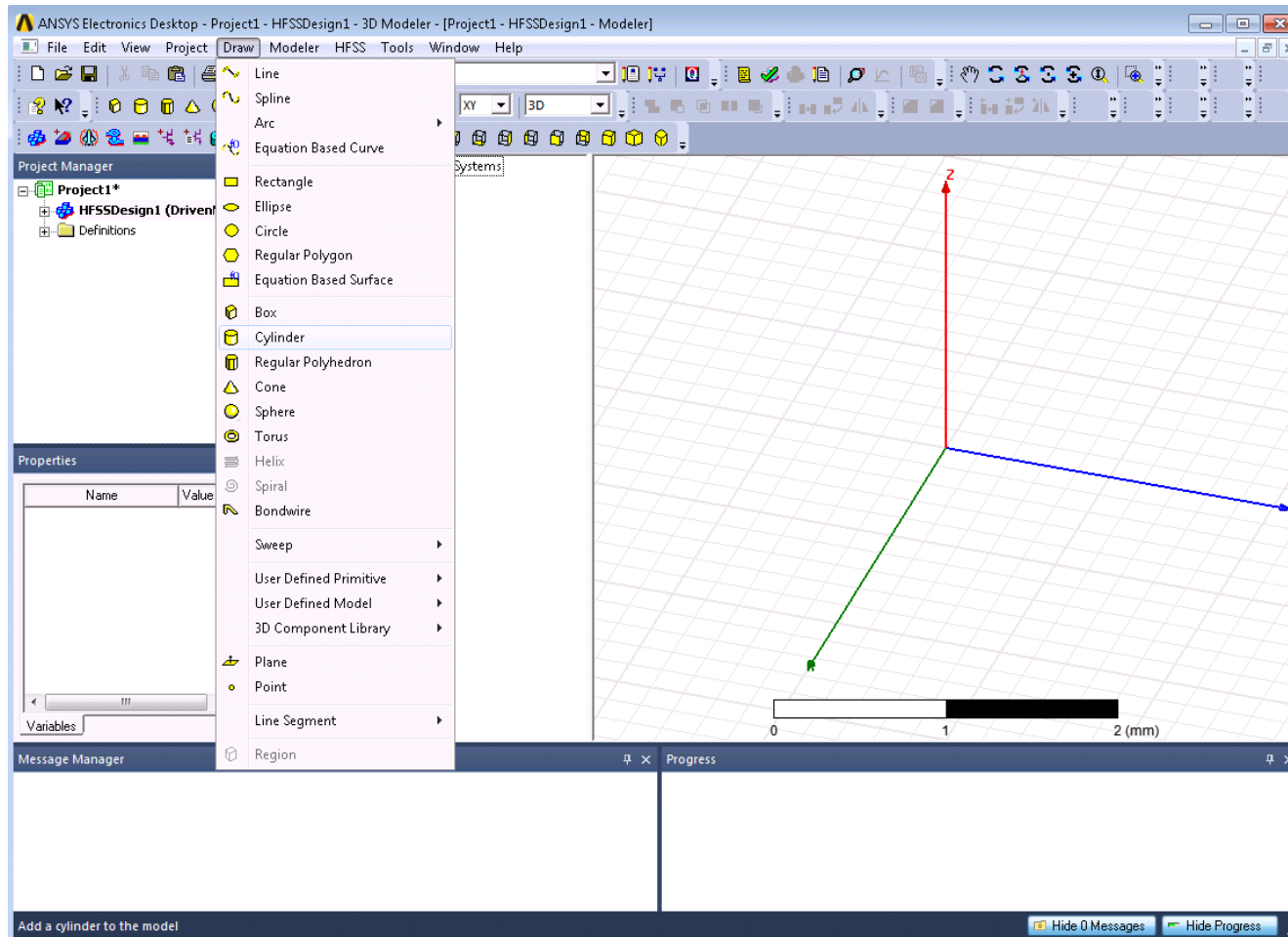
Set general options

Hide 0 Messages Hide Progress



6: Create Parameterized Antenna

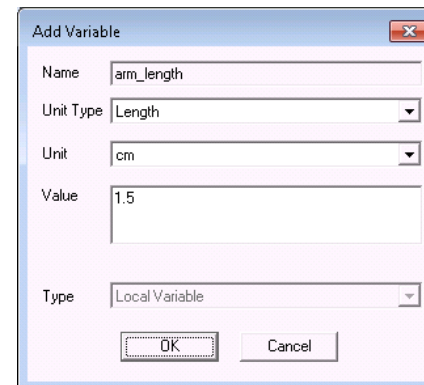
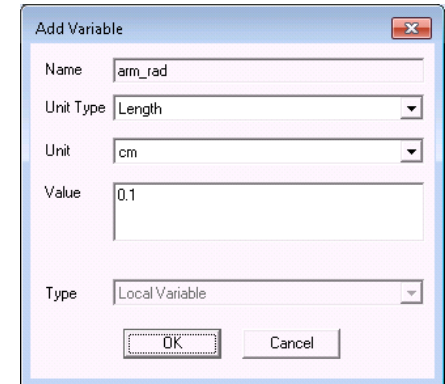
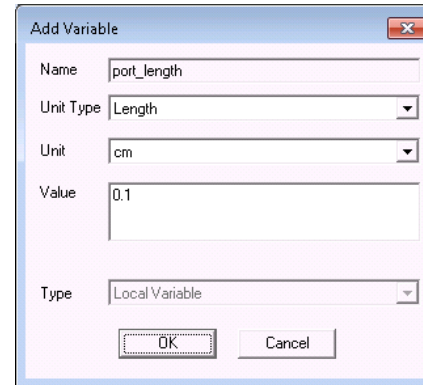
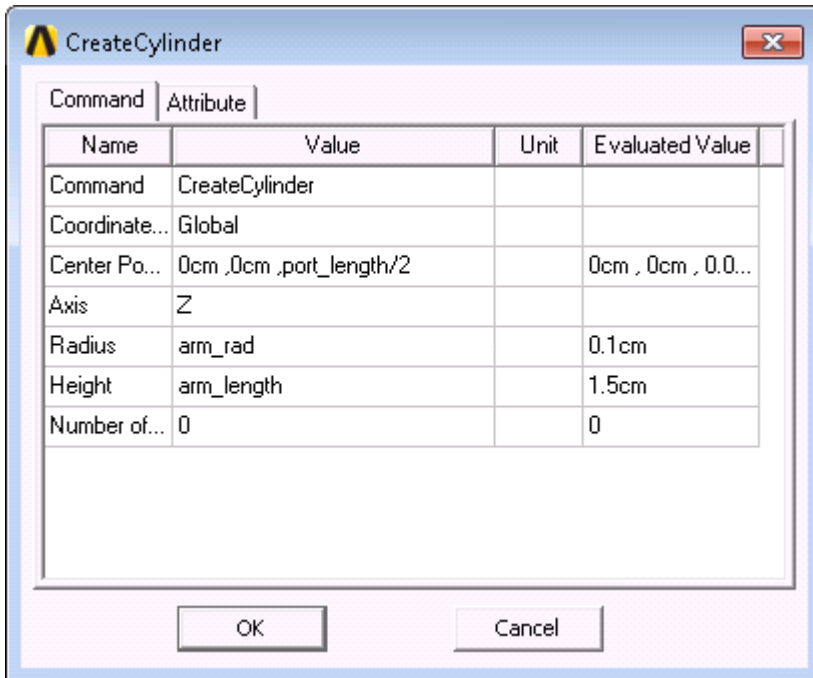
▶ Select *Draw > Cylinder*





6: Create Parameterized Antenna

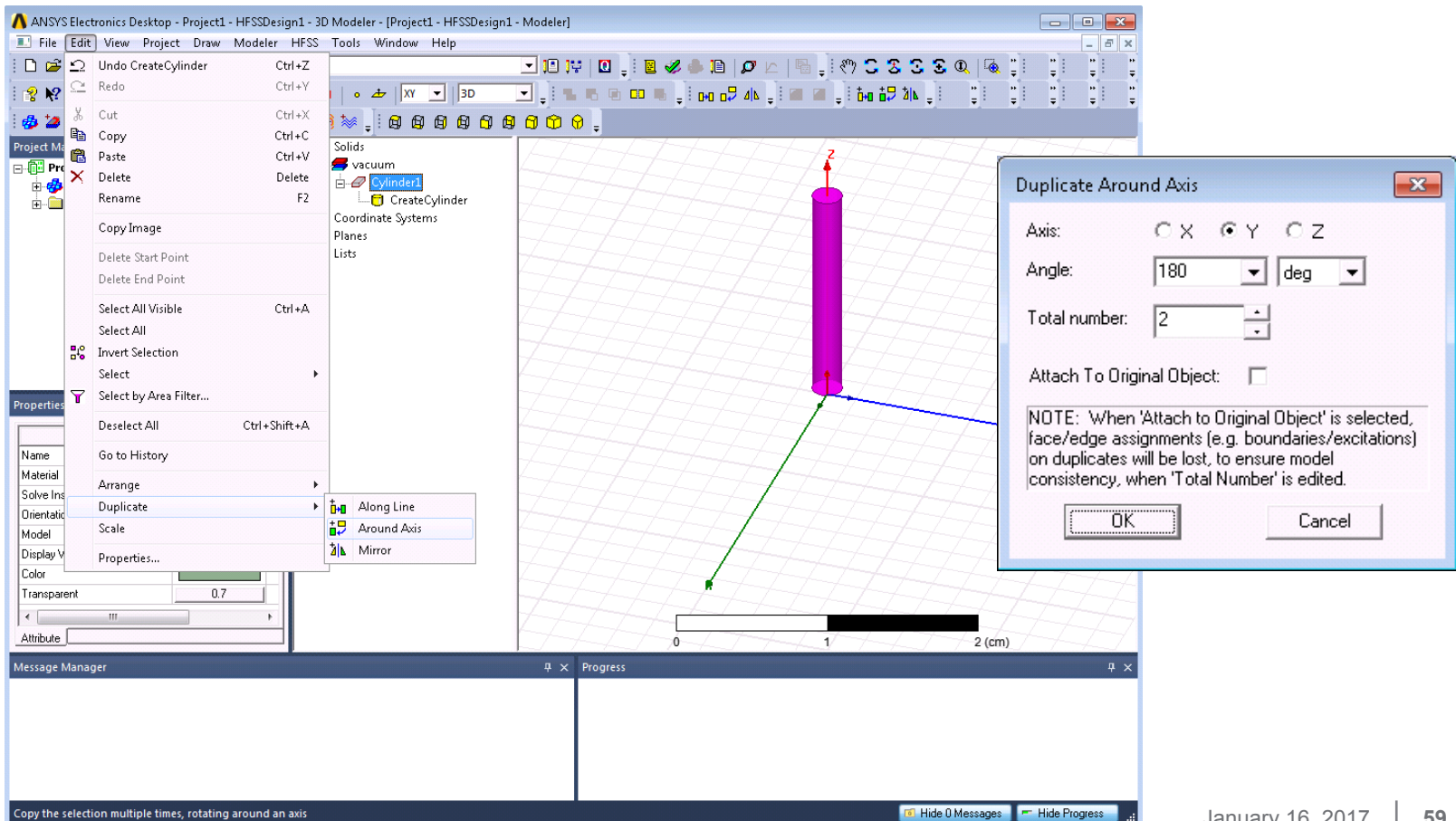
- ▶ $\text{port_length} = 0.1 \text{ cm}$
- ▶ $\text{arm_rad} = 0.1 \text{ cm}$
- ▶ $\text{arm_length} = 1.5 \text{ cm}$





6: Create Parameterized Antenna

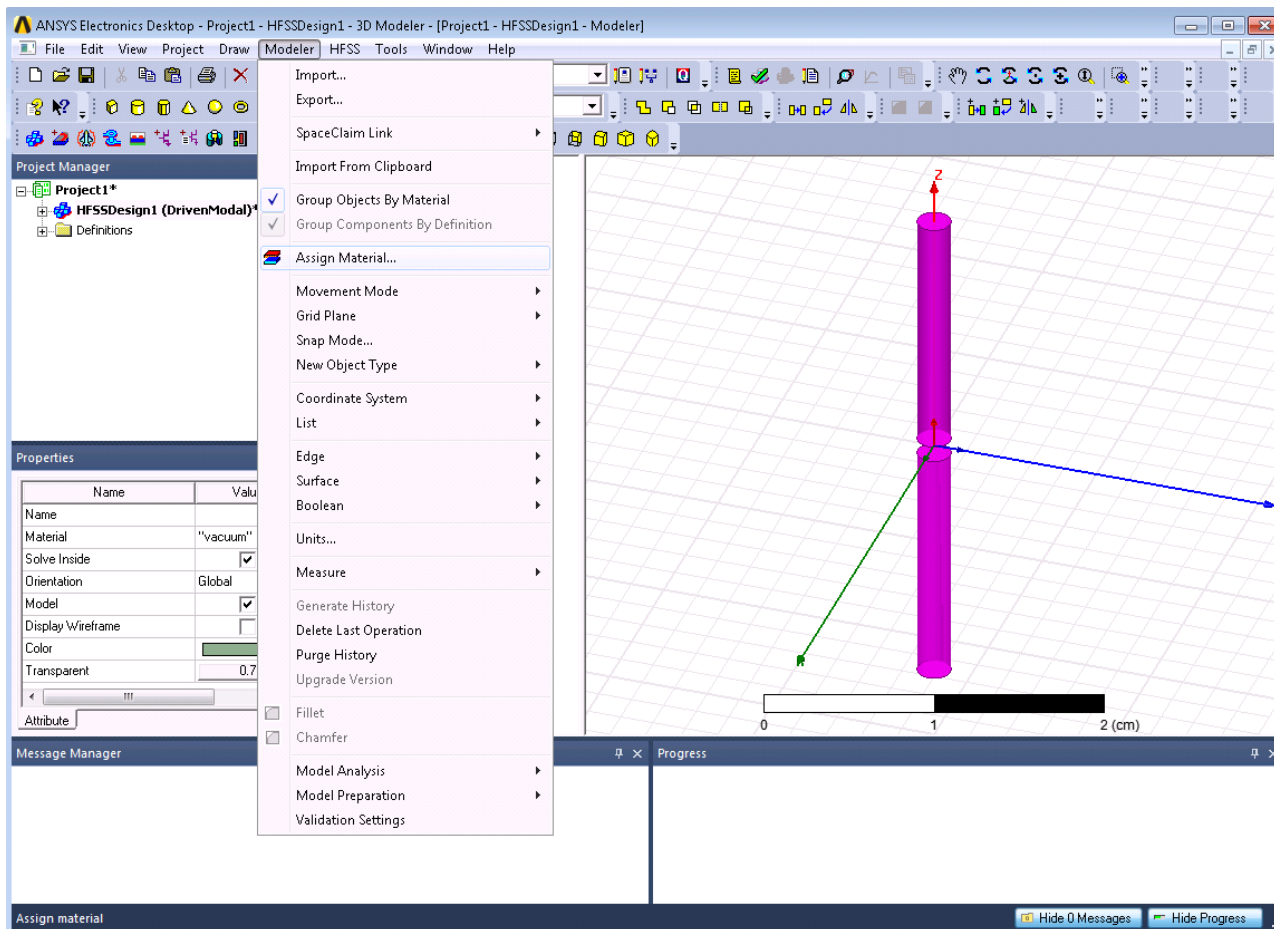
- ▶ Select Cylinder1 in 3D modeler tree and **Edit > Duplicate > Around Axis**
- ▶ Axis = y-axis, Angle = 180 degrees, Total number = 2





7: Assign Material to Antenna

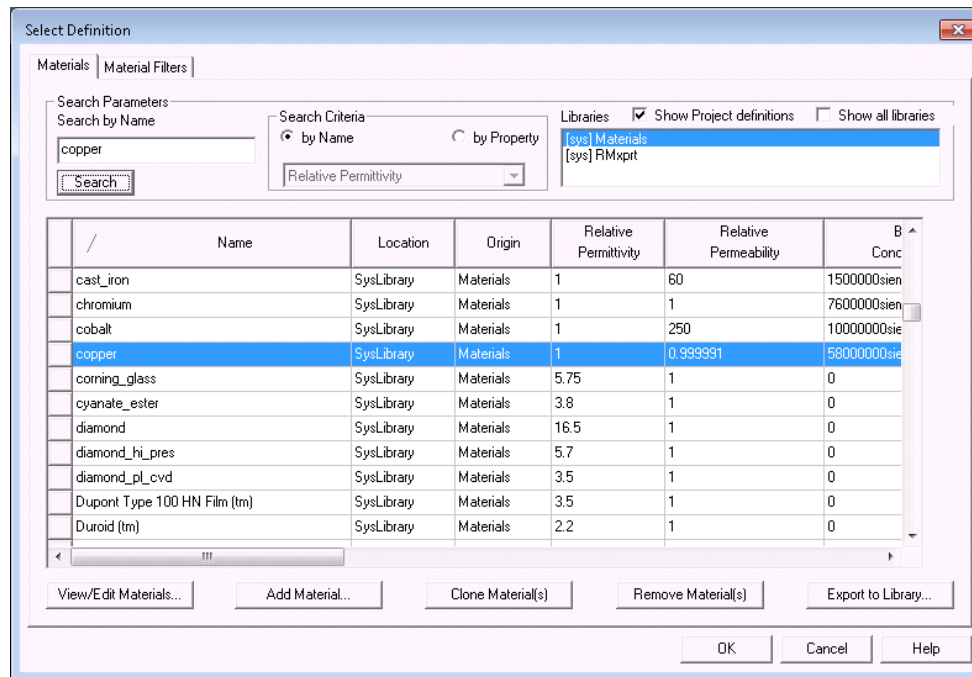
- ▶ Select Cylinder1 and Cylinder1_1 in 3D modeler tree
- ▶ Select **Modeler > Assign Material**





7: Assign Material to Antenna

- ▶ Enter “copper” in Search field of Materials Database and press OK





8: Create Port

▶ Select Draw > Rectangle

The screenshot shows the ANSYS Electronics Desktop interface. The 'Draw' menu is open, and 'Rectangle' is selected. The 'CreateRectangle' dialog box is displayed, showing the following table:

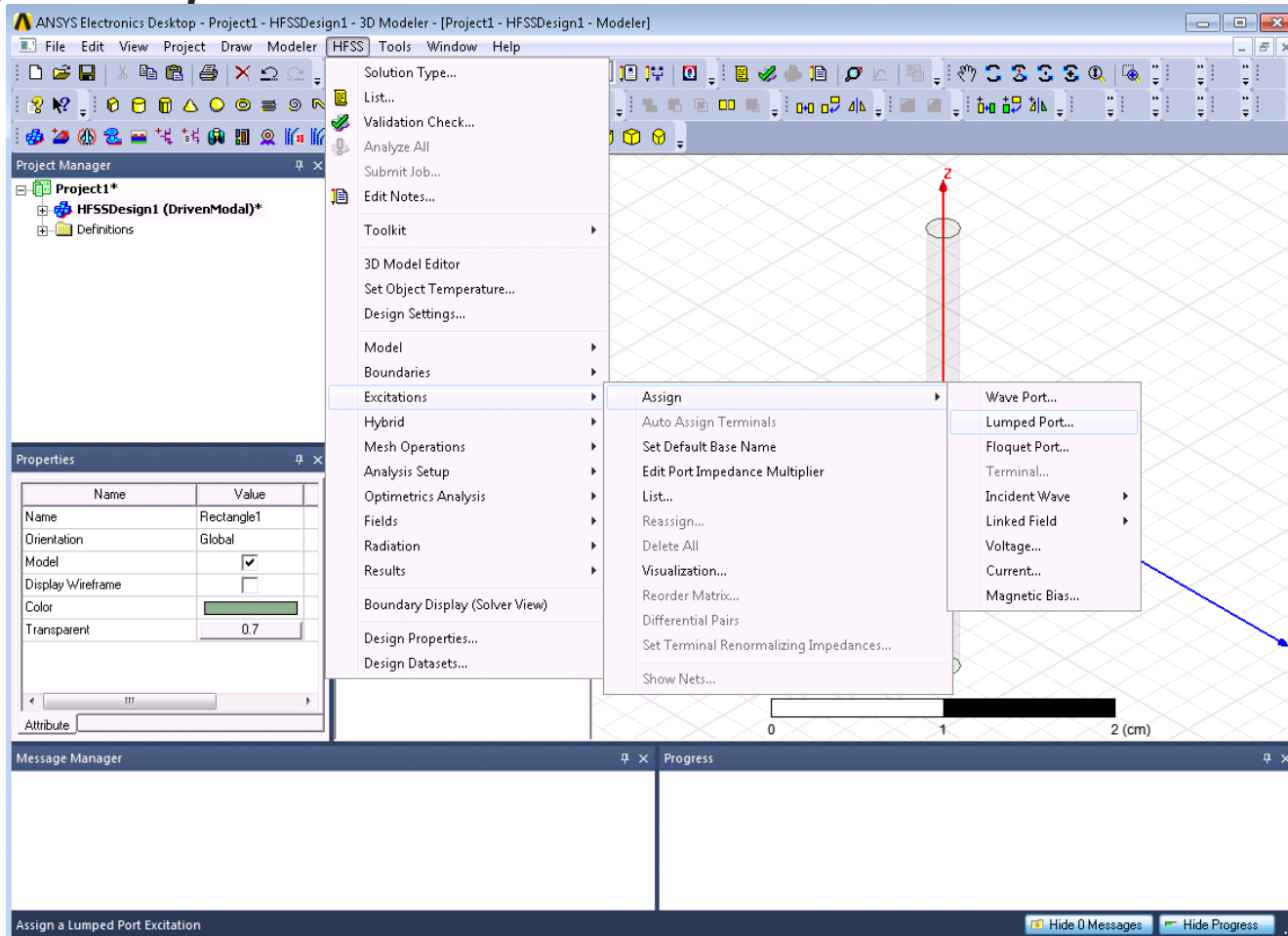
Name	Value	Unit	Evaluated Value
Command	CreateRectangle		
Coordinate...	Global		
Position	0cm , -arm_rad , port_length/2		0cm , -0.1 cm , ...
Axis	X		
Width	2*arm_rad		0.2cm
Length	port_length		0.1cm

The dialog box has 'OK' and 'Cancel' buttons at the bottom. The background shows a 3D model of a port on a grid, with a scale bar at the bottom indicating 0, 1, and 2 cm.



8: Create Port

- ▶ Select Rectangle1 in 3D modeler tree and then **HFSS > Excitations > Assign > Lumped Port**





8: Create Port

- ▶ Name Port P1
- ▶ Draw integration line along center of port between antenna arms
- ▶ Enter 73 Ohms for renormalization impedance

Lumped Port: General

Name: P1

Full Port Impedance

Resistance: 50 ohm

Reactance: 0 ohm

Use Defaults

< Back Next > Cancel

Lumped Port: Modes

Number of Modes: 1

Mode	Integration Line	Characteristic Impedance (Z ₀)
1	Defined	Z _{pi}

Use Defaults

< Back Next > Cancel

Lumped Port: Post Processing

Port Renormalization

Do Not Renormalize

Renormalize All Modes

Full Port Impedance: 73 ohm

Deembed Settings

Deembed

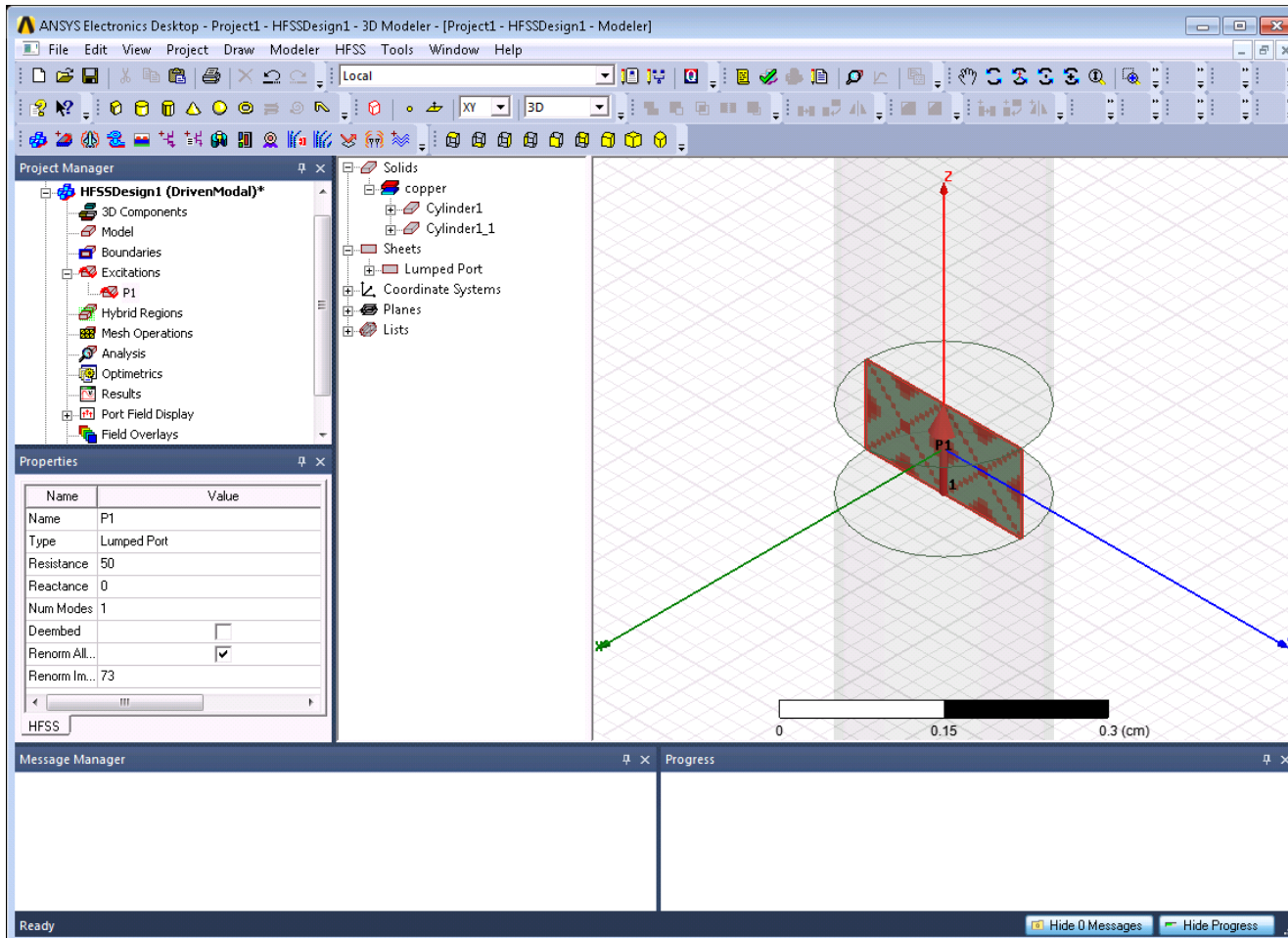
Use Defaults

< Back Finish Cancel



8: Create Port

- ▶ Should have port assignment as shown below





9: Create Open Region

- ▶ Select **HFSS > Model > Create Open Region**
- ▶ Enter Operating Frequency = 5 GHz and Radiation Boundary

ANSYS Electronics Desktop - Project1 - HFSSDesign1 - 3D Modeler - [Project1 - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Project Manager

- HFSSDesign1 (DrivenModal)*
 - 3D Components
 - Model
 - Boundaries
 - Excitations
 - P1
 - Hybrid Regions
 - Mesh Operations
 - Analysis
 - Optimetrics
 - Results
 - Port Field Display
 - Field Overlays

Properties

Message Manager

Progress

Number of strands (number of wires per conductor), 0 for auto-design

Hide 0 Messages Hide Progress

0 0.5 1 (cm)

Create Open Region

Region

Operating Frequency: 5 GHz

Apply infinite ground plane at Neg Z direction

Boundary

Radiation

FE-BI

PML

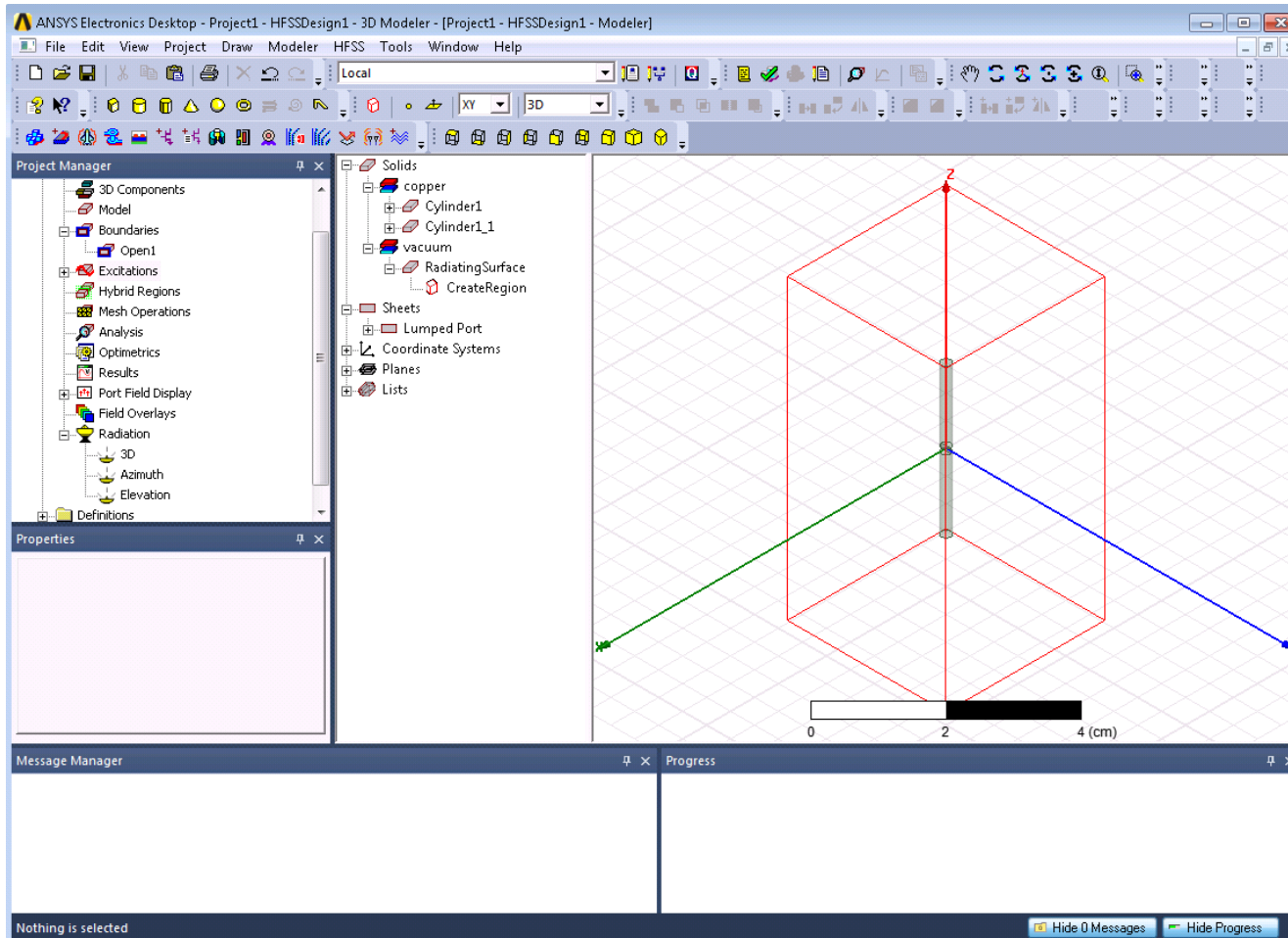
Save as default

OK Cancel



9: Create Open Region

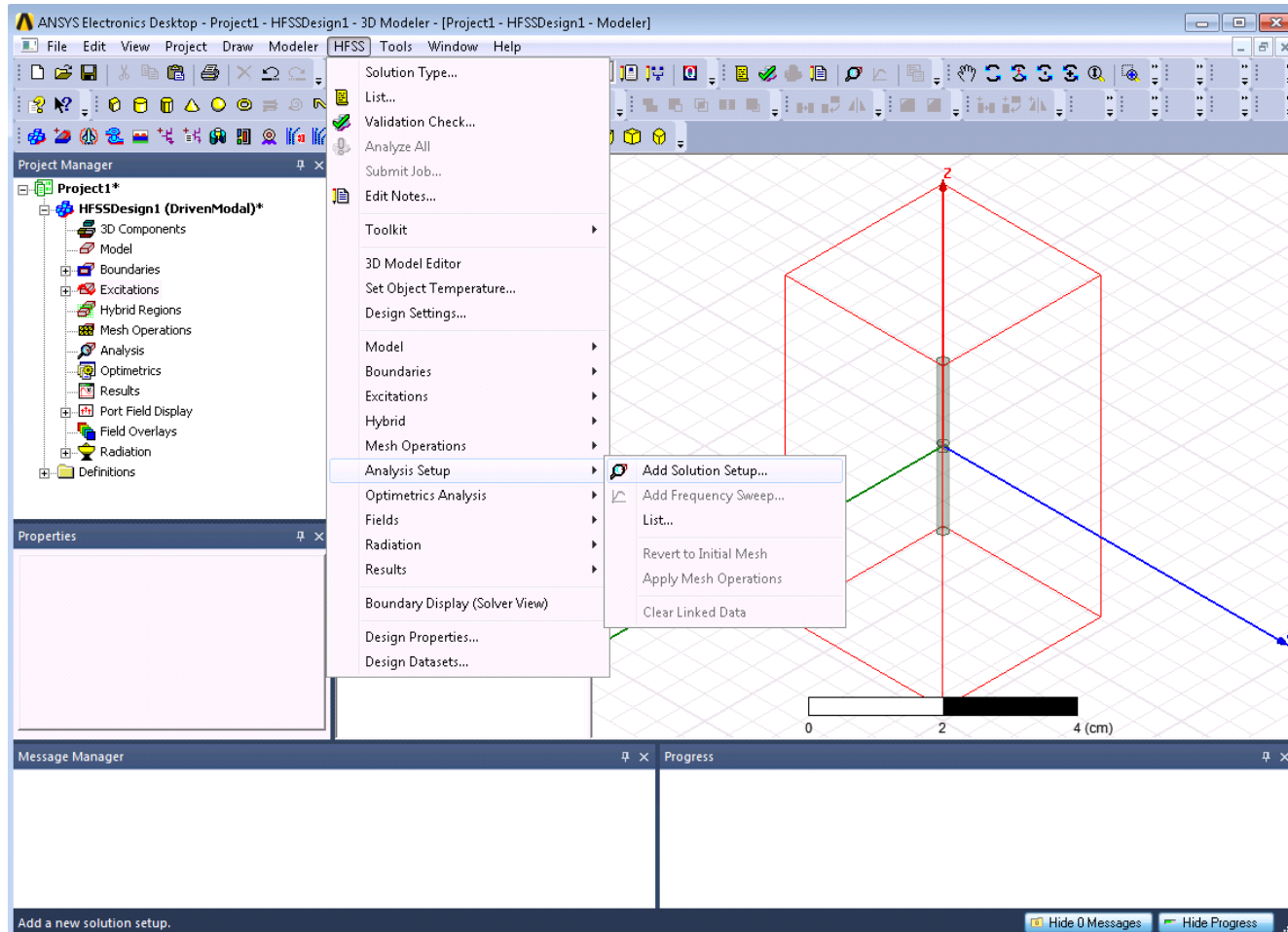
- ▶ Should have radiation boundary and far-field setups as shown below





10: Add Solution Setup

- ▶ Select **HFSS** > **Analysis Setup** > **Add Solution Setup**





10: Add Solution Setup

- ▶ Solution Frequency = 5 GHz, Maximum Passes = 12, Minimum Passes = 4
- ▶ Select Iterative Solver

Driven Solution Setup

General | Options | Advanced | Expression Cache | Derivatives | Defaults

Setup Name:

Enabled Solve Ports Only

Solution Frequency: GHz

Adaptive Solutions

Maximum Number of Passes:

Maximum Delta S

Use Matrix Convergence

Driven Solution Setup

General | Options | Advanced | Expression Cache | Derivatives | Defaults

Initial Mesh Options

Do Lambda Refinement
Lambda Target: Use Default Value

Use Free Space Lambda

Adaptive Options

Maximum Refinement Per Pass: %

Maximum Refinement:

Minimum Number of Passes:

Minimum Converged Passes:

Solution Options

Order of Basis Functions:

Direct Solver

Iterative Solver
Relative Residual:

Domain Decomposition
Relative Residual:



11: Add Frequency Sweep

- ▶ Select Setup1 in Project Manager and then **HFSS** > **Analysis Setup** > **Add Frequency Sweep**
- ▶ Interpolating sweep, Start = 3 GHz, End = 7 GHz, Step size = 0.01 GHz

ANSYS Electronics Desktop - Project1 - HFSSDesign1 - 3D Modeler - [Project1 - HFSSDesign1 - Modeler]

Project Manager

- Project1*
- HFSSDesign1 (DrivenModal)*
 - 3D Components
 - Model
 - Boundaries
 - Excitations
 - Hybrid Regions
 - Mesh Operations
 - Analysis
 - Setup1
 - Optimetrics
 - Results
 - Port Field Display
 - Field Overlays
 - Radiation
 - Definitions

Properties

Name	Value
Name	Setup1
Enabled	<input checked="" type="checkbox"/>
Passes	12
Percent R...	30

HFSS

Message Manager

Progress

Add a new frequency sweep to this setup.

Hide 0 Messages Hide Progress

Edit Frequency Sweep

General | Interpolation | Defaults

Sweep Name: Sweep Enabled

Sweep Type: Interpolating

Frequency Sweeps [401 points defined]

	Distribution	Start	End	Step size	
1	Linear Step	3GHz	7GHz	Step size	0.01GHz

Add Above Add Below Delete Selection Preview ...

Time Domain Calculation...

OK Cancel



12: Save Project

- ▶ Select **File > Save** and save project as “dipole_tutorial.aedt”

The screenshot shows the ANSYS Electronics Desktop interface. The 'File' menu is open, and the 'Save' option is highlighted. The 'Save As' dialog box is open, showing the file name 'dipole_tutorial' and the file type 'ANSYS Electronics Desktop Project File (*.aedt)'. The dialog box also shows a list of files in the 'Tutorial' folder, including 'cavity_tutorial.aedtrresults' and 'cavity_tutorial.aedt'.

Name	Date modified	Type	Size
cavity_tutorial.aedtrresults	1/6/2017 12:11 PM	File folder	
cavity_tutorial.aedt	1/6/2017 11:03 AM	Ansoft Electronics...	37 Ki

File name: dipole_tutorial
Save as type: ANSYS Electronics Desktop Project File (*.aedt)

Save the active project

Hide 0 Messages Hide Progress



13: Perform Validation Check

- ▶ Select **HFSS** > **Validation Check**
- ▶ Confirms that required steps to solve model have been performed

ANSYS Electronics Desktop - dipole_tutorial - HFSSDesign1 - 3D Modeler - [dipole_tutorial - HFSSDesign1 - Modeler]

File Edit View Project Draw Modeler HFSS Tools Window Help

Solution Type...
List...
Validation Check...
Analyze All
Submit Job...
Edit Notes...
Toolkit
3D Model Editor
Set Object Temperature...
Design Settings...
Model
Boundaries
Excitations
Hybrid
Mesh Operations
Analysis Setup
Optimetrics Analysis
Fields
Radiation
Results
Boundary Display (Solver View)
Design Properties...
Design Datasets...

Project Manager

dipole_tutorial
HFSSDesign1 (DrivenModal)
3D Components
Model
Boundaries
Excitations
Hybrid Regions
Mesh Operations
Analysis
Setup1
Optimetrics
Results
Port Field Display
Field Overlays
Radiation
Definitions

Properties

Name	Value
Name	Setup1
Enabled	<input checked="" type="checkbox"/>
Passes	12
Percent R...	30

HFSS

Validation Check: cavity_tutorial - HFSSDesign1

✓ HFSSDesign1

Validation Check completed.

- ✓ Design Settings
- ✓ 3D Model
- ✓ Boundaries and Excitations
- ✓ Mesh Operations
- ✓ Analysis Setup
- ✓ Optimetrics
- ✓ Radiation

Abort Close

Message Manager

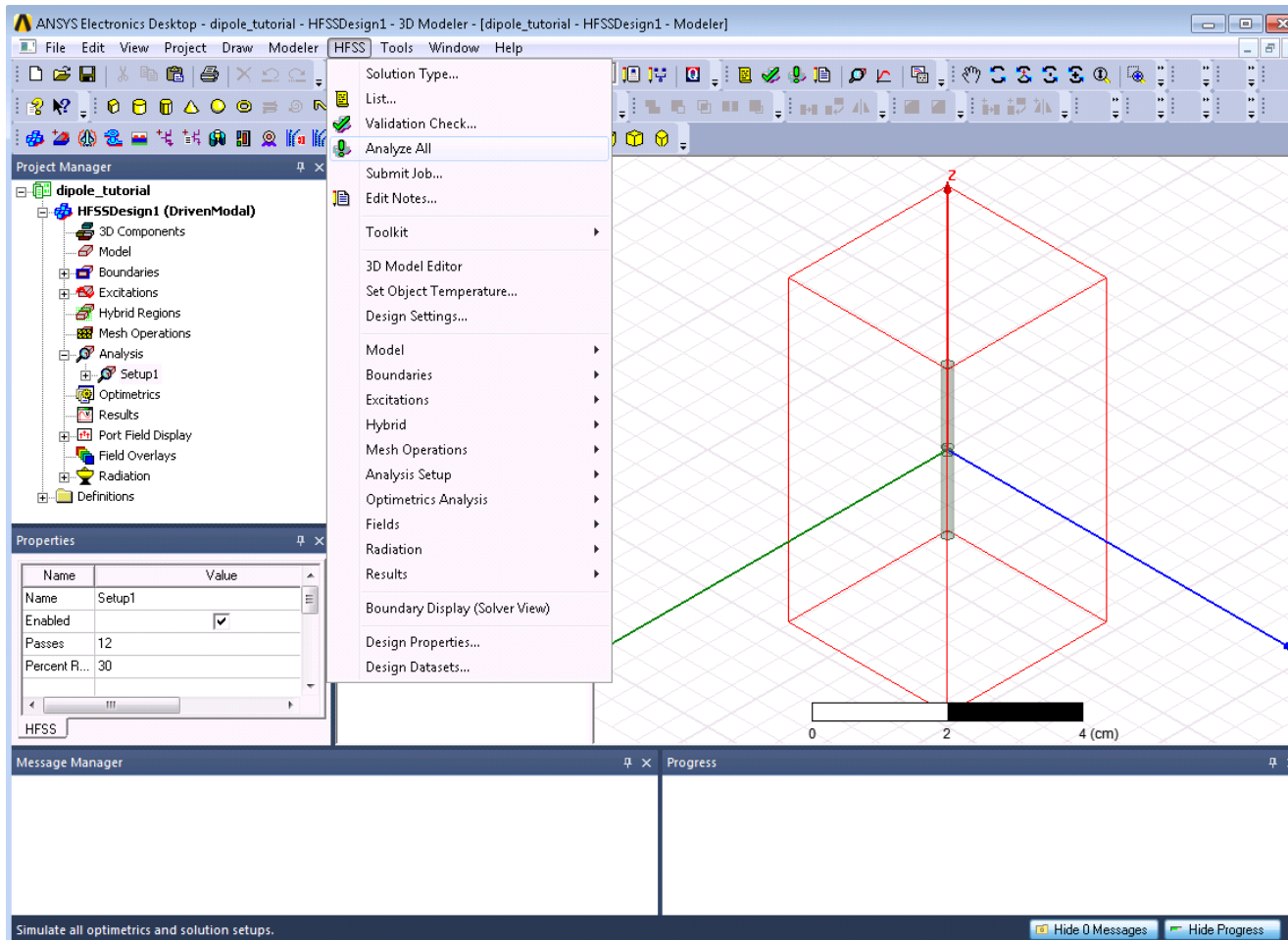
Progress

Perform validation check on design setup. Hide 0 Messages Hide Progress



14: Solve Model

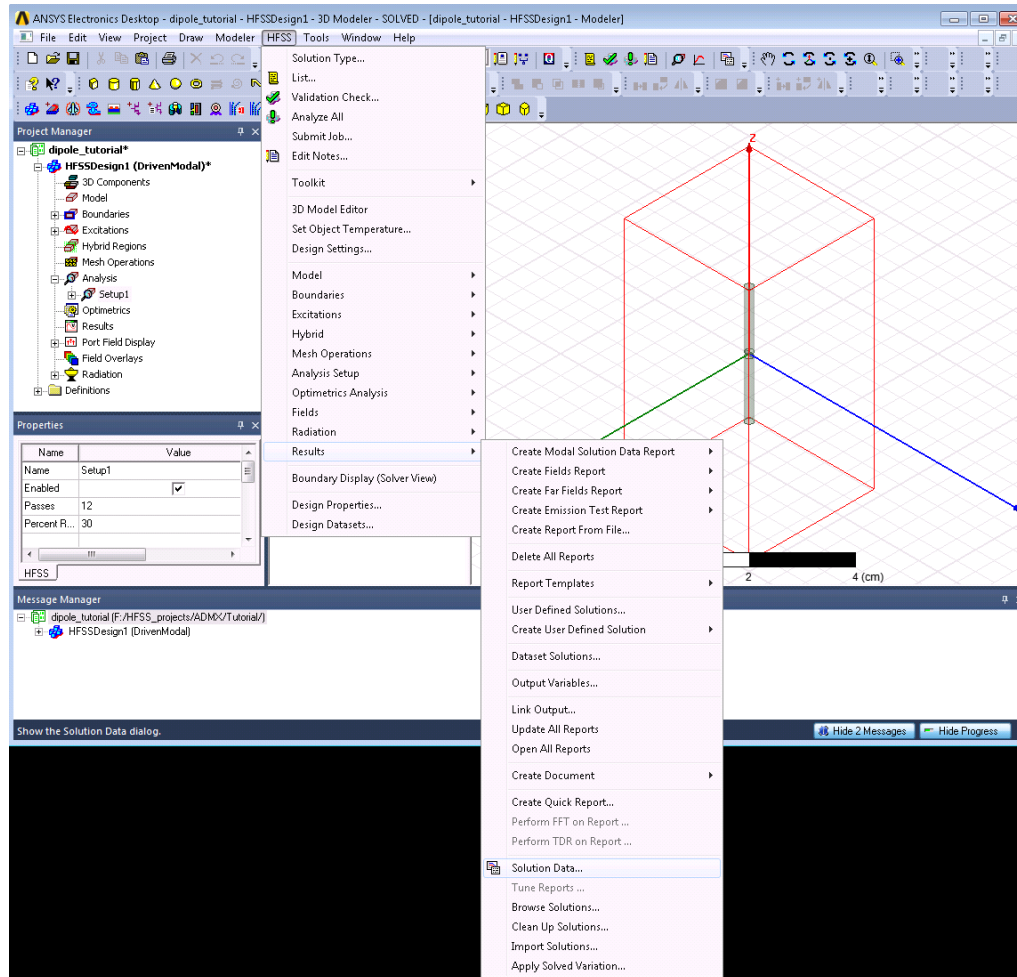
► Select **HFSS** > **Analyze All**





15: View Solution Data

▶ Select *HFSS* > *Results* > *Solution Data*





15: View Solution Data

- ▶ Select **Matrix Data** tab to view network parameters

The screenshot shows the 'Matrix Data' tab in the HFSS software interface. The window title is 'Solutions: dipole_tutorial - HFSSDesign1'. The 'Simulation' dropdown is set to 'Setup1' and the 'Sweep' dropdown is set to 'Sweep'. The 'Design Variation' field contains 'arm_length='1.5cm' arm_rad='0.1cm' port_length='0.1cm''. The 'Matrix Data' tab is selected, and the 'S Matrix' checkbox is checked. The frequency is set to '3 (GHz)'. The 'Passivity Tolerance' is set to '.0001'. The 'Export Matrix Data...' button is visible. The table below shows the results for the S Matrix at 3 GHz.

Freq	S:P1:1
3 (GHz)	P1:1 (0.83594, -61)



15: View Solution Data

- ▶ Select **Convergence** tab to view adaptive pass information

The screenshot shows the 'Solutions: dipole_tutorial - HFSSDesign1' window. The 'Convergence' tab is selected, displaying the following information:

Simulation: Setup1
Design Variation: arm_length='1.5cm' arm_rad='0.1cm' port_length='0.1cm'

Profile | **Convergence** | Matrix Data | Mesh Statistics

Number of Passes:
Completed 4
Maximum 12
Minimum 4

Max Mag. Delta S:
Target 0.02
Current 0.0078799

View: Table Plot

Export...

CONVERGED

Consecutive Passes:
Target 1
Current 2

Default Settings:
Save Defaults Clear Defaults

Pass Number	Solved Elements	Max Mag. Delta S
1	2435	N/A
2	3179	0.034516
3	4150	0.018734
4	5393	0.0078799

Close



15: View Solution Data

- ▶ Select **Profile** tab to view run log file (20 sec runtime)

Simulation: Setup1

Design Variation: arm_length='1.5cm' arm_rad='0.1cm' port_length='0.1cm'

Profile | Convergence | Matrix Data | Mesh Statistics

Task	Real Time	CPU Time	Memory	Information
Field Recovery	00:00:00	00:00:00	105 M	Disk = 0 KBytes, 1 excitations
				Interpolation Error: S Matrix error 3.88005 %
Frequency: 3.5 GHz				Full Solution # 5
Simulation Setup	00:00:00	00:00:00	46.1 M	Disk = 0 KBytes
Matrix Assembly	00:00:00	00:00:00	69.9 M	Disk = 0 KBytes, 5393 tetrahedra , P1: 30 triangles
Solver DCS8-L2	00:00:00	00:00:00	104 M	Disk = 0 KBytes, matrix size 36293 , matrix bandwidth 21
Field Recovery	00:00:00	00:00:00	104 M	Disk = 0 KBytes, 1 excitations
				Maximum Passivity Error for S Matrix : 0
				Interpolating sweep converged and passive within tolerar
Solution Process				Elapsed time : 00:00:06 , Hfss ComEngine Memory : 60.4
Total	00:00:01	00:00:03		Time: 01/06/2017 13:55:30, Status: Normal Completion

Export...

Close



16: Create Plot of Return Loss

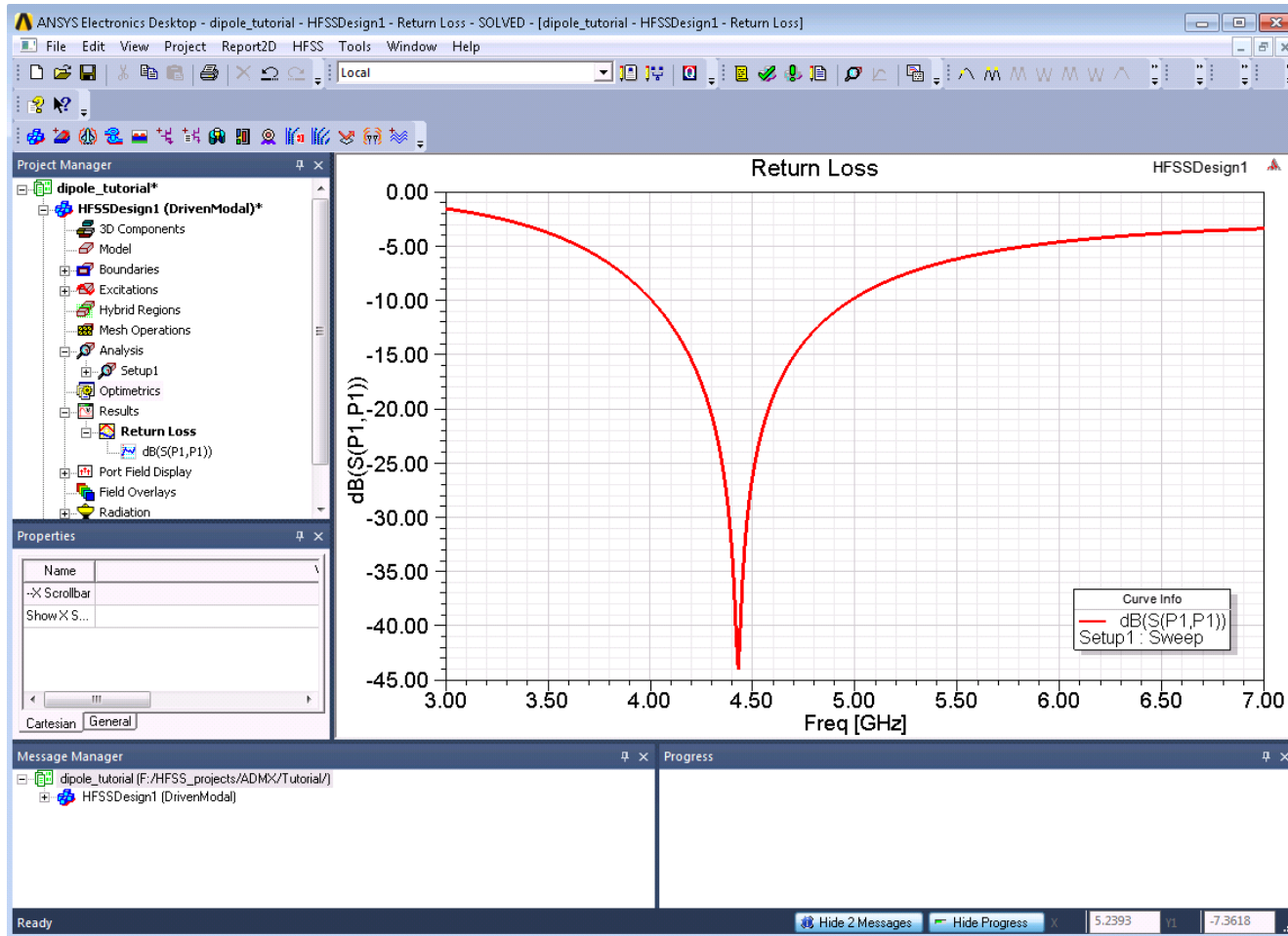
- ▶ Select **HFSS** > **Results** > **Create Modal Solution Data Report** > **Rectangular Plot**

The screenshot illustrates the steps to create a Rectangular Plot of Return Loss in ANSYS HFSS. The main window shows the 3D model of a dipole antenna. The 'Results' tab is active, and the 'Create Modal Solution Data Report' > 'Rectangular Plot' path is highlighted in the context menu. The 'Report: dipole_tutorial - HFSSDesign1 - New Report - New Trace(s)' dialog box is open, showing the configuration for the report. The 'Context' section shows 'Solution: Setup1: Sweep' and 'Domain: Sweep'. The 'Y' axis is set to 'dB(S(P1,P1))'. The 'Quantity' list includes 'S(P1,P1)' and 'Function' includes 'dB10normalize'.



16: Create Plot of Return Loss

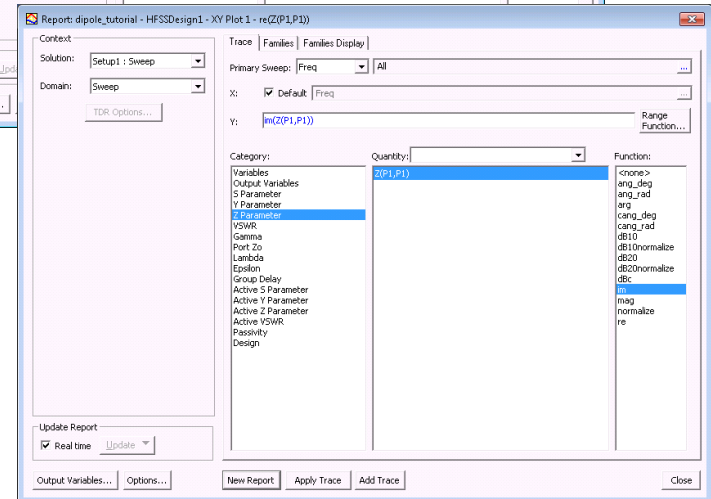
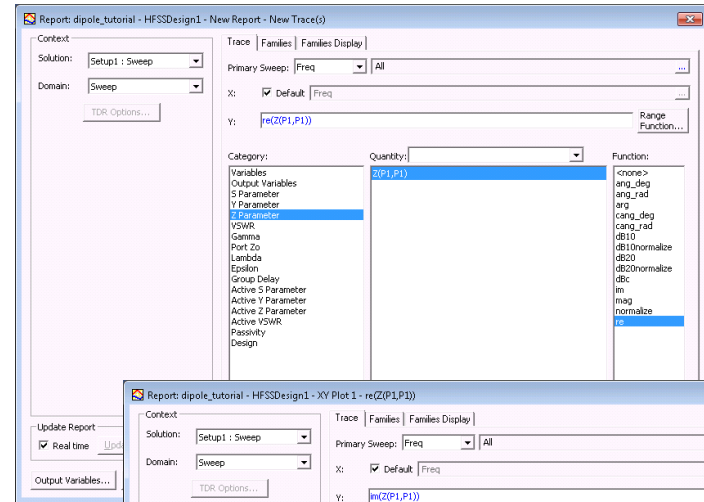
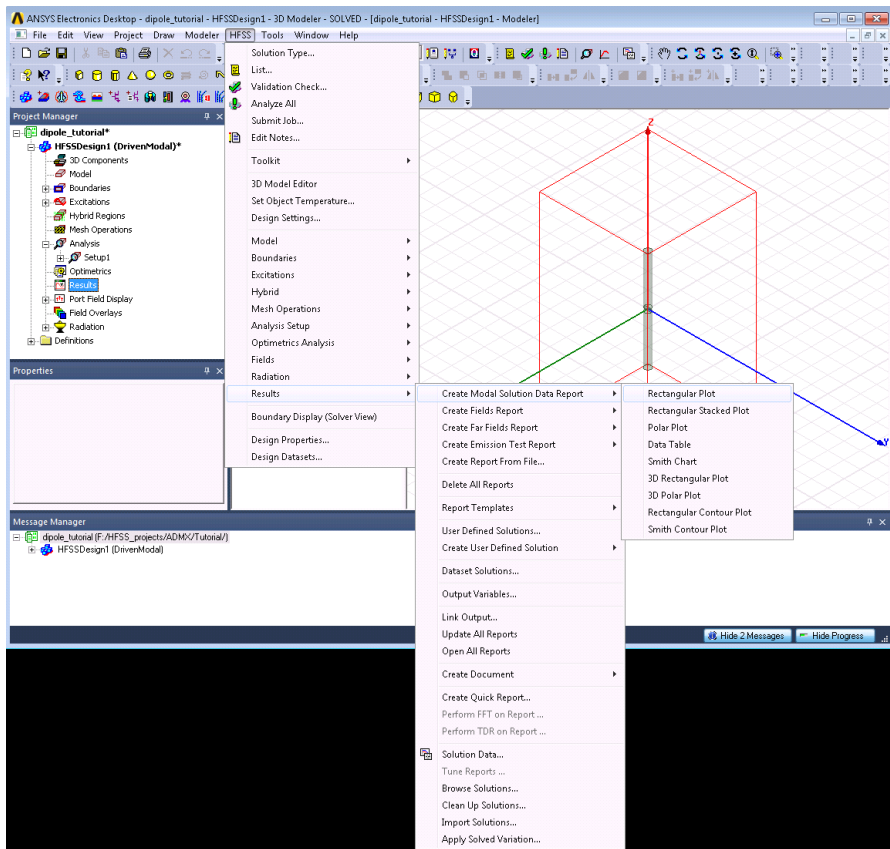
- ▶ Antenna resonates near 4.4 GHz





17: Create Plot of Input Impedance

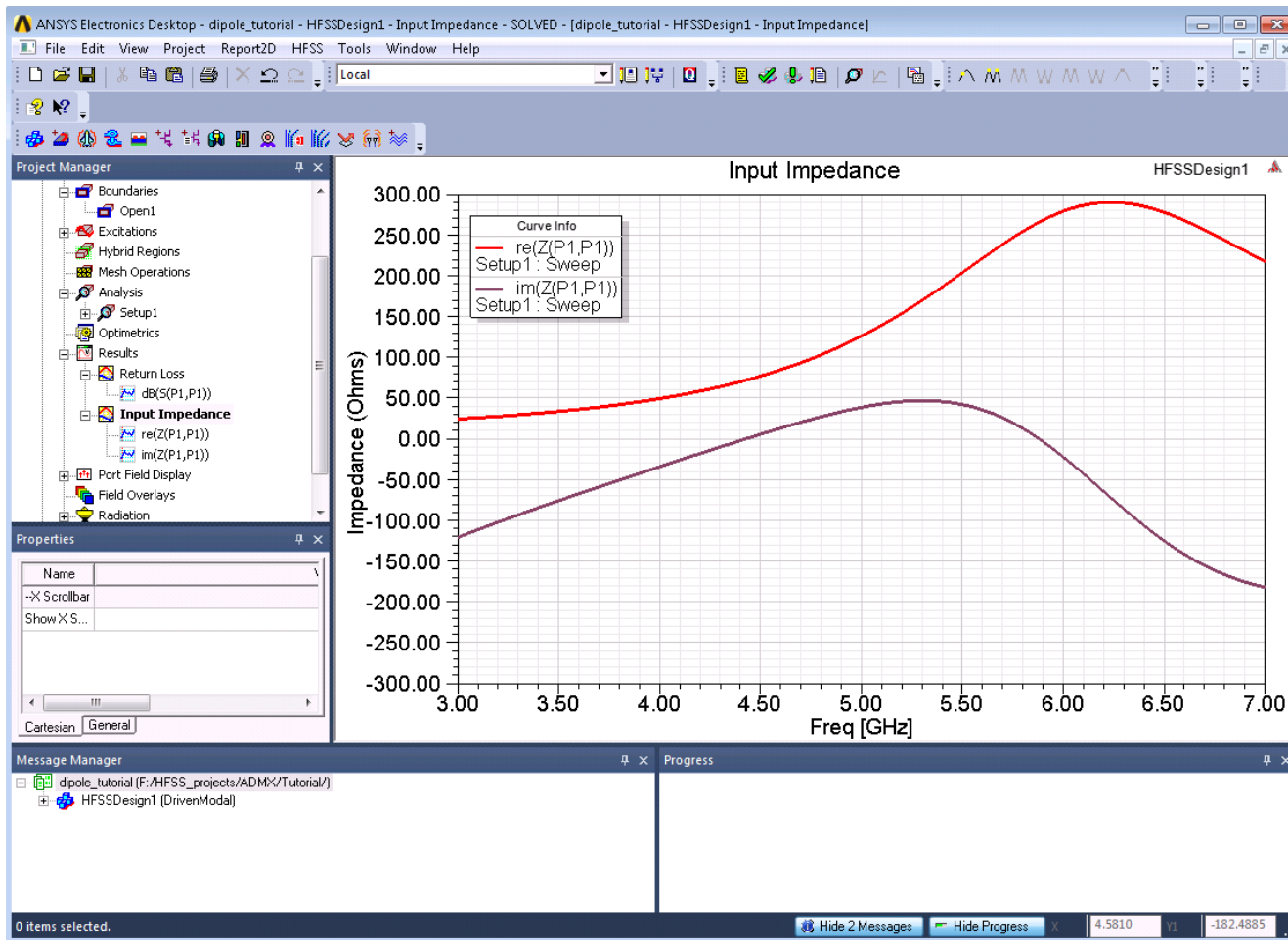
- ▶ Select **HFSS** > **Results** > **Create Modal Solution Data Report** > **Rectangular Plot**





17: Create Plot of Input Impedance

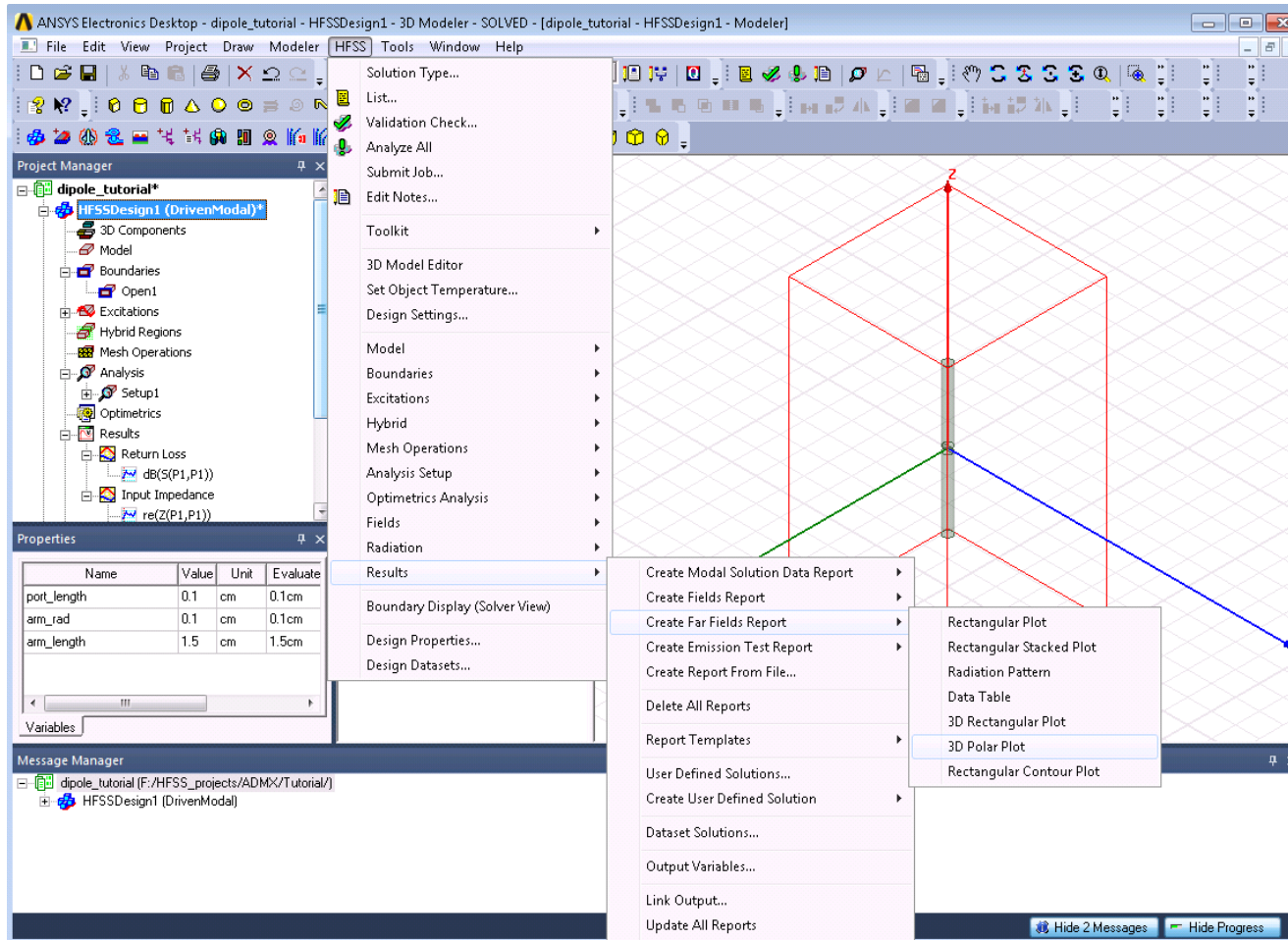
- ▶ Antenna resonates near 4.4 GHz





18: Create Plot of Far-field Pattern

- ▶ Select **HFSS** > **Results** > **Create Far Fields Report** > **3D Polar Plot**





18: Create Plot of Far-field Pattern

- ▶ Select **Gain** > **GainTotal** > **dB**

