# Laboratory 4

## Simulation in HFSS, Dipole Antenna, part I

Start ANSYS Electronics Desktop, then start HFSS by clicking on the corresponding icon in the ribbon. If you forgot how, access laboratory 2.

Check (and change if necessary) that the working mode is Driven Modal, HFSS >

#### Solution Type > Modal .



Set Modeler length units to **mm** , **Modeler > Units > mm** .



Change the default drawing mode to **Dialog** Data Entry mode (the coordinate input window), **Tools > Options > General Options > 3D Modeler > Drawing > Dialog**. Remember that the F3/F4 keys let you switch between this mode and the **Point** mode (using the mouse).



For a better visualization, set a default transparency of 0.7, **Tools > Options > General Options > Display > Rendering**.



Draw the antenna in Parameterized mode. This mode assumes that variables will be used for the dimensions of the structure. For the dipole, three variables are needed, the radius and length of the two wires plus the distance between the wires. The parametric mode has the advantage that the model dimensions are entered by variables that receive an initial value, and, if necessary, changing the numerical value of the variable leads to the automatic redraw of the structure taking into account the new value. Optimetric (Parametric) analysis is also available in HFSS (automatic variation of a variable, with comparison of the results - for example we change the radius in steps and we check the effect on the resonant frequency of the antenna).

Variables are defined while drawing the model. If a word is entered in one of the input boxes that require a length, if that variable does not exist, the window for its definition and entering the its initial value appears. To draw a parameterized wire, select **Draw** > **Cylinder** and enter names at radius and height position.

Name	Value	Unit	Evaluated Value	Name	lung	
Command	CreateCvlinder	- Or inc		, Kalilo		
Coordinate	Global			Unit Type	Length	
Center Po	0, 0, 0	mm	0mm , 0mm , 0mm	Unit	mm	_
Axis	Z					
Radius	raza		0.5mm	Value	39.5	
Height	lung	mm	Omm			
Number of	0		0		·	
				Туре	Local Variable	
				-	OK Cancel	

Check (and change if necessary) that the data type is Length with the unit of measurement mm. In the example, the names "raza", "lung", "port\_l" are used as variables, with the initial values of 0.5mm, 39.5mm, 1mm respectively. Draw the antenna along the OZ axis, symmetrical with respect to the origin (hence the positioning of the center 0, 0, port\_l/2).

**Important!** The names you use are not essential, you can use other words if you want, but it is essential to keep consistency throughout the lab (if you have chosen to name the variable "X" use the same "X" further where that value is used).

id					Add Variable	
Name	Value	Unit	Evaluated Value	Description	Name D	ort I
Command	CreateCylinder					
Coordinate Sys.	Global				Unit Type Le	ength
Center Position	0 ,0 ,port_1/2	mm	Omm , Omm , Omm	-	Linit Im	
Axis	Z				in the second se	
Radius	raza		0.5mm		Value 1	
Height	lung		39.5mm			
Number of Seg.	. 0		0			
					Туре	ocal Variable
						OK Cancel
				Show Hidden		

After drawing the first wire of the dipole we will create the second one by symmetry. Taking into account the particular way we chose the coordinates we can create the second cylinder by rotating the first one  $180^{\circ}$  around the OX or OY axis (! Not the default OZ). Select the first cylinder and then select **Edit > Duplicate > Around Axis**, choose the axis X or Y and enter the angle of  $180^{\circ}$ , Total number 2. The advantage of duplication is that the second cylinder will keep the same parameterized dimensions with the same variables names as the first cylinder, changing the variable value will redraw both.

E File	Edit	View Project Draw	Modeler HFSS	Tools	Windo	w Help		
11	17	Undo	Ctrl+Z		$\bigcirc$	+ Pan		
	64	Redo	Ctrl+Y	*	4	ARotate -		
Save	×	Cut	Ctrl+X		Zoom	Orient -		
Desktor	-	Сору	Ctrl+C	Re	sults	Automation		
Project Ma	3	Paste	Ctrl+V					
na3	x	Delete	Delete					
FSSDesi		Rename	F2					
finitions		Copy Image		inder				
		Delete Start Point						
		Delete End Point					1 mar 10 mar	10.200
		Select All Visible	Ctrl+A				Duplicate Arou	nd Axis
		Select All					Auio	CV GV CZ
	510 810	Invert Selection					CONS.	5 A 19 1 10 2
		Next Behind	B				Angle:	180 🔻 deg 🔻
		Selection Mode	+					
		Select Objects	•				Total number:	2
		Extend Selection	•					
	Y	Select by Area Filter					Attach To Orig	inal Object: 🛛 🥅
		Deselect All	Ctrl+Shift+A				NOTE: When	'Attach to Original Object' is s
		Go to History					face/edge assi	ignments (e.g. boundaries/exc
		Arrange	•			1-1-1	on duplicates v	vill be lost, to ensure model hen 'Total Number' is edited
		Duplicate	•	фю.	Along Lin	e	poondiction by, in	
roperties		Scale		謬。	Around A	xis	(OK	Cancel
And in case of the local division of the loc				+1				

As in laboratory 3, assign a material (cooper) for the two wires.

We must provide a input source to perform the simulation. The input signal will consist in defining a reference electromagnetic field between the two wires. The area where the input signal is applied (the generic name for this area is **Port**) is drawn as a rectangle. Select **Draw > Rectangle** and draw a rectangle (! it's a surface model not a volume one) between the two wires, as in the figures bellow.



Select the drawn rectangle and define it as the input signal area/port **HFSS** > **Excitations** > **Assign** > **Lumped Port**.



When defining the port, in three successive steps, choose the port name (P1), draw the integration line between the axes of the two cylinders, and enter  $73\Omega$  for renormalization impedance.

Lumped Port : General	Lumped Port : Modes
Name: P1 Full Port Impedance: 50 ohm 💌 Impedance ::= resistance + 1i * reactance	Number of Modes: 1 Mode Integration Line Characteristic Impedance (Zo) 1 None Zpi None New Line
Use Defaults	
	Use Defaults
< Back Next > Cancel	< Back Next > Cancel
Lumped Port : Modes	Lumped Port : Post Processing
Number of Modes: 1 Mode Integration Line Characteristic Impedance (Z	Port Renormalization C Do Not Renormalize Renormalize All Modes Full Port Impedance: 73 ohm Impedance := resistance + 11° reactance
	Deembed Settings
Use Defaults	Use Defaults
< Back Next >	< Back Finish Cancel

Check the success of port definition in **Project Manager > Excitations > P1**. The view on the screen should be identical to the figure bellow.



The computation area is defined ("Open Region": outside this area HFSS considers the simplified model of free propagation of the waves in vacuum, without influence from other objects, not even the antenna you drawn). "Open region" determines the transition from near/far field regions. Select **HFSS** > **Model** > **Create Open Region**. As in the figures bellow, choose the frequency of interest (around 1.9GHz) and Radiation boundary.



Operating Frequency:	1.9	▼ GHz	•
Apply infinite grour	nd plane at NegZ	- direction	
oundary			
Radiation			
C FE-BI			
C PML			

If successful, a rectangular radiation surface is obtained that will surround the two wires at the appropriate distance (for 1.9GHz near/far field regions).



Add a solution. Select **HFSS > Analysis Setup > Add Solution Setup**.



In the first two tabs of the window, choose the frequency around which you want to have the correct results (1.9GHz), the maximum number of passes 12, minimum number of passes 4 and select the iterative solver. The structure will be meshed at the indicated frequency.

Seneral     Options     Advanced     Hybrid     Expression Cache     Derivatives     Defaults       Setup Name     Setup 1       Image: Transled     Solve Ports Only	Driven Solution Setup  General Options Advanced Hybrid Expression Cache Derivatives Defaults  Initial Mesh Options  Do Lambda Refinement Lambda Target: 0.3333  Use Default Value Use Free Space Lambda
Solution Frequency: <sup>©</sup> Single <sup>©</sup> Multi-Frequencies <sup>©</sup> Broadband          Frequency          1.9 <sup>©</sup> GHz          Maximum Number of Passes          12 <sup>®</sup> Maximum Delta S           0.02 <sup>®</sup> Use Matrix Convergence           Set Magnitude and Phase             Use Defaults           Use Defaults	Adaptive Options         Maximum Refinement Per Pass:       30       %         Maximum Refinement:       1000000         Minimum Number of Passes:       4         Minimum Converged Passes:       1         Solution Options       1         Order of Basis Functions:       First Order         C Direct Solver       •         If terative Solver       Relative Residual:         Relative Residual:       1e-06         C Domain Decomposition       0.0001
OK Cance	OK Cance

After meshing, we define a broadband analysis. Select the previously defined solution (**Project Manager > Analysis > Setup1**) and add a broadband solution around the frequency corresponding to the single frequency ssolution **HFSS > Analysis Setup > Add Frequency Sweep**.

ANSYS Electronics Desktop - antena3 - HFSSDesig	n1 - 3D Modeler - [antena3 - HFSSDes HESS Tools Window Help	ign1 - Modeler]
Joint Coll     Joint Coll     Joint Coll       Joint Coll     Joint Coll     Select: Object       Save     Paste X Delete     Select       Desktop     View     Draw	Solution Type Solution Type Validation Check Solution Check	
Project Manager P × P & Model	Edit Notes	HAHA
B	Toolkit	·
	3D Model Editor	HHHH
E-	Design Settings	AHAH
Excitations	Model	·HHHH
Hybrid Regions     Hybrid Regions     H	Boundaries	
Analysis	Hybrid	· ATATA
Optimetrics	Mesh Operations	• #####################################
	Analysis Setup	<ul> <li>Add Solution Setup</li> </ul>
Port Field Display	Optimetrics Analysis	Add Frequency Sweep
	Fields	<ul> <li>List</li> </ul>
30	Radiation	<ul> <li>Revert to Initial Temperature</li> </ul>

eneral   Interpola	ation   Defa	aults			
Sweep Name:	Sweep				Enabled
Sweep Type:	Interpola	iting	•		
-Frequency Sv	weeps [281	points defined	]		
Dist	tribution	Start	End		
1 Linear St	tep	0.5GHz	3.3GHz	Step size	0.01GHz
		Add below	Delete :	selection	Preview
					20.0
				Time Domai	n Calculation
				Time Domai	n Calculation
				Time Domai	n Calculation
			1	Time Domai	n Calculation

Select **HFSS** > **Validation Check** to verify that all required steps have been completed.

HFSSDesign1	<ul> <li>Design Settings</li> <li>3D Model</li> <li>Boundaries and Excitations</li> <li>Mesh Operations</li> <li>Analysis Setup</li> <li>Optimetrics</li> </ul>
Abort	Y Hadiation

Select **HFSS** > **Analyze All** to start the simulation. Click the **Show Progress** button to view the progress of the solver.



When the simulation is finished, we plot the results that provide useful information about the antenna.

Select **HFSS** > **Results** > **Create Modal Solution Data Report** > **Rectangular Plot** to display the S parameter corresponding to input port P1 (in dB). The resonant frequency of the antenna can be found. To find out the exact value, you can place a marker on the screen in

the desired position, either by right-clicking on the graph and choosing the **Marker > Add Marker** command or from the menu **Report2D > Marker > Add Marker**.



# Activity in the laboratory

1. Using the formula c /  $f = \lambda$  calculate the length of the wires required to obtain the resonant frequency corresponding to your own data.



2. Create the HFSS project according to the instructions in this lab with the dimensions corresponding to your assigned data (length and radius). Analyze the project. It is recommended to work with parameterized model (using variables) as it will significantly ease the work for the next laboratory. Most likely the resonant frequency will not be exactly the desired one, which is not a mistake. The antenna design activity will be continued in laboratory 5 to obtain the required frequency.

3. Save the project with your assigned data. Create an archive from the menu ("\*.aedtz") with the simulation results included (set the "Results/solution files" option) and upload the archive to the lab server (it **must** be a **single** file).

**Very important!** If the archive size (including results) exceeds 10MB, it will not be possible to upload it directly to the server. In this case, use a cloud service (such as Google Drive or Microsoft OneDrive) where you upload the archive and submit to the server a download link to this file (check from another device/computer that the link allows download without username / password).

### **Reference**

MARK JONES, Ansys High Frequency Structure Simulator (HFSS) Tutorial <u>https://indico.fnal.gov/event/13068/contribution/10/material/slides/1.pdf</u>