# Ansoft High Frequency Structure Simulator



# Tutorial

# The Dipole Antenna

April 2004

In this tutorial, a dipole antenna will be constructed and analyzed using the HFSS simulation software by Ansoft. The example will illustrate both the simplicity and power of HFSS through construction and simulation of this antenna structure. The following notes will provide a brief summary of goals.

- ✓ General navigation of software menus, toolbars, and quick keys.
- ✓ Variable assignment.
- Overview of commands used to create structures.
- ✓ Proper design and implementation of boundaries.
- ✓ Analysis Setup.
- ✓ Report Creation and options.

From the Project Manager window Right-Click the project file and select Save As from the sub menu. Name the file "dipole" and Click Save.

		×
+ FE Pr	niect1	
	🛍 Paste	
	Insert	•
	⊆lose	
	Save .	Ctrl+S
	Save <u>A</u> s	
	Project Variables	
	Datasets	
	Remove Unused Definitio	05.55
Project		

To begin working with geometries, you must insert an HFSS design. Right-Click the project file and select Insert -> Insert HFSS Design from the menu.



> Note:

Always create a personal folder to store all HFSS projects. You may find that you do not have access rights to some portions of the hard drive. This will also allow the user to quickly backup/copy data from projects.

Due to the nature of this design we will use Driven Modal as the solution type. From the HFSS menu select Solution Type and Driven Modal. The units are chosen as *mm* by choosing the heading 3D modeler and Units from the menu.





HFSS relies on variables for any parameterization / optimization within the project. Variables also hold many other benefits which will make them necessary for all projects.

- ✓ Fixed Ratios (length, width, height) are easily maintained using variables.
- Optimetrics use variables to optimize the design according to user-defined criteria.
- All dimensions can be quickly changed in one window as opposed to altering each object individually.

Click the HFSS heading and select Design Properties at the bottom of the menu.



Variable Definition

This will open the variable table. Add all variables shown below by selecting Add. Be sure to include units as needed.

Name	Value	Unit	Description	Read-only	Hidden
lambda	10	mm			
dip_rad	lambda/200				
radiation_rad	dip_rad+(lambda/4)				
gap_src	0.125	mm			
dip_length	res_length/2-(gap_src/2)				
radiation_height	gap_src/2 + dip_length + lambda/10				
res_length	.475*lambda				
	Population Service				

### > Note:

Creating variables before defining the structure will allow the user to build the geometry much faster than using a fixed system. The 3D Modeler toolbars play a vital role in the creation of geometric structures within HFSS. By default, the 3D modeler toolbars should be visible on the screen. If you cannot locate the toolbar then right-click on the upper border of the form and select them from the drop down menu.

^∨ ^∨ `) `) □ ○ ○ ○ 10 17 17 10 △ ○ ◎ ≓ ∅ № ∘ *±* XY ▼ 3D ▼

We will start by creating the dipole element using the Draw Cylinder button from the toolbar. Choose 3 Arbitrary points inside the drawing area. These points will be defined using the variables created previously so there is no need to specify points.



Follow the format below for structure size. Give the name *dip1* to this object. Assign the material *PEC* and click  $\mathbb{OK}$ . *PEC* (Perfect Electric Conductor) will create ideal conditions for the element.

Di proc		Unit	Description	Read-only
Indine	dip1			
Material	pec			
Solve Inside				
Urientation	Global			
Model	<ul> <li>Image: A start of the start of</li></ul>	1. I.		
Display Wireframe				
Color	Edit			
Transparent	0			

Name	Value	Unit	Description
Command	CreateCylinder		
Coordinate System	Global		
Center Position	Omm , Omm , gap_src/2		
Axis	Z		
Radius	dip_rad		
Height	dip_length		

Under the Command tab, enter the following information:

The next command is essential when building symmetric structures. Right-Click the drawing area and select Edit -> Duplicate -> Around Axis.

<ul> <li>Select Objects</li> <li>Select Eaces</li> <li>F</li> </ul>	(FFFF
Next <u>B</u> ehind B <u>A</u> ll Object Faces	THE ALL
Measure	· ####
⊻iew	· 7444
Edit	Ctrl+C
Assign Material Assign Boundary	Reste Ctrl+V
Assign Excitation	Properties
Assign Mesh Operation	<u>Arrange</u>
Plc 🛺 Along Line	Duplicate
Plc 🔁 Around <u>A</u> xis	<u>S</u> cale
Cc	<u>S</u> urface <u>B</u> oolean
	Sweep
	Delete Last Operation

A mirror image is produced by enter the following:

Axis:	OX CY CZ
Angle:	180deg 💌 deg 💌
otal number:	2

The dipole structure is illustrated below:



Ideally, the structure is one solid geometry. A slot has been created at the origin in this example. This will allow later placement of a source for excitation.

In the section the user will create a Lumped Gap Source. This will provide an excitation to the dipole structure. Begin by selecting the YZ plane from the toolbar. Using the 3D toolbar, click Draw Rectangle and place two arbitrary points within the model area.

▶ <u>∫</u> ±	1 4	0	~m
o 🕁	XY 💌	In Pl	ane
+	XY YZ		15.2
-2	XZ	5	2

8 🧆 🗈	P 12	a 🚮, '
<u>)</u>	000	00
Drav	vrectangle	•

### Enter the following:

Name	Value	Unit	Description	Read-onl
Name	source			
Orientation	Global			
Model				
Display Wireframe				
Color	Edit			
Transparent	0.2			

Name	Value	Unit	Description
Command	CreateRectangle		
Coordinate System	Global		
Position	0mm,-dip_rad,-gap_src/2		
Axis	X		
YSize	dip_rad*2		
ZSize	gap_src		

Click the Command tab and enter the following:

Please note that the variable *gap\_src* was chosen relatively small in comparison to the dipole structure. This was done to minimize effects due to the source and place emphasis upon the structure. The source is depicted below.



With the source geometry in place, the user must provide an excitation. A lumped port will be used for the dipole model. This excitation is commonly used when the far field region is of primary interest. In the project explorer, right-click Excitation -> Assign -> Lumped Port.



Name the port *source* and leave the default values for impedance. Click Next and enter the following:

mode	Integration Line	Characteristic Imp. (20)
	None	Zpi
	None New Line	
	INOW EITON.	

Using the mouse, position the cursor to the bottom-center of the port. Ansoft's snap feature should place the pointer when the user approaches the center of any object. Left-click to define the origin of the E-field vector. Move the cursor to the top-center of the port. Left-click to terminate the E-field vector. Click finish to complete the port excitation. In this section, a radiation boundary is created so that far field information may be extracted from the structure. To obtain the best result, a cylindrical air boundary is defined with a distance of  $\lambda/4$ . From the toolbar, select Draw Cylinder and choose 3 arbitrary points within the model window.

a, o 👯 📠 🎉 📱	<b>1</b>
	ĝ.
Draw cylinder 🖬 🔳	ŧ
🖯 🖉 Model	~

Enter the following:

Name	Value	Unit	Description	Read-on
Name	radiation			
Material	air			
Solve Inside	v			
Orientation	Global			
Model	✓			
Display Wireframe				
Color	Edit			
Transparent	0.8			
				Show Hir

Name	Value	Unit	Description
Command	CreateCylinder		
Coordinate System	Global		
Center Position	Omm , Omm , -radiation_height		
Axis	Z		
Radius	radiation_rad		
Height	radiation_height*2		

Click the *Command* tab and enter the following:

Assuming all steps were properly completed, the boundary should resemble the illustration below:



With the geometry complete, the actual radiation boundary may now be assigned. From the 3D toolbar select face from the drop down window as shown below.

			S Fa	ce 🔽	
Click	S	and se	elect all faces	as follows:	
			Select Face		×
			Object name: dip1 dip1 1 radiation source	Face ID: Face38 Face39 Face40	
			ОК	Cancel	

With all faces selected, right-click the Boundary icon in the object explorer and select Boundary -> Assign -> Radiation.



Leave the default name Rad1 and click OK.

In this section a solution must be defined to display the desired data. We are primarily interested in the frequency response of the structure. We will also explore HFSS's ability to calculate general antenna parameters such as directivity, radiation resistance, radiation efficiency, etc... . From the project explorer, select Analysis -> Add Solution Setup.



Enter the following:

Solution	Frequency:	29.9792		GHz	-	
🗖 Solv	e Ports Only					
Adaptive	Solutions					
Мах	imum Numbe	er of Passes:	20			
Мах	imum Delta S	i Per Pass:	0.02			
1		Use D	efaults			

Leave all other settings as default. Click  $\mathbb{O}\mathbb{K}$  when complete.

To view the frequency response of the structure, a frequency sweep must be defined. From the project explorer select Setup1 -> Add Sweep.



## Enter the following:

Dweep Type	DC Extrapolation Options
C Discrete	Extrapolate to DC
Fast	Minimum Solved Frequency 0.1 GHz
C Interpolating	Snap Magnitude to 0 or 1 at DC
Error Tolerance 0.5 %	Snapping Tolerance 0.01
Type: Linear Count 💌	Frequency

At this point, the user should be ready to analyze the structure. Before running the analysis, always verify the project by selecting 3D toolbar. If everything is correct the user should see:

V HFSSModel1	<ul> <li>ID Model</li> <li>Boundaries and Excitations</li> <li>Mesh Operations</li> </ul>
/alidation Check completed.	<ul> <li>Analysis Setup</li> <li>Optimetrics</li> <li>Radiation</li> </ul>
Abort Close	

Analyze the structure by clicking

Allow 5-20 minutes for the analysis, depending on the machine.

elle,

After completion of the analysis, we will create a report to display both the resonant frequency and also the radiation pattern. Click on the heading HFSS and select Results -> Create Reports.

Optimetrics Analysis	4744
Analyze	44
Results	Create <u>R</u> eport
<u>F</u> ields	🗟 Solution <u>D</u> ata
Radiation	Output <u>V</u> ariables
Boundary Display (Solver View)	Update Reports
Design Properties	Open All Reports
E Createrateus	Browse Solutions
	⊆lean Up Solutions
element	Import Solutions
	Import Soldtons

Choose the following in the Create Report window:

Create Repor	t 🛛 🔀
Target Design:	HFSSModel1
Report Type:	Modal S Parameters
Display Type:	Rectangular Plot 💌
0	Cancel

×	()	Y		Y-axis Add Blank Trac
Freq	im(Z(sou	rce,source))		Y1
Freq	re[Z(sou	ce,source))		Y1 Remove Trace
				Remove All Trac
ontext		Sweeps X Y	1	
Design: HFSSModel1	<u>×</u>	Category:	Quantity:	Function:
Solution: Setup1 : Swee	p1 💽	Variables	Z(source,source)	ang
Domain: Sweep	•	Output Variables S Parameter		ang_rad dB10
10		Y Parameter Z Parameter		dB20 im
TDR Options.	<i>11</i>	VSWR Gamma		mag re
		Port Zo		-2013t
			1000	
Output Variable	s			Set Lerminations
			Add Trace Re	place Trace

Select the following highlighted parameters and click  ${\rm Add}$   ${\rm Trace}$  to load the options into the Trace window.

Click Done to display a graph of impedance vs. frequency.



Looking at the graph below, both real and imaginary components of the impedance are displayed.

The input resistance can be directly determined from the graph. We will mark the point at which imaginary component crosses zero. This mark will allow the user to determine input impedance at the point of resonance. Right-Click the graph and select 200m ln.



Using the mouse select a zoom window around the imaginary component as it crosses zero.

Mark the zero point by right-clicking the plot window and selecting Data Marker.



Select a point as close as possible to zero along the imaginary line. You will not be able to choose exactly zero due to the resolution chosen (1000 points) in the solution setup. Left-click to mark the point as shown below:



Right-click the plot window and select Fit All. Follow the same procedure to mark the real component at exactly the same frequency of the imaginary component.



Both marked data can be seen in the graph below:

The input resistance of the antenna is 78.83 ohms according to the graph. Performing calculations from a text, the user should compute a resistance between 65 and 75 ohms. The port was previously defined with an impedance of 50 ohms. This will produce sub-optimal results due to mismatched impedance. This will be corrected shortly. In the next step, we will plot  $S_{\rm 11}$  vs. frequency. Create a Report as previously shown and add the following trace:

🛦 Traces			
1 Freq	dB(S(source,source))	Y-ax Y1	Add BlankTrace
<u>.</u>			Remove All Traces
Context Design: HFSSModel1 Solution: Setup1 : Sweep1 Domain: Sweep TDR: Options	Sweeps X Y Category: Category: Category: Variables S Parameter Y Parameter Y Parameter VSWR Gamma Port Zo	Quantity: S(source,source)	Function: ang rad rdB im mag re
Output Variables		Add Trace Replace	Set Terminations
	spply Done	Cancel	

Click Done when complete.



The point of resonance was marked at -14.57dB. In order to compute accurate antenna parameters, the input must be matched. From the project explorer, right-click *source* and select Properties. Adjust the port impedance as shown:

Name: sourc	e			
Resistance:	79	[	Ohm 💌	
Reactance:	0	[	Dhm 💌	

We will now re-analyze the structure with a properly matched port. In order to preserve memory and calculation time, right-click Analysis -> Revert to Initial Mesh in the project explorer.



Re-analyze the structure. When complete create another plot of  $S_{11}$  vs. frequency as shown below:



Note the improved response of -55.81dB at resonance.

HFSS has the ability to compute antenna parameters automatically. In order to produce the calculations, the user must define an infinite sphere for far field calculations. Right-click the Radiation icon in the project manager window and select Insert Far Field Setup -> Infinite Sphere.



Accept all default parameters and click Done. Right-click Infinite Sphere1 -> Compute Antenna Parameters... from the project explorer as shown:

ė 🏆	Radiation	
ject	🕁 Infinite Sp	Rena <u>m</u> e X Delete
		Properties
Name	Value	Compute Antenna Parameters

nputs				2	
Setup Name:	Infinite Sphere1		i i i	пк	
Frequency:	29.9792GHz LastAdaptive				0.0
Solution:				- 11 - 11 - 11 - 11 - 11 - 11 - 11 - 1	Export
Design Variation:	dip lengt	dip length='2.36875mm'			
Array Setup:	None				
itenna Parameters:					
Quantity		Value		Units	
MaxU	Max U		0.1471		W/sr
Peak Directivity	Peak Directivity		1.6826		
Peak Gain		1.8634			
Peak Realized Gain		1.8485			
Radiated Power		1.0986			W
Accepted Power		0.992			W
Incident Power		1		W	
Radiation Efficiency		1.1074			
aximum Field Data:	1 10		fricad	AL DIL:	1
IE Field	Ve	nue	Units	ACENE	At Theta
Tatal	10 501		52	2004	004
Total	10.531	1	V	200deg	90deg
Total X	10.531 4.7775		V V	200deg 180deg	90deg 130deg
Total X Y	10.531 4.7775 4.7607	8	V V V	200deg 180deg 270deg	90deg 130deg 50deg
Total X Y Z	10.531 4.7775 4.7607 10.531		V V V	200deg 180deg 270deg 200deg	90deg 130deg 50deg 90deg
Total X Y Z Phi	10.531 4.7775 4.7607 10.531 0.08711	9	V V V V	200deg 180deg 270deg 200deg 90deg	90deg 130deg 50deg 90deg 90deg
Total X Y Z Phi Theta	10.531 4.7775 4.7607 10.531 0.08711 10.531	9	V V V V	200deg 180deg 270deg 200deg 90deg 200deg	90deg 130deg 50deg 90deg 90deg 90deg

Select all defaults and results are displayed as follows:

From the chart the Peak Directivity is 1.68. Calculations from standard antenna texts will show this model to be approximately 1.63. All other parameters can be seen as slightly elevated above the expected. Adjustments to the radiation boundary might provide more accuracy.

Next, the far field will be plotted. Create Reports as previously shown. Modify the following:

Create Report	X
Target Design: HFSSModel1	*
Report Type: Far Fields	•
Display Type: Radiation Pattern	<b>•</b>
OK Ca	ncel

### Enter the following:

		第二次のない はいみついい
		Hemove Trac
		Remove All Tra
ext	Sweeps Ang Mag	
esian: HESSModel1	Use current Design and Project variable values	
	Sweep Design and Project variable values	🗖 All Values
ution: Setup1 : LastAdaptive 🗾	Name Type Description 0de	g 🔺
netry: Infinite Sphere1	Phi Primary Sweep All Values 10d 20d	eg ea
	Theta Point(s) 90deg 30d	eg
	Freq Point(s) 29.9792GHz 50d	eg
	60d 70d	eg
	800	eg
	100	deg deg
	120	deg
	130	deg
	Edit Sweep Reset 150	deg 📃
Output Visiobles	Apply To All Selected Traces	
oupur vanabies		
	Add Trace Replace T	race

Select the Mag and enter the following:

Ang		М	Add Blank Trac	
Phi	dB(Ga	inPhi)		Remove Trace
Context Design: HFSSModel1 Solution: Setup1 : LastAr	Japtive 💌	Sweeps Ang Ma Category: Variables	g   Quantity: GainTotal	Function:
ieometry: Infinite Sphere1	Y	Output Variables rE Directivity Gain Realized Gain Axial Ratio Polarization Ratio Antenna Params	GainPhi GainTheta GainX GainX GainLHCP GainLHCP GainL3X GainL3Y	abs acosh asin asin atan atanh atanh cos cosh dB dBm dBm dBW even exp int j0
Output Variables	har i	E	Add Trace Repla	Set Terminations

Select Add Trace and click Done when complete.



# The radiation pattern is displayed below:

This tutorial was intended to introduce the user to the basic commands and the general procedure for simulation. The software is complex in nature and requires extensive knowledge Electromagnetic Theory to fully utilize its capabilities. The dipole antenna was intended to accelerate the software learning curve and generate interest in the subject.

# 射频和天线设计培训课程推荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立,致力并专注于微 波、射频、天线设计研发人才的培养;我们于 2006 年整合合并微波 EDA 网(www.mweda.com),现 已发展成为国内最大的微波射频和天线设计人才培养基地,成功推出多套微波射频以及天线设计经典 培训课程和 ADS、HFSS 等专业软件使用培训课程,广受客户好评;并先后与人民邮电出版社、电子 工业出版社合作出版了多本专业图书,帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、 研通高频、埃威航电、国人通信等多家国内知名公司,以及台湾工业技术研究院、永业科技、全一电 子等多家台湾地区企业。

易迪拓培训课程列表: http://www.edatop.com/peixun/rfe/129.html



#### 射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电 路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材; 旨在 引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和 研发设计能力。通过套装的学习,能够让学员完全达到和胜任一个合格 的射频工程师的要求…

课程网址: http://www.edatop.com/peixun/rfe/110.html

#### ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程,共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系 统设计领域资深专家讲解,并多结合设计实例,由浅入深、详细而又 全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设 计方面的内容。能让您在最短的时间内学会使用 ADS,迅速提升个人技 术能力,把 ADS 真正应用到实际研发工作中去,成为 ADS 设计专家...



课程网址: http://www.edatop.com/peixun/ads/13.html



#### HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程,是迄今国内最全面、最 专业的 HFSS 培训教程套装,可以帮助您从零开始,全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装,更可超值赠送 3 个月 免费学习答疑,随时解答您学习过程中遇到的棘手问题,让您的 HFSS 学习更加轻松顺畅…

课程网址: http://www.edatop.com/peixun/hfss/11.html

#### CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出,是最全面、系统、 专业的 CST 微波工作室培训课程套装,所有课程都由经验丰富的专家授 课,视频教学,可以帮助您从零开始,全面系统地学习 CST 微波工作的 各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装, 还可超值赠送 3 个月免费学习答疑…



课程网址: http://www.edatop.com/peixun/cst/24.html



#### HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书,课程从基础讲起,内容由浅入深, 理论介绍和实际操作讲解相结合,全面系统的讲解了 HFSS 天线设计的 全过程。是国内最全面、最专业的 HFSS 天线设计课程,可以帮助您快 速学习掌握如何使用 HFSS 设计天线,让天线设计不再难…

课程网址: http://www.edatop.com/peixun/hfss/122.html

#### 13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程,培训将 13.56MHz 线圈天线设计原理和仿 真设计实践相结合,全面系统地讲解了 13.56MHz 线圈天线的工作原理、 设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体 操作,同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过 该套课程的学习,可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹 配电路的原理、设计和调试…



详情浏览: http://www.edatop.com/peixun/antenna/116.html

#### 我们的课程优势:

- ※ 成立于 2004 年, 10 多年丰富的行业经验,
- ※ 一直致力并专注于微波射频和天线设计工程师的培养,更了解该行业对人才的要求
- ※ 经验丰富的一线资深工程师讲授,结合实际工程案例,直观、实用、易学

#### 联系我们:

- ※ 易迪拓培训官网: http://www.edatop.com
- ※ 微波 EDA 网: http://www.mweda.com
- ※ 官方淘宝店: http://shop36920890.taobao.com

专注于微波、射频、大线设计人才的培养 **房迪拓培训** 官方网址: http://www.edatop.com

淘宝网店:http://shop36920890.taobao.cor