ADS Fundamentals - 2001

LAB 7: Harmonic Balance Simulations

Overview - This exercise continues the amp_1900 design and shows the fundamentals of using the Harmonic Balance simulator to look at the spectrum, analyze compression, calculate TOI, and perform other non-linear measurements.

OBJECTIVES

? ***Set up and perfrom a 1 tone HB simulation.***

? ***Set up and perform a 2 tone HB simulation.***

? ***Use variables for simulation and source control.***

? ***Test Gain, Compression, Available Power, Noise Figure, IP3, and other specifications.***

? ***Use the ts transform on HB data.***

? ***Work with equations, plots, and the Mix table.***

Agilent Technologies

TABLE OF CONTENTS

PROCEDURE

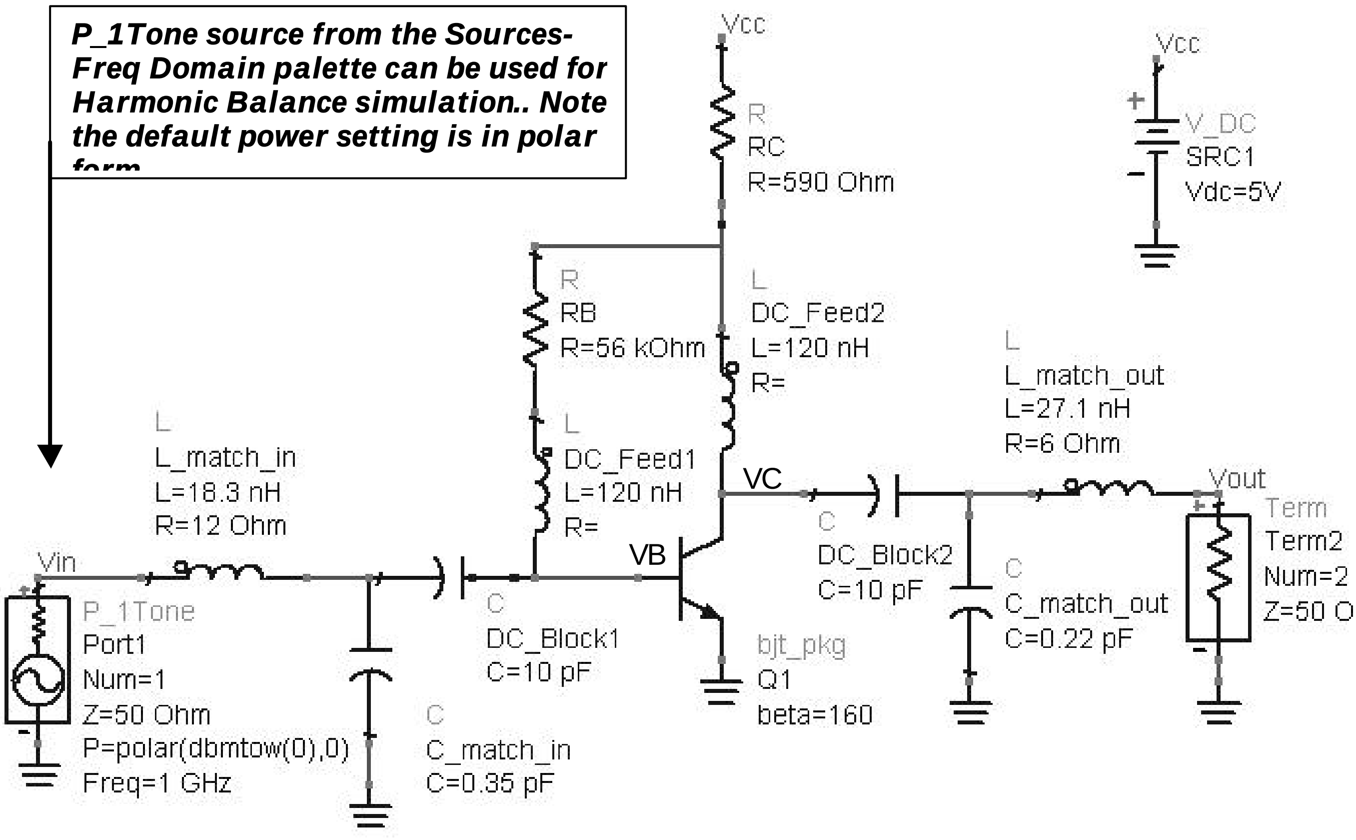
1. Set up the circuit with a P_1Tone source.

a. Open the amp_1900 project and schematic: s_final_match

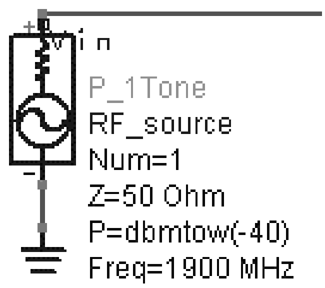
b. Save the s_final_match schematic with a new name: hb_basic. Then delete all the simulation and measurement components and the input termination to begin building the setup shown here.

c. Insert a P_1Tone source for the RF input.

d. Insert 4 pin labels (node names) Vin, Vout. VC and VB as shown so that the voltages will be available in the dataset.



e. Set the RF source as shown where the polar function is removed and only the dbm-to-watts function remains as: P=dbmtow (-40). Also, rename the source



RF_source as shown. The port number is defined by Num=1.

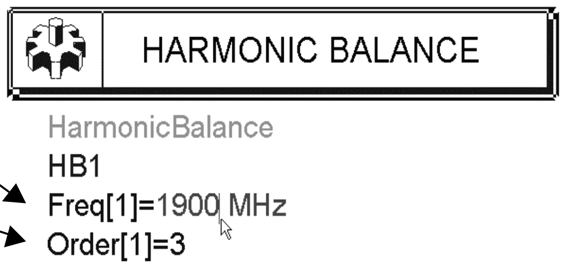
2. ***Set up a one-tone Harmonic Balance simulation .***

   a. ***Go to the Simulation_HB palette and insert HarmonicBalance simulation controller as shown here.***

   b. ***Edit the Freq setting on the screen: change it to Freq [1] = 1900 MHz so that it matches the Freq setting in the P_1Tone source.***

Freq[1]must be set to match the source frequency.

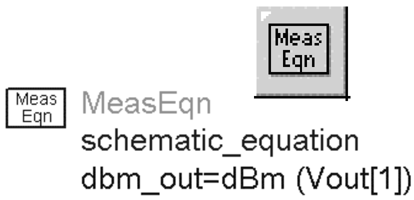
Order is the number of harmonics.



3. ***Write a measurement equation for dBm of Vout and simulate.***

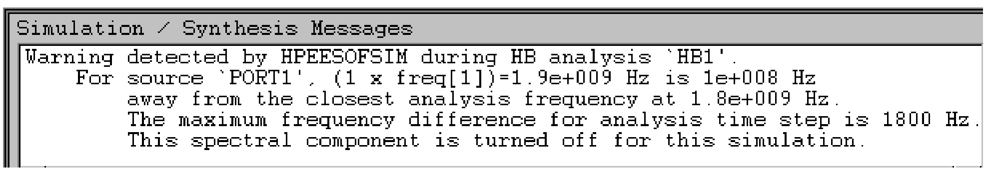
   a. ***From the simulation palette, insert a measurement equation.***

   b. ***Write the equation to calculate the output power at Vout in dBm: dbm_out = dBm (Vout[1]). The bracked [1] refers to the index value the calculated frequencies in the analysis. With Order = 3, the index values are: index [0] is the DC component, inded [1] is 1900 MHz, index [2] is the second harmonic or 3800 MHz and index [3] is the third harmonic. Therefore, the equation should produce the output power in dBm for 1900 MHz only.***



   c. ***Simulate – you should have no warnings or error messages.***

   d. ***Change the HB controller to: Freq[1]= 1800 MHz. Now, simulate again and read the error message - the source is 100 MHz away from the HB frequency of 1800 MHz. This is a common error when the source and controller do not agree.***

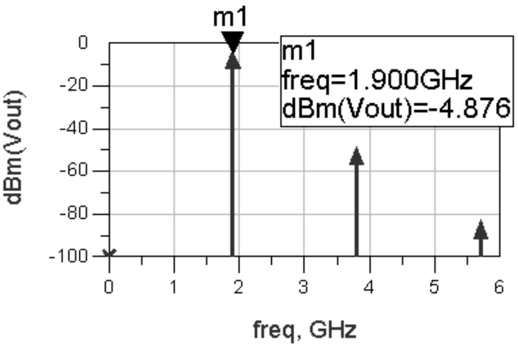


e. ***Reset the HB controller Freq[1] = 1900 MHz and simulate again.***

4. ***Plot the spectrum, equation, and ts of the node voltages.***

   a. ***In the data display, plot dBm of Vout. Also, insert a list of dbm_out. Whenever you write a measuremet equation, it will appear in the dataset. The two values should be the same as shown here.***

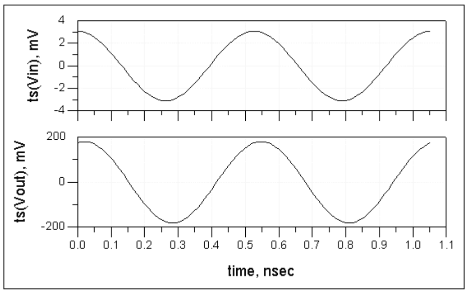
   b. ***Put a marker on the fundamental and verify that your amplifier has about 35 dB of Gain with output power in dBm = -4.876 at 1900 MHz.***



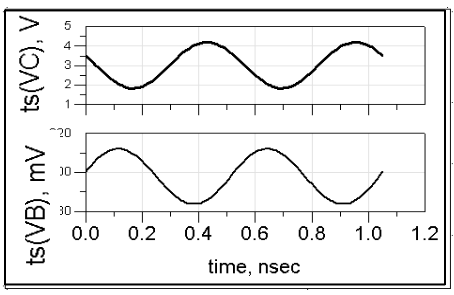
dbm_out
-4.876

NOTE on results: With Order set to 3 in the simulation controller, you get 3 tones: fundamental plus two harmonics. The DC component also shows up on the plot because Harmonic Balance always computes DC for convergence.

   c. ***Insert a stacked rectangular plot and insert two data traces as time domain signals: Vin and Vout. The ts (time series) function operates on HB and transforms it into the time domain. In this case, you can see that the amplifier does not invert the signal as you might expect. These will be two separate plots in one frame. Put markers on the same time point as shown.***

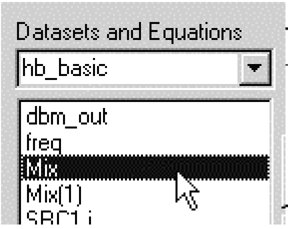


   d. ***Edit the Y axis label on the trace by changing Vout to VC and changing Vin to VB as shown here. Now you can see the inversion. This means that the matching network probably has a great effect on the phase for Vout.***



5. *Operate on Vout and Mix using functions and indexing.*

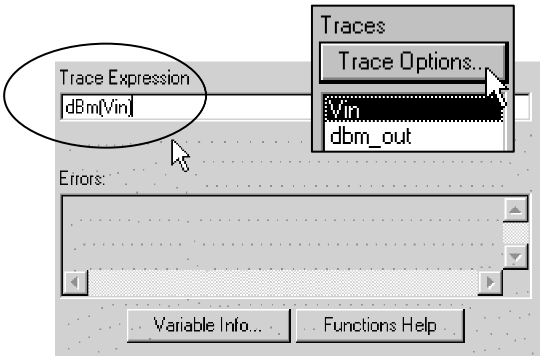
a. Insert a list of Vout and Mix as shown here. Notice that Vout is always complex (mag and angle) until you operate on it using dB, dBm, etc. Also, whenever a HB simulation is performed, a Mix table is created in the dataset. In the next steps, you will learn how to write equations to display or operate on precise tones in the Mix table. This is especially useful whenever mixing occurs.



freq	Vout	Mix
0.0000 Hz	0.000 / 0.000	0
1.900GHz	0.180 / -14.199	1
3.800GHz	0.001 / -170.939	2
5.700GHz	1.963E-5 / 46.135	3

Index value of 1900 MHz = 1. The DC

b. Edit the list and add Vin. Then select Trace Options and edit Vin with the dBm function: dBm(Vin) and click OK. If necessary you can get variable information (dependencies ,etc.) or Help (documentation manuals) on functions this way or by inserting an equation.



c. Your list should contain the schematic equation dbm_out and the expression dBm (Vin) for all frequencies as shown. Now, edit the dBm(Vin) data by inserting the index value [1] in the Vin argument and you get the single value of interest at 1900 MHz.

a. Insert the [1] in the dbm_out equation - it becomes invalid because it was indexed on the schematic.

freq	dBm(Vin)	dbm_out
0.0000 ...	<invalid>	-4.876
1.900GHz	-40.214	
3.800GHz	-82.943	
5.700GHz	-111.161	

dBm(Vin[1])	dbm_out
-40.214	-4.876

dBm(Vin[1])	dbm_out[1] <invalid>
-40.214	<invalid> <invalid>

Adding [1] to the Vin data returns the 2nd index value = 1900 MHz.

Adding [1] to the measurement equation makes

b. ***Remove [1] from the invalid dbm_out equation to make it valid again.***

c. ***Insert the cursor in the expression dBm(Vin[1]) and add a comma and 50 as shown here. The second argument in the dBm function is Zin. If no argument is given, the default is 50 ohms.***

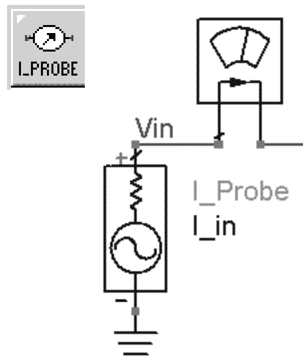
dBm(Vin[1],50)	dbm_out
-40.214	-4.876

Insert Zin: 50 as the 2nd argument separated by a comma.

Important NOTE on dBm function and Zin of your designs - The dBm function converts a voltage into dBm. But it assumes an exact 50 ohm system. For your amplifier, if Zin was exactly 50 ohms, then the delivered power to the load would be the value listed. However, because the dBm function is so widely used with Harmonic Balance and AC analysis, it is important to insert the correct Zin in the equation as you will see in the next step.

6. ***Simulate Power Delivered and Zin.***

a. ***In the schematic, insert a current probe from the Probe Components palette. Change the instance name to: I_in. You will use this in an equation.***



b. ***Simulate and when completed, write a <u>data display</u> equation for average delivered power using the probe I_in. Note that 0.5 gives the average of the peak value, the conj function converts the complex current to its conjugate because V&I must be in phase to dissipate power and + 30 converts the value to dBm (same as dividing by 0. 001):***

Eqn P_del_dBm = 10*log (0.5*real (Vin[1]*conj(I_in.i[1])))+30

c. ***Write another equation to calculate Zin from the probe current and list the value. This Zin is calculated using –40 dBm RF in, so it will be slightly different from the small signal S-parameter simulation value you got in the last lab.***

Eqn Z_in = Vin[1] / I_in.i[1]

Z_in
47.619 / 0.686

d. ***Edit your list and add the P_del_dBm equation and the dBm(Vin[1],Z_in) equation where [1] is 1900 MHz and you use the Zin argument instead of 50. The two equations using Zin are different from the dBm of Vin which uses a default 50 ohms***

Uses Zin from HB	***Uses default 50 ohms***	***Uses log of power + 30***
dBm(Vin[1],Z_in)	dBm(Vin[1])	P_del_dBm
-40.003	-40.214	-40.003

input impedance.

7. Test for Gain Compression using the XDB simulator.

The XDB simulation controller is a special use Harmonic Balance simulation for gain compression.

a. Save all you current work: schematic and data. Then save the schematic with a new name: hb_compression. Afterward, close the hb_basic window and the data display.

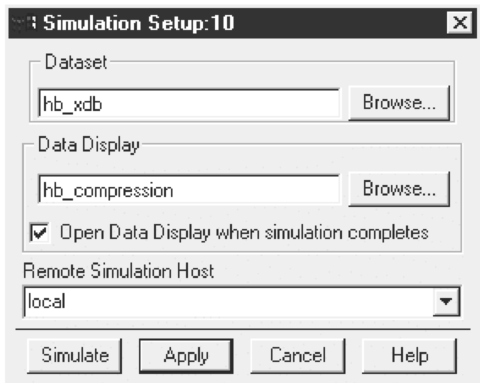
b. In the new schematic, deactivate the HB1 controller.

c. Go to the Simulation-XDB palette and insert the XDB controller. Edit the controller on screen so that Freq[1] and GC input and output frequencies are all 1900 MHz as shown. The parameter GC_Xdb = 1 means that the test will be for 1 dB compression. Later, if you wanted 3 or 6 dB compression, simply change the value.

XDB
HB2
Freq[1]=1900 MHz
Order[1]=3
GC_XdB=1
GC_InputPort=1
GC_OutputPort=2
GC_InputFreq=1900 MHz
GC_OutputFreq=1900 MHz
GC_InputPowerTol=1e-3
GC_OutputPowerTol=1e-3
GC_MaxInputPower=100

d. In the Simulation Setup, change the Dataset name to hb_xdb and then Simulate.

e. When the data display opens, insert a list of inpwr and outpwr. Then edit directly on the list by inserting a bracketed one [1] on each and title the plot as shown here. Whenever you want a quick test of gain compression, simply use XDB. For this amplifier, it is biased quite high and therefore the 1dB compression point occurs when the input power is about –29 dBm as shown here.

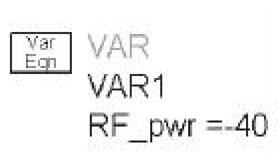


freq	inpwr	outpwr
0.0000 Hz	-30.67 dBm	3.498 dBm
1.900GHz	-30.67 dBm	3.498 dBm
3.800GHz	-30.67 dBm	3.498 dBm
5.700GHz	-30.67 dBm	3.498 dBm

Gain Compression at 1900 MHz

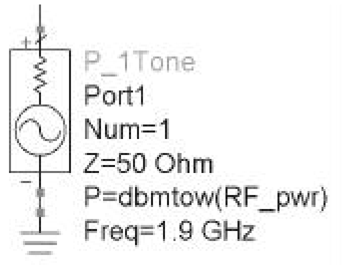
inpwr[1]	outpwr[1]
-30.671	3.498

8. Simulate compression with a power sweep.



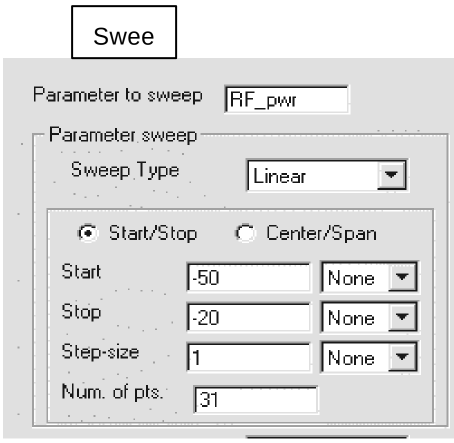
a. Insert a variable equation VAR for RF_pwr = -40.

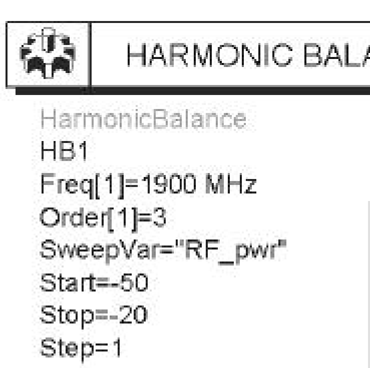
b. Set the RF source power to the variable: P=dbmtow (RF_pwr).



c. <u>Deactivate the XDB</u> <u>and activate the HB controller.</u>

d. Edit the HB controller. In the sweep tab, set the RF_pwr sweep as shown from –50 to –20, step 1.

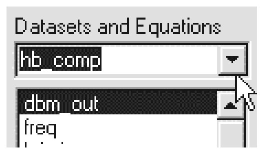


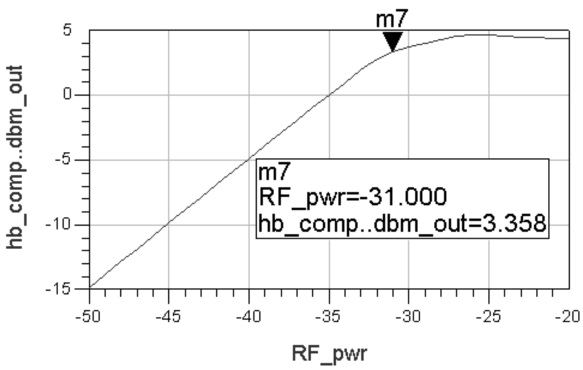


e. Go to the Display tab and set the SweepVar and its values to be displayed on the HB component.

f. Change the dataset name to: hb_comp and simulate. When the data display window opens, answer No to changing the dataset - this will keep the XDB data valid as the default dataset, so you will have to explicitly plot the hb_comp data – this is common practice.

g. In a plot and select the hb_comp dataset. Then plot the schematic measurement equation dbm_out. Insert a marker on the trace where the value of RF_pwr is near the XDB inpwr value. As you can see, the two values are close but they differ because the sweep resolutions are





1 dB compression point from XDB

inpwr[1]	outpwr[1]
-30.671	3.498

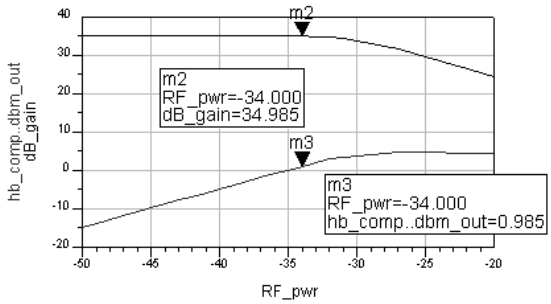
different – the XDB simulation used more points.

9. Write equations for gain and an ideal output power line.

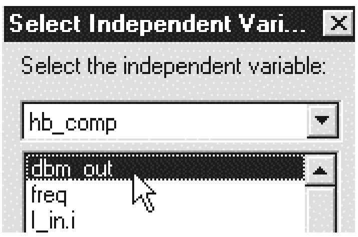
a. Write an equation dB_gain that uses the dbm_out measurement equation. By subtracting linear inpu RF_pwr from output power (dbm_out), the result is the gain.

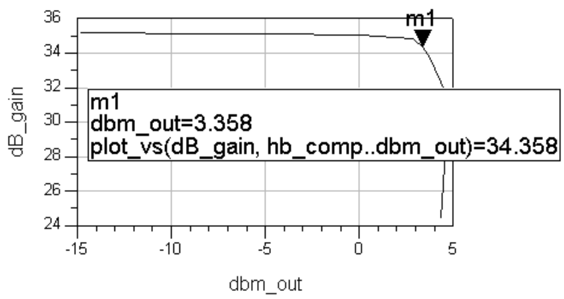
Eqn dB_gain = hb_comp..dbm_out - hb_comp..RF_pwr

b. Add the equation to the plot of dbm_out the Y axis scale will adjust. You can add markers to see both values at one RF power level.



c. Plot dB_Gain against output power. Insert a new plot and select the dB_gain equation and then click AddVs. Next, select the independent variable for the X axis as hb_comp..dbm_out using the dialog. Click OK and the sharp fall of gain will be plotted as shown.

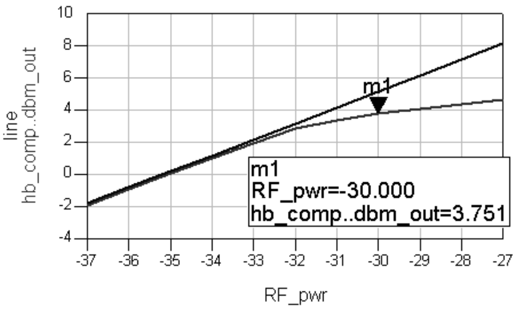




Use markers to read the values.

d. Write one more equation, line, to create a linear line (extrapolated data) that represents the ideal output power with no compression. By adding the uncompressed gain at the first data point [0] to the RF power at every point, you get the ideal gain or line.

e. Insert anew plot of dbm_out and line Then zoom in on the X-axis as shown here.



_gain [0]

f. Save all your work.

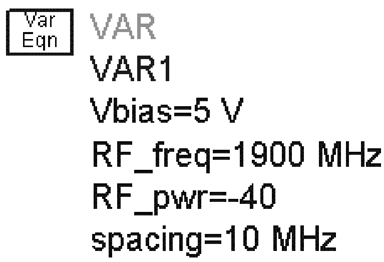
NOTE on controller swept variables – they are output to the dataset by default.

10. Set up a 2-tone simulation with variables

The next few steps show how to use variable instead of "hard coded" numbers in a simulation setup. This is important for more complex circuit refinement, calculations in the remaining labs, and working with ADS examples which use this method.

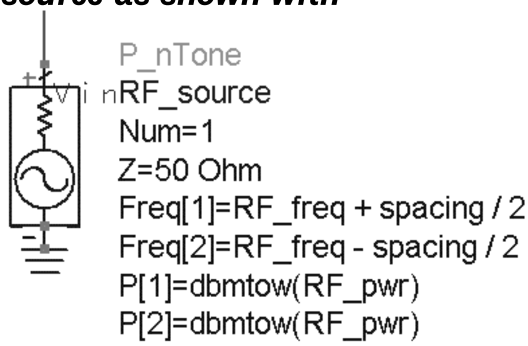
a. Save the last schematic design with a new name: hb_2Tone.

b. Edit the VAR and add variables for RF_freq and spacing as shown here. Vbias is not required - you may or may not have Vbias if you did an optional step earlier.

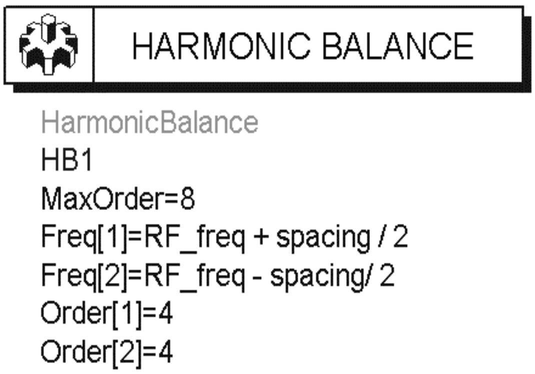


NOTE on units in VARs – If you set units here do not set them anywhere else or they may multiply in the simulation.

c. Change the source to a P_nTone. Edit the source as shown with two tones and their power levels. Set the values with the variables you just created for freq and power as shown.



d. Edit the Harmonic Balance controller as shown here by adding another frequency Freq[2] and set both Order = 4. Also, set MaxOrder = 8. In this case, the two RF tones are spaced 5 MHz



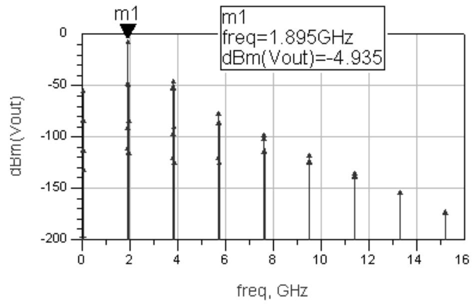
MaxOrder = number of mixing products.

Freq[1] is a variable or a number. Order [1]= 4 means Freq[1] will be calculated with 4 harmonics.

apart (channel spacing).

e. Remove the RF_pwr sweep by erasing it on-screen or in the dialog and display.

f. Simulate and plot the spectrum of Vout in dBm. Put a marker on a tone near 1900 MHz. Notice that you cannot clearly see the adjacent tones. To see the inter-modulation tones, you can either zoom in on the plot as needed or try changing the X or Y axis scaling. Try both of these methods quickly because the next step shows another technique using equations.

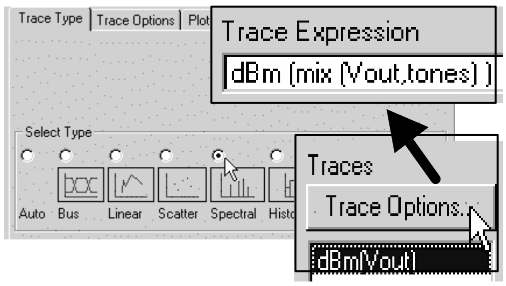


g. Create a matrix with vectors (index values) to the desired data tones. To do this, write the following two equation shown here. The tones equation creates a matrix using the square brackets. Within the brackets are numbers which are the vectors or index values. In this case the number 1 represents an RF tone with the spacing. Zero means no other tone is desired or DC, and 2

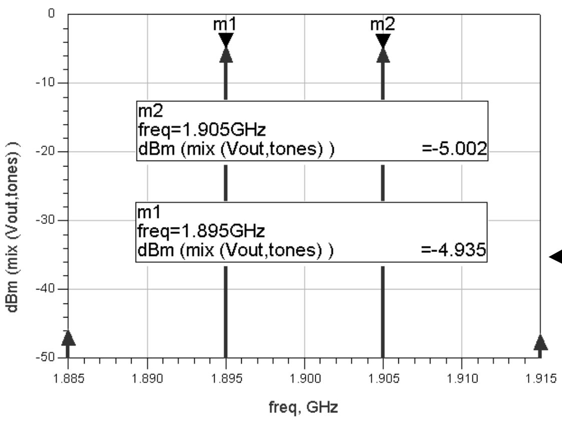
Eqn tones=[{1,0},{0,1},{2,-1},{-1,2}] ← ***Use curly braces within***

is two times an RF tones.

h. Insert a rectangular plot of Vout – spectrum in dBm. Then edit the Trace Expression as shown here using parenthesis: dBm(mix(Vout,tones)). Also, be sure the Trace Type is Spectral.



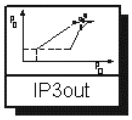
i. The plot should now show only the four tones you specified (10 MHz apart). To verify this, insert a list of Mix (Mix table). The index values from the Mix table are the tones that you specified with the tones eqn.



freq	Mix	
	Mix(1)	Mix(2)
0.0000 Hz	0	0
10.00MHz	1	-1
20.00MHz	2	-2
30.00MHz	3	-3
40.00MHz	4	-4
1.865GHz	-3	4
1.875GHz	-2	3
1.885GHz	-1	2
1.895GHz	0	1
1.905GHz	1	0
1.915GHz	2	-1
1.925GHz	3	-2
1.935GHz	4	-3

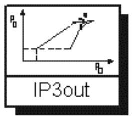
11. Simulate IP3 or TOI (Third Order Intercept)

a. On the schematic, insert two Harmonic Balance IP3out measurement equations: one for the upper and one for the lower spaced tone. Many measurements require two-tones so name the instances upper and lower as shown here.



IP3out
ipo_upper
upper_toi=ip3_out(Vout,{1,0},{2,-1},50)

b. Note the default node label (vout), vectors {1,0}, and impedance 50. To match these values to your circuit, be sure the node is Vout (uppercase V). Then set vectors for the two-tone simulation – these correspond to your Mix table shown here from the last simulation.



IP3out
ipo_lower
lower_toi=ip3_out(Vout,{0,1},{-1,2},50)

2-tone Mix HB data

1.885GHz	-1	2	
1.895GHz	0	1	
1.905GHz	1	0	
1.915GHz	2	-1	

c. Check the equations to be sure they are correct and then simulate.

d. In the Data Display, list the two measurement equation values as shown here. Remove the independent variable using Plot Options. Here the amplifier TOI values appear reasonable and almost symmetrical.

lower_toi	upper_toi
15.679	15.914

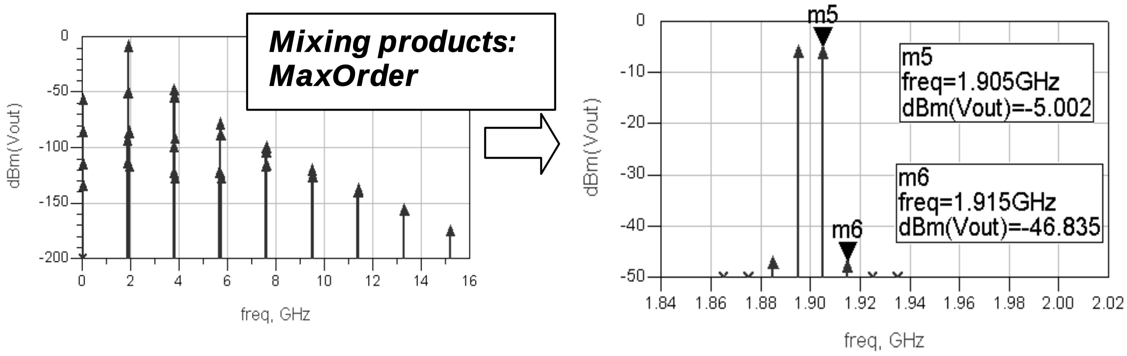
e. As an exercise in controlling data and using ADS functions, write an equation for the same upper_toi measurement in the Data Display and list is as shown here. This step increases your skill in controlling data using ADS functions before and

Eqn my_toi = ip3_out(Vout,{1,0},{2,-1},50)

my_toi
15.914

after simulation.

f. ***<u>Plot the spectrum of Vout in dBm</u> and then zoom in on the plot to see the two tones you just simulated. Put markers on the upper fundamental and the 3rd order tone – these should match the frequency values in the Mix table.***



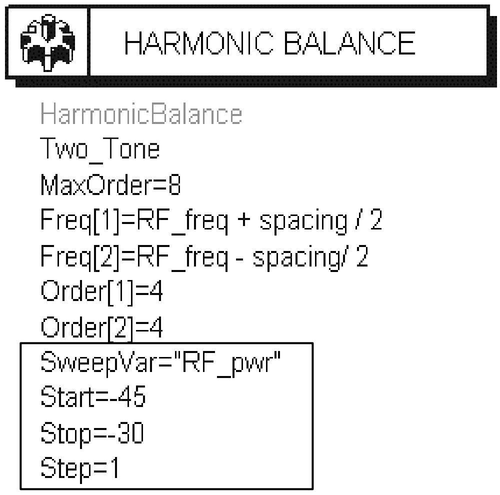
NOTE – You could easily go back to the schematic, change the spacing var and resimulate. All the equations, plots and tables would simply fill up with the new data. This is the value of using variables for simulation and data displays.

g. ***Save the schematic and data display.***

NOTE for Mixer measurements – If you design mixers, the LO should be Freq[1] in the simulation controller because it has the most power. Also, in measurement equations, you will have to treat 2-tone data as if it were 3-tone: LO, RF1 and RF2 for upper and lower tones. For example, the an upper IP3 equations for a down-converter would vector into the data as {-1,1,0},{-1,2,1} where –1 in the first position in each vector represents:minus the LO frequency.

12. OPTIONAL - sweep RF power against the TOI measurement

This step shows the effects on TOI when the input power begins to drive the device toward compression. In general, many measurements can be refined to get a better measure of circuit performance, beyond the required specifications. To do this, you must have a powerful non-linear simulator and data display tool: ADS.



a. ***Using the same design hb_toi, set up the HB simulation controller to sweep the RF power as shown here from –45 to –30 dBm. You already tested 1 dB compression to be about –31 dBm RF input power and you just finished measuring TOI which was about 15dBm.***

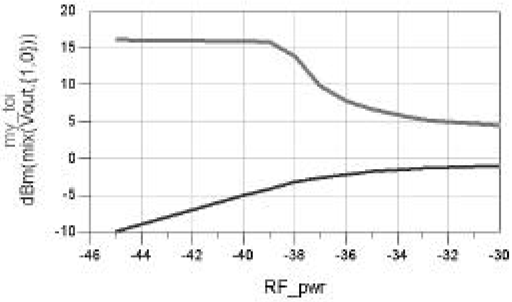
b. ***Simulate and watch the changes in the data display.***

c. ***Edit the my_toi list to include the independent data (RF_pwr). Then increase the list size so that all the values appear. As you can see, TOI begins to change greatly as RF_pwr moves higher. However, the change is not linear. The next step will show this with more refinement.***

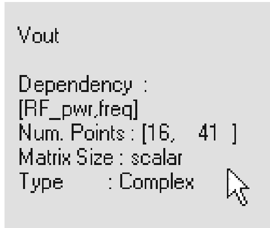
RF_pwr	my_toi
-45.000	16.110
-44.000	16.086
-43.000	16.057
-42.000	16.019
-41.000	15.973
-40.000	15.914
-39.000	15.841
-38.000	13.852
-37.000	9.824
-36.000	7.795
-35.000	6.637
-34.000	5.929
-33.000	5.305
-32.000	4.888

d. ***Change the list of my_toi to a rectangular plot (Plot Options - click the plot type icon). Then, on the same plot, insert Vout in dBm and edit the trace expression to return the upper RF tone using curly braces for the vector:***

dBm(mix (Vout, {1,0}))



2 traces: my_toi and dBm of upper RF tone.

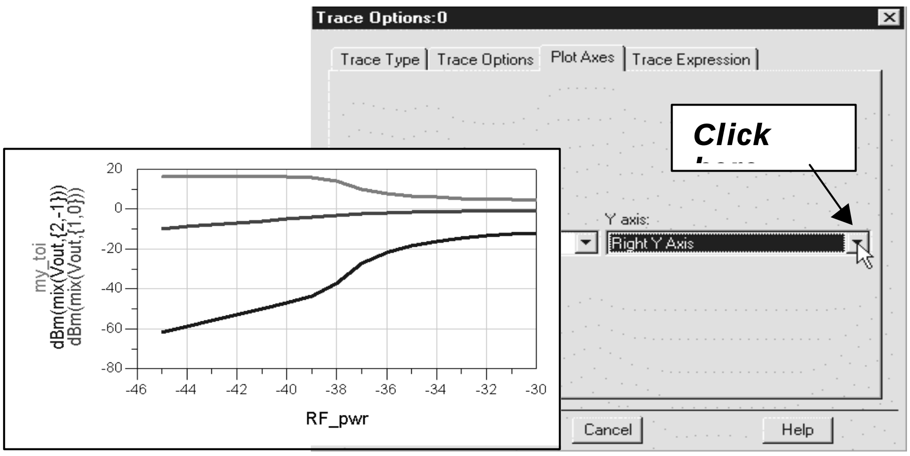


NOTE on Vout data – You must use the mix function because Vout contains 41 total frequency tones: 2 spaced fundamentals with 4 harmonics (this means 8 tones), with 8 max_order (this means 32 more intermod tones), plus the dc component. These 41 tones are present at each of the 16 values of RF power.

e. ***Add one more Vout trace to the plot. Again, edit the trace (Trace Expression) and this define the upper 3^{rd} order product:***

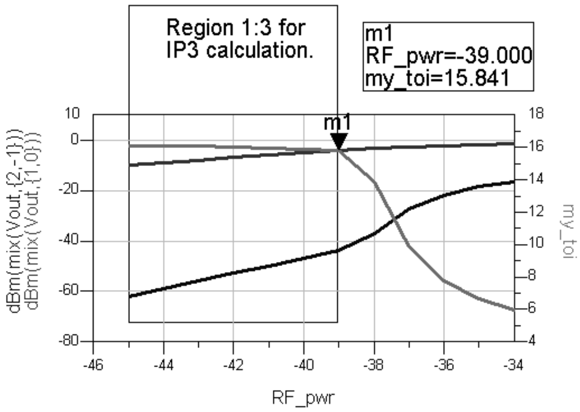
dBm (mix (Vout, {2,-1}))

f. ***Your plot should now look like the one shown here. It should contain the upper RF_freq, the upper 3^{rd} order product, and the equation my_toi (upper toi). Now, edit the my_toi trace and select Plot Axes as shown here. Then select Right Y axis for this trace and watch the change.***

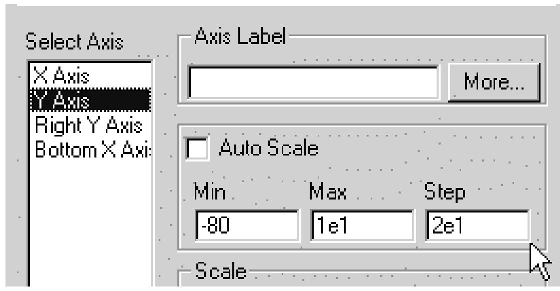


g. ***Your plot should now have the TOI value from your equation on the Right Y axis and the two tones used to calculate TOI on the left. Now, use Plot Options, select Y Axis, and remove the Auto Scale (uncheck the box). Then increase the Max to 10 and click OK. Finally, place a marker on the point where the slope of the two tones is no longer 3:1. As you can see, IP3 was calculated in the correct region. However, after the marker, the 3^{rd} order product begins to rise at a sharper rate. This is a good***

example of using ADS to learn more about the performance of your design, beyond the specification.

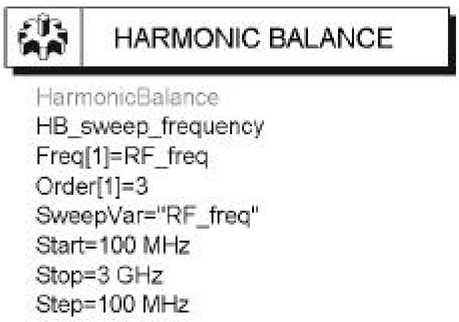


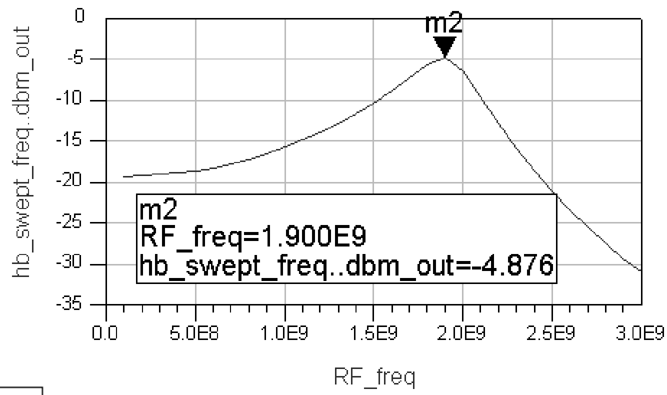
Make your plot look similar using the Data Display text and drawing features.

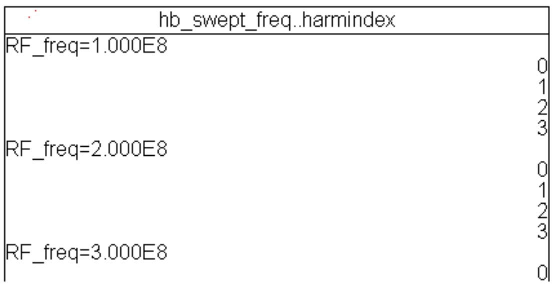


EXTRA EXERCISES:

1. ***<u>Swept RF frequency</u> - Copy the schematic and then change the swept variable from RF power to one tone RF freq. To do this, set up the VAR for RF_freq in both the controller and the source. Sweep RF_freq from 100 MHz to 3GHz in 100 MHz steps. Be sure to change the dataset name, then simulate and plot the output power equation against the swept freqnency as shown. Also, note that the dataset will contain a list of the harmonic index as shown.***







	hb_swept_freq..harmindex
RF_freq=1.000E8	
	0
	1
	2
	3
RF_freq=2.000E8	
	0
	1
	2
	3
RF_freq=3.000E8	
	0

2. ***Try writing an equation to pass all the 5th order products to a spectral plot.***

3. ***Use the pspec function to calculate power gain to the load. To do this, first look at the Help for pspec. Then insert a current probe at the Vout node.***

THIS PAGE IS INTENTIONALLY BLANK.

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

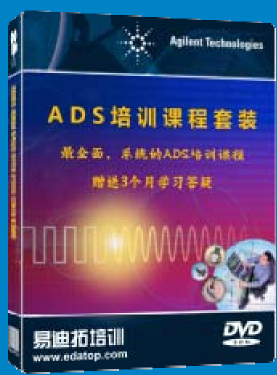
射频工程师养成培训课程套装

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…

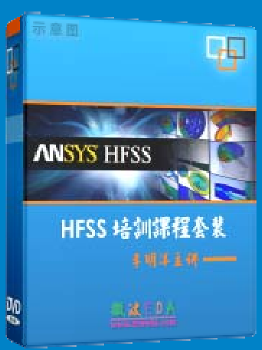
课程网址：http://www.edatop.com/peixun/rfe/110.html

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