ADS Fundamentals - 2001

LAB 8: Circuit Envelope Simulation

Overview - This chapter shows the basics of Circuit Envelope to measure time and frequency of an output signal when the input is a pulsed or modulated source such as GSM, CDMA, etc.

OBJECTIVES

? ***Set up Circuit Envelope simulations using a behavioral amp***

? ***Use various start and stop times for the simulation.***

? ***Add and test for distortion.***

? ***Use demodulation components and equations.***

? ***Simulate the 1900 MHz amp with a GSM signal and Envelope.***

? ***Plot carrier and baseband data.***

? ***Operate on CE data in the frequency and time domain***

Agilent Technologies

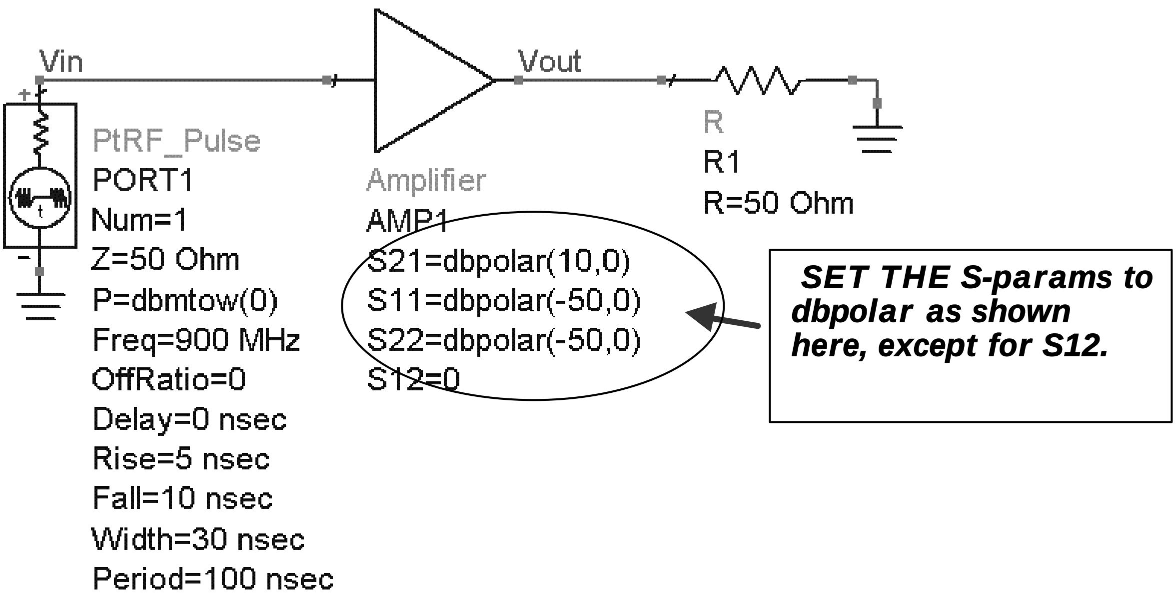
TABLE OF CONTENTS

PROCEDURE

1. ***Set up a PtRF source and behavioral amp.***

   a. ***Create a new schematic and name it: ckt_env_basic. You are still in the amp_1900 project. This amplifier circuit will be used to cover the basics of envelope simulation. Build the circuit shown here using the following steps:***

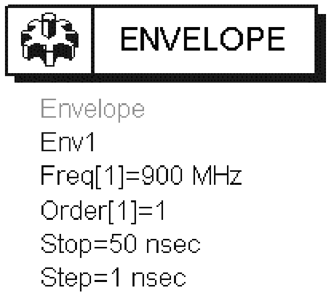
   b. ***Insert a behavioral amplifier (Amplifier) from the System-Amps & Mixers palette. Set the S-parameters as shown where S21 is 10dB of gain with 0 phase (dB and phase are separated by a comma). S11 and S22 are –50 (dB return loss), and 0 phase. Finally, S12 can remain set to 0 to indicate no reverse leakage. Be sure to use dbpolar for S21, S11, and S22 as shown here.***



   c. ***Insert a pulsed RF source (Sources-Modulated) and set it to 0 dBm at 900 MHz. Also, edit the following settings and be sure to check the display box for each setting: Off Ratio = 0, Delay= 0 ns, Rise time=5 ns, Fall time = 10 ns, Pulse Width = 30 ns, and the period is 100 ns.***

   d. ***Insert a 50 ohm resistor, node names, grounds, and wire as needed.***

2. Set up the Envelope Simulation controller.

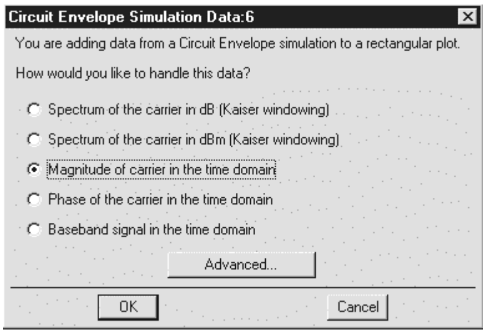


a. Set the frequency to 900 MHz and Order=1. Later on, you will add distortion and increase the order.

b. Set stop = 50 ns. This is enough time to see the entire pulse width, including the rise, fall, and delay.

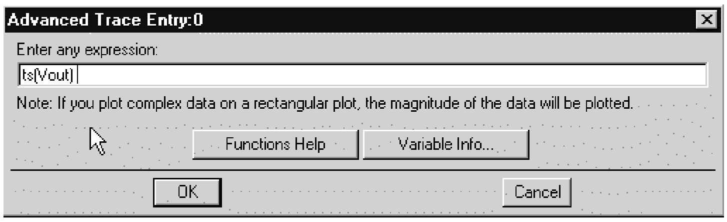
c. Set the step = 1 ns. This means the signal will be sampled every 1 ns resulting in 51 points of time sampled data.

3. Simulate and plot the time domain response.

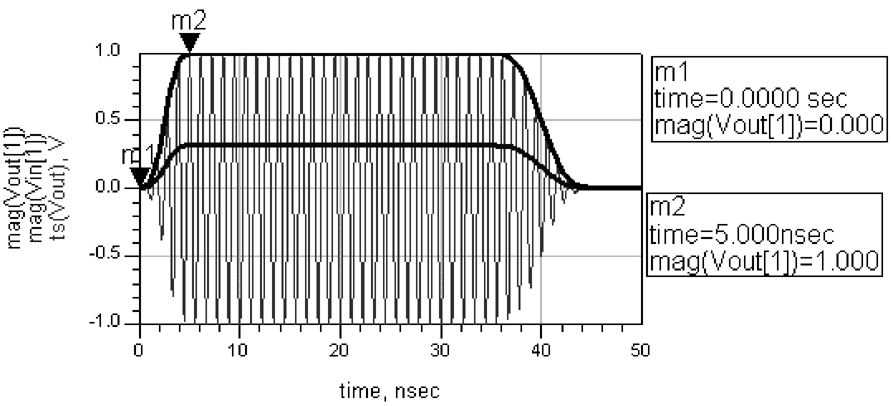


a. Simulate and watch the status window. You will see each time step calculated until the final result of 50 ns. After the data display opens, plot Vin and Vout in a rectangular plot as the Magnitude of the Carrier in the time domain.

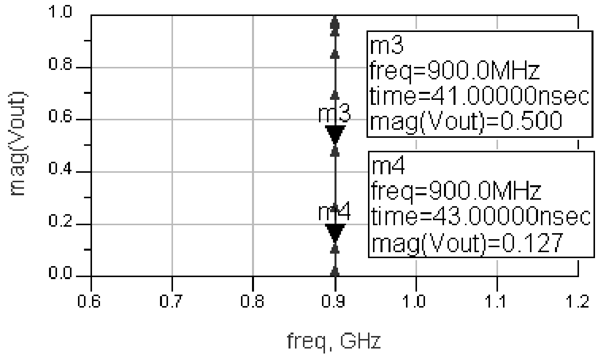
b. Also, add a third trace of Vout by selecting it and using the Advanced button to edit the expression as: ts (Vout) which gives the composite waveform. The index [1] in the other two mag traces gives you the magnitude of the 900 MHz carrier.

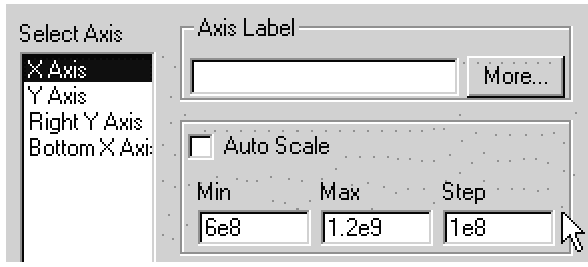


c. Put two markers on the plot to verify the rise time of 5 ns.



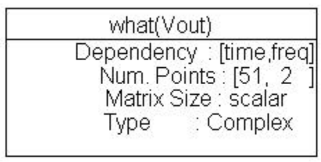
d. ***In a separate plot, insert mag of Vout again, and edit the trace to remove the indexing: mag (Vout). Also, edit the Plot Options, and turn off X-axis Auto Scale: set X-axis from 600 to 1200 MHz as shown here to center the trace. Notice that without the index value, you get the magnitude of the fundamental (900 MHz) in the frequency domain. The increasing arrows represent the increasing magnitude of the pulse carrier as it rises during the time (5 ns).***

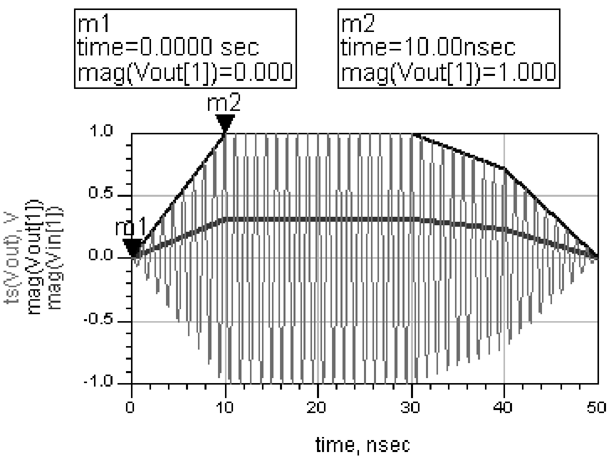




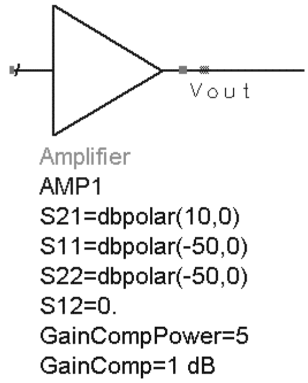
e. ***Next, insert a List. When the dialog box appears, use the Advanced button and edit the expression to be: what (Vout). Click OK and you will see what dependencies there are for Vout. The purpose of this is to again use the what function and to show that both time and frequency exist in the circuit envelope data. There are 51 time points of the two frequencies: 0 (dc) and 900 MHz. The Matrix Size refers to the 1x1 matrix (ADS calls it scalar) and the data is complex (mag and phase of the 900 MHz).***

f. ***Set the time step to 10 ns and simulate. Now, watch what happens to your plot when you under-sample the envelope. With the time step greater than the rise-time, you still get the carrier but not the correct envelope. On the plot, the X-axis has increased and the markers are on the first two time points: 0 and10 nsec.***





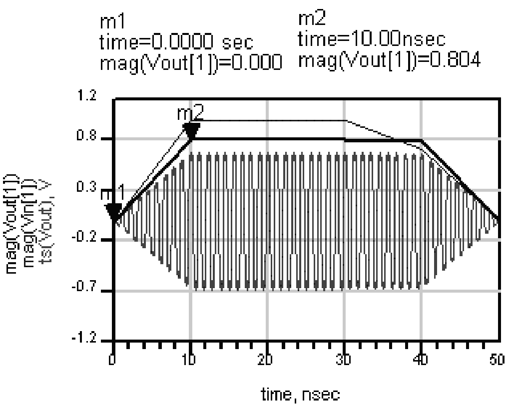
4. Add distortion to the behavioral amplifier.

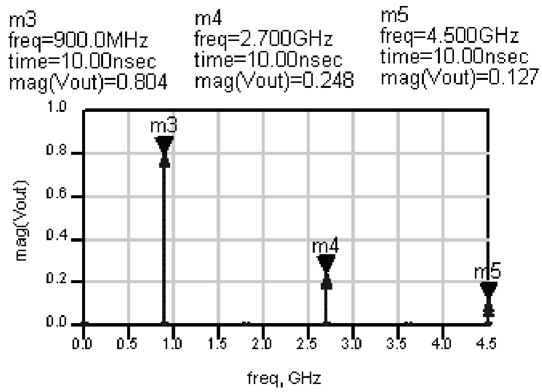


a. Edit the Amplifier by setting: Gain Compression Power = 5 (dBm is the default) and Gain Compression = 1 dB. These values are only used to show how the settings work. Be sure to display these settings.

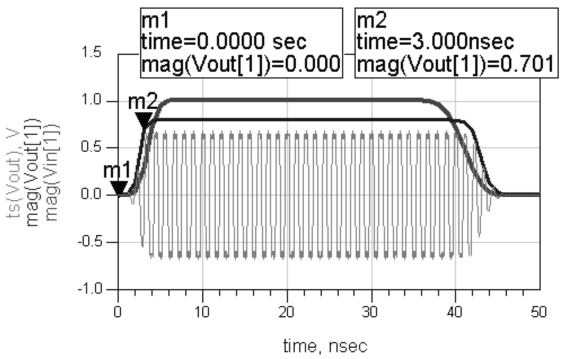
b. Set the CE controller Order = 5 and keep the time step at 10 ns. Also, set the source input power to 10 dBm: dbmtow (10).

c. Simulate and view the data. The time domain plot will adjust if autoscale is on. On the frequency domain plot, set the X-axis back to Auto Scale and place the markers as shown, where strong odd harmonics result from the amplifier distortion (summing out-of-phase). This results in the envelope amplitude being smaller than the magnitude of the Vin or Vout magnitude. Also, the envelope shape is not accurate because the sampling rate is too coarse.





d. Set the time step to 1 ns and Simulate again. After updating, the plot shows the correct envelope. But Vin and Vout still greater than the envelope magnitude, due to the compression. To prove this, insert a List of Vout and <u>Suppress Table Format</u>. Then scroll down to the 5 nanosecond data. Now, you can see that the third harmonic is 180 degrees out-of-phase, making the envelope smaller than the magnitude of the fundamental.



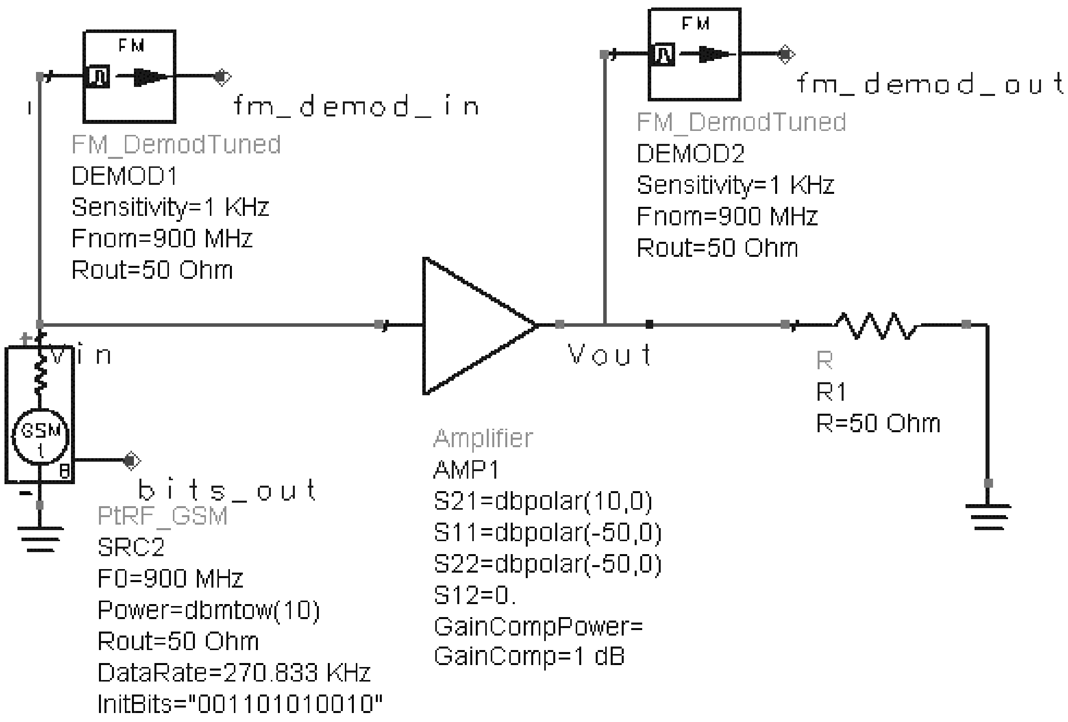
freq	Vout
time=5.000nsec	
0.0000 Hz	0.000 / 0.000
900.0MHz	0.803 / 2.624E-15
1.800GHz	0.000 / 0.000
2.700GHz	0.246 / -180.000
3.600GHz	0.000 / 0.000
4.500GHz	0.124 / -9.696E-15

5. Set up demodulators and a GSM source.

Note on GSM modulation: This is a phase modulation of the carrier (typically 900 MHz) where the phase variation represents 1 or 0.

a. From the Sources-Modulated palette, insert the GSM source and put a pin label (node name) at the B output as shown: bits_out. It looks like a non-connected pin but it is OK. Also, set the source FO=900 MHz and Power = dbmtow (10). Also, remove the compression: GainCompPower = (blank).

b. Go to the System-Mod/Demod palette and insert two demodulators: FM_DemodTuned as shown. Set the value of Fnom on the two demodulators as shown: 900 MHz. Also, insert label names at each output: fm_demod_in and fm_demod_out as shown. These will be used to look at the demodulated GSM

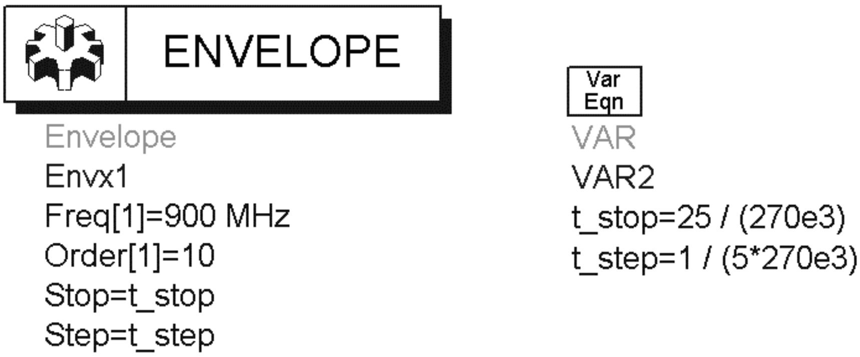


signal (baseband).

Note on Demodulators – You could use phase-demodulators but the FM demodulators are easier to use for this example. If you design demodulators, you could use this type of setup to test your circuits. In addition, refer to the Example directory for modulator/demodulator simulation examples.

6. Set up the Envelope Simulation with variables.

a. Insert a variable equation VAR and set up the stop and step times for approximate 270 kHz modulation BW as shown. The variable: t_stop is set to cover approximately 100 us. It is convenient to use the BW value as the denominator but not necessary. The sample rate t_step is 5 times the BW. Also, note that the default ADS Envelope time units (seconds) does not have to be specified.

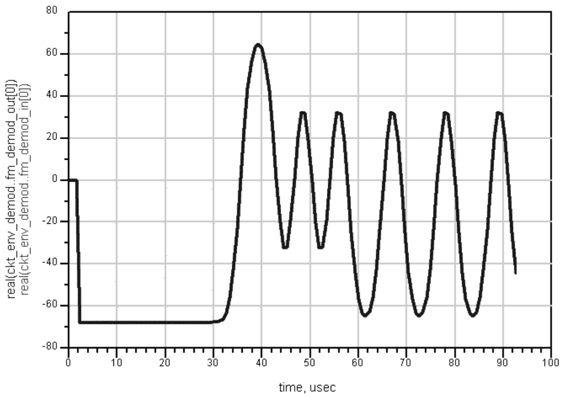


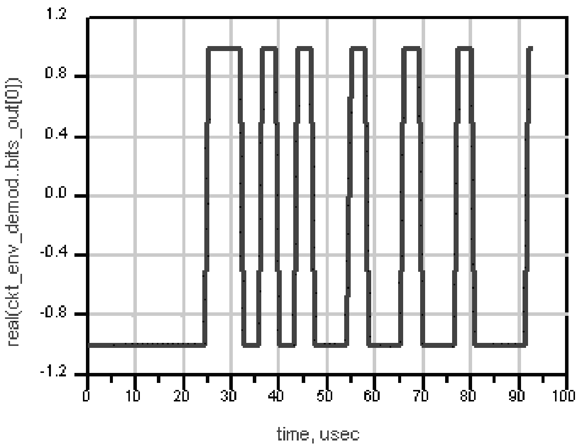
7. Simulate and plot the results demodulated results.

a. Simulate with the dataset name: ckt_env_demod.

b. Your previous plots are not set up to display this data so use a new dataset name to keep the data in separate plots but in the same data display window. So, plot the two FM nodes as Baseband signal in the time domain. These traces will be the real part indexed to [0]. The demodulator only outputs a signal at baseband (similar to the dc component). Notice they are the same because there is no distortion at this time.

c. In a separate graph, plot the real part of bits_out. Except for some delay,





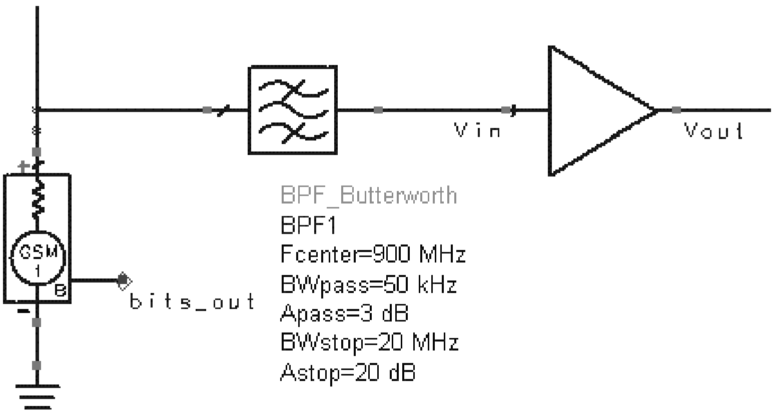
you should see the 001101010010 pattern.

8. Use a filter to simulate phase distortion.

a. On the amplifier, set the GainCompPower on the amplifier to 5 (this is 5 dBm at the amp output) and set the GainComp to 1 dB.

b. Be sure the GSM source power is set to 10 dBm.

c. Insert a Butterworth filter (Filters-Bandpass) between the amplifier and the source and set it as shown. This will create some distortion as only the narrower bandwidth passes to the amplifier and the full signal goes to the first demodulator.

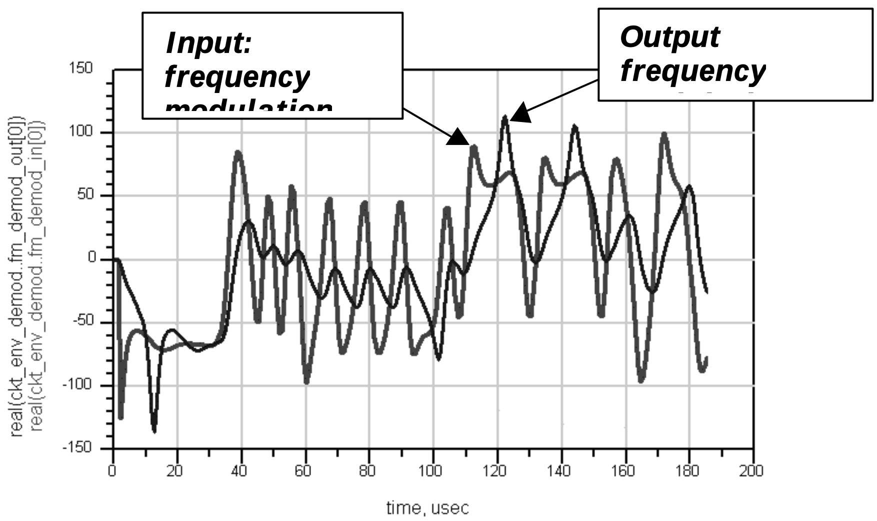


d. Change the t_step to 10 times the 270KHz BW as: t_step = 1 / (10*270e3)

e. Change the t_stop numerator to 50 (200 us): t_stop = 50 / (270e3)

9. Simulate and plot input and output modulation.

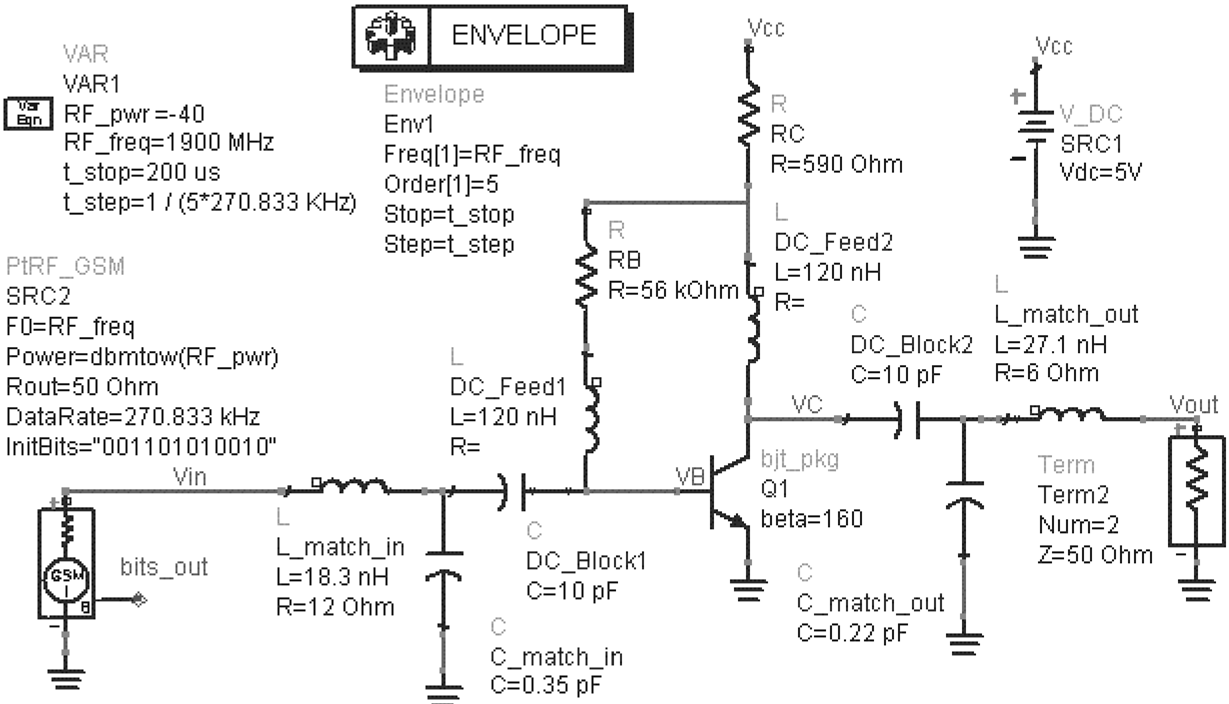
Your plot should show the distortion and delay from the input to the output similar to the one shown here.



10. Simulate amp_1900 with a GSM source

a. Open the previous schematic design (hb_2Tone) and save it with a new name: ckt_env_gsm.

b. Delete any previous simulation controllers, variables, etc. Then modify the schematic by inserting: 1) an Envelope controller, 2) a PtRF_GSM source, 3) VarEqn set as shown here. The simulation components and variables are similar to the last Envelope simulation setup so you could use the Edit > Copy/Paste commands in schematic. Also, be sure to label the



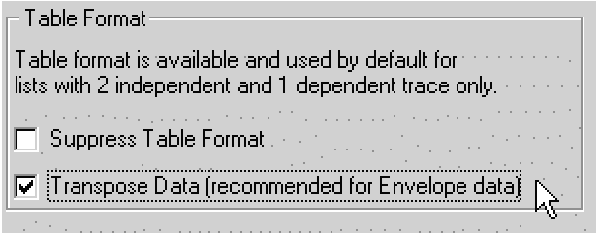
bits_out node on the GSM source.

NOTE on CE setup values: In this simulation, t_stop of 200 us (twice as long as the previous simulation) will give you better spectral resolution. The t_step is set using an exact multiple for the BW (270.833 KHz). Generally, this is not necessary but it can be done if you want more exact frequency calculation for phase. Also, the default start time for CE is always zero seconds and it is not recommended to change it. However, if you want to see this, use the Display tab setting and turn on Start.

c. Check your setup and then Simulate and watch the status window.

11. Plot the GSM data and spectrum.

a. In the data display, insert a list of Vout and use the Plot Options to set the format for Engineering and select the Transpose Data feature as shown here. Now, you can see that CE calculates each tone specified at each time step. Scroll to the end and you will see that the last point is at the end of the t_stop time.

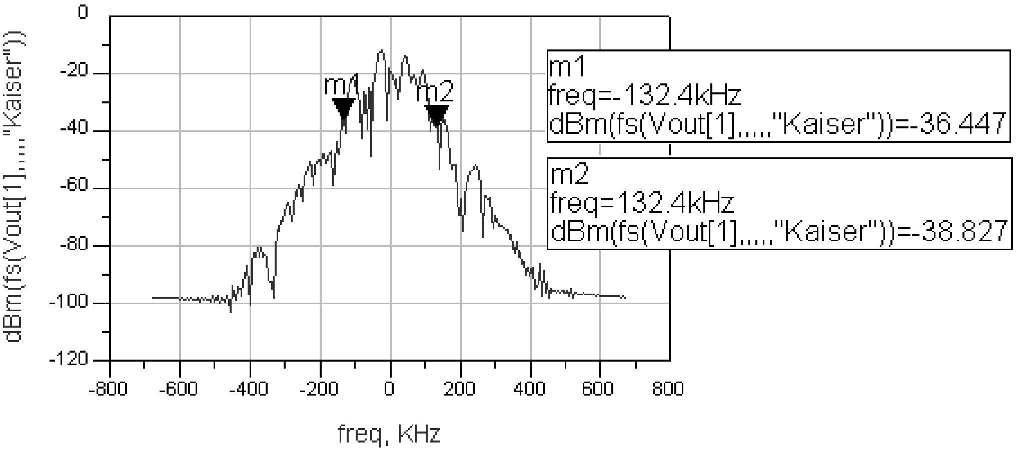


time	Vout					
	freq=0.0000 Hz	freq=1.900GHz	freq=3.800GHz	freq=5.700GHz	freq=7.600GHz	freq=9.500GHz
197.2usec	101.1p / 180.0 V	180.4m / 165.9 V	1.019m / -170.7 V	19.59u / -133.3 V	862.7n / -142.1 V	52.08n / -156.2 V
197.9usec	5.660p / 0.0000 V	180.4m / -176.2 V	1.019m / -134.9 V	19.59u / -79.64 V	862.5n / -70.53 V	51.96n / -66.81 V
198.6usec	25.44p / 0.0000 V	180.4m / -158.2 V	1.019m / -98.96 V	19.59u / -25.72 V	862.5n / 1.348 V	51.94n / 23.05 V
199.4usec	41.18p / 180.0 V	180.4m / -140.3 V	1.019m / -63.02 V	19.59u / 28.19 V	862.5n / 73.24 V	51.94n / 112.9 V
200.1usec	46.14p / 0.0000 V	180.4m / -122.4 V	1.019m / -27.24 V	19.59u / 81.85 V	862.5n / 144.8 V	51.97n / -157.6 V

Scroll

b. Plot the Vout data as: <u>Spectrum of the carrier in dBm with a Kaiser window</u>. Then insert two markers across the GSM bandwidth (about 270 kHz) to measure the BW. This is the output spectrum around the fundamental frequency (0 Hz on the plot). The window helps ensure that the first and last time data points equal zero. This improves the dynamic range of the computed spectrum. Also, with windowing, the noise floor is

NOTE on trace: Set trace options to Linear.

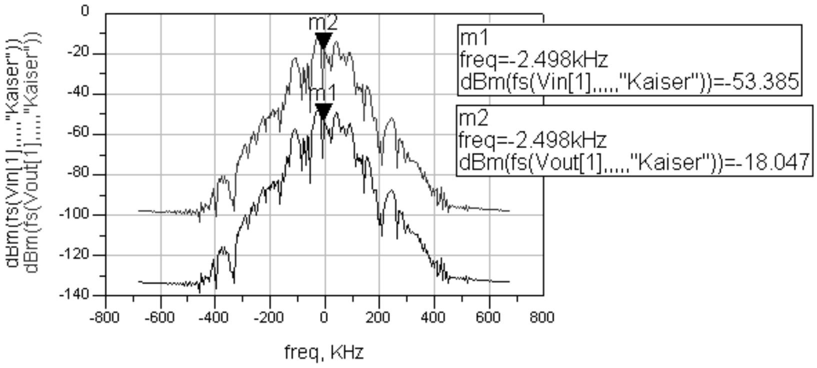


lowered.

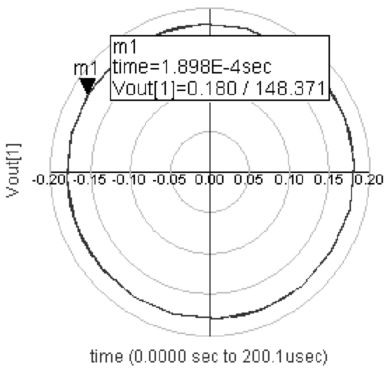
Note on CE for mixers - The Kaiser window is used by default for spectral data using the dialog. It assumes that the carrier is index value [1].

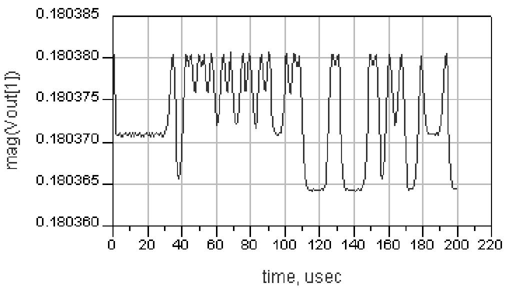
However, for a mixer, you may need to edit the trace and replace [1] with the correct index value from the Mix table

c. ***On the Vout plot, insert Vin (same data format type) and use markers to verify that the gain is very high (about 35 dB) which corresponds to previous simulations – this is true because the model is ideal.***



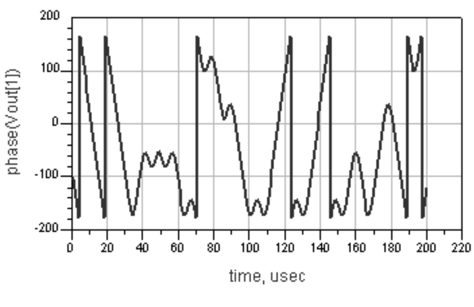
d. ***Insert two more plots: a polar plot of Vout[1] at all time points and a rectangular plot of Vout magnitude in the time domain as shown here. As you can see, the magnitude on both plots shows little variation in amplitude. For GSM, this means that the amplifier is adding little or no distortion to the baseband***

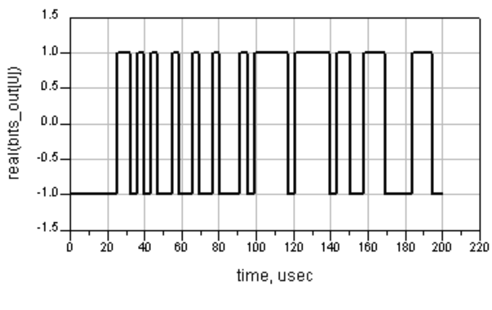




because GSM is a phase modulation.

e. ***Insert another two plots: a plot of Vout phase to see the phase variation during 200 us. Notice the phase plot Y-axis is +/- 180 from zero (similar to network analyzer). Also, insert a plot of the bits_out data. These are the raw bits from the source. In the next step, you will operate on this data to see the relationship between them.***

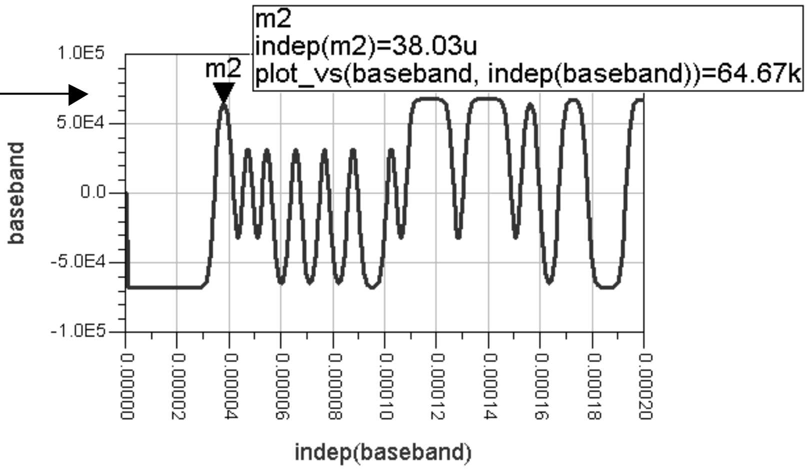




f. Write an equation to demodulate the data. In the equation shown, the unwrap function will remove the +/- 180 transition format from the absolute phase and the diff function will differentiate unwrapped slope. Then, dividing by 360 will give the value in Hz – this is essentially the demodulated output.

Eqn baseband=diff (unwrap (phase (Vout [1])) / 360)

After about 20 us of delay in the source, the marker shows about 65 kHz deviation from the carrier at 38 micro seconds = X axis .

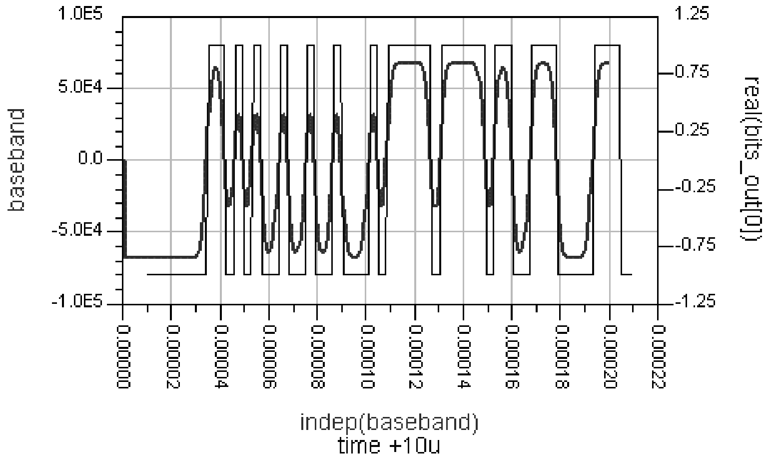


Plot the equation as shown here.

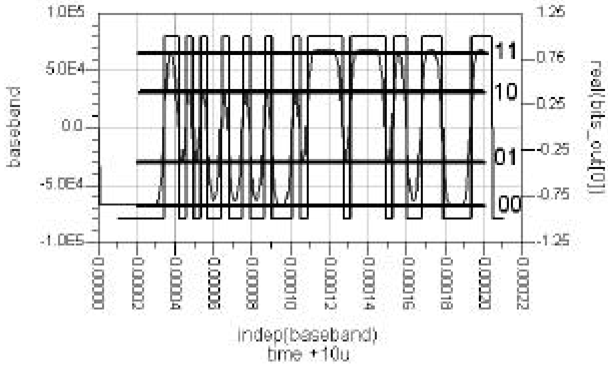
g. Onto the baseband plot, add a trace of the bits_out in the time domain. It will be near zero until you edit the trace. Then go to the Plot Axes tab and select Right Y-axis for this trace.

h. Next, in Plot Options, remove autoscale and reset the right Y-axis from -1.25 to 1.25. Finally, shift the time + 10us as shown by adding it directly to the axis label using the cursor type in: +10u and enter. The shift is the delay through the amplifier. Now, you have a comparison of input to output baseband integrity.

<u>Compare GSM results</u>: Unwrapped and differentiated phase data results are the same as using FM_DemodTuned demodulator



NOTE: You can draw the possible states of any baseband signal (four lines) and label possible states (00, 01, 10, 11) with text as shown, directly on the plot.



i. Save all your work now and also if you do the optional exercise.

12. OPTIONAL – Channel power calculations

a. Create a new page in the data, Page > New Page, and name it as shown here.

b. Write two equations to calculate the power in the spectrum using the ADS channel_power function. The first equation, limits, defines the modulation bandwidth. The second equation, channel_pwr, uses the ADS channel_power_vr function where vr means that it uses voltage instead of current in the calculation. Vout[1] is the 1900MHz tone in the equation. Also, 50 is the system impedance, the “Kaiser” argument is a window that lowers the noise floor, and +30 converts the final value into dBm (where 0 dBm = 0.001W).

Eqn limits = {-(270KHz / 2), (270KHz /2)}

Eqn channel_pwr=10*log(channel_power_vr(Vout[1],50,limits,"Kaiser"))+30

c. List the channel_pwr equation and you now have the power in the GSM signal bandwidth. This calculation can be applied to many other modulation schemes using Circuit Envelope.

channel_pwr
-4.886

Channel power for amp_1900 for a GSM sign...

EXTRA EXERCISES:

1. ***Sweep RF power in the schematic and watch the change in the output.***

2. ***Use the FM demodulator on the output and re-run the simulations. Compare the bits in and the bits out for the amp_1900.***

3. ***Go to the example file: examples\Tutorial\ModSources_prj\Pi4DQPSK and copy the source and data display into your directory and try that source on the amplifier, using the data display as a reference to guide you.***

THIS PAGE IS LEFT 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

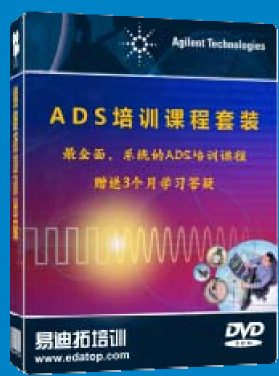
射频工程师养成培训课程套装

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…

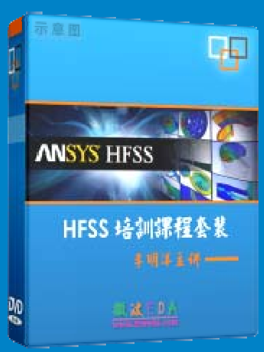
课程网址：http://www.edatop.com/peixun/rfe/110.html

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

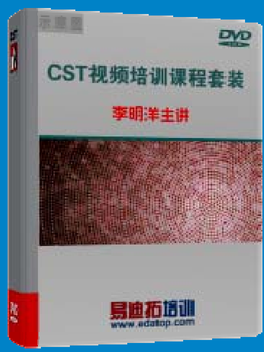


HFSS 学习培训课程套装

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