ADS Fundamentals - 2009

LAB 8: Circuit Envelope Simulation

Overview - This chapter covers the basics of Circuit Envelope simulation to simulate 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
- Experiment with simulation parameters
- Test for distortion
- Use demodulation components and equations
- Simulate the 1900 MHz amp with a GSM signal
- Plot carrier and baseband data
- Operate on datasets in the frequency and time domain

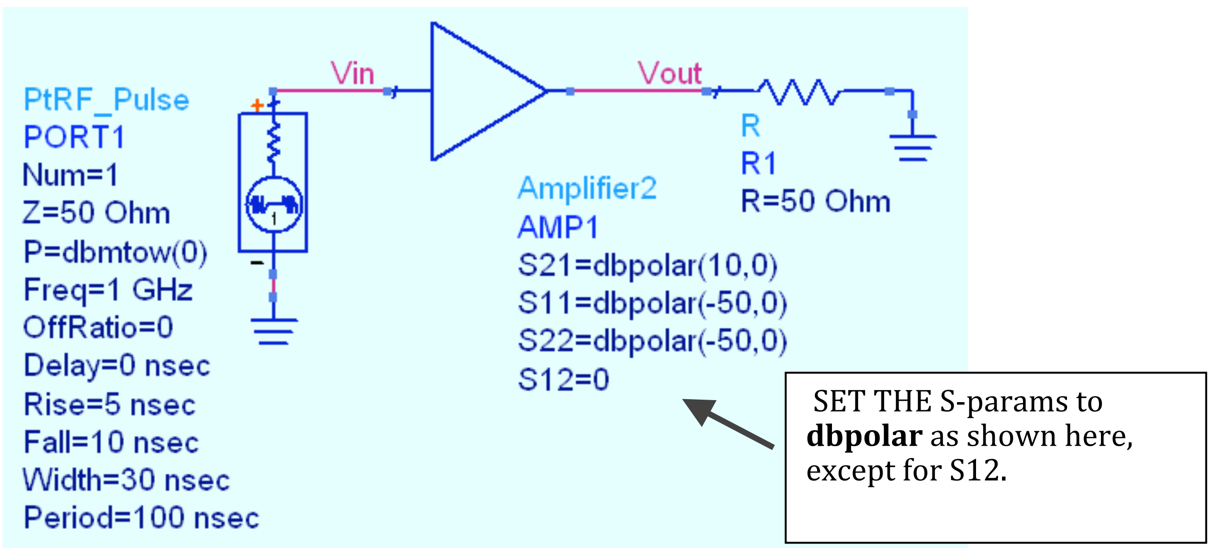
TABLE OF CONTENTS

PROCEDURE

1. Set up a PtRF source and behavioral amp.

a. In the amp_1900 project, create a new schematic and name it: **ckt_env_basic**. Build the circuit shown here using the following steps:

b. Insert a behavioral **Amplifier** from the **System-Amps & Mixers** palette: Amplifier2. Set the S-parameters as shown where **S21 = 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 **PtRF_Pulse source** (Sources-Modulated) and set the power as **P =dbmtow(0)** and **Freq = 1 GHz**. 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 = 100 ns**.

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

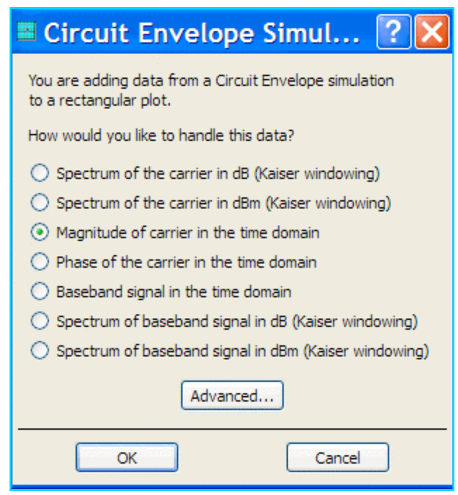
2. Set up the Envelope Simulation controller.

a. Insert the controller and set the calculation frequency to**1** frequency to**1 GHz** and **Order=1**.   Later on, you will add will add distortion and increase the order.

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

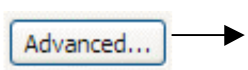
c. Set the **step = 1 nsec**. This means the signal will be sampled every 1 ns resulting in 51 points of time sampled data.

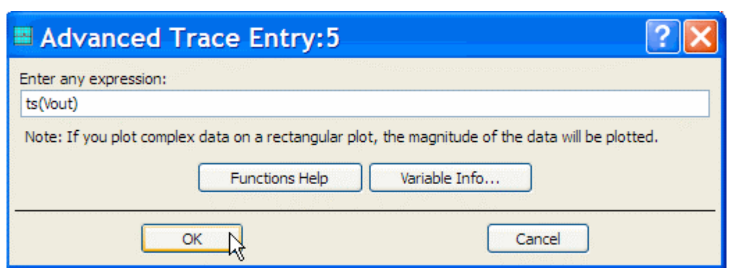
3. Simulate and plot the time domain response.



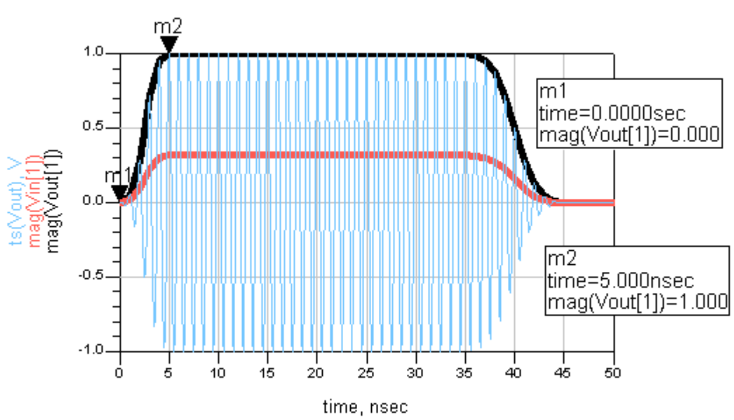
a. Simulate and view the status window. You will see see each time step calculated until the final result of 50 result of 50 ns. After the data display opens, plot **Vin** plot **Vin** and **Vout** in a rectangular plot as the Magnitude of the Carrier in the time domain.

b. Also, add a third trace using the **Advanced** button and button and type in the expression: **ts (Vout)** which which gives the composite waveform. The index [1] in [1] in the other two traces gives you the magnitude of magnitude of the 1GHz carrier.

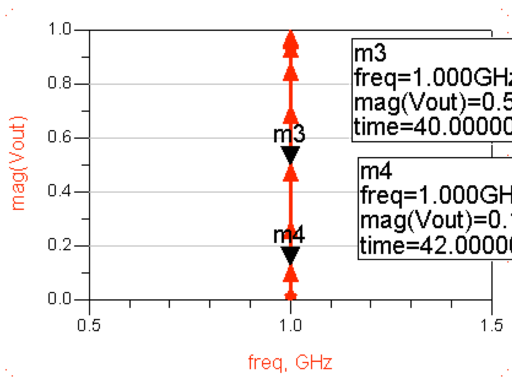


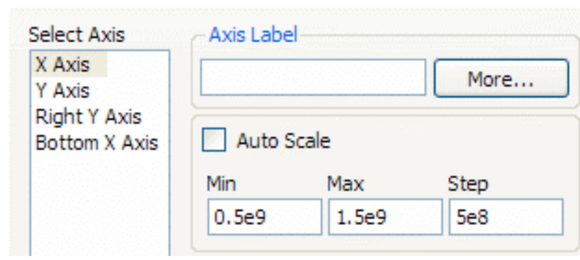


c. Put two markers on the on the plot to verify the verify the rise time of 5 time of 5 ns.



d. In a separate plot, insert the magnitude of **Vout** (time domain) again. Now, edit the plot, select the trace, and use **Trace Options > Trace Expressions** to remove the index [1] so that the expression is: **mag (Vout)** and click OK. Now use the **Plot Options** tab and turn **off** the X-axis Auto Scale: set **X-axis** from **0.5 GHz** to **1.5 GHz** as shown here to center the trace. By removing the index value, you get the magnitude of the fundamental (1 GHz) in the frequency domain. The increasing arrows represent the increasing magnitude of the pulse carrier as it rises during the time (5 ns).

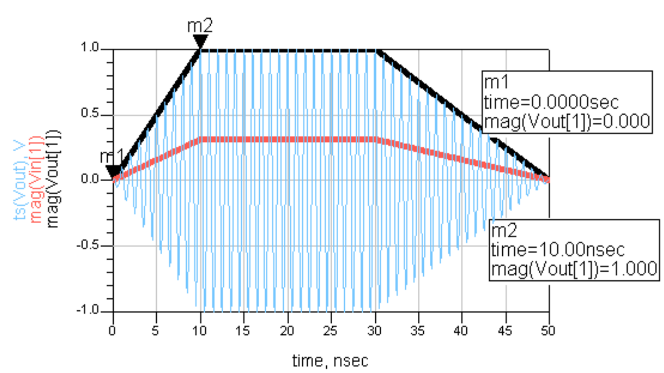




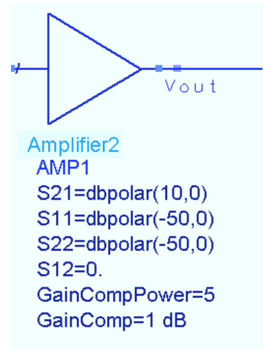
e. Next, insert a **List**. When the dialog box appears, use the **Advanced** button and type in the expression: **what (Vout)**. Click **OK** and you will see what dependencies there are for Vout. The purpose of this is to show that both time and frequency exist in the circuit envelope data. There are 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). Also, the mix table contains all data. Try inserting the mix table and surpressing the table format to see this!

what(Vout)
Dependency : [time,freq]
Num. Points : [51, 2]
Matrix Size : scalar
Type : Complex

f. Go back and 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 and 10 nsec.



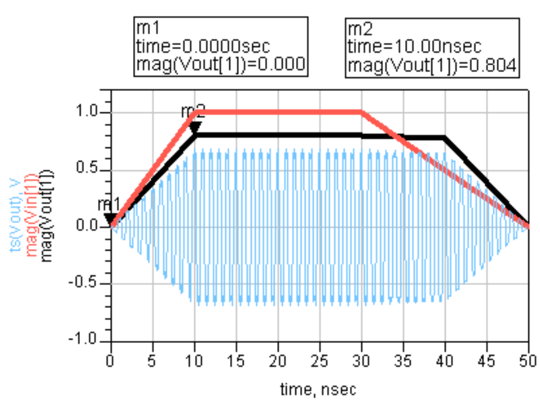
4. Add distortion to the behavioral amplifier.

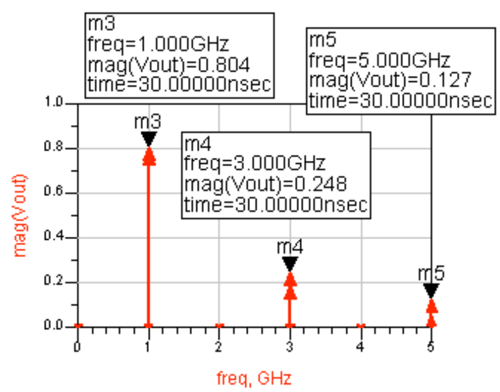


a. Edit the Amplifier by setting: Gain Compression Power = **5** (dBm is the default) and Gain Compression = **1 dB.** These values are only used to show how the settings work. Be sure to **display** these settings.

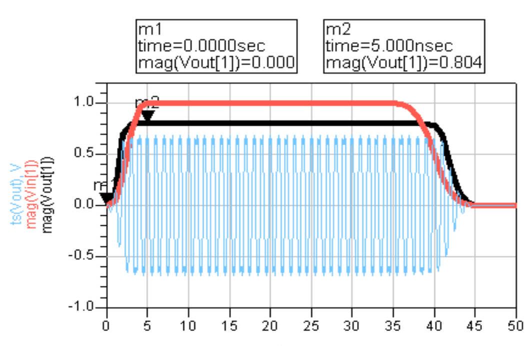
b. Set the CE controller **Order = 5** and keep the time step at 10 ns. Also, set the source input power to 10 dBm: dbmtow (**10**).

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 are still greater than the envelope magnitude, due to the compression. To prove this, insert a **List** of **Vout** and Suppress Table Format. Then scroll down to the 5 nanosecond data. Now, you can see that the large third harmonic is 180 degrees out-of-phase, making the envelope smaller than the magnitude of the fundamental.



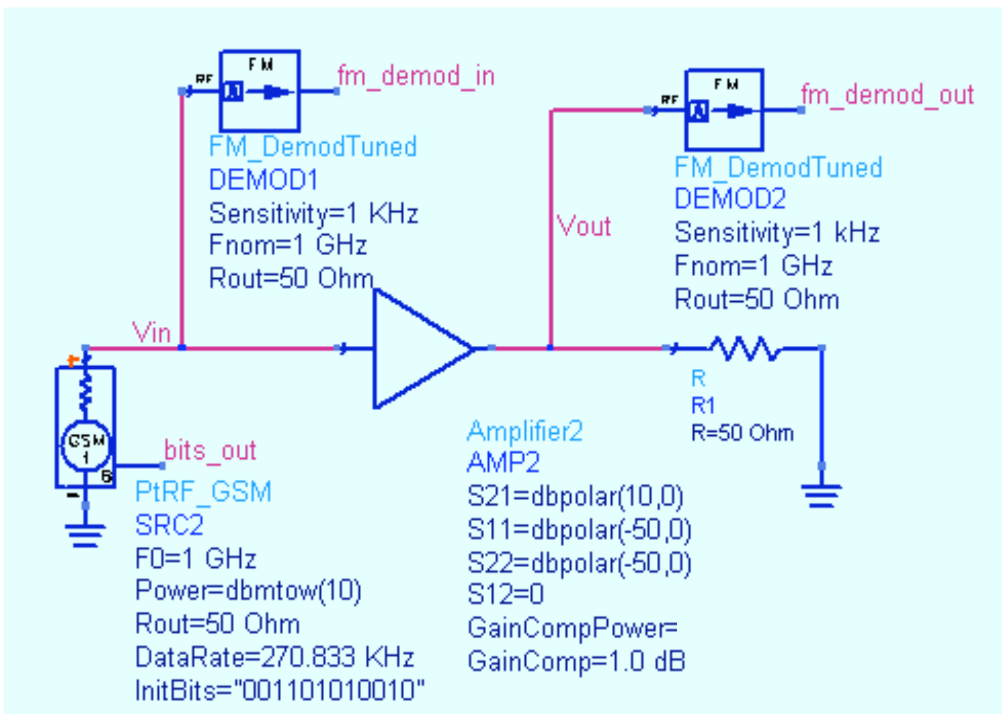
freq	Vout
time=5.000 nsec	
0.0000 Hz	0.000 / 0.000
1.000 GHz	0.804 / 0.000
2.000 GHz	0.000 / 0.000
3.000 GHz	0.248 / 180.000
4.000 GHz	0.000 / 0.000
5.000 GHz	0.127 / 0.000

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=1 GHz** and **Power = dbmtow (10).** Also, remove the amplifier compression (previously set to 5) so that: **GainCompPower = (blank)** as shown.

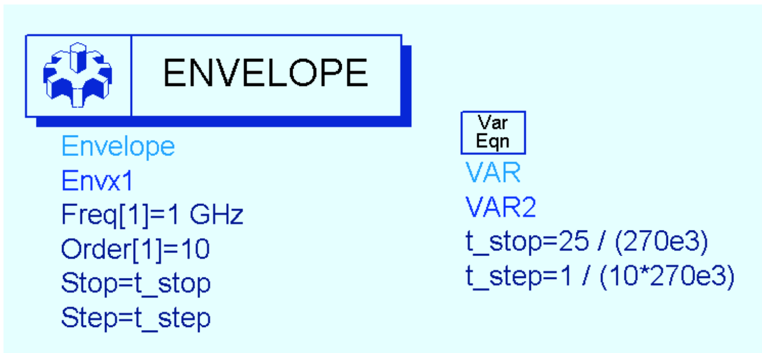
b. Go to the **System-Mod/Demod** palette and insert two demodulators: **FM_DemodTuned** as shown. Set the value of Fnom on the two demodulators as shown:**1 GHz**. Also, insert label names at each output: **fm_demod_in** and **fm_demod_out** as shown. These will be used to look at the demodulated GSM 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.

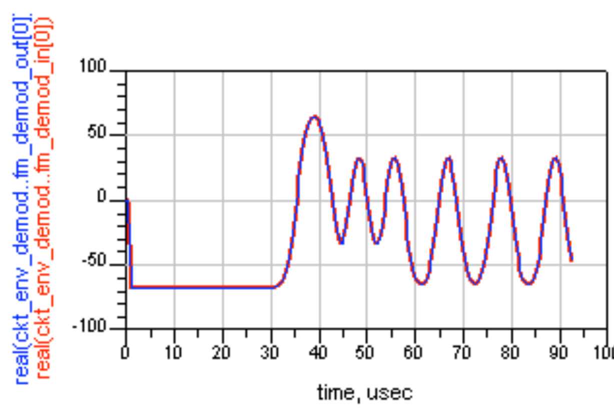
a. Insert a variable equation **VAR** and set up the stop and step times for approximate 270 kHz modulation BW as shown. The variable: t_stop is set to cover approximately 100 us. It is convenient to use the BW value as the denominator but not necessary. The sample rate t_step is 10 times the BW. Also, note that the default ADS Envelope time units (seconds) does not have to be specified.



7. Simulate and plot the demodulated results as described below

a. Simulate with the dataset name: **ckt_env_demod**.

b. Open a data display window. Plot the two FM nodes as the **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.



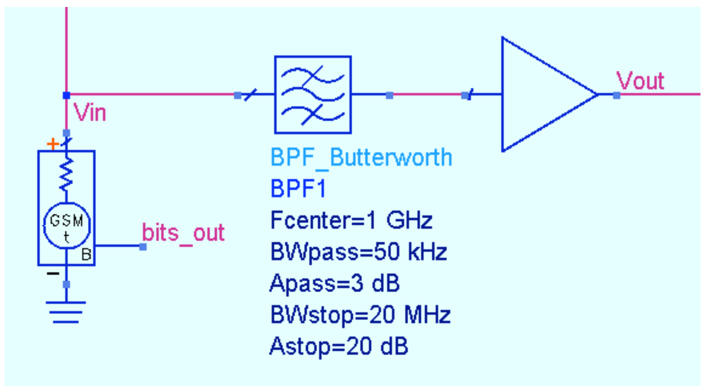
c. In a separate graph, plot the real part of **bits_out**. Except for some delay, you should see the 001101010010 pattern.

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** = **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.

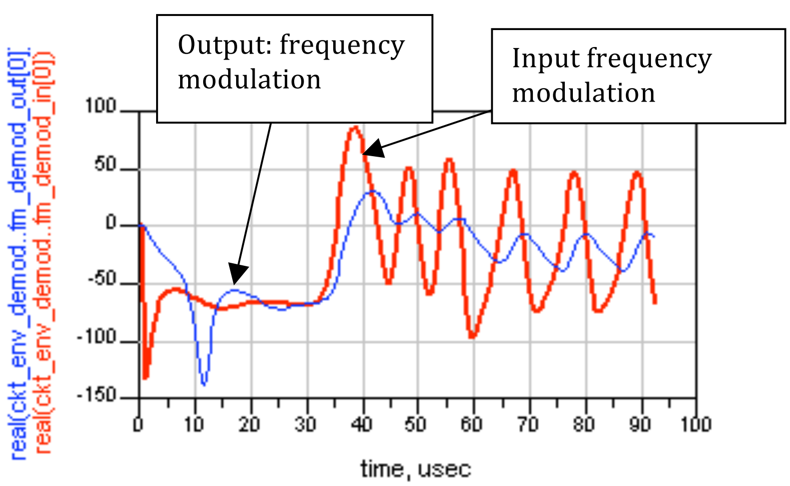


d. Be sure that: **t_step = 1 / (10*270e3)**.

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

9. Simulate and plot input and output modulation.

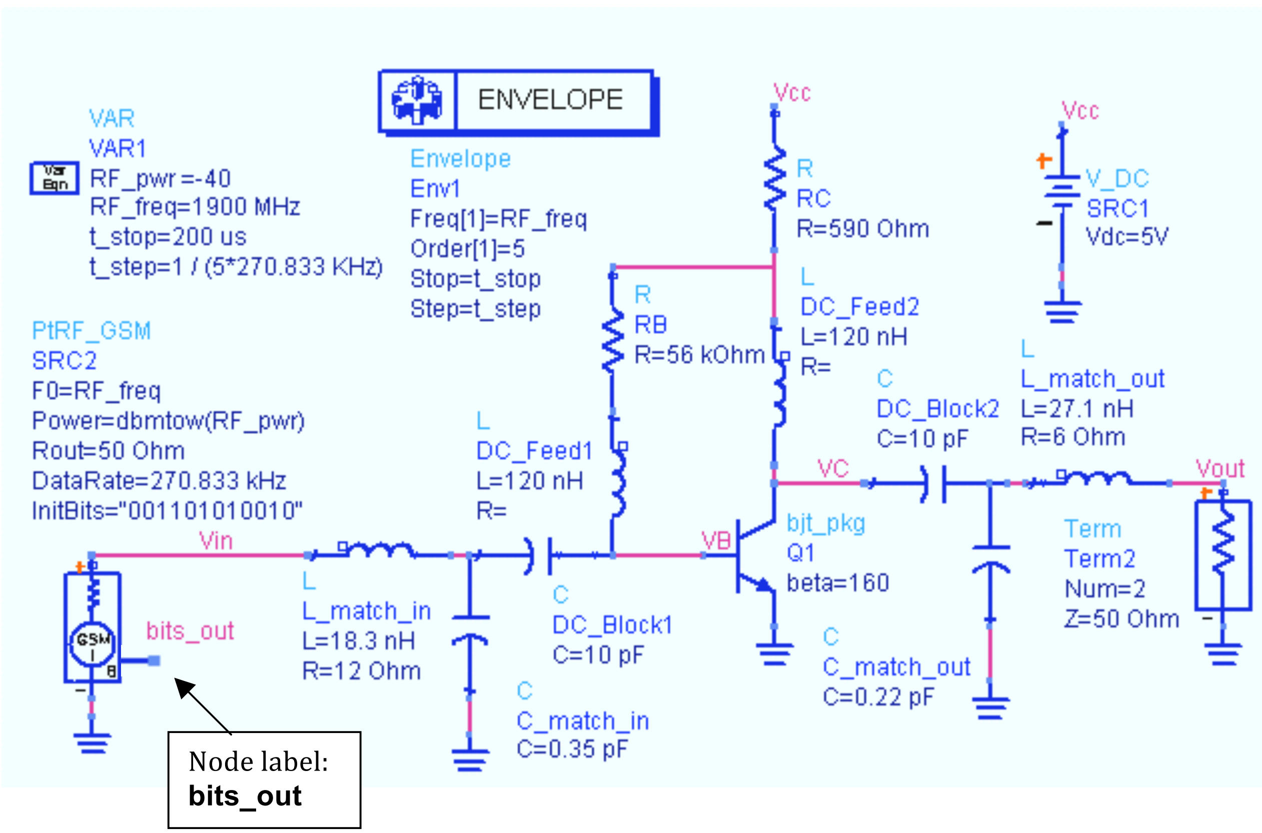
Your plot should show the distortion and delay from the input to the output similar to the one shown here.



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) and set up the **VAR** as shown here. The simulation components and variables are similar to the last Envelope setup. Therefore, you can use the Edit > Copy/Paste commands in schematic. Also, be sure to label the **bits_out** node on the GSM source.

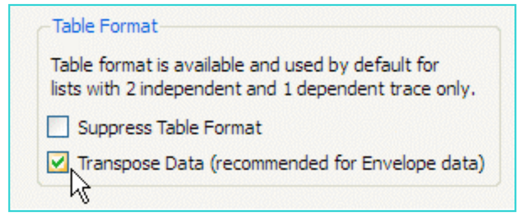


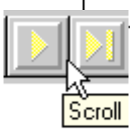
NOTE on CE setup values: In this simulation, **t_stop** of 200 us (twice as long as the previous simulation) will give you better spectral resolution. The **t_step** is set using an exact multiple for the BW (270.833 KHz). Generally, this is not necessary but it can be done if you want a more exact frequency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

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 and use the **Plot Options** to set the format for format for **Engineering** and select the **Transpose Data** feature as shown here. Now, here. Now, you can see that CE calculates calculates each tone (freq and order) at each each time step. Scroll to the end and you will you will see that the last point is at the end of the t_stop time.

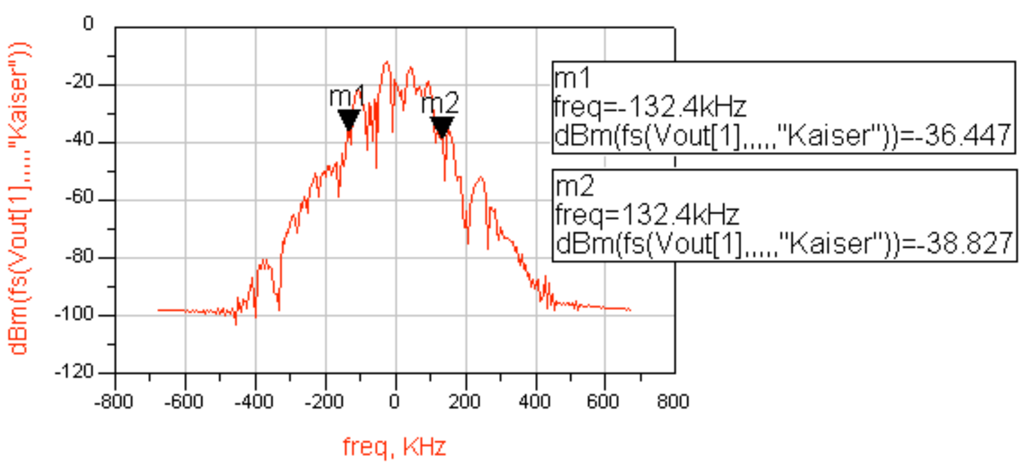




time	Vout					
	freq=0.0000 Hz	freq=1.900GHz	freq=3.800GHz	freq=5.700GHz	freq=7.600GHz	freq=9.500GHz
197.2usec	101.1p / 180.0 V	180.4m / 165.9 V	1.019m / -170.7 V	19.59u / -133.3 V	862.7n / -142.1 V	52.08n / -156.2 V
197.9usec	5.660p / 0.0000 V	180.4m / -176.2 V	1.019m / -134.9 V	19.59u / -79.64 V	862.5n / -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

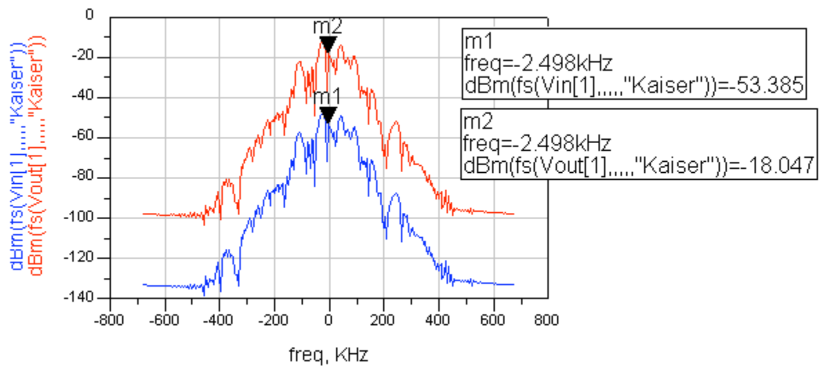
b. Plot the **Vout** data as**: Spectrum of the carrier in dBm with a Kaiser window**. Then insert two markers across the GSM bandwidth (about 270 kHz) to measure the BW. This is the output spectrum around the fundamental frequency (0 Hz on the plot). The Kaiser window helps ensure that the first and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

NOTE: Use **Trace Options > Trace Type** and select trace type: **Linear**.

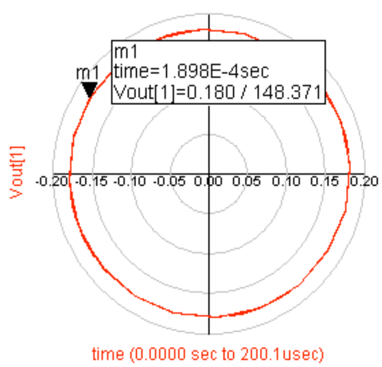


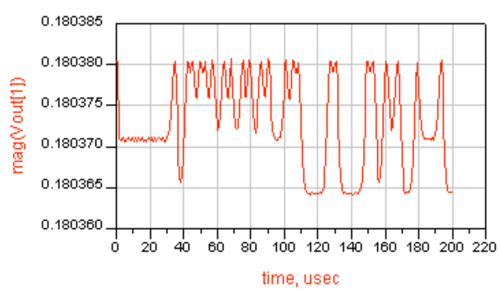
Note on CE for mixers - The Kaiser window is used by default for spectral data using the dialog. It assumes that the carrier is index value [1]. However, for a mixer, you may need to edit the trace and replace [1] with the correct index value from the Mix table for your IF or RF frequency.

c. On the Vout plot, insert **Vin** (same data format type) and use markers to verify that the gain is about 35 dB. This corresponds to previous simulations for amp_1900 using the ideal Gummel-Poon model.

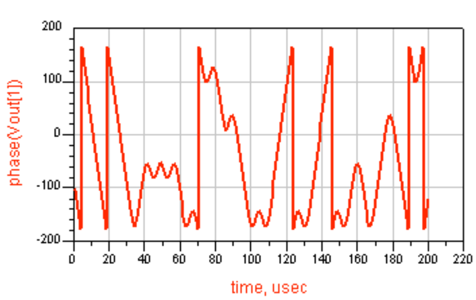


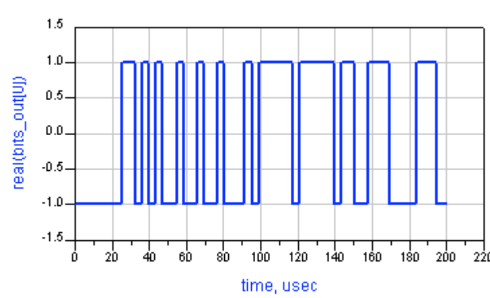
d. Insert two more plots: a **polar plot of Vout[1]** at all time points and a **rectangular plot of Vout magnitude in the time domain** as shown here. As you can see, the magnitude on both plots shows little variation in amplitude. For GSM, this means that the amplifier is adding little or no distortion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





e. Insert two more plots: the **phase of Vout** to see the phase variation during 200 us. Notice the phase plot Y-axis is +/- 180 from zero (similar to a 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.

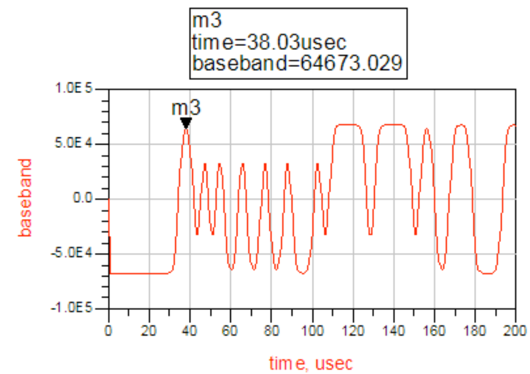




f. An equation can demodulate the data. In the **baseband** equation shown here, the **unwrap** function removes the +/- 180 degree transition format from the absolute phase. The **diff** function will differentiate the unwrapped slope. Dividing by 360 will give the value in Hz. This is essentially the demodulated output. <u>Write and plot the equation shown</u>:

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

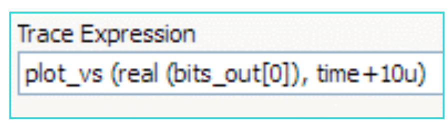
After about 20 us of delay in the source, the marker shows about 65 kHz deviation from the carrier at 38 micro seconds = X axis .



g. Edit the baseband equation plot and add a trace of **bits_out** (baseband) in the time domain. It will be near zero until you **edit the trace**. Go to the **Plot Axes** tab and select **Right Y-axis** for this trace.

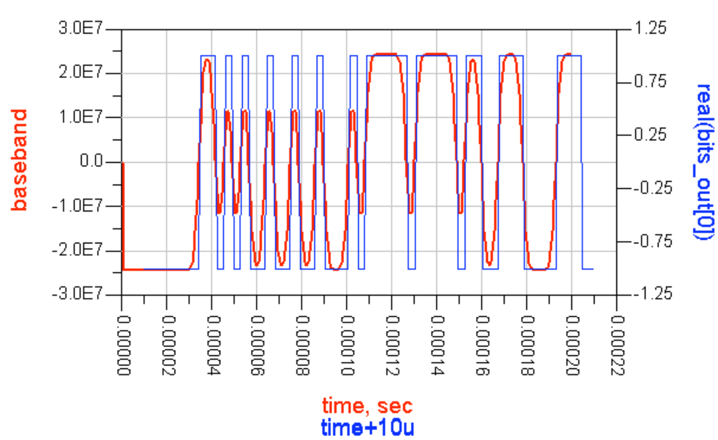
h. Next, in **Plot Options**, remove autoscale and reset the right Y-axis from **-1.25** to **1.25**.

i. To remove the effects of the source delay, and and shift the time +10usec, edit the bits_out trace, bits_out trace, using Trace Expression as shown shown here:
plot_vs(real(bits_out[0]),time+10u)

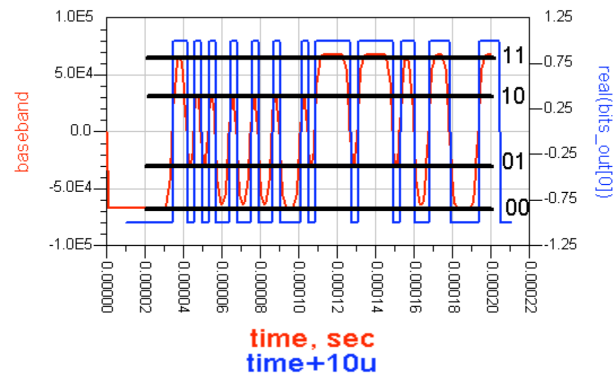


Now, you have a comparison of input to output baseband integrity.

<u>GSM results</u>: Unwrapped and differentiated phase data results are the same as using FM_DemodTuned demodulator component on the behavioral amp.

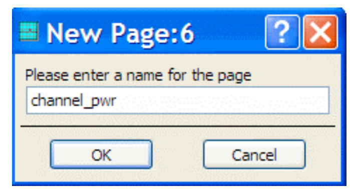


NOTE on data: You can draw lines to show the possible states of the baseband signal and label the states (00, 01, 10, 11) directly on the plot as shown here.



j. **Save** all your work. At this point in the course, you have covered all the fundamental processes in ADS. The final lab exercise you will put all the circuits together for final testing.

12. OPTIONAL – Channel power calculations



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

b. Write two equations to calculate the power in the spectrum spectrum using the ADS channel_power function. The first equation, **<u>limits</u>**, defines the modulation bandwidth. The second equation, **<u>channel_pwr</u>**, 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 will see the power in the GSM signal bandwidth. This calculation can be applied to other modulation schemes using Circuit Envelope simulation.

Channel power for amp_1900 for a GSM signal:

channel_pwr
-4.886

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.