Harmonic Balance Simulation on ADS

General Description of Harmonic Balance in Agilent ADS [1]

Harmonic balance is a frequency-domain analysis technique for simulating nonlinear circuits and systems. It is well-suited for simulating analog RF and microwave circuits, since these are most naturally handled in the frequency domain. Circuits that are best analyzed using HB under large signal conditions are:

- ❖ power amplifiers
- ❖ frequency multipliers
- ❖ mixers
- ❖ oscillators
- ❖ modulators

Harmonic Balance Simulation calculates the magnitude and phase of voltages or currents in a potentially nonlinear circuit. Use this technique to:

- ❖ Compute quantities such as P1dB, third-order intercept (TOI) points, total harmonic distortion (THD), and intermodulation distortion components
- ❖ Perform power amplifier load-pull contour analyses
- ❖ Perform nonlinear noise analysis
- ❖ Simulate oscillator harmonics, phase noise, and amplitude limits

In contrast, S-parameter or AC simulation modes do not provide any information on nonlinearities of circuits. Transient analysis, in the case where there are harmonics and or closely-spaced frequencies, is very time and memory consuming since the minimum time step must be compatible with the highest frequency present while the simulation must be run for long enough to observe one full period of the lowest frequency present.

Harmonic balance simulation makes possible the simulation of circuits with multiple input frequencies. This includes intermodulation frequencies, harmonics, and frequency conversion between harmonics. Not only can the circuit itself produce harmonics, but each signal source (stimulus) can also produce harmonics or small-signal sidebands. The stimulus can consist of up to twelve nonharmonically related sources. The total number of frequencies in the system is limited only by such practical considerations as memory, swap space, and simulation speed.

The Simulation Process [1] (FYI – skip to next section if you want to get started now)

The harmonic balance method is iterative. It is based on the assumption that for a given sinusoidal excitation there exists a steady-state solution that can be approximated to satisfactory accuracy by means of a finite Fourier series. Consequently, the circuit node

---

[1] From Agilent ADS Circuit Simulation Manual, Chap. 7, Harmonic Balance.

voltages take on a set of amplitudes and phases for all frequency components. The currents flowing from nodes into linear elements, including all distributed elements, are calculated by means of a straightforward frequency-domain linear analysis. Currents from nodes into nonlinear elements are calculated in the time-domain. Generalized Fourier analysis is used to transform from the time-domain to the frequency-domain.

A frequency-domain representation of all currents flowing away from all nodes is available. According to Kirchoff's Current Law (KCL), these currents should sum to zero at all nodes. The probability of obtaining this result on the first iteration is extremely small.

Therefore, an error function is formulated by calculating the sum of currents at all nodes. This error function is a measure of the amount by which KCL is violated and is used to adjust the voltage amplitudes and phases. If the method converges (that is, if the error function is driven to a given small value), then the resulting voltage amplitudes and phases approximate the steady-state solution.

- Designers are usually most interested in a system's steady-state behavior. Many high-frequency circuits contain long time constants that require conventional transient methods to integrate over many periods of the lowest-frequency sinusoid to reach steady state. Harmonic balance, on the other hand, captures the steady-state spectral response directly.
- The applied voltage sources are typically multitone sinusoids that may have very narrowly or very widely spaced frequencies. It is not uncommon for the highest frequency present in the response to be many orders of magnitude greater than the lowest frequency. Transient analysis would require an integration over an enormous number of periods of the highest-frequency sinusoid. The time involved in carrying out the integration is prohibitive in many practical cases.
- At high frequencies, many linear models are best represented in the frequency domain. Simulating such elements in the time domain by means of convolution can results in problems related to accuracy, causality, or stability.

Harmonic Balance Setup

The HB method depends on calculating currents and voltages at many harmonically related frequencies for each fundamental signal under consideration. Since we are interested in the steady state solution of a nonlinear problem, we must allow the HB simulator to use enough harmonics so that a Fourier series constructed from these harmonic amplitudes and phases can reproduce a reasonable replica of the time domain solution.

Figure 1 illustrates a very basic HB simulation setup. The Harmonic Balance controller specifies several key simulation parameters. In the example below, one fundamental frequency, Freq[1]=450 MHz, is specified as an input. The index [1] shows that only one fundamental frequency is being considered. Order[1] specifies the number of harmonic frequencies to be calculated (15) for the first (and only) frequency in this case. One of the most common errors in HB simulation setup is to use too low of an order. You can
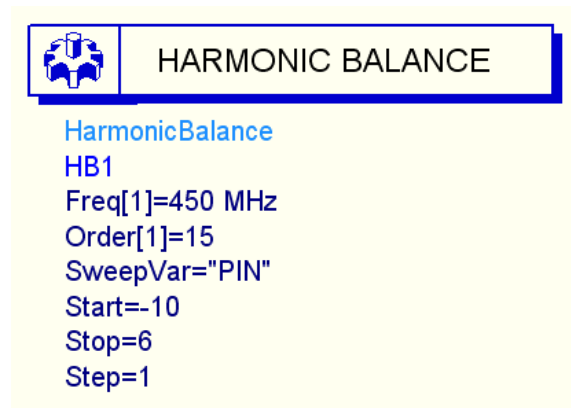
determine what order is optimum if you first simulate your circuit with a small order then increase the order in steps of 1 or 2 harmonics. When the solution stops changing within a significant bound, you have reached the optimum order. Using too high of an order is wasteful of memory, file size and simulation time, so it is not efficient to just clobber the problem with a very high order. Some user discretion is advised.

Figure 1. Example of the HB controller used for a very simple single tone (frequency) simulation. In addition, power PIN is being swept from -10 to 6 dBm.

ADS does not automatically pass parameters from the schematic to the display panel. Calculated node voltages are automatically transferred, but the input parameters used for independent voltage, current or power sources are not (unless they are being swept by a sweep controller setting. Then they become a parameter that is automatically passed to the display.). To transfer parameters from schematic to display, open the controller symbol, select output tab, and then add the variables to the list as seen below:
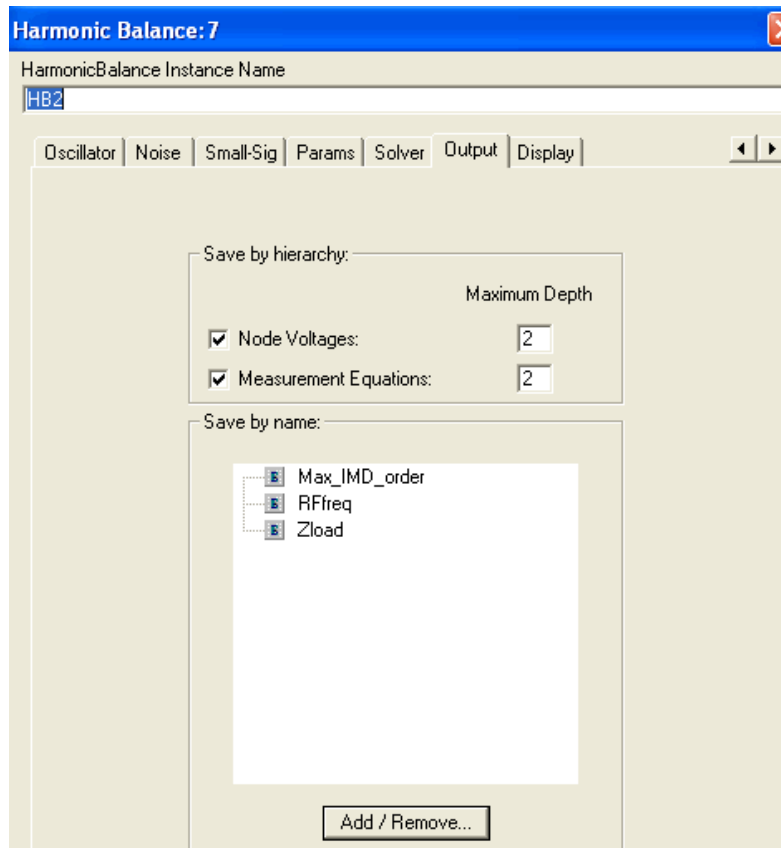
The fundamental frequency of the input source must be the same as specified on the controller. The *Sources-Freq Domain* palette includes many sources suitable for use with HB. The single tone source *P_1Tone* is illustrated below. This source provides a single frequency sinusoid at a specified available power. Here we see that the internal source resistance (50 ohms) is included. The available source power is provided as PIN (in dBm, which will be converted to Watts by the dbmtow function) and degrees of phase.

Figure 2. Single frequency source. Frequency, source impedance, and available power must be specified.

You could also have selected a voltage source, *V_1Tone*, or for multiple frequency simulations, there are *V_nTone* and *P_nTone* sources. These are often used for intermodulation distortion simulations. Voltage or current sources require an external source resistance or impedance whereas the power sources include an internal source resistance or impedance, *Z*.

Nodes must be labeled in the harmonic balance simulation in order to transfer their voltages to the display. If currents are to be used in calculations as well, a current probe must be inserted from the *Probe Components* menu. An example of a PA output network is shown in the next figure. Nodes Vce and Vload are labeled using the Insert pulldown menu: *Insert > Wire/Pin Label.* This opens a text box where you can enter the node name you want. I_ce and I_load were measured with the current probes as shown.
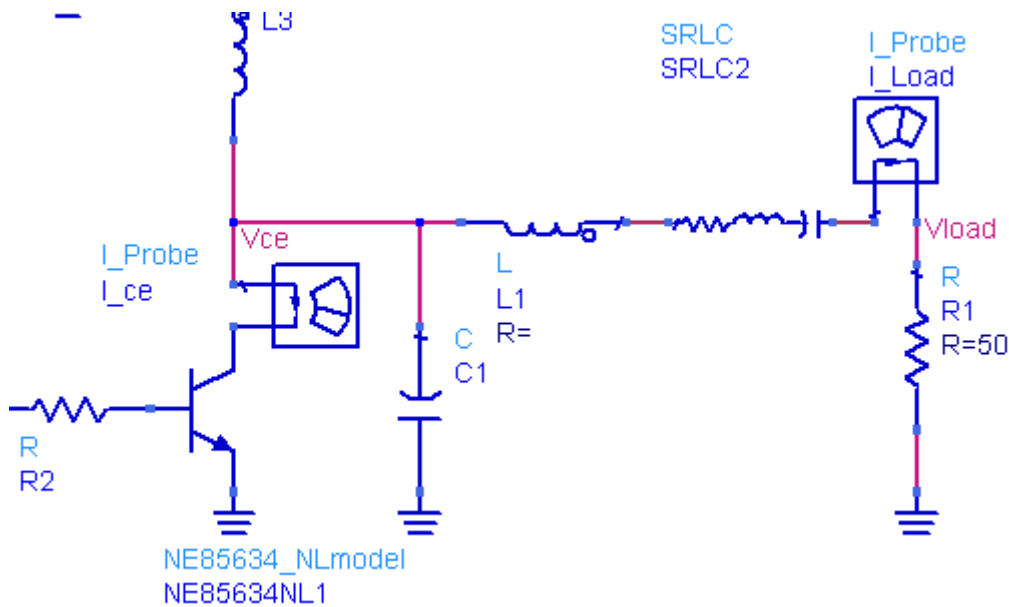

Figure 3. PA output circuit showing node voltage and current labels and probes.

Displaying results

The output voltages and currents calculated by the HB analysis will contain many frequency components. You can display all of them in a spectral display by just plotting the voltage or current on an X-Y plot. Markers can be used to read out the spectral line amplitudes or powers.
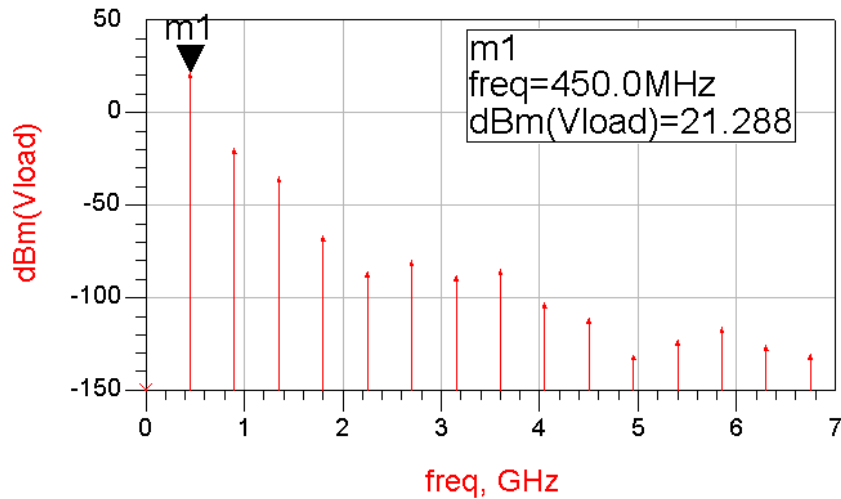
Figure 4. Spectrum in dBm is plotted for Vload. You can see the 15 harmonics.

Often you will want to plot power in dBm. If your load impedance is real, you can use the dbm function in an equation. If the load has a complex impedance, then use the definition for sinusoidal power.

$$P_{out\_dBm} = 10 * \log(0.5 * real(Vload * conj(I\_load.i))) + 30$$

This will give you the power in dBm in all cases. This is the preferred method. Note that calculated quantities much below – 100 dBm are probably not very reliable due to the limited precision of the device models

To perform calculations of power and efficiency, you will want to be able to select specific frequency components. The harmonic index (harmindex) can be used to do this. If you plot your output variable in a table format, you will see a list of frequencies.

freq	Pout	Pout_dBm
0.0000 Hz	0.0000	<invalid>
450.0MHz	134.5m	21.29
900.0MHz	11.47u	-19.41
1.350GHz	336.8n	-34.73
1.800GHz	219.8p	-66.58
2.250GHz	2.250p	-86.48
2.700GHz	9.389p	-80.27
3.150GHz	1.465p	-88.34

Figure 5. Table showing the value of Pout and Pout_dBm at several harmonic frequencies. The frequencies are printed in order and can be designated by an index, ranging from 0 for DC to Order – 1 for the highest harmonic frequency.

The first frequency in the table is DC and has index 0. Fundamental is index 1. So, to select the voltage at the fundamental frequency, for example, you could write Vload[1] or to select power, Pout[1] or Pout_dBm[1] in this example. The second harmonic would be

Pout[2]. Of course, we do not need to draw a table to use the index. For example, the DC component of the power supply voltage can be extracted by using the 0 index: VCC[0]. Then, if the supply voltage and current were measured and passed to the output display, you could calculate DC input power by

$$\text{Eqn} \quad \text{P\_DC=real(VCC[0]*I\_DC.i[0])}$$

To display the results of equations such as this, you use the table or rectangular plot features in the display panel. The data set must be changed to Equations as shown in order to find the result of the calculations.
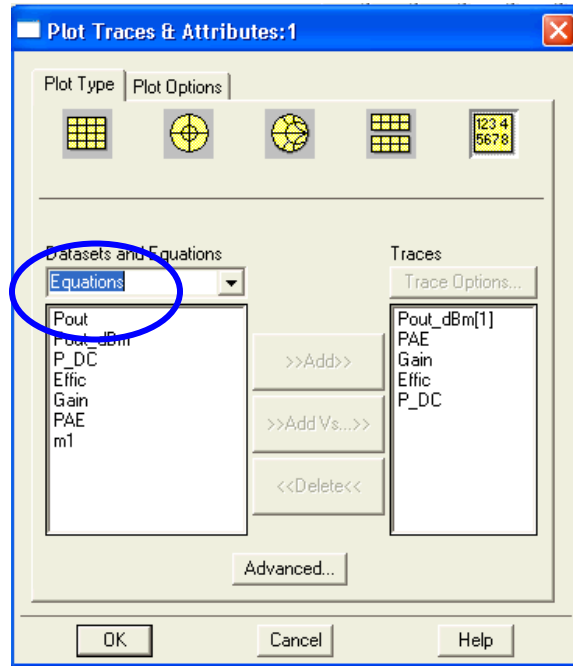
Figure 6. To plot the results of an equation in the data display, select Equations in the data set

If you want to see the time domain version of a voltage or current, the display can perform the inverse Fourier transform while plotting. Select the Time domain signal option.
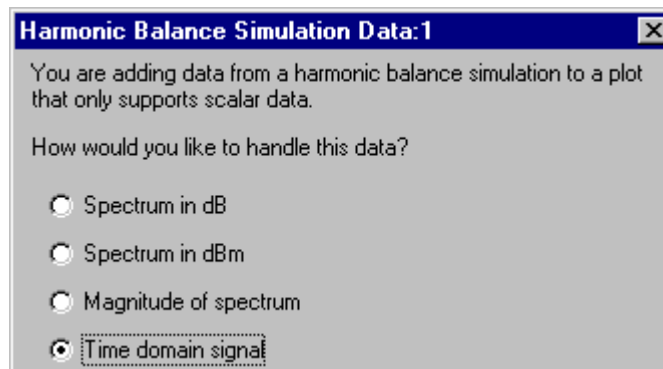

Figure 7. When plotting HB data, you must convert it to a scalar quantity (dB, dBm, or magnitude). Notice that a time domain conversion can also be performed by an FFT if requested.
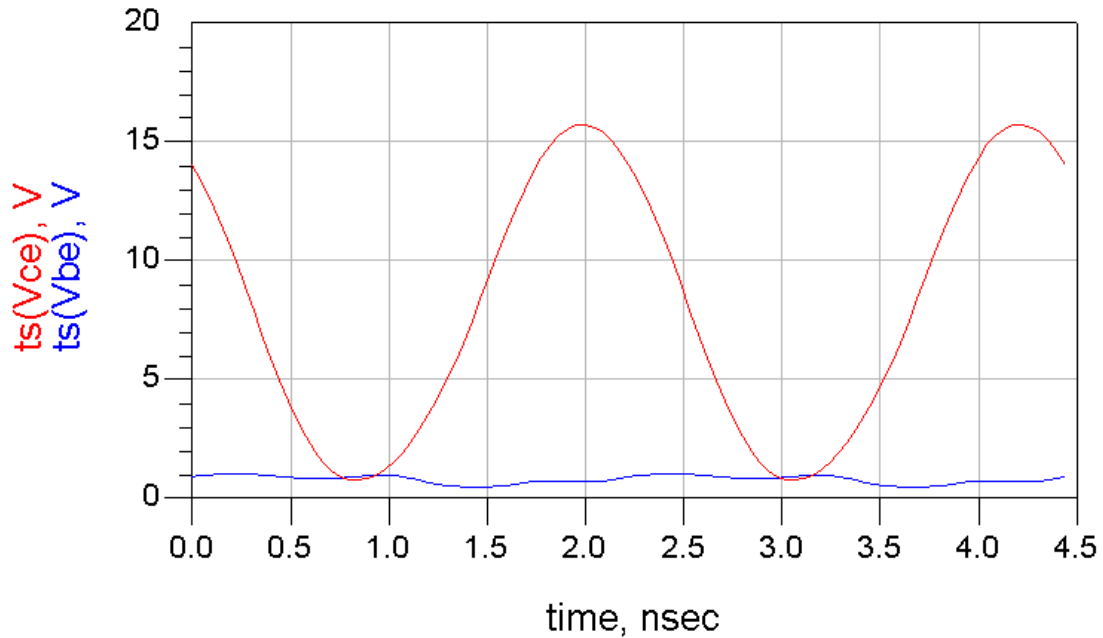
Figure 8. Example of a time domain plot from a HB simulation.

Once the simulation has been run, the data is available on the display panel. You can use equations to calculate power, gain, and power added efficiency. Note the use of the indices once again.

Eqn Pout=0.5*real(Vload*conj(I_Load.i))   Eqn Pout_dBm=10*log(Pout)+30   Eqn P_DC=real(VCC[0]*I_DC.i[0])

Eqn Gain=Pout_dBm[1]-PIN   Eqn PAE=(Pout[1]-dbmtow(PIN))/P_DC   Eqn Effic=mag(Pout[1]/P_DC)

Parameter Sweeps

It is possible to sweep any of the independent parameters in the HB simulation. To set up the sweep, double click on the Harmonic Balance Controller.
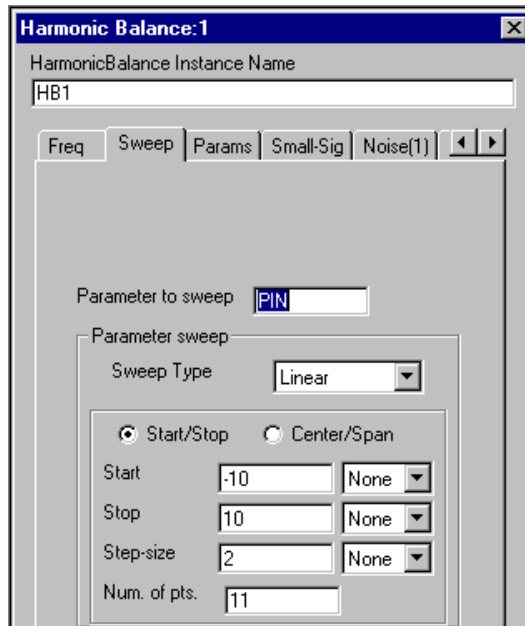
Figure 9. Selecting the Sweep tab allows you to sweep one independent variable.

Click on the Sweep tab. Choose the parameter to be swept, the sweep type (Linear, Log), and the Start, Stop, and Step variables (or number of points instead of step-size). In this example, we are sweeping the input power to the amplifier to determine the gain compression behavior. The more sweep points chosen, the longer the simulation time and the greater the data file size.[2]
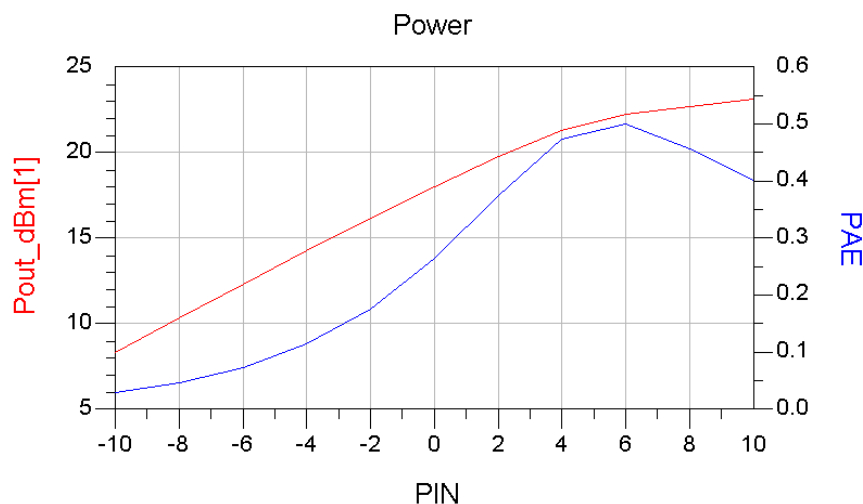

Figure 10. A double axis plot of Pout vs Pin and PAE vs. Pin for a power amplifier.

---

[2] If two variables are to be swept, a ParamSweep controller icon from the HB menu must be added to the schematic.
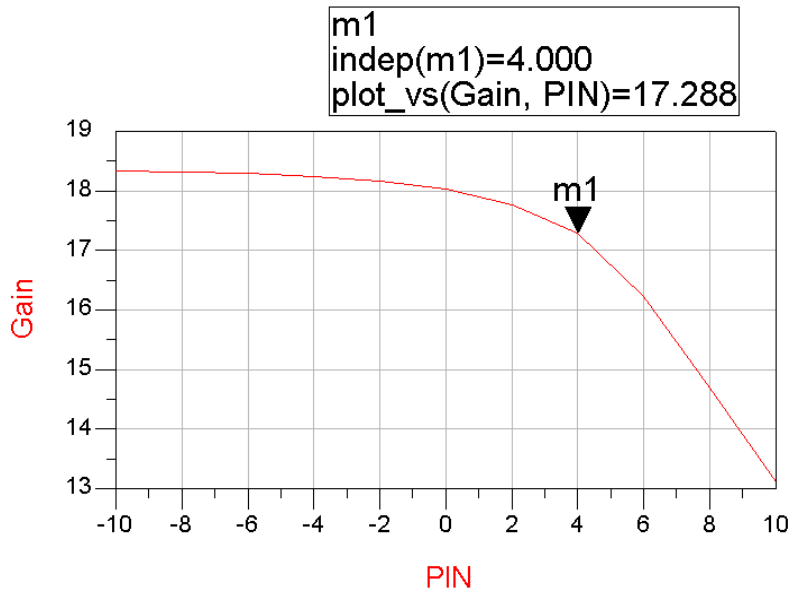
Figure 11. Amplifier gain vs. Pin.

From this plot, we can see that the P_{1dB} compression input power is about 4 dBm.

Multiple frequency simulations

Multiple frequencies or "tones" (mainly two-tone) are widely used for evaluation of intermodulation distortion in amplifiers or mixers. In fig. 12, you can see that now two frequencies have been selected, Freq[1] and Freq[2]. Each frequency must also declare an order (number of harmonic frequencies to be considered).
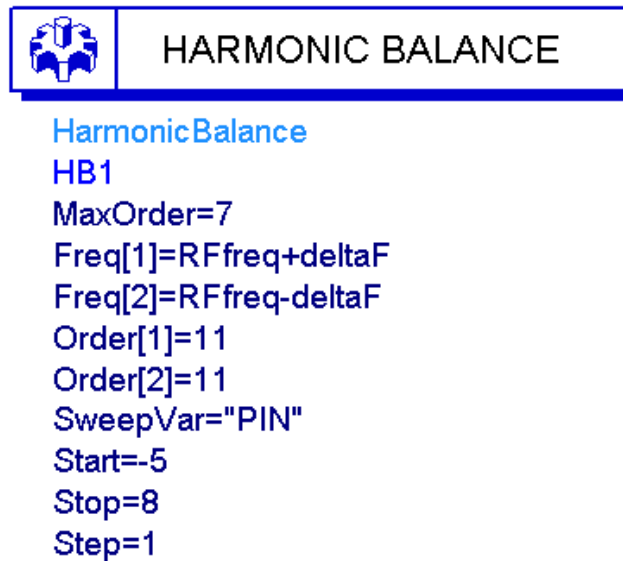

Figure 12. HB controller example for a two-tone PA simulation.

Intermodulation distortion occurs when more than one input frequency is present in the circuit under evaluation. Therefore, additional frequencies need to be specified when setting up for this type of simulation. Two-tone simulations are generally performed with two closely spaced input frequencies. In this example, the two inputs are at 449.8 and 450.2 MHz. The frequency spacing must be small enough that the two tones are well within the signal bandwidth of the circuit under test.

Maximum order corresponds to the highest order mixing product (n + m) to be considered (n*freq[1] ± m*freq[2]). There will be a frequency component in the output file corresponding to all possible combinations of n and m up to the *MaxOrder* limit. The simulation will run faster with lower *MaxOrder* and fewer harmonics of the sources, but may be less accurate. Often accurate IMD simulations will require a large maximum order. In this case, a larger number of spectral products will be summed to estimate the time domain waveform and therefore provide greater accuracy. This will increase the size of the data file and time required for the simulation. Increase the orders and *MaxOrder* in increments of 2 and watch for changes in the IMD output power. When no further significant change is observed, then the order is large enough. If large asymmetry is noted in the intermodulation components, higher orders are indicated.
.
Sometimes, increasing the *oversampling ratio* for the FFT calculation (use the *Param* menu of the HB controller panel) can reduce errors. This oversampling controls the number of time points taken when converting back from time to frequency domain in the harmonic balance simulation algorithm. A larger number of time samples increases the accuracy of the tranform calculation but increases memory requirements and simulation time. Both order and oversampling should be increased until you are convinced that further increases are not worthwhile.

For multiple frequency simulations, the simulation time will be reduced substantially by using the Krylov option which can be selected on the *Display* tab of the HB controller.

The two tone source frequencies are provided with a *P_nTone* generator from the Sources – Freq Domain menu. The two frequencies are sometimes specified in a *Var* block. The same approach is used to specify frequencies in the HB controller so that the effect caused by changes in deltaF could be evaluated by changing only one variable. The available power, PIN, is specified in dBm for each source frequency.
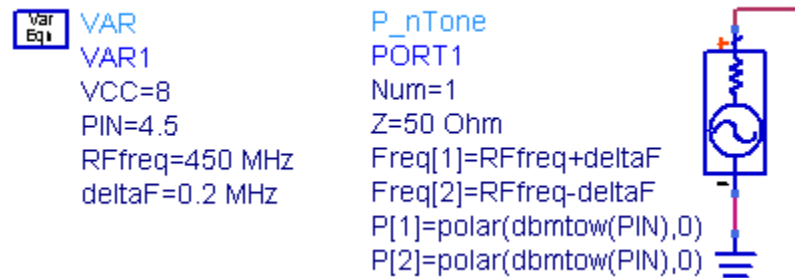

Figure 13. Two-tone source example.

Displaying Results of Multitone Simulations

You can view the result around the fundamental frequencies by disabling the autoscale function in the plot and specifying your own narrow range. The display below shows intermodulation products up to the 7th order (MaxOrder specified on the HB controller).
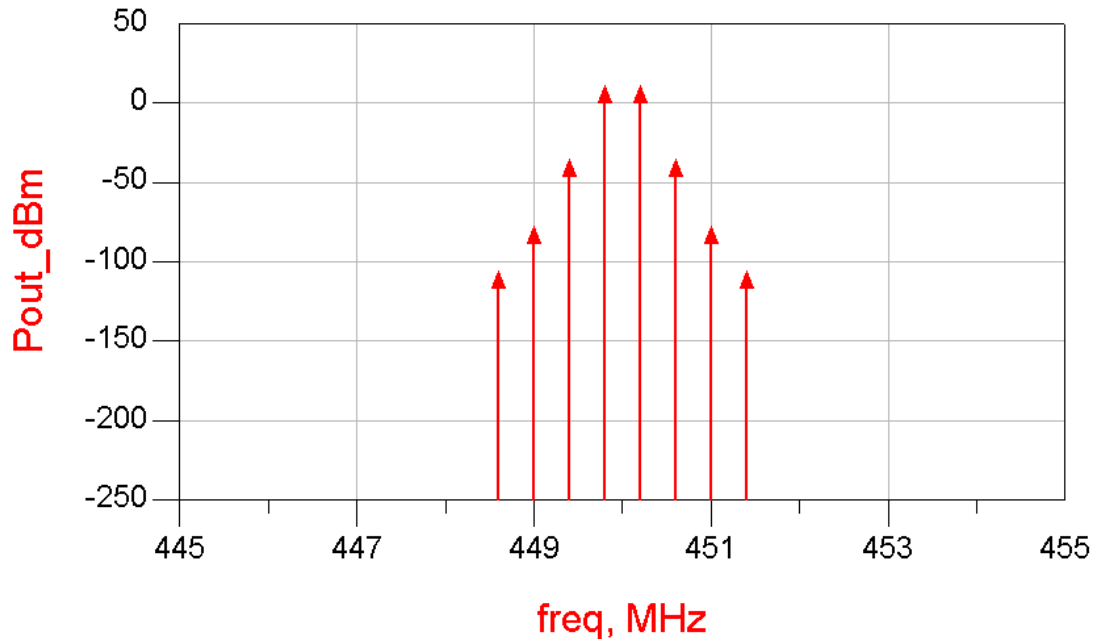

Fig. 14.

We would like to study the output voltage at the fundamental frequencies and the third-order IMD product frequencies. This can be selected from the many frequencies in the output data set by using the *mix* function. The desired frequencies could be selected by:

(RFfreq + deltaF)	Vfund1 = mix(Vload,{1,0}).
(RFfreq - deltaF)	Vfund2 = mix(Vload,{0,1})
(2*(RFfreq +deltaF)-RFfreq - deltaF)	VIM1 = mix(Vload,{2,-1})
(2*(RFfreq - deltaF)-RFfreq + deltaF)	VIM2 = mix(Vload,{-1,2})

The respective indices used with the mix function to select this frequency are shown to the right. The indices in the curly brackets are ordered according to the HB fundamental analysis frequencies.

Mixer Simulations

In the case of a mixer simulation, at least 2 frequencies are always needed: LO and RF. Figure 16 shows an example of the setup used for a two-tone simulation of a mixer. The format is similar to that described above for power amplifier two tone simulations except now 3 frequencies are required. The frequency with the highest power level (in this

example, the LO) is always the first frequency to be designated in the harmonic balance controller. Other inputs follow sequencing from highest to lowest power.

SIMULATION CONTROL

HARMONIC BALANCE

HarmonicBalance
HB2
MaxOrder=11
Freq[1]=LOfreq
Freq[2]=RFfreq+Fspacing/2
Freq[3]=RFfreq-Fspacing/2
Order[1]=9
Order[2]=3
Order[3]=3
Oversample[1]=1
Oversample[2]=1
Oversample[3]=1
UseKrylov=yes

Increase the LO order and max order until there are no significant changes in the IMD power predictions. Oversampling can also be increased as needed.

Figure 16. HB controller example for a mixer simulation.

The harmonic order should be higher for high amplitude signals. For the example above, the LO order is highest because it is intended to switch the mixer. The RF orders can be smaller since they are rarely of high amplitude compared with the LO.
In the case of a mixer simulation, we would like to study the output voltage at the IF frequency. This must be selected from many frequencies in the output data set. A particular frequency is selected by using the *mix* function. In this example, the desired IF frequencies could be:

LOfreq – (RFfreq + Fspacing/2)	V_{IF} = mix(Vout,{1,-1,0}).
LOfreq – (RFfreq - Fspacing/2)	V_{IF} = mix(Vout,{1,0,-1})
LOfreq + (RFfreq + Fspacing/2)	V_{IF} = mix(Vout,{1,1,0})
LOfreq + (RFfreq - Fspacing/2)	V_{IF} = mix(Vout,{1,0,1})

and the respective indices used with the mix function to select this frequency are shown to the right. The indices in the curly brackets are ordered according to the HB fundamental analysis frequencies. Thus, {1,-1,0} selects $1*LOfreq – 1*RFfreq[1] +0*RFfreq[2]$.

DesignGuides

There are so many types of simulations that could be performed on a mixer that it is not reasonable to try to describe them all in this tutorial. Instead, you can use the Mixer DesignGuide, a set of schematic and display templates that can be pulled into your project file. Go to the DesignGuides pulldown menu and select Mixer DesignGuide. Choose a representative sample mixer schematic to modify if you want to create your own mixer circuit. Determine whether your mixer is single ended or differential. The differential circuit templates include baluns; single-ended do not. Choose from a large set of simulation types, some with parameter sweeps, some without.

Refer to the Mixer Design Guide tutorial for more information.

Convergence Woes

Any user of the harmonic balance simulator will eventually encounter convergence problems. Unfortunately, when this happens, no useful information is provided by the simulator. Problems with convergence generally arise when the circuit under simulation is or becomes highly nonlinear. In the case of mixers, there are inherent nonlinearities that are required for the mixing process, but these are usually not so bad unless you are seriously overdriving one of the inputs. If the simulation fails, check the biasing of the transistors. HB doesn't do well with BJTs driven into their saturation region. If that is not the problem, then try decreasing either LO power or the RF power sweep range. You may be driving the mixer well beyond saturation when using default power levels in mixer DG templates. As a last resort, you can try using the Direct solver rather than Krylov, but this will increase simulation time by a large factor.