# ECE145B Design Lab #1
Mixer Design Project - Winter 2005
Report Due Jan 28

You are to design, fabricate and test a BJT mixer. The goal of this lab is for you to become familiar with the design tradeoffs that influence conversion gain, noise figure, gain compression and intermodulation distortion in mixers, not necessarily to produce the world's best mixer. The inputs and outputs are wideband in this design, but the target application is for an FM receiver at the frequencies shown in the table.

Specification	Low end	High end
RF input frequency	88 MHz	108 MHz
LO frequency	98.7	118.7
IF frequency	10.7	10.7
Minimum Conv Gain	6 dB	6 dB

A suggested circuit diagram: All source and load impedances are 50 ohms. Capacitors are for bypass and DC blocking. Choose appropriate values for the frequencies in use.
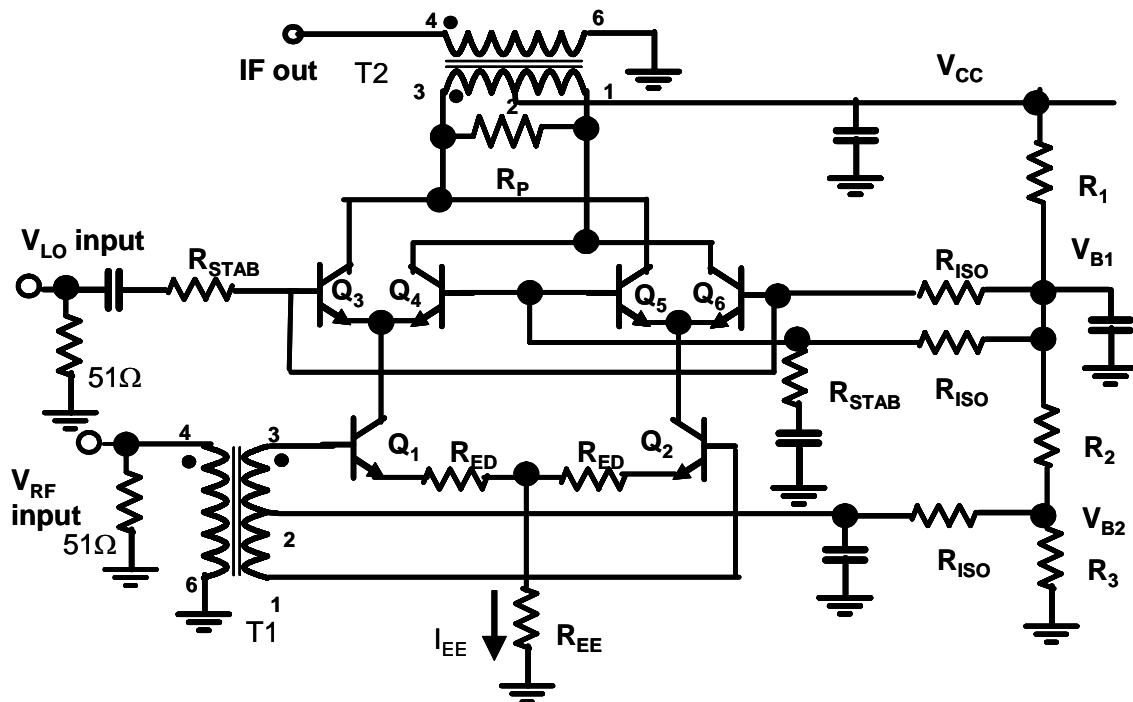
1. Design
A. Do a hand analysis of the DC biasing. R1 – R3 set the bias voltages of the RF and LO sections of the Gilbert cell. R_{EE} and V_{B2} determine the collector current. R_{ISO} is used to avoid losing input power into the bias network. Thus, $R_{ISO} \gg 50$ ohms. Start with I_{EE} = 10 mA. (You will need to explore the dependence of I_C on P_{1dB}, so this may change later) V_{B1} should be set so that the Q1 and Q2 are always in the active region. V_{CC} must

be set to avoid saturation of the LO transistors. Also, with inductive feed, the peak AC collector voltage will exceed V_{CC}, so watch out for breakdown.

MAX RATINGS:
- $V_{BR,CBO} = 20V$ and $I_{Cmax} = 50$ mA for the 2N5179. (Refer to the data sheet for other parameters.)
- Note that the RF transformer has a maximum DC current rating of 30 mA. Data sheets for both are attached at the end of this lab.

B. The RF to IF signal path of the mixer resembles a cascode. An RF transformer is used to transform the 50 ohm IF load impedance to a higher AC load impedance at the collectors. T2 is an RF transformer (T16-6T-W38) that has a turns ratio of 4, for a 16 to 1 impedance transformation. Each collector sees ½ of the transformed impedance. Refer to the MiniCircuits App Note: "How RF Transformers Work", linked from the course web page. The transformer also increases the conversion gain by 6 dB by using the differential output.

Transformer T1 is a Minicircuits T4-1-W38, with turns ratio of 2, whose purpose is to balance the input and increase conversion gain by driving both RF inputs.

The main difficulty with this design has been stability. The circuit will oscillate at about 500 MHz if suitable stabilization is not provided. R_P and Rstab are intended to stabilize the mixer RF to IF path and must be used to avoid this condition. Past experience has shown that a compact layout also is helpful.

Make an estimate of the conversion gain using small signal analysis and assuming the LO transistors are ideal switches.

The load impedance presented to Q3 – Q6 will determine the maximum current and voltage swing, similar to the large signal load line concept in the Class A power amp. At maximum drive, avoid signal swings into the saturation region or into breakdown. Estimate the peak current that will be possible for your given load impedance and your choice of I_{EE} and V_{B1}.

2. Circuit Simulation
Design Goal: Choose V_{CC}, I_{EE}, V_{B1}, LO power, R_{ED} to obtain the highest figure of merit, defined for purposes of this lab as

$$FOM = P1dB - SSB \text{ Noise Figure}$$

with a small signal conversion gain of at least +6 dB. You can assume that P1dB and IP3 will be related, but simulation time for P1dB is considerably shorter. While you should determine power dissipation, it is not necessary to minimize it in this application.

You will be using the 2N5179 bipolar transistor for this mixer. A model for this device can be found in the Analog Parts library of ADS. Since the frequency range is rather low for this application, the models in this library should be adequate for simulation.

After drawing the schematic diagram of your mixer, you will need to make it into a subnetwork. Pay attention to the comment on ADS transformers while you are constructing this. See the pages at the end of this document for directions on how to do this. The benefit will be that you can easily insert the mixer subnetwork into all of the ADS Mixer DesignGuide schematic templates. This will save you hours of work because the setup for all of the ADS simulations you will need is already done. Harmonic Balance method will be used for most of these simulations. Also see the appended document on use of the Mixer DesignGuide and the mixer harmonic balance tutorial on the web site.

You may want to proceed with the design of the mixer using the following steps.
   a. Determine the optimum LO input power by simulating P1dB as a function of LO power. You will use the LO sweep option for this simulation.
   b. Fix the LO power at the lowest value that gets you close to the highest P1dB. Evaluate P1dB (conversion gain and gain compression simulation) and single-sideband mixer noise figure as a function of your other circuit parameters: I_{EE}, R_{ED}, V_{B1}, V_{CC}. There will be tradeoffs between conversion gain, noise figure and P1dB. For example, larger I_{EE} gives higher P1dB but also higher NF. Higher R_{ED} will increase P1dB but also increases noise figure and reduces conversion gain.
   c. Once the design space has been narrowed down, you must determine the optimum value of R_P and Rstab for stability. The mixer is potentially quite unstable and will oscillate at VHF frequencies if stabilizing resistors are neglected. To explore stability, make and rename a modified mixer subnetwork where Q4,Q5 are biased off all of the time. Ground the LO input and now you have a cascode amplifier. Use the same small signal S parameter stability analysis that you used in last quarter's amplifier designs to plot the k factor and stability circles. Don't make Rp unnecessarily low or you will forfeit conversion gain.
   d. Simulate the IIP3 of the mixer. Since FM station carriers are separated by 200 kHz, use this as the frequency separation for the two-tone third-order IMD simulation.
   e. Examine the output spectrum and identify spectral lines up to order 4 (with two input frequencies). Determine LO – RF, LO – IF, and RF – IF isolation.

3. Construction of the mixer.
We will be fabricating custom PC boards for lab 1. This service has been graciously provided by the internet PCB vendor, ExpressPCB. The boards are subject to the following restriction:

   • The PCB dimensions must be 3.8 x 2.5 inches. This is their standard miniboard size and the only one they will support for this class.

- Board material is FR-4. You should assume $\varepsilon_r = 4.5$.

You can access the layout software on the PCs in 5162 or ECI. Or you can download the software onto your own PC. In either case, visit http://www.expresspcb.com for an orientation on their layout software and board specifications. If you use your own PC, then you must also download the component footprint (W38.p) for the transformers that is found on the course website. Copy this file into the following directory:
    Programs\ExpressPCB\PCBComponents_Custom

When your layout is ready to submit, carefully check it against your schematic. Then generate a plot file for manual order (Layout menu). This will catch some of the layout errors if they exist.

email **BOTH** the .pcb and the .plt file to: long@ece.ucsb.edu. I will submit all of the layouts for the class in one batch. DO NOT SEND AS A .zip FILE. The department server will automatically delete these.

THE BOARD LAYOUTS ARE DUE BY 9 AM ON MONDAY, JAN. 17.

4. Testing the mixer.
Use generators to test the mixer performance with the IF output connected directly to the spectrum analyzer. Perform the measurements indicated below.

1. Plot conversion gain vs. LO drive power.
2. Plot 1 dB compression input power level (P_{1dB}) vs. LO drive power.
3. Measure the frequency response of the RF port. Adjust both the LO and RF frequencies to find out the upper 3 dB frequency for your mixer while keeping the IF frequency fixed at 10.7 MHz.
4. Input intercept point (two-tone IMD measurement). Use a 2 tone test configuration to determine the third order input IMD intercept. Use the same frequency plan as you used for the simulation. (Note that you will need to use 3 generators for this measurement; one for the LO and 2 for the inputs).
5. Identify spectral lines up to 1 GHz and measure LO – RF, LO – IF, and RF - IF isolation.

Finally, connect the mixer to the FM IF/demodulator/audio amplifier box, attach a clip lead to the mixer input, and see how many FM stations you can tune in. Or, if you want to try to measure DSB NF with the spectrum analyzer, see the TAs or Prof. Long.

5. Report.
Refer to the course introduction for expectations on reports. The report must include a section on your design and simulation efforts. Describe the effect of the various tradeoffs in design parameters, predict the performance of your mixer, and compare measurement with simulation.

Parts list for ECE145B   Mixer Design Lab   Winter 2005

1	Minicircuits T16-6T-W38 RF Transformer
1	Minicircuits T4-1-W38 RF Transformer
6	2N5179
3	10 uF Tantalum Caps
3	board mounted SMA connectors
24"	Pair of insulated wires for power connection.

NPN

2N5179

1. EMITTER
2. BASE
3. COLLECTOR
4. CASE
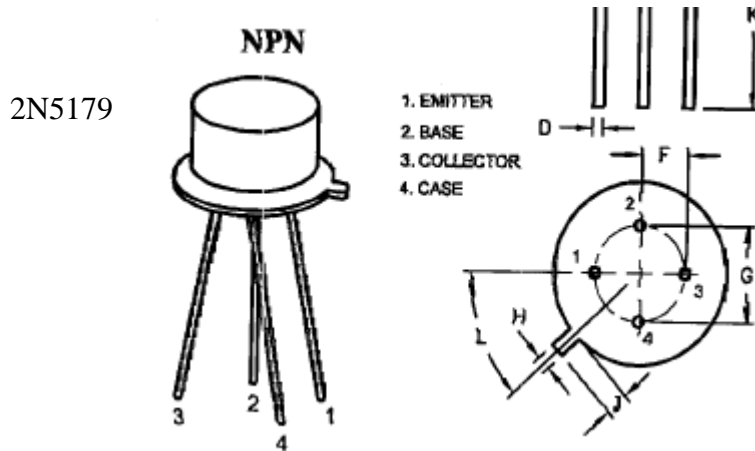
Comments on transformers in ADS.

TF3
TF1
T1=0.5
T2=0.5
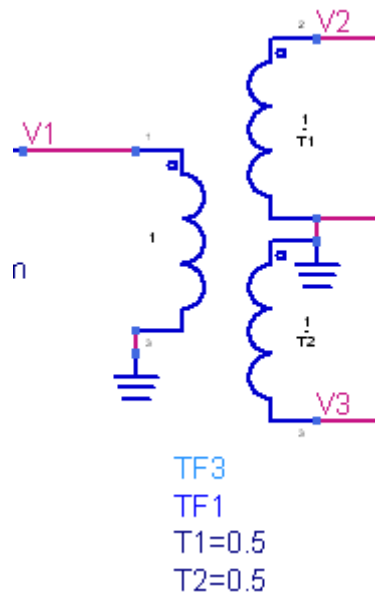
In ADS, the ideal center-tapped transformer is the TF3 icon in the components menu. For some unknown reason, however, the turns ratio is inverted from what you would expect to see. The primary side always is 1. The secondary windings are specified as 1/T1 and 1/T2. Hence, the T16-6T transformer, which has a total turns ratio of 4

(impedance transformation of 16), is specified as above. Each half of the secondary should have a ratio of 2, thus T1 = T2 = 0.5. This is a frequent source of confusion!

The ratio of V2/V1 and V3/V1 should ideally be N = 1/T1 = 1/T2. The impedance presented to ports 2 and 3 is ½ of the total transformed impedance. Thus, if the source impedance at the primary is 50 ohms, you should see 400 ohms at ports 2 and 3.

Appendix 1: Defining a Subnetwork in ADS:

Subnetworks allow you to take advantage of the heirarchy possible in ADS. You can save circuits or portions of circuits that you plan to reuse as subnetworks which can then be accessed in the Component Library.

Here are the steps:

Begin from your schematic of the circuit that you want to represent as a subnetwork.

1. Insert and label connectors from the tool bar. In a mixer, port 1 is the RF port, 2 is the IF and 3 is the LO port.
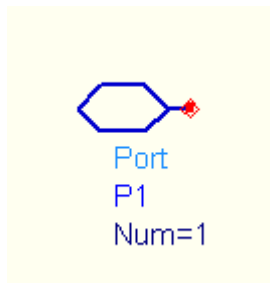
Complete the construction of your schematic for the subnetwork. Note that you will need to include transistor models into the subnetwork if your circuit requires transistors. Name and save the subnetwork.

2. . Create Subnetwork symbol:
> *View > Create/Edit Schematic Symbol*
> The Symbol Generator panel opens up. Click on the *Copy/Modify* Tab
> There is a library of symbols already available. For a mixer, use the *System-Amps & Mixers* lib.
> To return to the schematic, choose *View > Create/Edit Schematic Symbol* (again)
> Save the file

3.   Name the circuit and set the design parameters
> *File > Design Parameters*

> General tab.   The component name will be shown, BJT_SB in this example.
> Name the component instance. For example: Mix.   When the component is
> included in a schematic, each instance will be numbered sequentially.
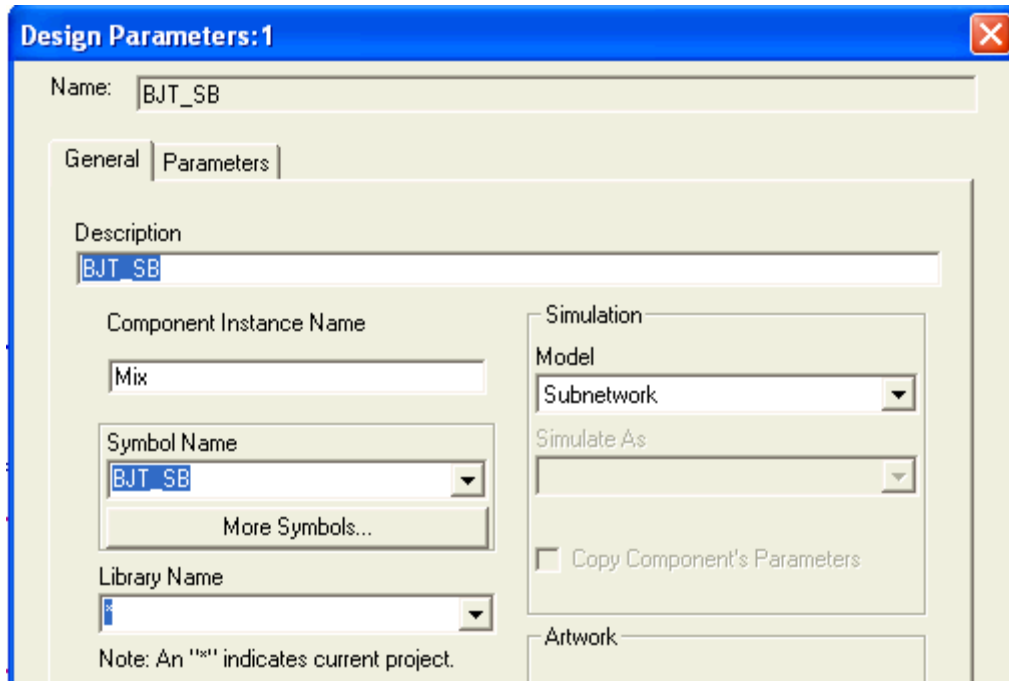
5. If you want to allow adjustable parameters from outside the subnetwork, use the Parameters menu tab. This is useful when you want to sweep some parameter that is inside the subnetwork. In this example, 4 parameters are declared to be adjustable. Their values are given a default, but can be specified again outside the model in the schematic where it is being used.
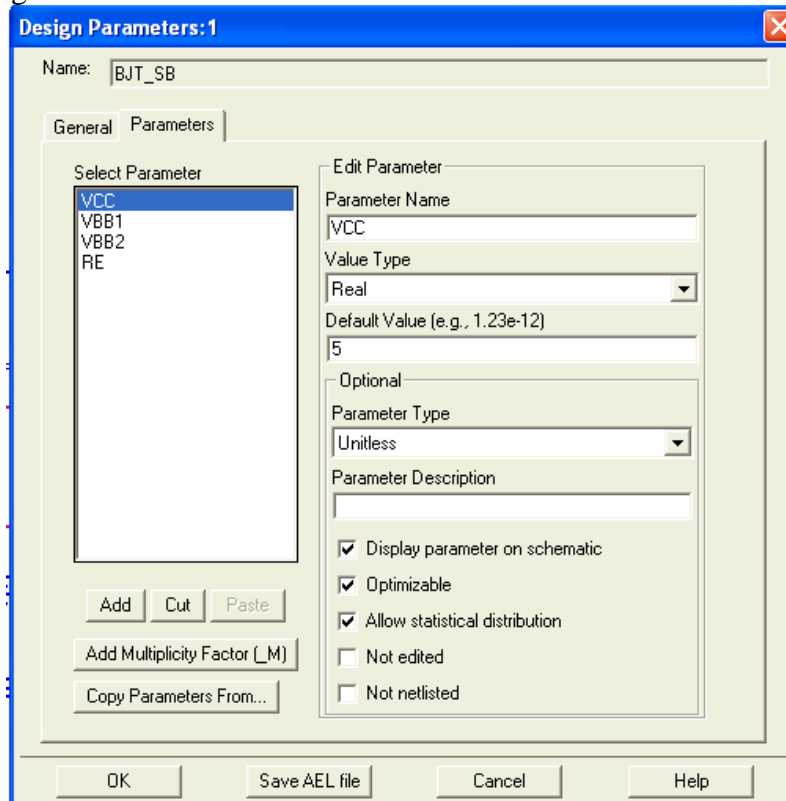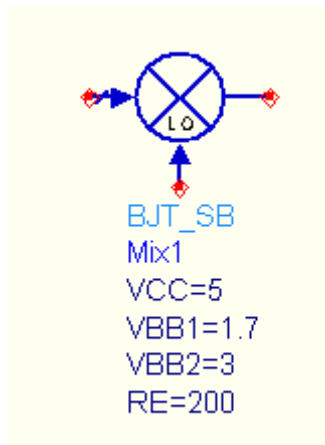
6. Save the subnetwork again.

7. To insert your mixer into the schematic, use the component library icon (the bookshelf icon) on the tool bar to open the library. Select your project, then the component you wish to insert. You can then point and click into the schematic.
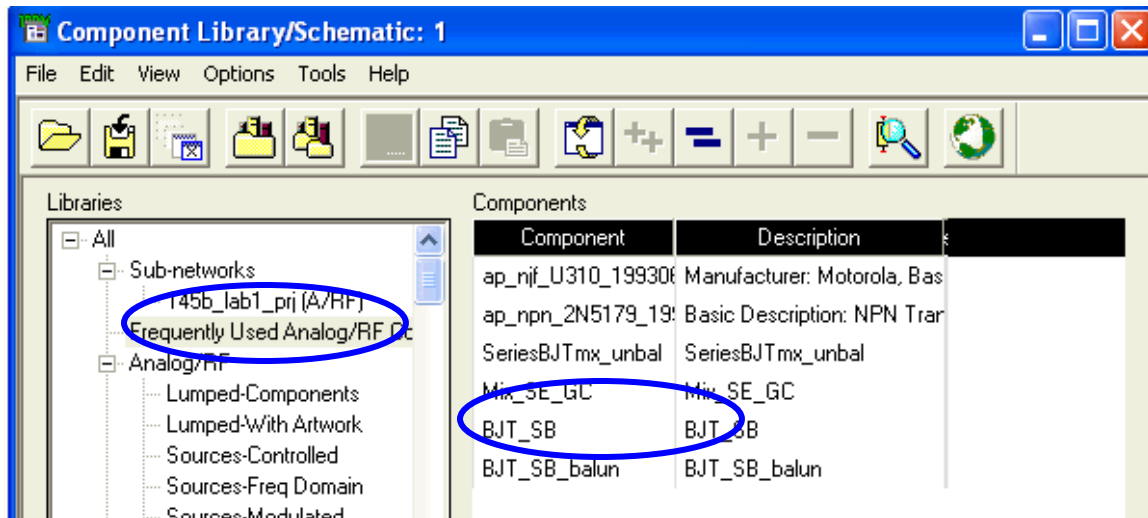
The symbol appears like this in the schematic:

7. You can push into the subnetwork to view or modify it with the down arrow icon on the toolbar.


Rev: January 2003

8
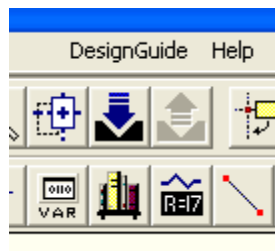
ADS Tutorial: Using the Mixer Design Guide

Agilent provides all ADS users with a large selection of "Design Guides" covering many areas of interest in microwave circuits and communications. These DGs consist of a collection of schematic and display templates that are preconfigured for relevant simulations. As you will quickly see, you can save days or weeks of time by importing the most relevant schematic/display pair and modifying it for your own application instead of starting from nothing.

To import into your ADS project, use the DesignGuide menu.
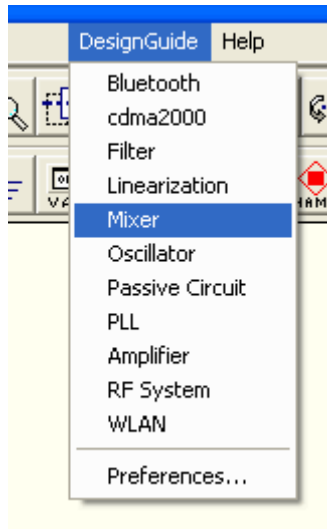
When you highlight and click, the menu expands as shown below.
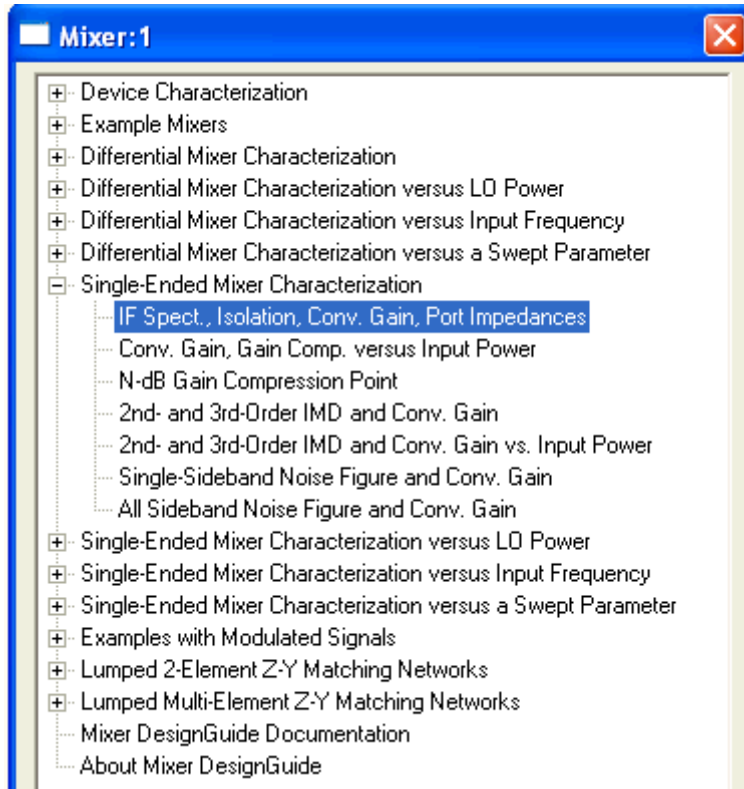
You will find a large selection of mixer characterization templates and example mixers. These are grouped according to differential or single-ended mixer interfaces. Within each group, you will find 4 categories of measurements, all using the harmonic balance simulation mode.

> **1. Mixer Characterization**. These are either fixed (non-swept) simulations or they are RF input power sweeps.
> **2. Mixer Characterization vs. LO Power**. The same set of simulations with the LO power swept.
> **3. Mixer Characterization vs. Input Frequency**. RF Input frequency is swept.
> **4. Mixer Characterization vs. Swept Parameter**. Use this if you want to evaluate some aspect of mixer performance as a function of any other swept parameter (bias voltage or current, component value, device widths, etc.)

Within each category, there is a set of 7 measurement templates.
> a. **IF Spect, Isolation, Conv. Gain, Port Impedances**. Spectral display of the IF output; LO – IF, LO – RF, and RF – IF isolation; Conversion gain at fixed input power; small signal input impedances.
> b. **Conv. Gain Gain Comp. vs. Input Power**. Simulate conversion gain plot gain compression while sweeping the RF input power.
> c. **N-dB Gain Compression**. A more limited version of b. No plots, just a table.
> d. **2nd and 3rd order IMD and Conv. Gain**. Simulate two-tone intermodulation power and intercepts with a fixed RF and LO input power.

e. 2nd and 3rd order IMD and Conv. Gain. Vs. Input Power. This performs the RF power sweep and plots fundamental, 2nd order, and 3rd order powers on a dB vs. log freq. plot.

f. SSB Noise Figure and Conv. Gain.   A harmonic balance single sideband noise figure simulation. This accounts for large signal effects and mixing of noise from LO harmonics.

g. All Sideband Noise Figure and Conv. Gain.   This uses the hot-cold noise figure simulation method and calculates a double-sideband NF, again using harmonic balance.

When you select one of the template pairs, it will import the files into your ADS project. You are then free to modify it for your own application. Paste in your own mixer in place of the default one. Modify frequencies, powers, change parameters, sweeps etc. as needed.

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; try decreasing either LO power or the maximum RF power input. You may be driving the mixer well beyond saturation when using default power levels in mixer DG templates.

If this does not succeed, you can use a transient assisted harmonic balance technique. See the course web site for an example.

Rev: January 2005