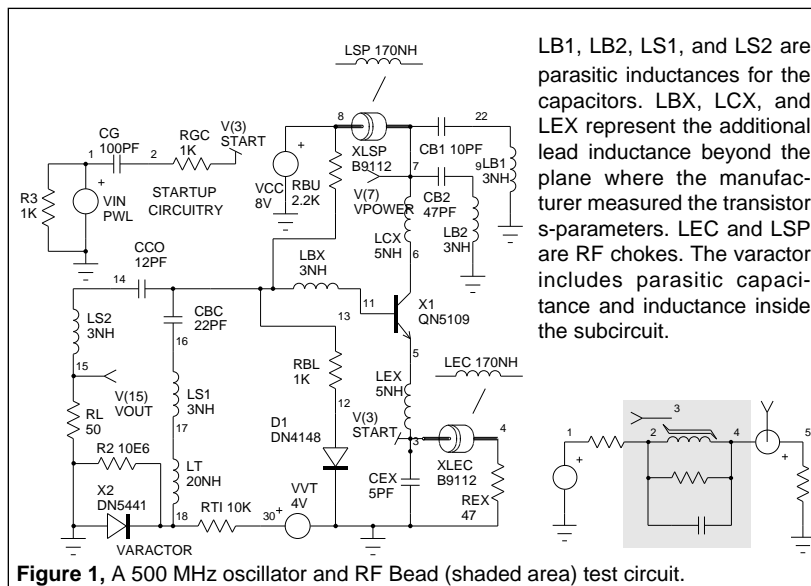


New Models Simulate RF Circuits

It's no news to those who simulate that the accuracy of SPICE is directly related to the accuracy of the models. What may be news is that simulation of high frequency circuits well into the gigahertz range is now possible due to the introduction of some new RF SPICE models.

To illustrate the improved simulation accuracy, the RF models were used to analyze a 500 MHz oscillator (Figure 1). The oscillator generates a relatively high power output with a very distorted output waveform (typical at power levels over the 1-2mW range). The simulation goals were to study the start-up characteristics, oscillating frequency and amplitude, and the resulting harmonic distortion. Two simulations were run. The first using the standard Gummel-Poon BJT model and simple inductor chokes and a second using an improved 2N5109 model along with a new Intusoft RF bead model from the RF Device Library.

Above approximately 100-200 MHz, the built-in SPICE BJT model, based on the Gummel Poon model, fails to accurately predict the real device performance. The BJT must be remodeled as a subcircuit (Table 1) in order to accurately model the package and bond wire parasitics which are of greater significance at higher frequencies. The improved model includes the package parasitics and matches the s-parameters up to 2 GHz. The RF library was created by Analog & RF Models, specialists in the creation of RF models, for Intusoft [1].



Modeling An RF Bead

Although SPICE does contain a model for a nonlinear inductor it does not have a built-in bead model. Fortunately, one can be created using the a nonlinear magnetics model and passive elements. The new model, as shown in Figures 1 & 2, accurately simulates the impedance vs. frequency response and the change in impedance vs. temperature. The change in impedance with DC bias is also modeled due to the addition of the core model instead of a simple inductor. The bead model gives a very low DC impedance while providing a large impedance at higher frequencies. The model used here is for a ferrite bead made of a medium permeability nickel zinc material from Fair-Rite, P#2743009112. Other types of beads including beads on leads, wound beads and surface mount beads can be found in the RF library [2].

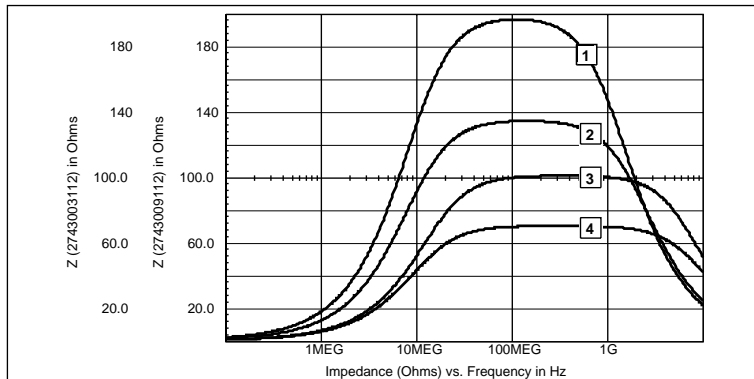
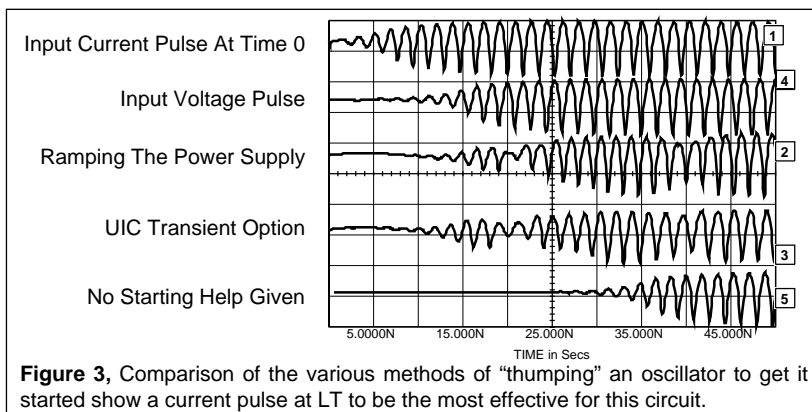


Figure 2, The Intusoft RF Bead model simulates the proper impedance vs. frequency characteristics. The graph displays the response for several devices.

Starting An Oscillator

Simulation of oscillators present a variety of challenges, not the least of which is getting the oscillator to oscillate. When IsSPICE performs an AC or Transient analysis it first performs a DC analysis in order to establish the starting initial operating point for the circuit. If a stable operating point is found, which is the goal of the DC analysis, the oscillator may not oscillate during the transient (time domain) analysis unless some random disturbance is encountered.

There are a number of ways to start an oscillator; each with varying results and consequences (Figure 3). The method chosen here was to introduce a voltage pulse into the circuit, specifically, at the emitter of the transistor (`VIN 1 0 PWL...`). Another possible method is to insert a current pulse somewhere in the resonant portion of the circuit, for example at LT (`I1 18 17 PULSE .01 0`). If the DC analysis does not converge, a sign that the circuit



is unstable and may want to oscillate without any help, the UIC keyword can be issued in the .TRAN statement. For example, `.TRAN .1NS 50NS UIC`. This will cause the simulation to proceed directly to the Transient analysis bypassing the DC analysis. The .IC and “IC=” parameters can then be used to set initial transient conditions and unbalance the oscillator. One problem with using UIC is that no DC operating point will be produced inhibiting study of the circuit bias. The last method is similar to the first and involves ramping of the power supply (`VCC 8 0 PULSE 0 8V 0 5NS`). This method may not work well, however, due to the bypass capacitors. In general, when ramping a source, make sure to give the ramp a realistic slope in order to avoid “timestep too small” errors. The first two methods are the most often recommended as the other methods may not work or may introduce transient start-up residues [3].

Circuit Modeling

The oscillator circuit was first simulated with two inductor chokes, LSP and LEC (Figure 1), and a standard .MODEL statement for the 2N5109 transistor. In SPICE, the standard representation for a transistor uses the Gummel-Poon model. Various parameters in the SPICE .MODEL statement are altered in order to cause the “generic nature” of the Gummel-Poon template to represent a particular device.

In a second simulation, the chokes were each replaced with the new RF bead model. The simple transistor model was replaced with a subcircuit representation. Since the Gummel-Poon model can not adequately represent BJT behavior above approximately 200MegHz, a composite model must be assembled. The SPICE subcircuit, containing a BJT model and various parasitic elements, is utilized for this purpose.

When simulating in SPICE it is best to use a subcircuit representation for a device rather than forcing model parameters to have unreasonable values. If model parameters are used outside their physical bounds, the model may work well in one area, but incorrectly in another. For example, the device may behave properly during the AC small signal analysis, but poorly during the nonlinear transient analysis. Some vendors who produce SPICE model use this approach and the user should beware. Models for RF transistors that are going to be used for both linear and nonlinear analysis can not be produced this way. A subcircuit representation must be used!

Note: Parasitics for the passive elements were maintained for both simulation cases.

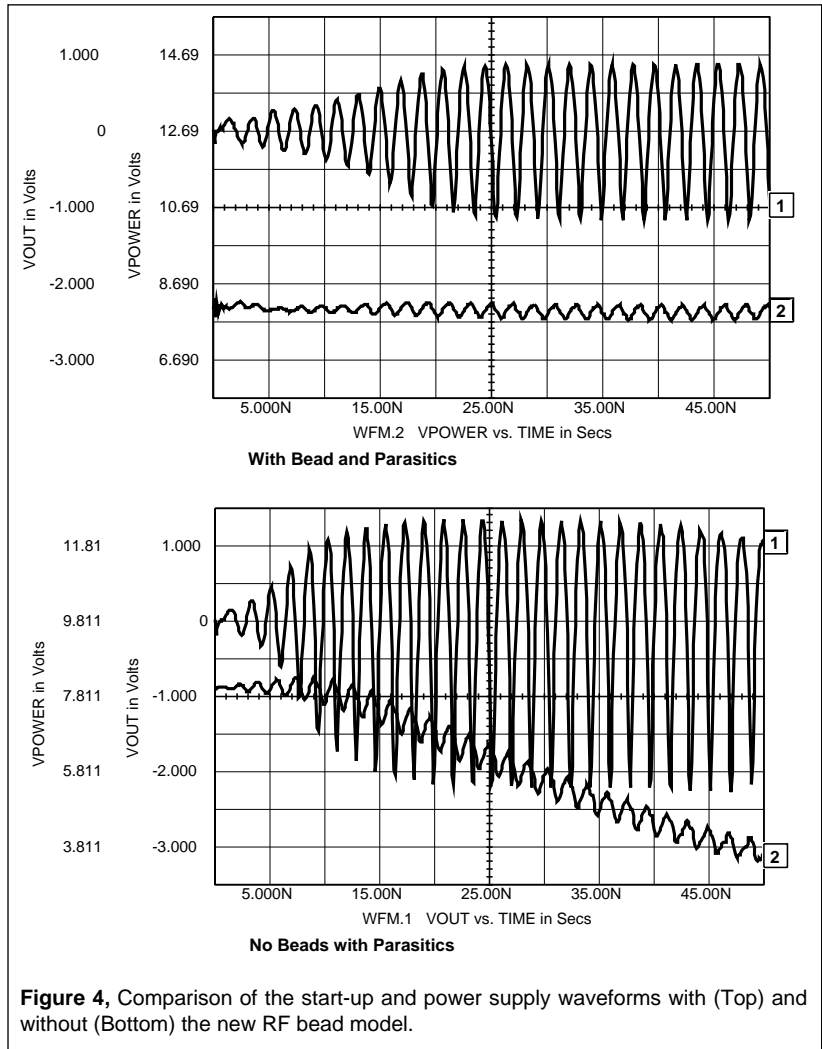


Figure 4, Comparison of the start-up and power supply waveforms with (Top) and without (Bottom) the new RF bead model.

Results

In order to isolate VCC and not contaminate the power supply with the 500 MHz oscillating waveform, adequate bypassing is required. A voltage generator represents a perfect bypass because it is zero ohms at all frequencies. This is quite different from the real world.

As shown in Figure 4, use of an inductor causes a droop in the VCC voltage. The reason for the droop is that the 170nH represents a large impedance. As the oscillator starts up, the transistor wants more current. Because of the large inductance it can't draw adequate current so it starts to discharge the bypass capacitors. This appears as a drooping in the VCC power (lower graph). The bead, on the other hand, has a very low DC impedance and a high AC impedance. By choosing the proper bead, a frequency response can be selected that will block all the AC around the oscillation frequency. With the bead (upper graph), the VCC line doesn't droop and shows that the size of the ripple stays the same revealing the imperfections in the bypassing. Also note that the oscillation starts slower and does not have quite as much power out with the bead inserted (to be expected) as it does when the inductors are used.

The oscillator with no BJT parasitics or beads still oscillates because it was made to be tolerant of package parasitics, but the results predicted were inaccurate in several important areas. As shown in Figure 5, the FFT and transient response of the circuit with beads and new RF BJT subcircuit model reveals that the frequency of oscillation is lower and the distortion higher.

Conclusions

From the simulations performed, it is clear that modeling the proper circuit parasitics is of vital importance, especially at RF frequencies. It is recommended that initial simulations run for many cycles in order to verify that stable oscillation is actually taking place. As for performing an FFT, the `IsSPICE .TRAN tstart` parameter can be used to delay the start of data taking until steady state oscillation has been reached.

With the new bead and BJT models in the RF library, `IsSPICE` is able to show the peak component stresses when power is applied, the transient start-up performance, and the variations in the power supply. In contrast to the linear analysis programs commonly used by RF designers, `IsSPICE` simulations can reveal many important circuit properties such as efficiency, power dissipation, start up characteristics, and harmonic distortion. Characteristics that would be either difficult or impossible to measure.

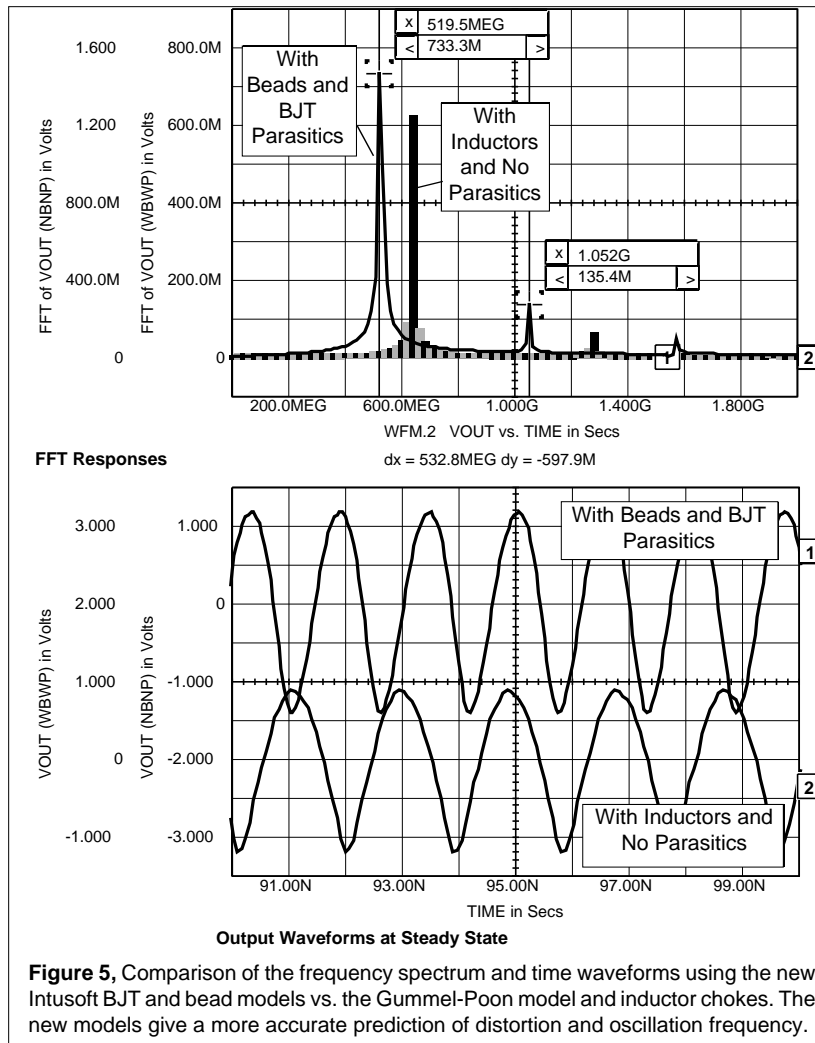


Figure 5, Comparison of the frequency spectrum and time waveforms using the new Intusoft BJT and bead models vs. the Gummel-Poon model and inductor chokes. The new models give a more accurate prediction of distortion and oscillation frequency.

Table 1, IsSpice Subcircuit for an RF Transistor & Bead	
<pre> .SUBCKT QN5109 1 2 3 LC 1 4 0.875E-9 LB 2 6 1.590E-9 LE 5 3 2.650E-9 CC 4 3 1.410E-12 CB 4 6 0.561E-12 Q1 4 6 5 QR33 .MODEL QR33 NPN (BF=44 VAF=160 VAR=16.0 RC=0.69 +RB=1.57 RE=2.75 IKF=0.28E+00 ISE=0.36E-13 TF=0.111E-09 +TR=0.80E-08 ITF=0.82E-01 VTF=0.66E+01 CJC=2.758E-12 +CJE=1.822E-12 XTI=3.0 NE=1.5 ISC=0.12E-13 EG=1.11 +XTB=1.5 BR=1.14 VJC=0.75 VJE=0.75 IS=0.40E-14 +MJC=0.33 MJE=0.33 XTF=4.0 IKR=0.28E+00 KF=0.1E-14 +NC=1.7 FC=0.50 RBM=1.1 IRB=0.40E-01 XCJC=0.5) .ENDS </pre>	
<pre> .SUBCKT BEAD 1 2 R4 1 2 220 TC=-.00333 C2 1 2 .9PF RX 3 2 1E12 CB 3 2 7.432N F1 1 2 VM1 1 G2 2 3 1 2 1 E1 4 2 3 2 1 VM1 4 5 RB 5 2 341.5 RS 5 6 3.7904 VP 7 2 300 D1 6 7 DCLAMP VN 2 8 300 D2 8 6 DCLAMP </pre>	

References

- [1] Analog & RF Models, Bill Sands (602) 575-5323, FAX (602) 297-5160
- [2] RF Device Library, Intusoft, 1990
- [3] "SIMULATING WITH SPICE", L.G. Meares, C.E. Hymowitz, Intusoft, 1988