

Intusoft Newsletter



Personal Computer Circuit Design Tools
Copyright © Intusoft, January 1990

New IntuScope Update

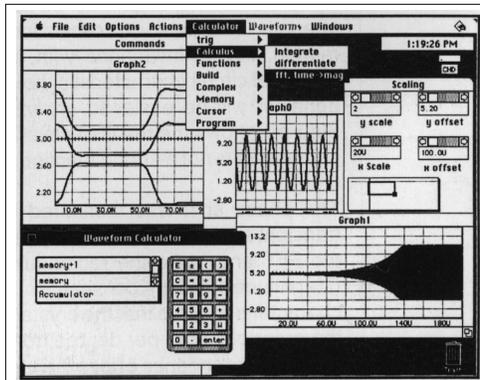
Intusoft will be shipping a major upgrade to the IntuScope program beginning this March. Some of the new features being added to IntuScope version 3.0 include:

- Unlimited Data Points (> 20,000)
- Multiple Graphs with Multiple Waveforms from Multiple Output files
- Improved Output Device and DTP support

plus: Pop-Up Menus using the new AT&T open look interface, Transcript Window with waveform manipulation using equations in either algebraic or reverse polish notation, all functions accessible from menus, support for multiple temperature runs, improved waveform manipulation and scaling, and file compatibility between the Macintosh and PC. Both Macintosh and PC versions (shown right) will be available and have exactly the same menu interfaces providing easy learning for users of both platforms. Upgrades from the current version, 2.11, are free if IntuScope was purchased after 11/89.

In This Issue

- 1 New IntuScope Update
- 1 New Spice Applications Handbook
- 2 **Simulating a VCXO;**
- 3 AC Analysis Issues
- 8 Modeling Crystals
- 8 Modeling Varactor Diode
- 10 SPICE Opamp Models



Do You Like The Spice Newsletters ?

Do you find the Intusoft newsletter useful? Would you like to know how you can obtain back issues? Would you like to have all of the Spice models and circuits? If the answer to any of these questions is yes,

Continued on page 16-10

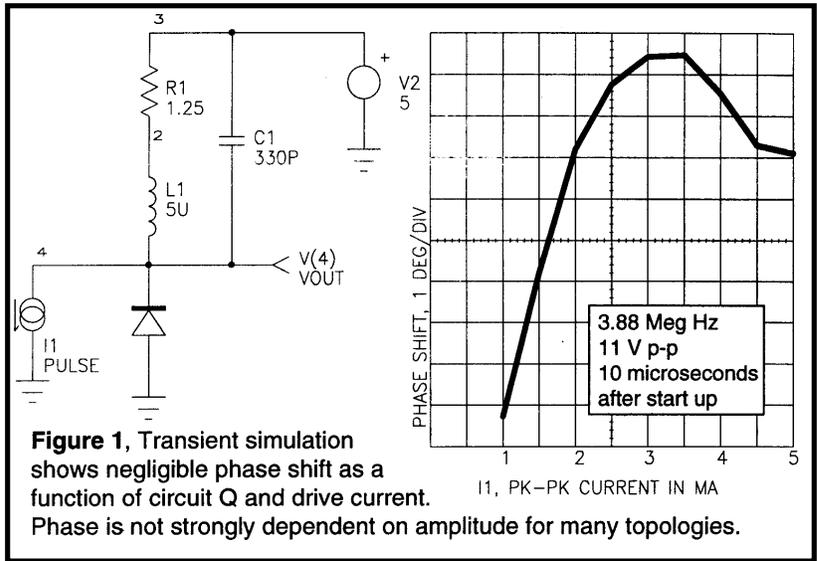
Simulating Crystal Controlled Oscillators

Designing precision oscillators using IsSPICE presents some interesting challenges. Measurement of performance in a laboratory is accomplished using time domain instrumentation, for example, a counter and oscilloscope. On the other hand, the cumulative propagation of small numerical errors prevents a similar approach when analyzing a circuit design with a simulator. An alternate approach is available using AC analysis techniques for predicting circuit performance.

Before describing this method in detail, it is first necessary to recall how stability criteria can be used to predict an oscillators frequency. When a circuit has negative gain and phase margin, any random disturbance will grow exponentially until the resulting signal reaches a limit. The limit will be dependent on device operating points, power supply voltages, and circuit topology. The small signal circuit gain will drop toward zero as the nonlinear limits are approached. One can then assert that when the average loop gain is unity, the circuit will oscillate at a frequency whose average fundamental frequency phase margin is zero.

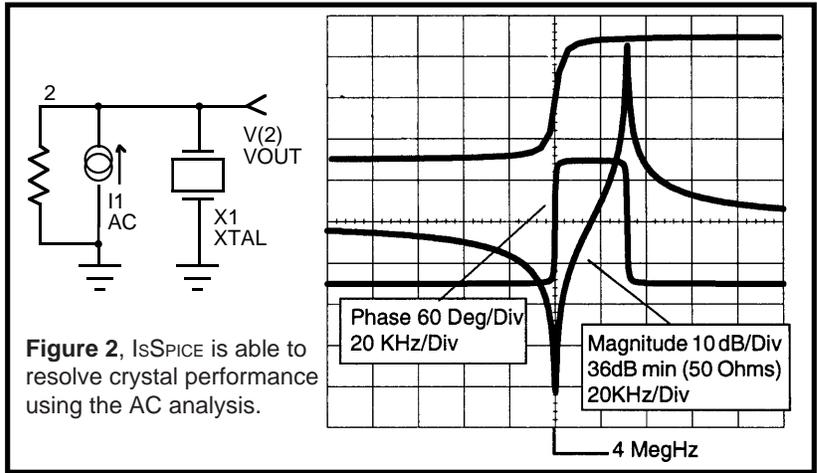
We won't attempt to prove this assertion, however, it can be readily seen to be true when the nonlinearities are very small and the small signal gains are near unity. Moreover, there is a large class of circuit topologies for which phase changes are small as nonlinearities are encountered. For these phase invariant circuits, the uncertainty in predicting oscillation frequency is small and the small signal gain will be automatically reduced as signal strength increases with oscillation taking place at zero phase margin. Phase invariance with amplitude is by no means a universal circuit characteristic; however, circuits that exhibit this behavior will be easier to analyze using linear circuit theory. Figure 1 shows the result of several simulations, illustrating that phase shift is not a strong function of amplitude for this topology.

The phase shift was measured using the last 5 cycles of a transient simulation run for 40 cycles. INTUSCOPE was used to take the FFT and make the rectangular (Real, Imaginary) to polar (Magnitude, Phase) conversion in order to obtain the phase characteristic. Note that the crystal provides a phase gradient on the order of 1 ppm per degree from Figure 2, providing a signal strength dependance of less than 10 parts per billion of frequency per percent change in signal strength when combined with the data in Figure 1.



AC Analysis Resolves Oscillator Performance

Precision oscillators, especially crystal oscillators, have a steep phase vs. frequency gradient. This characteristic must be accurately resolved in order to use the AC analysis features of IsSPICE to perform the simulation. The IsSPICE simulator was updated back in 1988 with version 1.41, increasing the independent variable (Time, Frequency) resolution to 6 digits. Figure 2 shows that the Bode plot of crystal impedance is sufficiently resolved to predict oscillation frequency.



AC Analysis *(continued)*

Since the oscillation frequency will be predicted based on phase shift through the “loop”, it is necessary to somehow identify a control loop, excite it, and make the phase margin and gain margin measurements. The simplest way to excite a loop for an AC analysis is to find a cut point in the loop and insert a series voltage source. This contrasts the more complex real life approach where extra circuitry is required to eliminate loop instabilities so that the closed loop response can be measured. The loop gain is then defined as the output node voltage divided by the input node voltage at each frequency.

When IsSPICE performs an AC analysis, it first computes a DC operating point, then it computes the small signal transfer function at each frequency, substituting the appropriate complex values for inductors, capacitors, and transmission lines. The transfer function will be the same for any non-zero signal strength since the circuit has been linearized about its nonlinear operating point. It is therefore common practice to make AC sources have unity magnitude.

Complex arithmetic performed using INTUSCOPE relies on its unique capability to extend phase past the first cycle (360 degrees) of the complex plane. Phase extension works by assigning the phase of the first frequency point to the first phase plane; subsequent jumps of greater than 180 degrees are assumed to be phase plane jumps, and phase is adjusted by 360 degrees. After acquiring the magnitude in dB and extended phase in degrees for the input node and the output node, both sets of waveforms must be subtracted to get the loop gain and phase.

A Crystal Controlled Oscillator

The following topology fits the criteria for low amplitude distortion because the current available to pump the tank circuit is limited by RE1 when Q2 saturates. The circuit is unstable at its DC bias point as shown in the Bode plot of Figure 3.

The loop gain will drop when Q2 collector current nears cutoff and also as the transistor saturates. The amplitude limit is then equal to $V_{CC} + .6$ as long as the Q of the tank circuit is high enough to be pumped by the available current. Working this backward, the peak-peak current, I_{p-p} , required is:

$$I_{p-p} = 2 * C1 * \frac{dv}{dt_{max}} / Q$$
$$\frac{dv}{dt_{max}} = (V_{CC} + .6) \omega$$

where: Q = tank circuit Q , approximately $= \omega * L1 / R2$
 ω = the resonant frequency in 1 / Sec

The components are defined in Figure 3 and result in:

$$I_{p-p} = 2 * 330P * 5.6 * 25.133Meg / 100.53 = .924 \text{ ma}$$

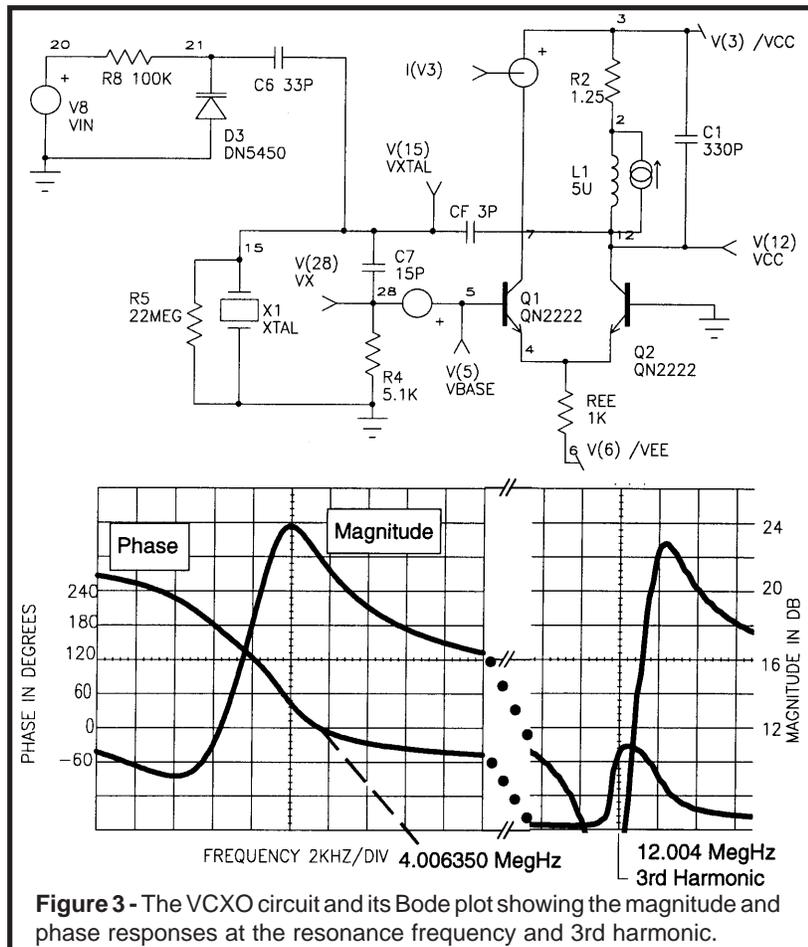
This current is less than the bias current $((V_{EE} - .6)/R_{EE})$ so that the nonlinear limits will govern amplitude.

Since feedback is explicit, it is fairly easy to break the circuit at the base of Q1 in order to find the AC parameters. Several device models must be generated. The basic crystal model comes from our P_{RES}PICE Model Libraries. We will modify the crystal model to include temperature sensitivity by altering its equivalent inductance and capacitance to include a TC parameter (ppm per deg. C). We will also develop temperature dependent capacitor and inductor models. The varactor diode was modeled using the standard I_SPICE diode and the techniques described in our P_{RES}PICE manual. All of these models are shown on pages 16-8 and 16-9. The circuitry will be developed for an oscillator running at the fundamental crystal frequency; however, we will include a 3rd harmonic model in order to make sure the circuit will not oscillate at a crystal overtone.

Simulation Results

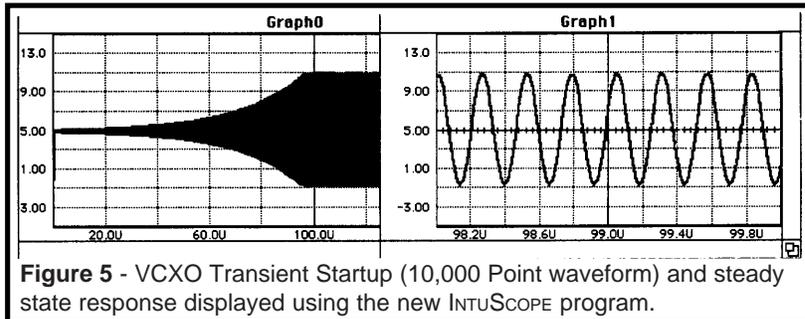
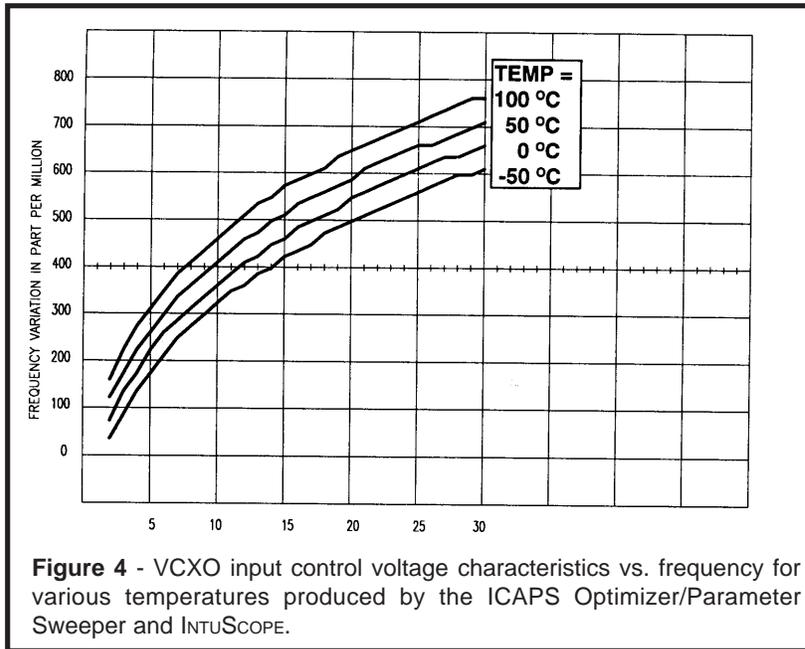
Because of the difficulty encountered with the transient simulation, the design will be performed using AC analysis. When things like stability and control range have been established, the transient simulation will be run to show start up and validate the assumptions that oscillation occurs at the crystal fundamental frequency.

The first simulation performed is an open loop AC analysis. Shown in Figure 3, the predicted oscillation frequency is the frequency where there is no phase shift around the loop. It will be assumed that when the amplitude is increased that loop gain will go down without any phase change. Notice that the third harmonic gain is lower than the fundamental and more importantly that phase does not satisfy the criteria for instability. The initial lagging 180 degree phase shift near the 3rd overtone is due to the 90 degrees from the collector capacitance and an additional 90 degrees from transistor F_t . If you want the circuit to oscillate at the 3rd overtone, then it would be necessary to adjust the tuned circuit in the collector to be resonant at slightly greater than the oscillation frequency in order to shift the phase in the leading direction.



Next, a sweep was run using the ICAPS Parameter Sweeper in order to study the linearity of frequency versus control voltage over temperature (Figure 4). Without the automatic data taking and netlist manipulation features of the ICAPS program, this simulation would have required far more effort and user interaction; in contrast, once setup, the analysis is able to run unattended until completion. Oscillation frequency was recorded for each run as the offset above 4.0 Meg Hz. The control voltage was swept from 2 to 30 Volts for each of 4 temperatures ranging from -50 °C to 100 °C. The crystal temperature coefficient was set to 1 ppm/°C for these sweeps. For these runs, only semiconductor and crystal temperature effects were included. If the TC's of other components are known, they can be accounted for by using a subcircuit expansion similar to that shown for the crystal.

The graph in Figure 5 shows the transient start up of the oscillator



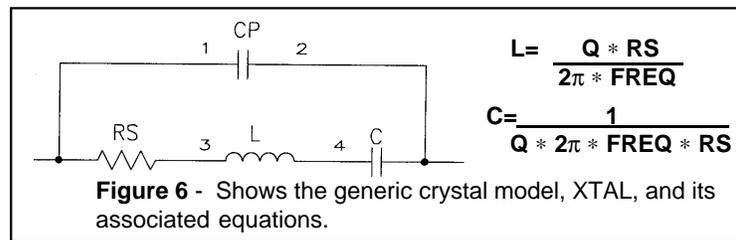
and verifies that oscillation occurs at the crystal fundamental as predicted by the AC analysis.

Conclusions

The capability of IS_{SPICE} to predict oscillator performance with part per million resolution was demonstrated. Techniques were developed that rely on using AC analysis features for predicting precision and Transient simulation for gross performance, such as start up time, harmonic distortion and identification of unwanted oscillation models. The ICAPS Optimizer was shown to be a valuable addition to IS_{SPICE}; providing the capability to automatically acquire data from analyses that required varying input voltage during AC and transient analyses for several values of temperature.

Modeling Crystals

Modeling a crystal is quite simple. Figure 6 shows the generic model topology and the equations for the L and C values. The values for CP and RS must be explicitly supplied. Adding the temperature dependency to the frequency requires multiplying the FREQ value by the expression $(1+(TEMP-27)*TC*1U)$, where TC is in ppm per degrees C. Thus, TC will cause the resonant frequency to increase or decrease with temperature. The 3rd harmonic is produced by adding a shunt branch containing an R, L, and C with the FREQ parameter multiplied by 3. The resistor for the 3rd harmonic is reduced by half in order to provide for the higher Q ($=2*Q$) value seen at the 3rd harmonic.



Modeling Varactor Diodes

Varactors, or voltage variable capacitors, are very similar to most pn junction diodes and can therefore be modeled by using a standard IsSPICE diode model that contains three parameters responsible for governing charge storage behavior. Charge storage in the diode is governed by **CJO**, the zero bias depletion region capacitance, **VJ**, the built-in junction potential, and **M**, the junction capacitance exponential factor. The value of CJO is dependent on many factors, while VJ is usually in the range of 0.2V to 1.0V. M varies from .33 for a linearly graded junction to .5 for a step grading. To calculate the parameter values you will need a graph of Capacitance versus Reverse Voltage (Figure 7). If this data is not available you will not be able to get an accurate model for the varactor. Other diode parameters are usually not necessary. The varactor is usually reversed biased, so there is no need for the forward biased parameters N, IS, RS. The device is also not switched on and off, so the transit time parameter TT is not critical. Also, the reverse voltage applied to the device should not be enough to break it down; therefore, the reverse parameters, BV, and IBV are not needed.

There are a number of parasitics that can be added to the basic varactor model. They include the case capacitance, usually about .2-.3 PF, the board capacitance which depends on several factors, and the lead inductance, typically 2-10nH.

Calculating The Charge Storage Parameters

- 1 M is the slope of the log(C) vs. log(Vr) curve and should be taken at the high reverse voltage end (assuming VJ is very small (0) compared to VD) using:

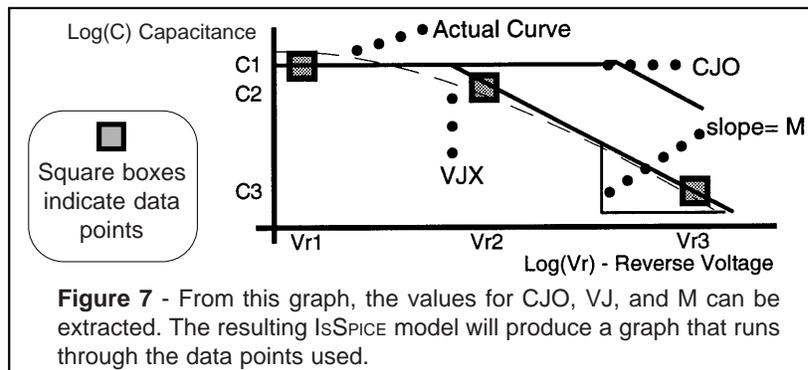
$$M = \frac{\ln(C3) - \ln(C2)}{\ln(Vr3+VJ) - \ln(Vr2+VJ)}$$

- 2 Compute VJ using two data points, one near Vr = 0 using:

$$VJ = \frac{[Vr2 * (\frac{C1}{C2})^{\frac{-1}{M}} - Vr1]}{[1 - (\frac{C1}{C2})^{\frac{-1}{M}}]}$$

- 3 Finally, compute CJO at one data point close to Vr = 0.

$$CJO = \frac{C1}{[(1 + Vr1/VJ)^{-M}]}$$



Model Listings For The VCXO Circuit

The parameters that define the crystal model are CP=parallel capacitance in farads, RS=series resistance in ohms, TEMP=temperature of the crystal in degrees C, FREQ=frequency of oscillation in Hz., Q=Q of the crystal, and TC=temperature coefficient in ppm per degrees C.

```
*****
* Model for a Crystal with Temperature Coefficient dependence and 3rd Harmonic
.SUBCKT XTAL 1 2 {TEMP=27 TC=1}
CP 1 2 {CP}
RS 1 3 {RS}
L 3 4 {Q*RS/(6.283185*FREQ*(1+(TEMP-27)*TC*1U))}
C 4 2 {1/(Q*6.283185*FREQ*(1+(TEMP-27)*TC*1U)*RS)}
RS3RD 1 30 {RS/2}
L3RD 30 40 {Q*RS/(6.283185*3*FREQ*(1+(TEMP-27)*TC*1U))}
C3RD 40 2 {1/(Q*6.283185*3*FREQ*(1+(TEMP-27)*TC*1U)*RS)}
.ENDS
*****
.MODEL DN5445 D(CJO=32P M=.42 VJ=.66)
*****
.MODEL DN5450 D(CJO=80P M=.42 VJ=.66)
*****
.MODEL DN5456 D(CJO=280P M=.42 VJ=.66)
```

Varactor Diode Models

Intusoft Newsletter Handbook, *Continued from page 16-1*

here's a bit of important news. A compilation of all the newsletters published by **intusoft** (from 6/86 to 1/90, 16 in all) is now available. The book, entitled "SPICE Applications Handbook, Vol.1" contains all the application notes, models, circuits information, and simulation techniques from all the past newsletters. If you are not familiar with the power of SPICE, this book will introduce you to its range of applicability. If you need modeling or applications assistance, the handbook will provide you with invaluable tips and techniques for taming SPICE's capabilities. In addition, models, circuits, and SPICENET symbols contained in the newsletters will be available on floppy disk, making access to the information all the more easier. A special introductory offer is available till 4/90.

Intusoft's Current Feedback Op-amp Models

In the September issue of the **intusoft** newsletter, models for current feedback opamps were described in detail. The models, as usual, are generic in nature, and therefore have applicability to op-amps from various vendors. The parameters needed to complete the model description are RF in ohms (feedback resistor), DVDT in Volts/sec (slew rate), and FC in Hz (-3db bandwidth). Unfortunately, the parameter units were not explicitly stated. We hope that this has not caused any confusion.

More Op-amp Models

Through some negotiations, **intusoft** has obtained the right to distribute op-amp models from various manufacturers. Op-amps from HARRIS, PMI, and COMLINEAR are now available on floppy disk directly from **intusoft**. The models are included in two forms, as standard separated ascii files and as an **intusoft** compatible ascii library (.LIB) file. SPICENET symbols for each op-amp are also included. To obtain the models, contact **intusoft** directly. Please include a handling charge of \$20 per library.

Intusoft encourages users to utilize the generic op-amp models available in PRESPICE. The OPAMP (bipolar) and FET (JFET) models can be used to model thousands of op-amps for AC, DC, and nonlinear Transient analyses and are far superior to most models from other sources. Some op-amp and CAE software vendors still use an outdated or modified version of the BOYLE op-amp model. Intusoft's generic op-amp models use a superior topology and can be used to create new models easier than other parameter estimation type programs. If anyone is interested, the **intusoft** op-amp models are well documented in our new "SPICE Applications Handbook" and PRESPICE manual.