

Transceiver Block Simulation with SmartSpice RF

Introduction

Microwave circuits have three main parameters that establish its performance: dynamic range, sensitivity, and selectivity. The first two are dictated by the noise figure and linearity of the devices. The selectivity depends mostly on the noise contribution, either from a signal generator or signal processor. Noise contribution is commonly described in term of phase noise, which is mainly due to the semiconductor materials, device selection criteria, and circuit design techniques. Therefore, for the highly non-linear type of circuit, phase noise is an important characteristic and its accurate prediction is critical.

SmartSpice RF Features

Phase noise simulation of the autonomous circuits is a two-stage process:

- 1) Finding the Frequency of oscillation, Amplitude (Power), and Shape of the produced signal (carrier) by providing Periodic Steady-State Analysis (PSS);
- 2) Simulating phase noise due the device noise contribution by providing Noise Analysis over the given frequency sideband.

SmartSpice RF has implemented two methods of PSS analysis, which allow the effective handling of different topologies of autonomous circuits, such as a high-Q LC (crystal) tank oscillators, and ring oscillators:

- Frequency-domain Harmonic Balance Optimization method (.HOSCIL analysis statement);
- Time-domain Shooting method can be used with any of the one-tone analyses (.HARM, .HNOISE, etc. analyses statements).

The simulation examples are applied to the 3 GHz range voltage-controlled ring oscillator, which consists of 50 MOS devices and is a part of PLL circuit design. The project uses the TSMC 90nm CMOS logic salicide (1P9M 1.0V/3.3V) process that provides models of MOSFET, bipolar, diode, interconnect, and passive components.

Periodic Steady-State Analysis

To setup PSS analysis for batchprint mode the following Dot statements have to be included into the input deck:

a) By Harmonic Balance method:

```
.hoscil Probe(xi0feedback2) v(FOUT)
+ Fund_Osc=3GHz nharm=20
+ Newton_Accuracy=highest
+ UseTIG=3 TIGtstop=15n TIGtmax=0.1n
+ oscillator=RING
```

b) By Shooting method:

```
.harm v(FOUT)
+ Fund=3GHz nharm=20
+ method=shooting tstab=15n tstep=0.1n
+ oscillator=RING
```

Where: FOUT is the output port name;

Probe(xi0.feedback2) is the probe, connected to the given node;

Fund_Osc, Fund <=3GHz> is the estimated oscillation frequency.

To set up PSS analysis in GIU mode, the dialog windows, shown in Figures 1 – 3, can be used.

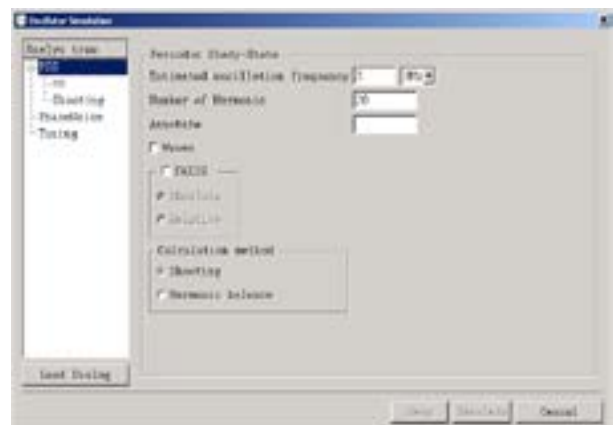


Figure 1. PSS analysis dialog window.

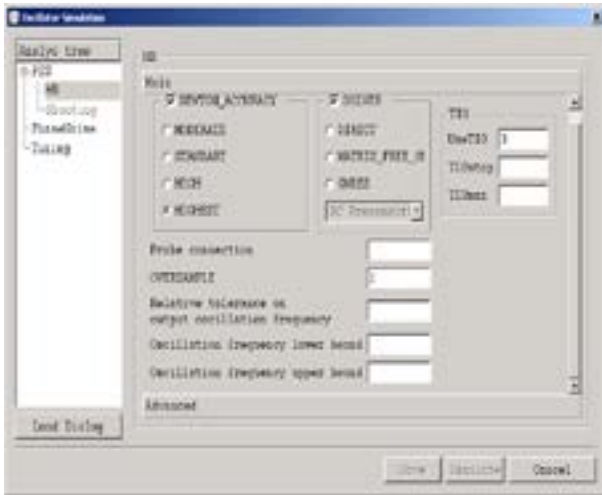


Figure 2. .HOSCIL analysis dialog window.

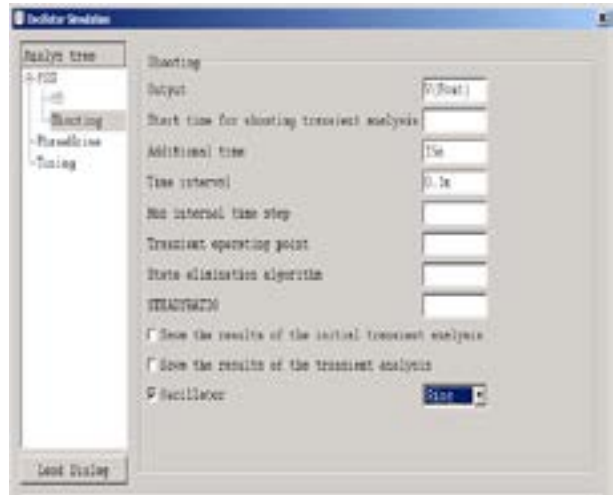


Figure 3. Shooting method dialog window.

The results of simulation are:

HOSCIL analysis: Oscillation Frequency = 3051397515.80643 Hz

Carrier: Power Psig = 2.221370e-01 Wt

Effective amplitude Ac = 6.665387e-01 V

Oscillator Analysis by Shooting method:

Frequency of Oscillation Fund = 3.035513e+09 Hz

Carrier: Power Psig = 2.215548e-01 Wt

Effective amplitude Ac = 6.656648e-01 V

The output waveform is shown in Figure 4. Normalized output waveform and Impulse Sensitivity Function plot are shown in Figure 5.

Figure 5 indicates that the output noise level will increase because of the asymmetric waveform.

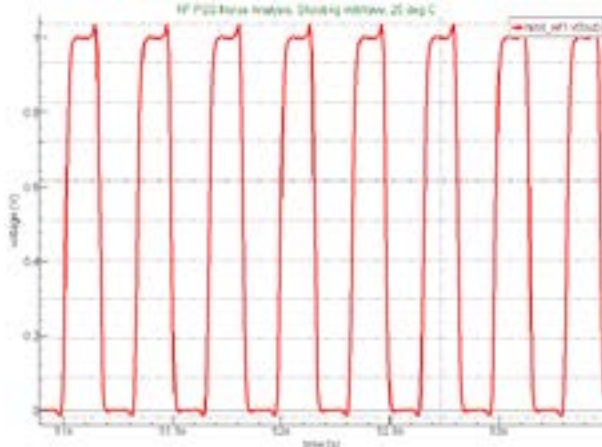


Figure 4. Waveform at the node FOUT.

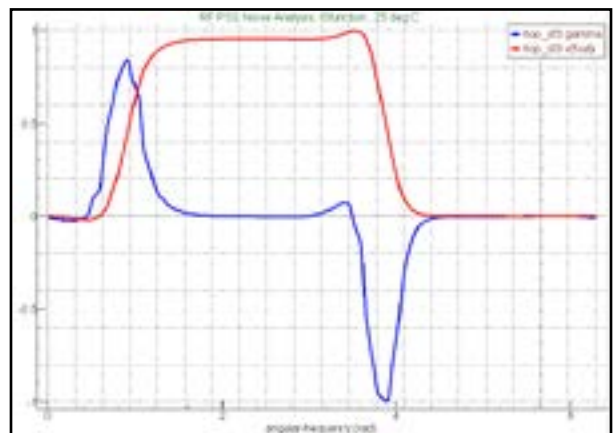


Figure 5. Normalized waveform and impulse sensitivity function at the node FOUT.

PSS Analyses Troubleshooting

SmartSpice RF employs advanced methods and algorithms to guarantee accurate simulation results, but at the same time it offers the definition of additional accuracy parameters to resolve non-convergence issues. They are:

1. "Newton_accuracy" = <moderate/standard/high/highest>
 "Moderate" better for typical circuits; "Highest" for sensitive analog circuits.
2. "MAXITER" defines the maximum iterations number for shooting method.
3. "abstol" "vntol" "reltol"

These help to control tolerances of the node currents and voltages or its relative value. The higher values usually provide the higher accuracy.

If any non-convergence issues occur during simulation, SmartSpice RF outputs a list of suggestions which could help users to resolve them. In any case, detailed analysis of the warning or error messages is sometimes the key component to start with.

Phase Noise Simulation

To set up Phase noise simulation, it is necessary to define frequency-domain simulation parameters in the Dot statement (sideband frequency sweep, axis types, level of output results, etc.). For example:

```
.hnoise v(FOUT)
+ DEC_ENG 10 10kHz 1GHz
+ Fund=3GHz nharm=20
+ method=shooting tstep=0.1n tstab=75n
+ Devpts = 1
+ DevCon = 5
+ PhaseNoise
+ oscillator=RING
```

or use Phasenoise dialog window shown in Figure 6.

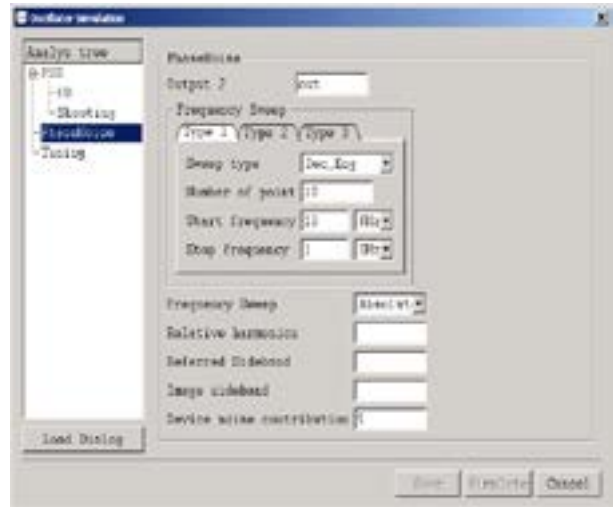


Figure 6: Phasenoise dialog window.

All simulation setups could be saved for the further usage – download, correction, or just to repeat simulation. The example of saved statement is shown in Figure 7.

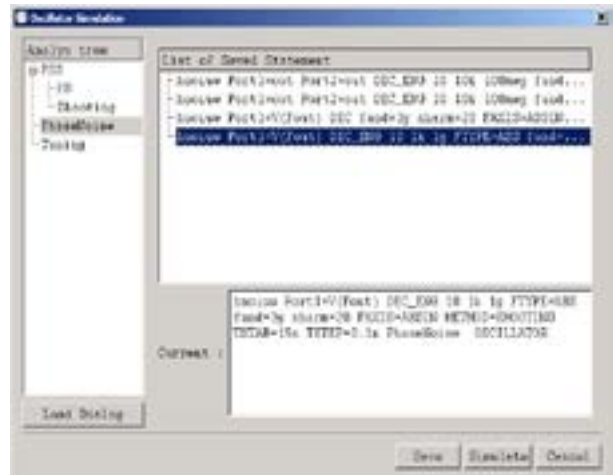


Figure 7. Saved oscillator simulation statement

SmartSpice RF produces device noise contribution for the Output and its integrated noise in the following form:

Spot Noise Summary (V²/Hz) at 1 MHz Sorted by Noise Contributors

Device	Noise Contribution	% Of Total
m.xi0.xi98.xi4.mm0	8.29033e-17	79.88
m.xi0.xi98.xi4.mm1	1.59017e-17	15.32
m.xi0.xi98.xi13.mmn1	1.78661e-18	1.72
Total Output Noise = 1.03781e-16 V ² /Hz		

Integrated Output Noise (V²) Sorted by Noise Contributors

Device	Noise Contribution	% Of Total
m.xi0.xi98.xi4.mm0	3.26900e-09	58.256
m.xi0.xi98.xi4.mm1	2.15835e-09	38.463
m.xi0.xi98.xi13.mmn1	5.00422e-11	0.892
Total Integrated Output Noise = 5.61149e-09 V ²		

The resulting Phase noise plot shown in Figure 8 is very close to competing benchmark results (Figure 9).

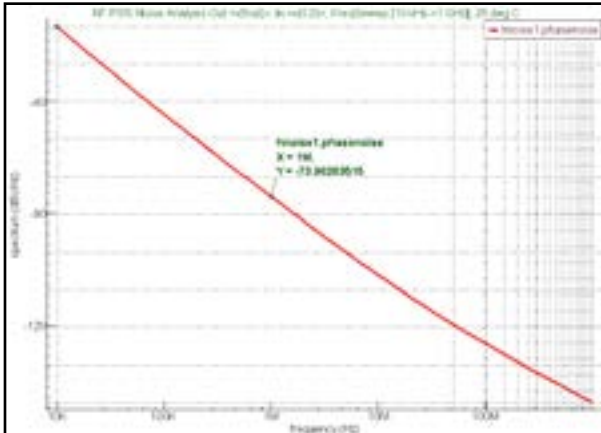


Figure 8. SmartSpice RF phase noise plot.

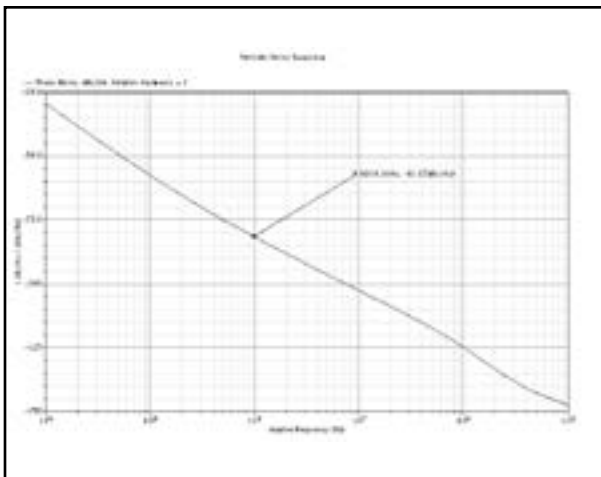


Figure 9. Phase noise plot from competitor benchmark.

Conclusion

This application note has described SmartSpice RF's capabilities in providing oscillator simulation. The simulation results show that SmartSpice RF has good convergence with high accuracy, and faster run-times than other tools. Simulated phase noise at a given offset is very close to the measurement result. Output vectors and plots can be used by circuit designers to improve performance up to desired parameters. SmartSpice RF can be widely used for simulation of autonomous circuits.