

The normal procedure for simulating oscillators are as follows:

1. Run a transient analysis to make sure that the circuit oscillates! Once it oscillates, check if the estimated operating frequency is as desired (this is approximately done by performing a Fast Fourier Transform (FFT) on the output waveform using the calculator in Cadence).
2. Run a Periodic Steady State (PSS) analysis, this analysis will give you the correct operating frequency and several other important performance Parameters, such as the output power and the harmonic spectrum.
3. Run a Periodic Noise (PNoise) analysis in order to measure the phase noise performance of the circuit.

Normally the steps above are repeated several times and changes are made in the circuit to optimize the parameters. The following descriptions will give you how to perform PSS and Phase noise analysis

For PSS analysis follow the instructions below.

1. In the simulation window, choose Analyses - Choose... and click on pss for the analysis. The form changes to display options needed for PSS analysis.
2. In the field next to the Beat Frequency and Beat Period buttons, type your best estimate of the oscillation frequency. Normally it is easier for the simulator to converge if you give a higher estimation, for instance if the oscillation frequency is 1 GHz you may type 1.1 GHz.
Make sure that the Auto Calculate button is *not* highlighted.
3. In the Output harmonics cyclic field choose Number of harmonics and type in a reasonable number, such as 20.
4. Highlight conservative for the Accuracy Default (errpreset).
5. In order to make it easier for the simulator to converge we may give a optional stabilization interval of the length tstab, this is the time when you estimate that the output of the oscillator has reached steady state. Normally this time is extracted from the Transient Analysis made earlier.
6. Highlight Oscillator.
7. Click on the Oscillator node Select button. Then click on the appropriate wire in the schematic window to choose the oscillator node.
8. Set the Reference node field to /gnd.
9. Click on the Options button and specify the step length for the simulator in the Maxstep field (set this value to 10 ps).

For PNoise analysis follow the instructions below.

1. At the top of the Choosing Analysis form, click on pnoise.
2. In the Sweeptype cyclic field, choose Relative. The choice specifies that the Frequency Sweep Range (Hz) values you choose represent frequency values away from the fundamental frequency. For example, if you specify 100k, you choose a value 100k away from the fundamental frequency.
3. Choose Start-Stop for the type of Frequency Sweep Range (Hz). Type in reasonable values, such as 10 and 1G as the Start and Stop values.
4. In the Sweep Type cyclic field, choose Automatic.
5. In the Sidebands cyclic field, choose Maximum sideband and type in 0 in the Sidebands field.
6. In the Output cyclic field, choose voltage.
7. Click on the Positive Output Node Select button and click on the output wire of the oscillator.
8. Set the Negative Output Node field to /gnd.
9. Press Ok and run the simulations.

Now the configuration of the PSS and PNoise are done. When the simulations are completed you may find the results by clicking on Results -> Direct Plot -> PSS....