

# Simulations on VCO

---

Using Cadence SpectreRF Simulator

PSS, Pnoise, ... analyses

---

# OSCILLATORS

Phase Noise analysis using PSS and Pnoise

# Autonomous PSS Analysis

---

- ◆ PSS analysis lets you simulate both **driven** and **autonomous** circuits
- ◆ Driven circuits; amplifiers, filters, or mixers,
  - require a **stimulus** to create a time-varying **response**
  - Analysis period is a multiple of drive signal period
- ◆ Autonomous circuits: oscillators,
  - have time-varying responses even though the circuits are time-invariant.
  - No analysis period due to unknown oscillation period
  - specify an **estimate** of the oscillation period
  - PSS analysis uses this estimate to compute the precise period
    - ◆ PSS analysis monitors the voltage of the specified nodes and refines its estimate of the period

# Phases of Autonomous PSS Analysis

---

- ◆ A transient analysis phase to initialize the circuit
  - A beginning interval
  - A second, optional stabilization interval of length *tstab*
  - A final interval (that is four times the estimated oscillation period).
  - During the final interval, the PSS analysis monitors the waveforms in the circuit and improves the estimate of the oscillation period.
- ◆ A shooting phase to compute the periodic steady state solution.
  - During this phase, the circuit is simulated repeatedly over one period.

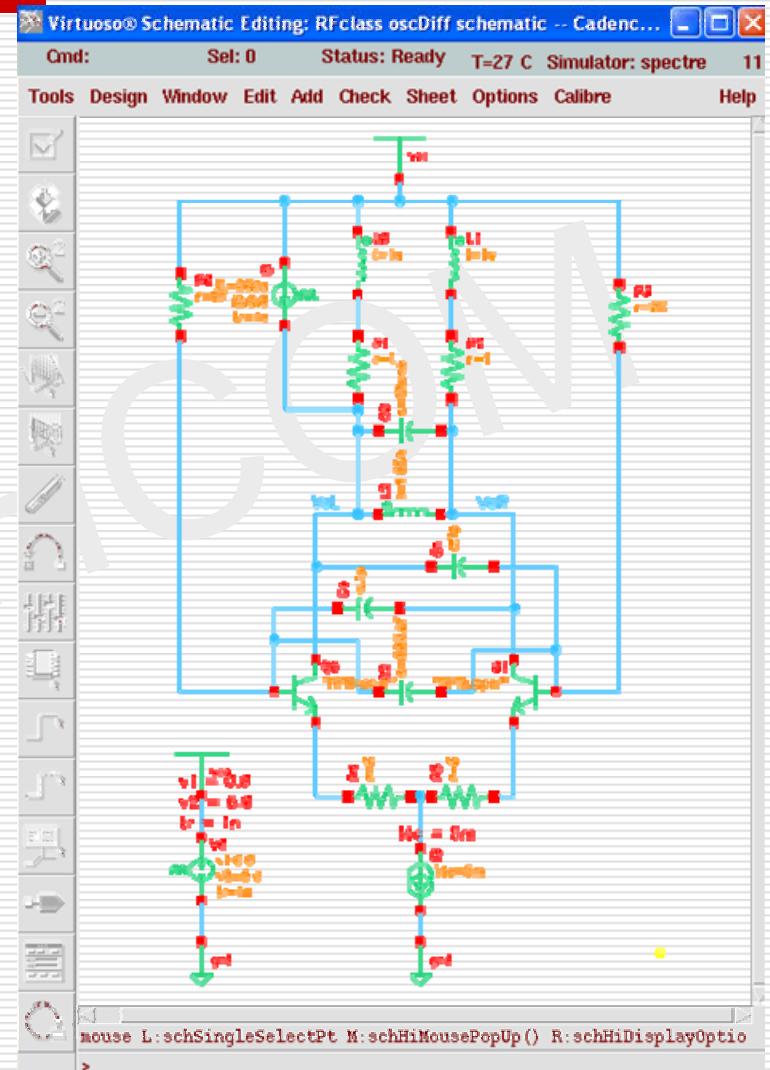
# Starting and Stabilizing the Oscillator

---

- ◆ To simulate an oscillator using PSS analysis, you must first start it.
- ◆ Start an oscillator by supplying either
  - A brief impulse stimulus
  - A set of initial conditions for the components of the oscillator's resonator
- ◆ Allow the oscillator to run long enough to stabilize before you start the steady state solution.
- ◆ Adjust the *tstab parameter to supply the additional stabilization period.*

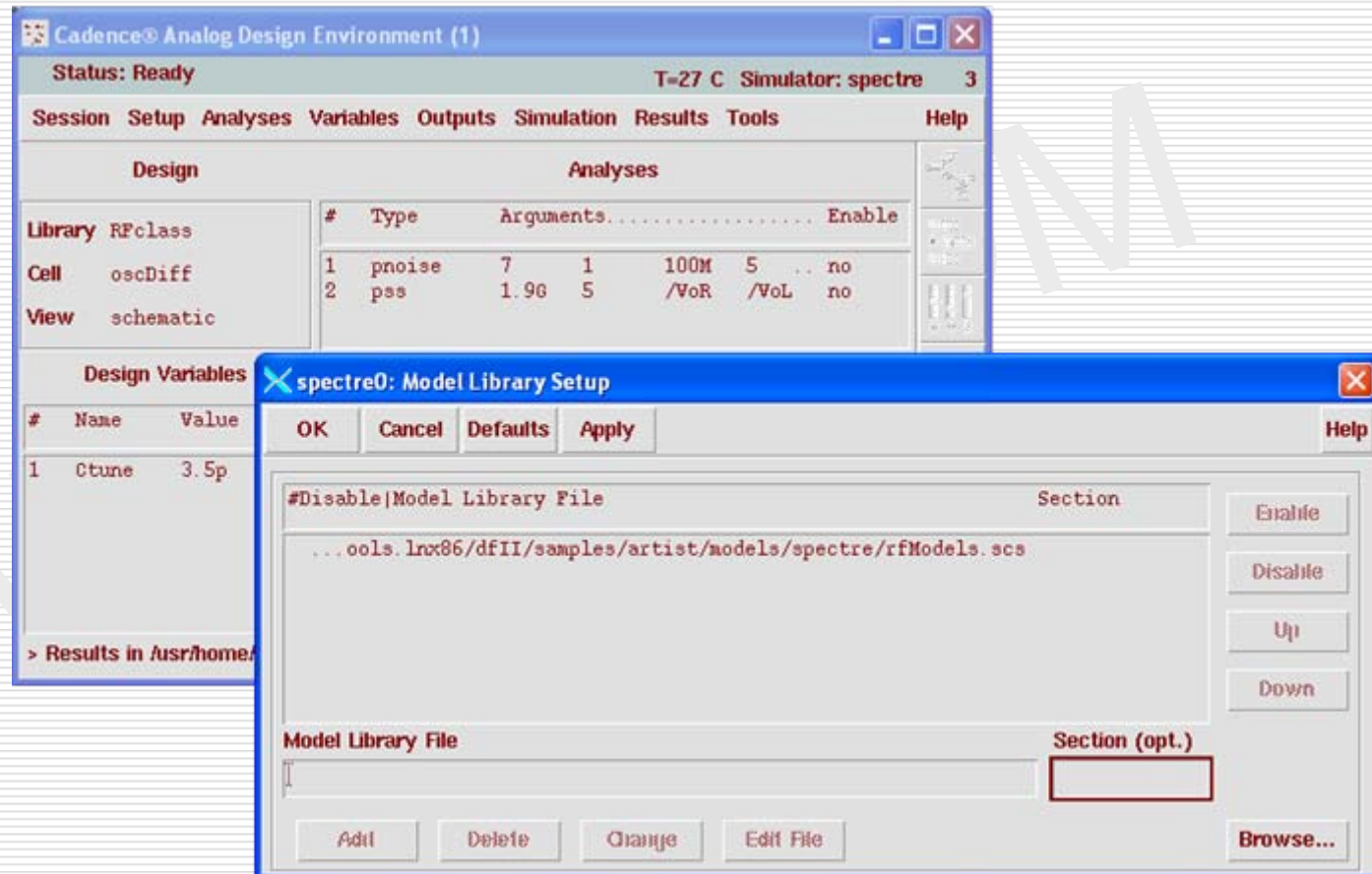
# Differential Bipolar Oscillator

This example is from  
*rfExamples* library of Cadence



# Analog Environment

- ◆ Set model library (rfModels.scs)



# Setting Up PSS Analysis

- ◆ Specify an estimate frequency
- ◆ Specify an additional time for stabilization, *tstab* (optional for more accuracy)
- ◆ Highlight *Oscillator*
- ◆ Select the oscillator nodes on schematic

The screenshot shows the 'Choosing Analyses' dialog box in the Cadence Analog Design Environment. The 'Analysis' section has several checkboxes, with 'pss' selected. The 'Periodic Steady State Analysis' section contains the following settings:

- Fundamental Tones:** A table with columns #, Name, Expr, Value, Signal, and SrcId. Below the table are buttons for 'Clear/Add', 'Delete', and 'Update From Schematic'. A text input field shows '1.9G' and is circled in red.
- Beat Frequency:** A checkbox that is checked, with a text input field showing '1.9G' and is circled in red.
- Beat Period:** A checkbox that is unchecked.
- Auto Calculate:** An unchecked checkbox.
- Output harmonics:** A section with a 'Number of harmonics' input field set to '10'.
- Accuracy Defaults (empreset):** Three radio buttons: 'conservative' (unchecked), 'moderate' (checked), and 'liberal' (unchecked).
- Additional Time for Stabilization (tstab):** A text input field showing '100.5n' and is circled in red.
- Save Initial Transient Results (saveinit):** Two radio buttons: 'no' (checked) and 'yes' (unchecked).
- Oscillator:** A checked radio button, circled in red.
- Oscillator node:** A text input field showing '/VoR' and is circled in red.
- Reference node:** A text input field showing '/VoL' and is circled in red.
- Sweep:** An unchecked checkbox.
- Enabled:** A checked radio button.
- Options...** A button.

# Setting Up Pnoise Analysis

- ◆ Choose relative sweeptype and harmonic 1
  - Shows frequency values relative to the fundamental frequency.
- ◆ *Maximum sideband = 7*
  - Increase the *Maximum sideband value until the output noise stops changing*
  - *Maximum sideband must be at least 1 to see any flicker noise up conversion.*
- ◆ Output: voltage
- ◆ Input: none

Choosing Analyses -- Cadence® Analog Design Environment...

OK Cancel Defaults Apply Help

☐ pz ☒ pnoise ☐ envlp ☐ pss  
☐ pac ☐ qpac ☐ pxf ☐ psp  
☐ qpss ☐ qpnoise ☐ qpxf  
☐ qpsp

Periodic Noise Analysis

PSS Beat Frequency (Hz)

Sweeptype relative Relative Harmonic 1

Frequency Sweep Range (Hz)

Start-Stop  Start 1 Stop 100M

Sweep Type Logarithmic ☒ Points Per Decade 5  
☐ Number of Steps

Add Specific Points ☐

Sidebands

Maximum sideband 7

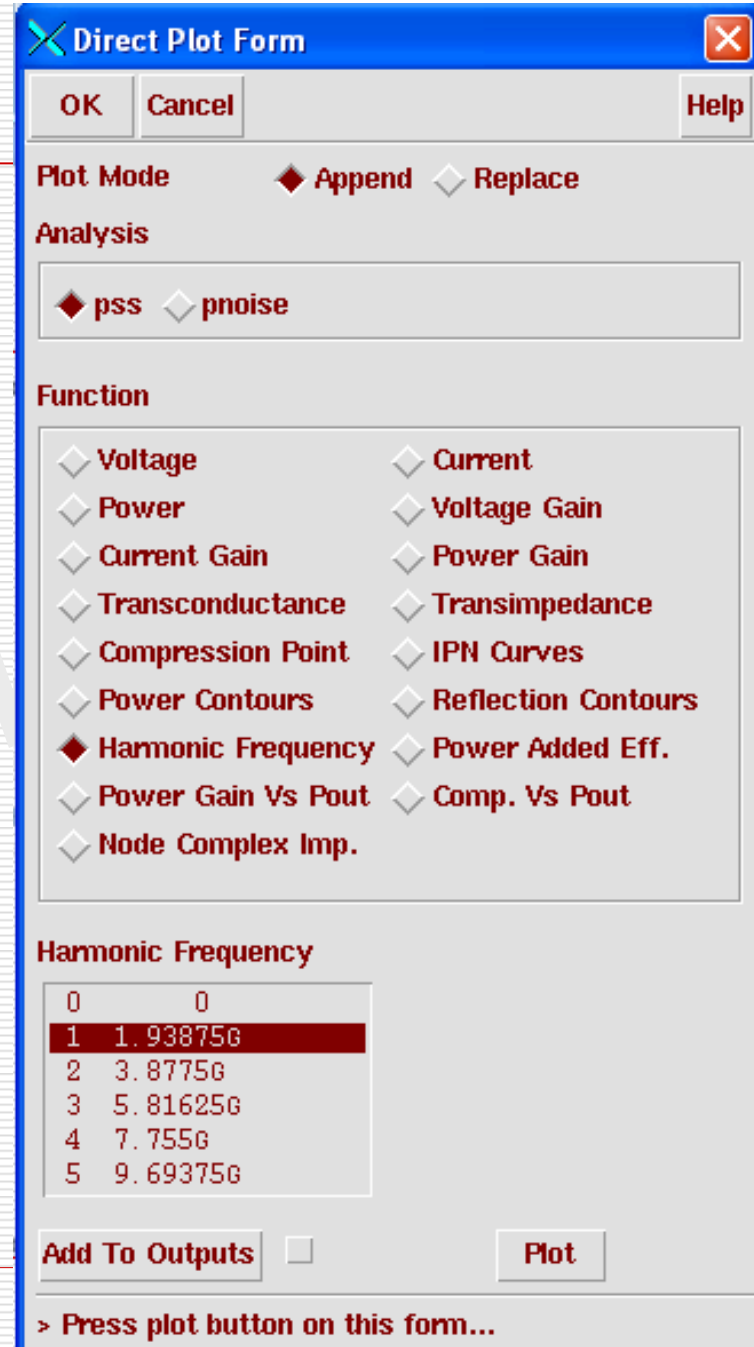
Output voltage Positive Output Node /VoR1 Select  
Negative Output Node /VoL1 Select

Input Source none

Noise Type sources

# Simulation Results

- ◆ After running the simulation
- ◆ You can see the fundamental frequency
  - Results → direct plot → main form
  - → pss, Harmonic Frequency
- ◆ In this example:  
 **$f = 1.93875 \text{ GHz}$**



The image shows a 'Direct Plot Form' dialog box with a blue title bar and standard Windows window controls. It contains several sections for configuring a simulation plot. The 'Plot Mode' section has 'Append' selected. The 'Analysis' section has 'pss' selected. The 'Function' section has a list of 16 options, with 'Harmonic Frequency' selected. The 'Harmonic Frequency' section contains a table with 6 rows and 2 columns, where the first row is highlighted. At the bottom, there are 'Add To Outputs' and 'Plot' buttons, and a note to press the plot button.

**Direct Plot Form**

OK Cancel Help

Plot Mode ☒ Append ☐ Replace

Analysis

☒ pss ☐ pnoise

Function

<input type="radio"/> Voltage	<input type="radio"/> Current
<input type="radio"/> Power	<input type="radio"/> Voltage Gain
<input type="radio"/> Current Gain	<input type="radio"/> Power Gain
<input type="radio"/> Transconductance	<input type="radio"/> Transimpedance
<input type="radio"/> Compression Point	<input type="radio"/> IPN Curves
<input type="radio"/> Power Contours	<input type="radio"/> Reflection Contours
<input checked="" type="radio"/> Harmonic Frequency	<input type="radio"/> Power Added Eff.
<input type="radio"/> Power Gain Vs Pout	<input type="radio"/> Comp. Vs Pout
<input type="radio"/> Node Complex Imp.	

Harmonic Frequency

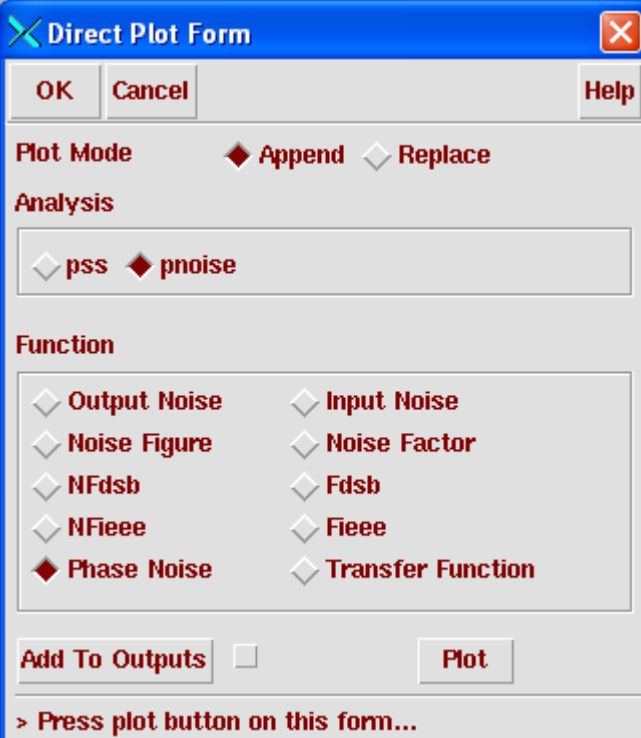
0	0
1	1.93875G
2	3.8775G
3	5.81625G
4	7.755G
5	9.69375G

Add To Outputs ☐ Plot

> Press plot button on this form...

# Phase Noise & Output Noise

- ◆ You can plot Phase Noise & Output Noise (dB) on the same graph
  - & see the linear relationship between them.
  - The output noise is scaled by the carrier amplitude to produce the phase noise value.
- ◆ The phase noise is:
  - -80.5 dBc/Hz @ 10kHz
  - -107.2 dBc/Hz @ 100kHz
  - -129.1 dBc/Hz @ 1MHz



The image shows a 'Direct Plot Form' dialog box with a blue title bar and standard Windows window controls. It contains several sections: 'Plot Mode' with 'Append' (selected) and 'Replace' options; 'Analysis' with 'pss' and 'pnoise' (selected) options; 'Function' with a grid of checkboxes including 'Output Noise', 'Input Noise', 'Noise Figure', 'Noise Factor', 'NFdsb', 'Fdsb', 'NFeee', 'Feee', 'Phase Noise' (selected), and 'Transfer Function'; and 'Add To Outputs' with an unchecked checkbox. At the bottom, there are 'OK', 'Cancel', and 'Help' buttons, and a 'Plot' button. A status bar at the very bottom says '> Press plot button on this form...'.

Direct Plot Form

OK Cancel Help

Plot Mode ☒ Append ☐ Replace

Analysis

☐ pss ☒ pnoise

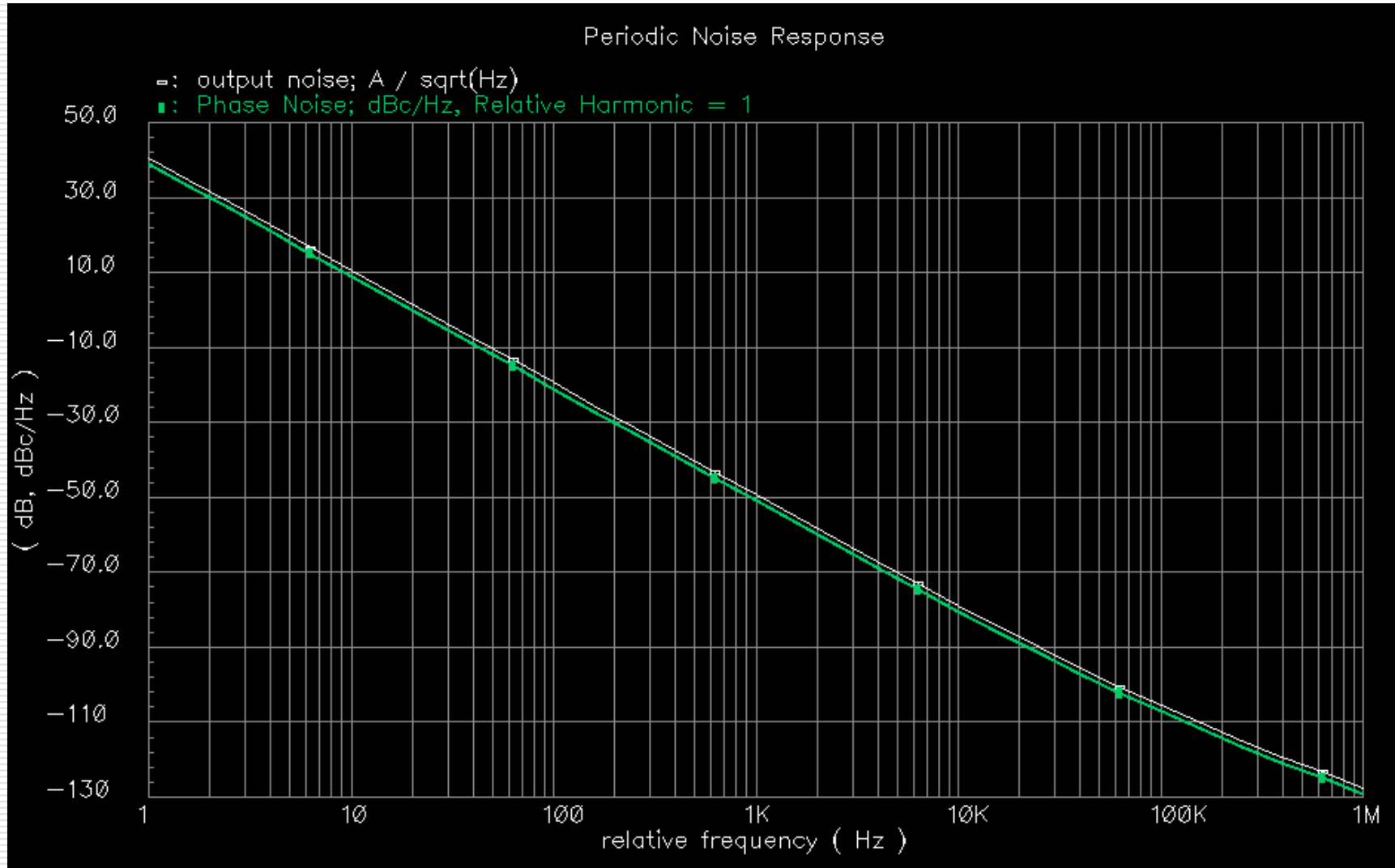
Function

<input type="radio"/> Output Noise	<input type="radio"/> Input Noise
<input type="radio"/> Noise Figure	<input type="radio"/> Noise Factor
<input type="radio"/> NFdsb	<input type="radio"/> Fdsb
<input type="radio"/> NFeee	<input type="radio"/> Feee
<input checked="" type="radio"/> Phase Noise	<input type="radio"/> Transfer Function

Add To Outputs ☐ Plot

> Press plot button on this form...

# Phase Noise & Output Noise

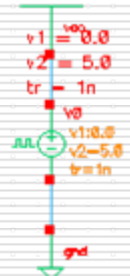


# Convergence Issue

---

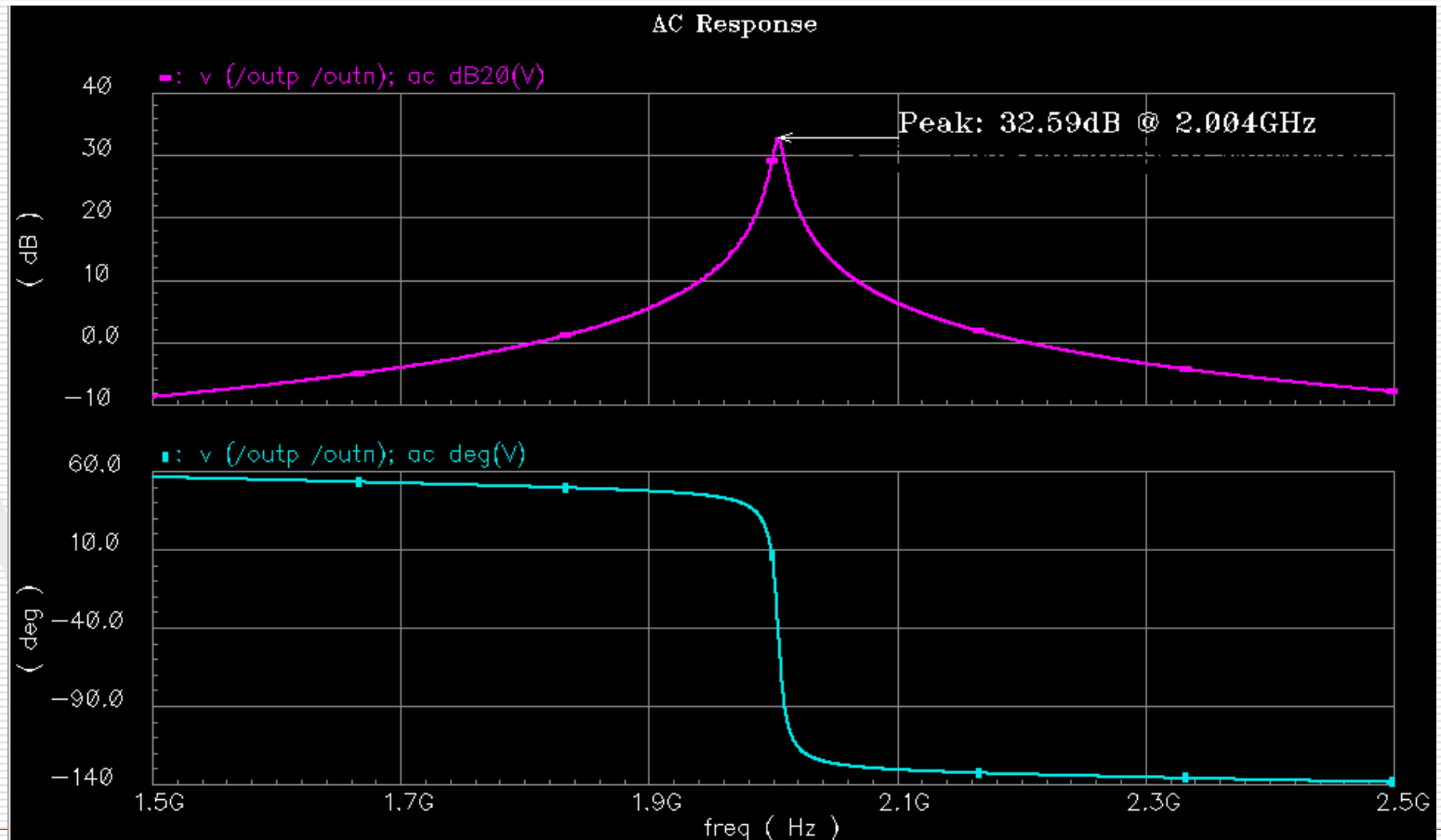
- ◆ Increase the value of the tstab parameter.
- ◆ Be sure that you successfully started the oscillator.
- ◆ Improve your estimate of the period.
- ◆ Increase the value of the maxperiods parameter to increase the maximum number of iterations
- ◆ Tighten the normal simulation tolerances by changing the values of the maxstep, reltol, lteratio, and errpreset parameters. Avoid setting *errpreset to liberal*.

\_\_\_\_\_



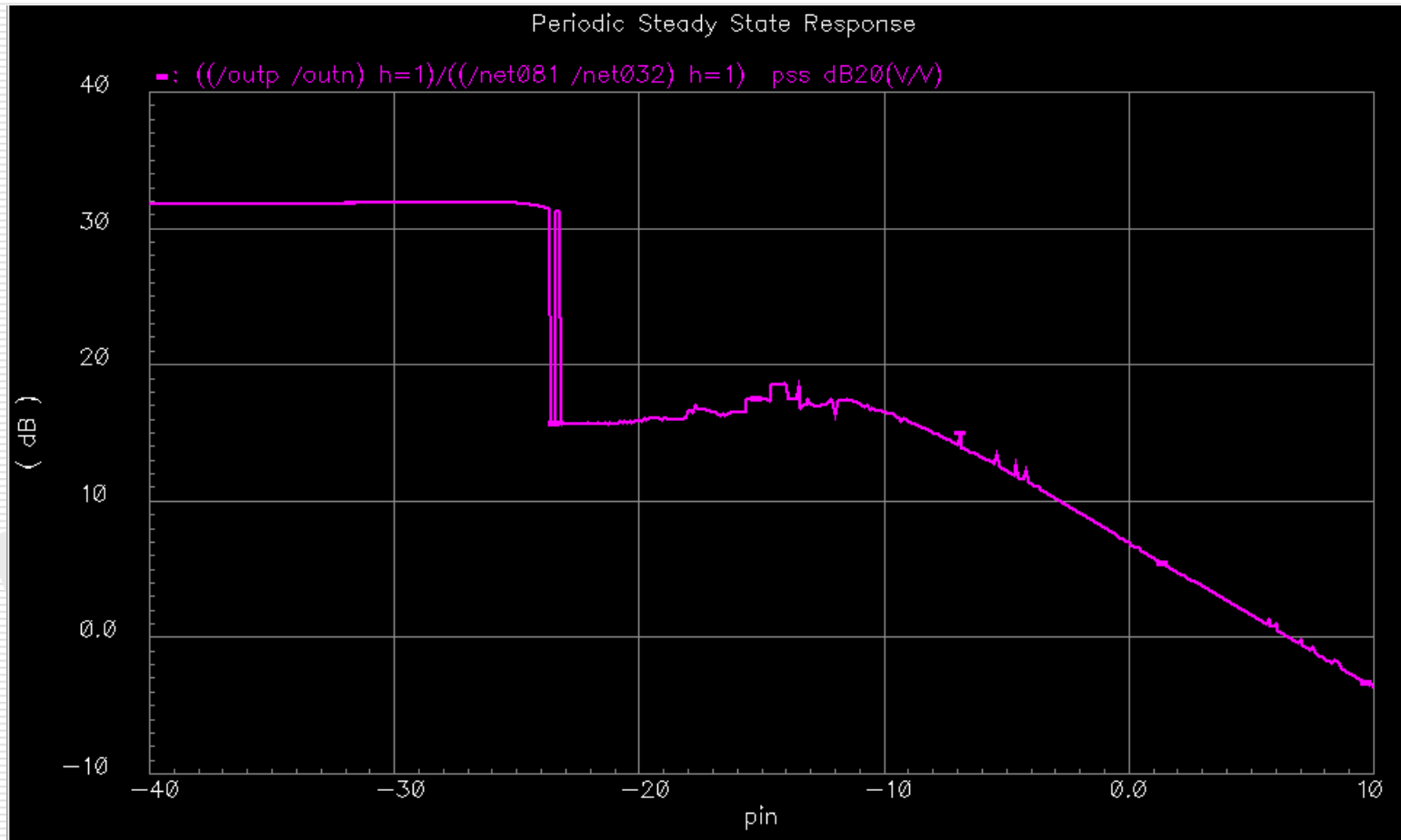
# Loop Gain with AC Analysis

- ◆ Peak freq is at ~2G (higher than oscillation freq)
- capacitive load is lower because the loop is opened



# Loop gain vs. Amplitude (PSS)

- ◆ Loop gain decreases with higher amplitudes



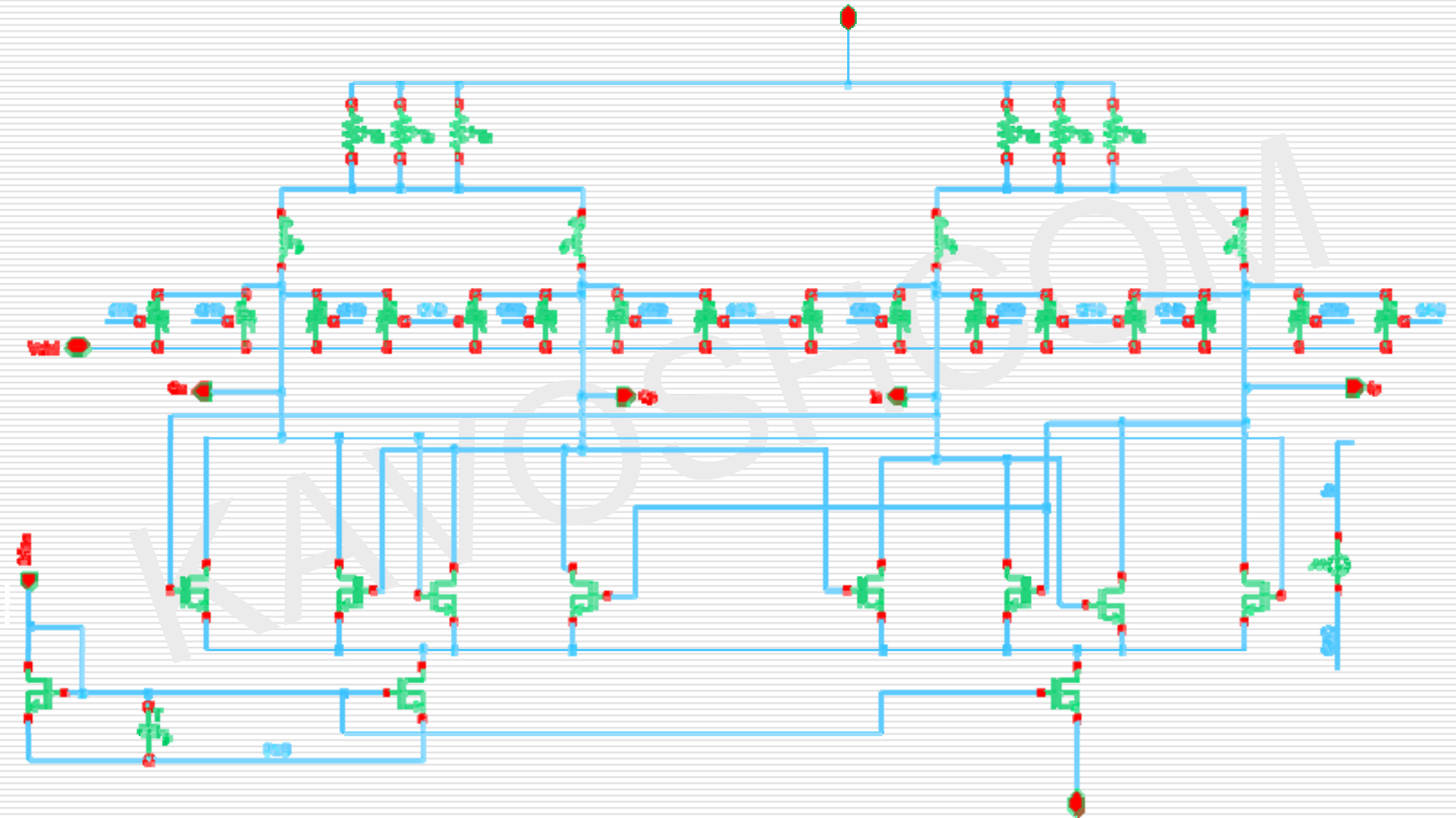
---

# VOLTAGE CONTROLLED OSCILLATORS

Time-domain response using *Tran* or *PSS* analyses  
Tuning range using *PSS* analysis  
Phase Noise using *PSS* and *Pnoise* analyses

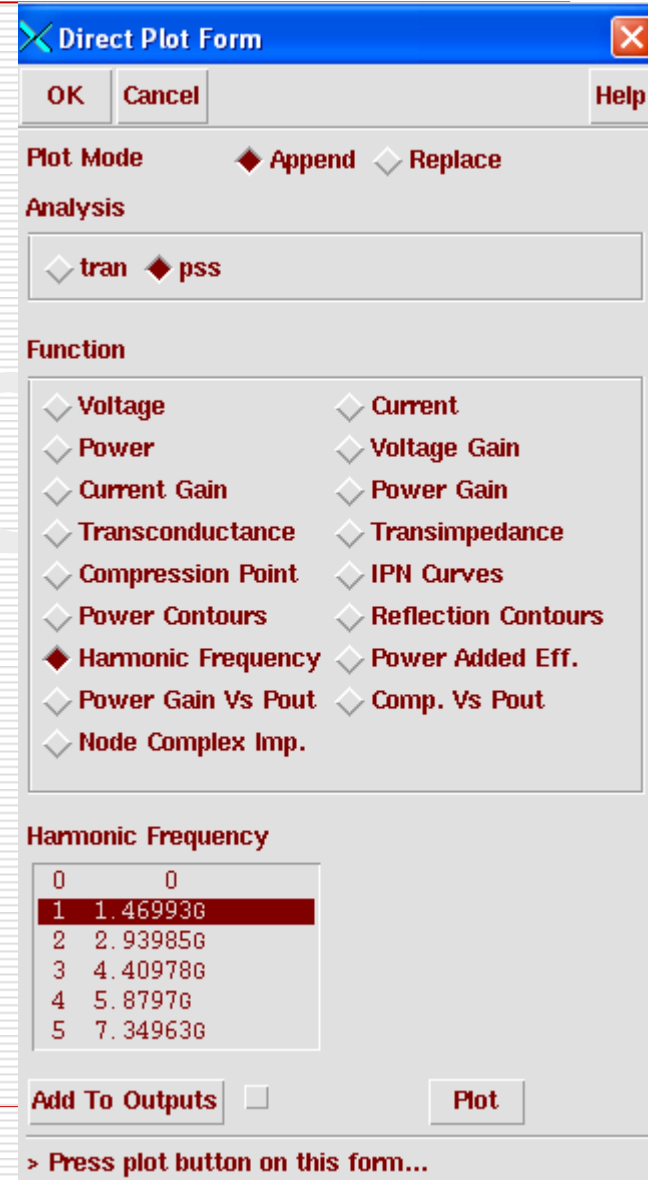
# Quadrature CMOS VCO

---



# Tran and PSS Analyses

- ◆ Setup and run these analyses as described before
- ◆ The oscillation frequency for  $V_{cont}=0.7V$  is  $\sim 1.47GHz$



The image shows a 'Direct Plot Form' dialog box from a simulation software. It has a blue title bar with a close button. The dialog contains several sections: 'Plot Mode' with 'Append' selected; 'Analysis' with 'pss' selected; 'Function' with a list of analysis options where 'Harmonic Frequency' is selected; and 'Harmonic Frequency' with a table of harmonic data. At the bottom, there are 'Add To Outputs' and 'Plot' buttons, and a note to press the plot button.

**Direct Plot Form**

OK Cancel Help

Plot Mode ☒ Append ☐ Replace

Analysis

☐ tran ☒ pss

Function

☐ Voltage ☐ Current  
☐ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☒ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp.

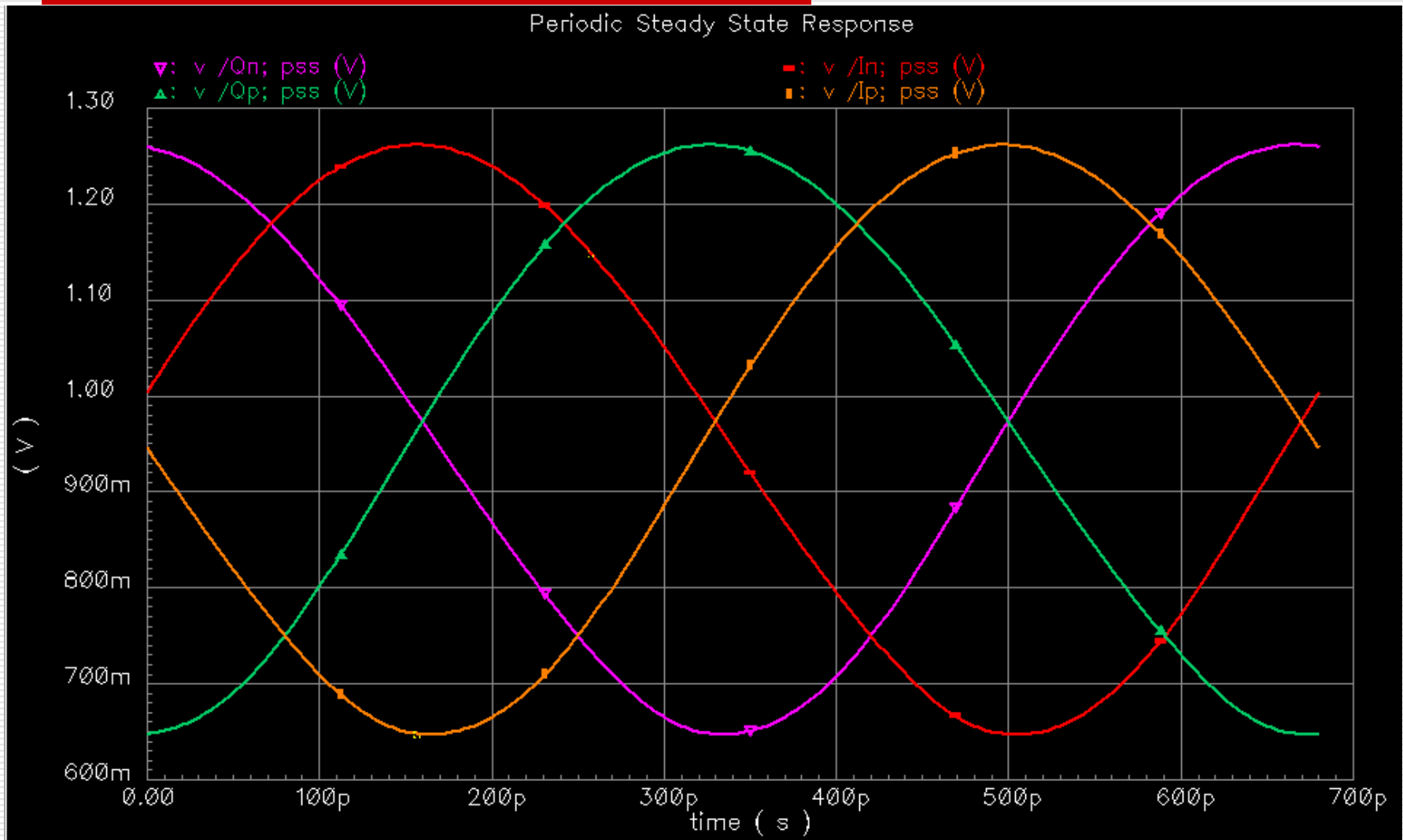
Harmonic Frequency

0	0
1	1.46993G
2	2.93985G
3	4.40978G
4	5.8797G
5	7.34963G

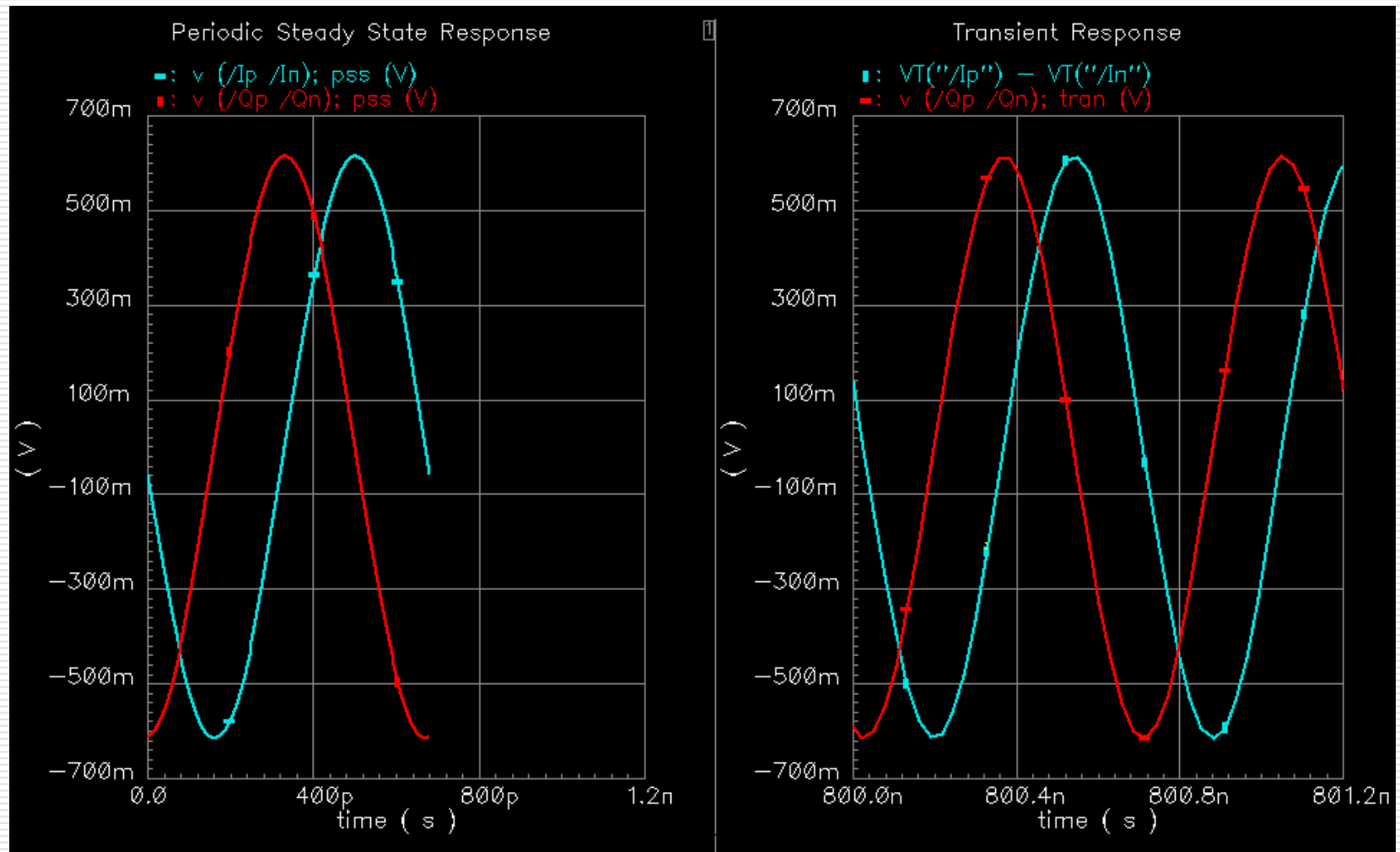
Add To Outputs ☐ Plot

> Press plot button on this form...

# PSS Response in time (1 period)



# PSS vs. Transient Response (diff)



# Swept PSS to plot frequency vs. Vcont

- ◆ To plot oscillation frequency vs. control voltage
  - PSS is much faster & easier than *Tran*
- ◆ Highlight *sweep* and specify sweep variable and range

Choosing Analyses -- Cadence® Analog Design Environment...

OK Cancel Defaults Apply Help

◆ Beat Frequency 1.4G Auto Calculate ☐

◆ Beat Period 1.4G

Output harmonics

Number of harmonics 5

Accuracy Defaults (empreset)

☐ conservative ☒ moderate ☐ liberal

Additional Time for Stabilization (tstab) 100n

Save Initial Transient Results (saveinit) ☐ no ☐ yes

Oscillator ☒ Oscillator node /Ip Select

Reference node /In Select

Sweep ☒ Frequency Variable? ◆ no ◇ yes

Variable  Variable Name Vcont Select Design Variable

Sweep Range

◆ Start-Stop Start 0 Stop 1.4G

◆ Center-Span

Sweep Type

◆ Linear Step Size .1

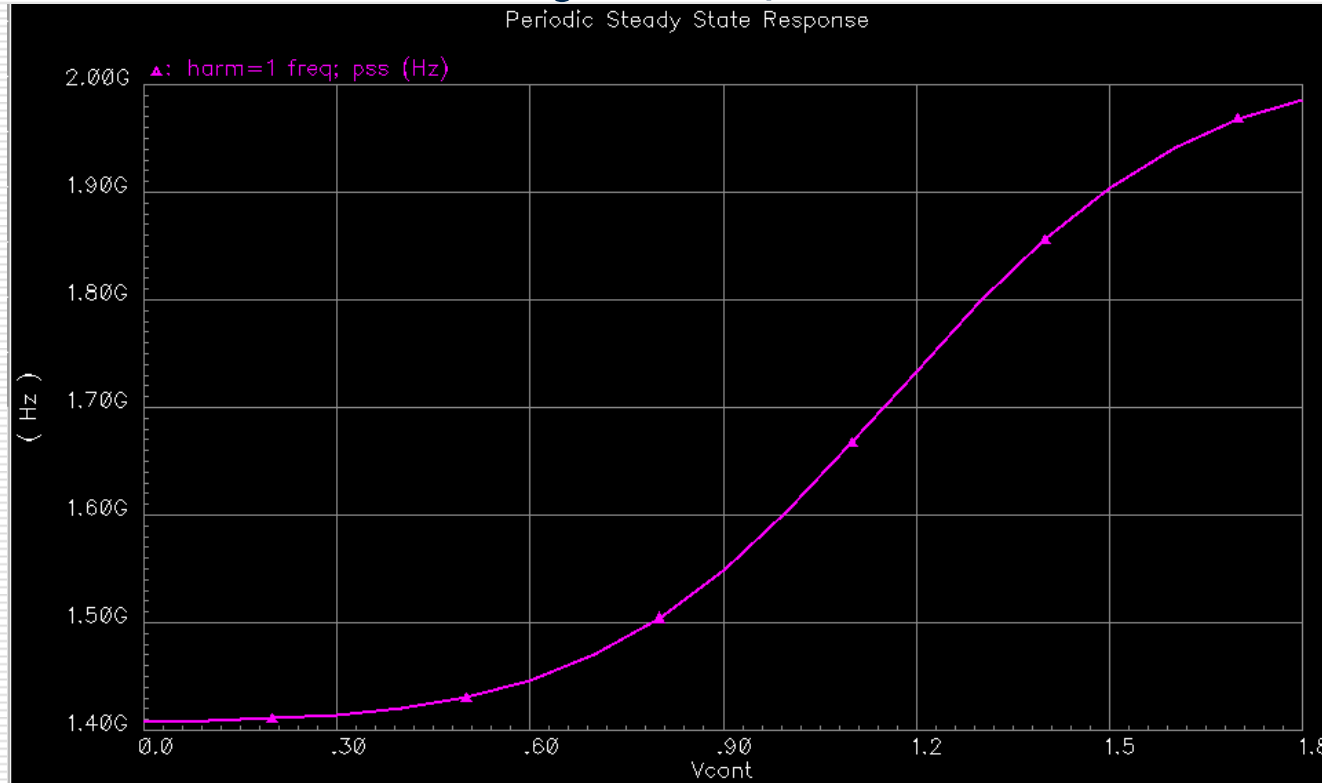
◆ Logarithmic Number of Steps 1

Add Specific Points ☐

Enabled ☒ Options...

# VCO frequency vs. control voltage

- ◆  $f = 1.407 \text{ GHz @ } 0\text{V}$
- ◆  $f = 1.987 \text{ GHz @ } 1.8\text{V}$
- ◆ Nonlinearity  $\propto$  slope



Direct Plot Form

OK Cancel Help

Plot Mode ☒ Append ☐ Replace

Analysis

☒ pss

Function

☐ Voltage ☐ Current

☐ Power ☐ Voltage Gain

☐ Current Gain ☐ Power Gain

☐ Transconductance ☐ Transimpedance

☐ Compression Point ☐ IPN Curves

☐ Power Contours ☐ Reflection Contours

☒ Harmonic Frequency ☐ Power Added Eff.

☐ Power Gain Vs Pout ☐ Comp. Vs Pout

☐ Node Complex Imp.

Harmonic Frequency

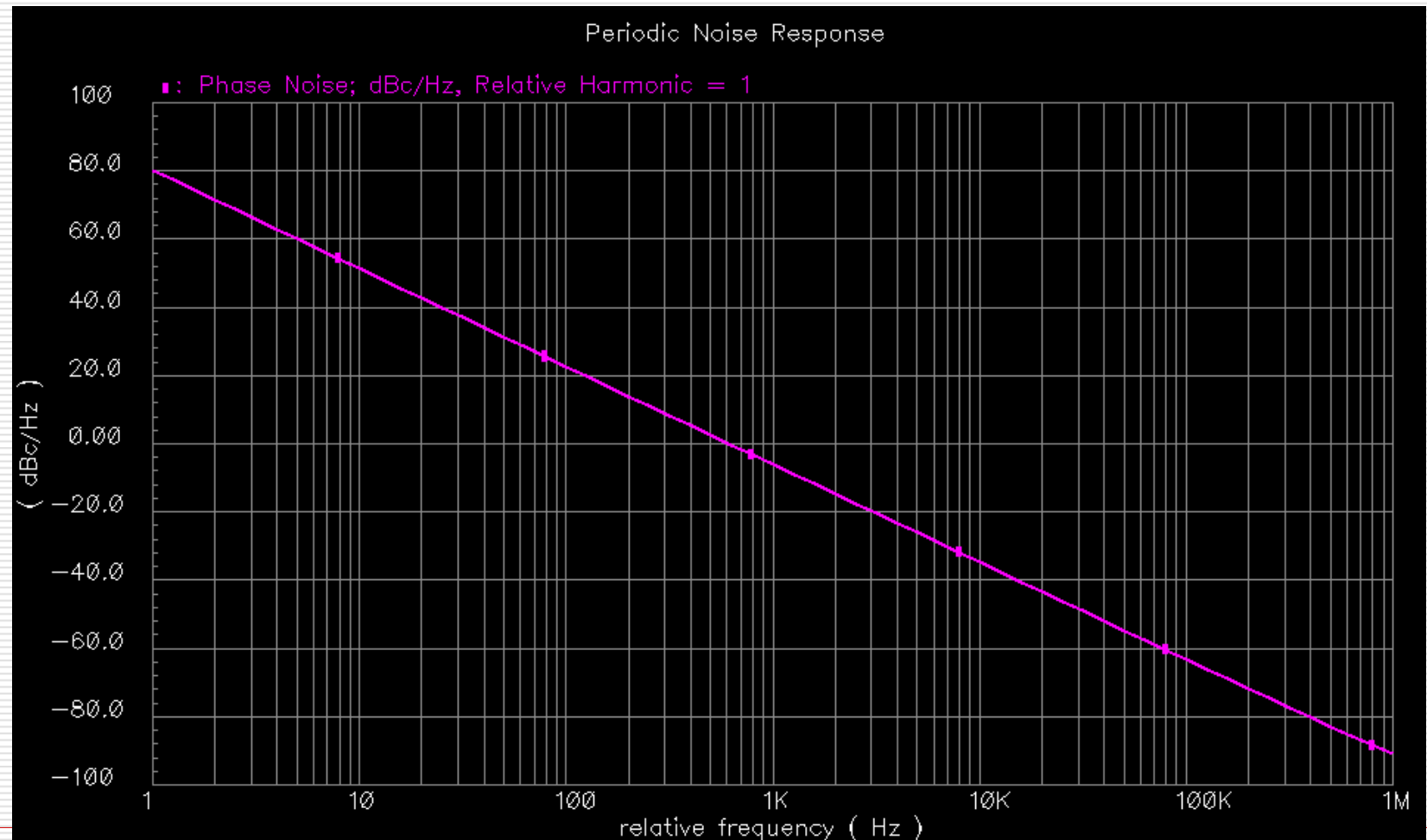
Order	Frequency (Hz)	Amplitude (dBm)
0	0	0
1	1.407496	-1.0
2	2.814986	-3.0
3	4.222476	-5.0

Add To Outputs ☐ Plot

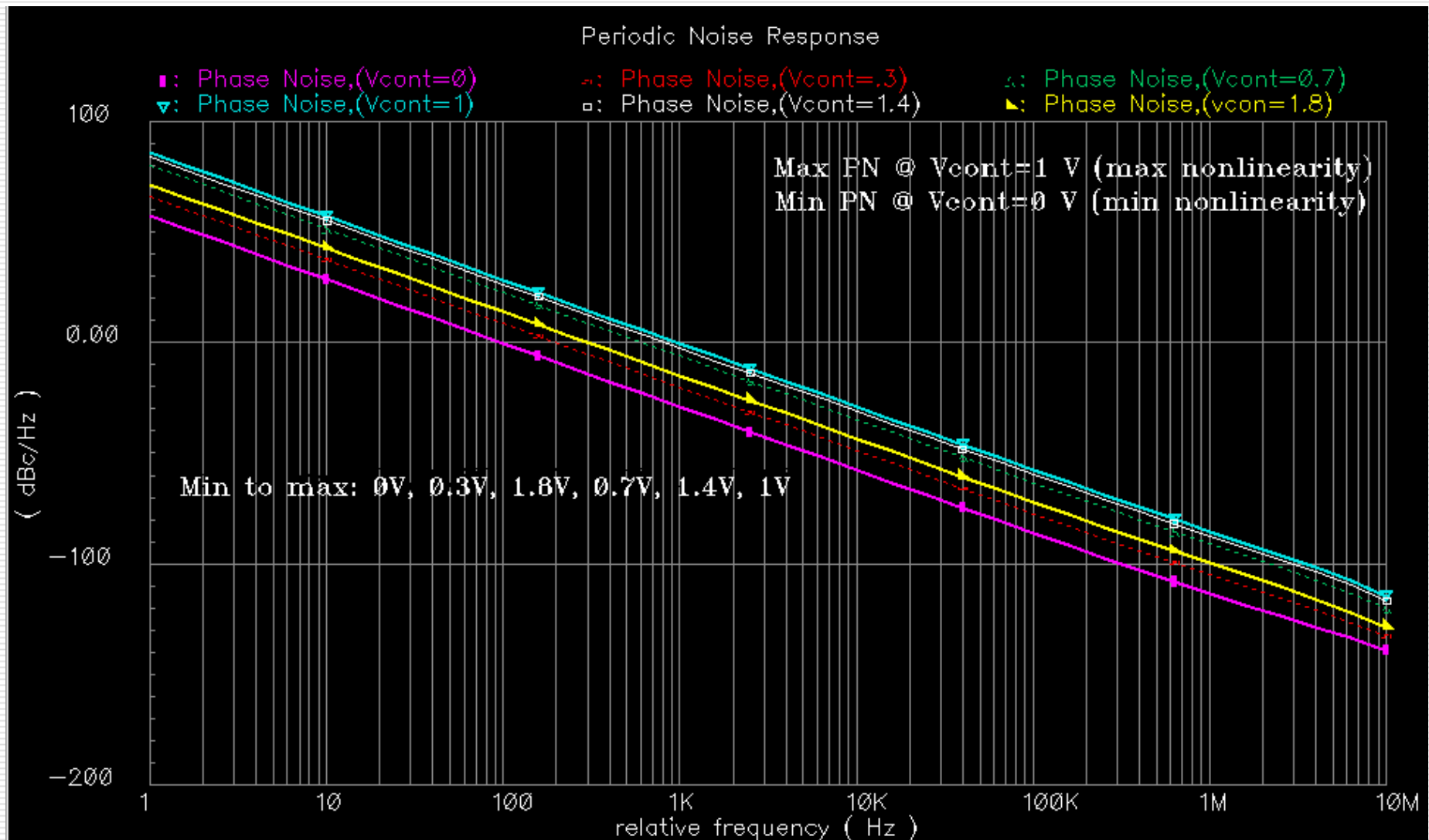
> Press plot button on this form...

# Phase Noise (@ $V_{\text{cont}}=0.7\text{V}$ )

- ◆ -63.3 dBc/Hz @ 100kHz      -91dBc/Hz @ 1MHz



# Effect of Varactor Nonlinearity



# Periodic Noise Response

