1.5 GHz Oscillator Design

RF Circuit Design

Salvatore Levantino

2020/2021

Differential CMOS LC Oscillator

Parameters

$V_T = 0.35V$

$\mu_n C_{ox} = 120\mu A/V^2$

$\mu_p C_{ox} = 60\mu A/V^2$

Requirements

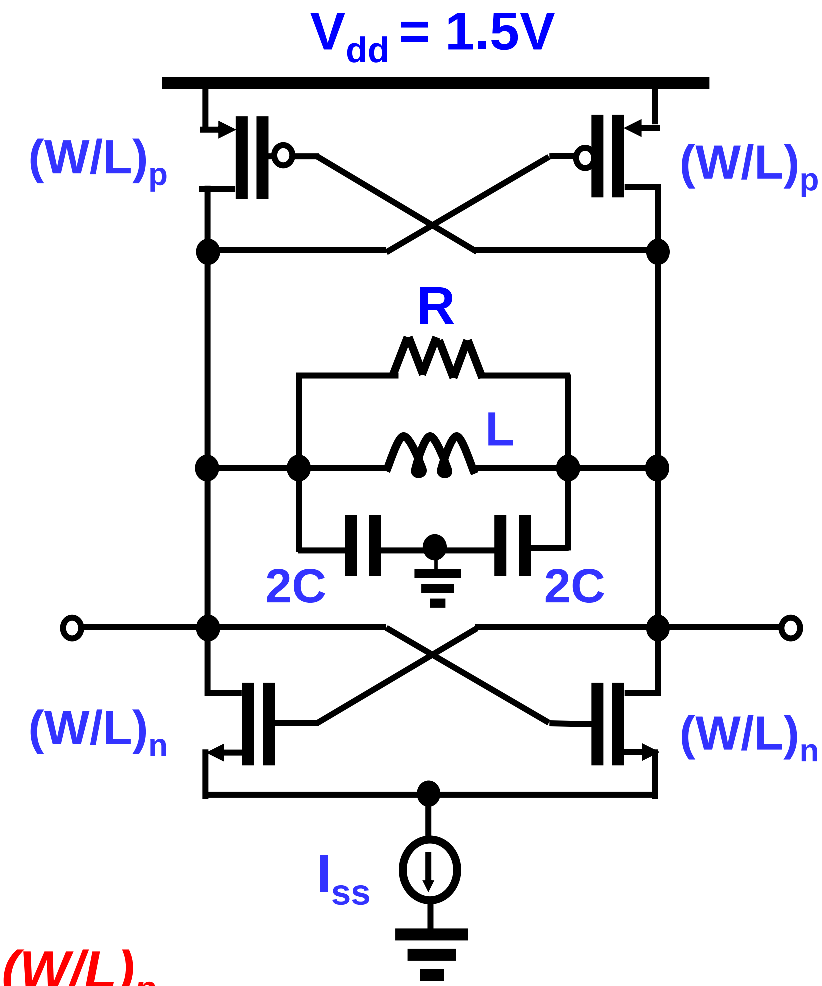
$f_0 = 1.5GHz$

$Q = 20$

$I_{ss} = 3mA$

EGF = 5

Maximize the FoM



Derive values of R, L, C, $(W/L)_n$, $(W/L)_p$

Oscillator Phase Noise and Figure of Merit

$$\mathcal{L}(\Delta\omega) = \frac{S_v(\omega_0 + \Delta\omega)}{S_v(\omega_0)} = \frac{\frac{1}{2}\frac{4K_bT}{R}\frac{1}{(2C\Delta\omega)^2}F_a}{\frac{1}{2}A_0^2} = \frac{K_bTRF_a}{Q^2A_0^2}\left(\frac{\omega_0}{\Delta\omega}\right)^2 = \boxed{\frac{K_bTF_a}{Q^2 2\eta P_{diss}}\left(\frac{\omega_0}{\Delta\omega}\right)^2}$$

$$C = Q/(\omega_0 R)$$

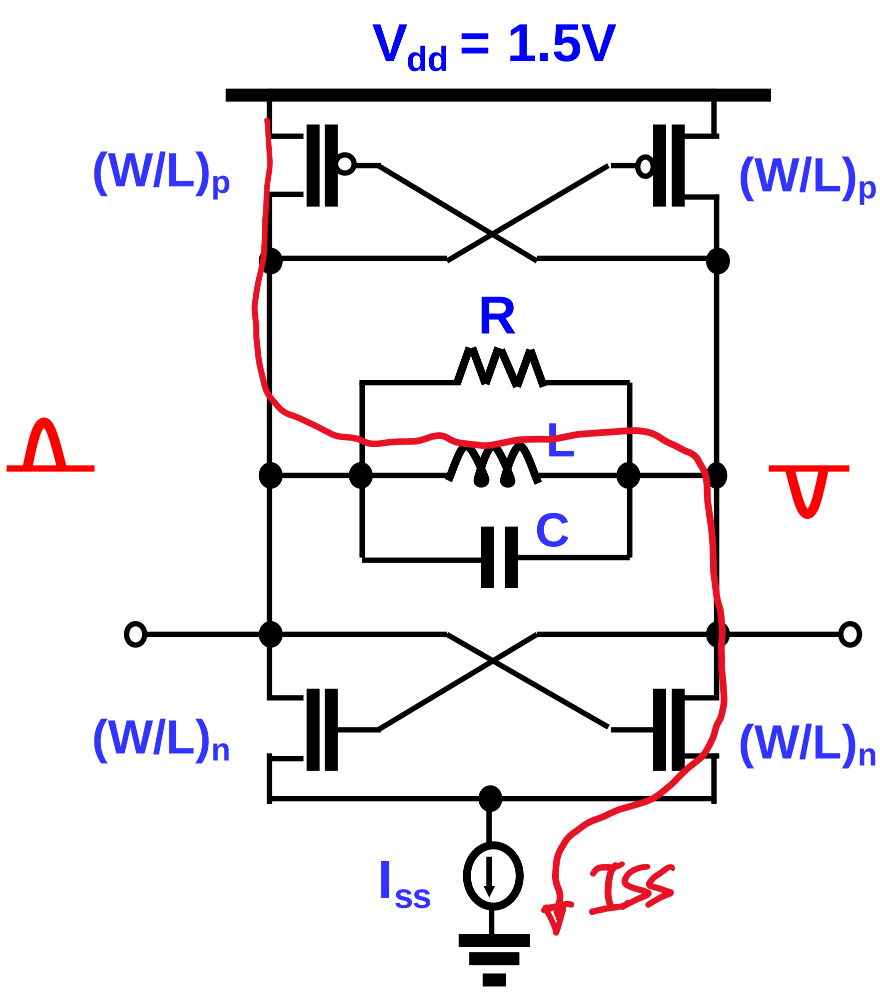
$$\eta = \frac{P_{out}}{P_{diss}} = \frac{\frac{A_0^2}{2R}}{V_{dd}I_{ss}} = \frac{2A_0^2}{\pi A_0 V_{dd}} \propto \boxed{\frac{A_0}{V_{dd}}}$$

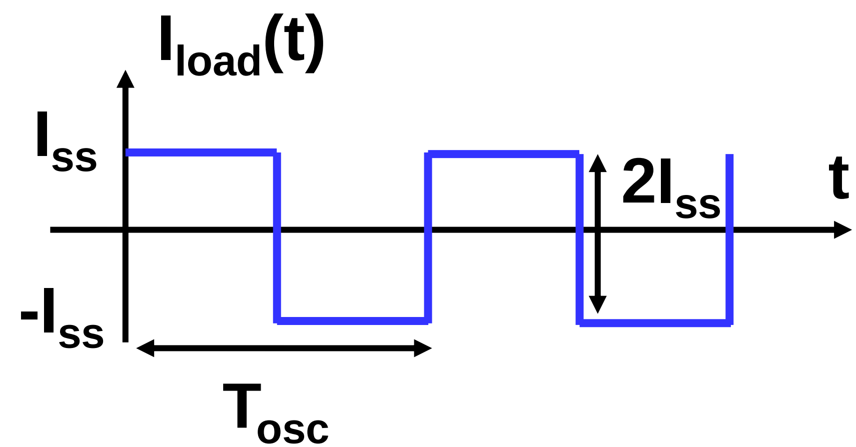
$$A_0 = \frac{4}{\pi} I_{ss} R$$

$$FoM = \frac{1}{\mathcal{L}(\Delta\omega)\, P_{diss,mW}}\left(\frac{\omega_0}{\Delta\omega}\right)^2 = 10^{-3}\frac{2\eta Q^2}{KTF_a} \boxed{\propto \eta}$$

To maximize the FoM, A_0 has to be maximized

Oscillation Amplitude





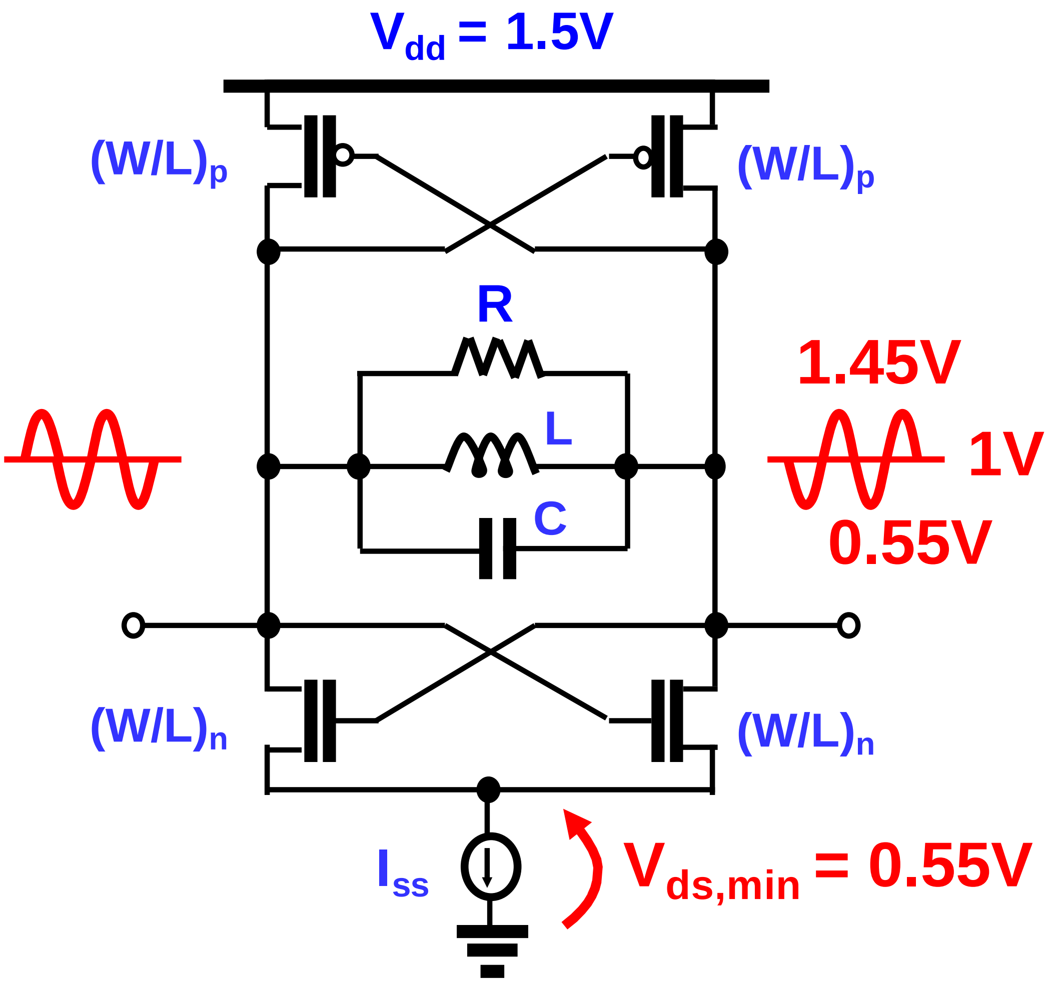
$$I_1 = \frac{2}{\pi} 2 I_{ss}$$

$I_{load}(t)$ 1st harmonic

Oscillation Amplitude:

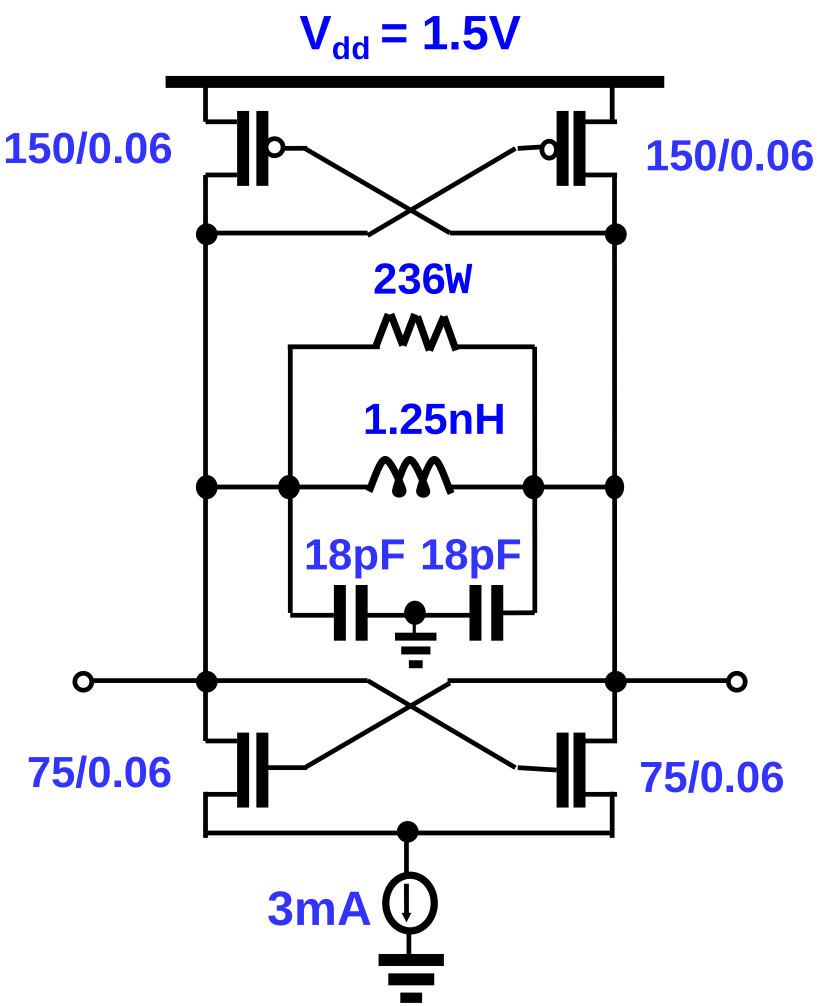
$$A_0 = I_1 R = \frac{4}{\pi} I_{ss} R$$

Maximizing the Amplitude



$A_{0,max}$ = 0.9V

Deriving the component values



$$A_0 = 0.9V = \frac{4}{\pi} \cdot I_0 R \rightarrow R \approx 236\Omega$$

$$Q = 20 = \frac{R}{\omega_0 L} \rightarrow L \approx 1.25nH$$

$$f_0 = 1.5\text{GHz} = \frac{1}{2\pi\sqrt{LC}} \rightarrow C \approx 9pF$$

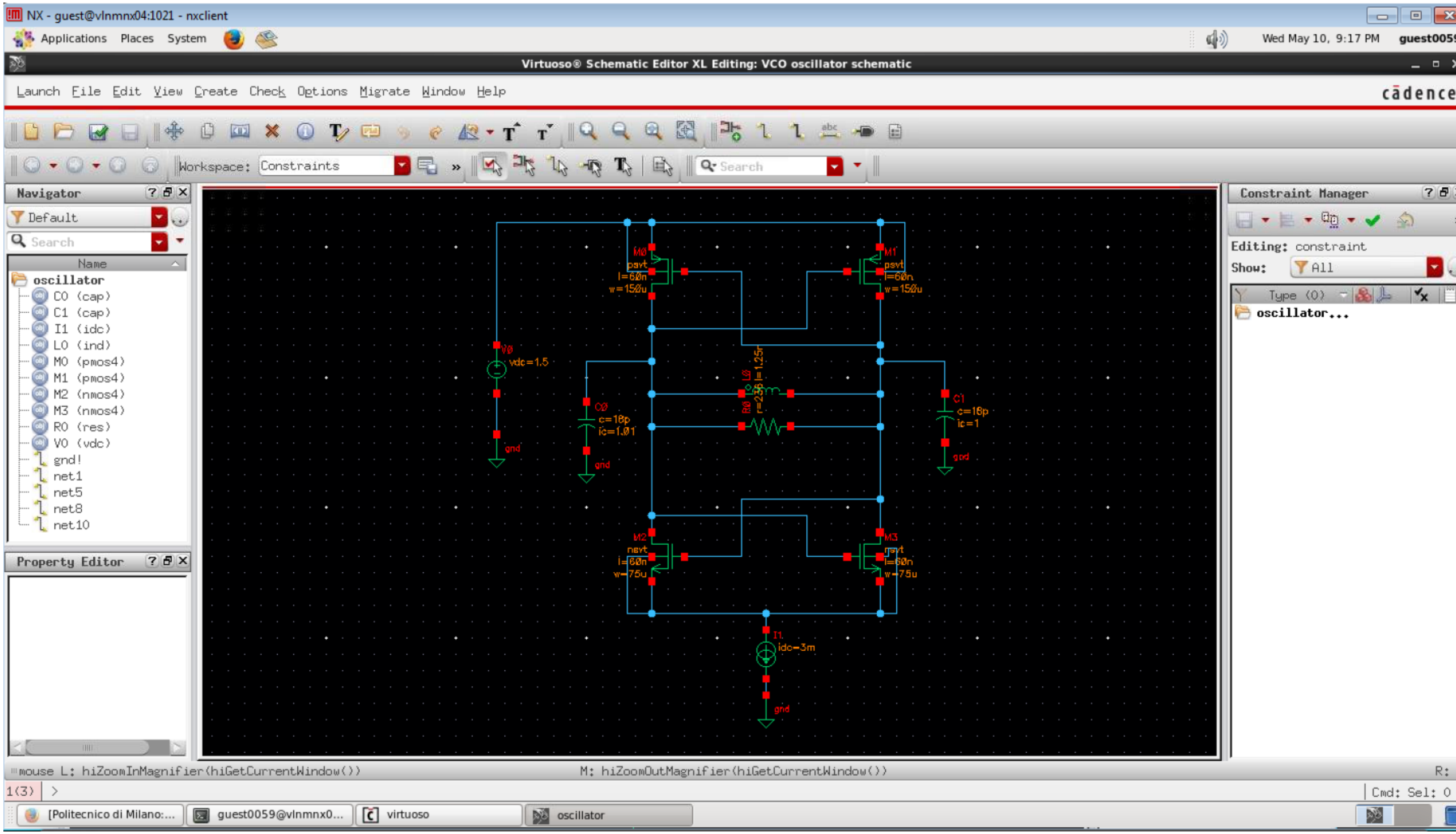
$$g_m R = 5 \rightarrow \left(\frac{W}{L}\right)_n = 1250 = \left(\frac{75\mu m}{60nm}\right)$$

Try using a <u>spreadsheet</u> to play around with the values

Cadence Simulation

- Start virtual desktop: *NX Client*
- Applications ▸ Terminal ▸ *virtuoso.sh*
- File ▸ New ▸ Library (do not need process)
- File ▸ New ▸ Cellview (schematic – schematic XL)
- “i” to add instances
- Use devices from “*analogLib*” library (*res*, *cap*, *ind*, *nmos4*, *pmos4*, *vdc*, *gnd*, *idc*)
- Use model name nsvt and psvt for nmos4 and pmos4, respectively

Draw Schematic



Model File

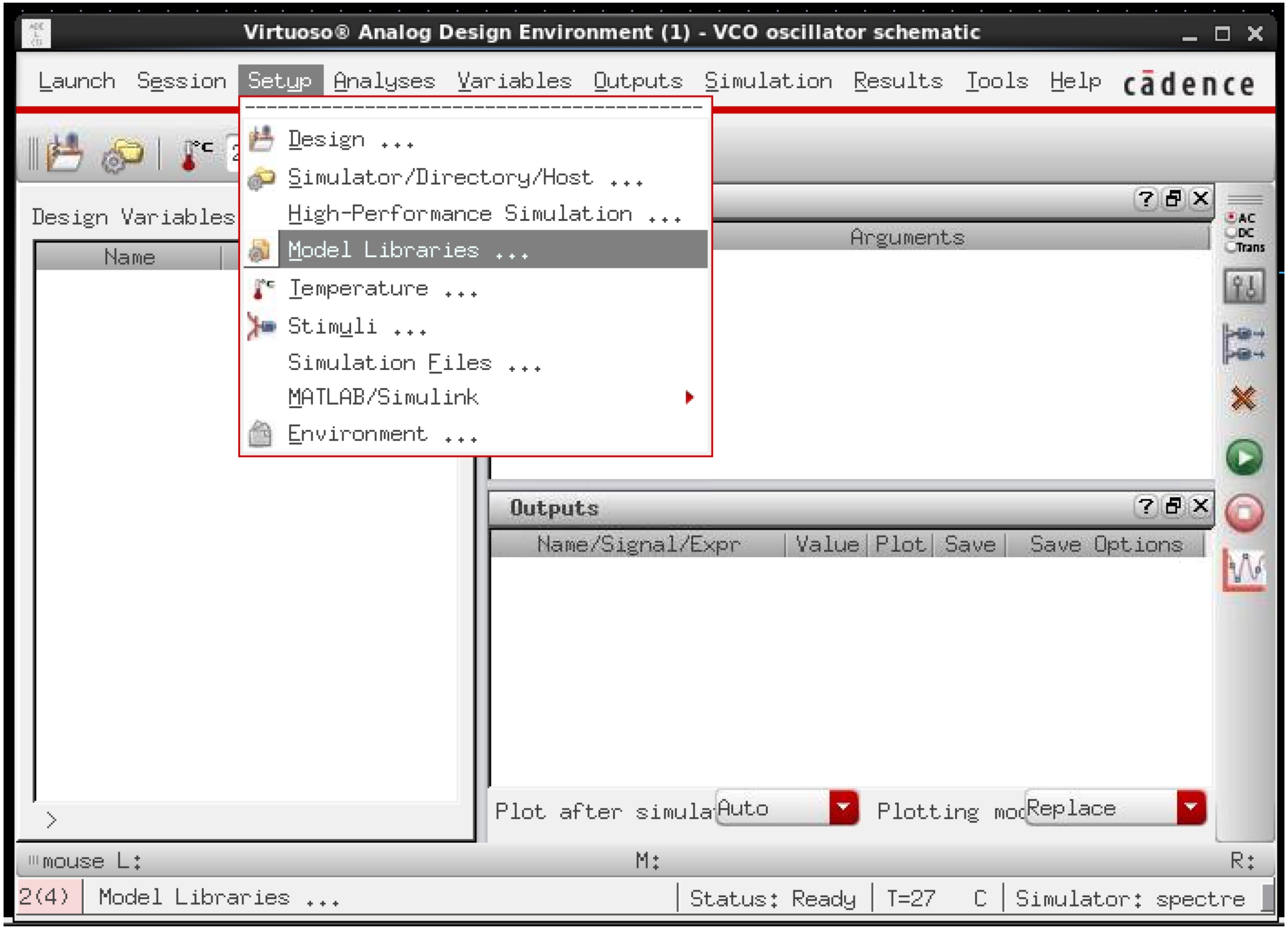
- **Create a text file:**

“models.scs”

```
simulator lang=spectre
model nsvt mos1 type=n vto=0.35 kp=120e-6 lambda=5e-2 \
cgso=2e-10 cgdo=2e-10 cgbo=2e-10
model psvt mos1 type=p vto=-0.35 kp=60e-6 lambda=5e-2 \
cgso=2e-10 cgdo=2e-10 cgbo=2e-10
```

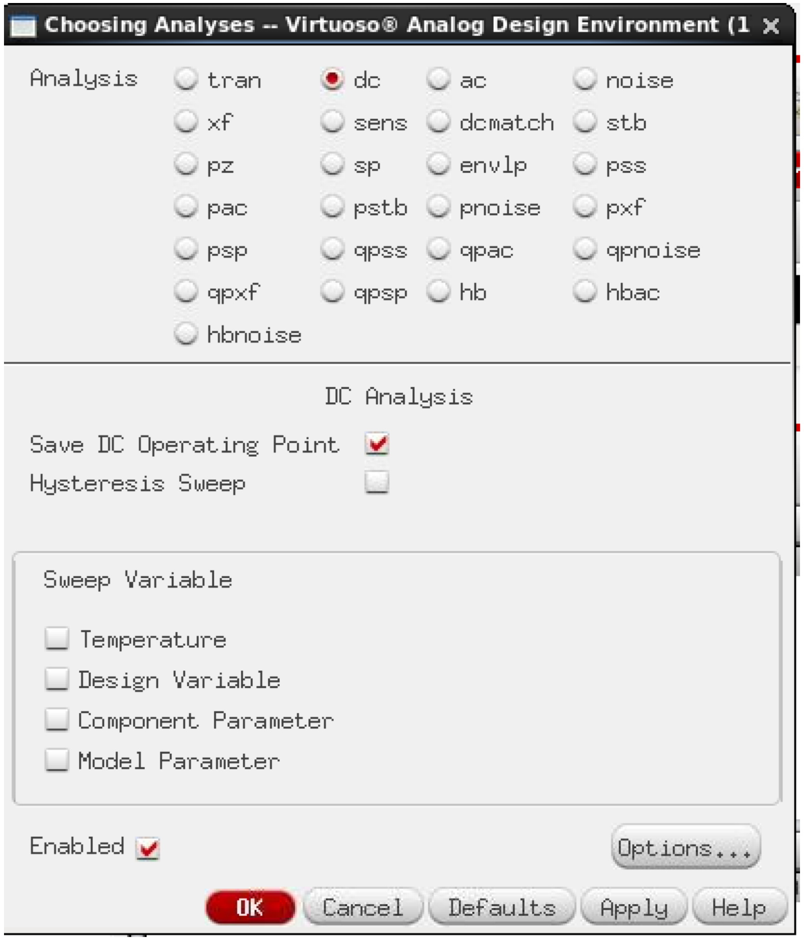
- **Open ADE L**
- **Menu Setup ▸ Model Libraries (path to your model.scs file)**

Model Library

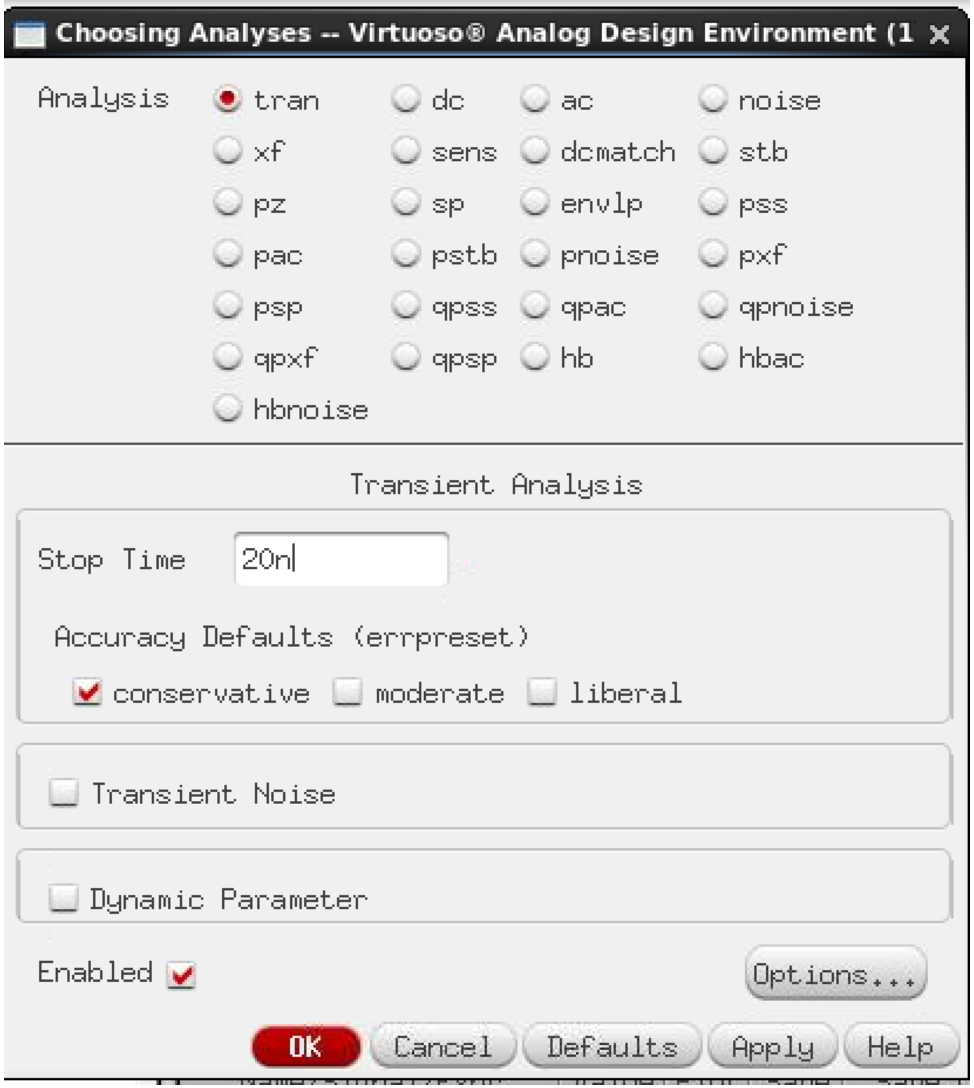


1) DC Analysis

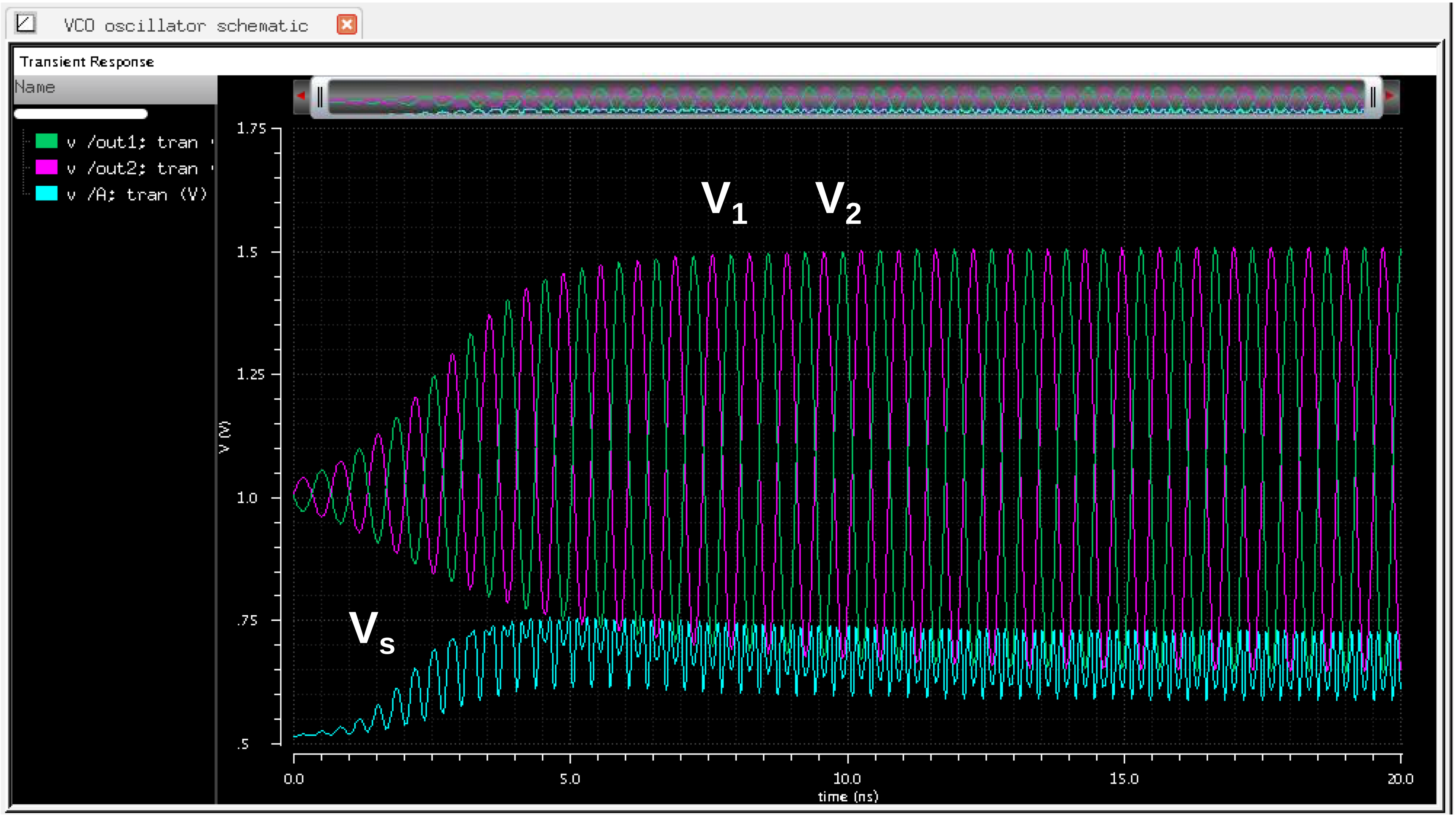
- **From ADE L tool**
- **Menu *Analysis*: dc**
- **Menu *Results* ▸ Print Operating Point**
  - **$g_m \approx 21.5$ mA/V**
  - **$c_{gs,n} \approx 15$ fF, $c_{gs,p} \approx 15$ fF**



2) Transient Analysis

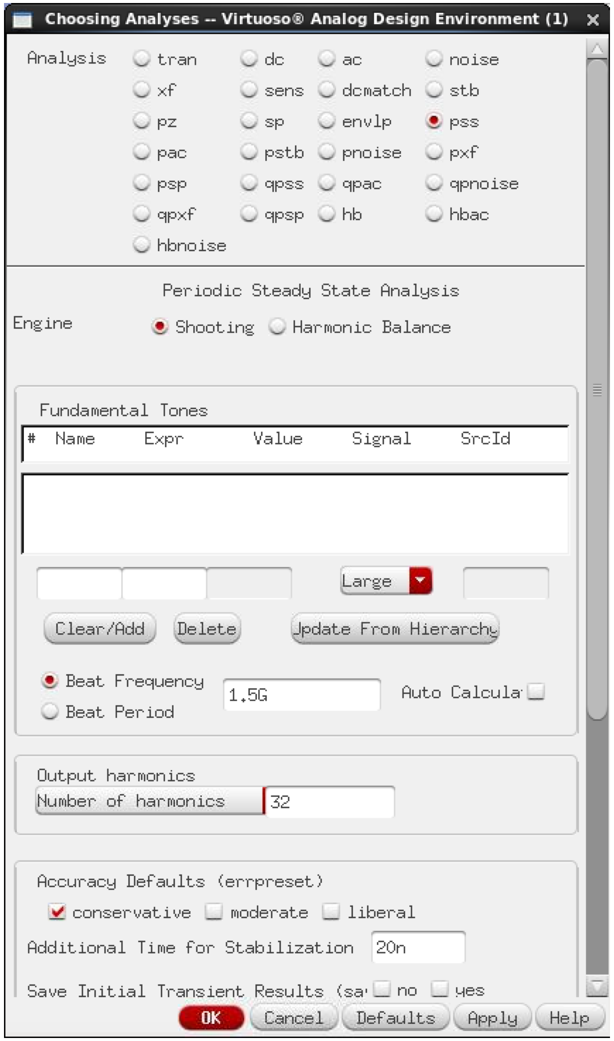


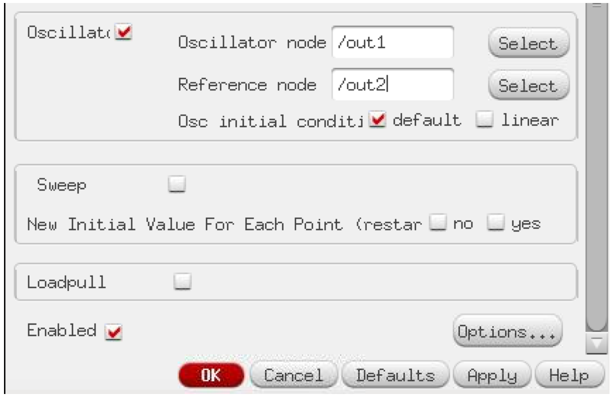
Result of Transient Analysis



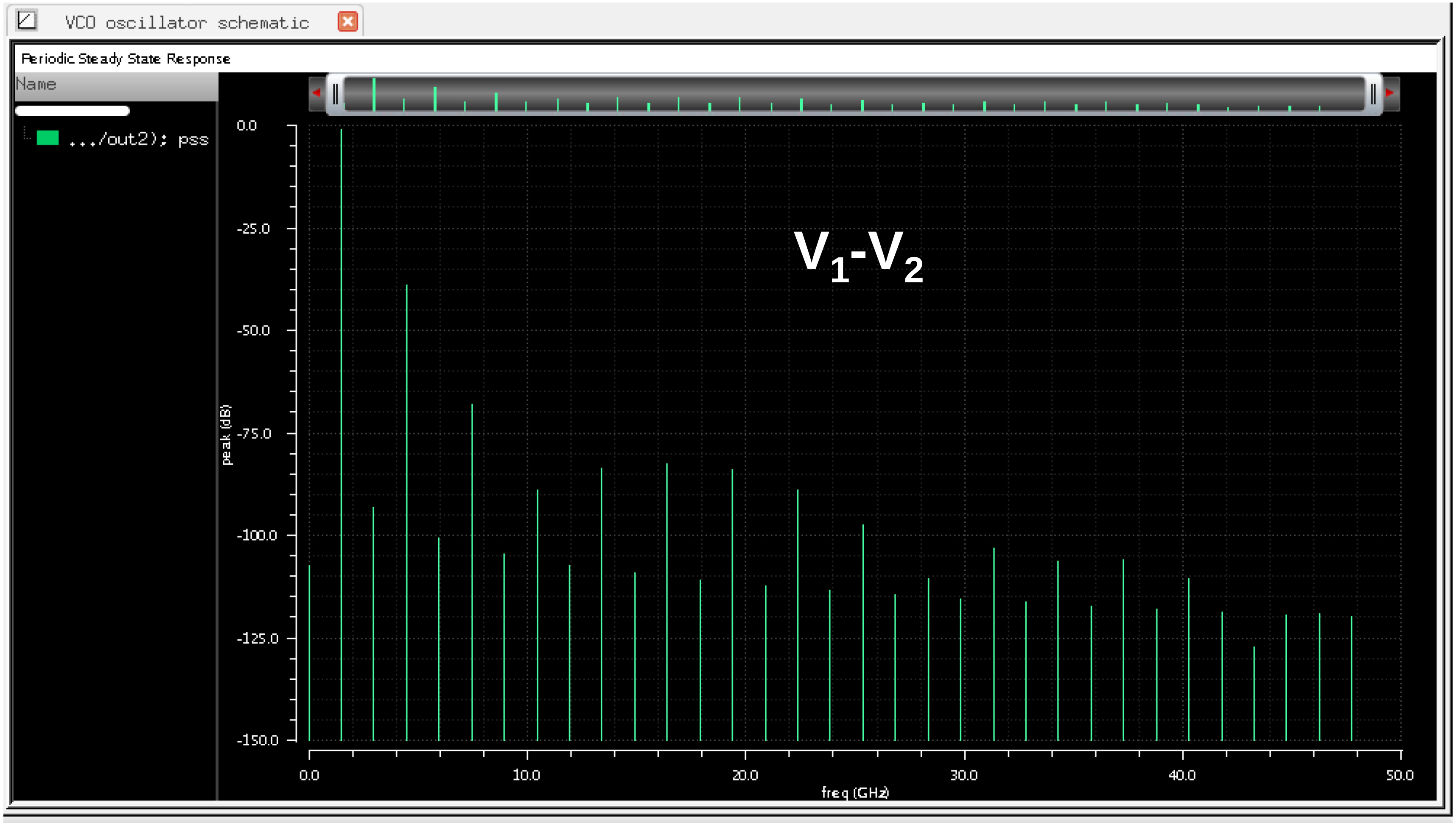
Set initial conditions!

3) Periodic Steady State

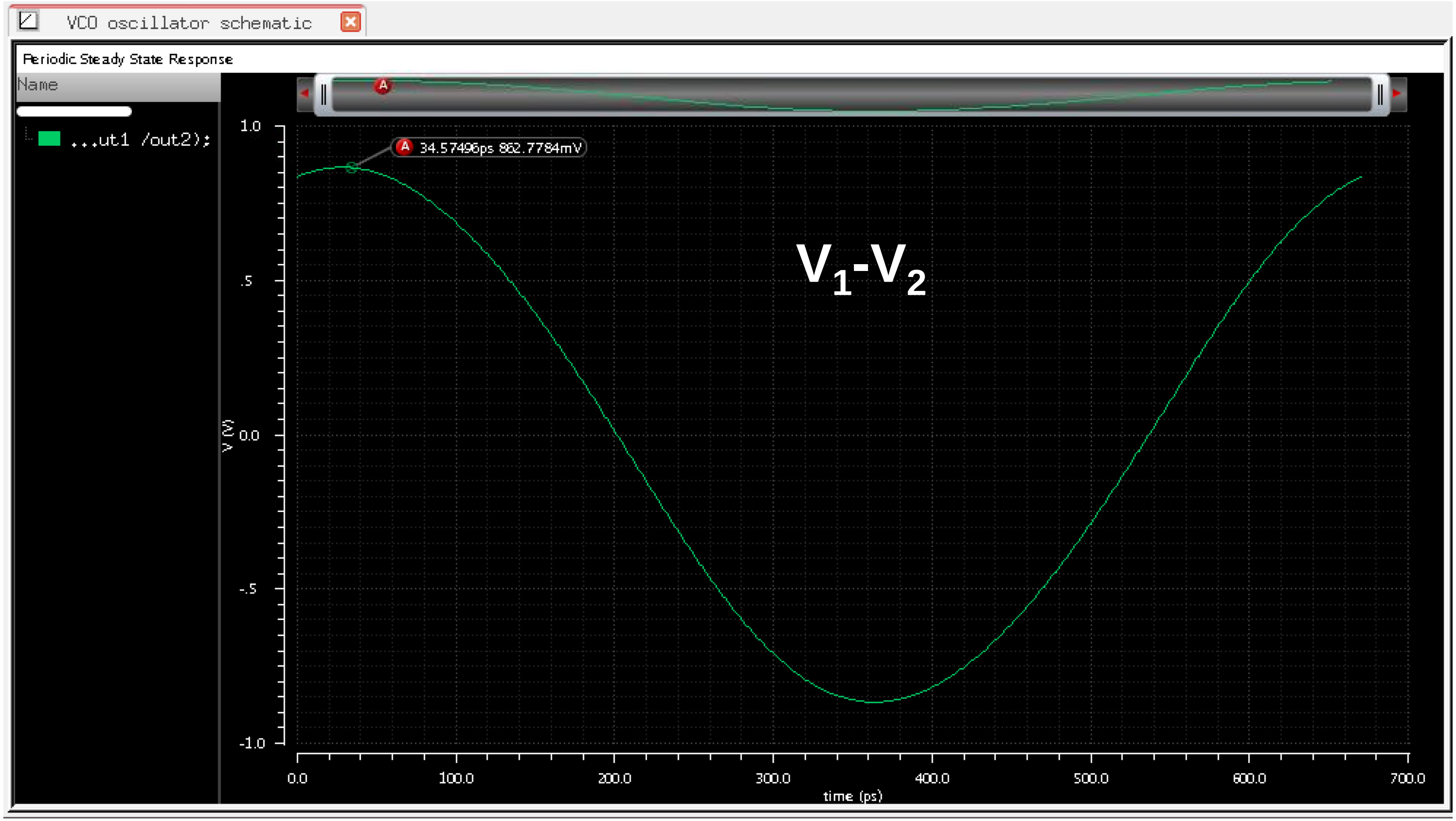




Result of PSS Analysis (frequency domain)

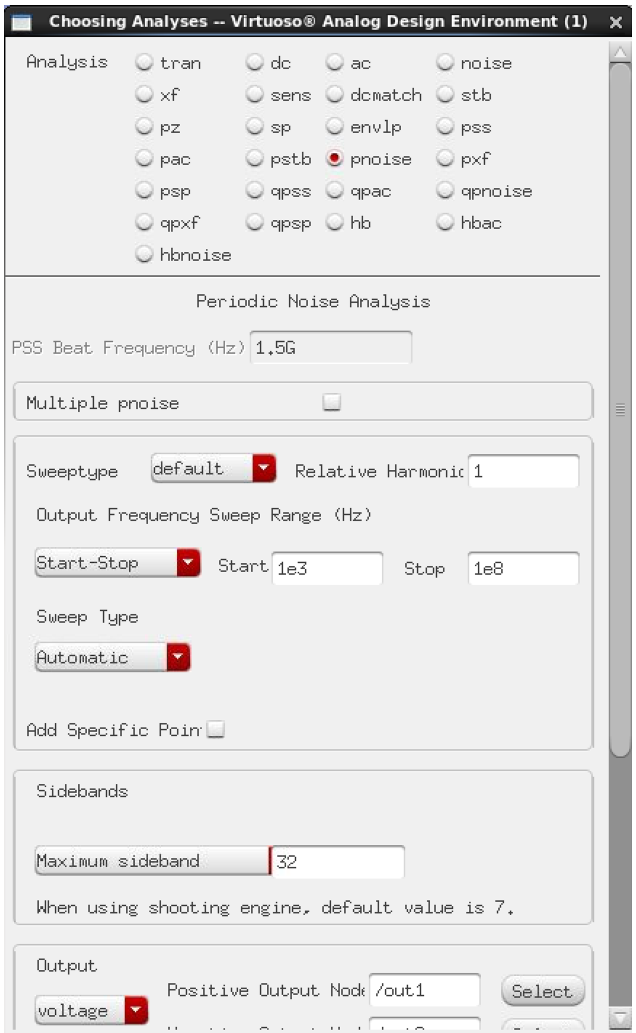


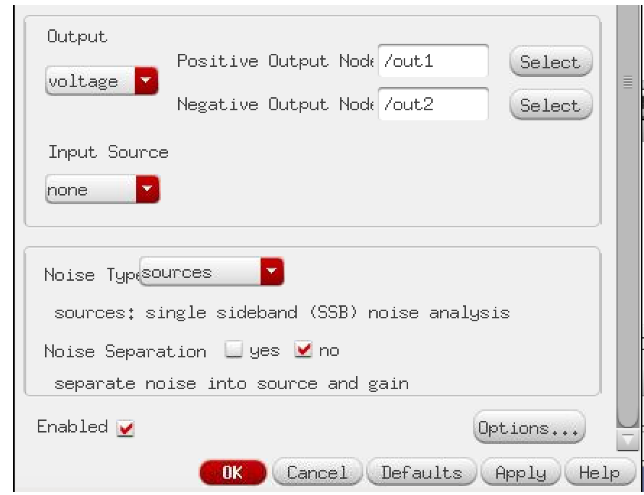
Result of PSS Analysis (time domain)



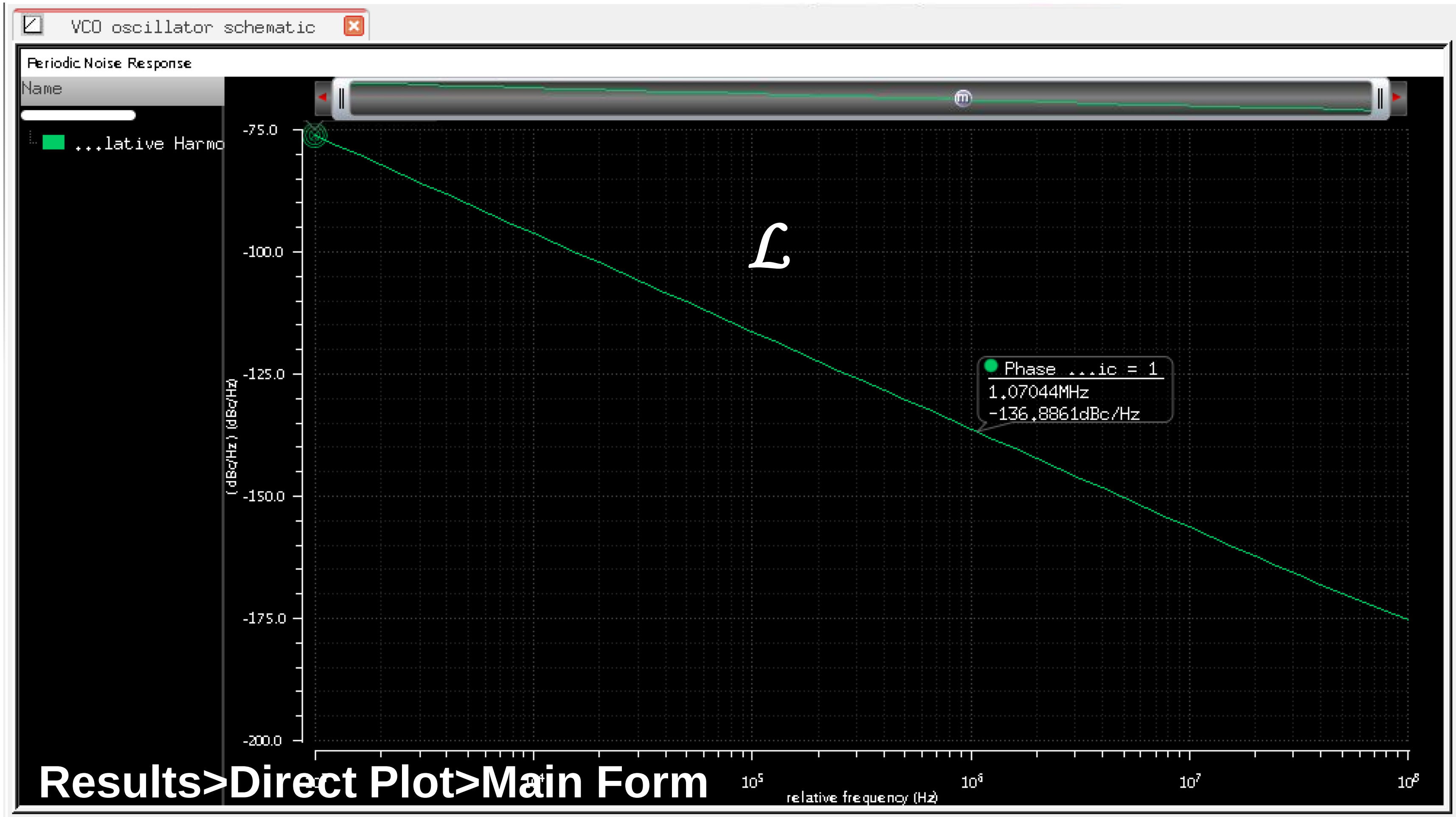
- A_0 **= 0.89V**

4) Periodic Noise (PNOISE)





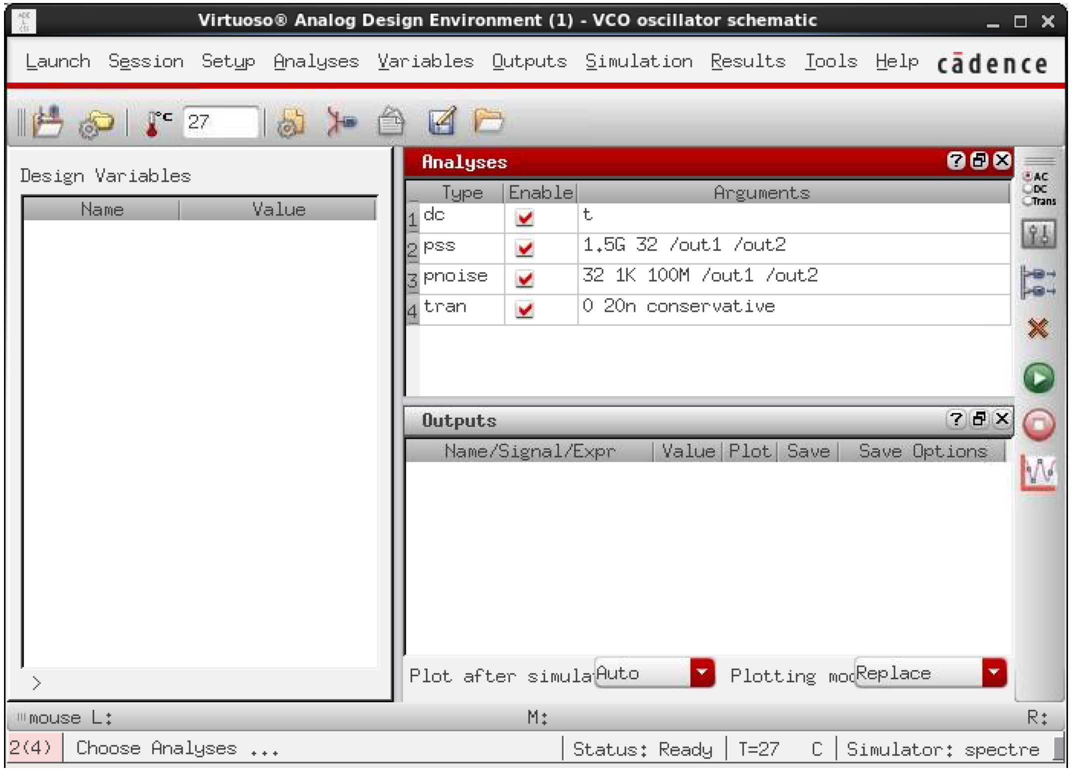
Result of PNOISE Analysis



Results>Direct Plot>Main Form

- $\mathcal{L}$ **= -136.9 dBc/Hz**

Analog Design Environment (ADE)

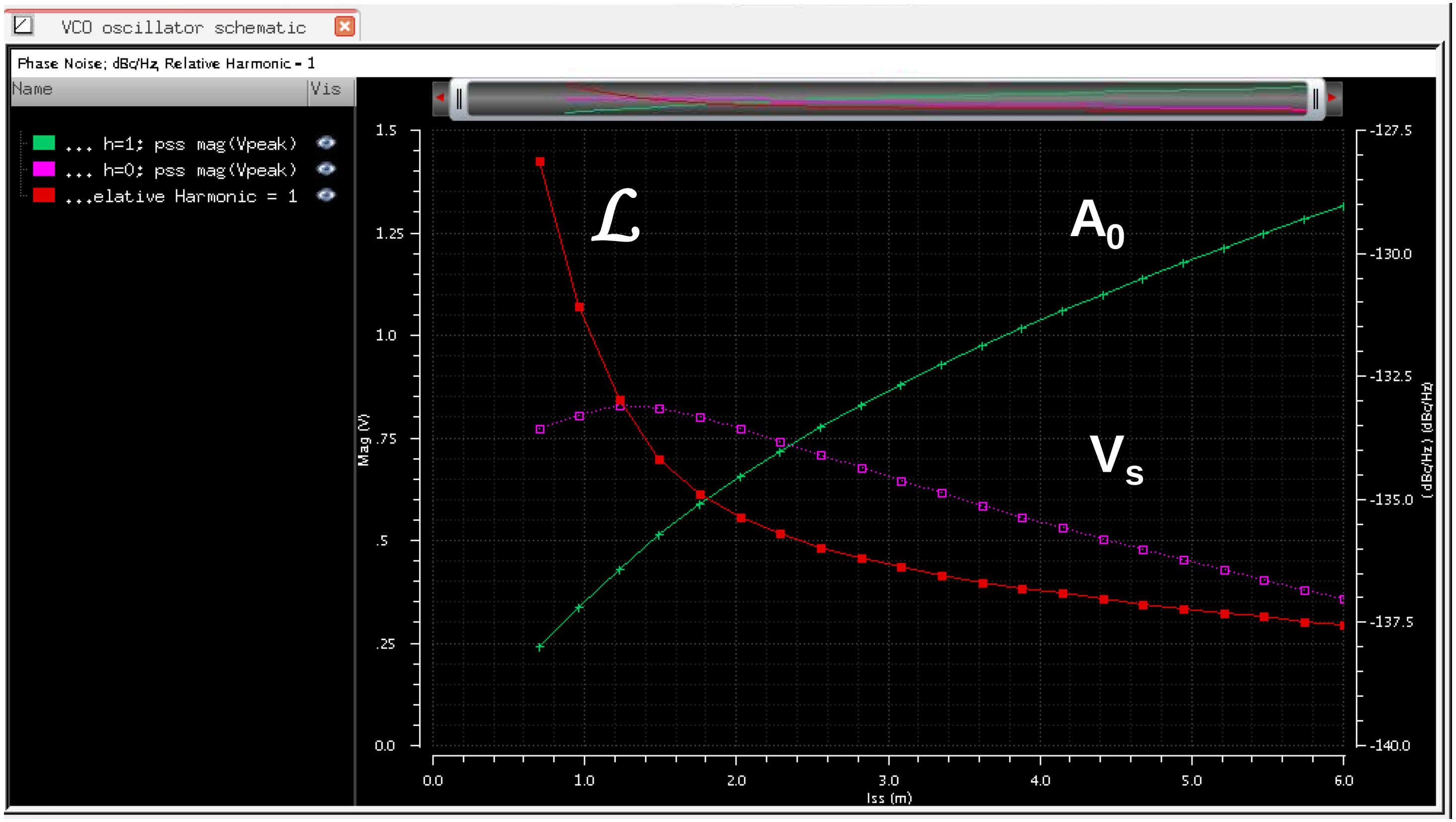


- **Set analyses:**

  PSS (sweep variable)

  PNOISE

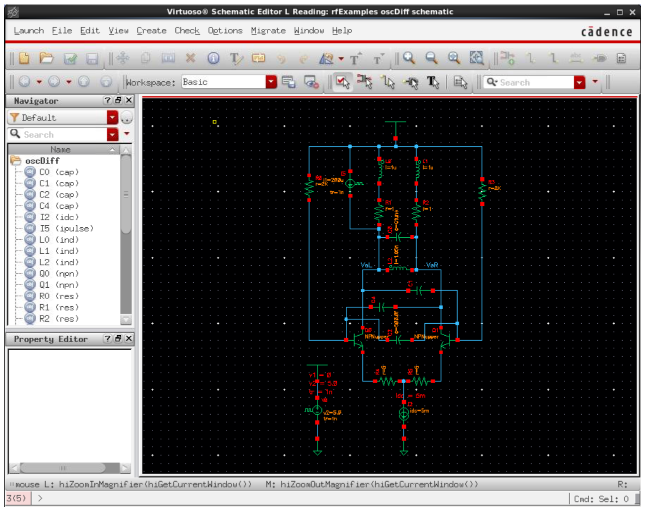
Exercise: sweep I_{ss} from 0.7 to 6.0 mA



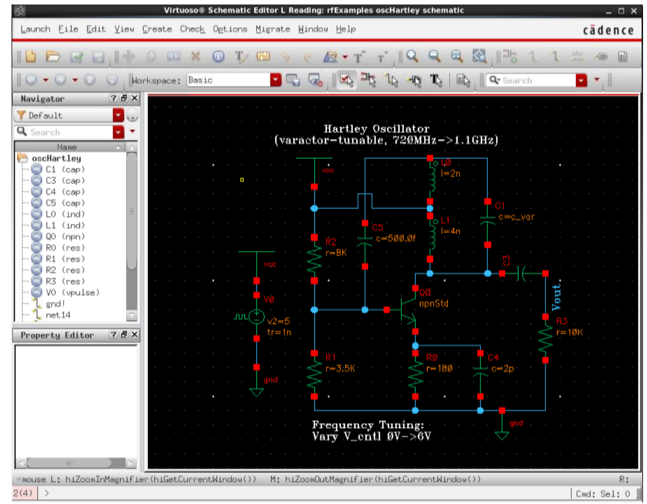
- **Amplitude of differential output increases (but saturates)**
- **Source average voltage goes down**
- **Phase noise decreases (but saturates)**

Library: rfExamples

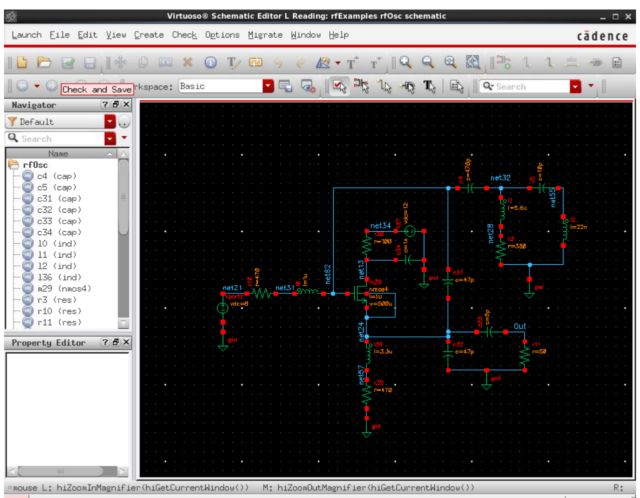
oscDiff



oscHartley



rfOsc



vp_osc