

SmartSpiceRF

Switched-Capacitor Filter Simulation

Tutorial

Introduction

Information presented in this Tutorial is based on the assumption that the user is familiar with the following:

- Basics of the computer operating system and hardware employed
- Basic terminology of semiconductor process and device operation
- Circuit design, schematic capture, and simulation

The simulation flow described in Tutorial uses Silvaco Process Design Kit (PDK) based on a fictional process called SBCD (Silvaco Bipolar Cmos Dmos). SBCD process includes the following devices: two low-voltage Cmos, two high-voltage Dmos, four resistors, capacitor, and inductor. SBCD libraries provide all the data needed to demonstrate a real design flow.

This Tutorial presents front end design flow of Switched-Capacitor Filter (SC) circuit and guides you through the following circuit design steps:

- Schematic Capture with Gateway
- Simulation with SmartSpiceRF
- Results postprocessing with SmartView

The following Silvaco EDA tools are needed to work with Tutorial:

- GATEWAY
- SMARTSPICERF
- SMARTVIEW

1: GATEWAY Schematic Capture

Start Gateway and load the workspace file SC.workspace by selecting **File→Open→Workspace**. When the browser window appears, navigate to the directory where RF Demo PDK Examples were installed, descent into ./SC directory, and select SC.workspace file.

1.1: Loading SCF Schematic

This Tutorial presents the first order low pass switched-capacitor filter (SCF) simulation.

SCF and OP Amplifier circuit schematics, shown on Figure 1 and Figure 2 accordingly, were used to create two testbench circuits.

To load these files, select **File→Open→ Schematic**. When the file browser appears, navigate to the ./libraries/symbols/sc. directory and select SC_test.schlr, then OpAmp.schlr.

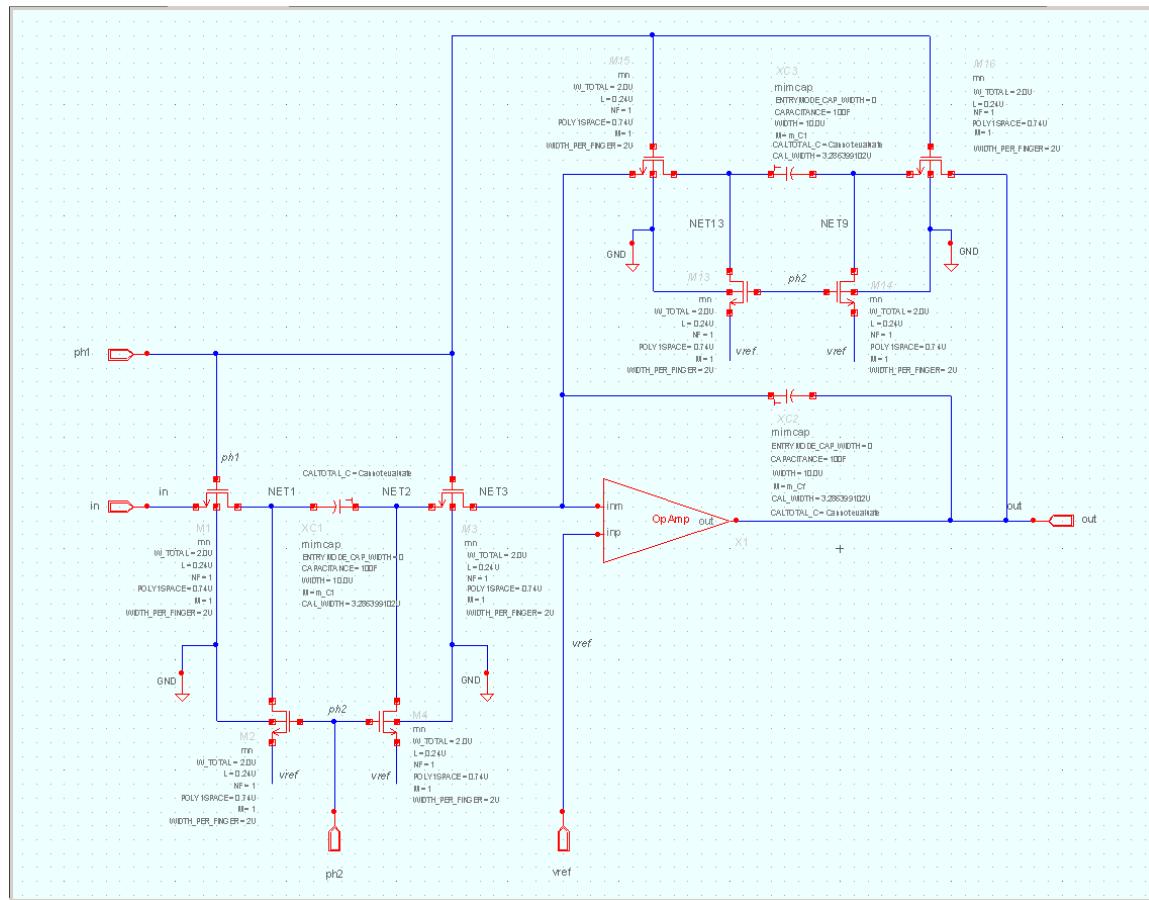


Figure 1: Switched-Capacitor Filter Schematic

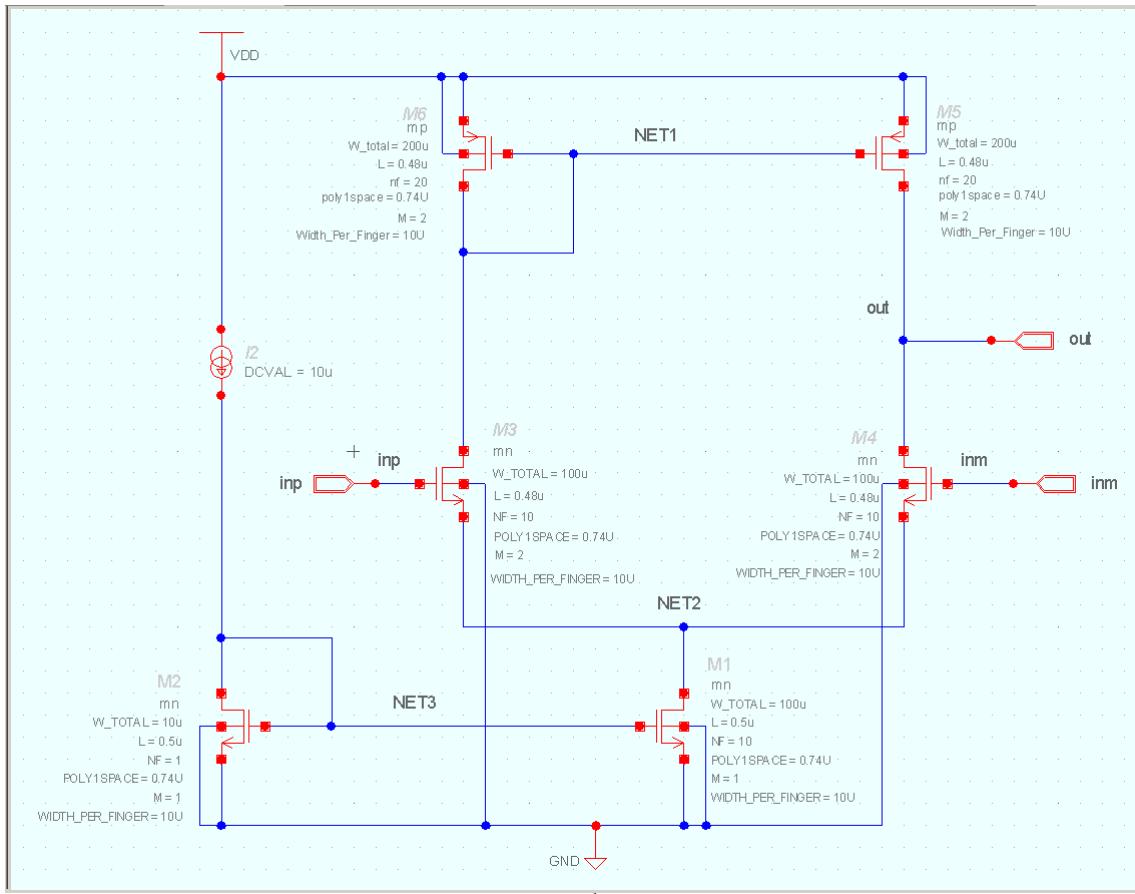


Figure 2: OP Amplifier Schematic

1.2: Simulator Setup

Check GATEWAY that your preferred simulator is set to SMARTSPICERF. In GATEWAY click on **Edit -> Preferences** to open the preferences setup window (Figure 3). Choose **Tools-> Simulator** and set (check) Simulator to SMARTSPICERF. Version number is set to Default (the latest installed version) in **Version** window. You can define any specific version too.

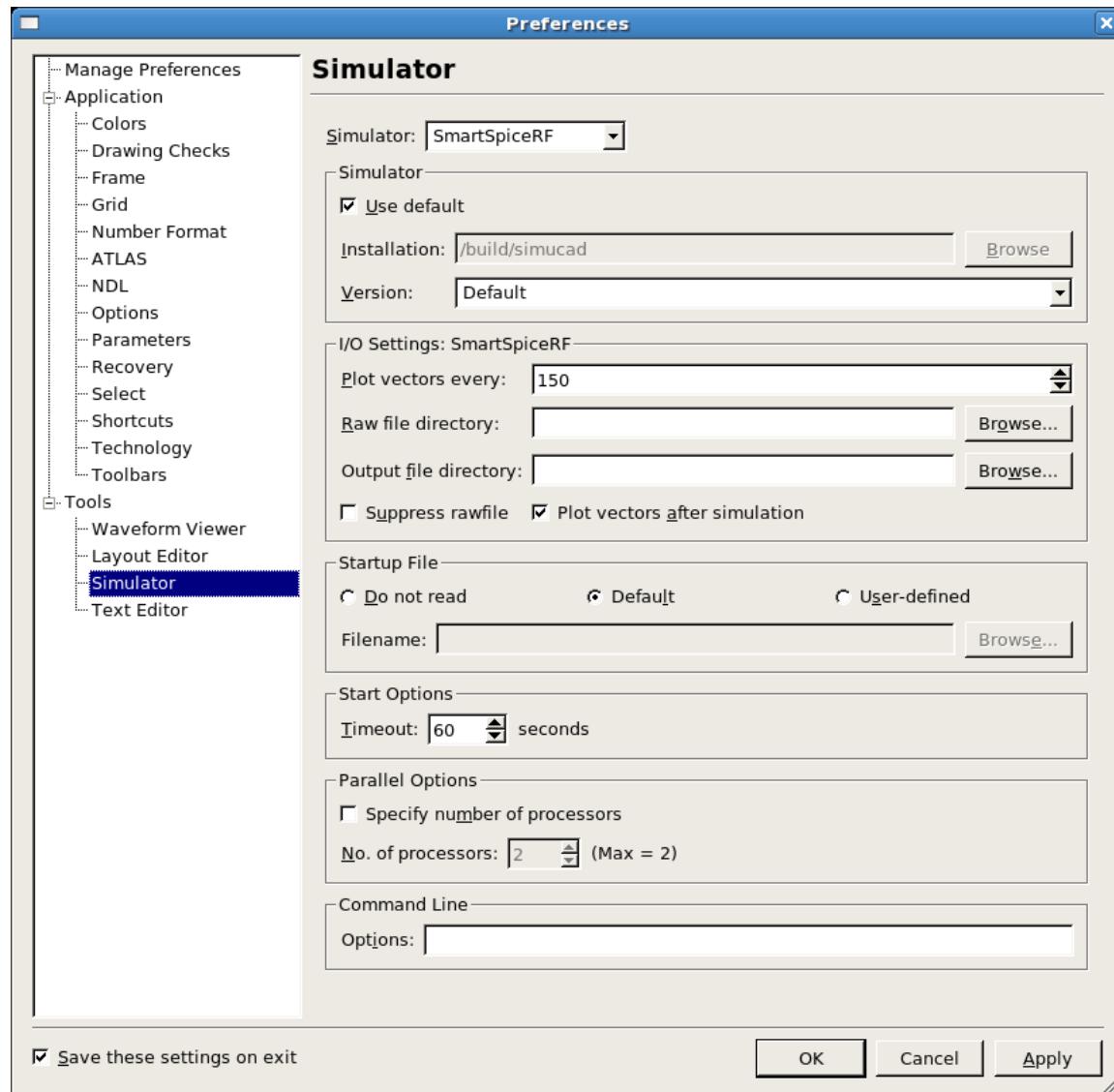


Figure 3: Simulator Preferences Window

2: SmartSpiceRF Simulations

This Tutorial illustrates how SmartSpiceRF can simulate switched-capacitor filter circuits and provide basic measurements.

2.1: Transient Analysis

Transient analysis is provided with testbench file `TB_SC_test.schlr`.

2.1.1: Loading Testbench Schematic

To load the first testbench schematic, select **File→Open→ Schematic**. When the file browser appears, navigate to the `./SC` directory and select the file `TB_SC_test.schlr`. The SCF testbench circuit schematic will appear in the window (Figure 4).

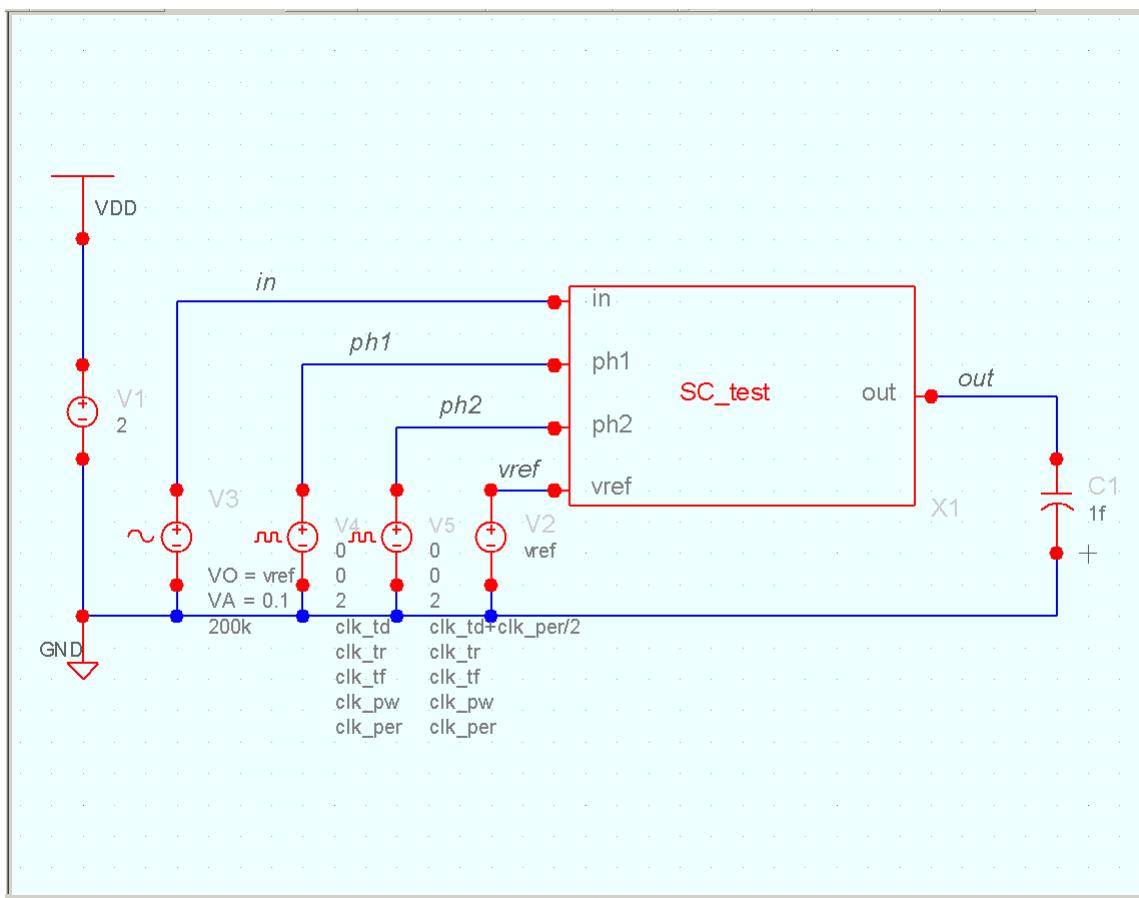


Figure 4: Testbench of Switched-Capacitor Filter for Transient Analysis

2.1.2: Generating Netlists

Netlists are the text files used to describe device connectivity and element properties of the circuit. GATEWAY can create two different netlist formats: SMARTSPICE for circuit simulation, or GUARDIAN for layout design. See the GATEWAY USER'S MANUAL for more information on these formats.

To generate netlist in SMARTSPICE format, either select **Simulation→Create Netlist**, or click on **SmartSpice Netlist** icon (Figure 5) in **Tool Bar**.



Figure 5: Create Netlist Icon

SMARTSPICE netlist then will appear in a new window:

```

File Edit View Bookmarks Spice Help
[Icons] AC DC [ ] [ ] H[ ] N[ ] OP PZ X[ ] F[ ]
* Gateway 2.8.25.R Spice Netlist Generator
* Workspace name: D:\susanz\silvaco\pdk\simucad-radiofrequency-demo\1.1.5.R\gateway\simucad-radiofrequency-d
* Simulation name: D:\susanz\silvaco\pdk\simucad-radiofrequency-demo\1.1.5.R\gateway\examples\sc\TB_SC_test.s
* Simulation timestamp: 31-Jul-2009 17:06:51

* Schematic name: TB_SC_test
C1 out GND lf
V1 VDD GND DC 2
V2 vref GND DC vref
V3 in GND SIN(vref 0.1 200k )
V4 ph1 GND DC 0 PULSE(0 2 clk_td clk_tr clk_tf clk_pw clk_per)
V5 ph2 GND DC 0 PULSE(0 2 "clk_td+clk_per/2" clk_tr clk_tf clk_pw clk_per)
X1 in out ph1 ph2 vref SC_test

* Schematic name: SC_test
.SUBCKT SC_test in out ph1 ph2 vref
M1 NET1 ph1 in GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M2 NET2 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M3 NET3 ph1 NET2 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M4 NET2 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M13 NET13 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M14 NET9 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M15 NET13 ph1 NET3 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M16 out ph1 NET9 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
X1 NET3 vref out OpAmp
XC1 NET2 NET1 mimcap LT=3.2864U M=m_C1
XC2 NET3 out mimcap LT=3.2864U M=m_Cf
XC3 NET13 NET9 mimcap LT=3.2864U M=m_C1
.ENDS SC_test

* Schematic name: OpAmp
.SUBCKT OpAmp inm inp out
I2 VDD NET3 DC 10u
M1 NET2 NET3 GND GND mn W=10U L=0.5u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=10
M2 NET3 NET3 GND GND mn W=10U L=0.5u AS=6.6P AD=6.6P PS=21.32U PD=21.32U M=1
M3 NET1 inp NET2 GND mn W=10U L=0.48u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=20
M4 out inm NET2 GND mn W=10U L=0.48u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=20
M5 out NET1 VDD VDD mp W=10U L=0.48u AS=3.99P AD=3.7P PS=11.798U PD=10.74U M=40
M6 NET1 NET1 VDD VDD mp W=10U L=0.48u AS=3.99P AD=3.7P PS=11.798U PD=10.74U M=40
.ENDS OpAmp

* Global Nodes Declarations
.GLOBAL GND VDD
|
* End of the netlist

```

Figure 6: SCF Testbench Netlist

2.1.3: Control File

SMARTSPICERF needs more than just a circuit netlist to perform a meaningful simulation. It needs Circuit Netlist, Voltage or Current stimulus, Options, Analysis Statements, and active device Model parameters or Libraries. All this information comes together in the form of an Input Deck file (*.in, *.inp, *.cir, *.sp, *.sm, *.scs, etc.), which provides SMARTSPICERF with all needed information to run simulation and generate output data.

Click on the simulation **Tool Bar** icon **Edit Control File** (Figure 7):



Figure 7: Edit Control File Icon

The control file will be opened in the text editor Sedit window. The control file TB_SC_test.ctr (Figure 8) consists of path to SILVACO RF Demo PDK model library file as well as SMARTSPICERF .OP and .TRAN Analyses statements. The control file combined with the circuit netlist and list of vectors to be saved creates an Input Deck for SMARTSPICERF.

```

File Edit View Bookmarks Spice Help
[Icons] AC DC [un] [un] H(s) [N] OP PZ X+1 F{0-4}
|.inc .....\models\sbcd_rf.lib
.OP

.param vref = 1_V
.param clk_td = 1n
.param clk_tr = 1n
.param clk_tf = 1n
.param clk_pw = 45n
.param clk_per = 100n
.param m_C1 = 15
.param m_C2 = 10
.param m_Cf = 30

*>>>>>>>      Simulation Control      <<<<<<<<<<<<<<<<<<
.OPTIONS format savemeasures NOMOD=1 NODECK=1 NOPAGE=1 rawpts=500 numdgt=8 accurate seed=101
.TEMP 27

.TRAN 0.1n 20u

```

Line 1 Col: 1 INS Unix RW SEdit 3.12.2.R © Simucad 2009

Figure 8: TB_SC Control File

Transient analysis statement is shown as follows:

```

*****
.TRAN 0.1n 20u
*****
```

Click on

(**Gateway -> Simulation -> Run**) to begin the SMARTSPICERF simulation.

2.1.4: Simulation Results

Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be open.

Open plot `tran1`, then select vector `v(out)` and click **Plot**. The waveform plot is shown on Figure 9.

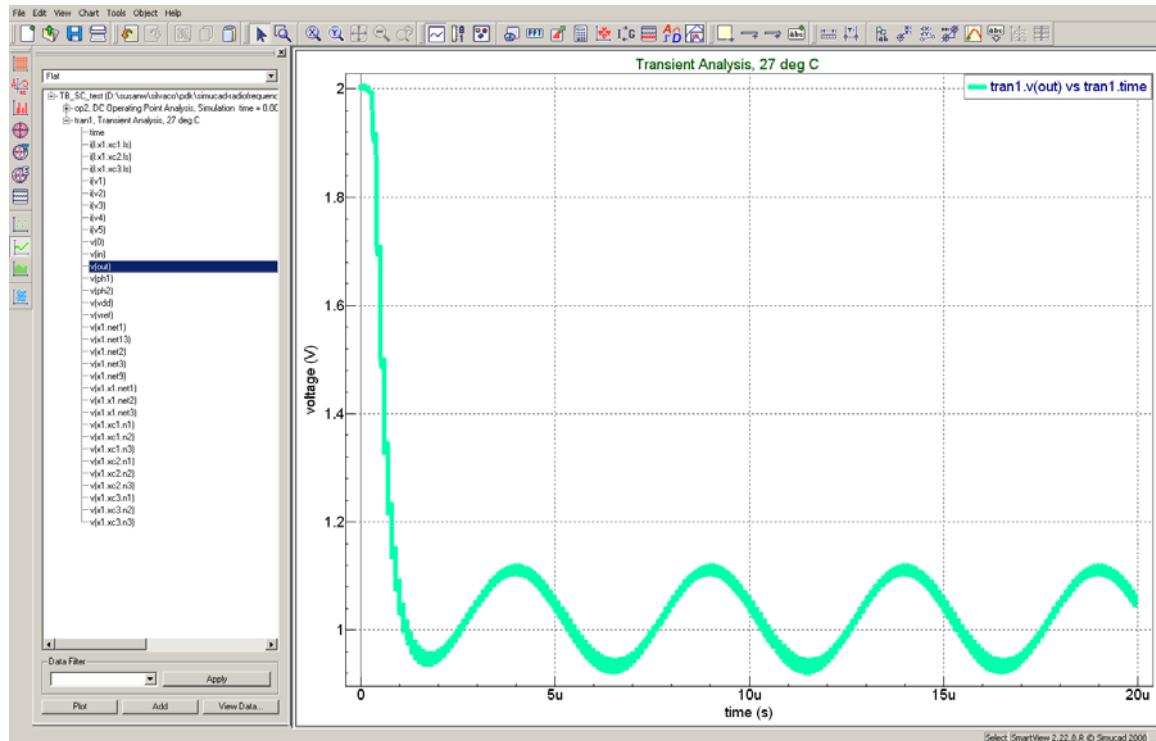


Figure 9: Waveform $v(\text{out})$ Plot

2.2: Transfer Function and Noise Analyses

Transient analysis is provided with testbench file `TB_SC_test_PSS.schlr`.

2.2.1: Loading Testbench Schematic

To load testbench schematic, select **File→Open→ Schematic**. When the file browser appears, navigate to the `./SC` directory and select the file `TB_SC_test_PSS.schlr`. The SCF testbench circuit schematic will appear in the window (Figure 10).

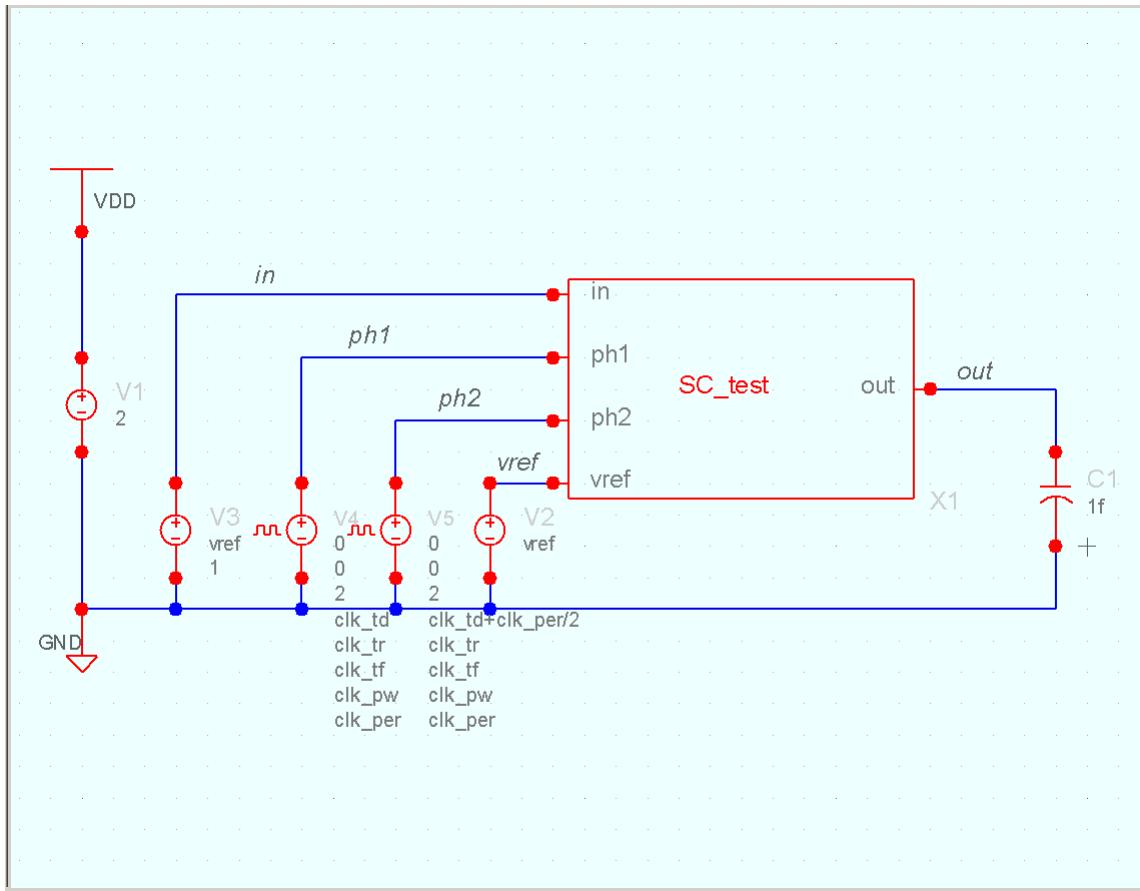
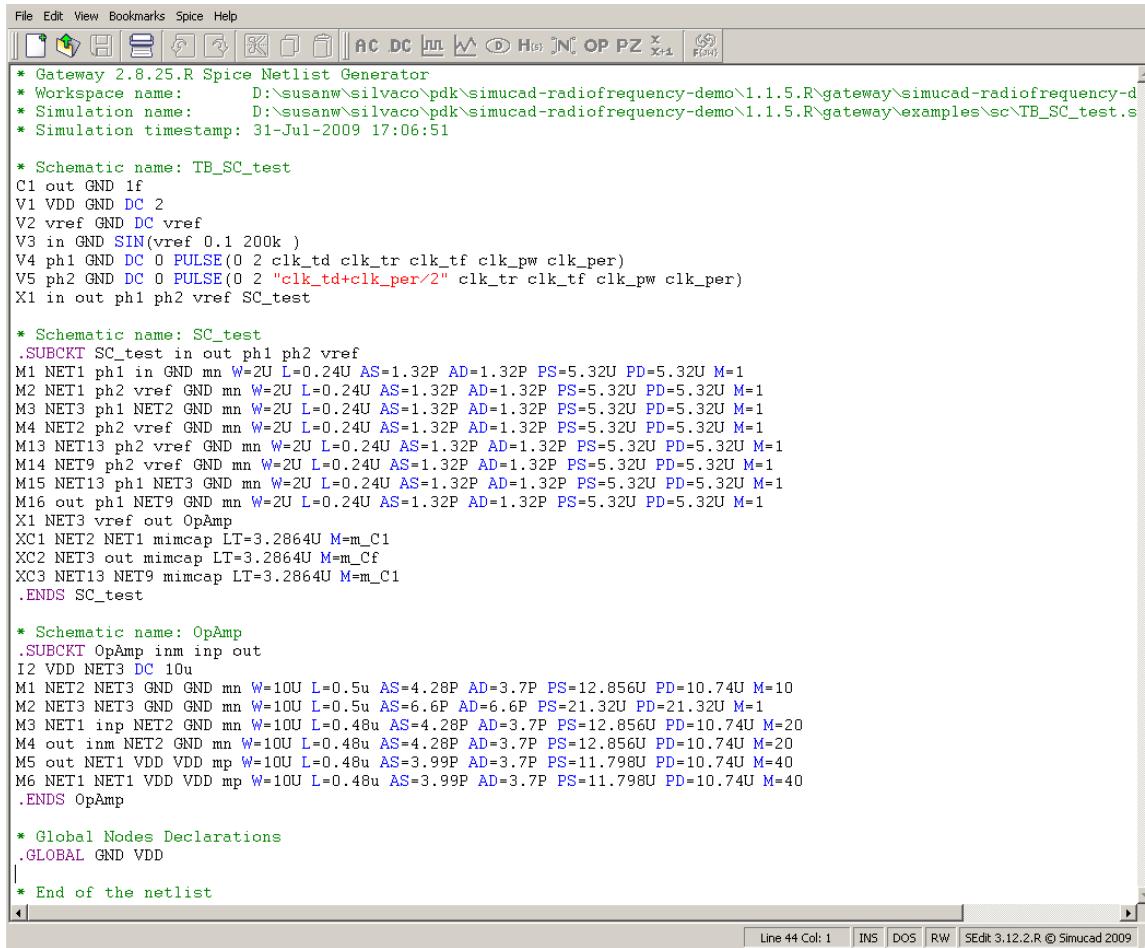


Figure 10: Testbench of Switched-Capacitor Filter for RF Analyses

2.2.2: Generating Netlists

To generate netlist in SMARTSPICE format, either select **Simulation→Create Netlist**, or click on **SmartSpice Netlist** icon (Figure 5) in **Tool Bar**.

SMARTSPICE netlist then will appear in a new window:



```

File Edit View Bookmarks Spice Help
| AC DC [ ] [ ] H[ ] N[ ] OP PZ X[ ] F[ ]
* Gateway 2.8.25.R Spice Netlist Generator
* Workspace name: D:\susanz\silvaco\pdk\simucad-radiofrequency-demo\1.1.5.R\gateway\simucad-radiofrequency-d
* Simulation name: D:\susanz\silvaco\pdk\simucad-radiofrequency-demo\1.1.5.R\gateway\examples\sc\TB_SC_test.s
* Simulation timestamp: 31-Jul-2009 17:06:51

* Schematic name: TB_SC_test
C1 out GND 1f
V1 VDD GND DC 2
V2 vref GND DC vref
V3 in GND SIN(vref 0.1 200k )
V4 ph1 GND DC 0 PULSE(0 2 clk_td clk_tr clk_tf clk_pw clk_per)
V5 ph2 GND DC 0 PULSE(0 2 "clk_td+clk_per/2" clk_tr clk_tf clk_pw clk_per)
X1 in out ph1 ph2 vref SC_test

* Schematic name: SC_test
.SUBCKT SC_test in out ph1 ph2 vref
M1 NET1 ph1 in GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M2 NET1 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M3 NET3 ph1 NET2 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M4 NET2 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M13 NET13 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M14 NET9 ph2 vref GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M15 NET13 ph1 NET3 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
M16 out ph1 NET9 GND mn W=2U L=0.24U AS=1.32P AD=1.32P PS=5.32U PD=5.32U M=1
X1 NET3 vref out OpAmp
XC1 NET2 NET1 mimcap LT=3.2864U M=m_C1
XC2 NET3 out mimcap LT=3.2864U M=m_Cf
XC3 NET13 NET9 mimcap LT=3.2864U M=m_C1
.ENDS SC_test

* Schematic name: OpAmp
.SUBCKT OpAmp inm inp out
I2 VDD NET3 DC 10u
M1 NET2 NET3 GND GND mn W=10U L=0.5u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=10
M2 NET3 NET3 GND GND mn W=10U L=0.5u AS=6.6P AD=6.6P PS=21.32U PD=21.32U M=1
M3 NET1 inp NET2 GND mn W=10U L=0.48u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=20
M4 out inm NET2 GND mn W=10U L=0.48u AS=4.28P AD=3.7P PS=12.856U PD=10.74U M=20
M5 out NET1 VDD VDD mp W=10U L=0.48u AS=3.99P AD=3.7P PS=11.798U PD=10.74U M=40
M6 NET1 NET1 VDD VDD mp W=10U L=0.48u AS=3.99P AD=3.7P PS=11.798U PD=10.74U M=40
.ENDS OpAmp

* Global Nodes Declarations
.GLOBAL GND VDD
|
* End of the netlist

```

Figure 11: SCF RF Testbench Netlist

2.2.3: Control File

Click on the simulation **Tool Bar** icon **Edit Control File** (Figure 7):

The control file will be opened in the text editor Sedit window. The control file TB_SC_test_PSS.ctr (Figure 12) consists of path to SILVACO RF Demo PDK model library file as well as SMARTSPICERF .OP, .HAC, and HNOISE Analysis statements. The control file combined with the circuit netlist and list of vectors to be saved creates an Input Deck for SMARTSPICERF.

```

File Edit View Bookmarks Spice Help
New Open Save Print Find Copy Paste Undo Redo AC DC HAC OP PZ X+1 F0-1
.inc ..\models\sbcd_rf.lib
.OP

.param vref = 1_V
.param clk_td = 1n
.param clk_tr = 1n
.param clk_tf = 1n
.param clk_pw = 45n
.param clk_per = 100n
.param m_C1 = 15
.param m_C2 = 10
.param m_Cf = 30

*>>>>>> Simulation Control <<<<<<<<<<<<<<<
.OPTIONS format savemeasures NOMOD=1 NODECK=1 NOPAGE=1 rawpts=500 numdgt=8 accurate seed=101
.TEMP 27

.HAC $ PSS AC Analysis
+ DEC 10 100Hz 5MegHz $ Frequency sweep Number of points per decade=100 fstart=1 fstop=10Mhz
+ fund=10MHz $ Fundamental frequency
+ nharm=10 $ Maximum number of computed harmonics
+ SB(0) $ Set of n integer numbers representing the sidebands
+ KMAX=20 $ Ignores noise at frequencies <= fout - Kmax x fund and >= fout + Kmax x fund
+ method=shooting $ Steady-state analysis by SHOOTING method. A must be for SC circuits
+ tstep=10n $ Time step to define timestep control parameters
+ tstab=10u $ Time of Initial transient analysis to let the circuit settle
+ annotate=4 $ Prints information in the output file
+ waves $ Generate time domain OP waveforms

.HNOISE V(out) V(in)
+ DEC 10 100Hz 5MHz
+ fund=10MHz
+ nharm=10
+ SB(0)
+ KMAX=20
+ method=shooting
+ tstep=10n
+ tstab=10u
+ annotate=4
+ waves

```

Line 1 Col: 1 INS Unix RW SEdit 3.12.2.R © Simucad 2009

Figure 12: SCF RF Testbench Control File

Transfer function analysis statements are shown as follows.

```
*****
.HAC
+ DEC 10 100Hz 5MegHz
+ fund=10MHz
+ nharm=10
+ SB(0)
+ KMAX=20
+ method=shooting
+ tstep=10n
+ tstab=10u
+ annotate=4
+ waves
*****
```

.HNOISE analysis statement to calculate output noise is shown as follows.

```
*****
.HNOISE V(out) V(in)
+ DEC 10 100Hz 5MHz
+ fund=10MHz
+ nharm=10
+ SB(0)
+ KMAX=20
+ method=shooting
+ tstep=10n
+ tstab=10u
+ annotate=4
+ waves
*****
```

Click on  (**Gateway -> Simulation -> Run**) to begin the SMARTSPICERF simulation.

2.2.4: Simulation Results

SmartSpiceRF outputs simulation results of Large-Signal PSS, Small-Signal HAC and NOISE analyses, and statistic information into Output window and creates number of plots with Waveforms, Spectra, Measurement results, etc. Simulation results in form of plots will be loaded into SMARTVIEW, and SMARTVIEW **Data Browser** window will be open.

Open plot hac1, then select vector $v(out)$, and click **Plot**.

The transfer function is shown in Figure 13.

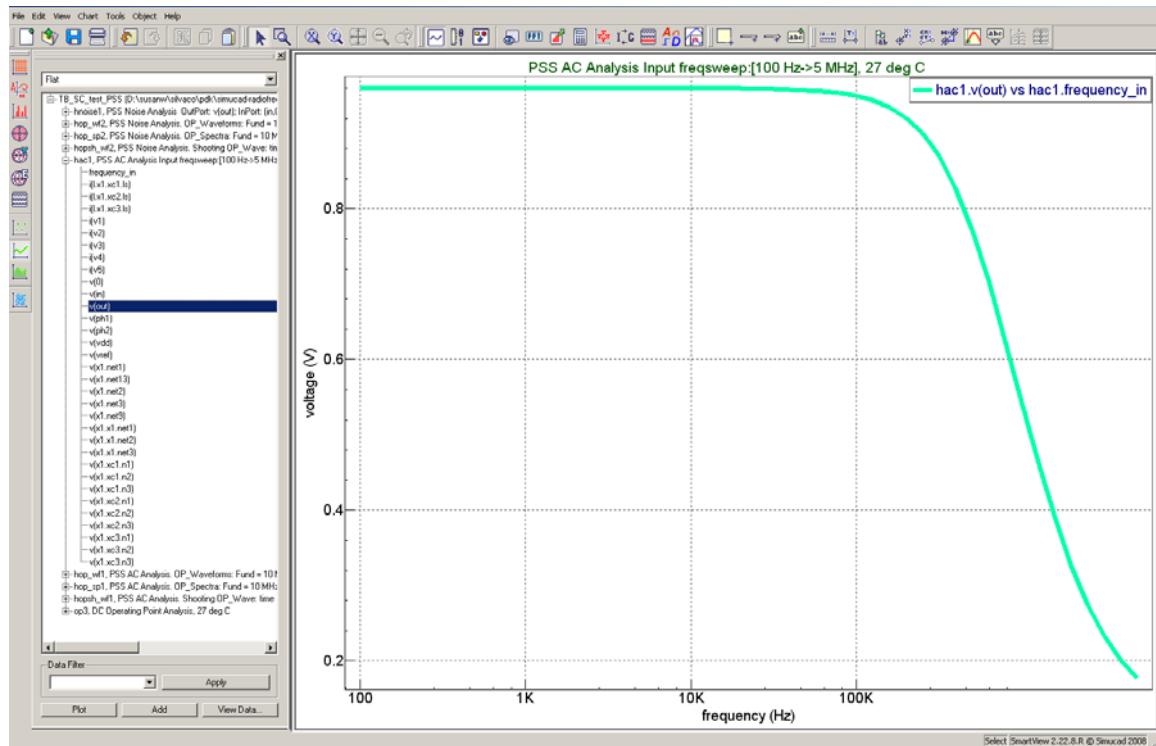


Figure 13: Transfer Function Plot

Open plot hnoise1, then select vector onoise_s, and click Plot.

The output noise plot is shown in Figure 14.

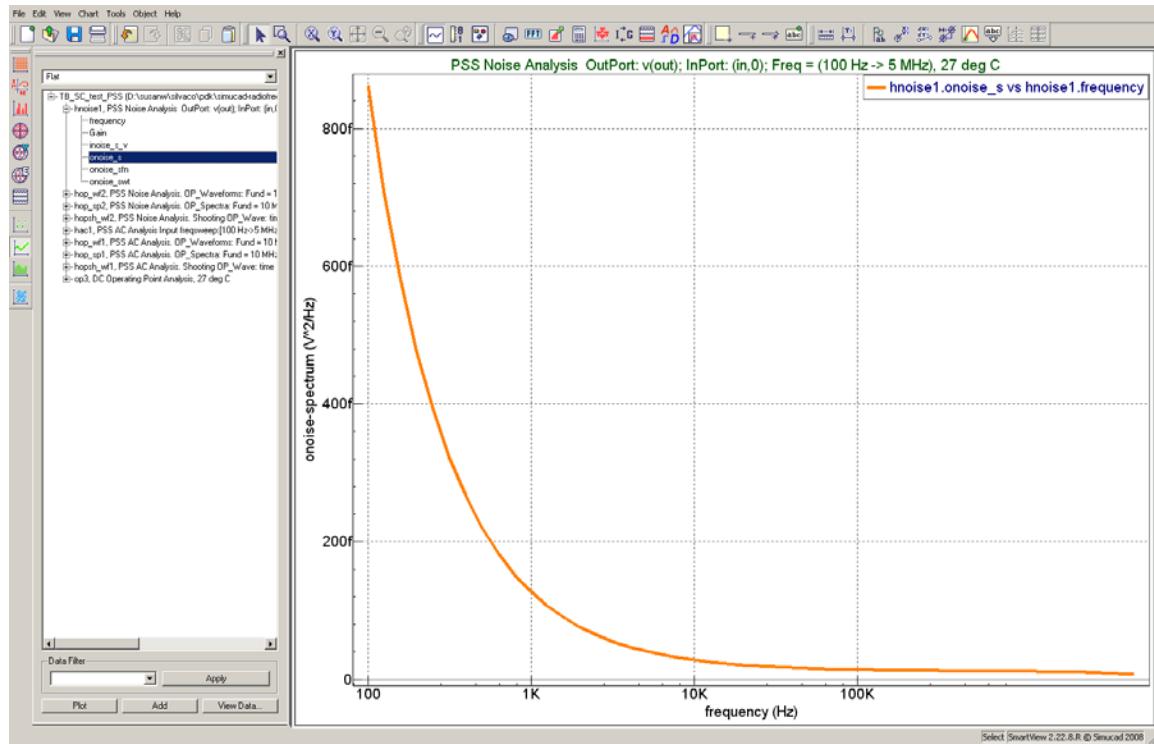


Figure 14: Output Noise Plot