AC analysis of switched capacitor filters in SPICE-family programs

Dalibor Biolek[1, 2], Viera Biolkova[3], and Zdenek Kolka[3]

[1] University of Defence, Dept. of EE, Kounicova 65, 662 10 Brno, Czech Republic
[2] Brno University of Technology, Dept. of Microelectronics, Technicka 10, 616 00 Brno, Czech Republic
[3] Brno University of Technology, Dept. of Radio Electronics, Technicka 12, 616 00 Brno, Czech Republic
dalibor.biolek@unob.cz
{biolkova, kolka}@feec.vutbr.cz

Abstract. The paper describes a method for the analysis of the frequency characteristics of switched-capacitor circuits in SPICE-family programs, which do not provide a direct AC analysis of such networks. The method starts from the z-domain charge equations of switched circuit, which are modified to current equations. Based on these equations, macromodels of capacitor and operational amplifier are constructed and implemented as SPICE subcircuits. The method generates correct results for two-phase switched circuits containing ideal switches and ideal transforming cells such as operational amplifiers. For a convenient drawing of the model via schematic editor, special schematic symbols of the components are designed such that the user can define how the nodes are interconnected by switches during each switching phase, and also select the input and output samples for a definition of the corresponding frequency characteristics.

Keywords: Switched capacitor, SC, AC analysis, frequency characteristic, SPICE.

1 Introduction

The methods of “Hand-and-Paper” (flow graph and matrix) and particularly computer-aided analyses of switched-capacitor filters experienced explosive development in the nineteen eighties and nineties [1]. However, the very first problem was a fundamental limitation of SPICE-family programs, which do not enable a direct analysis of the frequency responses of circuits employing periodically operated switches via a classical AC analysis [2]. That is why special programs for such analyses were developed [3]. As another possibility, the in-built SPICE analyses can be utilized for indirect computation of the frequency responses, namely either via repeated transient analyses for various frequencies of the sinusoidal excitation with subsequent sensing of the amplitude and the initial phase of the response [4] or by no less arduous preparation of a substitution model of the circuit, constructed from the original schematic of the switched filter [5]. The above procedures can be combined and automated. However, it is necessary to build a special software interface of the classical simulation program [6]. An attempt to build the model of the switched circuits directly in the schematic editor of SPICE-like program has been published in [7]. The schematic symbol of each component was doubled for each switching phase. The drawback consisted in the fixed distance between the replicated parts, which is a source of irresolvable problems when creating models of complicated circuits.

We propose below a simple method for the frequency analysis of switched capacitor circuits by an arbitrary SPICE-family program which is equipped with a schematic editor for drawing the model. The schematics of the components are drawn such that the corresponding pins are split into pairs for two switching phases. The user interconnects them via wires according to how these nodes are really interconnected by switches in the original circuit. In the opinion of the authors, this is an optimum procedure of constituting a switched circuit in the environment of conventional circuit simulating program which can result in a correct analysis of the frequency responses in standard AC operation. All other procedures aimed at increasing the user's comfort while compiling the model would amount to either a direct modification of the program or developing its interface environment according to [6].

2 Capacitor in switched circuit as a four-terminal device

In circuits with ideal switches, i.e. switches with zero-R_{ON} and with infinitely short switching times, it is necessary to consider the so-called inconsistent initial conditions [8], when, after interconnecting two nodes by an ideal ON-state switch, the current is a Dirac impulse. As a consequence, the capacitor voltages can vary in a discontinuous way. That is why the capacitor inside such two-phase switched circuits should be considered a four-terminal element with four generally independent nodal voltages according to Fig. 1 (a). The switches 1 and 2 symbolize the time-domain multiplex in the circuit according to Fig. 1 (b), formed by the sequence of switching states 1 and 2 with potential discontinuities of the circuit variables at switching instants. Let us consider hereinafter the switching regime with the switching period T and the duty cycle $D = 1 – D'$, where typically $D = 0{,}5$.

The individual symbols of the circuit quantities in Fig. 1 (a) denote the following:

v_c: instantaneous capacitor voltage;
v^1 or v^2: instantaneous capacitor voltage in the time-domain multiplex 1 or 2;
q^1 or q^2: electric charge conveyed via the gate "1" or "2" into the capacitor during the phase 1 or 2;
$<i>^1$ or $<i>^2$: average current flowing via the gate " 1" or "2" computed within the switching period, thus

$$< i >^1 = \frac{q^1}{T},\; < i >^2 = \frac{q^2}{T}\,. \tag{1}$$

Then the following equation holds for the charge conveyed into the capacitor within the switching phase 1 from the time instant kT of switching from phase 2 to phase 1 until the end of phase 1 $kT+DT^-$, where the symbol – denotes the left-side limit, i.e. just before the subsequent commutation to phase 1:

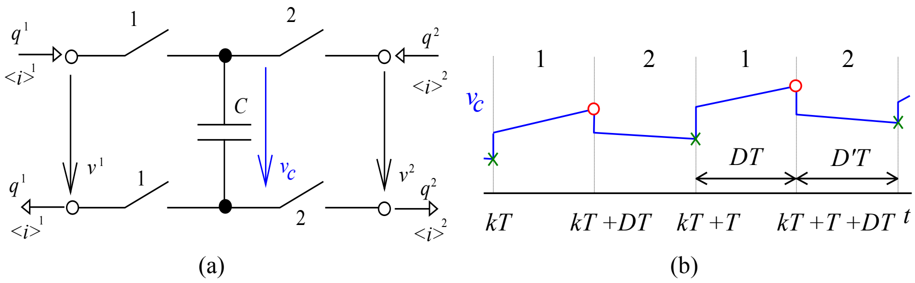


Fig. 1. (a) Capacitor model in two-phase switched circuit, (b) switching multiplex with marked limit values of voltages at the ends of switching phases 1 (o) and 2 (x).

$$q^1(kT+DT) = C[v^1(kT+DT^-) - v^2(kT^-)]. \tag{2}$$

An analogous equation holds for the charge conveyed into the capacitor at switching phase 2 from the time instant $kT+DT$ until the instant $kT+T$ of the end of phase 2:

$$q^2(kT+T) = C[v^2(kT+T^-) - v^1(kT+DT^-)]. \tag{3}$$

Before implementing the equations into SPICE, the charge must be converted into the current according to Eq. (1). The modified equations will be in the form

$$<i>^1 = \frac{C}{T}[v^1(kT+DT^-) - v^2(kT^-)],\ <i>^2 = \frac{C}{T}[v^2(kT+T^-) - v^1(kT+DT^-)] \tag{4}$$

For the AC analysis in SPICE, the above equations can be used after their z-domain transform, where $z = \exp(j\omega T)$ [7]:

$$I^1 = \frac{C}{T}[V^1 - V^2 z^{-D}],\ I^2 = \frac{C}{T}[V^2 - V^1 z^{-D'}] \tag{5}$$

Figure 2 shows the proposed schematic symbol of the capacitor in the switching regime, distinguishing between voltages in phases 1 and 2, and the corresponding macromodel resulting from Eq. (5). Example of the PSpice subcircuit is below (note that $f_s = 1/T$ is the switching frequency):

```
*capacitor
.subckt SCapacitor 1p1 2p1 1p2 2p2 params: C=10p fs=100k D=0.5
R1 1p1 2p1 {1/(C*fs)}
G1 2p1 1p1 LAPLACE {V(1p2,2p2)*C*fs} {exp(-s*D/fs)}
R2 1p2 2p2 {1/(C*fs)}
G2 2p2 1p2 LAPLACE {V(1p1,2p1)*C*fs} {exp(-s*(1-D)/fs)}
.ends SCapacitor
```

If one of the capacitor terminals is permanently grounded, then the model and also the schematic symbol can be simplified.

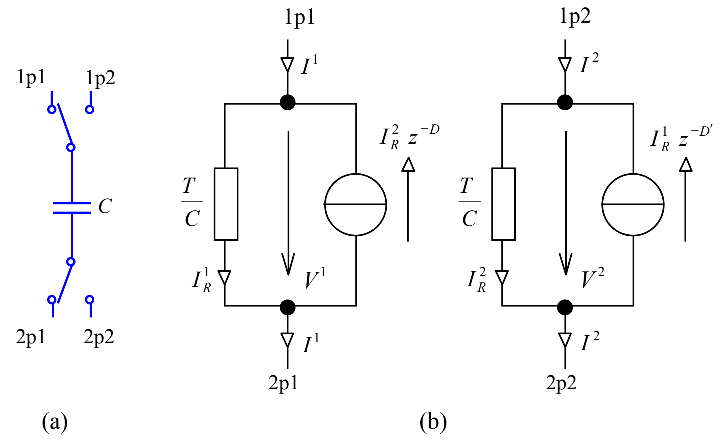


Fig. 2. (a) Schematic symbol of capacitor as a four-terminal device in circuit with two-phase switching, containing pseudo-switches which symbolize voltage multiplexing, (b) *z*-domain AC model of the capacitor. The node notation 1p2 means: node No. 1 at phase No. 2.

3 Ideal OpAmp in switched circuit as a six-terminal device

The above technique of node splitting can also be applied to ideal differential-input operational amplifier (OpAmp) with the model

$$V_{out} = A V_d \,, \tag{6}$$

where V_d is the difference voltage and A is the finite frequency-independent gain. Such an OpAmp behaves in circuits with two-phase switching as a six-terminal device. As is obvious from the model in Fig. 3, there are two independent OpAmp models for the switching phases 1 and 2. This follows from the fact that these models are non-inertial. If the input node is permanently grounded, one can use a simplified model (see Fig. 4 in the next section as an example).

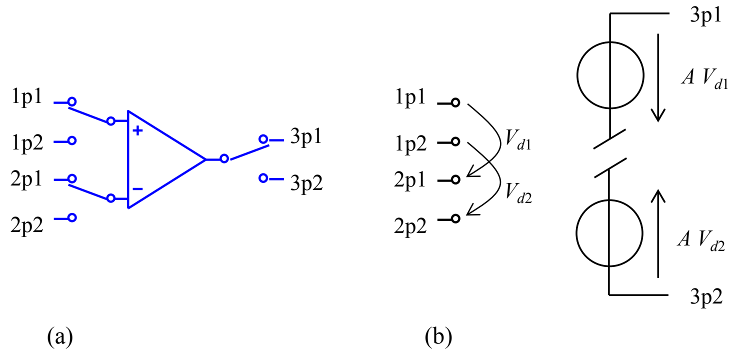


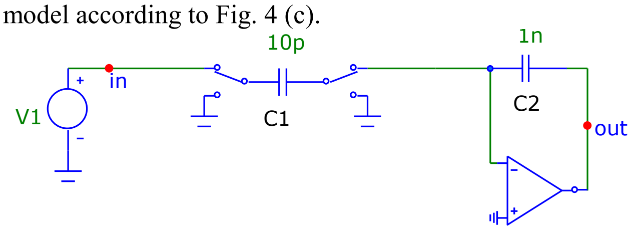
Fig. 3. (a) Schematic symbol of OpAmp as a six-terminal device in circuits with two-phase switching, (b) AC model, made up of an independent couple of models for each switching phase.

The corresponding PSpice code of the model is given below.

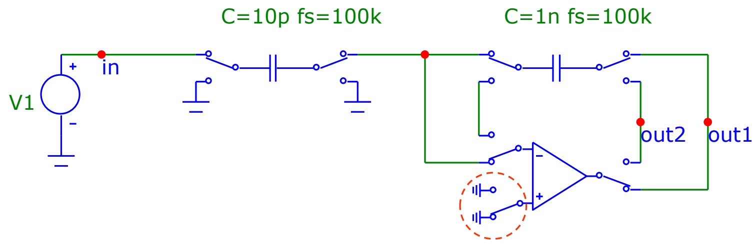
```
*ideal OpAmp
.subckt SCOpAmp 1p1 2p1 3p1 1p2 2p2 3p2
+ params:  A=200k
E1 3p1 0 value={A*V(1p1,2p1)}
E2 3p2 0 value={A*V(1p2,2p2)}
.ends SCOpAmp
```

4 A demonstration of utilizing the models

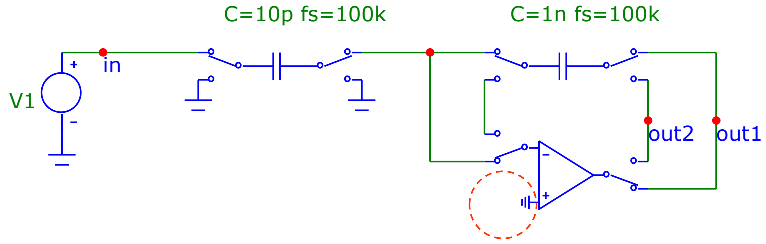
The schematic of a switched capacitor integrator and its AC model are shown in Figs 4 (a) and (b), respectively. The switches in the model of the capacitor C2 and also the switches belonging to the OpAmp are fictive. On the other hand, the fictive switches around C1 correspond to the real switches in Fig. 4 (a). Since the non-inverting input is permanently grounded, it is possible to use a simplified OpAmp model according to Fig. 4 (c).



(a)



(b)



(c)

Fig. 4. (a) Switched-capacitor integrator, (b) its AC model, c) simplified AC model.

The results of the classical SPICE analysis of the AC model from Fig. 4 (c) are shown in Fig. 5. The amplitude frequency responses are identical for samples selected in phases 1 and 2. The phase responses for switching phases 1 and 2 correspond, according to the theory [1], to the BD and LDI transforms, respectively.

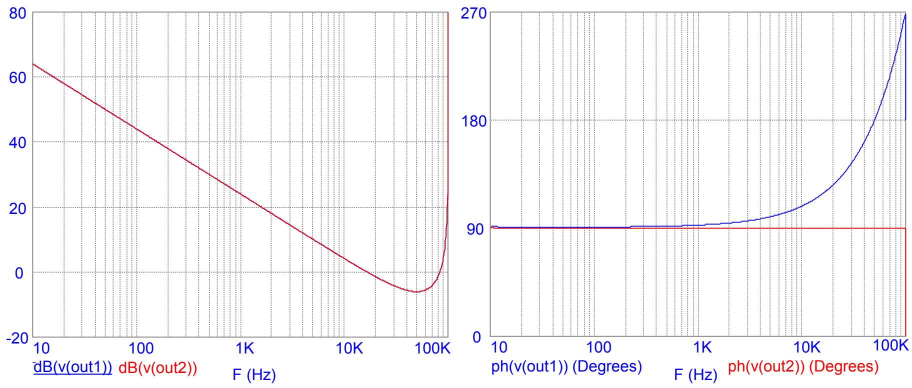


Fig. 5. AC analysis of switched-capacitor integrator from Fig. 4. The phase frequency response at output out1 (samples in phases 1), linear for the linear frequency axis, corresponds to BD integrator (Backward-Difference). The phase shift is zero at output out2 (samples in phases 2), which corresponds to LDI integrator (Lossless Discrete Integration).

References

1. Unbehauen, R., Cichocki, A.: MOS switched-capacitor and continuous-time integrated circuits and systems. Springer-Verlag, 1989
2. Vlach, J., Singhal, K.: Computer Methods for Circuit Analysis and Design. Van Nostrand Reinhold Company, New York, 1987
3. Valsa, J., Vlach, J.: SWANN - A Program for Analysis of Switched Analog Nonlinear Networks. In Proc. of ISCAS 1995, IEEE, 1995, 1752--755
4. Bičák, J., Hospodka, J.: Frequency response of switched circuits in SPICE. In Proc. of ECCTD'03, Krakow, IEEE, 2003, I-333--336
5. Nelin, B. D.: Analysis of Switched-Capacitor Networks Using General-Purpose Circuit Simulation Programs. IEEE Trans. on CAS 30(1), 43--48 (1983)
6. Biolek, D., Kadlec, J., Biolkova, V., Kolka, Z.: Interactive command language for OrCAD PSpice via Simulation Manager and its utilization for special simulations in electrical engineering. WSEAS Trans. on Electronics 5(5), 186--195 (2008)
7. Biolek, D., Biolková, V., Kolka, Z.: AC Analysis of Idealized Switched-Capacitor Circuits in Spice-Compatible Programs. In Proc. of CSCC07, Crete, 2007, 222—226
8. Opal, A., Vlach, J.: Consistent initial conditions of linear switched networks. IEEE Trans. on CAS 37(3), 364--372 (1990)

Acknowledgments. This work has been supported by the Technology Agency of the Czech Republic under grant agreement No. TA04011279, and by the Project for the development of K217 Department, UD Brno, Czech Republic. Research described in this paper was also financed by Czech Ministry of Education in frame of National Sustainability Program under grant LO1401.
For research, infrastructure of the SIX Center was used.