

AC analysis of real circuits with external switching in PSpice

DALIBOR BIOLEK^{1,2}, VIERA BIOLKOVÁ³, ZDENĚK KOLKA³

¹ Dept. of EE, University of Defense, Brno, Czech Republic

² Dept. of Microelectronics, Brno University of Technology, Czech Republic

³ Dept. of Radio Electronics, Brno University of Technology, Czech Republic
dalibor.biolek@unob.cz <http://user.unob.cz/biolek>

Abstract: The inability to provide direct small-signal AC analysis of circuits with periodical external switching belongs to well-known limitations of Spice-compatible programs. This paper refers to novel methods of modeling which enable the above analyses also in PSpice simulators with the aid of the so-called Simulation Manager.

Keywords: - PSpice, simulation, analysis, circuit.

1 Introduction

A direct small-signal AC analysis of switched-capacitor (SC), switched-current (SI) and other circuits with periodically controlled analog switches belong to well-known limitations of Spice-compatible programs.

A method of direct AC analysis of idealized two-phase SC filters in PSpice has been described in [1]. The frequency responses are acquired neither via the repeated TRANSIENT analysis nor by the method of multi-tone excitation [2], but through a direct application of the conventional AC analysis to a special model of switched circuit.

AC analysis of general linear switched circuits while taking into consideration influences of real phenomena, e.g. nonzero switch on-resistances, parasitic inductances, frequency dependent OpAmp gains, etc., is hardly applicable in Spice-compatible programs. The method described in [1] is based on the assumption of immediate changes of capacitor voltages at the switching instants. That is why it cannot be used for such cases when the lengths of transient phenomena caused by switching processes cannot be neglected. The new method presented below can be used only on the assumption that the PSpice features will be extended by the utilization of the so-called Simulation Manager (SiM) [3]. SiM is a special program which controls – on the basis of the so-called Manager Control File (MCF) – the operation of OrCAD PSpice program in the sequential mode. In other words, PSpice is run repeatedly for various types of analyses, when the results generated by a given analysis serve as input data for subsequent analyses [3].

The paper is confined to circuits with two-phase switching. However, the method described is easily applicable also to circuits with multiphase switching.

2 Conception of AC analysis of switched circuits in PSpice

Consider a linear switched circuit with two-phase switching, i.e. a circuit that can be modeled by a pair of linear circuits, separately for switching phases 1 and 2. Let the lengths of switching phases 1 and 2 be denoted T_1 and T_2 , respectively. Their sum is equal to the switching period $T=1/F_s$, where F_s is the switching frequency.

Exclusion of the so-called inconsistent initial conditions (IIC) [4] is a basic assumption of PSpice simulation of real switched circuits. The IIC can arise in the case of idealized modeling, e.g. when two capacitors with different initial voltages are connected in parallel by an ideal switch with zero on-resistance. Accepting this assumption is a necessary consequence of the fact that the internal algorithms of PSpice cannot resolve numerical problems which are associated with the IIC. One can easily avoid the IIC, e.g. by defining nonzero on-resistances of all the switches inside the circuit.

Let us define state variables within each switching phase of the circuit such that they are continuous in time at instants between the switching phases. Let the vectors of such state variables for phases 1 and 2 be denoted $\mathbf{x}_1$ and $\mathbf{x}_2$, respectively. In the case of avoiding the IIC, the vectors of state variables can be compounded of capacitor voltages and inductor currents.

As shown in [5], the behavior of real switched networks is also affected by variations of the input signals within the relatively short switching phases. However, when the real phenomena are not extremely significant, then such an influence on the frequency

response can be neglected, which substantially simplifies the subsequent computer analysis. This neglecting is implemented by introducing the assumption that the input signal v_{in} is of the Sample-Hold (SH) character [5], e.g. with discontinuities at switching instants. The consistence of initial conditions and the continuity of state variables should be ensured while modeling the input gate of the switched circuit.

Under the above assumptions, the switched circuit can be described in each switching phase by linear equations (1) and (2):

End of switching phase No. 1 at time $t = kT + T_1$, $k = 0, 1, 2, \dots$

$$\mathbf{x}_1(kT + T_1) = \mathbf{A}_1 \mathbf{x}_2(kT) + \mathbf{B}_1 v_{in}(kT + T_1). \quad (1)$$

End of switching phase No. 1 at time $t = kT + T$, $k = 0, 1, 2, \dots$

$$\mathbf{x}_2(kT + T) = \mathbf{A}_2 \mathbf{x}_1(kT + T_1) + \mathbf{B}_2 v_{in}(kT + T), \quad (2)$$

where $\mathbf{A}_1$, $\mathbf{A}_2$, $\mathbf{B}_1$, and $\mathbf{B}_2$ are the matrices/vectors whose elements depend on the character of transient phenomena in the circuit within the corresponding switching phases.

Utilizing the theory of generalized transfer functions [5], equations (1) and (2) can be converted to the z-domain:

$$\mathbf{X}_1 = \mathbf{A}_1 \mathbf{X}_2 z^{-T_1/T} + \mathbf{B}_1 V_{in,1}, \quad (3)$$

$$\mathbf{X}_2 = \mathbf{A}_2 \mathbf{X}_1 z^{-T_2/T} + \mathbf{B}_2 V_{in,2}, \quad (4)$$

where $\mathbf{X}_1$, $\mathbf{X}_2$, $V_{in,1}$, and $V_{in,2}$ are the z-transforms of signals $\mathbf{x}_1$, $\mathbf{x}_2$, v_{in} , sampled at time instants in which the switching phases 1 or 2 are terminated.

We can conclude that the AC analysis of the switched circuit should be accomplished in the following consecutive steps:

1) Computing the elements of matrices and vectors $\mathbf{A}_1$, $\mathbf{A}_2$, $\mathbf{B}_1$ and $\mathbf{B}_2$.

2) AC analysis of equations (3) and (4), utilizing the well-known substitution $z = \exp(j\omega T)$.

A possible method of computing the $\mathbf{B}_1$ vector implies from Eq. (1): The conventional TRANSIENT analysis is executed during switching phase No. 1 on the assumption of $v_{in} = 1V$ and under zero initial condition $\mathbf{x}_2(kT)$. Then the vector $\mathbf{x}_1$ at the end of this analysis will contain the elements of vector $\mathbf{B}_1$.

When the TRANSIENT analysis of circuit in phase 1 is performed under the condition of $v_{in} = 0$ and with state variable No. i being set to one, then the vector $\mathbf{x}_1$ at the end of this analysis will contain the elements of column No. i of matrix $\mathbf{A}_1$.

An analogous procedure can be repeated for phase No. 2 in order to compute vector $\mathbf{B}_2$ and matrix $\mathbf{A}_2$.

After computing the above vectors and matrices, item 2) will be performed via behavioral modeling of equations (3) and (4) and the following AC analysis.

The sequence of PSpice simulation tasks can be described as follows:

For $k=1..2$

**computing the $\mathbf{B}_k$ vector*

- Modeling the circuit within phase No. k , $v_{in}=1V$, zero initial conditions.
- TRANSIENT analysis till the time T_k .
- Reading the values of state variables and saving them to the $\mathbf{B}_k$ vector.

**computing the $\mathbf{A}_k$ matrix*

- Modeling the circuit within phase No. k , $v_{in}=0V$, zero initial conditions.

For $i=1..N$; N is the number of state variables

- Setting state variable No. i to one.
- TRANSIENT analysis till the time T_k .
- Reading the values of state variables and saving them to the i th column of $\mathbf{A}_k$ matrix.

end

end

It should be noted that PSpice cannot provide the above algorithm independently, without the user's interventions. That is why a demonstration of the cooperation between PSpice and the so-called Simulation Manager (SiM) [3] is described. SiM should provide an automated run of the analyses according to the above algorithm. The user will write this algorithm into the Manager Control File (MCF), which is a generalization of the well-known PSpice circuit file.

3 Demonstration of AC analysis of Sample-Hold circuit

A simple model of the Sample-Hold circuit is shown in Fig. 1 together with the waveforms of input and output voltages and switching impulses. Also the so-called equivalent signals v^{1e} and v^{2e} are indicated here. They represent the continuous-time equivalents of discrete-

time signals at time instants at the ends of switching phases 1 and 2, respectively [5]. The frequency responses for the selection of output samples in phase 1 or 2, computed in AC analysis, are defined by the frequency dependence of the magnitude ratios and the differences of initial phases of the corresponding equivalent signal and the input signal.

We select the only state variable in the circuit, i.e. the capacitor voltage v . The matrices and vectors of **A** and **B** types in equations (1)-(4) are then reduced to scalars a_1 , a_2 , b_1 , and b_2 . Since the switch in Fig. 1 separates the entire circuit from the input signal during phase 2, the relation $b_2 = 0$ holds and thus there is no need to compute this quantity.

A list of the MCF is given below. The individual lines are numbered for clarity (these numbers are not part of the MCF). The syntax details of the SiM are described in [3], [6].

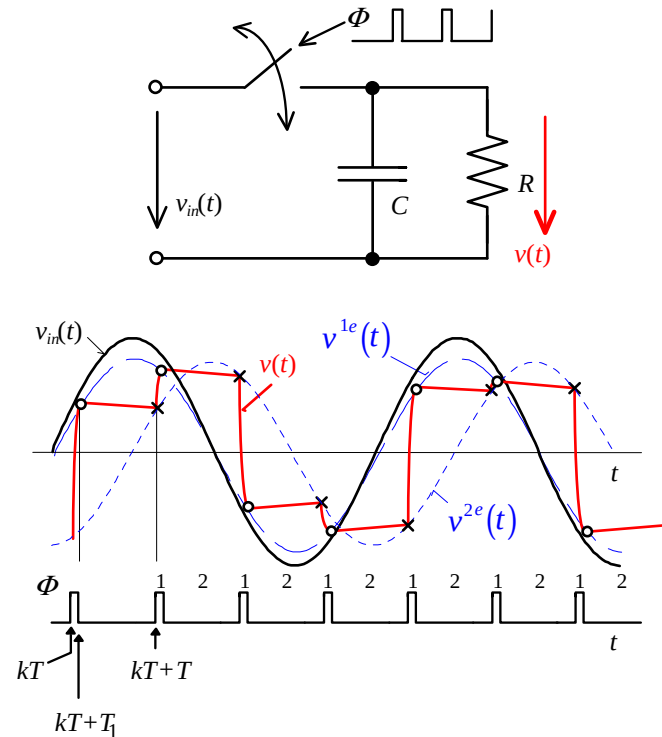


Fig. 1: Model of SH circuit and a demonstration of circuit waveforms.

```

1: *AC analysis of Sample-Hold circuit
2: #set Ron 5k fs 100k T1 0.1/fs T2 1/fs-T1
3: #beginnet SH1
4: Ron 1 2 #Ron$
5: Rs 2 3 10m
6: C 3 0 1n
7: Rz 2 0 100k
8: #endnet

```

```

9: #beginnet SH2
10: Rs 2 3 10m
11: C 3 0 1n
12: Rz 2 0 100k
13: #endnet
14: #defsim tran1 .TRAN 0 #T1$ 0 #T1/100$ skipbp
15: #defsim tran2 .TRAN 0 #T2$ 0 #T2/100$ skipbp
16: #defsim AC .AC dec 100 10 #fs*2$
17: #assemblycir run1.cir
18: Vin 1 0 1V
19: #use SH1
20: #runsim tran1
21: #endassembly
22: #getprobe b tran1 V(3) #T1$
23: #assemblycir run2.cir
24: Vin 1 0 0V
25: #use SH1
26: .IC V(3) 1V
27: #runsim tran1
28: #endassembly
29: #getprobe a1 tran1 V(3) #T1$
30: #assemblycir run3.cir
31: #use SH2
32: .IC V(3) 1V
33: #runsim tran2
34: #endassembly
35: #getprobe a2 tran2 V(3) #T2$
36: #assemblycir run4.cir
37: Vin 1 0 AC 1
38: Ec1 c1 x
+LAPLACE {V(c2)} {#$a1$*exp(-s*#T1$)}
39: Ex x 0 value={V(1)*#b$}
40: Ec2 c2 0
+LAPLACE {V(c1)} {#$a2$*exp(-s*#T2$)}
41: #runsim AC/nocsdf
42: #endassembly

```

The MCF starts by a header (line 1). The definition of variables via #set command is on line 2. This command is an analogy with the PSpice command .param and serves for the definition of the Manager global variables. These variables can form formulas, which are bordered by a pair of symbols \$, and SiM converts the formulas to numerical values. The SiM commands start with the # symbol and they can be combined with the PSpice commands and syntax. The subcircuits SH1 (SH2), defined within the lines 3 and 8 (9 and 13), model the SH circuit as two linear circuits at phases 1 and 2. Three types of analyses are defined on lines 14 to 16. They will be used later in the frame of automatically generated PSpice input files: the TRANSIENT analysis within switching phase 1 or 2 (line 14 or 15), and the AC analysis within the frequency range from 10Hz to the double of switching

frequency, i.e. to 200kHz (line 16). The commands for the generation of PSpice input file RUN1.CIR are on lines 17 to 21. This circuit file is for the transient analysis of SH circuit within switching phase 1, for $v_{in} = 1V$ and zero initial conditions. After executing the command on line 21, this circuit file is created, the simulator is run automatically and the corresponding analysis is performed. The value of output voltage at time T1, i.e. at the end of the analysis, is saved to variable b (see line 22). This variable is labeled as b1 in equations 1 and 3. The other circuit file RUN2.CIR is defined on lines 23 to 28 for computing the variable a1 (A1 matrix reduced to a scalar). Here the input voltage is zero and the natural response to the initial condition of state variable $V = 1V$ is computed. Accordingly, the analysis of circuit file RUN3 within switching phase 2 leads to variable a2. Equations (3) and (4) are modeled on lines 36 to 42 by means of E-type controlled sources. This model is then analyzed via the AC analysis.

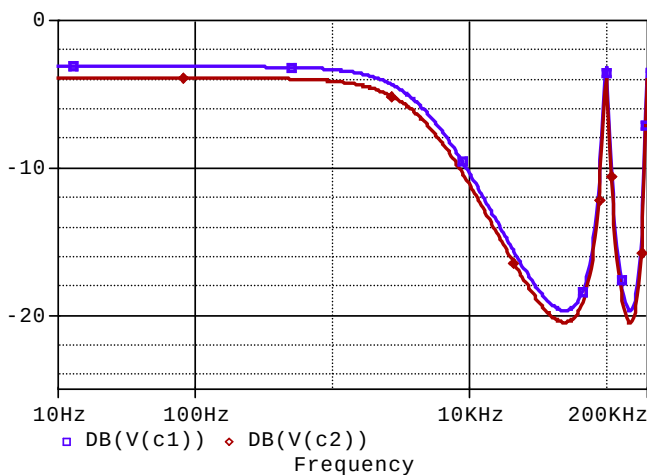


Fig. 2: Results of PSpice AC analysis: Amplitude frequency response with output samples at phases 1 ($V(c1)$) and 2 ($V(c2)$).

The entire sequence of the simulation runs takes fractions of a second on AMD Athlon™ 64 3500+ 2.21GHz, 2GB RAM with installed OrCAD PSpice 16. The resulting frequency responses in Fig. 2 are equivalent to results obtained from a special SPIN program [5].

4 Conclusions

The Simulation Manager (SiM) increases the application range of the OrCAD PSpice such that we can program an arbitrary algorithm and combine the results of basic PSpice analyses (DC, AC, and TRANSIENT). Such combining can be used for advanced analyses of special electronic systems. The paper demonstrates how this approach can be used for programming atypical simulation tasks, namely the AC analysis of real switched circuits.

Acknowledgment

This work is supported by the Grant Agency of the Czech Republic under grant No. 102/08/0784, by the research programmes of BUT MSM0021630503, MSM0021630513, and by the research programme of UD Brno MO FVT0000403.

References

- [1] Biolek, D., Biolková, V., Kolka, Z. 'AC Analysis of Idealized Switched-Capacitor Circuits in Spice-Compatible Programs', In: Proc. of Int. Conf. CICC'07, Greece, 2007, pp. 1-4.
- [2] Bičák, J., Hospodka, J. 'Frequency response of switched circuits in SPICE', Proceedings of ECCTD'03, Krakow, IEEE, 2003, pp. I-333-336.
- [3] Jaroš, M., Kadlec, J., Biolek, D. 'Unconventional Simulation tasks in OrCAD PSpice via Simulation Manager', submitted to the 12th WSEAS Int. Conference on Circuits (CICC'08), Greece, 2008.
- [4] Wojciechowski, J., Vlach, J., Opal, A. 'Analysis of Nonlinear Networks with Inconsistent Initial Conditions', IEEE Transactions on CAS-I, 1995, vol. 42, no. 4, pp. 195-200.
- [5] Biolek, D. Modeling of Periodically Switched Networks by Mixed s-z Description. IEEE Transactions on CAS-I, 1997, vol. 44, no. 8, pp. 750-758.
- [6] Jaroš, M. 'Simulation manager for SPICE-compatible programs', Bachelor's Thesis, UMEL FEKT VUT Brno, 2007 (in Czech).