A SPICE Compatible Behavioral Model of SEPIC Converters

Sam Ben-Yaakov*, Daniel Adar[1] and Gil Rahav

Department of Electrical and Computer Engineering
Ben-Gurion University of the Negev
P. O. Box 653, Beer-Sheva 84105
ISRAEL

Tel: +972-7-461561; Fax: +972-7-472949; Email: sby@bguee.bgu.ac.il

Abstract - An average model of SEPIC converters operating in Continuous Current Mode (CCM) was developed and verified against cycle-by-cycle simulation. The proposed model is compatible with SPICE and other modern electronic circuit simulators and can be used to run DC (static transfer function), AC (small signal, frequency domain) and TRAN (large signal, time domain) analyses. The model is developed in terms of the average, large signal behavior while the small signal (AC) response is worked out automatically by the simulator. An extension to current programmed SEPIC for the case of Peak Current Mode (PCM) Control is also presented. This paper treats the case of SEPIC with uncoupled inductors.

I. INTRODUCTION

Average modeling of switch mode converters is a vital tool for the examination of their dynamic response and for designing the feedback loop. Various approaches for average modeling of PWM converters have been suggested. The PWM switch model [1] was recently applied to model the small signal response of a SEPIC converter [2].

A powerful approach to average modeling and simulation of switch mode system is the equivalent circuit methodology. In this technique the switched system is converted into a continuous, albeit non-linear, equivalent circuit that is compatible with any modern electronic circuit simulator (e.g. HSPICE, MetaSoftware Inc.; ISSPICE, Intusoft Inc.; PSPICE, MicroSim Inc.). The large signal model can then be used as is to run time domain simulation or to run small signal, frequency domain simulation by applying the AC analysis capability embedded in all modern simulators.

Averaging can be conveniently carried out by isolating the switched sub-system and then applying the Switched Inductor Model (SIM) [3] to obtain the continuous equivalent circuit. It has already been demonstrated that the SIM approach can be easily applied to all basic PWM topologies operating in continuous or discontinuous inductor current mode [4-6]. Furthermore, a simple extension of this methodology can be used to emulate the behavior of peak and average current mode control [7]. Recently, the equivalent circuit approach combined with the power of behavioral dependent sources, was also used to model the series-parallel resonant converter [8].

The PWM cases treated hitherto by the equivalent circuit approach were confined to systems which include one switched inductor. However, some PWM converters (e.g. the C'uk and SEPIC topologies) may include a number of switched inductors. To model these cases one needs to extend the basic SIM approach to more than one element. This was investigated in this study by considering one such case, the uncoupled SEPIC converter.

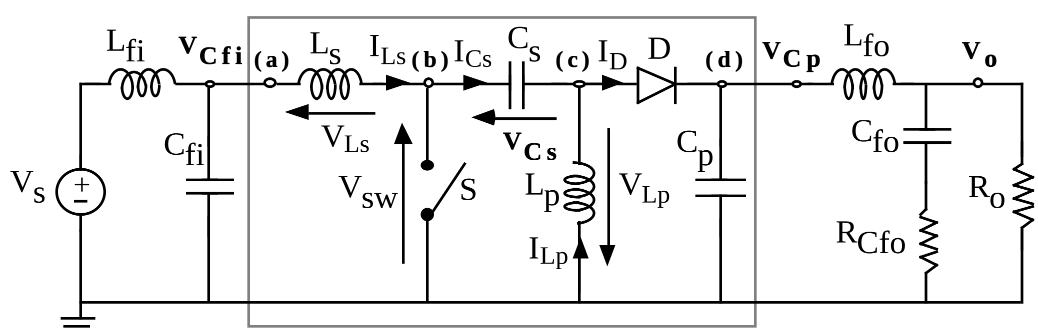


Fig. 1. Basic topology of a SEPIC converter.

* Corresponding author.
1 Presenting author.

II. AVERAGE MODEL DEVELOPMENT

Close examination of the SEPIC power stage (Fig. 1) reveals that it includes three switching elements: two switched inductors (L_s, L_p) and one switched capacitor (C_s). These switched elements are depicted separately in Fig. 2. The proposed average modeling of these switched assemblies is based on the concept of the SIM presented earlier [3]. This approach hinges on the observation that by applying the average voltage across an inductor one gets the average current flowing through it. This current can then be incorporated in dependent current sources to emulate the average currents of the other two ports of the switched inductor.

Following the basic SIM philosophy, the average behavior of a switched capacitor can be derived by injecting into the capacitor (via dependent sources) the expected average current. This will reproduce the average voltage across it. Applying this methodology, one can convert the three switching blocks of Fig. 2 into the three average circuits of Fig. 3.

To be more specific, the inductor L_s of Fig. 1 is connected at terminal (a) to a relatively large capacitor (C_{fi}) which is assumed to represent a constant voltage source during one cycle (but can vary from cycle to cycle). The other terminal of L_s, (b), is switched between ground (through the switch) and the sum of the voltages across capacitors C_s and C_p (via the diode D). These are again assumed to be large enough so that their voltages do not change significantly during one cycle. Therefore, we can emulate the average voltage across the inductor (L_s) by applying a dependent voltage source (E_{Ls}) as shown in the upper model of Fig. 3. The resultant average inductor current (I_{Ls}) can now be used to generate the average current flowing into capacitor C_s ($I_{Ls}D_{off}$) at terminal (b).

The second inductor (L_p) is switched to terminal (c), during the 'on' time, and to terminal (d) during the 'off' time. Again, the inductor is switched between two constant voltages (V_{Cs} at the 'on' time and $-V_{Cp}$ at the 'off' time, approximately) therefore, a dependent voltage source (E_{Lp}) can be used to impose the average voltage across L_p, as shown in Fig. 3. The resultant current (I_{Lp}) is used to generate the average currents flowing into capacitor C_s at terminal (c) and into the output section (C_p and the output filter) at terminal (d).

The capacitor C_s is switched between two current sources (Fig. 2) (assuming that the average currents of the inductors do not change significantly during one cycle). Therefore, the capacitor's average voltage (V_{Cs}) can be emulated by injecting into the capacitor C_s (Fig. 3) the average currents obtained by the upper two switched inductor models (at terminals (b) and (c)).

The complete expressions for the voltage dependent sources (of Fig. 3) imposed on inductor L_s and L_p are as follows :

$$E_{Ls} = V_{sw(on)}*D_{on} + (V_{Cs} + V_{D(on)} + V_{Cp})*D_{off} \quad (1)$$

$$E_{Lp} = (V_{Cs} - V_{sw(on)})*D_{on} - (V_{D(on)} + V_{Cp})*D_{off} \quad (2)$$

where:

$V_{sw(on)}$ is the 'on' switch voltage

$V_{D(on)}$ is the diode 'on' voltage

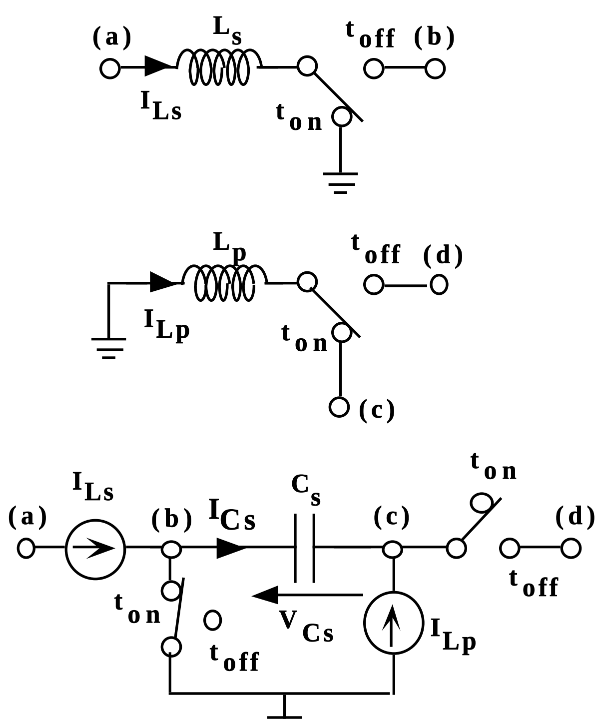


Fig. 2. The switched elements of a SEPIC converter (refer to Fig. 1 for notations).

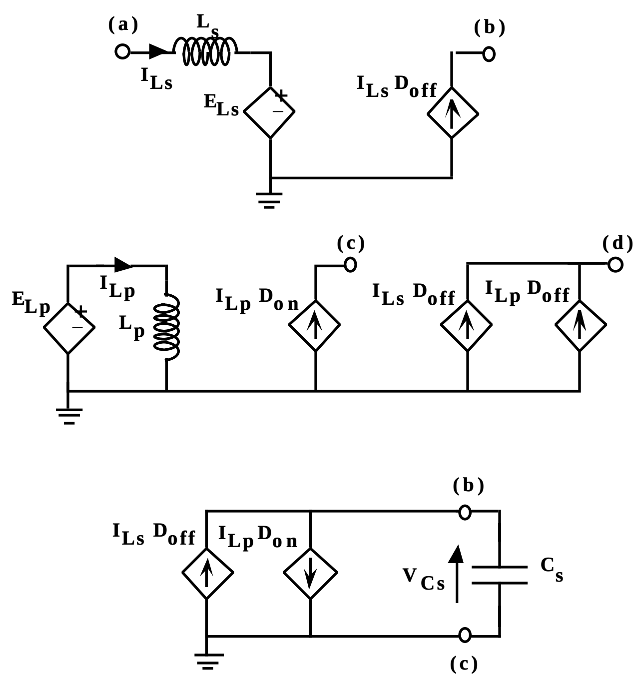


Fig. 3. The average models of the SEPIC switched elements (as shown in Fig. 2).

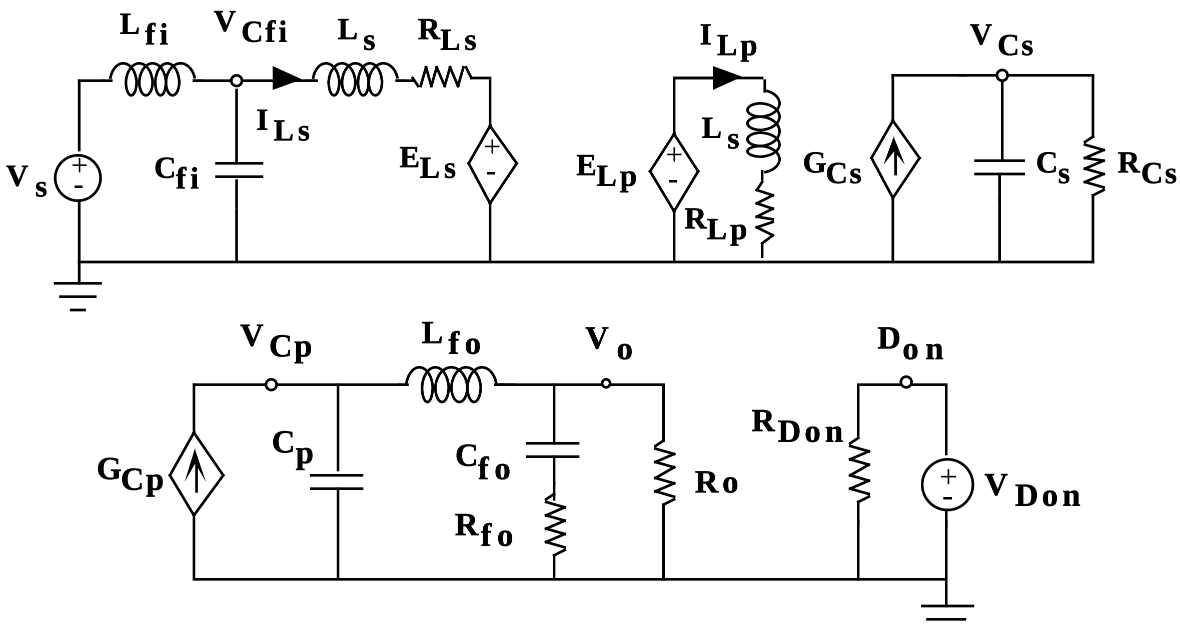


Fig. 4. The proposed average model of an opened loop SEPIC converter.

Combining the three average circuits of Fig. 3, we obtain the complete average model of the SEPIC converter as shown in Fig. 4. Note that the dependent current sources that appeared in parallel in Fig. 3, have been combined into one dependent current source that is described by an expression that reflects the sum of the currents.

The duty cycle is emulated by a continuous independent voltage source for open loop simulations. This source can be replaced by a dependent voltage source and the corresponding circuitry for closed loop simulation.

The voltage across the switch can be emulated dynamically by the following expression:

$$V_{sw(on)} = (I_{Ls} + I_{Lp}) * R_{s(on)} \quad (3)$$

where $R_{s(on)}$ is the 'on' switch resistance.

The diode voltage drop ($V_{D(on)}$) during the 'on' time is assumed here (eq. 4-5) to be constant (0.7V) but can be emulated more accurately by injecting the diode's 'on' current into the appropriate diode model as shown in Fig. 5.

The expressions of the dependent sources for the complete SEPIC average model of Fig. 4 are thus as follows :

$$E_{Ls} = (I_{Ls} + I_{Lp}) * R_{s(on)} * D_{on} + (V_{Cs} + 0.7 + V_{Cp}) * D_{off} \quad (4)$$

$$E_{Lp} = (V_{Cs} - (I_{Ls} + I_{Lp}) * R_{s(on)}) * D_{on} - (0.7 + V_{Cp}) * D_{off} \quad (5)$$

$$G_{Cs} = -I_{Lp} * D_{on} + I_{Ls} * D_{off} \quad (6)$$

$$G_{Cp} = (I_{Ls} + I_{Lp}) * D_{off} \quad (7)$$

III. EXTENSION TO CURRENT PROGRAMMING

As presented earlier [9, 10], Current Programmed models need a 'Duty Cycle Generator' to represent the relationship between the control voltage and the duty cycle. The waveforms for the case of SEPIC with PCM control, where the current is sensed through the switch, are shown in Fig. 6. The duty cycle (D_{on}) can be determined by the intersection of the voltage associated with the sensed current ($K_s I_{sw}$) and the slope compensated control voltage (V_e $-M_c t$). The intersection can be expressed by the following equation:

$$K_s(I_{Ls} + I_{Lp}) + K_s\left(\frac{V_{CFi} - V_{sw(on)}}{L_s} + \frac{V_{Cs} - V_{sw(on)}}{L_p}\right)\frac{T_{on}}{2} = V_e - M_c T_{on} \quad (8)$$

where:

$$T_{on} = D_{on} T_s \quad (9)$$

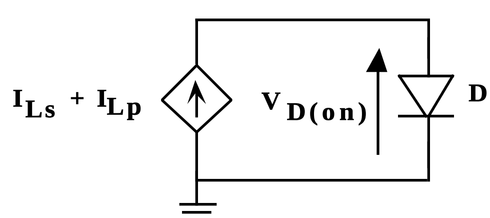


Fig. 5. A circuit for emulating the average diode voltage drop $V_{D(on)}$.

Substituting eq. (9) into eq. (8) and rearranging we get the required relationship :

$$D_{on} = \frac{V_e - K_s(I_{Ls} + I_{Lp})}{T_s\left[M_c + \frac{K_s}{2}\left(\frac{V_{CFi}-V_{sw(on)}}{L_s} + \frac{V_{Cs}-V_{sw(on)}}{L_p}\right)\right]} \quad (10)$$

where:

V_e is the error voltage of the outer (voltage) loop (V).
K_s is the sensed current conversion gain (V/A).
I_{Ls} is the average current of L_s (A).
I_{Lp} is the average current of L_p (A).
T_s is the switching period time (Sec).
M_c is the slope compensation constant (V/Sec).

Consequently, for a current programmed SEPIC converter, in which the current through the switch is sensed, the independent voltage source V_{Don} (Fig. 4) need to be replaced by an equivalent circuit that includes a dependent voltage source (E_{Don}) that emulates eq. (10). This dependent source is driven by an independent voltage source V_e (for open outer loop) as shown in Fig 7. Again, this source (V_e) can be replaced by a dependent voltage source and the auxiliary circuitry associated with the outer (voltage) loop.

IV. SPICE SIMULATION

Since the proposed average model is SPICE compatible, it can be used directly to simulate the large and small signal responses of SEPIC converters in open or closed loop configuration.

The average model was verified against a cycle-by-cycle PSPICE (V. 6.1, MicroSim Co.) simulation of the switched SEPIC converter. The parameters of the converter were as follows:

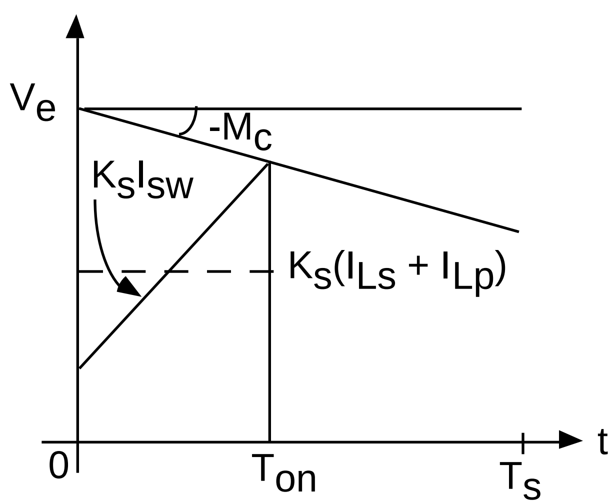


Fig. 6. The wave forms (one cycle) for SEPIC with PCM control.

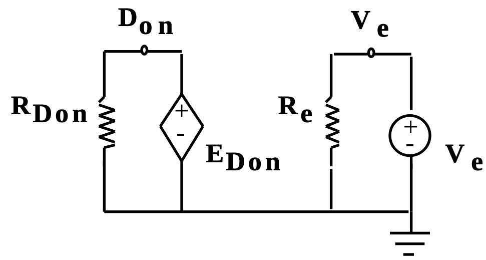


Fig. 7. 'Duty Cycle Generator' for Current Programmed SEPIC. E_{Don} is defined by equation (10).

V_s = 36V, L_{fi} = 2.75µH, C_{fi} = 0.2µF, L_s = 9.75µH, L_p = 9.75µH, C_s = 0.3µF, C_p = 0.44µF, L_{fo} = 3.8µH, C_{fo} = 940µF, R_{Cfo} = 90 mΩ, $R_{o(nominal)}$ = 5 Ω (see Fig. 1 for notations).

Fig. 8 shows the DC characteristics of the SEPIC power stage for two values of the switch resistance ($R_{s(on)}$). For large duty cycles, the switch resistance has significant effect on the transfer ratio and therefore should be taken into account. As can be seen, in both cases the model and cycle-by-cycle simulations agree very well.

Fig. 9a demonstrates the validity of the small signal control-to-output responses. The continuous line is the output of the average model simulation. The few discrete points (phase and gain) were tediously collected one by one for each frequency using a straightforward cycle-by-cycle simulation of the switching circuit. This was carried out by modulating the input to a pulse width modulator by a sine wave and measuring the resulting small signal at the output. The running time for each point was few orders of magnitudes longer than the complete average simulation done by the proposed model. A similar comparison was carried out for the current programmed case (Fig. 9b). The results demonstrate the validity and accuracy of the proposed average model of SEPIC converters when operated in the voltage or current programmed modes.

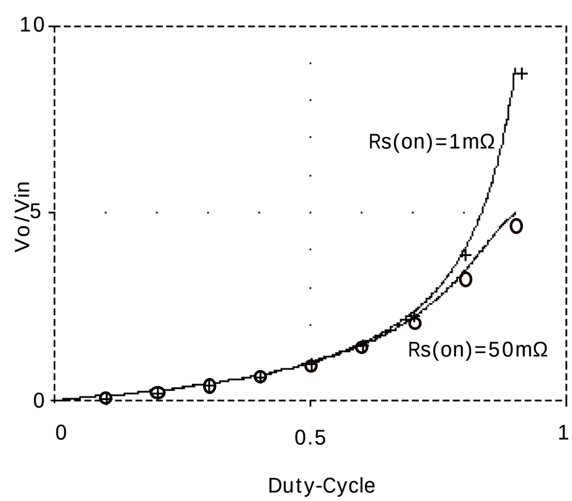
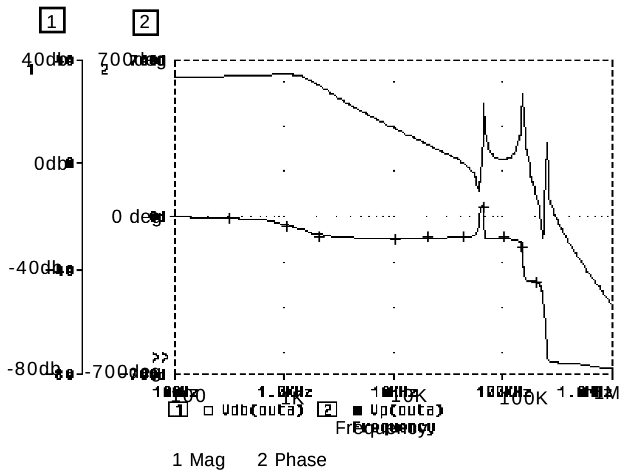


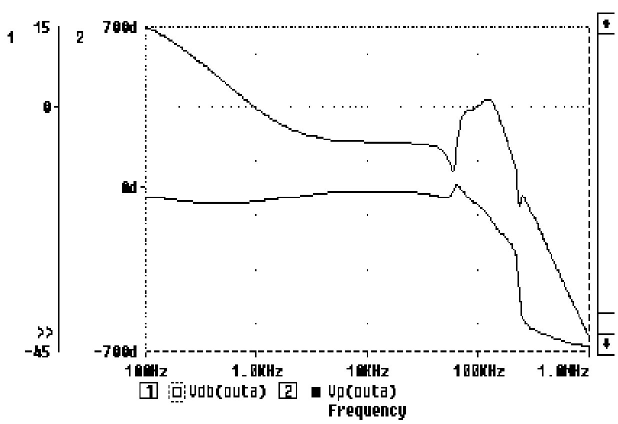
Fig. 8. DC characteristics of SEPIC topology. Cycle by cycle simulations (**o** for $R_{S(on)}$=50mΩ, + for $R_{S(on)}$=1mΩ) vs. average model simulations.

Large signal responses for a load step change obtained by cycle-by-cycle simulation and by average model simulation are shown in Fig. 10. Again, there is almost a perfect agreement between the two, except for the ripple that appears, as expected, only in the cycle-by-cycle results. The speed up ratio of the average model simulation was about 300.

A PSPICE [11] schematics of the proposed model is depicted in Fig. 11. This representation is compatible with the schematics capture front end of recent PSPICE versions (6.1 and up). The schematics of Fig. 11 is a direct translation of Fig. 1 per the values given in Section IV. The behavioral dependent sources emulate eq. (4-7).



(a)



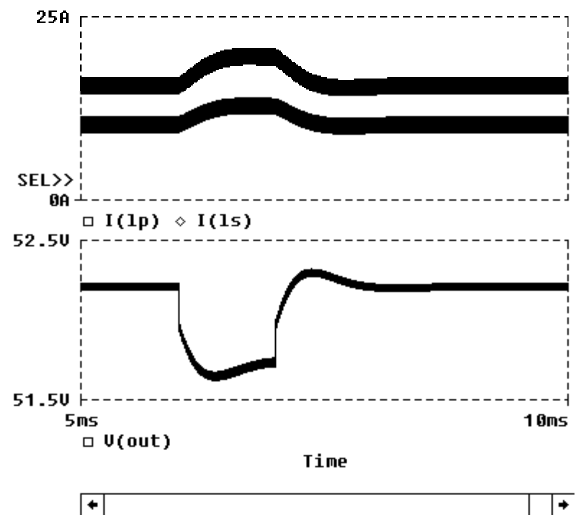
(b)

Fig. 9. Small signal response of a SEPIC converter. (a) Control (D_{on}) to output response of Voltage Mode control (operating point: D_{on}=0.125, R_o=5Ω). (b) Control (V_e) to output response of Current Mode control (Operating point: V_e=0.625V, R_o=1.666Ω). Continuous curve: average model. Discrete points: Cycle-by-cycle simulations **(o)** for Magnitude[dB], **(+)** for Phase[deg].

The 'Schematics' diagram of Fig. 11 covers the open loop power stage operating in voltage mode. For current mode control one has to define an additional behavioral dependent sources that emulates equation (10). The output of this module will replace VDon in Fig. 11.

VI. CONCLUSIONS

This study extends the methodology of average modeling by the SIM approach to systems that cannot be reduced to a single switched inductor. By applying the same reasoning, average models of the ZETA and C'uk converters can be easily developed. The treatment of coupled inductor cases will be presented in a subsequent paper.



(b)

Fig. 10. Transient response for a load step of 20% (from R_o=5Ω down to 4Ω and back). Upper plots: inductor currents (I_{Ls} (top) and I_{Lp}). Bottom plots: output voltage. (a) Cycle-by-cycle simulation .(b) Average Model simulation.

REFERENCES

[1] V. Vorperian, "Simplified analysis of PWM converters using the model of PWM switch, Part I: Continuous conduction mode," *IEEE Trans. on AES*, Vol. 26, No. 3, May 1990.

[2] W. M. Moussa , "Modeling and performance evaluation of a DC/DC SEPIC converter," APEC '95 , Vol. 2, pp. 702-706.

[3] S. Ben-Yaakov, "SPICE simulation of PWM DC-DC converter systems: Voltage feedback, continuous inductor conduction mode," *IEE Electronics letters*, Vol. 25, No. 16, 1989, pp. 1061-1063.

[4] Y. Amran, F. Huliehel and S. Ben-Yaakov,“ A unified SPICE compatible average model of PWM converters,“ *IEEE Trans. on Power Electronics*, vol. 6, pp. 585-594, Oct. 1991.

[5] D. Edry and S. Ben-Yaakov, "A SPICE compatible model of Magamp post regulator," APEC '92, pp. 793-800.

[6] D. Edry, O. Mor, M. Hadar and S. Ben-Yaakov, "A Spice compatible model of Tapped-Inductor PWM converters," APEC '94, pp. 1021-1027.

[7] S. Ben-Yaakov and Z. Gaaton, "A unified model of current feedback in switch mode converters," International Symposium on Circuits and Systems (ISCAS '92), 1992, Vol. 4, pp. 1891-1894, San Diego,.

[8] S. Ben-Yaakov and G. Rahav, "Average modeling and simulation of Series-Parallel Resonant Converters by SPICE compatible behavioral dependent sources," APEC '96, vol. 1, pp. 116-120.

[9] S. Ben-Yaakov, "Average modeling of PWM converters by direct implementation of behavioral relationships," APEC '93, pp. 510-516.

[10] S. Ben-Yaakov and D. Adar, "Average models as tools for studying the dynamics of switch mode DC-DC converters," PESC '94, vol. 2, pp. 1369-1376.

[11] PSPICE: MicroSim Inc.,20 Fairbanks, Irvine, California.

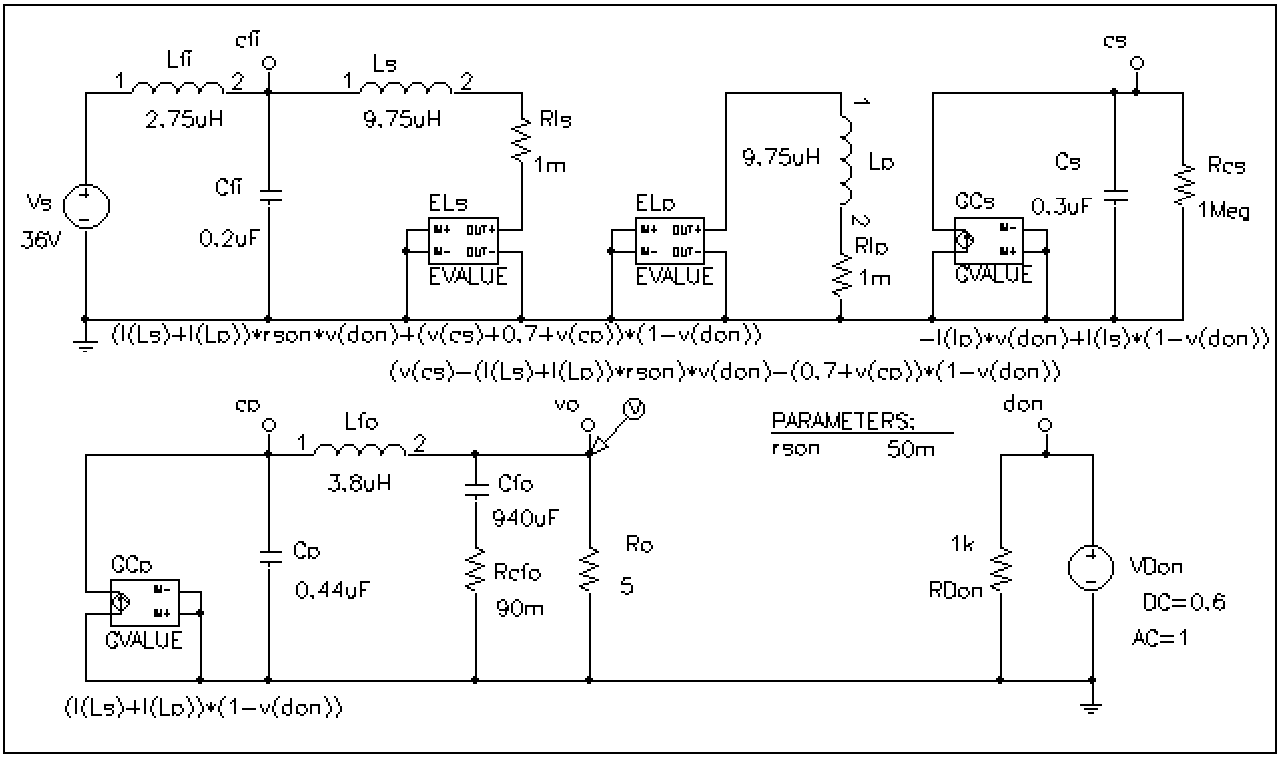


Fig. 11 'Schematics' (MicroSim Inc.) diagram of SEPIC power stage operating in open loop.