

# A simulation model for parallel buck DC/DC converters with separate capacities created in MS Excel

<sup>1</sup>Dávid SCHWEINER, <sup>2</sup>Dobroslav KOVÁČ

<sup>1,2</sup> Department of Theoretical and Industrial Electrical Engineering, Faculty of Electrical Engineering and Informatics, Technical University of Košice, Slovak Republic

<sup>1</sup>david.schweiner@tuke.sk, <sup>2</sup>dobroslav.kovac@tuke.sk

**Abstract** — The paper describes a creation of the simulation model for two buck DC/DC converters with separate capacities connected in parallel. The simulation model is created in Excel, which is part of MS Office package. For calculations of the current in both converters and for the output voltage is used transient analysis. Created simulation model graphically shows the simulation results and it is also using a makro code for further calculations.

**Keywords** — buck DC/DC converter, flowchart, MS Office, Laplace-Carson transformation, transient analysis

## I. INTRODUCTION

The switch-mode power supplies (SMPS) are recently very popular because of their high effectivity, lower cost, smaller size and lower weight compared to conventional linear power supplies. The essential and vital part of every SMPS is DC/DC converter. The mostly used DC/DC converter in SMPS is buck (step-down) converter. This type of converter is also used for analyze in this article [1], [2].

In the previous issues of this journal were published two articles [3], [4], which contain the transient analysis of the buck DC/DC converter. This analysis was conducted for aperiodic and also for quasi-stationary course of values. The analysis can be used as a fundamentals for creating own, flexible simulation tool.

The parallel connection of more DC/DC converters has a potential of bringing power increase, but also can be problematic in terms of stability and reliability. It is the reason why it is useful to further research such possibilities. Therefore, created model in MS Excel is designed to simulate two buck DC/DC converters connected in parallel, with individual capacitors. In the previous issue of the journal [5] was also published article, which described design of the simulation model in Excel for only one buck DC/DC converter.

## II. FLOWCHART

When the equations of transient analysis are used in the proper algorithm, then the algorithm can be written to the Excel. According to how the algorithm is designed, more topologies can be programmed into Excel model. For example, there can be cascade connection, parallel connection with mutual capacity or parallel connection with individual capacities. Flowchart for the topology of two DC/DC converters connected in parallel with separate capacitors can be seen in the Fig. 1 bellow.

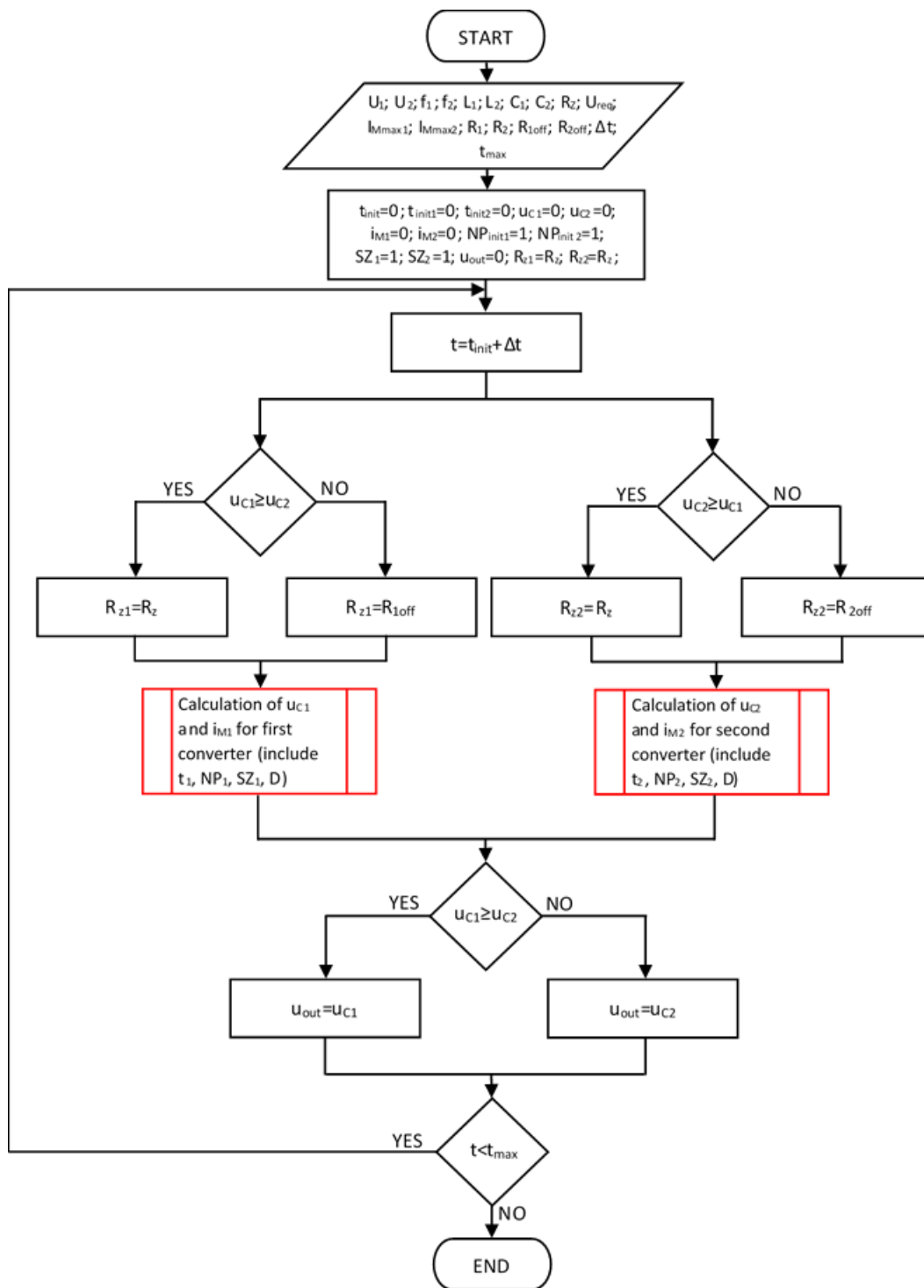


Fig. 1 Flowchart for simulation model of parallel connected DC/DC converters with separate capacitors

The first block of the flowchart defines required parameters (constants), such as input voltages  $U_1$  and  $U_2$ , capacity of both converters  $C_1$  and  $C_2$ , inductance of both converters  $L_1$  and  $L_2$  etc. The second block defines initial values of variables, which are subsequently calculated. The third block is calculating current time  $t$  by adding time step  $\Delta t$  to the initial time  $t_{init}$ . This topology uses so called summing diodes, which should ensure stable operation of parallel connection, by preventing overflow of energy between both converters. Therefore, next blocks of the flowchart decide, which of the two output voltages ( $u_{C1}$  or  $u_{C2}$ ) is higher. The output load (resistivity) for each converter is set according to that. Converter, which has higher output voltage, is connected to the output load  $R_z$ , which is defined by the user. Converter with lower output voltage is connected to the high resistivity  $R_{off}$ , which represents resistivity of the summing diode in non-conducting state (positive potential on its cathode). Below are two red blocks, which contain subroutine for main calculations of the output voltage  $u_C$  and converter's current

$i_M$  for both parallel converters. Flowchart of the subroutine can be seen in previous issue of the journal [5]. Next block determines overall output voltage  $u_{out}$  by deciding, which of the two output voltages  $u_{C1}$  and  $u_{C2}$  is higher (higher voltage is equal to output voltage). The last block decides if the maximum time  $t_{max}$  was reached. In case the time was reached, algorithm is going back before the third block, where the another time step  $\Delta t$  is added to the current time  $t$ .

### III. ILLUSTRATION AND VERIFICATION

The final version of the created Excel simulation model for parallel DC/DC converters with individual capacities can be seen in the Fig. 2.

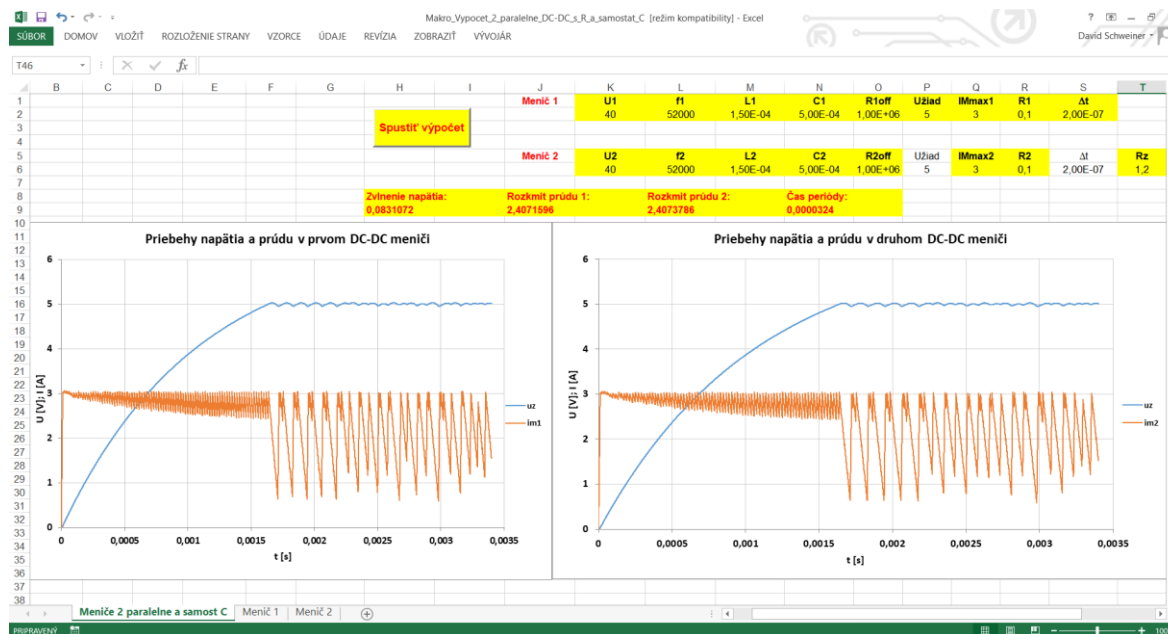


Fig. 2 Final simulation model of parallel connected DC/DC converters with separate capacities

The model contains two graphs, one for each converter. The graphs contains calculated voltage and current as a function of time. For both converters can be set input voltage  $U$ , switching frequency  $f$ , inductance  $L$ , capacity  $C$ , maximum allowed converter's current  $I_{Mmax}$  and internal resistivity of the converter  $R$ . Common parameters, which can be set are output load  $R_z$ , required output voltage  $U_{req}$  and time step  $\Delta t$ . These cells, which can be filled by a user, have yellow background color. The simulation model is also using macro code (VBA) for calculating overall voltage ripple, amplitude of the current and also time of the voltage ripple periode. These calculations can be launched by button (ActiveX element), which has macro code mapped on itself. Cells with these macro calculations are filled with red font (also with yellow background color).

The created model can be verified by comparison with other trustworthy and well established software. Specifically, for this verification is used software PSpice, which is part of the OrCAD. In the Fig. 3 bellow can be seen simulation scheme in OrCAD.

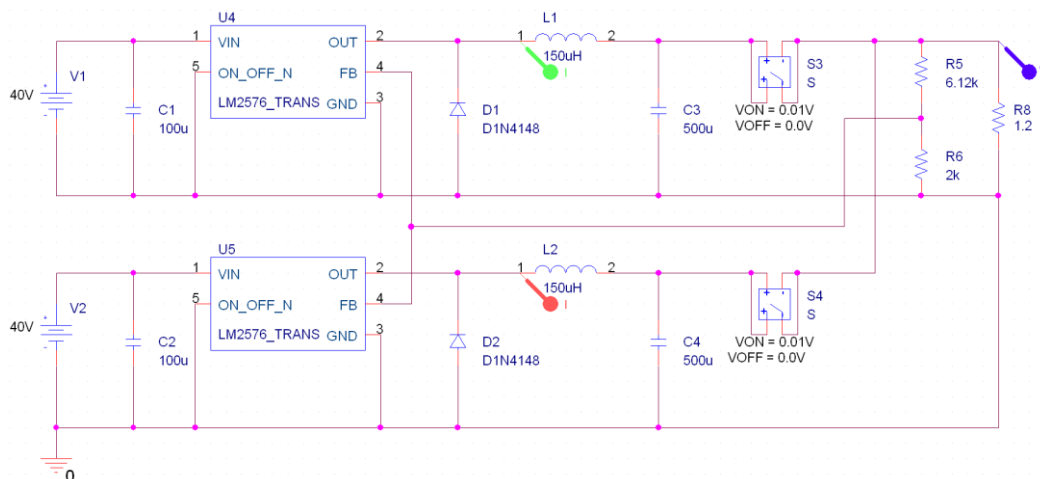


Fig. 3 Scheme in the OrCAD for the PSpice simulation

For the simulation is used model of LM2576 switching regulator, which has 52kHz switching frequency, it is the same as have been set in the Excel model. Input voltage supplies have 40V, inductances for both converters are 150µH and capacity for each converter are 500µF. Output load is set to 1,2Ω and output voltage is set to 5V by voltage divider. Consequently, all the parameters are equal for both simulations, Excel simulation and Pspice simulation. In the Fig. 4 and Fig. 5 can be seen result from PSpice simulation.

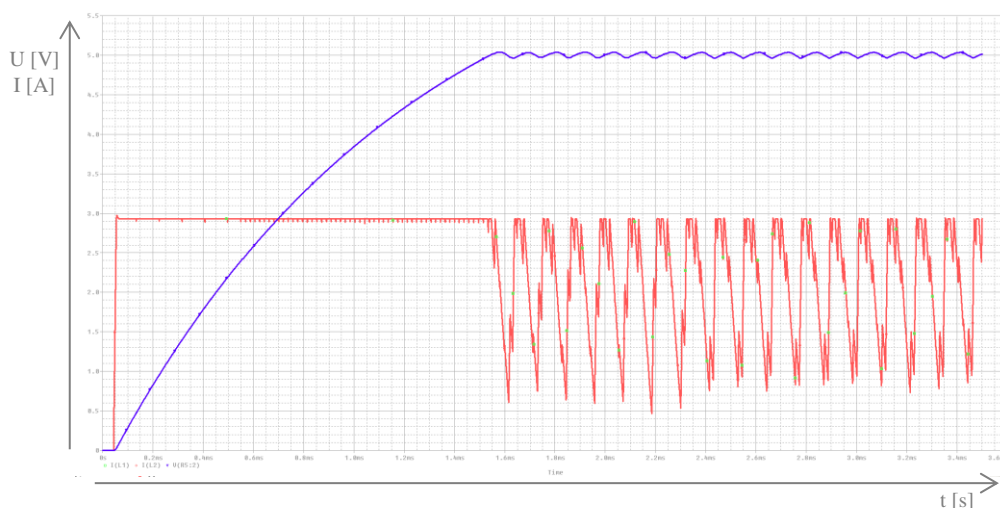


Fig. 4 Simulation results from PSpice

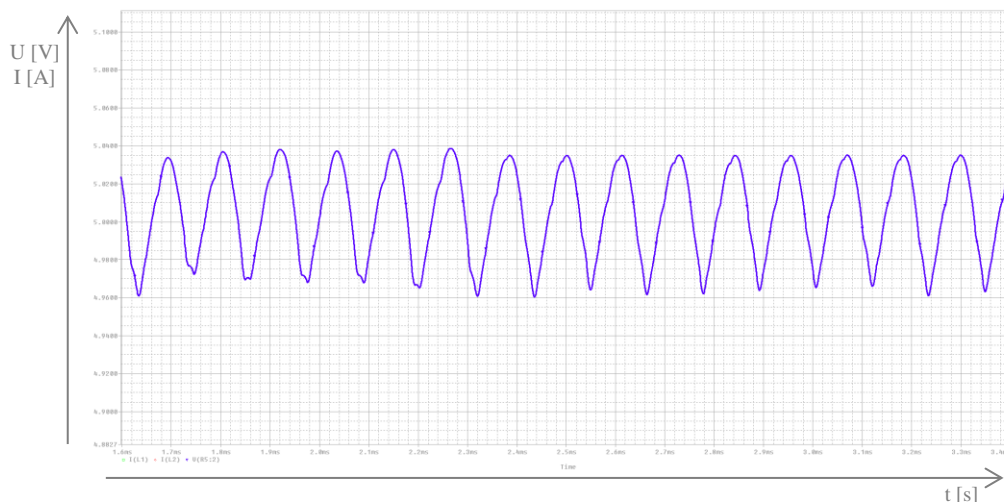


Fig. 5 Detail of voltage ripple in PSpice

Differences in both simulations are quite small. The amplitude of the converter's current is almost the same in both simulations and it oscillate in the interval from 0,5A to 3A. The rise time (time needed to get required voltage) is approximately the same in both simulations, specifically it is 1,6ms. Output voltage ripple in the Excel simulation varies in the interval from 4,95V up to 5,03V. In the PSpice simulation it is from 4,96V up to 5,04V. Thus, the difference in the output voltage ripple is maximum 10mV, while overall voltage ripple is the same in both cases (80mV).

Overall results are satisfactory considering versatility of the designed Excel model. Accordingly, time step  $\Delta t$  also determines accuracy of the simulation, if the  $\Delta t$  would be set to lower value, results could be even more precise, although simulation would take a little bit longer.

#### IV. CONCLUSION

Created simulation model in Excel has a significant advantage in speed of the simulation. User can change one of the parameter and results can be observed in real time with delay only a few seconds. The model is also versatile, it can be used for any buck DC/DC converter. Another advantage is customizability, so user can freely change the model, by changing it's conditions and macro code.

The model can be used for research of possibilities for the parallel connection of DC/DC converters. In the future should be designed and created other topologies as parallel connection with individual capacities or cascade connection of more DC/DC converters.

#### ACKNOWLEDGMENT

The paper has been prepared under support of grant project FEI-2018-50.

#### REFERENCES

- [1] S. Kumar, P. Jain and S. Akashe, "Design and analysis of switch mode power supply using CMOS single-phase full bridge rectifier," *2015 International Conference on Communication Networks (ICCN)*, Gwalior, 2015, pp. 374-380. doi: 10.1109/ICCN.2015.72
- [2] X. Xu, A. J. Collin, S. Z. Djokic, R. Langella and A. Testa, "Operating Cycle Performance, Lost Periodicity, and Waveform Distortion of Switch-Mode Power Supplies," in *IEEE Transactions on Instrumentation and Measurement*. doi: 10.1109/TIM.2018.2813761
- [3] D. Schweiner, D. Kováč, "Výpočet prechodného deja pri spínaní znižovacieho DC/DC meniča pre aperiodický priebeh veličín," *JIEE Časopis priemyselnej elektrotechniky*. Roč. 2, č. 1, 2018. ISSN 2454-0900.
- [4] D. Schweiner, D. Kováč, "Výpočet prechodného deja pri spínaní znižovacieho DC/DC meniča pre kvazistacionárny priebeh veličín," *JIEE Časopis priemyselnej elektrotechniky*. Roč. 2, č. 2, 2018. ISSN 2454-0900.
- [5] D. Schweiner, D. Kováč, "Vytvorenie modelu DC/DC meniča v programe Excel," *JIEE Časopis priemyselnej elektrotechniky*. Roč. 2, č. 4, 2018. ISSN 2454-0900.