*Eastern Academic Journal*

*ISSN: 2367-7384*
*Issue 2, pp.62-68, June, 2016*

# IMPROVED SIMULATION APPROACH FOR ANALYSIS OF ELECTRONIC SWITCHES IN POWER ELECTRONIC CONVERTERS

Angel St. Marinov

Technical University of Varna
Varna Bulgaria

**ABSTRACT—** *The paper presents an accelerated simulation approach that allows analysis of power electronics circuits and more specifically the characteristics of their electronic switches. The approach allows improved simulation times for detailed analysis of voltages and currents during switching, while longer overall simulation time takes place – analysis of different modes of operation for the specific circuit. The algorithm on which the suggested approach is developed is presented in details. It includes a pilot run for the complete simulation time using ideal switches, where based on a specified number of discrete values for inductor currents and capacitor voltages in the circuit a series of small time, precise step simulations with the real switches are made. Based on that series of simulation conclusion for the overall performance of the switches can be made. An example based on a boost converter is provided. Results from this example are given as a verification of the suggested approach.*

**Keywords:** electronic switches, power electronics, power losses, simulation

## 1. INTRODUCTUON

Power electronics is a specific field of electronics that deals with the conversion, conditioning and control of electric energy. Most modern electrical and electronic devices rely and have in some part a power electronic device. At the core of power electronic devices is the power electronic converter. Having the correct and precise design of the converter allows for high efficiency, better thermal management and reliability, not only for the power electronic device but also for the device that utilize it [1, 2].

As with most electronic devices, modern approaches for designing power electronic converters include computer aided analysis in the form of simulations. Simulations allow for a fast and accurate evaluation of the given design and its modes of operation [3, 4].

A specific issue with simulations of power electronic converters is selecting the accurate simulation time and simulation step. Selecting a longer simulation time allows designers to study the various modes of operation of the converter, while having a small simulation step allows for precise analysis of the switching transients of the electronic switches used in the converter.

For example in order to simulate a converter until its steady state is reached a simulation time of a several tens to several hundred of milliseconds might be required. If the said convertor uses a modern power electronic switch (for example a MOSFET), in order to examine its switching characteristics - so an accurate evaluation of power loss can be achieved – simulation steps in the range of several nanoseconds might be required (as the switching process usually takes several tens of nanoseconds). This means that having an analysis of the switch for whole duration of the simulation time will require both long simulation time and small simulation step that will leads to a large computing time and high amounts of generated data.

Corresponding author at: *Technical University of Varna, Varna, Bulgaria, e-mail:* **a.marinov@tu-varna.bg**

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

Some solutions to this issue include: compromise between simulation time and simulation step; using a set of simulations with varying simulation parameters; using various step simulation. Those solutions however do not always allow for a fast and accurate detailed analysis for both the modes of operation of the evaluated circuit and the specifics of the components used.

Considering those points the paper aims at suggesting and evaluating a simulation approach that allows for a fast analysis of electronic converters using large simulation times while having accurate evaluation of the switching characteristics of the electronic switches.

## 2. SUGGESTED APPROACH

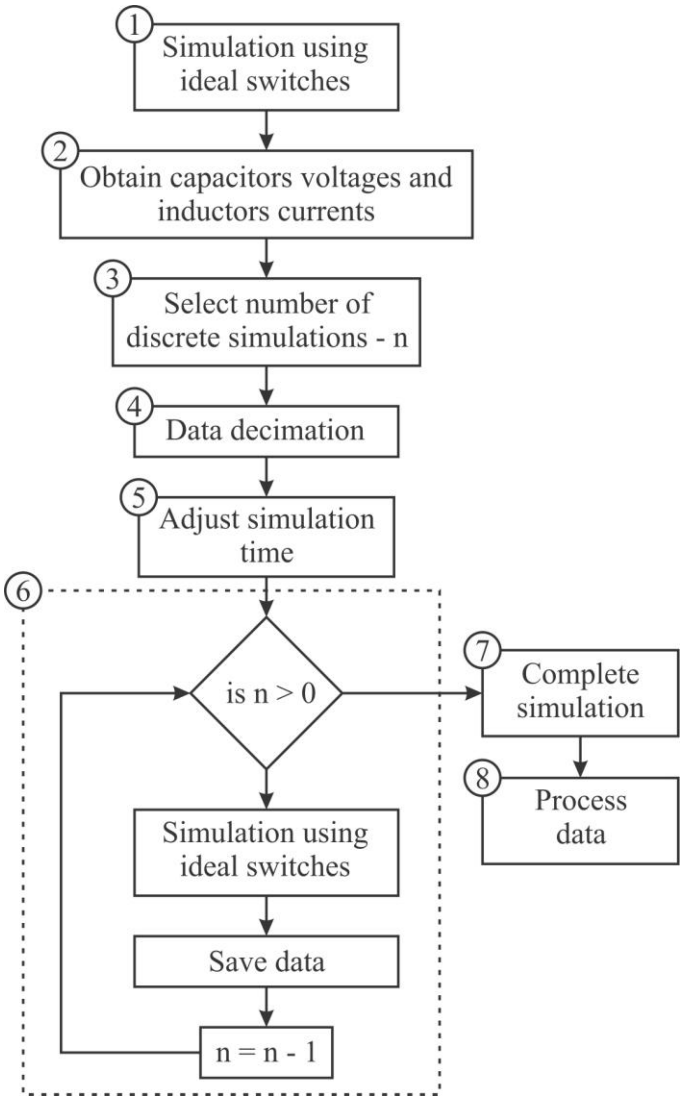


**Figure 1.** Algorithm of the suggested simulation approach

The algorithm that the suggested simulation approach follows is presented at figure 1. It includes the following steps:

(1) The examined design is modeled and simulated for the required simulation time using ideal switches. This is simulation is carried using a large simulation step. Parameters of the switches are not considered and taken into account.

(2) Based on the simulation form step 1 datasets for the inductors currents and capacitors voltages are obtained.

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

(3) A number of sub-simulations *n* is selected. The larger the number *n* is the better the accuracy will be, but at the expense of a larger computing time.

(4) Based on the selected number of simulations *n* an equal number of discrete values for the inductors currents and capacitors voltages is taken. Those values can be taken at even on varied intervals (for example where only specific regions need to be examined in detail).

(5) Simulation time for the selected design is readjusted to encompass only a few working periods of the given circuit. The ideal switches are replaced with real. Simulation step time is adjusted so an accurate analysis of the switch transients can be achieved.

(6) *n* simulations are made where each discrete value of the inductors currents and capacitors voltages are set as initial conditions for the given simulation. Data for analyzed switches is saved for each simulation instance.

(7) After the simulation is complete data is reconstructed using based on the total simulation time and the number of simulations.

## 3. SIMULATIONAL VERIFICATION

In order to demonstrate and verify the suggested simulation approach a dedicated example is presented in this section. The example is based on a boost DC/DC converter. The converter is presented at figure 2. Where *Circuit 1* is the boost converter developed using an ideal switch (used for step one of the algorithm presented in section 2 of the paper), while *Circuit 2* is the converter developed using a real switch (used of step six of the algorithm) – in the case of the example a MOSFET. It has to be noted that for simplification purposes for both circuits the inductor L, the capacitor C, the load R the diode D are modeled as ideal components. The boost converter is designed based on [5]. Values for the parameters of its components and the way they are modeled is given in table 1.

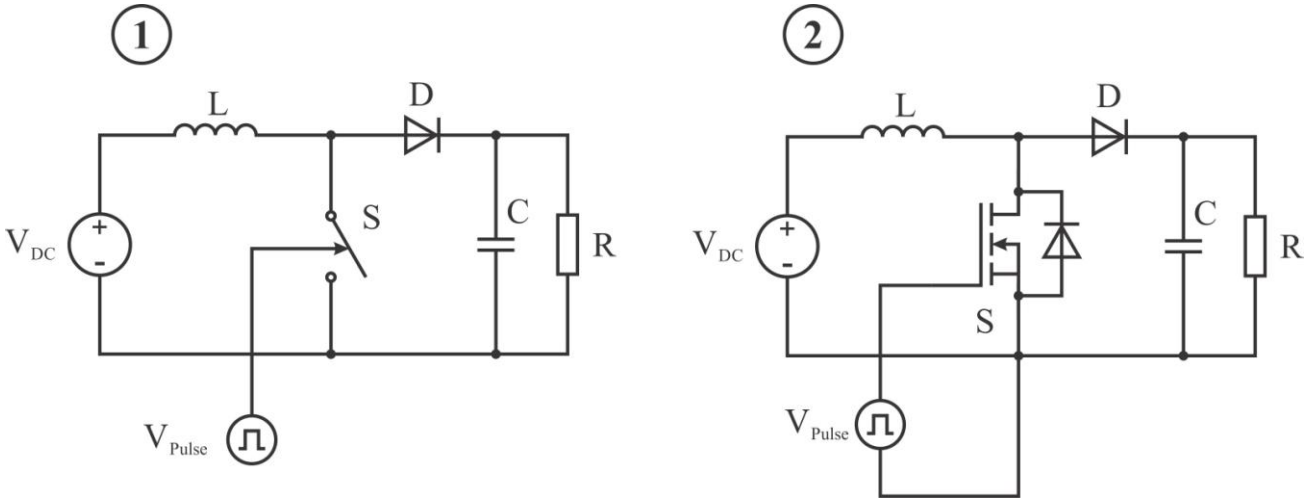


**Figure 2.** Circuits for the developed example. Circuit 1 – boost converter with ideal switches; Circuit 2 boost converter with real switches

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

| Component | Value | Description |
|---|---|---|
| C | 10μF | Modeled as ideal capacitor – equivalent series resistance is not considered |
| L | 10μH | Modeled as ideal inductor – series resistance and magnetic material properties are not considered |
| R | 100Ω | Load resistance – purely active load is used. |
| $V_{DC}$ | 100V | Ideal voltage source |
| $V_{Pulse}$ | *Frequency*: 50kHz or 25kHz<br>*Duty Ratio*: 0.5<br>*Amplitude*: logical level for Circuit 1; 15V for Circuit 2 | Pulse controlled for either the ideal switch or the examined MOSFET. |
| S | *On resistance at* 10V *gate source voltage* $V_{GS}$*:* 0.18Ω<br>*Input Capacitance Ciss:* 1300pF<br>*Output Capacitance Coss:* 430pF<br>*Reverse transfer Capacitance Crss:* 130pF | Parameters set based on component IRF640. Modeled using [6, 7, 8] |
| D | *Forward voltage drop:* 0.6V | Modeled as ideal diode – reverse recovery is not considered. |

**Table 1.** Values and modeling specifics of the components used in the boost circuit

Table 2 presents the simulation parameters in terms of simulation time and simulation step based on the approach suggested in section 1.

| Parameter | Value |
|---|---|
| Solver | Variable step |
| Simulation time for Circuit 1 | 15ms |
| Maximal simulation step for Circuit 1 and Circuit 2 | 1ns |
| Simulation time for Circuit 2 – algorithm application | 2 periods of the switch, respectively: 40μs for 50kH; 80 μs for 25kHz; |
| Simulation time for Circuit 2 – Verification | 15ms |
| Number of discrete simulations - *n* | 20 |

**Table 2.** Algorithm of the suggested simulation approach

The analysis presented bellow includes: (1) A simulation using the algorithm suggested in section 2; (2) A conventional simulation using the same simulation time and the same simulation step. The suggested simulation approach is compared against the convention simulation for two work frequencies of the switch – 50kHz and 25kHz. Both simulations are executed in a specialized

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

simulation software using the same solver the operates under variable step size. The simulation time is selected so a steady state of the circuit is reached after the simulation step is complete. The step size is selected so it is compatible with the switching times of the selected transistor.

Results for the inductor current and the capacitor voltage from simulation with Circuit 1 are presented at figure 3. *n* discrete values from those waveforms are taken at even intervals and applied as initial currents and voltages respectively for the inductor and the capacitor for *n* simulations with parameters specified as in table 2.

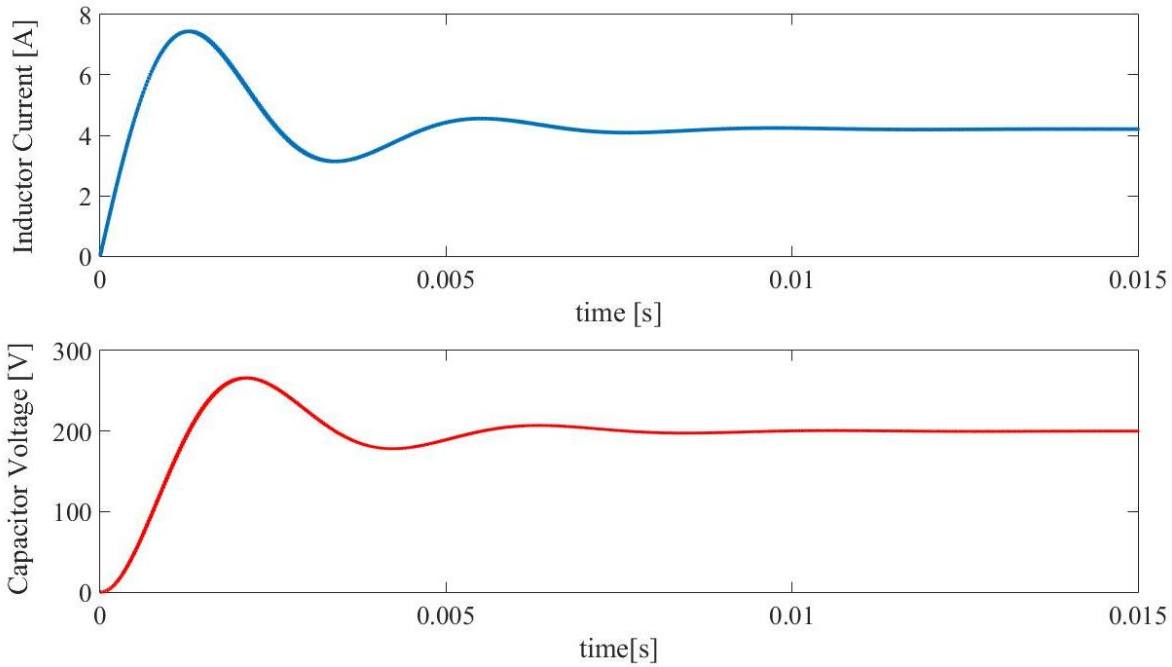


**Figure 3.** Algorithm of the suggested simulation approach

For further evaluation and comparison, the average total power losses on the MOSFET [] are considered. Those are the sum of the switching and conduction losses of the MOSFET. The switching losses are formed during the switching processes of the MOSFET. For this circuit the average total power loss will be highly depended of the inductor current and capacitor voltage – they will highly depend on the mode of operation of the circuit – start up or steady state. This makes average total power losses of the switch a parameter that will allow sufficient evaluation and comparison for between the suggested approach and the conventional simulation.

Results from the execution of the suggested approach and its comparison with a conventional simulation are presented at figures 4 and 5, where figure 4 shows the analysis for 50kHz switching frequency, while figure 5 shows the analysis for 25kHz switching frequency. The total power loss is averaged over 2 periods of the switch for the suggested approach and over 20 periods of the switch for the conventional simulation

It can be seen from the presented results that the suggested approach provides results close to those of the obtained through the conventional simulation. Both are comparable with small amount of error.

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

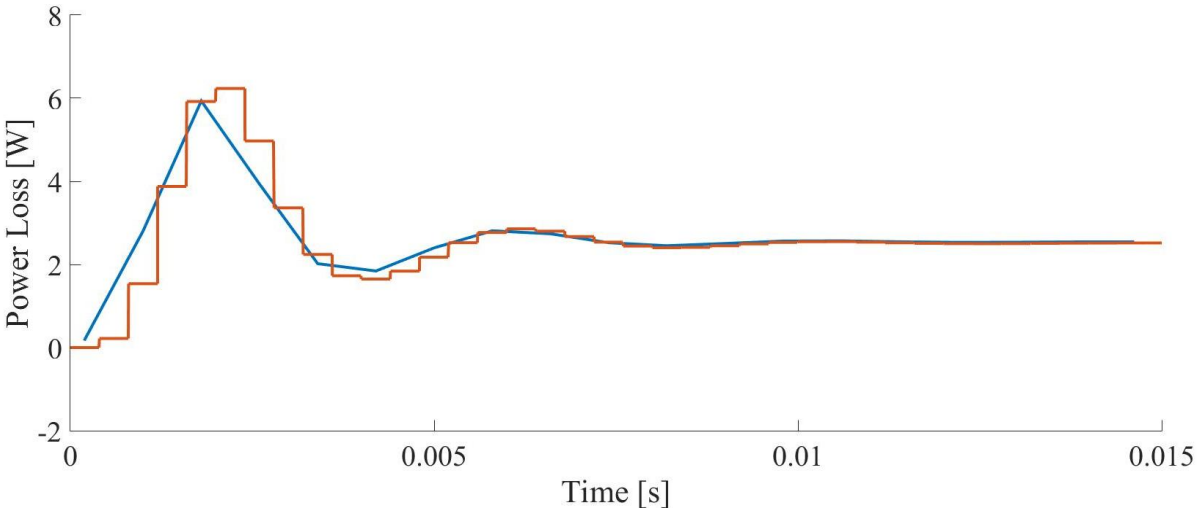


**Figure 4.** Results for simulation with 50kHz switching frequency: Orange – average total power losses for the conventional simulation; Blue – average total power losses for the suggested approach.

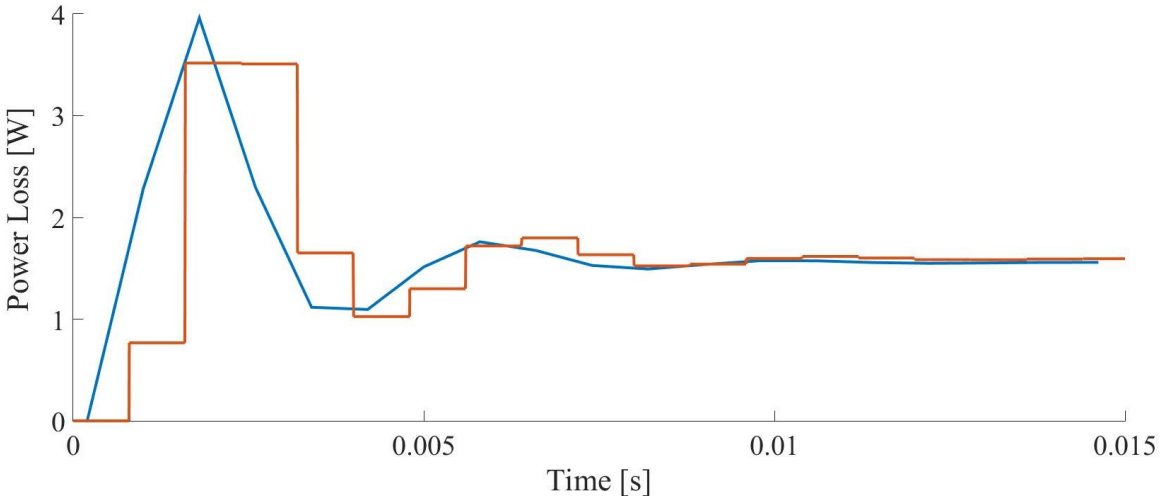


**Figure 5.** Results for simulation with 25kHz switching frequency: Orange – average total power losses for the conventional simulation; Blue – average total power losses for the suggested approach.

Results for the computing times for the carried simulations is given in table 6. It can be seen that the suggested approach significantly reduces computing time (especially for high frequencies).

| Parameter | Time at 50kHz | Time at 25kHz |
|---|---|---|
| Average time for simulation using complete analysis | 445s | 430s |
| Average time for simulation using ideal components | 0.88s | 0.53s |
| Average time for simulation for the suggested algorithm (n=20) | 44s | 64s |
| Average time for simulation for instance part of the suggested algorithm | 2s | 3.2s |

**Table 6.** Computing times for the carried simulations

*ISSN: 2367-7384*
*Issue 2, pp. 62-68, June, 2016*

## 4. CONCLUSIONS

The paper presents a specialized simulation approach applicable to power electronic convectors that allows for relatively small computing times for long simulations with small step times.

Results from the provided example show the following advantages: (1) Accuracy comparable to the one a conventional simulation; (2) Reduced computing times – up to several times compared to a conventional simulation.

During the application the following disadvantages were noted: (1) The approach takes longer time to set when a high number of capacitors and inductors are presented in the analyzed circuit; (2) The approach is applicable only to simulation software that allows scripted simulations (for example MATLAB/Simlink); (3) The designer that utilizes the approach needs to have extended knowledge of the operation of power electronic converters in order to set it up.

Future work on this topic may include: (1) Development of more complex scripts based on the suggested algorithm that can provide better automation when setting the simulation approach; (2) Applying and testing the suggested approach for more complex circuits.

## 5. ACKNOWLEDGEMENTS

The paper was developed in the frames of project NP, Scientific Fund of TU Varna, 2016 “Synthesis, modeling and analysis of new and improved solution for design of electronic devices"

## 6. REFERENCES

[1]. Georgiev A., T. Papanchev, D. Kovachev, „Evaluation of Technical Condition of a Radar with Redundancy“, International Journal of Engineering and Innovative Technology (IJEIT), Vol. 4, Issue 6, Dec. 2014, pp. 185-192, ISSN: 2277-3754
[2]. Georgiev A., N. Nikolov, Structural Reliability Analysis of A Star Structured Complex Electronic System, Electronic Journal for Computer Science and Communications, Vol.4, No: 4, pp. 44-51, ISSN: 1314-7846, 2015
[3]. Y. Kang and J. D. Lavers, "Power electronics simulation: current progress and future development," Computers in Power Electronics, 1994., IEEE 4th Workshop on, Trois-Rivieres, Que., 1994, pp. 169-174.
[4]. T. Kato, K. Inoue and T. Hashimoto, "Practical Modeling and Simulation of a Power Electronic System," Power Conversion Conference - Nagoya, 2007. PCC '07, Nagoya, 2007, pp. 477-484.
[5]. Ned Mohan, Tore M. Undeland, William P. Robbins, ”Power Electronics. Converters, Applications and Design”, Third Edition, John Wiley and SONS, ISBN: 978-0-471-22693-2
[6]. Massorio G., P. Antognetti (1993). Semiconductor device modeling with SPICE. 2nd Edition, McGraw-Hill
[7]. Shichman H., D. A. Hodges (1968). Modeling and simulation of insulated-gate field-effect transistor switching circuits. IEEE J. Solid State Circuits, SC-3
[8]. Lauritzen, P., C. Ma (1991), A simple diode model with reverse recovery, IEEE Transaction on power electronics, Vol. 6, N2