Mateusz ZAPART[1], Cezary WOREK[1]

AGH University of Science and Technology, Departments of Electronics (1)

doi:10.15199/48.2021.02.20

A mixed–mode analog–digital simulation of LLC resonant power converter with the quality factor limiter

Streszczenie. *Stany przejściowe w przetwornicach rezonansowych są trudne do wyznaczenia analitycznie. Również pomiary z wykorzystaniem modeli fizycznych są bardzo czasochłonne oraz wymagają specjalistycznego sprzętu. Jednak gdy wykorzysta się narzędzia symulacyjne proces analizy i projektowania układów energoelektronicznych ulega znacznemu usprawnieniu. W artykule przedstawiono wyniki symulacji sterowanej cyfrowo przetwornicy rezonansowej typu LLC o mocy 350W. W symulacjach wykorzystano oprogramowanie Ngspice, a jako edytor schematów KiCad. Aby otrzymać krótkie czasy symulacji i móc przeprowadzić badania kompletnej przetwornicy wykorzystano uproszczone modele podzespołów. W pracy otrzymano dobrą zgodność wyników symulacji i pomiarów. Zaprezentowany model symulacyjny wraz z narzędziami tworzy kompletne środowisko, które pozwala skrócić czas projektowania i testowania cyfrowych zasilaczy. Wyróżnikiem przedstawionego podejścia jest połączenie analogowej symulacji przetwornicy rezonansowej wraz z implementacją w pełni cyfrowego regulatora. (**Symulacje analogowo-cyfrowe przetwornicy rezonansowej LLC z ogranicznikiem dobroci**).*

Abstract. *. Electrical transients in high-order resonant power converters are difficult to solve with classical analytic analysis. It is also costly and challenging to verify some phenomena empirically. Thus, design and analysis of electronic circuits can be remarkably improved by using appropriate simulation tools, especially in the field of power electronics. In our work, a complete simulation of digitally controlled 350W LLC resonant power converter is presented. Ngspice was utilized as a simulator and KiCad was used as applicative schematic editor. In particular, the use of simplified element models in time domain analysis provide short simulation time. We achieved good consistency between simulation results and hardware measurements. The introduced simulation environment with the LLC converter model is a compelling tool that shortens design time and reduces deployment costs. Among many published scientific papers, this work is distinguished by the fact that the simulation includes a converter with full resonant circuit, quality factor limiter and digital feedback loop.*

Słowa kluczowe: symulacja, ngspice, spice, LLC, przetwornica rezonansowa, xspice, mixed-signal, cyfrowe sterowanie, ogranicznik dobroci.

Keywords: simulation, ngspice, spice, LLC, resonant converter, xspice, mixed-signal, digitally controlled converter, quality factor limiter.

Introduction

Modern power converters are becoming increasingly complex due to higher requirements, including high efficiency under wide load conditions, small dimensions and low EM emissions. An application of resonant LLC and LCLC converters allows to meet these needs. Due to resonant character of such circuits, the flowing currents are sinusoidal with low harmonics. Inductive character above resonant frequency of resonant tank gives ideal conditions for transistors in half or full bridge topologies to operate with soft switching. The circuit can operate in Zero Voltage Switching (ZVS) mode, which reduces transitioning losses to near zero. Lower switching losses eliminates the need for heatsink and results in reduced production costs and increased reliability. To achieve optimal results in a specific application, conventional analog controllers are insufficient. It is hard or even impossible to implement new design strategies, such as adaptive regulation, frequency/duty cycle regulation or diagnostics, using analog circuitry. The most relevant solution for controlling the power converter is to apply dedicated digital controllers. This class of controllers is a new trend in the power electronics industry [1]. Such a controller usually consists of a specialized microcontroller incorporating, for example, a Cortex-M4F core with peripherals designed for controlling power converter circuitry. Those peripherals include high resolution timers, fast comparators, operational amplifiers, which can autonomously cooperate with high speed ADC and DAC as well as can provide fast short circuit protection paths without CPU interaction. Timers generate precise control signals for switching transistors with resolution about hundreds of picoseconds. The microcontroller samples the input voltage signals and generates a driving signal of given frequency and duty cycle and it is responsible for appropriate output voltage and current regulation. In some solutions this also allows to design a feedback loop with hardware digital PID controllers. Controllers of this type are provided by several companies, including Texas Instruments, ST, Microchip, Infineon and other. Their application costs are significantly lower than the use of FPGA, although FPGA solutions provide more flexibility in creating unconventional projects.

A design process of digitally controlled power converters is arduous and consumes considerable amount of time. Moreover, it is often necessary to build several prototypes and test them using costly equipment. In-system testing of new driving and controlling strategies is problematic - it requires expensive power sources, electronic loads, current probes, differential voltage probes and high bandwidth oscilloscopes and above all. Any algorithm omission or program execution failure may cause disastrous hardware damage. To mitigate the above-mentioned problems, physical hardware needs to be displaced from the early stage of development process.

In this article, we present our solution that simulates an LLC power resonant converter. It incorporates STM32F334 microcontroller. Current research focuses on simulated power converters without closed feedback loops or on modeling individual parts of a complete system or even on individual electronic components [2, 3]. Lingering simulation time is caused by high circuit complexity and the usage of Spice models of electronics components. Reducing the simulations execution time is a real challenge. There are several commercial electronic simulators, for example SystemVision from Mentor Graphics. It delivers extensive base of exemplary topologies and provides VHDL AMS tool to create new element models. Its main disadvantage is often price which disqualifies it from use in small or medium-sized companies. In this article, an open source solution is described. We used Ngspice as a simulation engine and XSPICE extension, which is part of Ngspice simulator, was used to make simple transistors, diodes and digital controller models. The usage of behavioral models significantly reduces simulation time due to smaller number of nodes. Netlists were generated by KiCad schematic tool which was slightly modified to support XSPICE elements.

Closed loop system stability test using a Middlebrook's method [4] can be performed and compared with Bode 100 measurements. The introduced toolset with a convergent converter model is a new quality in the construction and development of resonant power supplies.

LLC resonant converter

Figure 1 presents a simplified circuit of the analyzed half-bridge LLC power converter. The design of this resonant tank is based on our earlier project [5].

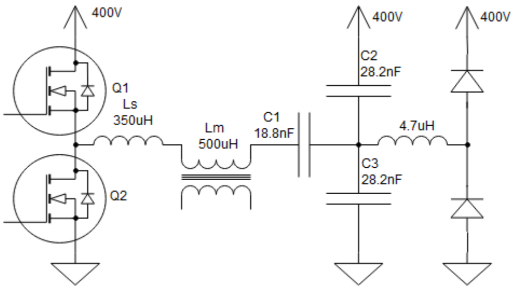

Fig.1. Overview of the components in the LLC resonant converter with quality factor limiter [5]

In the LLC resonance circuit, leakage inductance Ls is often integrated into the transformer inductance through appropriate wire winding. However, to simplify the simulation, in our solution inductance Ls is realized physically as a separate component. The purpose of this solution is to increase the Ls inductance, which significantly reduces short-circuit current in the resonant circuit. Transformer ratio is modified to obtain lower output voltage. Real circuit and simulations also includes the quality factor limiter [6]. Its role is to limit the amount of energy in resonant tank in case of working close to resonant frequency. The Quality factor limiter protects tank elements from potential damage caused by high currents or voltages.

The examined LLC resonant converter is supplied with an input voltage of 400 VDC. It provides an output voltage of 12V under a 30 A rated load. The operating frequency varies from approximately 76 kHz at full load to 120 kHz under low load conditions.

Simulation model

Figure 2 presents a schematic diagram of a simulation model. Introduced model is characterized by a short simulation time even though the time step was 10 ns. An exemplary 100 ms transient simulation took approximately 6 minutes on a modern laptop. The feedback loop parameterization paves the way for stability testing using step response or transmittance analysis. All parasitic elements use linear model of resistance, inductance or capacitance.

To achieve short simulation time, we created our own simplified generic models of diode, switch and output voltage controller. The use of this solution gives us the ability to quickly change key parameters to meet the desired requirements. The models were implemented in C language in conformance with the simulator interface.

A diode is described with Shockley equation with series resistance. A switch operates with high impedance input, with configurable on and off levels. Transition time is also specified - sharp edges are smoothed using logarithmic function. Static parameters such as on and off resistances are also included. Inductors and transformer are created with usage of classical reluctance model. Additional RC elements were added in parallel to obtain similar resonance parameters (amplitude and frequency) compared to the real resonant circuit [7]. Losses in resonant circuit were introduced as series resistors. In simulation, the change of temperature is not considered, but the inductance drift due to current change is accomplished by providing core B-H curve. The introduced model consists of half-bridge incorporating two switches with additional gate-drain capacitances to simulate the Miller effect. Switches are driven with square wave directly by the controller. Switching bridge is working in class DE [8]. Therefore, drain-source capacitance value is a sum of internal transistor capacitance and externally added capacitors.

Secondary side is made with center tapped synchronous rectifier. The ESR of output capacitors is modelled as a series resistance. The LC filter smoothing the output voltage also has its own series resistors modeling the parasitic resistance. The feedback voltage to the controller is taken from the rectifier output.

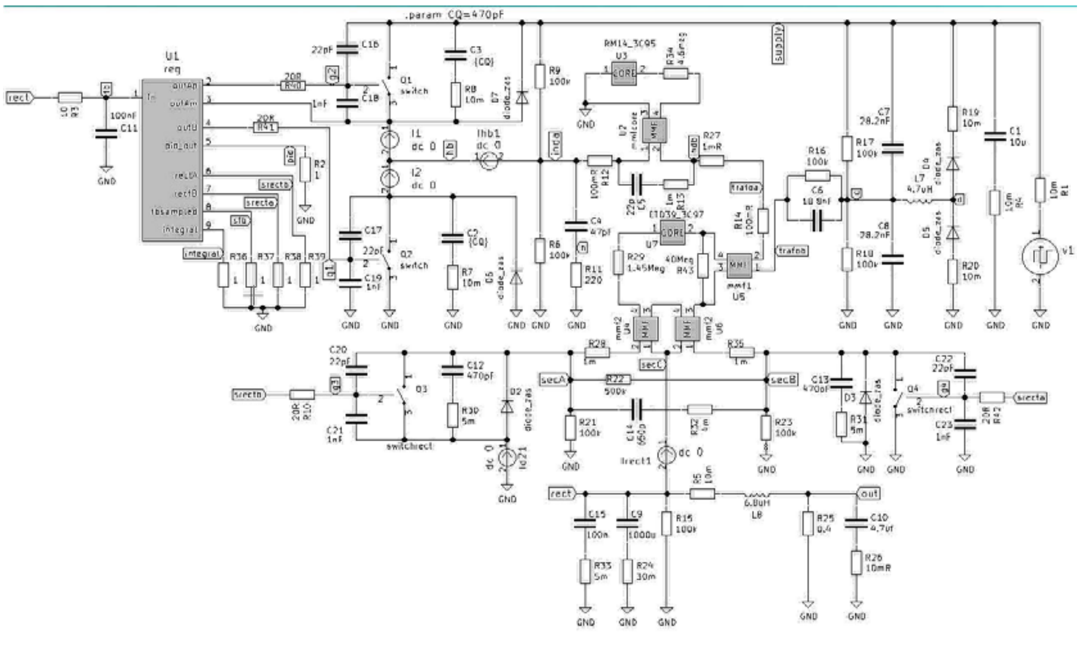

Fig.2. Schematic of the simulated system with measured values of inductances and parasitic elements of real LLC resonant circuit.

Controller

This part of simulation has been implemented in C language as the XSPICE extension model. The basic function of this controller is to maintain a stable output voltage by changing the half-bridge frequency. Its block diagram is presented in Figure 3.

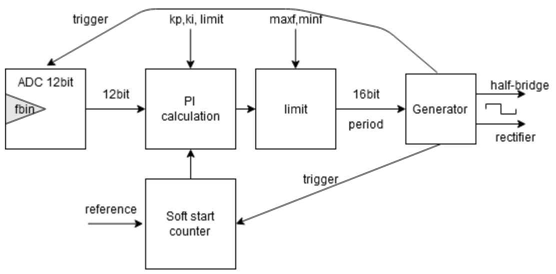

Fig. 3. Output voltage controller block diagram

The output voltage from the rectifier is provided through an analog filter to obtain desired loop bandwidth. From this signal an error voltage is generated and fed to the PI controller. The controller also implements a soft-start functionality based on linear increase of reference voltage. It raises from 0 V with a parametrized time constant. This mechanism is necessary in real power converters because of high output capacitance. The slower the output voltage increases, the lower the initial charging current.

Next block implements a classical PI controller with anti-windup. Proportional, integral coefficients are parametrized. Thus, they can be changed at any time, even during simulation. This provides the foundation for adaptive algorithms. A value of the PI controller output signal is clamped to the predefined range. The controller output signal is a measure of the period of the signal. The generator block forms an appropriate control signal to the half-bridge. Output waveforms are present only when simulation time is above a specified delay value. Controller block has four driving outputs. One with reference to ground, which drives the bottom transistor. The second one is inverted and level shifted to drive the upper transistor. Switching slopes of both signals are time-shifted to include the dead time. The dead time is a period of time when both outputs are turned off to give time for the transistors to turn on and off completely which eliminates cross conduction in the half-bridge circuit. The resonant converter works with soft switching. Therefore, transistor losses are minimalized. This mechanism cannot work properly without a dead time which must be long enough to discharge additional capacitors at light load conditions.

The controller model interposes additional two outputs to drive synchronous rectifier. The outputs have independent deadtime parameter. Switching points are synchronous with half bridge but have reduced duty due to larger deadtime.

Model provides a sufficiently accurate result and is simple enough to reduce calculation time. The potential of such a simulation model is huge. For example, the stability can be tested under various operating conditions, e.g., with a drop-in supply voltage, interference of input signals, variable load and many others.

Hardware

The examined LLC resonant converter is supplied with an input voltage of 400 VDC. It provides an output voltage of 12 VDC under a 30 A rated load.

Figure 4 presents block diagram of a physical model. The resonant circuitry is driven by a half-bridge MOSFET switch. The rectifier at the secondary side is based on a centered tap topology and it is realized as a synchronous rectifier incorporating low channel resistance MOS transistors. Secondary microcontroller integrates frequency-controlled oscillator which drives switching transistors and synchronous rectifier.

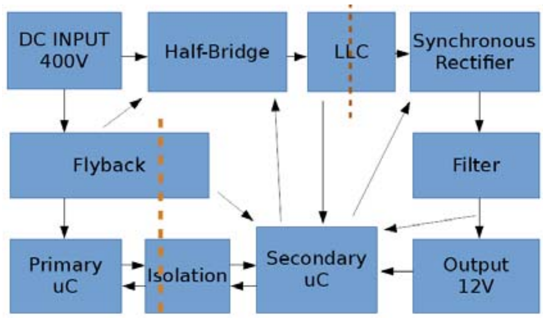

Fig. 4. Physical model block diagram

Switching frequency is set by a feedback loop and the difference between output voltage and a reference creates error signal which is provided to a software PID controller.

The block diagram shows additional microcontroller on the primary side. It is connected with the secondary microcontroller through isolated serial interface. It serves to monitor the input voltage and prevent the resonance converter from working under too high or too low supply voltage. On the primary side, the temperature of half-bridge transistors is monitored. The primary microcontroller only transmits data to secondary controller.

The power converter requires additional power supply for the microcontroller, gate drivers and temperature sensors. The required voltages are generated by the built-in flyback converter with multiple secondary windings. Hardware design allows to measure the power consumption of individual functional blocks. Therefore the operation of all components of the system can be measured. The transformer is wound on the ETD39 core based on 3C97 ferrite material. The inductor is created with RM14 core based on 3C95 ferrite material.

Fig. 5. Picture of the simulated hardware.

The photograph presented in Figure 5 shows the actual physical realization of the simulated power converter. The board is an evaluation model, it has many testing points and vast amount of space to use elements in different packages. Interesting feature is that the controller board is mounted with a connector to the motherboard. This allows testing of various controllers in the same prototype.

The proposed design was tested with STM32F334 as a driving microcontroller. The controller generates signals for half-bridge transistors and a synchronous rectifier. Moreover, it implements feedback loop. STM32F334 consists of Cortex-M4F core clocked with 72 MHz. The high-resolution timer allows generation of transistor gate signals with a resolution of 217 ps. The peripheral device allows to generate dynamically tuned dead time. Voltage measurements for feedback loops are measured using the 12 bits, 5 MSPS integrated analog-to-digital converter.

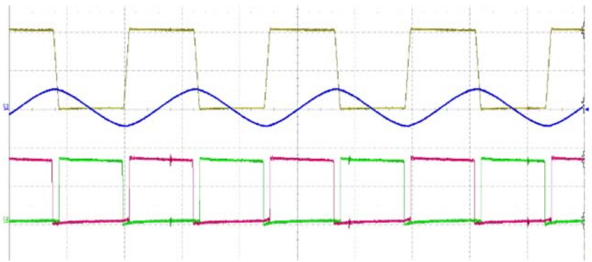

Fig. 6. Converter measurements for 100mA output load (5 us/div).Yellow: half-bridge voltage (200 V/div); blue: Ls current (2 A/div); pink and green: half-bridge driving signals (2 V/div).

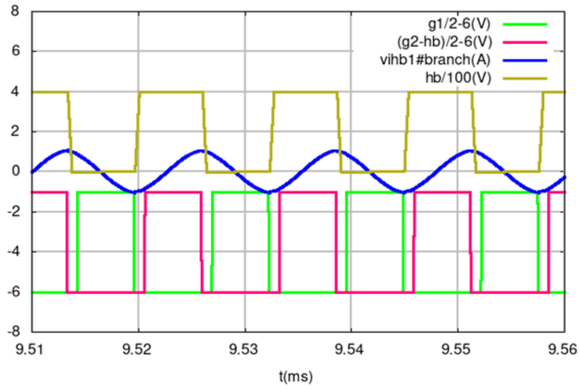

Fig. 7. Converter simulations for 100mA output load

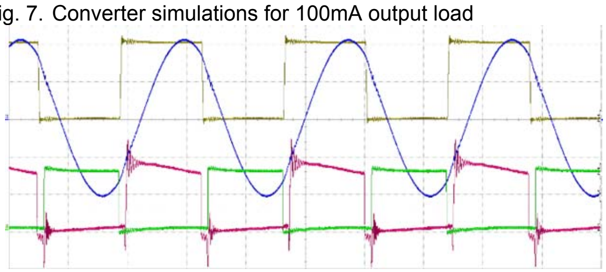

Fig. 8. Converter measurements for 30 A output load (5 us/div). Yellow: half-bridge voltage (200 V/div), blue: Ls current (2 A/div); pink and green: half-bridge driving signals (2 V/div).

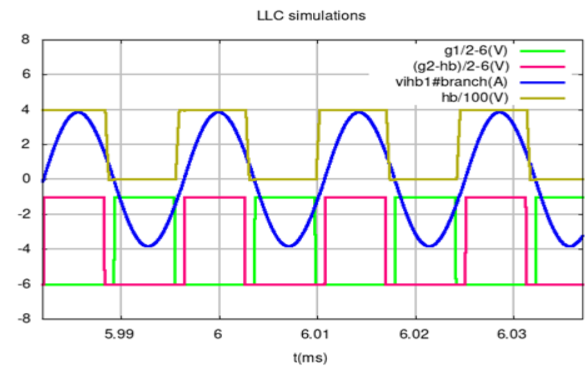

Fig. 9. Converter simulations for 30 A output load.

Simulations

Presented simulations are based on a simplified model. Hence there are some differences between the simulation and hardware. The magnetic elements do not contain information about core material and copper losses. However, this model is accurate enough to perform a comparative study (from Figure 6 to Figure 9).

Waveform comparison

A half-bridge current was measured and compared with simulation results. It is also called a tank current and is measured on inductor Ls shown in Figure 1. Two attempts were performed, one for small output current, second for maximum load.

Figure 6 presents measurements for small current. The output load current was set to 100 mA. Half-bridge voltage waveform is tabbed with yellow color. It changes from 0 V to 400 V. Both top and bottom transistor driving signals are presented on the diagram. Long dead time values can be observed. Measurements in Figgure 6 can be compared with simulation results shown in Figure 7. Current shape and amplitudes are almost identical. The tank current phase shift remains the same for simulation and measurement.

Current in the same circuit was also measured at full load conditions. The results are presented in Figure 8. Resonant curve amplitude is slightly above 4 A. A simulated version from Figure 9 shows the current amplitude of the same circuit. Its value is slightly below 4 A. We expected a small difference between measurement and simulation results in such conditions of the power supply.

Efficiency comparison

The next stage of comparative research was the assessment of efficiency.

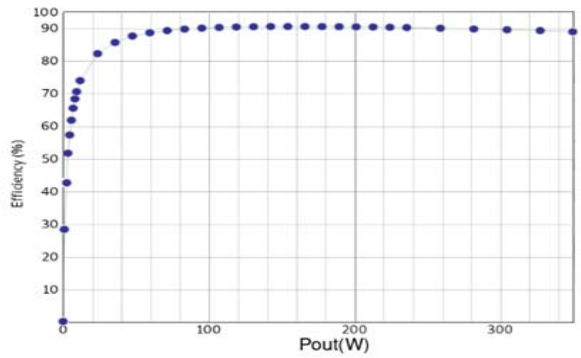

Fig. 10. Measured converter efficiency.

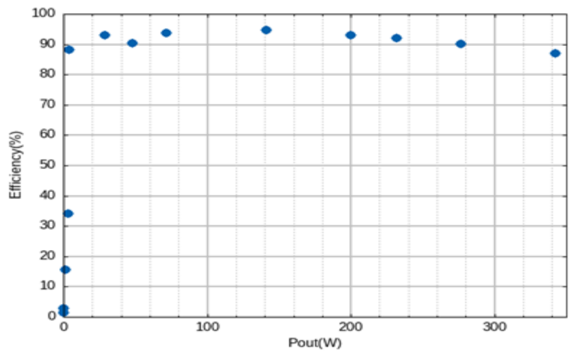

Fig. 11. Simulated converter efficiency.

Figure 10 shows the physical measurements, while simulated results are in Figure 11. Efficiency of the simulated model does not converge with the real measurement. It differs significantly at low and high loads. At this point, it can only be presumed that at low loads the simulation shows better results due to the lack of a core loss mechanism. At high output current, the simulated efficiency is slightly different.

Conclusions

The presented simulation environment and the LLC power converter model is a powerful tool that significantly streamlines the design and testing process. Despite the use of simplified models, simulation results are in consistency with hardware measurements. In addition, short simulation times accelerate the design work. Our future plans include measuring the stability of the power supply for varying parameters of capacitors and parasitic elements. Capacitor parameters change with operating conditions, including temperature, mean voltage, operating frequency (especially for X5R and X7R dielectrics) and time. This phenomenon is particularly important in current control systems, in which the output voltage varies over a wide range. It is equally important in frequency-controlled LLC and LCLC resonant power converters due to dependence of capacitance on frequency. In spite of applying some simplifications, the presented environment proved to be very useful in development of mixed-mode analog–digital simulation of resonant power converter with digitally controlled feedback loop.

This work was supported by AGH University of Science and Technology.

***Authors**: mgr inż. Mateusz Zapart, AGH University of Science and Technology, al. Adama Mickiewicza 30, 30-059 Kraków, E-mail:* mzapart@agh.edu.pl*; dr hab. inż. Cezary Worek, AGH University of Science and Technology, al. Adama Mickiewicza 30, 30-059 Kraków, E-mail:* worek@agh.edu.pl*.*

REFERENCES

[1] Liu Y., Meyer E., Liu X., Recent Developments in Digital Control Strategies for DC/DC Switching Power Converters, *IEEE Transactions on Power Electronics*, 24 (2009), n.11, 2567-2577
[2] Pereira D., Braga H., SPICE simulation and evaluation of a GaN-based synchronous full-bridge resonant converter, 12th *IEEE International Conference on Industry Applications (INDUSCON),* (2016)
[3] Li X., A LLC-Type Dual-Bridge Resonant Converter: Analysis, Design, Simulation, and Experimental Results, *IEEE Transactions on Power Electronics*, 29 (2014), n.8, 4313-4321
[4] Mysinski W., Pragłowska-Ryłko N., Use of middlebrook's method in frequency analysis of power electronic circuits, *Technical Transactions, (2016)*
[5] Worek C., Ligenza S., Design and implementation of a LLC resonant converter with high dynamic controls for LED applications, *Przeglad Elektrotechniczny*, 93 (2017), n.8, 46-49
[6] Worek C., Ligenza S., Widórek R., LLC resonant converter with quality factor limiter, *18th European Conference on Power Electronics and Applications (EPE'16 ECCE Europe*), (2016)
[7] Cox F.L., Kuhn W.B., Murray J.P., Code-level modeling in XSPICE, *IEEE International Symposium on Circuits and Systems*, (1992)
[8] Hamill D.C., Class DE inverters and rectifiers for DC-DC conversion, *27th Annual IEEE Power Electronics Specialists Conference*, (1996)