Extending the Possibilities of the PSpice Simulation in the Power Electronics Using Postprocessing in the Graphical Analyzer Probe

Elissaveta D. Gadjeva[1], Georgi Tz. Kunov[2] , Mario E. Ivanov[3]

Abstract – **The possibilities for advanced simulation of power electronic circuits are investigated using the PSpice program and the incorporated functions of the graphical analyzer Probe. The proposed investigations are applied to the basic power electronic circuits: thyristor rectifiers (single phase, three-phase one-wave, three-phase full-bridge rectifiers), bridge resonant inverter as well as current fed inverter. Similarly, the extended possibilities can be applied to another power electronic circuits.**

Keywords – **Power electronics, Computer modeling, PSpice simulation, Power converters, Parametric analysis.**

I. Introduction

The general-purpose *PSpice*-like simulators are widely used for the analysis of variety electrical and electronic circuits [1,6,7]. The time-domain analysis is often applied due to the specific characteristics of the electromagnetic processes which take place in the power electronics. The analysis results are highlighted with the voltage and current waveforms of the building elements in the time domain. These dependencies, however, can not describe some important circuit characteristics such as the regulation abilities, the load characteristics, as well as the circuit stability as a function, for instance, of the load variation [2,3,4,5].

The parametric analysis is widely used for a detailed investigating of electronic circuit of different kinds. It is applied to construct the regulation characteristics of the thyristor rectifiers.

The *Append* function allows the obtaining of family of curves and gives the possibility to compare the regulation characteristics of different rectifiers.

The load and regulation characteristics in power electronics are analyzed using generalized parameters such as quality factor Q (for the resonant inverters) and load coefficient B=1/(fRC) for the current fed inverters. The parametric analysis allows the load resistance calculation depending on the introduced generalized parameter.

The usage of macro-definitions in the graphical analyzer *Probe* allows to extend significantly the possibilities for constructing of generalized parameters. Based on macros, using the incorporated analog functions in *Probe*, a number of characteristics can be calculated: **RMS**, mean values, powers, time intervals and another quantities, which characterize the power electronic circuit.

II. Determination of Rectifier Regulation Characteristics

A. *Single phase rectifier* (*Fig. 1*)

In order to construct the regulation characteristics, the on-time delay with respect to the start value of the corresponding positive half-wave, is defined as a parameter.

The following parameter values are used for the sources:

– Vac is a sinusoidal voltage source of **VSIN** type [magnitude V=50V, frequency F=50Hz, time delay TD=0, initial phase PHASE=0]

– Ig1, Ig2 are pulse current sources of **IPULSE** type [I1=0, I2=0.5A, TD={TDvar}, TR=TF=1µs, PW=500µs, PER=20ms]

The on-time delay TD with respect to the start value of the corresponding positive half-wave, is defined as a parameter TDvar.

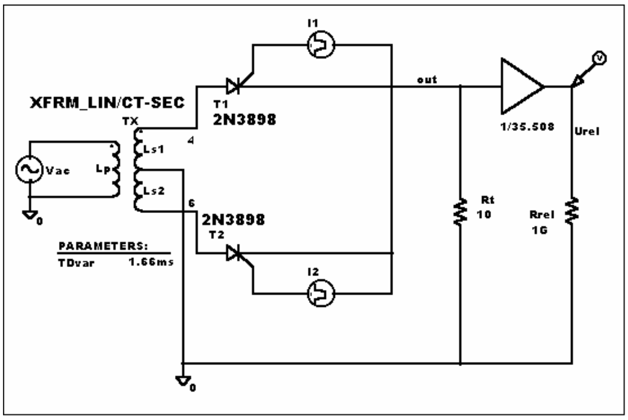

Fig. 1. Single phase rectifier

TD={TDvar} for the thyristor T1 and TD={TDvar + 10ms} for the thyristor T2. A linear variation is defined for TDvar from 0ms to 10ms by an increment of 0.5ms.

The **GAIN** block is introduced in order to obtain the output voltage in normalized form: GAIN=1/Vout(TDvar=0).

[1]Elissaveta D. Gadjeva is with the Faculty of Electronic Engineering, Technical University of Sofia, Kliment Ohridski Str. No 8, 1000 Sofia, Bulgaria, E-mail: egadjeva@tu-sofia.bg

[2]Georgi Tz. Kunov is with the Faculty of Electronic Engineering, Technical University of Sofia, Kliment Ohridski Str. No 8, 1000 Sofia, Bulgaria, E-mail: gkunov@tu-sofia.bg

[3]Mario E. Ivanov is with the Faculty of Electronic Engineering, Technical University of Sofia, Kliment Ohridski Str. No 8, 1000 Sofia, Bulgaria, E-mail: mario@tu-sofia.bg

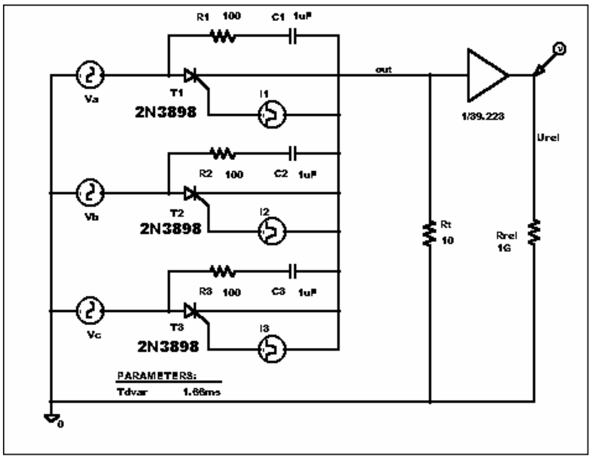

Fig. 2. Three-phase one-wave rectifier

B. Three-phase one-wave rectifier (*Fig. 2*)

The following parameter values are used for the sources:
– Vac1, Vac2 and Vac3 have identical parameters with the source used in Fig. 1. The only difference is in the phase. The values PHASE1=0, PHASE2=-120^o and PHASE3=120^o define the three-phase system of voltages.
– Ig1, Ig2, Ig3 have identical basic parameters with the sources used in Fig. 1. The difference is in the delay TD. It defines the beginning and the interval of regulation.

TD={TDvar+1.66ms} for the current Ig1, TD ={TDvar+ 8.33ms} for Ig2 and TD={TDvar+15ms} for Ig3. A linear variation is defined for TDvar from 0 ms to 8.5ms by an increment of 0.5ms.

C. Three-phase full-bridge rectifier (*Fig. 3*)

The differences in the source values with respect to those used in Fig. 1 and Fig. 2, are in the duration of the on-time pulses PW=3.5ms and in the time delays TDvar, as shown in Table 1. A linear variation is defined for TDvar from 0 ms to 7.5ms by an increment of 0.5ms.

TABLE I
TIME-DELAY VALUES

Ig1	Ig2	Ig3	Ig4	Ig5	Ig6
1.66ms	5.0ms	8.33ms	11.66ms	15ms	18.33ms

A linear variation is defined for TDvar from 0 ms to 8.5ms by an increment of 0.5ms.

D. PSpice simulation of rectifier regulation characteristics

As a result of the simulation, the regulation characteristics are obtained as a dependence of the normalized mean value of the rectified voltage regulation characteristics with respect to the on-time angle (represented by TDvar). This characteristic is shown in Fig. 6. The value of the rectified voltage in the normalized form are obtained at the output of the block **GAIN**. The gain is defined in the form:

$$GAIN = \frac{1}{V_{avg,out}\big|_{TD=0}}$$

where $V_{avg,out}\big|_{TD=0}$ is average value of the rectified voltage for *TD*=0.

The calculation of the average normalized value of the rectified voltage *Uoutrel* is obtained using the postprocessing in the graphical analyzer *Probe*. It is defined using the following macrodefinition in the *Trace/Macros* menu:

Uoutrel = Max(Avg(V(Uout))).

The family of regulation characteristics is obtained following the next procedure:

1. Plot/Axis Settings/X_Axis/Performance Analysis
2. Trace/Add…/Macros/Uoutrel.

Using the *File/Append* option, it is possible to visualize on the same screen and compare the regulation characteristics of the rectifier circuits in Fig 1, Fig. 2 and Fig. 3. The corresponding result is shown in Fig. 4.

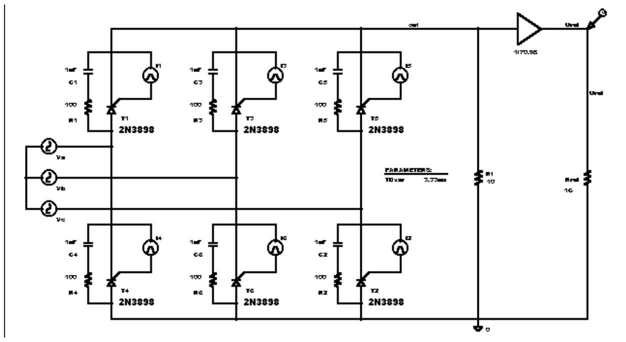

Fig. 3. Three-phase full-bridge rectifier

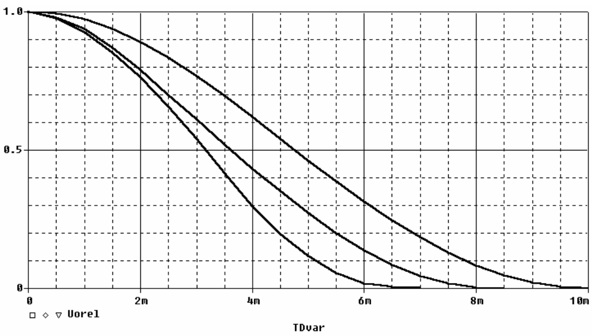

Fig. 4. Regulation characteristics

III. DETERMINATION OF TIME INTERVALS

The *PSpice* simulator allows the investigation of time intervals depending on parameters. As an example, the on-state time interval is obtained depending on the quality factor *Q* fir the full-bridge resonant inverter shown in Fig. 5. Ig1, Ig2, Ig3 and Ig4 are pulse current sources of IPULSE type with the following parameters: I1=0, I2=0.5A, TR=TF=1μs, PW=50μs, PER=200μs.

TD=0μs for Ig1 and Ig2 and TD=100μs for Ig1 and Ig2.

The circuit parameters are: Lvar = 200μH; Cvar = 2μF. Hence

Rvar = {SQRT(2*Lvar/Cvar)/Q}. (1)

The quality factor *Q* of the series resonant circuit is defined as a global parameter with a linear variation from 1.0 to 2.0 by an increment of 0.1. The load resistor value Rt is automatically calculated from (1).

The waveforms of the voltage across one of the thyristors, the current through the load, as well as the voltage across the load, are shown in Fig. 6.

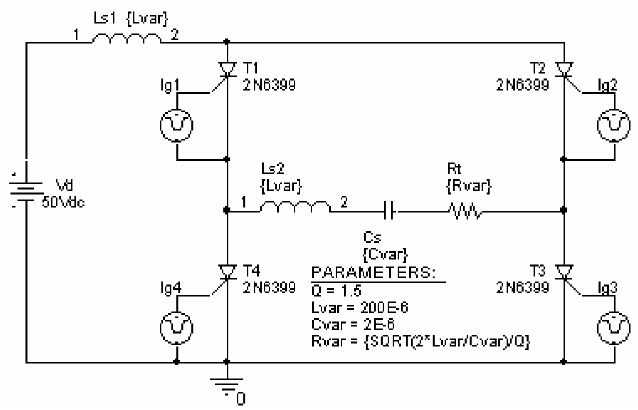

Fig. 5. Bridge resonant inverter

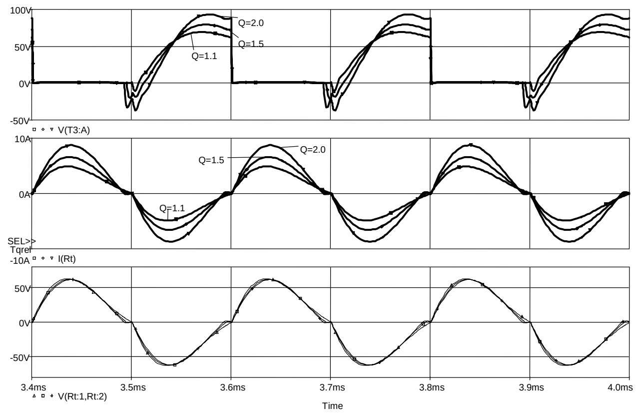

Fig. 6. The basic waveforms of tne resonant inverter

It is seen that the increasing of *Q* leads to increasing of the thyristor voltage and current. The turn-on time T_t decreases and the turn-off time T_q increases. The load voltage does not change its value significantly.

The normalized values $T_{q\,rel}(Q)$ and $T_{t\,rel}(Q)$ of the corresponding parameters T_q and T_t with respect to the commutation period PER, as well as the load power $P_r(Q)$ are shown in Fig. 7.

The following macro-definitions are constructed for this purpose:

PER=200E-6
NUM=20
Tqrel = (XatNthYp(V(T3:A),0.1, NUM)-XatNthYn(V(T3:A),-0.1, NUM))/PER
Ttrel = (XatNthYn(V(T3:A),-10, NUM-1)-XatNthYn(V(T3:A),10, NUM-1))/PER
Pr = Max(RMS(I(Rt)))*Max(RMS(V(Rt:1,Rt:2)))

The *Probe* function **XatNthYp(1,Y_value,n_occur)** is used to obtain the argument **X** when the function **1** reaches the **Y_value** for the **(n_occur)**-th time in the positive direction. Similarly, the function **XatNthYn(1, Y_value,n_occur)** is used for the negative direction. The **RMS** function is used for the load power calculation.

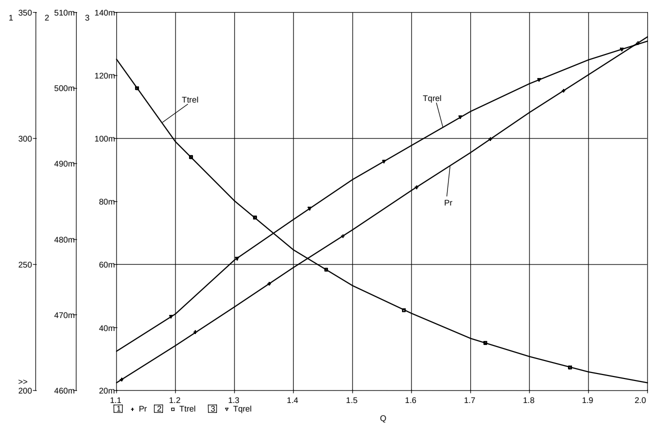

Fig. 7. The normalized values Tqrel Ttrel and the load power

IV. DETERMINATION OF THE LOAD CURVES

The bridge current fed inverter (Fig. 8) is used as an example, where the circuit elements are calculated for a given load coefficient *B*, defines as a parameter.

The current fed inverter circuit is simulated for the frequency F = 1kHz, corresponding to PER=1ms, and the time-delay of the pulse sources Ig2 and Ig4 is TD=0.5ms. The load coefficient *B* is defined as a global parameter.

$$B = \frac{1}{F.R_{var}\ C_{var}} \tag{2}$$

It is changed using value list:

The circuit parameters are: Cvar = 10μF; F = 1kHz. Hence

Rvar = {1/(F*Cvar*B)}. (3)

The normalized values of the load voltage *Urtrel*, the DC fed current from the source *Idrel*, as well as the turn-off time *Tqrel* are shown in Fig.9 depending on the load coefficient.

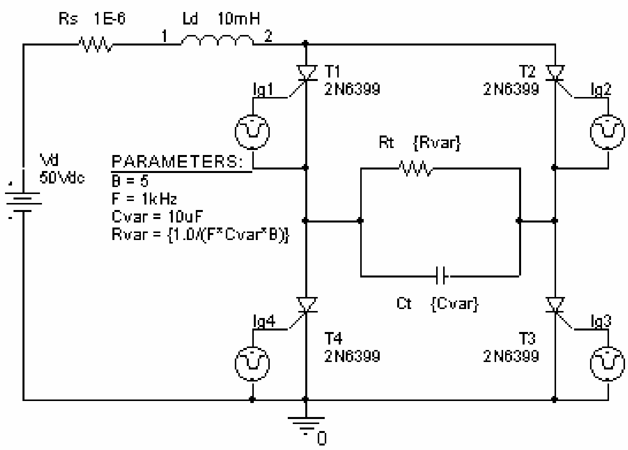

Fig. 8. Bridge current fed inverter

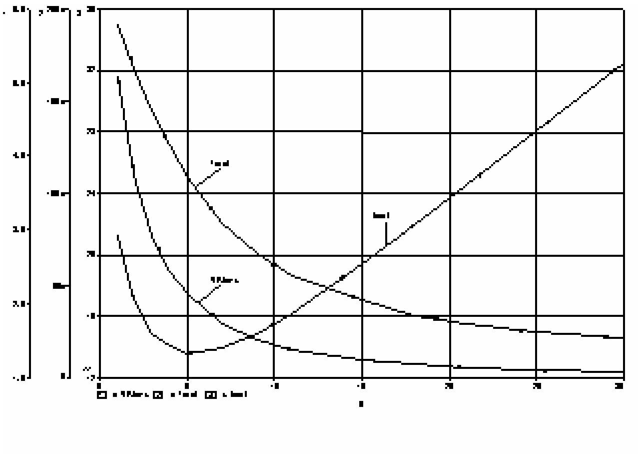

Fig. 9. Load characteristics of the current fed inverter

The following macro-definitions are constructed for this purpose:

PER=1E-3
NUM=20
Vd=50V
Cvar=10uF
Tqrel = (XatNthYp(V(T3:A),0.1, NUM)-XatNthYn(V(T3:A),-0.1, NUM))/PER
VRtrel = Max(RMS(V(Rt:1,Rt:2)))/Vd
Idrel = Max(I(Rs))*PER/(Vd*Cvar)

The *Probe* functions **XatNthYp** and **XatNthYn** are used for the *Tqrel* calculation, and the **RMS** function is used to the *VRtrel* determination.

V. CONCLUSION

An approach is proposed to computer-aided investigation of power electronic circuits using the *PSpice* simulator and the extended possibilities for postprocessing using the graphical analyzer *Probe*. Based on macromodel approach, the families of regulation characteristics are constructed for rectifier circuits. The dependence of the turn-on time and turn-off time of the thyristors for resonant inverters is obtained, as well as the load characteristics of the current fed inverters. The proposed approach can be extended to another power electronic circuit types.

REFERENCES

[1] R. Ramshaw and D. Schuurman, “PSpice Simulation of Power Electronics Circuits”, Chapman & Hall, 1997.

[2] N. Mohan, T. Undeland, W. Robbins, “Power Electronics: Converters, Applications and Design”, John Wiley & Sons, Inc., 1989.

[3] N. Nachev. G. Malleev, “Power Electronics”, Technika, Sofia, 1979 (in Bulgarian)

[4] S. Tabakov, “Power Electronic Circuits and Devices”, Technika, 1987 (in Bulgarian).

[5] Bobcheva M., S. Tabakov, P. Goranov, Converter Devices, Tekhnika, Sofia, 2002. (In Bulgarian).

[6] G. Kunov, G. Staichev, E. Gadjeva, “Spice Modeling and Optimization of a Frequency-Controlled Inverter/Double Current Rectifier System", 3-th Intern. Workshop “Compatibility in Power Electronics” CPE’2003, Gdansk, pp.289-292.

[7] OrCAD PSpice and Basics, Cicuit Analysis Software, OrCAD Inc., USA, 1998.