Innovative Lifelong e-Learning for Professional Engineers

(e-ProfEng)

586391-EPP-1-2017-1-SE-EPPKA2-CBHE-JP

Training in Electrical Engineering Discipline

Modelling and Simulation in Electrical Engineering

Teaching Materials for Topic 4

Modelling and Simulation of Power Electronic Converters

Authors:

Denis Pelin, P9 FERIT

Andrej Brandis, P9 FERIT

Contents

1. Introduction

Power electronic converters are devices that connect the network of default performance characteristics with the loads of the desired characteristics. The electrical engineering discipline that deals with the design and application of power electronic converters is called power electronics. The demand for electricity saving as well as the increasingly cost-effective characteristics of the loads caused the faster and wider application of the power electronic converters, from the battery charger of the mobile phones powered by a few watts to the converters for application in renewable energy sources like control of electrical energy production from wind generators powered by a several megawatts. As a result, the number of electrical engineers (power engineering, electromechanical engineering and electronics) who are in practice encountered with power electronic converters is constantly increasing. This also increases the importance of creating competent engineer personnel.

This course is intended to give knowledge of modelling and simulation the basic topology of power electronic converters. First, in theoretical part of the course, analysis of the mathematical model of chosen converters will be performed. Three characteristic topology of the power electronic converter will be analysed: DC-DC converter topology, rectifier topology and finally the inverter topology. In the second part of the course the simulation of the introduced power converter circuits together with control circuits will be carried out by using Typhoon HIL software. Typhoon HIL's real time simulation HIL testing software provides tools such as Schematic Editor, HIL SCADA, Test suite and Power Systems Toolbox. The simulation results will be verified in laboratory conditions on chosen power converter circuits.

2. DC-DC converter: Boost converter topology

The DC-DC voltage converters are electrical devices that connect two DC systems of different voltages. Depending on the models of sources and loads with non-isolated converter can be connected the voltage source and inductive load or the current source and the capacitive load. The DC-DC voltage converter symbol is shown in Fig. 2.1.

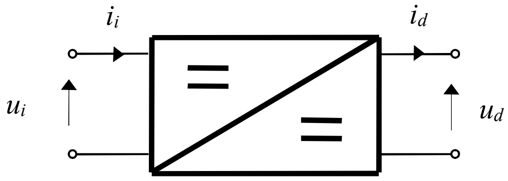


Fig.2.1: *Symbol of the DC-DC converter*

Two basic non-isolated DC-DC voltage converter topologies are buck or step-down and boost or step-up converter. The term topology refers to the way in which the converter components are interconnected.

2.1. Boost converter modelling

The converter switching components are modelled with ideal elements (V1 with control switch and V2 with ideal diode). The load can be considered as constant voltage source and labelled with U_d, ($C_d R_d \gg T_s$), Fig.2.2. The serial connection of the voltage source and inductivity is considered to act as a current source whose current has ripple due to the finite inductance of the L_E.

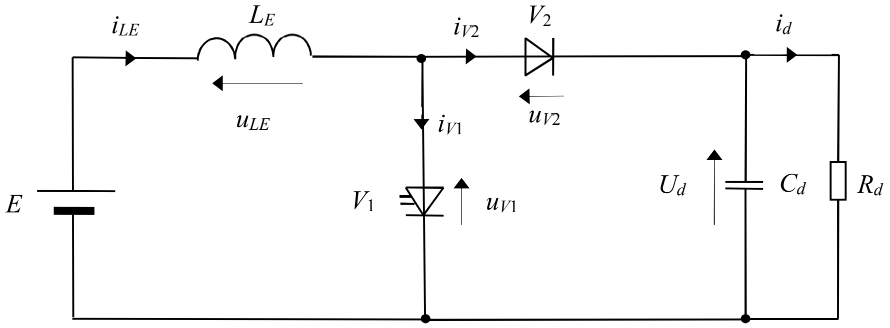


Fig.2.2 Equivalent circuit of the boost converter

For the presumed scheme of the DC converter circuit according to Fig.2.2 the following network equations can be applied:

$$E = u_{LE} + u_{V1} \tag{2-1}$$

$$u_{V1} = u_{V2} + U_d \tag{2-2}$$

$$i_{LE} = i_{V1} + i_{V2} \tag{2-3}$$

The continuous conduction mode of the boost converter is assumed. In continuous conduction mode the inductance current is higher than zero during the whole period of T_s. Since the continuous mode is assumed, in the analysis of the equivalent circuit there are two intervals within the switching period of operation:

- interval A; switch V_1 is in state ON and ideal diode V_2 is in state OFF (Fig. 2.3),
- interval B; ideal diode V_2 is in state ON and switch V_1 is in state OFF (Fig. 2.4).

Interval A

The interval A starts at time t = 0 when the impulse is given to switch V_1. Since in this interval switch V_1 is in state ON and ideal diode represent by V_2, is in state OFF when applying the network equations from 2-1 to 2-3; $u_{V1} = 0$ and $i_{V2} = 0$ will be set.

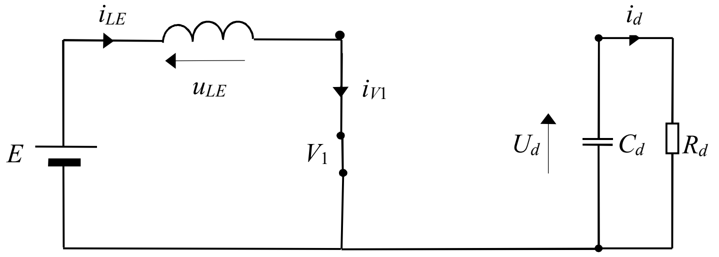


***Fig.2.3:** Equivalent circuit of the boost converter for interval A*

From equation 2-1 can be obtained: $E = L_E \frac{di_{LE}}{dt}$, and by using integration the expression for the current i_{LE} is calculated as:

$$i_{LE} = \frac{E}{L_E} t + i_{LE}(0) \tag{2-4}$$

where $i_{LE}(0) = I_0$, is initial value of the inductance current.

Current of control switch V_1 is determined from 2-3; $i_{V1}=i_{LE}$, and voltage of ideal diode V_2 is determined from 2-2 as $u_{V2} = -U_d$.

Interval A is ended when impulse for turn OFF came on controlled switch V_1.

Interval B

Since in this interval ideal diode V_2 is in state ON and control switch V_1 is in state OFF, when applying the network equations 2-1 to 2-3 will be worth: $u_{V2} = 0$ i $i_{V1} = 0$.

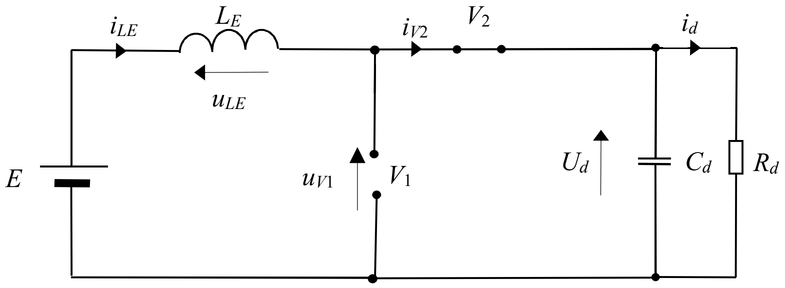


Fig.2.4: *Equivalent circuit of the boost converter for interval B*

From 2-1 and 2-2 follow: $E = L_E \frac{di_{LE}}{dt} + U_d$, and by using integration the expression for the current i_{LE} is calculated as:

$$i_{LE} = \frac{E - U_d}{L_E}(t - \alpha T_s) + i_{LE}(\alpha T_s) \tag{2-5}$$

where $i_{LE}(\alpha T_s) = I_1$, is initial value of inductance current but only on interval B and α is duty cycle: $\alpha = \frac{T_{V1}}{T_s}$.

The voltage of control switching V_1 is determined from 2-2; $u_{V1} = U_d$, and ideal diode current from 2-3; $i_{V2} = i_{LE}$.

Since the mean value of the inductance voltage in the periodic operating mode is equal to zero; $U_L(0)= 0$; for waveform of inductance voltage is obtained as it is shown on Fig. 2.5 :

$$\alpha E T_s = (1 - \alpha)(U_d - E)T_s$$

Which is condition for obtain control characteristic of boost converter:

$$\frac{U_d}{E} = \frac{1}{1 - \alpha} \tag{2-6}$$

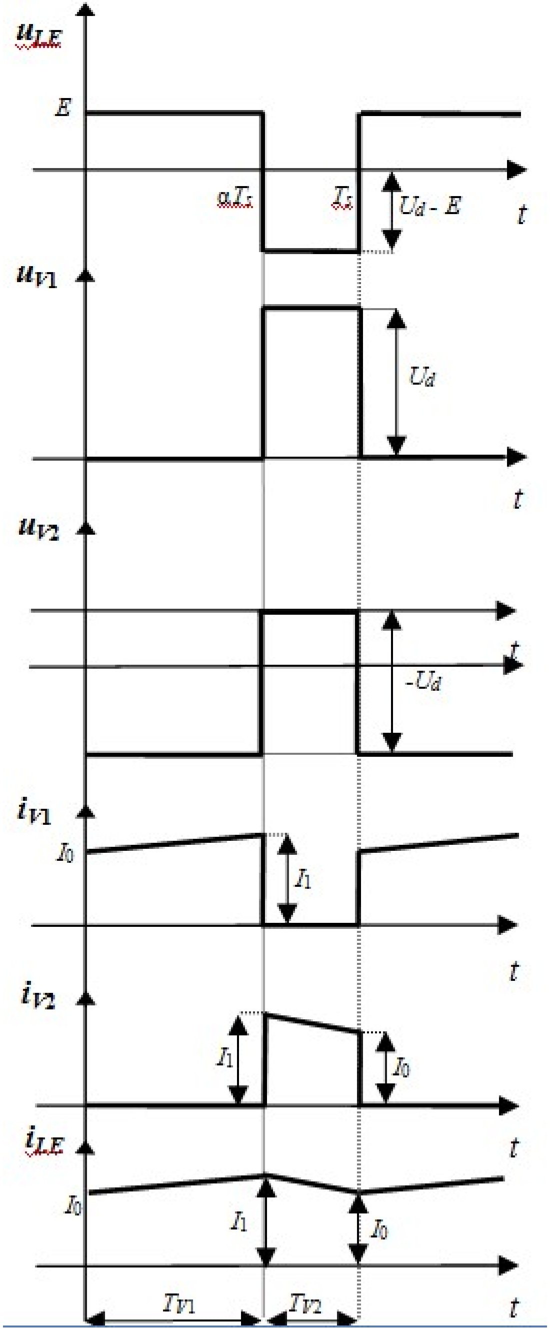


***Fig.2.5:** Characteristic waveforms of current and voltage for the boost converter*

2.2. Simulation with Typhoon HIL software - Introduction

A Typhoon HIL is the one of the test solutions for power electronics control systems using Hardware in the Loop (HIL) real time simulation. This package includes the hardware (interface) and the software Typhoon HIL control centre (Fig.2.6). The simulations can be performed without hardware, although not in the real time. The desktop icon of the Typhoon HIL Control Centre is shown in Fig.2.6.

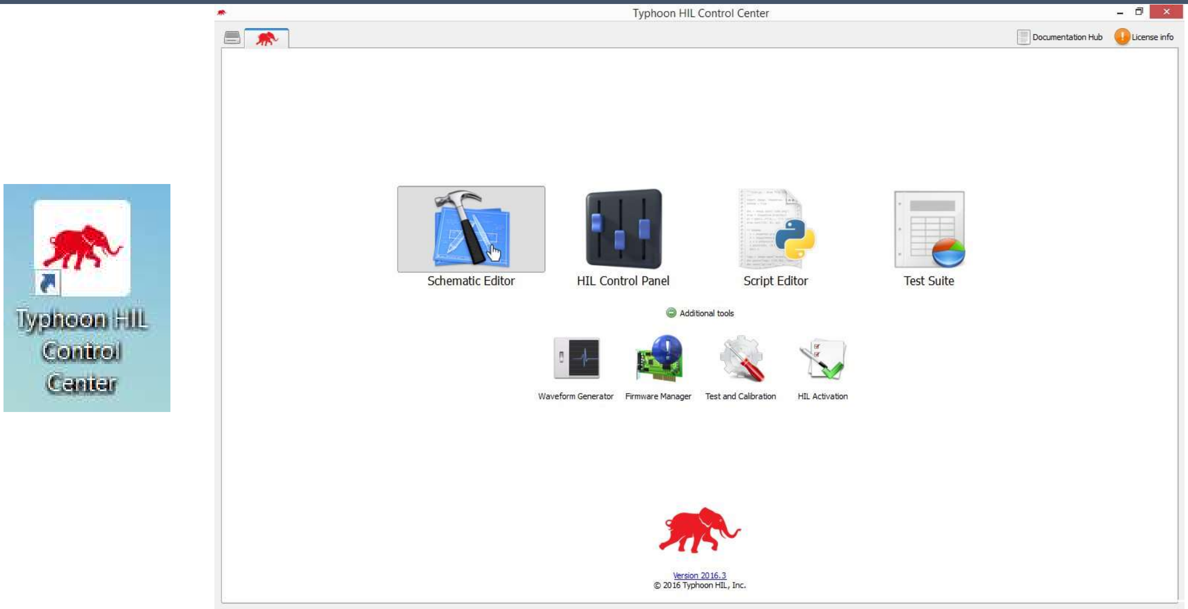
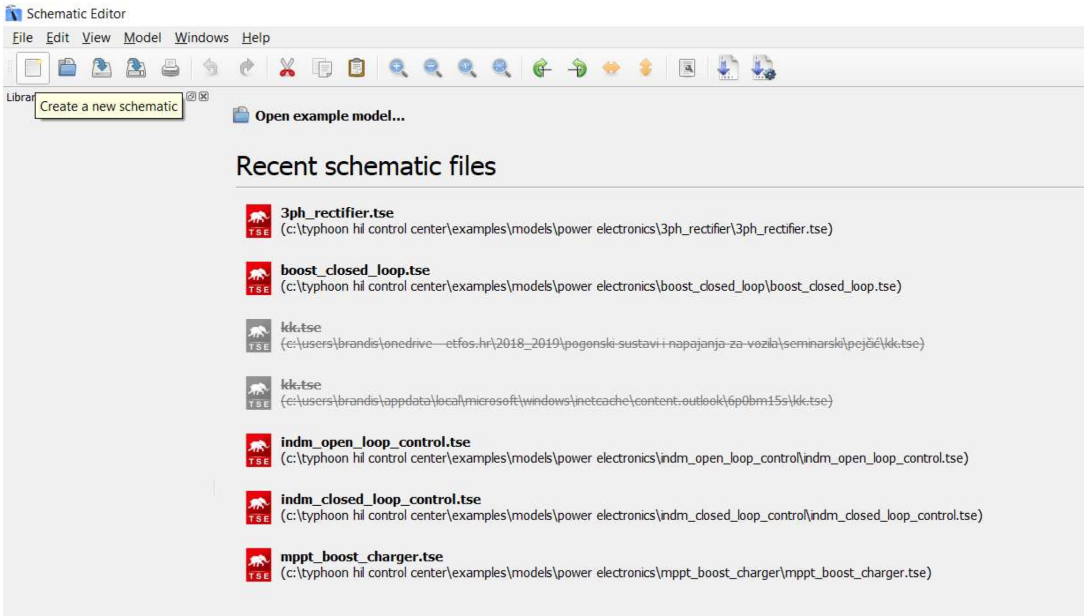


Fig.2.6: *Desktop icon of the Typhoon HIL Control Centre (left) and Typhoon HIL Control Centre (right)*

When the Typhoon HIL Control Centre is opened, to start draw the simulation model an icon of the Schematic Editor needs to be clicked (Fig.2.7). A Schematic Editor interface is shown in Fig.2.8.



Recent schematic files

3ph_rectifier.tse
(c:\typhoon hil control center\examples\models\power electronics\3ph_rectifier\3ph_rectifier.tse)

boost_closed_loop.tse
(c:\typhoon hil control center\examples\models\power electronics\boost_closed_loop\boost_closed_loop.tse)

~~kk.tse~~
~~(c:\users\brandis\onedrive - etfos.hr\2018_2019\pogonski sustavi i napajanja za vozila\seminarski\pejčić\kk.tse)~~

~~kk.tse~~
~~(c:\users\brandis\appdata\local\microsoft\windows\inetcache\content.outlook\6p0bm15s\kk.tse)~~

indm_open_loop_control.tse
(c:\typhoon hil control center\examples\models\power electronics\indm_open_loop_control\indm_open_loop_control.tse)

indm_closed_loop_control.tse
(c:\typhoon hil control center\examples\models\power electronics\indm_closed_loop_control\indm_closed_loop_control.tse)

mppt_boost_charger.tse
(c:\typhoon hil control center\examples\models\power electronics\mppt_boost_charger\mppt_boost_charger.tse)

Fig.2.7: *Schematic Editor interface*

By clicking on “Create a new schematic icon” (upper left corner), a Schematic wizard will be opened. Just click Next and Finish (the default settings are fine to start modelling). After the new blank Schematic Editor document is created (Fig.2.8) it needs to be saved (File – Save – filename). After this step a modelling can be started.

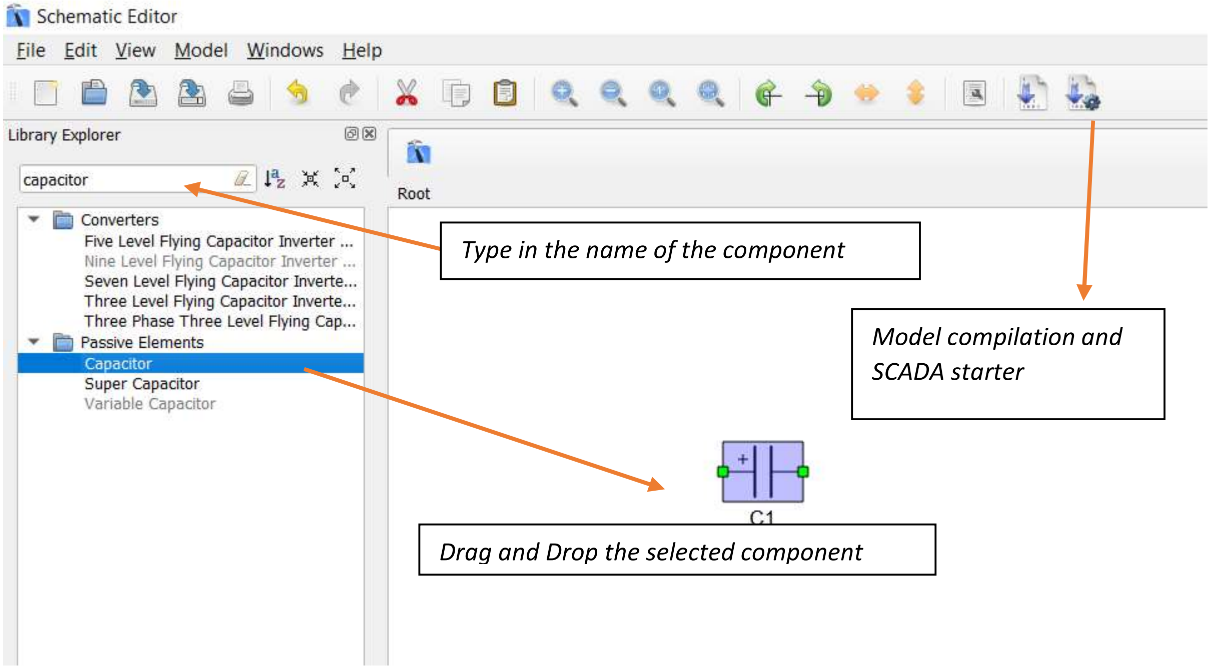


Fig.2.8 *Blank Schematic window*

A list of components for DC-DC converter will be given in form of table for faster model creation. The rectifier and inverter will be provided as finished models. The connections between schematic components are obtained by clicking on the first desired component receptacle and then on the second desired component receptacle (this will create a connection line).

After the schematic model is created, the component parameters need to be set. The component parameters will be given in converter specification. After this step, a compilation of the model needs to be done. This can be done by clicking on the Compile icon (Fig.2.8). When next window will pop-up, select the Load Model to the Virtual HIL Device. The SCADA window will be opened (Fig.2.9). Click on the Create New panel (Fig.2.9).

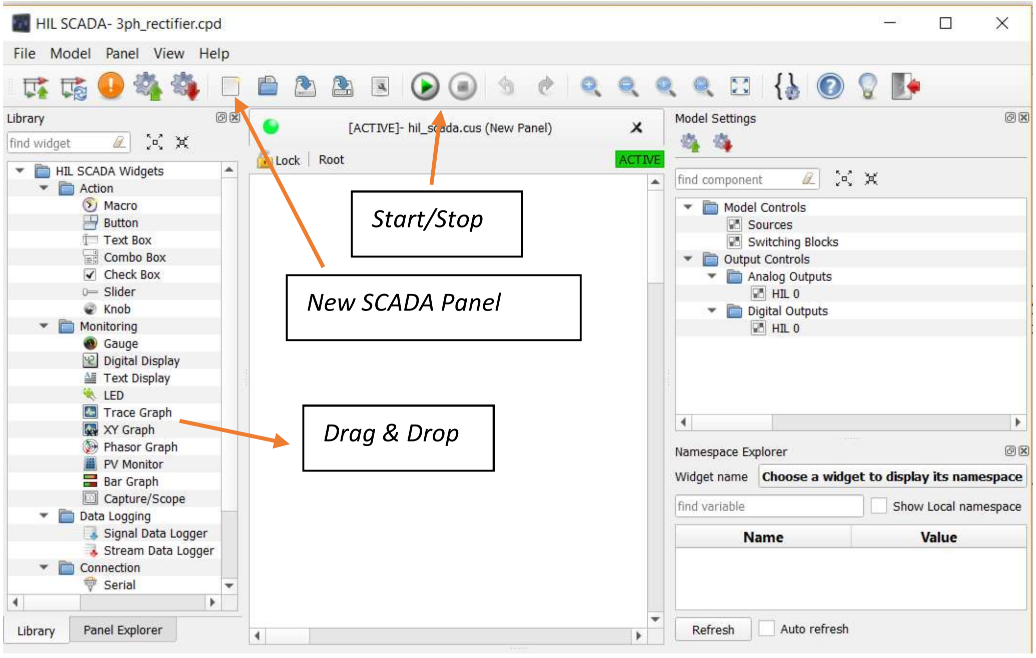
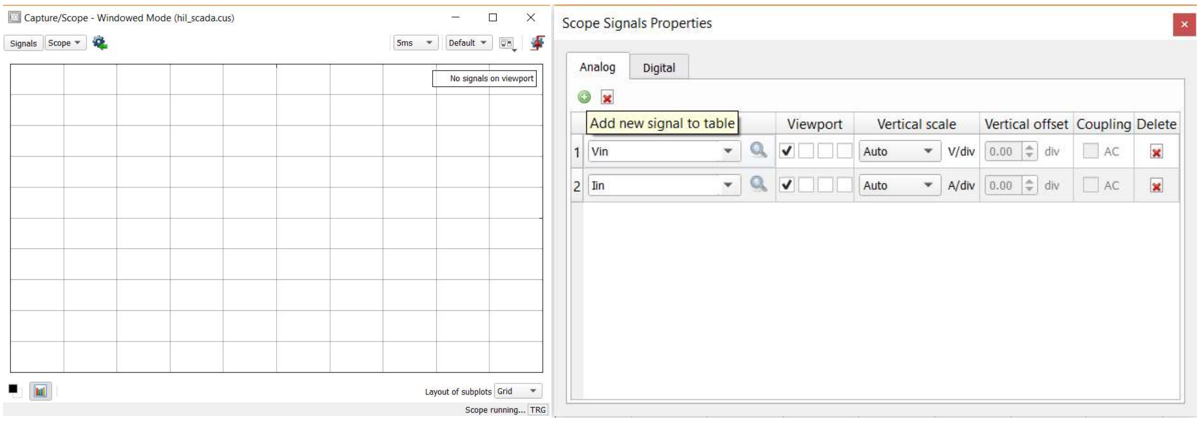


Fig.2.9: *A SCADA window*

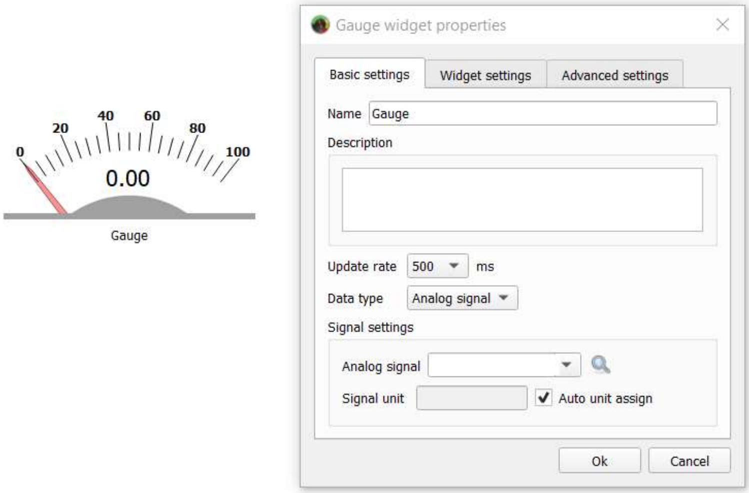
The principle of the SCADA model creation is the same as in the case of Schematic model. Just drag and drop the wanted widget to the main SCADA window. The model settings as well as the widget configurations will be given in workshop for every SCADA model of converter. After everything is set, the simulation is started by clicking on the Start button (Fig.2.9). Waveforms can be observed by opening the Capture/Scope widget of the model (Fig.2.10).



	Add new signal to table	Viewport	Vertical scale	Vertical offset	Coupling	Delete
1	Vin	☑ ☐ ☐ ☐	Auto V/div	0.00 div	☐ AC	☒
2	Iin	☑ ☐ ☐ ☐	Auto A/div	0.00 div	☐ AC	☒

Fig.2.10: *A Capture/Scope window (left) with opened Signals window (right)*

A Capture/Scope widget is essential for capturing desired waveforms. By clicking on Signals the new window will pop-up (Fig.2.10 – right). To add a new waveform, click on the green plus sign *Add new signal* to the table and select desired options (desired signal to show etc.). Also, for measuring the current and voltage values, the Gauge widget can be used (Fig.2.11).



Name: Gauge
Description:
Update rate: 500 ms
Data type: Analog signal
Signal settings
Analog signal:
Signal unit:
☑ Auto unit assign

Fig.2.11: *A Gauge widget (left) and its configuration window (right)*

To configure the Gauge widget, simply double click on the widget. In “*Analog signal* menu” pick the wanted signal which value will be measured. Also, in “*Advanced settings”*, a range of the instrument can be set.

For the end of the simulation, click on the STOP (same as the START). This were short instructions for Typhoon HIL Schematic Editor and SCADA. Detailed instructions will be given in the workshop.

2.3. Boost converter simulation

The aim of this exercise is to create a model of boost converter and to test it with given parameters. A modelling needs to be done according to the real device specifications which also will be tested later in laboratory. Finally, recorded waveform from both, simulation and real device needs to be compared with obtained waveforms from mathematical model. The Fig.2.12 shows the simulation model of the boost converter created in Typhoon HIL software. The schematic (Fig.2.12) include both, power and control section. The instructions how to create a schematic model are given in chapter 2.2. *Simulation with Typhoon HIL software – Introduction.*

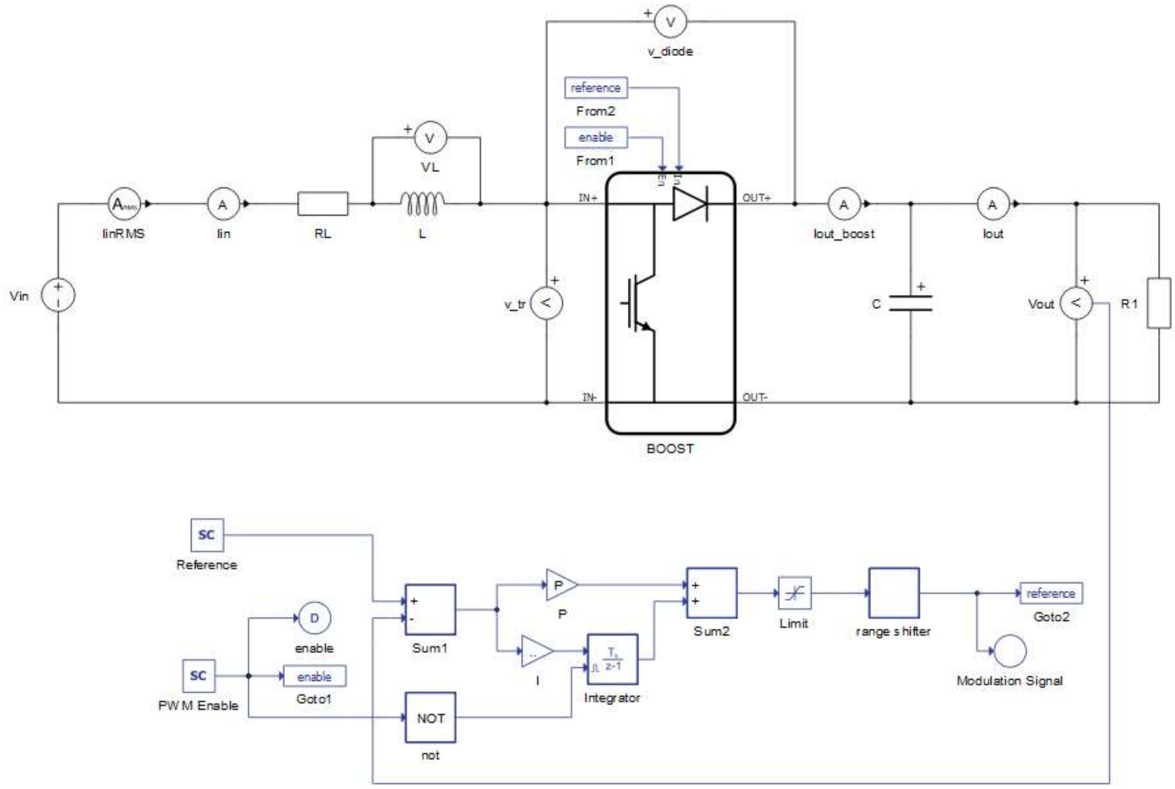


Fig.2.12: *Simulation model of the boost converter*

A complete component list of the converter (Fig.2.12) along with the parameter configuration for each component is shown in the Tab.2.1. Parameters configuration for each component is set by the double click on desired component and entering the given parameter.

Tab.2.1: *DC-DC components list and parameters configuration*

Library component name	**Parameters configuration**
Boost converter block	*Default*
Voltage Source *Vs*	*Source nature:* Constant *Voltage:* 14,5 V
Current measurement (Ammeter) *Iin, Iout_boost, Iout*	*Default*
Current RMS (RMS Ammeter) *IinRMS*	*Default*
Voltage measurement (Voltmeter) *Vin, VL, v_tr, v_diode, Vout*	*Default*
Capacitor *C*	470 µF (470e-6 F)
Resistor *R1*	56 Ω
Resistor *RL*	0,73 Ω
Inductor *L*	718 µH (718e-6 F)
Signal From *2*	*Name:* Reference
Signal From *1*	*Name:* Enable
Signal Goto *2*	*Name:* Reference
Signal Goto *1*	*Name:* Enable
SCADA Input *Reference*	*Signal type:* Real *Min:* 15

	Max: 30 *Default value:* 28 *Unit:* V *Execution rate:* Ts
SCADA Input *PWM Enable*	*Signal type:* Real *Min:* 0 *Max:* 1 *Default value:* 0 *Unit:* *Execution rate:* Ts
Sum1, Sum2	*Signs:* +-, ++
Logical operator *not*	*Operator:* NOT
Gain *P*	*Gain:* P
Gain *I*	*Gain:* I
Integrator	*Reset:* LEVEL
Limit	*Upper Level:* 1 *Lower Limit:* -1
Digital Probe *enable*	*Default*
Probe *Modulation Signal*	*Default*

After finishing the converter modelling and configuring, the model needs to be compiled. After this action, the SCADA window will open. Open the SCADA template for the boost converter with the guidance of the assistant. Open the Capture/Scope widget, add desired waveforms for recording (according to the Fig.2.13 – Fig.2.15) observed it and find appropriate waveforms. Read the values from the instruments and write down the results in Tab.2.2.

Reading results for a boost converter simulation model

Read the measurements from the instruments and write it down in the Tab.2.2.

Tab.2.2: *Measured mean values of current and voltage*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

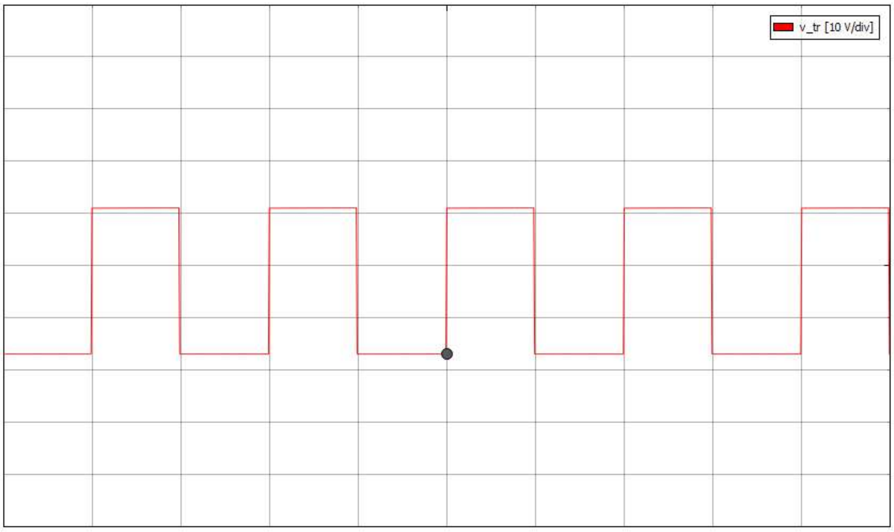


Fig.2.13: *Voltage waveform of the controlled switch*

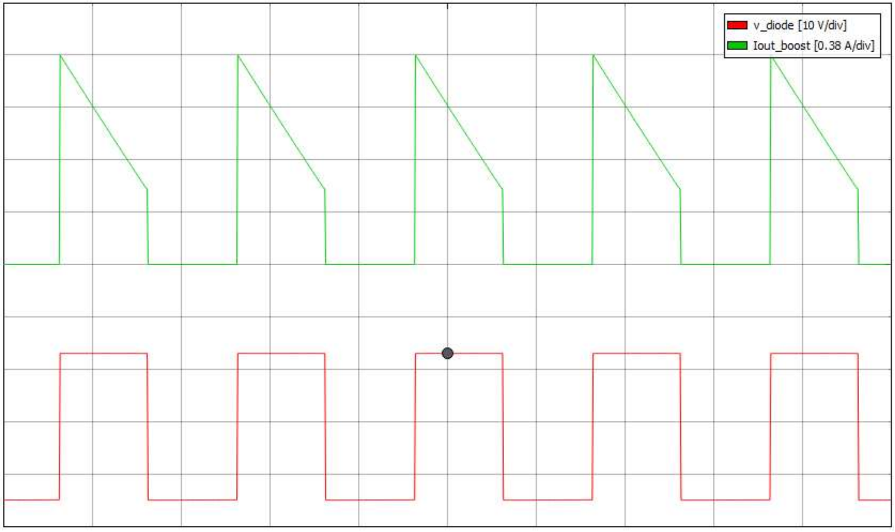


Fig.2.14: *Voltage and current waveforms of the diode*

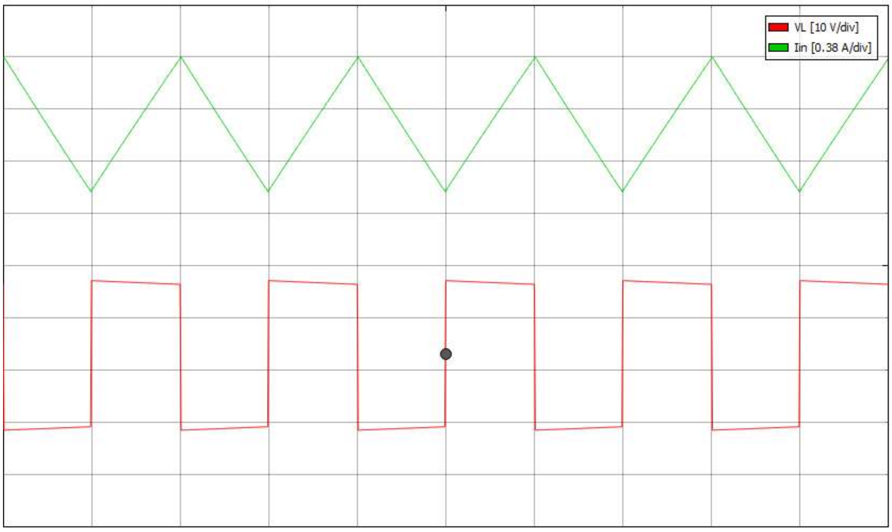


Fig.2.15: *Voltage and current of the inductor*

2.4. Boost converter for laboratory measurements

After the simulations are done and waveforms are obtained, the next step is to test the physical models of converters in laboratory with the same specifications as the simulation models. The aim of this measurements is to record required waveforms and finally to compare it with the waveforms obtained by simulation and by mathematical analysis. The Fig.2.16 shows the physical model (with wiring) of the boost converter done by students. Connect the measuring equipment according to the schematic shown in Fig.2.16, and then perform the measurements in the following order:

1. Apply the 14,5 V voltage from the DC voltage source located on the laboratory table to the input of the converter and select the frequency range *f*≤25 kHz with the switch
2. Turn the converter ON and observe the current waveform of the inductor LE on the oscilloscope. Set the converter frequency so that converter works in continuous mode with the switching frequency fs = 10 kHz. Read and write down the values shown by the ammeters and the voltmeter
3. Observe and record the current and voltage waveforms of the transistor, diode as well as the inductor with the help of the assistant
4. Turn OFF the boost converter.

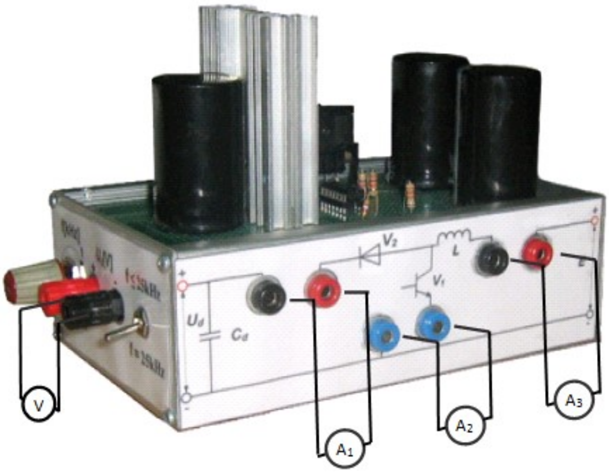


Fig.2.16: *Physical realization of the boost converter*

Measuring results from a boost converter

Read the measurements from the instruments and write it down in the Tab.2.3.

Tab.2.3: *Measured mean values of current and voltage*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

Compare recorded waveforms of voltage and current with waveforms shown in Fig.2.17 – Fig.2.18.

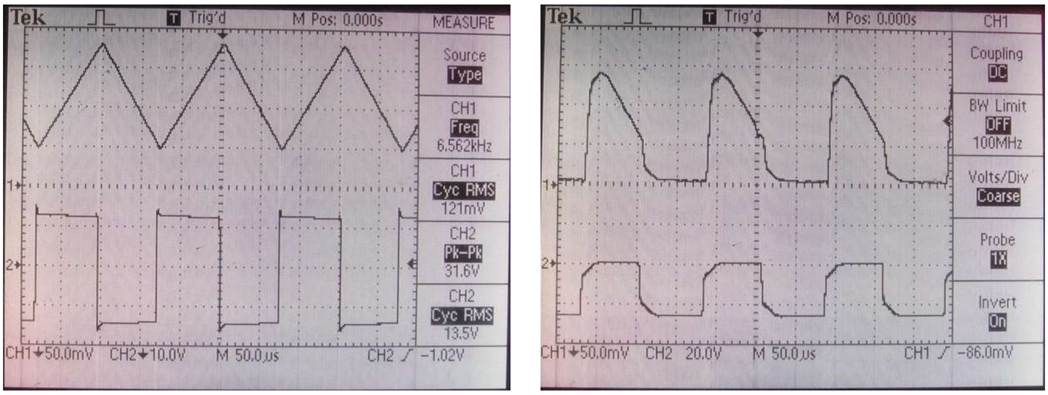


Fig.2.17: *Current and voltage waveform of the inductor (left) and diode (right)*

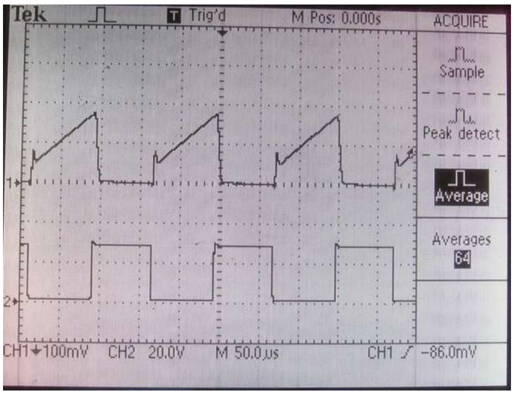


Fig.2.18: *Voltage of the transistor*

3. Single phase diode rectifier in bridge topology with capacitive load

The rectifiers are electrical devices that connect the AC network and DC load, whereby the energy is transmitted from the alternating network to the DC load (Fig 3.1 – symbol of the rectifier). The power converter switching components of non-controlled rectifiers are diodes. Depending on the number of AC network phases, non-controlled rectifiers are divided into single-phase or three-phase. For higher power loads most commonly used are three-phase non-adjustable rectifiers, and for less power-loads a single-phase non-controlled rectifiers are used.

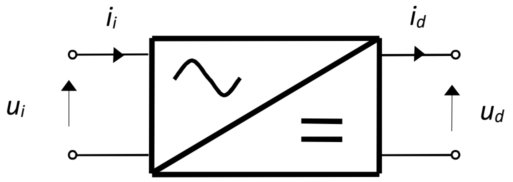


Fig.3.1. Symbol of the single phase rectifier

DC loads are divided into energy and electronic. Typical examples of power energy loads supply by non-controlled rectifiers are DC-motors, magnets, and battery packs when process of charging is going on. Power supplies of the electronic loads (personal and processor computers, instrumentation equipment, and audio-video devices) are considered to be electronic loads. Electronic loads require continuous voltage for supply and such loads are modelled by a parallel combination of capacitance and resistance, this is a reason why they are considered as capacitive loads. Power energy consumables should be provided with continuous current, and they are modelled with a serial combination of inductance and resistance circuit, which is often considered as inductive loads.

3.1. Diode rectifier modelling

The switching components are modelled with ideal diodes. The load is considered to be the constant voltage load U_d (because for capacity and the resistance of the load are ensured ($C_d R_d \gg T_s$), Fig.3.2. Power grid is modelled by AC voltage source with neglect of resistance and inductance of power grid. The NTC resistor is modelled with Linear Resistance R, which showed the best technical solution for reducing the inrush current for the first connection on power grid. At the moment of connection on to the power grid, the resistance value is several tens Ohm, which limits the inrush current. To obtain the analysis in the steady-state, resistance is considered to be several Ohm, which does not significantly increase the losses of the rectifier. The part of the load was modelled with non-linear resistance R_d with *i-u* characteristic that is ensured $U_d I_d(0)$ = const., i.e. $R_d = kU_d^2$.

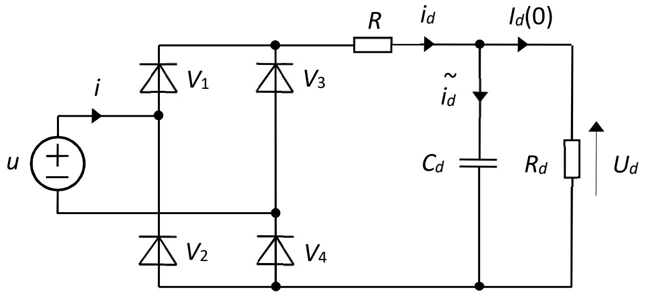


Fig.3.2: *The equivalent circuit of the diode rectifier with capacitive load*

The above assumptions substantially simplify the analysis of the rectifier. The load current was discontinued. The pair of diode (V_1 and V_4) is in state ON during a positive half-period of AC voltage: $\alpha_0 \le \omega t \le \pi - \alpha_0$, when the AC voltage is higher than the voltage of the load; U_d, Figure 3.3. In this regard:

$$\alpha_0 = \arcsin\frac{U_d}{\hat{U}}, \tag{3-1}$$

while the waveform of the load current was determine in mentioned interval with the expression:

$$i_d = \frac{u - U_d}{R} = \frac{\hat{U}\sin\omega t - U_d}{R} \tag{3-2}$$

The second pair of diodes (V_2 and V_3) is in state ON during the negative half-period of the AC voltage in the interval: $\pi+\alpha_0 \le \omega t \le 2\pi-\alpha_0$.

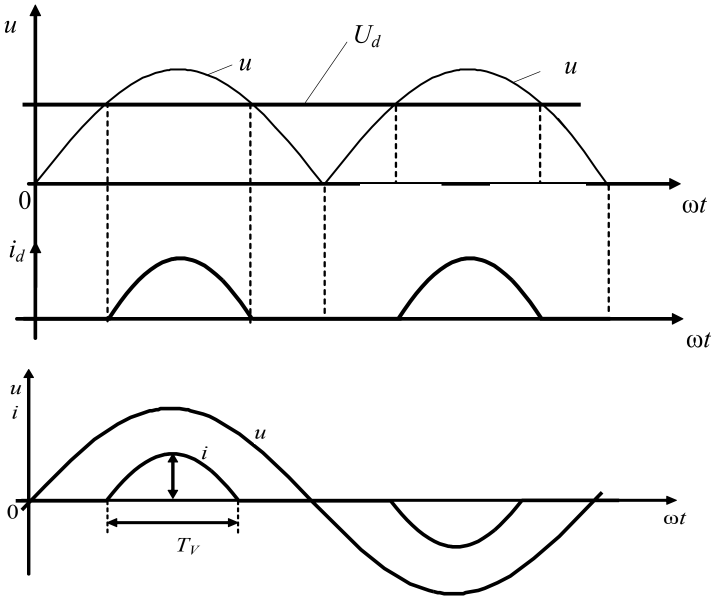


Fig.3.3: *Characteristic diode rectifier waveforms for capacitive load*

The duration of diode ON state, T_V is indicated on Fig.3.3. According to the Fig.3.3 is obviously: $\omega T_V = \pi - 2\alpha_0$ and using the expression (3-1) :

$$\frac{T_V}{T} = \frac{1}{2} - \frac{1}{\pi}\arcsin\frac{U_d}{\hat{U}} \tag{3-3}$$

Typical values of duration of diode ON-state in practice are 2 ms<T_s<3ms. For the mean value of the current value the expiration is:

$$I_d(0) = \frac{1}{\pi}\int_{\alpha_0}^{\pi-\alpha_0}\frac{\hat{U}\sin\omega t - U_d}{R}\,d\omega t = \frac{U_d}{R_d} \tag{3-4}$$

After integrating and equalizing as well as by using terms (3-1) and (3-3), the value of the load voltage was obtained:

$$U_d = \hat{U}\frac{1}{\sqrt{1+\frac{\pi^2}{4}\left(2\frac{T_V}{T}+\frac{R}{R_d}\right)}}, \tag{3-5}$$

Knowing the term of the load current (3-2), the current of the grid is determined, and within the half-period of the grid voltage $u = \hat{U}\sin\omega t$, $0 \le \omega t \le \pi$, can be described with sufficient accuracy as:

$$i = \begin{cases} 0, & 0 < \omega t < \alpha_0 \\ i_d, & \alpha_0 < \omega t < \pi - \alpha_0 \\ 0, & \pi - \alpha_0 < \omega t < \pi \end{cases} \tag{3-6}$$

3.2. Diode rectifier simulation

The aim of this exercise is to create a model of single-phase diode rectifier in the bridge topology with capacitive load in Typhoon HIL simulation software and to test it. A modelling needs to be done according to the real device specifications which also will be tested later in laboratory. Finally, recorded waveform from both, simulation and real device needs to be compared with obtained waveforms from mathematical model. The Fig.3.4 shows the simulation model created in Typhoon HIL software. This schematic will be created on the workshop with guidance of the assistant.

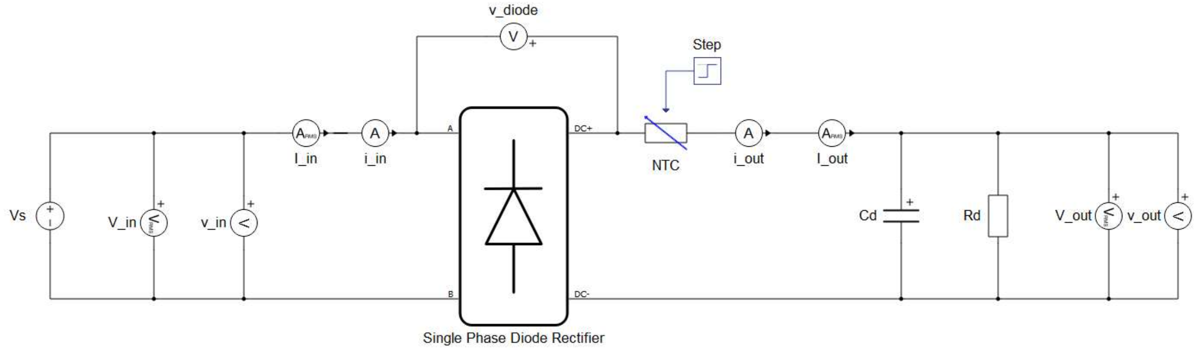


Fig.3.4: *Simulation model of single-phase diode rectifier with capacitive load*

After finishing the converter modelling and configuring, compile it following the instructions given before (click on the icon *Compile*). After this action, the SCADA window will open. Create new SCADA Panel as explained before. The widget needed for this converter is the Capture/Scope and Gauge widgets, so drag and drop it on the blank Panel window. Open the Capture/Scope widget, add desired waveforms for recording (according to the Fig.3.5 – Fig.3.7), observed it and find appropriate waveforms. Read the values from the instruments and write down the results in Tab.3.1.

Reading characteristic values form the single-phase diode rectifier simulation model

Read the values from the instruments and write it down in the Tab.3.1.

Tab.3.1: *Measured values of currents and voltages*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

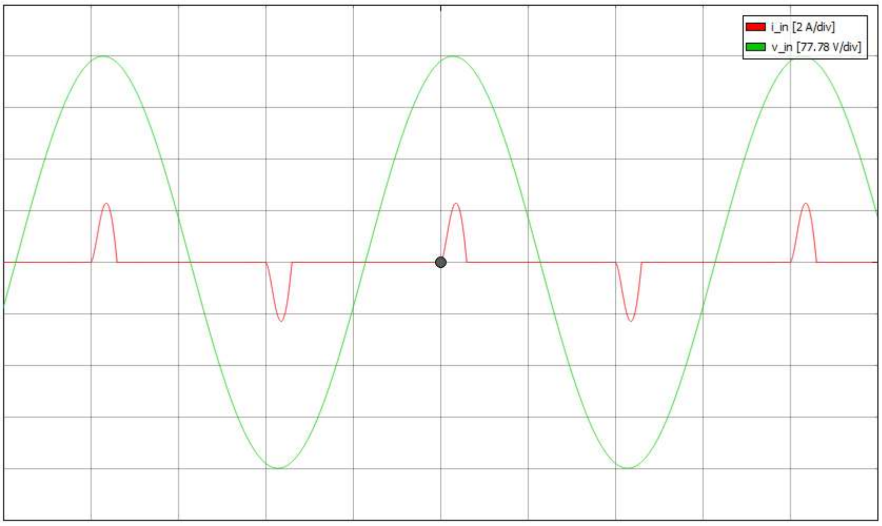


Fig.3.5: *Input current i_{in} and voltage v_{in}*

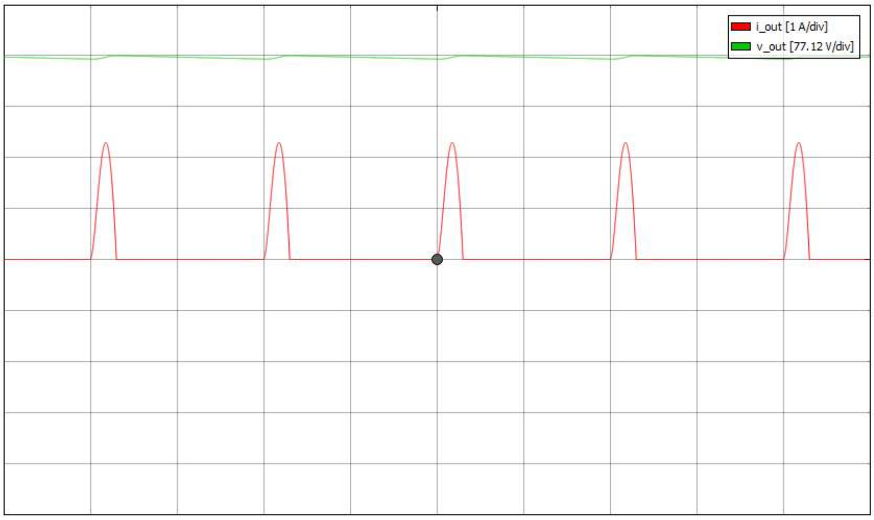


Fig.3.6: *Output current i_{out} and voltage v_{out}*

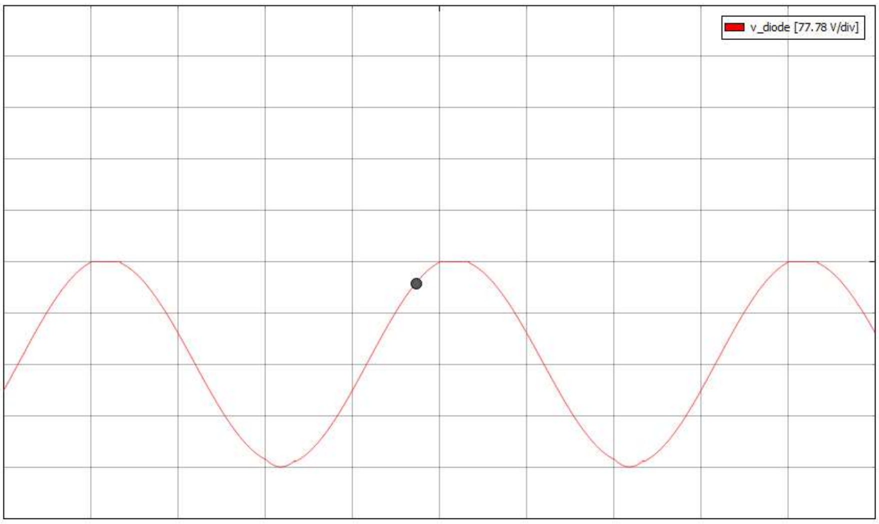


Fig.3.7: *Voltage of the diode v_{diode}*

3.3. Diode rectifier for laboratory measurements

After the simulations are done and waveforms are obtained, the next step is to test the physical model of converters in laboratory with the same specifications as the simulation models. The aim of this measurements is to record required waveforms and finally to compare it with the waveforms obtained by simulation and by mathematical analysis. The Fig.3.8 shows the physical model (with wiring) of the single-phase diode rectifier done by students. Connect the measuring equipment according to the schematic shown in Fig.3.8, and then perform the measurements in the following order:

1. Connect the single-phase rectifier to the single-phase power grid via connector on the laboratory table. Set power on, write down the values displayed by ammeters and voltmeters.
2. On oscilloscope observe and record the waveforms of current and voltage.
3. Power off the device, disconnect the rectifier from the lab`s table, and disconnect the measuring instruments.

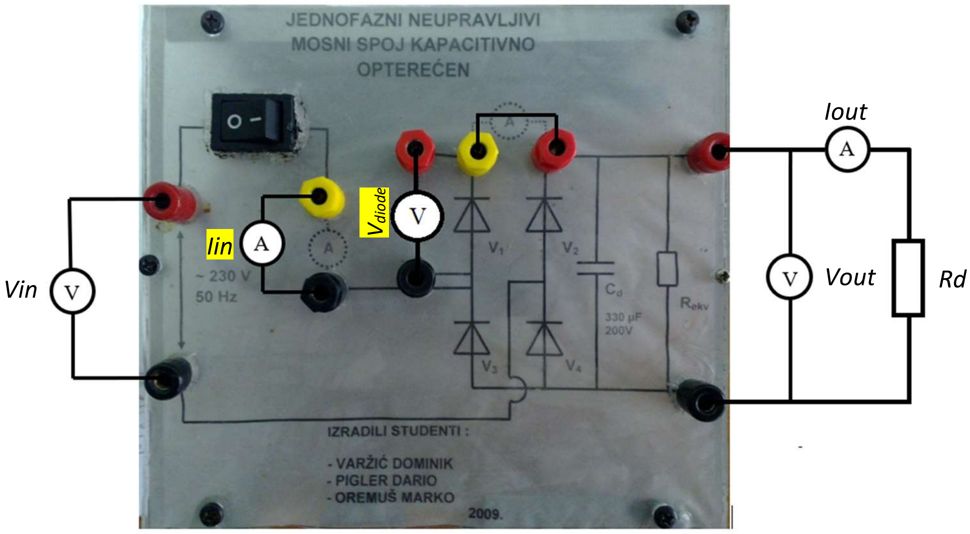


***Fig.3.8:** Physical model of single-phase diode rectifier with capacitive load*

Measurement results

Read the measurements from the instruments and write it down in the Tab.3.2.

***Tab.3.2:** Measured values of current and voltage*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

Observe recorded waveforms of voltage and current and compare them with the waveforms shown in Fig.3.9.

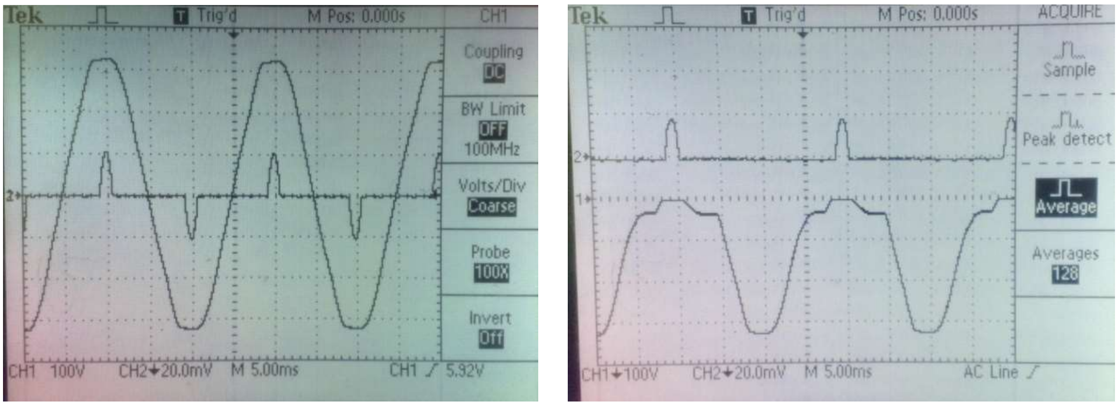


***Fig.3.9:** Input current i_{in} and voltage v_{in} (left) and voltage of the diode v_{diode}*

4. Single-phase autonomous voltage inverter in H-bridge topology with inductive load

DC source and AC loads are connected by inverters. If the AC load is AC network and if it is active, the inverters are non-autonomous. If the AC network is passive, the inverters are autonomous. Depending on the character of the DC source, autonomous inverters are divided into voltage and current controlled. Given the number of alternating network phases, the inverters can be single-phase or three-phase. The symbol of a single-phase voltage converter is shown in Fig. 4.1.

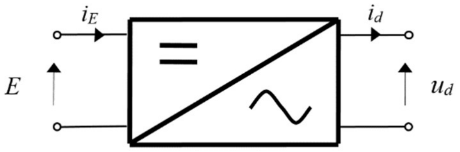


Fig.4.1: *Symbol of the single-phase voltage controlled inverter*

4.1. Voltage control inverter modelling

Let us assume that the power switching components are ideal and that the four MOSFET or IGBT as switching component in H-bridge configuration are modelled using an ideal diode and a control valve in anti-parallel connection. The inverter scheme is shown in Figure 4.2.

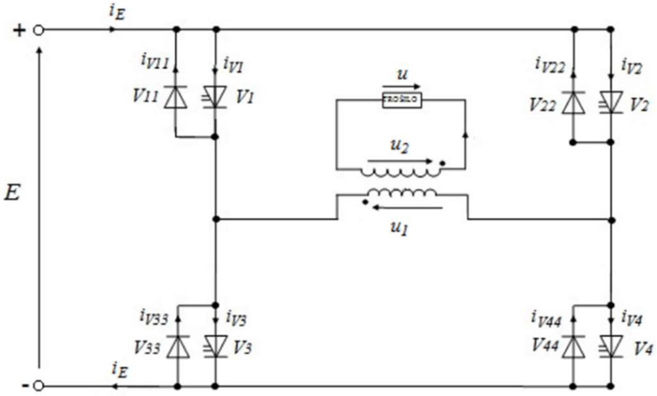


Fig.4.2*: Equivalent circuit of the single-phase voltage controlled inverter in H-bridge topology*

Analysis of the inverter in the steady-state

Valves V_1 and V_4 and V_2 and V_3 are controlled together. When a pair of V_1 and V_4 valves are in state ON, the pair of V_2 and V_3 valves are in the state OFF. The load voltage is equal to the $N{\cdot}E$ when V_1 and V_4 are opposite load voltage is equal to $-N{\cdot}E$, where N is transformation ratio.

Therefore, in the analysis of the work with regard to the load voltage there are two characteristic intervals:

A) Valves V_1 / V_4 or V_{11}/ V_{44} are in state ON, and V_2 /V_3 or V_{22}/ V_{33} are in state OFF ($0 \le t \le T/2$),

B) Valves V_2 /V_3 or V_{22}/ V_{33} are in state ON, and V_1 /V_4 or V_{11}/ V_{44} are in state OFF ($T/2 \le t \le T$).

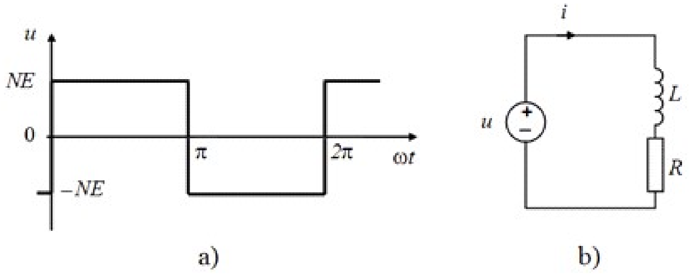


Fig.4.3: *Simplified equivalent circuit for analysis*

With propose control of the valves, the simplified equivalent circuit from Fig. 4.3b is obtained and load voltage is shown on Fig. 4.3a.

For the simplified equivalent circuit with inductive load by applying Kirchhoff`s voltage law is given: $u = u_L + iR$, along with the writing of the constitutive relation between current and voltage for inductance; $u_L = L\frac{di}{dt} = \omega L\frac{di}{d\omega t}$, a differential equation is obtained:

$$\mathrm{tg}\varphi \frac{di}{d(\omega t)} + i = \frac{NE}{R}\cdot\begin{cases}1 & +0\le\omega t\le\pi-0\\ -1 & \pi+0\le\omega t\le 2\pi-0\end{cases} \qquad (4\text{-}1)$$

where is: $\mathrm{tg}\varphi = \omega L/R$, and after solving the differential equation using the methods for solving multi-harmonic networks, the expression for the load current waveform is obtained:

$$i = \frac{NE}{R}\cdot\begin{cases}1-\dfrac{2e^{-\frac{\omega t}{\mathrm{tg}\varphi}}}{1+e^{-\frac{\pi}{\mathrm{tg}\varphi}}} & +0\le\omega t\le\pi-0\\[2ex] -\left(1-\dfrac{2e^{-\frac{\omega t-\pi}{\mathrm{tg}\varphi}}}{1+e^{-\frac{\pi}{\mathrm{tg}\varphi}}}\right) & \pi+0\le\omega t\le 2\pi-0\end{cases} \qquad (4\text{-}2)$$

The characteristic waveforms of voltage and current are shown in Fig. 4.4.

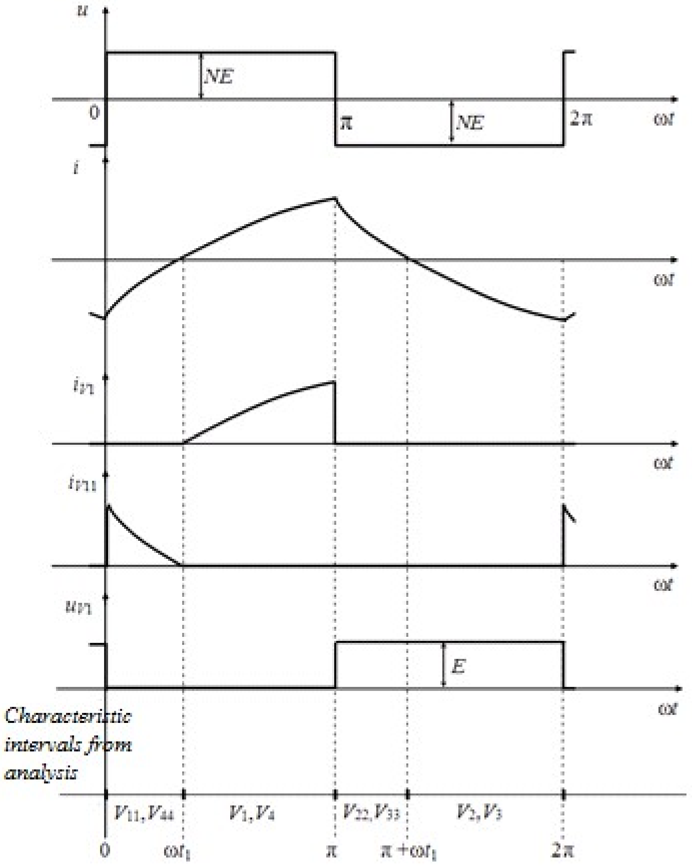


Fig.4.4: *Characteristic waveforms obtained by analysis*

By separating the load voltage form Fig. 4.3a in the Fourier line, the expression for the load voltage is given:

$$u = \frac{4NE}{\pi}\sum_{n=1}^{\infty}\frac{\sin n\omega t}{n}, \; n = 1, 3, 5, \dots \qquad (4\text{-}3)$$

4.2. Voltage control inverter simulation

The aim of this exercise is to create a model of the single-phase inverter and to test it with given parameters. A modelling needs to be done according to the real device specifications which also will be tested later in laboratory. Finally, recorded waveform from both, simulation and real device needs to be compared with obtained waveforms from mathematical model. The Fig.4.5 shows the simulation model of the inverter created in Typhoon HIL software. Due to limited course duration, a finished single-phase inverter model will be provided and explained in the workshop.

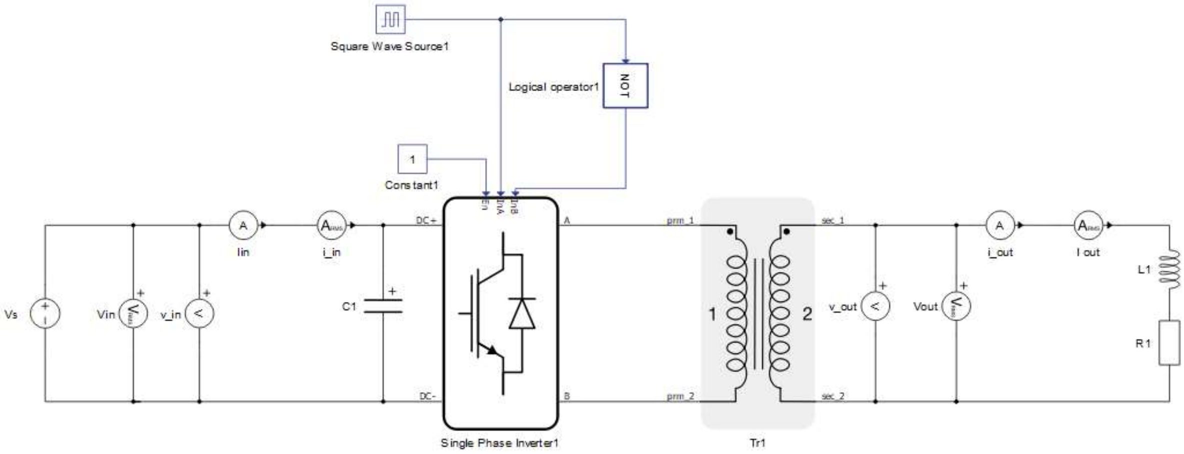


Fig.4.5: *Simulation model of the single-phase inverter*

After finishing the inverter modelling and configuring, the model needs to be compiled. After this action, the SCADA window will open. Open the SCADA template for the inverter with the guidance of the assistant. Open the Capture/Scope widget, add desired waveforms for recording (according to the Fig.4.6 – Fig.4.8), observed it and find appropriate waveforms. Read the values from the instruments and write down the results in Tab.4.1.

Readings from inverter simulation model

Read the characteristic value from the simulation model and write it down in the Tab.4.1.

Tab.4.1: *Characteristic values of currents and voltages*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

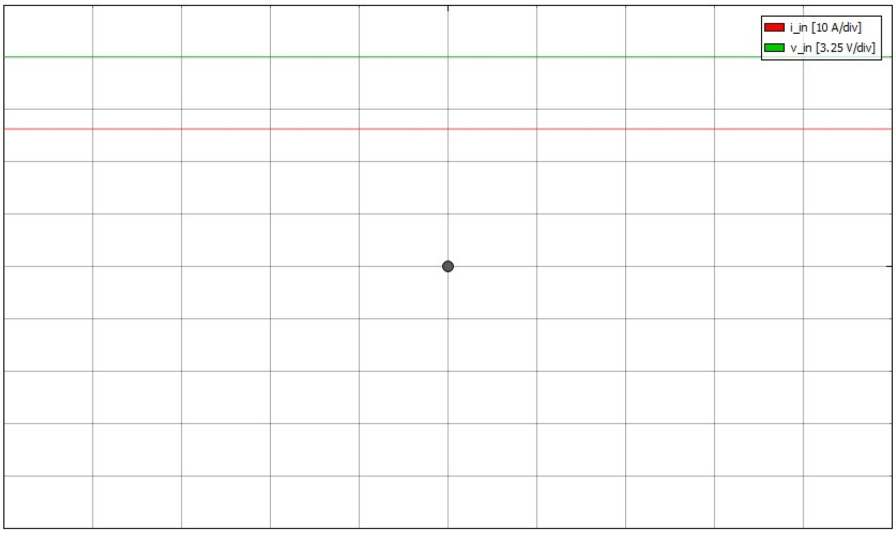


Fig.4.6: *Power grid voltage and current*

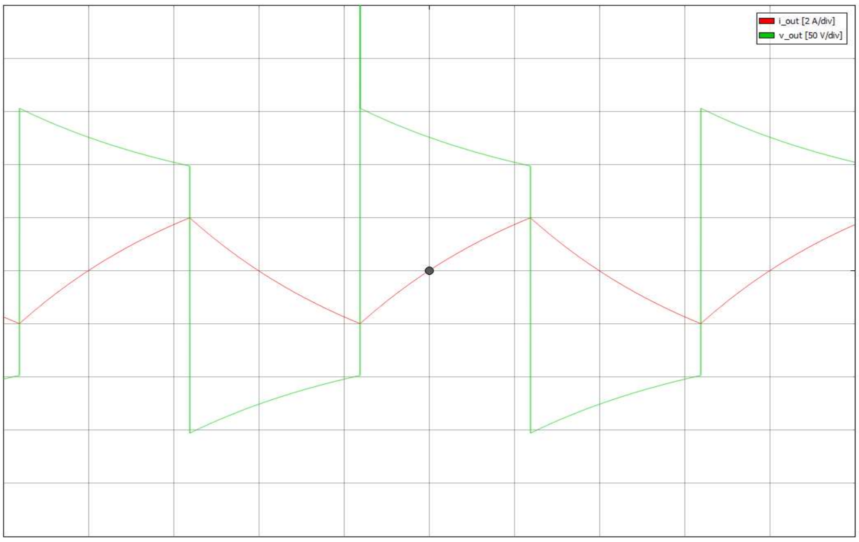


Fig.4.7: *Voltage and current of the load*

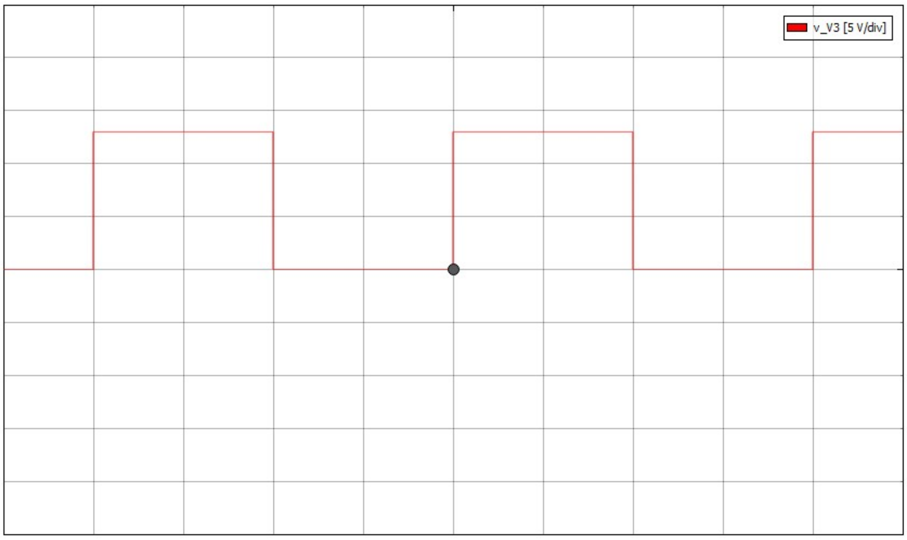


Fig.4.8: *IGBT voltage waveform*

4.3. Voltage inverter for laboratory measurements

After the simulations are done and waveforms are obtained, the next step is to test the physical models of converters in laboratory with the same specifications as the simulation models. The aim of this measurements is to record required waveforms and finally to compare it with the waveforms obtained by simulation and by mathematical analysis. The Fig.4.9 shows the physical model (with wiring) of the inverter done by students. Connect the measuring equipment according to the schematic shown in Fig.4.9, and then perform the measurements in the following order:

1. Set the input DC voltage to 12,5 V. Use the switch S on the workbench
2. On the oscilloscope observe and record the waveform of the current and voltage of the power grid, load and the voltage of the transistor V3
3. Turn OFF the switch, reduce the voltage of the source to zero, turn off the inverter and disconnect the measuring instruments.

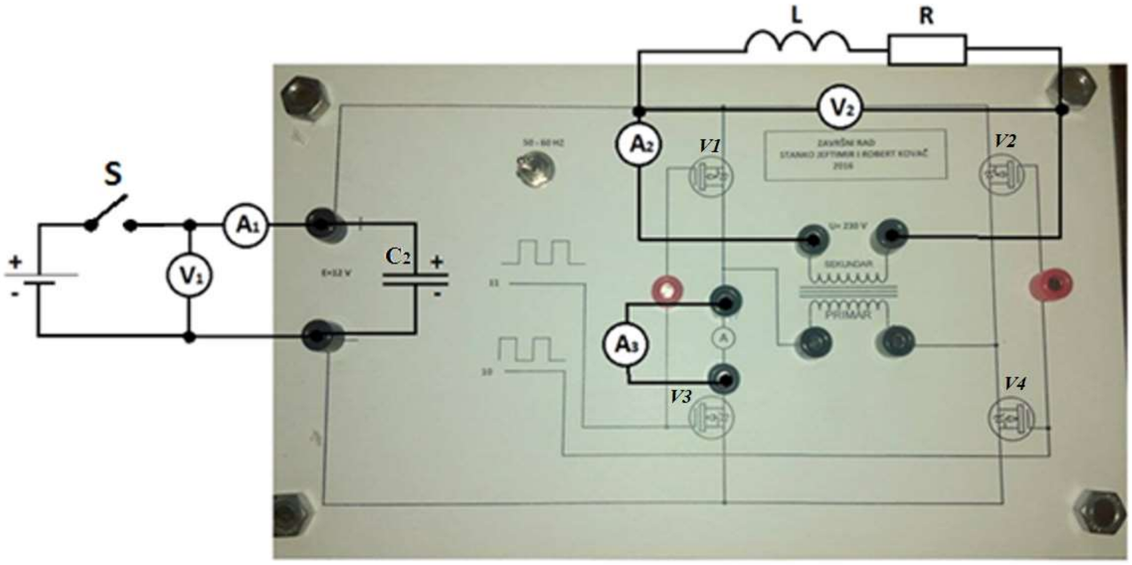


Fig.4.9: *Physical realisation of the inverter*

Measurements results

Read the measurements from the instruments and write it down in the Tab.4.2.

Tab.4.2: *Measured values of currents and voltages*

Vin [V]	*Iin* [A]	*Vout* [V]	*Iout* [A]

Observe recorded waveforms of voltage and current and compare them to the waveforms shown in Fig.4.10 – Fig.4.11.

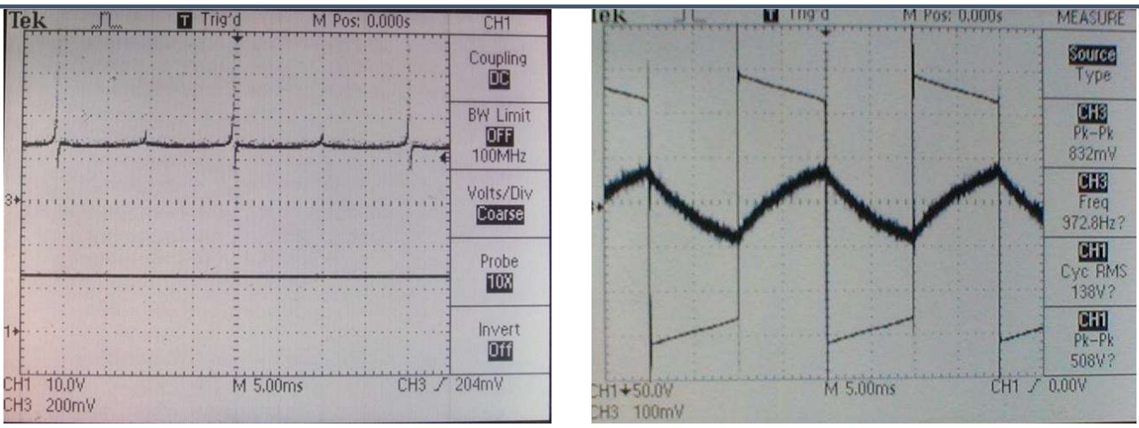


Fig.4.10: *Power grid voltage (left) and load (right) voltage and load current waveforms*

Fig.4.11: *Voltage waveform of the transistor*